Bricsys®

BricsCAD (V25)

Product documentation

Bricsys®

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1. Installation and licensing

1.1 BricsCAD system requirements

System requirements and recommendations.

Supported Operating Systems	• Windows 10 (x86-64), Windows 11 (x86-64). • ***Note:*** Windows 10 Operating System will reach end-of-support by Microsoft on 14 October, 2025. • Ubuntu 22.04 LTS and higher Ubuntu versions that are still supported by Canonical (if you want a stable experience, LTS tends to be a good choice). • openSUSE builds, later than April 2022, with glibc version >= 2.35. • Fedora builds, later than April 2022, with glibc version >= 2.35. • macOS 11.0 (Big Sur) or higher (see the **Apple compatibility for BricsCAD** article). ***Note:*** The Windows and Linux versions of BricsCAD are built and supported for the x86-64 (AMD64) architecture.

Minimum CPU: • Intel® Core™ i5 • AMD Ryzen™ 5	**Recommended CPU:** • Intel® Core™ i7 • Intel® Core™ i9 • AMD Ryzen™ 7 • AMD Ryzen™ 9	The CPU with the highest single-thread performance rating will always excel with BricsCAD. Additional CPU cores will bring in higher performance, as more and more functions get multithreaded computation in BricsCAD. ***Note:*** • **Intel® Xeon® and AMD Ryzen™ Threadripper™ CPUs**: These top-tier workstation CPUs support large on-chip memory cache, more PCIe® I/O lanes and multi-channel, high speed memory architectures. • Since V21.2.06, BricsCAD (Mac) works on Rosetta 2 (Intel emulation on **Apple silicon** chips). A native **Apple silicon** build is planned in a future version.
Minimum System Memory: 8 GB	**Recommended System Memory:** • 16 GB	For maximum performance, determine the memory configuration of your PC and ensure that each memory channel on the motherboard is populated with at least one memory module.

<table>
<tr>
<td>Minimum Disk Space:
3 GB for BricsCAD Ultimate, installed</td>
<td>Recommended Disk Space:
(As needed)</td>
<td>We strongly recommend using SSD (Sata 3 / PCI-E NVME) as system drives, for both BricsCAD and Windows. Drawings and support files can reside on slower mechanical hard disk drives, if necessary.

Note: Solid-state disks can be up to 10x faster reading and 20x faster writing data than a mid-range hard disk drive</td>
</tr>
<tr>
<td>Minimum Display System:
• Full HD (1920 x 1080) True Color display and a graphics card with 4 GB VRAM
• Apple Mac Displays</td>
<td>Recommended Display System:
• Multiple UHD (3840 x 2160) or 4k True Color displays, with a PCIe® graphics card w/GPUs and 8 GB VRAM
• Apple Mac Retina (4K and 5K) Displays</td>
<td>On Microsoft Windows systems, most graphics cards & GPUs from NVIDIA, AMD and Intel® are supported.

Note: If you work with point clouds and big models in the model space, we advise changing Windows' Graphic settings as follows:
• Go to Graphic settings
• Add BricsCAD as an app to set the preferences
• Set the option to High Performance

On Linux systems, most graphics cards and GPUs from NVIDIA and AMD are supported.
On Mac systems, native Apple display hardware is supported.
BricsCAD’s non-wireframe display modes are generated by RedSDK technology from Redway3D®. The GPU is used to render this modes.

Note: We suggest that you install the latest drivers available for your display hardware, as recommended by the manufacturer.

Note to Linux users: Hardware acceleration of 3D graphics on Linux systems is NOT supported on:
• Intel® graphics chipsets/cards.
• Laptops with dual graphics adapters.</td>
</tr>
</table>

1.2 Overview of licensing options

1.2.1 License levels

There are 5 license levels for BricsCAD:

- BricsCAD Lite
- BricsCAD Pro

- BricsCAD BIM
- BricsCAD Mechanical
- BricsCAD Ultimate

1.2.1.1 BricsCAD Lite

If you only create general 2D designs, BricsCAD Lite may be the right choice for you. Compared to AutoCAD LT®, it also allows to view, move, copy, or mirror 3D models. However, it is not possible to edit the model or create new 3D entities. BricsCAD Lite also runs Lisp routines. Lisp allows BricsCAD users to load customized routines or small applications that will run in BricsCAD.

1.2.1.2 BricsCAD Pro

BricsCAD Pro includes direct 3D modeling with a full 3D constraint system, rendering with materials and lighting and access to third party applications.

1.2.1.3 BricsCAD BIM

BricsCAD BIM includes all BricsCAD Pro features. You can use your current CAD skills to smoothly move to creating real Building Information Models in record time. One product, one workflow, all in industry-standard DWG.

1.2.1.4 BricsCAD Mechanical

On top of all BricsCAD Pro features, BricsCAD Mechanical includes a complete mechanical design toolkit, all in industry-standard DWG, including sheet metal functionality.

1.2.1.5 BricsCAD Ultimate

BricsCAD Ultimate combines the BricsCAD BIM and BricsCAD Mechanical features.

1.2.2 License types

There are 2 license types for BricsCAD:

- Perpetual license.
- Subscription license.

1.2.2.1 Perpetual license

When you purchase a perpetual license, you own the software. BricsCAD editions run on Microsoft Windows, macOS and Linux.

There are 2 maintenance options:

- The maintenance option includes automatic upgrades and support.
- You can purchase upgrades at your discretion.

1.2.2.2 Subscription license

When you choose the subscription option, you have access to the latest version of BricsCAD for 1 year. It includes Priority Support and access to all new versions that are released during the time your subscription is active.

1.2.3 License options

License options include academic, single-user, network, and volume licensing.

1.2.3.1 Academic license

Students, faculties, instructors, and educational institutions have free access to all BricsCAD software for 12 months. You can renew it yearly with a current student ID. Register as a student or school/teacher with your academic information.

Note: The academic license can be obtain for single-user, network or volume licensing.

1.2.3.2 Single-user license

The single-user license agreement is for a single user. You may install and use one copy of BricsCAD for a single operating system, on a single computer.

1.2.3.3 Network license

The network licensing agreement enables multiple users to access BricsCAD within a LAN (Local Area Network). The number of available network licenses defines the number of users who can concurrently use the software. The network license option is available for perpetual or subscription licensing and includes a 1-year maintenance contract.

1.2.3.4 Volume license

The volume licensing agreement allows you to use BricsCAD within a defined user base. You can install the software using a single license key valid for every user. The volume license option is available for perpetual or subscription licensing and includes a 1-year maintenance contract.

1.3 Installing BricsCAD®

1.3.1 Downloading BricsCAD®

1. If you have not already done so, browse to the **Bricsys website**, register and log in.
2. - Click the **Get started with BricsCAD** button.
   - Hover over **BricsCAD** in the title bar to open a drop-down menu and click the **Download** button.
3. The **Download BricsCAD® website** opens.
4. Select the platform, language, and version in the drop-down buttons below the **Download BricsCAD** button.

   Note: By default, the most recent version is selected. If you click the drop-down arrow, you can choose an older release.

5 Click the **Download BricsCAD** button once the desired version and language are displayed in the button.

1.3.2 Overview of installation options

- BricsCAD comes as a single download file for all license levels, including BricsCAD Shape.
- When you install a new BricsCAD version for the first time, you can run a 30-day trial mode.
- You can install each new major version next to a previous major version.

1.3.2.1 Interactive installation

In case of a normal (non-silent) installation, the user provides the necessary installation input via dialog boxes.

The user interface of Windows Installer queries the target system and displays Setup Wizard. Setup Wizard enables the user to change various options that affect the installation.

1.3.2.2 Silent installation

In case of a silent or quiet installation, the Setup Wizard user interface is disabled and the actions during the user interface sequence are not performed.

1.3.2.3 Interactive installation

1 Double click the installer file.

2 Click **Next**.

3 Tick the checkbox to accept the **License Agreement**.

4 Click **Next**.

5 Do one of the following:

  - Accept the default installation folder.
  - Click the **Change...** button and select an installation folder.

6 Click **Next**.

7 Click **Install** to start the installation.

8 Choose whether to add a shortcut on the desktop and/or to display the release notes when the installation process has finished.

9 Click **Next**.

10 Click **Finish** to finish the installation process.

BricsCAD is installed.

1.3.2.4 Silent installation (Windows only)

For silent installation, you need admin rights.

1 Open **PowerShell** or **Command Prompt** and change the directory (cd) to the location of the BricsCAD installer MSI file.

Note: On Windows 10 or higher, you can access the **Open PowerShell window here** option from the context menu that appears when you shift-right-click a folder in **Explorer**. This allows you to open **PowerShell** in the current window without having to change the directory.

2 Type msiexec /i and do one of the following:
   - Type the file name of the BricsCAD installer (for example, BricsCAD-V25.1.05-1-en_US(x64).msi).
   - Drag the BricsCAD installer file from the Windows Explorer dialog box to the command window.

3 Type /qn in the command prompt.

4 Type /norestart in the command prompt to suppress the restarting after the installation is complete.

5 (Optional) Add ADDDESKTOPSHORTCUT="" to suppress the addition of a desktop shortcut.

6 (Optional) Add SHOWRELEASENOTES="" to suppress the display of the Release Notes when the installation completes.

7 (Optional) Add APPLICATIONFOLDER="your_app_folder" to install BricsCAD in a custom folder.

8 (Optional) Add BRXLICENSEDESTFILE="path_to_lic_file" to specify a folder for the license file if any of the properties **BRXLICENSEKEY**, **BRXLICENSEFILE** or **BRXLICENSESERVER** are defined.

The default folder is: *C:\ProgramData\Bricsys*.

9 (Optional) Add BRXLICENSEKEY="your_license_key" to activate the license and store it in the folder specified by **BRXLICENSEDESTFILE**.

10 (Optional) Add BRXLICENSEFILE="your_license_file" to copy the license file to the folder specified by **BRXLICENSEDESTFILE**.

11 (Optional) Add BRXLICENSESERVER="host", or BRXLICENSESERVER="port@host" (*) to create a license file in the folder specified by **BRXLICENSEDESTFILE**, containing the server host specification.

(*) If your license server is using a different port than default port 5053.

12 (Optional) Add REBOOT=ReallySuppress to suppress a restart after the installation is complete.

13 (Optional on 32-BIT only) Add INSTALLVBA="0" or INSTALLVBA="" to not install VBA.

Note: The 32-BIT version is only available for BricsCAD V20 and older.

14 Press **Enter** to launch the installation.

Note:

- For more options, type: msiexec /? in the command prompt.
- You can find more information about the **msiexec** command on the Microsoft website.

Examples of command prompts:

- To perform a silent installation while suppressing the creation of a desktop shortcut:
- `msiexec /i "BricsCAD-Vxx.x.xx-x-en_US(x64).msi" /qn ADDDESKTOPSHORTCUT=""`
- To perform a silent installation while suppressing the creation of a desktop shortcut and the display of Release Notes:

- ```
  msiexec /i "BricsCAD-Vxx.x.xx-x-en_US(x64).msi" /qn ADDDESKTOPSHORTCUT=""
   SHOWRELEASENOTES=""
  ```
- To execute a silent installation and activate the license using the provided license key:
- ```
  msiexec /i "BricsCAD-Vxx.x.xx-x-en_US(x64).msi" /qn BRXLICENSEDESTFILE=
  "C:\ProgramData\Bricsys\BricsCAD.lic" BRXLICENSEKEY="****-****-****-****-****"
  ```

  The license file, **BricsCAD.lic**, is generated at the specified location, *C:\ProgramData\Bricsys*, ensuring that all BricsCAD versions automatically find it.
- To copy the existing **BricsCAD.lic** file to the specified directory, *C:\ProgramData\Bricsys*, ensuring that all BricsCAD versions automatically find the license file:

  ```
  msiexec /i "BricsCAD-Vxx.x.xx-x-en_US(x64).msi" /qn BRXLICENSEFILE=
  "D:\BricsCAD.lic" BRXLICENSEDESTFILE="C:\PROGRAMDATA\bricsys\BricsCAD.lic"
  ```
- To create the license file containing the server host specification in the assigned folder, *C:\ProgramData\Bricsys*, ensuring that all BricsCAD versions automatically find it:

  ```
  msiexec /i "BricsCAD-Vxx.x.xx-x-en_US(x64).msi" /qn
   BRXLICENSESERVER="port@host"
  BRXLICENSEDESTFILE="C:\PROGRAMDATA\bricsys\BricsCAD.lic"
  ```

1.3.3 Installation process of Linux RPM package on KDE desktop environment

To install BricsCAD on Fedora 35, KDE Plasma 5.23.3:

- Open Dolphin (KDE's file manager).
- Navigate to the folder where the downloaded installer is present.
- Right-click in Dolphin and select **Open Terminal** (it will open for the currently displayed folder).
  - Input:

    ```
    sudo dnf install <rpm installer name>
    ```
  - Press **Enter**
  - Input password and press **Enter**
  - Press **y**
  - Press **Enter**

    The installation completes.

To install BricsCAD on openSUSE Tumbleweed 20211202, KDE Plasma 5.23.4:

- Open Dolphin (KDE's file manager).
- Navigate to the folder where the downloaded installer is present.
- Right click in Dolphin, select **Actions** and **Open Terminal** (it will open for the currently displayed folder).
  - Input:

    ```
    sudo zypper install <rpm installer name>
    ```

- Press **Enter**
- Input password and press **Enter**
- Press **y**
- Press **Enter**
- Press **i** (to ignore Signature verification)
- Press **Enter**

  The installation completes.

To install BricsCAD on Fedora 35, Gnome 41.1:

- Double click the rpm installer in the default file manager.
- Software Install is launched.
- Select **Install**.
- Input password and press **Enter**.

  The installation completes.

1.4 Trial mode

BricsCAD can run in trial mode for 30 days. After this period, you need to activate a valid commercial license.

1.4.1 Running BricsCAD in trial mode

Each time you start BricsCAD in trial mode, the **BricsCAD Launcher** opens, indicating the days left.

Do one of the following:

- Click a workspace to continue using BricsCAD in trial mode.
- Click the **Activate** button to open the **License Manager** dialog box, which allows you to activate a Single or Volume license or to use a Network license.

1.4.2 Setting the license level

When running BricsCAD in trial mode, all features are available. It runs as a BricsCAD Ultimate license.

The SETLICENSELEVEL command allows you to select a lower license level.

1.4.3 Online activation (Trial)

The BricsCAD trial license activation requires an internet connection.

The first time you open a new major version of BricsCAD, it runs in trial mode.

Note: If no trial license is found, the **No active license**

message appears in the **BricsCAD Launcher** dialog box. In such cases:

1. Click the **Activate** button in the **BricsCAD Launcher** dialog box.
2. Click **Activate Trial** in the **License Manager** > **Activate BricsCAD** dialog box.

   The trial license is activated over the internet.

1.4.4 Offline/manual activation (Trial)

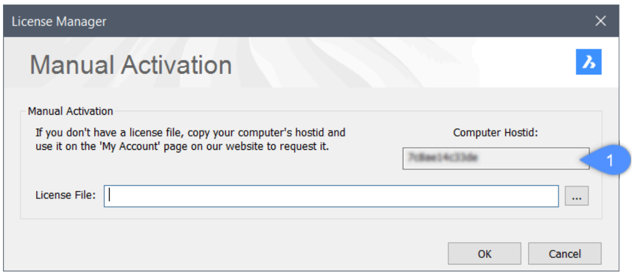
If your computer is not connected to the internet, you can activate the license manually via another computer that is connected to the internet.

1 Launch BricsCAD.

2 Click the **Activate** button in the **BricsCAD Launcher** dialog box.

3 Click **Activate manually...** in the **License Manager** > **Activate BricsCAD** dialog box.

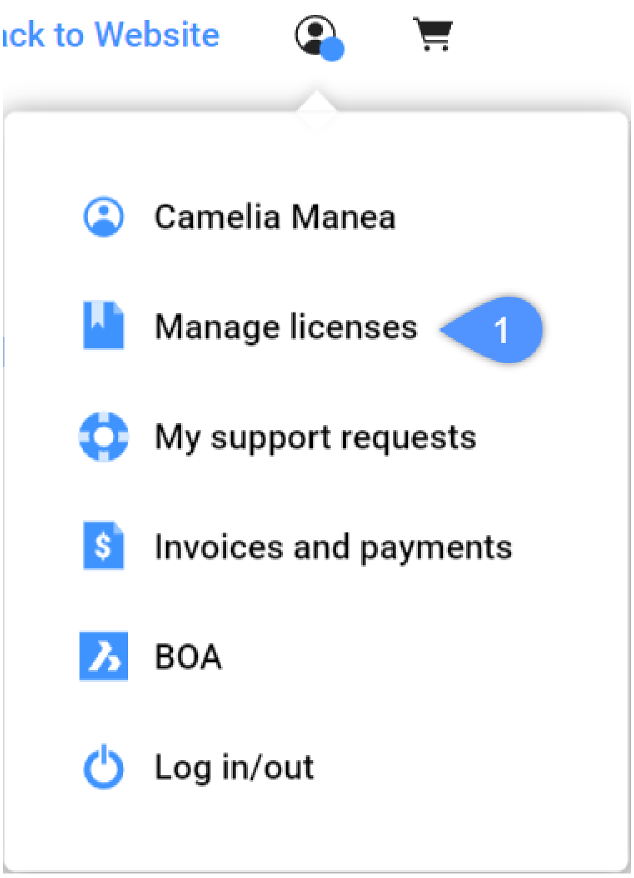
The **License Manager** > **Manual Activation** dialog box opens.



4 Copy the full content of the **Computer Hostid** (1) field.

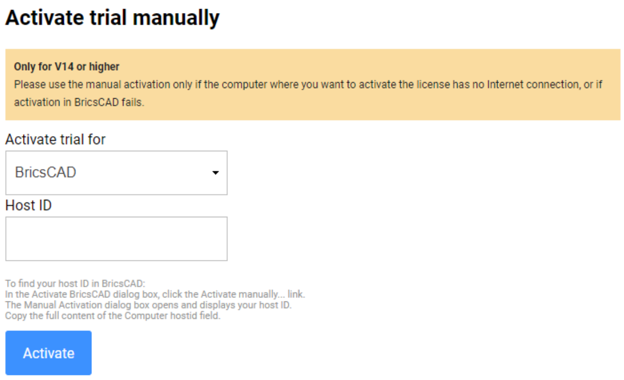
5 Log in to the Bricsys website with the computer connected to internet.

6 Click your user name and select **Manage licenses** (2) in the drop-down menu.



7 Choose **Activate BricsCAD Trial manually**.

8 In the **Host ID** field, paste the computer host ID of the computer you want to install BricsCAD on and click the **Activate** button.



A license file is generated and made available for download.

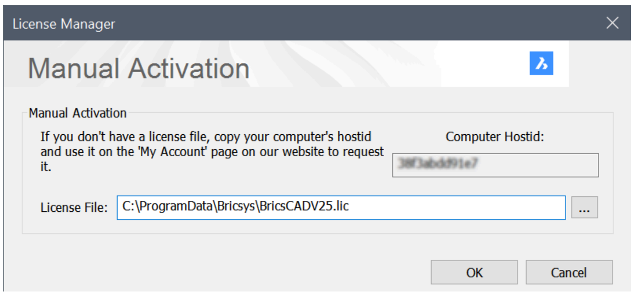
9 Click the **Download License File** button.

The license file is downloaded to your default download folder.

10 In the **License Manager** > **Manual Activation** dialog box, click the **Browse** button (...).

11 Open the folder where you copied the license file to and select the file.

12



13 Click the **OK** button.

BricsCAD is launched.

1.5 Working with Single/Volume licenses

1.5.1 Online activation (Single/Volume license)

The BricsCAD license activation requires an internet connection.

1 Launch BricsCAD.

Note: When your trial period has expired, click the **Activate** button in the **BricsCAD Launcher** dialog box, then click the **Activate Now...** button in the **License Manager** > **Activate BricsCAD** dialog box, and proceed to step 5.

2 Launch the LICENSEMANAGER command.

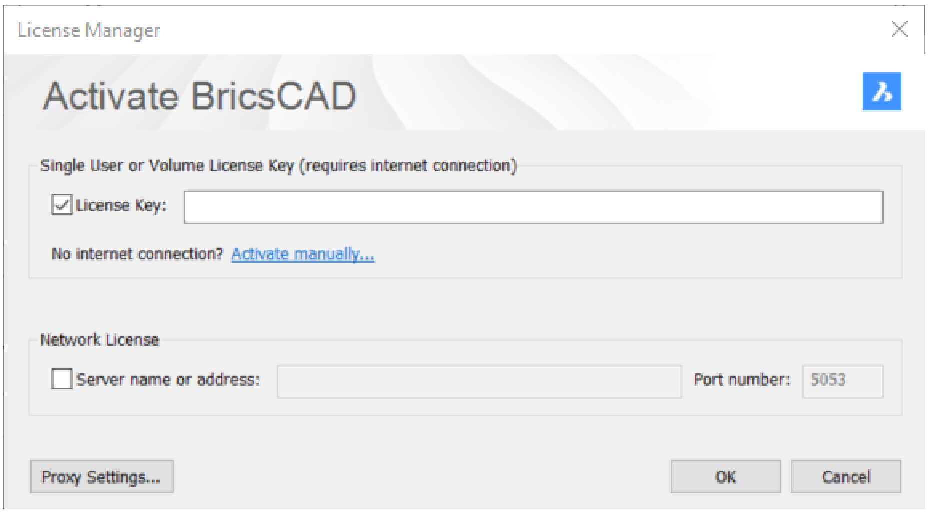
3 The **Bricsys License Manager** dialog displays.

4. Click the **Manage License** button.

   The **License Manager** > **Licensing Information** dialog displays.

5. Click the **Modify** button.

   The **License Manager** > **Activate BricsCAD** dialog box displays.



6. Retrieve your key from the **My Licenses** section in your account. See also the **Where can I find my licenses?** article.
7. Paste your license key in the **License Key** field.
8. Click the **OK** button.
   - Your license key is registered over the internet.
   - A dialog displays when the license key is successfully activated.
9. Read the text on the dialog box and click the **I understand** button.

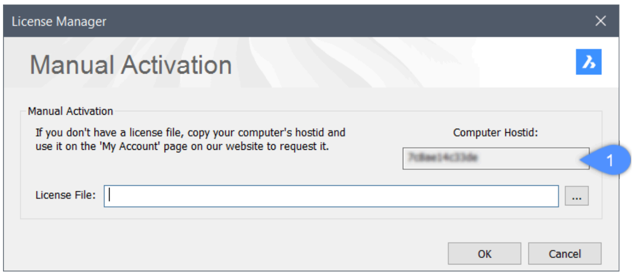
1.5.2 Offline/manual activation (Single/Volume license)

If your computer is not connected to the internet, you can activate the license manually via another computer that is connected to the internet.

Note: You cannot deactivate manually activated licenses.

1. Launch BricsCAD.
2. Do one of the following:
3. - If you start BricsCAD for the first time, click **Activate** in the **BricsCAD Launcher** dialog box.
   - If you replace or upgrade an existing license, launch the LICENSEMANAGER command, click **Manage License** and click **Modify...**.
4. Click **Activate manually...** in the **License Manager** > **Activate BricsCAD** dialog box.

   The **License Manager** > **Manual Activation** dialog box opens.

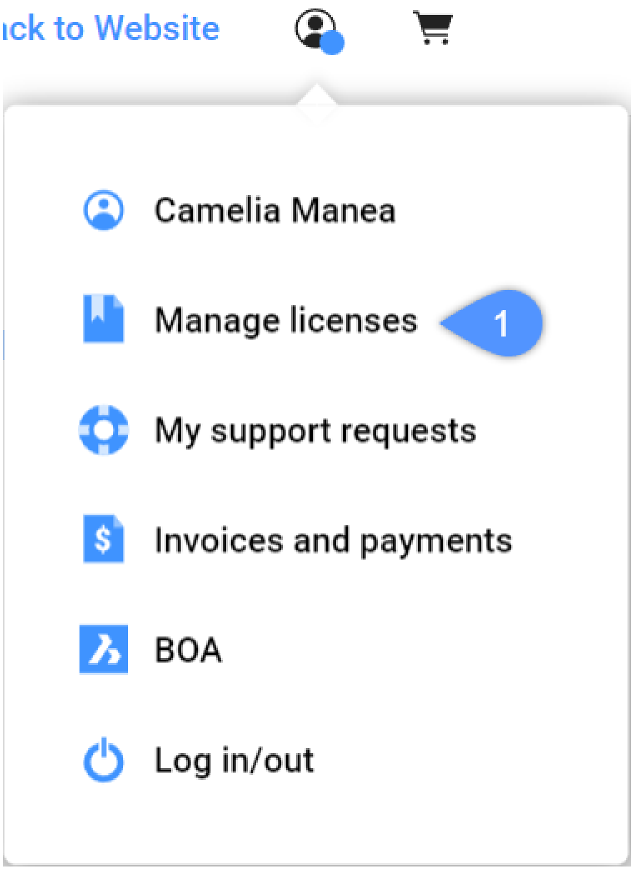


5 Copy the full content of the **Computer Hostid** (1) field.

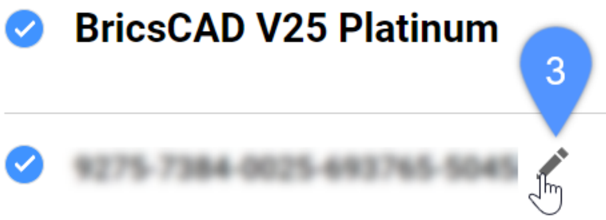
6 Log in to the Bricsys website with the computer connected to internet.

7 Make sure you log in with the Bricsys account which contains your licenses.

8 Click your user name and select **Manage licenses** (2) in the drop-down menu.



9 Hover over the license key you want to activate and click the pencil icon (✎) (3).



The **License details** page displays.

10 Click **Manual activation**.

11 In the **Host ID** field, paste the host ID of the computer you want to install BricsCAD on.

12 Click the **Activate** button.

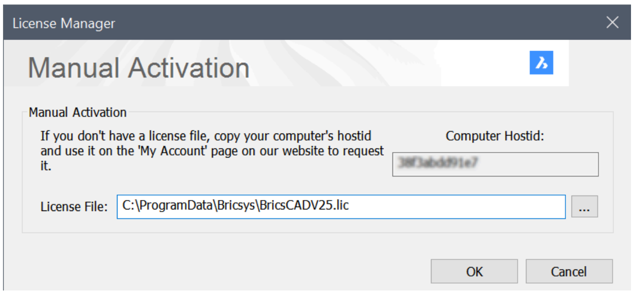
A license file is generated and made available for download.

13 Click the **Download License File** button.

The license file **BricsCADVxx.lic** is downloaded to your default download folder.

14 Copy the license file to the computer you want to install BricsCAD on.

15 In the **License Manager** > **Manual Activation** dialog box, click the browse button (...). Open the folder where you copied the **BricsCADVxx.lic** license file to and select the file.



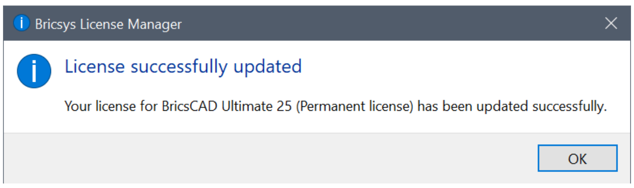
16 Click the **OK** button.

BricsCAD is launched.

1.5.3 Upgrading to the next BricsCAD version when having active maintenance (Single/ Volume license)

When a new major BricsCAD version is released, all licenses covered by an active maintenance contract are updated to run the newly released version.

After installation of BricsCAD *V25*, upon launch, you will see a pop-up saying that your license was updated to *V25*.



This shows after launching BricsCAD, before the BricsCAD Launcher dialog box appears.

Important:

- Only the V24 keys are automatically updated to V25 when covered by an active maintenance contract. If you have never activated your V24 license, you can retrieve your key from the **My Licenses** section in your account. Next, go to the **License Manager** dialog box, select **Activate Now** and enter your key.

  See more information in the **Online activation (Single/Volume license)** article.
- If the license was activated offline, you need to revoke it first. See the **Revoke a BricsCAD license (limited operation)** article for more information. If no more revokes are available, you have to contact support.

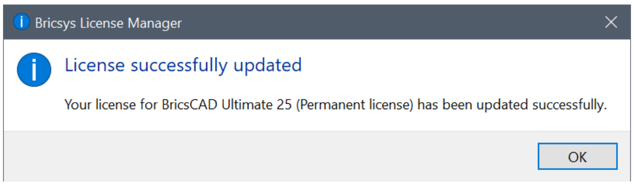
  Remember: Only 2 revokes are possible. For online activated licenses, deactivation and reactivation is unlimited.
- Communicator for BricsCAD® licenses are not automatically updated. You need to enter the key in the **License Manager** dialog box. See also the **Communicator for BricsCAD - Getting started** article.

If the update of your license was not automatically detected, retrieve your key from the **My Licenses** section in your account. Next, go to the **License Manager** dialog box, select **Activate Now** and enter your key.

1.5.4 Upgrading BricsCAD or extending your subscription (Single/Volume license)

When you have purchased a version upgrade for BricsCAD and you are connected to the internet

After installation of the new version, upon launch, a pop-up displays saying that your license was updated to the newer version.



This shows after launching BricsCAD, before the **BricsCAD Launcher** dialog box appears.

When you have purchased a product upgrade or have extended your subscription license

Retrieve your key from the **My licenses** section in your account. Next, go to the **License Manager** dialog box, select the **Activate Now...** button and enter your key.

Note: If the last version of BricsCAD installed was V24.2 or later, you will not need to retrieve your key first. BricsCAD will already prefill it in the **License Manager**.

When you can't connect to the internet

If the license was activated offline, you need to revoke it first. See the **Revoke a BricsCAD license (limited operation)** article for more information. If no more revokes are available, you have to contact support.

Remember: Only 2 revokes are possible. For online activated licenses, deactivation and reactivation is unlimited.

Next, follow the process described in the **Offline/manual activation (Single/Volume license)** article.

1.5.5 Migrating a Single or Volume license to another computer

All your valid licenses can be found on the Bricsys website (login required):

https://www.bricsys.com/protected/account/licenses.

1. Deactivate the license on the old computer.
2. Launch BricsCAD on the new computer.
3. Activate the license on the new computer.

1.5.6 Deactivate a BricsCAD license

Make sure you deactivate a license before:

- Migrating a license to another computer.
- Permanently removing BricsCAD from a computer.
- Installing a new hard disk.
- Restructuring the disk partitions.
- Upgrading the operating system.

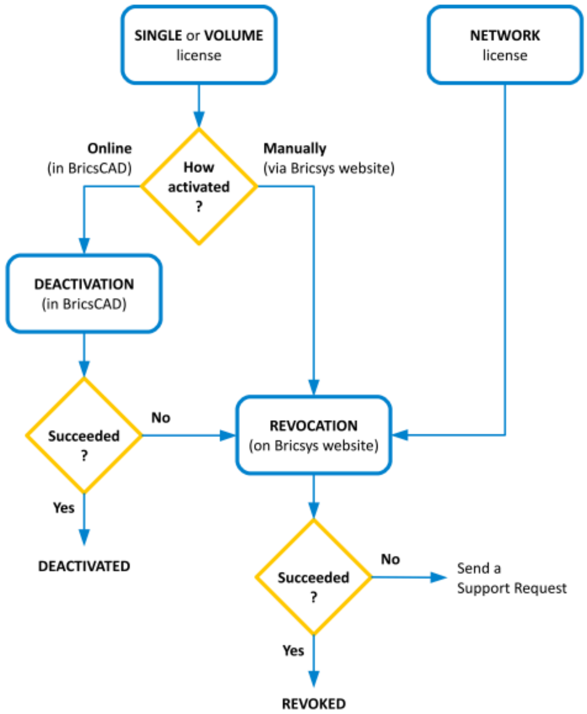
To deactivate a BricsCAD license, an internet connection is required. You cannot manually deactivate a license via another computer that is connected to the internet.

Note: You cannot deactivate:

- Single and volume licenses that are manually activated.
- Network licenses.
- Trial licenses.

The following chart explains how to deactivate a BricsCAD license. If it is not possible to deactivate a BricsCAD license, you can revoke the license.

Important: The revoke possibility is limited to two per key and the action cannot be undone. Consult also the **Revoke a BricsCAD license (limited operation)** article.



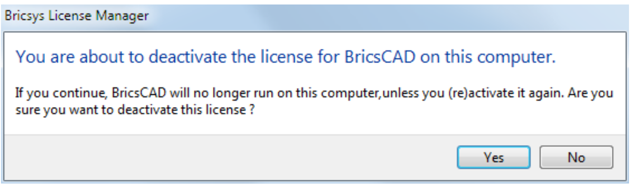
To deactivate a license:

1 Launch the LICENSEMANAGER command. Click the **Manage License** option in the **Bricsys License Manager** dialog box.

The **License Manager** > **Licensing information** dialog box displays.

2 If your computer connects to the license server through a proxy server, click **Proxy Settings...** to configure the proxy settings.

3 Click **Deactivate...**



4 Click the **Yes** button in the confirmation dialog box to deactivate the license.

The license is deactivated and can be activated on another computer.

Note:

- The number of times you can deactivate is unlimited. To move a license, always use the deactivate and reactivate options.
- When you uninstall BricsCAD, the **Bricsys License Manager** dialog box appears, giving you the option to deactivate the current local license or keep it active.

Tip: The deactivation of BricsCAD licenses can be automated with the LicTool application. See more information in the **How to automate activation and deactivation of local BricsCAD licenses using the LicTool application** article.

1.5.7 Revoke a BricsCAD license (limited operation)

Warning:

- Only revoke a license if deactivation is not possible.
- Support must be contacted first before revoking a license.
- *The revoke possibility is limited to two per key.*
- *There is no return possible if you revoke a license.*

Use this procedure to:

- Migrate a network license to a new server.
- Revoke a single or volume license that was manually activated.
- Reactivate BricsCAD after a hard disk crash.

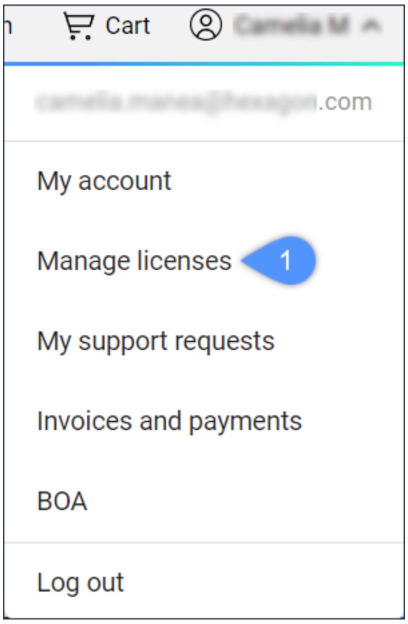
Important: When in doubt, https://boa.bricsys.com/protected/support/NewSupportRequest.do?categoryID=198 in the **BricsCAD** > **Licensing** > **Revoke license** category. See also the **How to submit a support request** article.

To revoke a license:

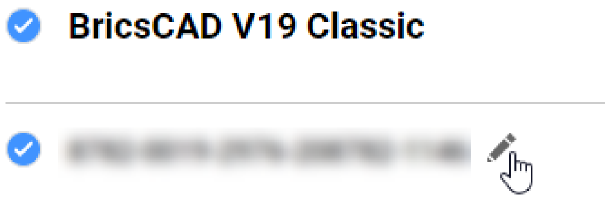
1 Log in to the Bricsys website.

1 Make sure you log in with the Bricsys account containing your licenses.

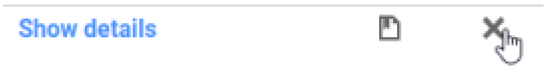
2 Click the **My Account** icon ( ) and select **Manage licenses** (1) in the menu.



3 Hover over the license key of the license you want to revoke and click on the pencil icon ( ).



4 Scroll down and click the **X** icon to revoke the license.



Note: If you are a Maintenance user, contact support to review your additional revoke request.

1.6 Working with Network licenses

1.6.1 About Network licenses

A network license allows multiple users on the same TCP/IP network to share access to product licenses. The Bricsys Network License Server manages how licenses are distributed to users.

When you launch BricsCAD, it requests a license from the license server through the network. If a license is available, the Bricsys Network License Server assigns a license to the computer, and the number of available licenses on the license server is reduced by one.

When you close BricsCAD, the license returns to the pool.

To make a network license available in BricsCAD on a client computer, the network administrator needs to install the Bricsys Network License Server, which contains the Bricsys Network License Manager, powered by a recent version of RLM framework (V15.2).

Overview and control over the license usage can be done via web-interface, while license activation and license update can be done via **Bricsys Network License Manager** dialog box.

The Bricsys Network License Server needs a license file that you can activate online or manually. The network license can only be activated once, and it will be locked to the machine.

Once the Bricsys Network License Server is set up and has an active license, it can distribute that license to the client computers that have BricsCAD installed. The number of computers that can use a license simultaneously is limited by the number of seats available on the network license.

1.6.1.1 System Requirements

The Bricsys Network License Server is available for Windows and Linux. It is not available for macOS. When you have the Bricsys Network License Server running on Windows or Linux, it can distribute licenses to client computers running Windows/Linux/macOS.

Supported platforms

Platform	Version
Windows	10*
Windows	11
Windows Server	2019
Windows Server	2022
Ubuntu Linux	22.04 LTS and higher**
openSUSE Linux	later than April 2022
Fedora Linux	later than April 2022

*Windows 10 Operating System will reach end-of-support on 14 October 2025.

**Ubuntu 22.04 LTS and newer Ubuntu versions that are still supported by Canonical (for a more stable experience, LTS is recommended).

1.6.2 Quick guide on using a Network license

In general, Bricsys Network License Server downloading, deployment and configuration is a duty for the network/system administrator.

1.6.2.1 Server machine actions

1 Download the Bricsys Network License Server from [here]() and install it with default settings.

Info related to server security policies:

After installation, the RLM.exe must be allowed to use 3 ports: **5053** for the license, **5054** for the web UI, and a **dynamically allocated port** by Windows for the ISV server which can be set fixed after activation as described in the **Advanced configurations** article.

2 When the **Activate New Network License** dialog box is displayed, use the license key from your Bricsys account. See also the **Where can I find my licenses?** article.

Info related to server security policies:

The Network License Manager tool is in the *C:\Program Files\Bricsys\Bricsys Network License Server \NetworkLicenseManager.exe* folder and must be able to reach the **license.bricsys.com/actpro** port **80** and **https://api.bricsys.com/** port **443**.

3 The **Reprise License Server Administration** page is on **localhost:5054**.

Note: Make sure to change the default credentials of the RLM login (username: *admin* and password: *admin*). See also the **Setup RLM login** article.

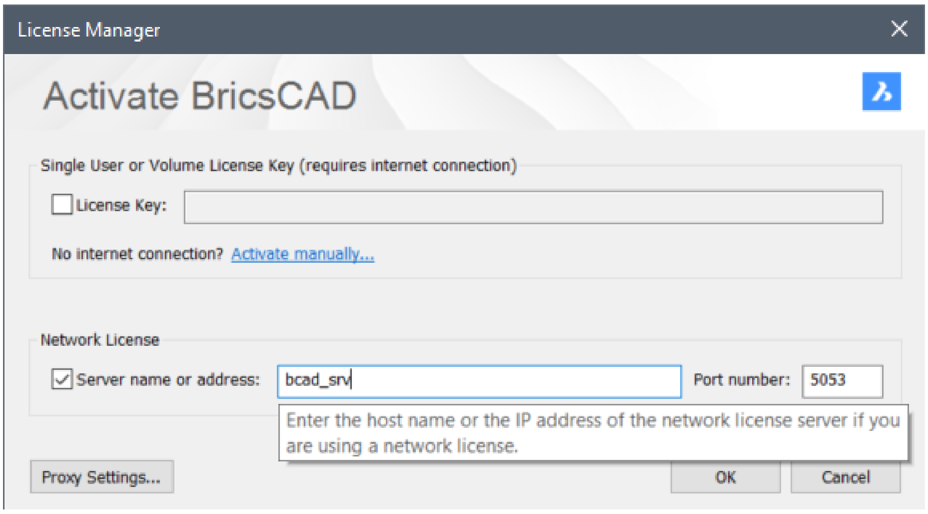
1.6.2.2 Client machine actions

The client machine must be:

- In the same **LAN** with the Bricsys Network License Server, physically or by VPN connection.
- Able to communicate with the Bricsys Network License Server on port **5053** by the **TCP/UDP** protocol.

1. Install BricsCAD with default settings.
2. Start BricsCAD and activate the Network License: <servername>/<ip number> and <port>.

   There may be no need for this as BricsCAD will check the network for a license server at the start.



1.6.3 Setting up a Network License Server

1.6.3.1 Setting up a Windows Network License Server

In general, Bricsys Network License Server downloading, deployment and configuration is a duty for the network/system administrator.

1. **Downloading Bricsys Network License Server**

Download the Bricsys Network License Server (Windows) from here.

Important:

- Starting with RLM V15.1, a default user with the username *admin* and password *admin* is created if rlm.pw does not already exist in the Bricsys Network License Server folder.
- If the *admin* password is still the default and/or there are any users without passwords for longer than 10 minutes, the server will shut down automatically.

  See also the **Setup RLM login** article.

Note:

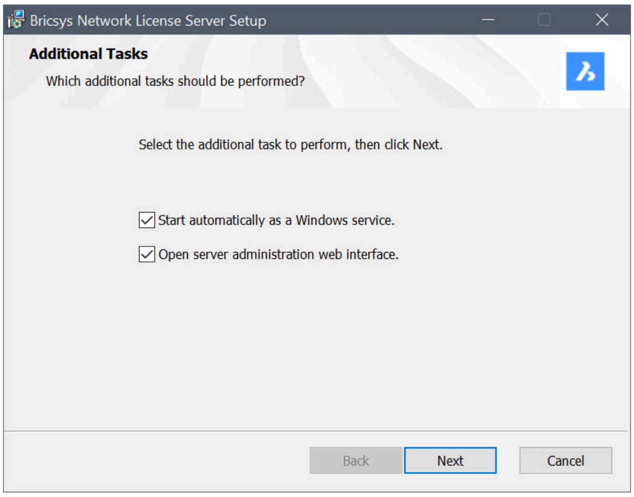
- The Bricsys Network License Server contains the RLM server and the Bricsys Network License Manager.

- The Bricsys Network License Server is powered by a recent version of RLM framework (V15.2). See more information in the Bricsys Network License Server - Release Notes.

Remember: Bricsys Network License Manager does not support updating licenses that are manually merged into one complex file, it supports the default setup: one Bricsys software license per file.

2 New installation of Bricsys Network License Server

In the **Bricsys Network License Server Setup** dialog box, make sure to tick all the checkboxes.



Start automatically as a Windows service

After the installation, the **Bricsys Network License Manager** dialog box opens. Click the **Add License** button to open the **Activate New Network License** dialog box and enter your network license to activate it. See more information the **Online activation (Network license)** article.

You can later open the **Bricsys Network License Manager** dialog box using the **Bricsys Network License Manager** desktop shortcut.

Note: If no active license is detected on your system, the **Activate New Network License** dialog box will show first.

Each time you start Windows, the license server is automatically started as a service. You can do this later with the following command in the elevated command prompt: rlm.exe -dlog rlm.log -install_service.

Open server administration web interface

After the installation, the Reprise License Server Administration web interface opens. It allows you to manage the network license. You can open the Reprise License Server Administration later by clicking the **License Server Administration** shortcut in the Start menu.

3 Upgrading Bricsys Network License Server

Download and install the Bricsys Network License Server on the computer used as Network License Server.

Note: The installation restarts the license server.

4 Installation from the Command line

You can also start the Bricsys Network License Server from a command window.

Install the Bricsys Network License Server using a terminal, such as Command Prompt or PowerShell:

- Type: msiexec /i "<FullPath>\Bricsys-NetworkLicenseServer-V1.0.1-1.msi"

  Note: Replace **<FullPath>** with the path where Bricsys-NetworkLicenseServer-V1.0.1-1.msi was downloaded.

By default, the options **Start automatically as a Windows service** and **Open server administration web interface** are disabled when installing via the Command line. To enable these options, use the respective parameters INSTALLSERVICE and OPENWEBINTERFACE.

For example, msiexec /qb /i "<FullPath>\Bricsys-NetworkLicenseServer-V1.0.1-1.msi" INSTALLSERVICE=1 installs as a service without showing the web interface.

Note:

- For more options, type **msiexec /?** at the command prompt.
- On the Microsoft website, you can find more information about the **msiexec** command.

1.6.3.2 Setting up a Linux License Server

In general, Bricsys Network License Server downloading, deployment and configuration is a duty for the network/system administrator.

Download and install the Bricsys Network License Server on the computer that will be used as Network License Server.

1 Download Bricsys Network License Server (Linux) from here.

  Note:

  - The Bricsys Network License Server contains the RLM server and Bricsys Network License Manager.
  - The Bricsys Network License Server is powered by a recent version of RLM framework (V15.2). See more information in the Bricsys Network License Server - Release Notes.

  Remember: Bricsys Network License Manager does not support updating licenses that are manually merged into one complex file, it supports the default setup: one Bricsys software license per file.

2 Extract **Bricsys-NetworkLicenseServer-V1.0.1-1.tgz** to any folder (Command line: tar xvzf Bricsys-NetworkLicenseServer-V1.0.1-1.tgz).

3 Go to that folder and start the Reprise License Manager (RLM) application (Command line: ./rlm).

  Note: If the ./rlm execution fails because of permissions, try and run sudo chmod u+x rlm to give execution permissions to the RLM file.

4 Run the launcher (Command line: ./NetworkLicenseManager.sh). Proceed and activate the key in the **Activate New Network License** dialog box that opens. If there are already some existing licenses activated on the server, use the **Add License** button on the top left of the **Bricsys Network License Manager** dialog box to activate the new key.

  Troubleshooting

  If you encounter an error on Ubuntu 22.04 when starting the Bricsys Network License Manager, install the extra package **libxcb-cursor0** (Command line: sudo apt install libxcb-cursor0).

Even if after installing the library the Bricsys Network License Manager launcher fails to start, follow this procedure:

a Open a browser at http://localhost:5054. Authentication is needed. A default user with the username admin and password admin is created in rlm.pw file under installation folder. Use those to login. You can then use **Change Password** in the left menu to change the password. See also the **Setup RLM login** article.

b On the Reprise License Server Administration web page, click **System Info** on the left. Copy the ethernet hostid (first if there are more) and use this to activate your license on your **My Account** > **Manage Licenses** page on the Bricsys website.

c When you download a license file from the Bricsys website, store it in the folder where you extracted the Bricsys Network License Manager archive.

d Click **Reread/Restart Servers** on the Reprise License Server Administration web page.

e Click **Status** to see the server status, on the next page click **Bricsys** in the table below to view Bricsys licenses.

Click **RLM Manual** to learn how to configure the license server in detail. RLM does not start automatically on a system boot. You need to start the Reprise License Manager (RLM) application manually. Among other things, it is explained on that page how to automatically start rlm when the server boots.

5 Once the Network license is activated and the server is started, the server is ready to serve licenses. It is now possible to use the Network license in BricsCAD on a client computer.

To open a port on Linux server:

1 Check the current active zone(s) on your system:

1
```
sudo firewall-cmd --get-active-zones
```

You will get output like "public" or "work", depending on your system's network configuration.

2 Add the port to the desired zone (replace **<PORT>** with the actual port number and **<ZONE>** with the active zone)

```
sudo firewall-cmd --zone=<ZONE> --add-port=<PORT>/tcp --permanent
```

Note: Replace "tcp" with "udp" if you want to open a UDP port.

3 Reload the firewall to apply the changes:

4
```
sudo firewall-cmd --reload
```

5 Verify that the port is now open:

6
```
sudo firewall-cmd --zone=<ZONE> --list-ports
```

The port is now opened in the specified zone, and you should be able to use services that utilize that port on your CentOS 9 system. Make sure to replace **<PORT>** and **<ZONE>** with the actual port number and zone name you want to use.

Tip: In case you need to update floating licenses that were already hosted on the server, you need to run the Bricsys Network License Manager by calling its launcher from the installation folder: *./NetworkLicenseManager.sh*

1.6.3.3 Online activation (Network license)

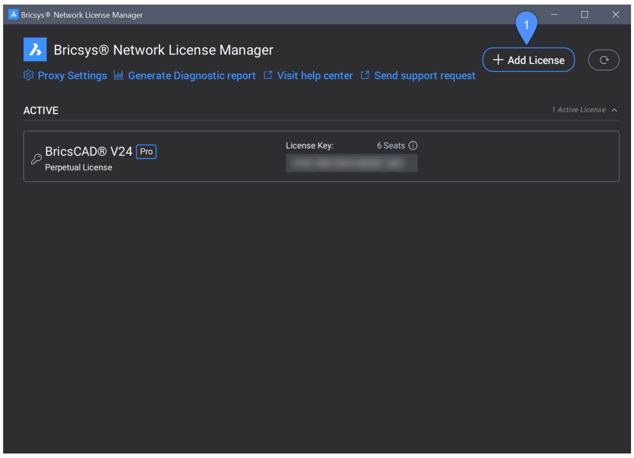
The BricsCAD license activation requires an internet connection at the moment of activation.

1 You can either:
   - Click the **Bricsys Network License Server** desktop shortcut.
   - Click the **Bricsys Network License Server** shortcut in the operating system Start menu.

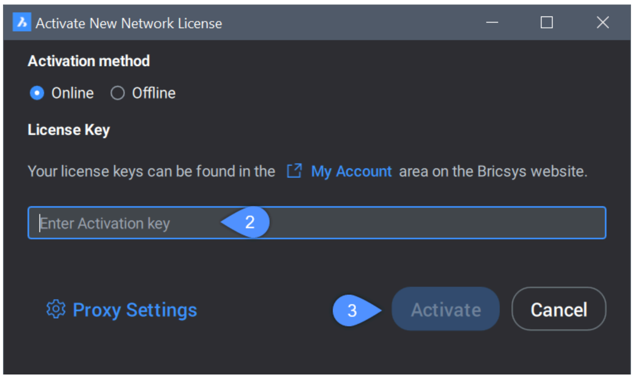
     C:\Program Files\Bricsys\Bricsys Network License Server\NetworkLicenseManager.exe

   The **Bricsys Network License Manager** dialog box opens.

2 Click the **Add License** button (1) to open the **Activate New Network License** dialog box.



Enter the new network license in the **Enter Activation key** field (2) and click **Activate** (3). Use the license key from your Bricsys account. See also the **Where can I find my licenses?** article.



Note: If no active license is detected on your system, the **Activate New Network License** dialog box will show first.

1.6.3.4 Offline/manual activation (Network license)

If your computer is not connected to the internet, you can activate the license manually via another computer that is connected to the internet. If you can have an internet connection available, even only during activation, we recommend using the online activation process.

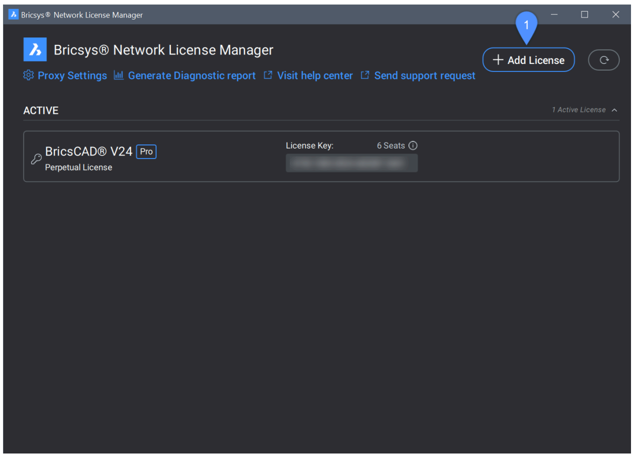
1 You can either:
   - Click the **Bricsys Network License Server** desktop shortcut.

- Click the **Bricsys Network License Server** shortcut in the operating system Start menu.

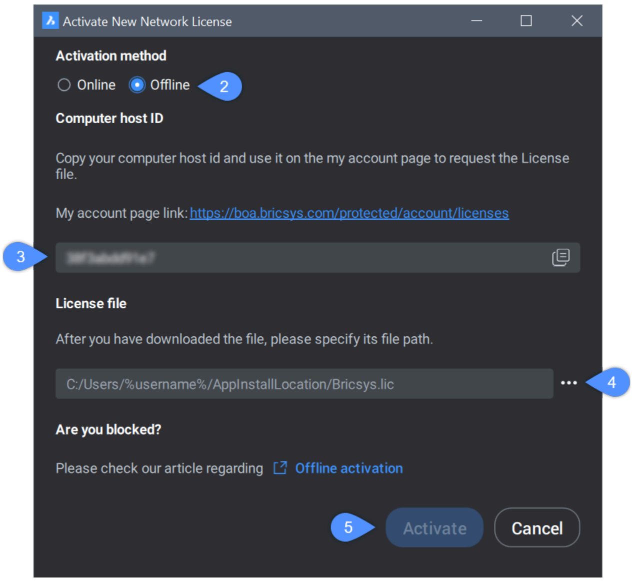
  C:\Program Files\Bricsys\Bricsys Network License Server\NetworkLicenseManager.exe

The **Bricsys Network License Manager** dialog box opens.

2 Click the **Add License** button (1) to open the **Activate New Network License** dialog box.



Select the **Offline** option (2) in the **Activate New Network License** dialog box and copy the full content of the **Computer Hostid** field (3).



Note: If no active license is detected on your system, the **Activate New Network License** dialog box will show first.

3 Log in to the Bricsys website with a computer connected to internet.

4 Click your name in the upper right corner and select **Manage licenses** from the menu.

Make sure you log in with the Bricsys account of your company which contains the Network license. Contact your Bricsys account manager if needed.

5 Hover over the license key you want to activate and click the pencil icon ( ).

The **License details** page displays.

6 Click **Manual activation**.

7 In the **Host ID** field, paste the computer host ID of the computer you want to install Bricsys Network License Server on.

8 Click the **Activate** button.

A license file is generated and made available for download.

9 Click the **Download License File** button.

The license file **BricsCAD.lic** is downloaded to your default download folder.

10 Go back to the **Activate New Network License** dialog box and click the browse button (4) to open the **Select BricsCAD license** dialog box. Select the previously downloaded LIC file and click **Open**.

11 Click the **Activate** button (5) to activate the license offline.

1.6.4 Upgrading a Network license

1.6.4.1 Upgrading to the next BricsCAD version when having active maintenance (Network license)

Install the new Bricsys Network License Server, which contains the Bricsys Network License Manager. In general, Bricsys Network License Server downloading, deployment and configuration is a duty for the network/system administrator. See the **Setting up a Windows Network License Server** article for more information.

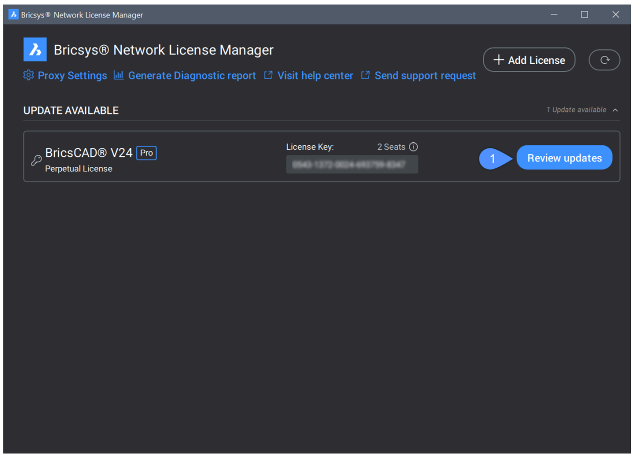
Note:

- The new Bricsys Network License Manager uninstalls the old one automatically and all the files from the old **Bricsys Network License Manager** folder are moved to the **Bricsys Network License Server** folder.

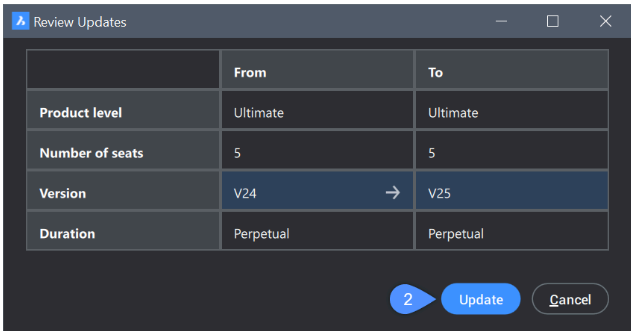
  Important:

  - Starting with RLM V15.1, a default user with the username *admin* and password *admin* is created if rlm.pw does not already exist in the Bricsys Network License Server folder.
  - If the *admin* password is still the default and/or there are any users without passwords for longer than 10 minutes, the server will shut down automatically.
  - If you created a rlm.pw file with RLM V14.2 or earlier, that PW file is no longer valid. To fix this, you must delete the old PW file and create a new one.
  - See the **Setup RLM login** article for more information.
- The active licenses are kept and displayed in the new **Bricsys Network License Manager** dialog box.

All licenses under maintenance, where you are entitled to a version upgrade, are shown under the **Update available** category.



Click the **Review updates** button (1) to open the **Review Updates** dialog box to have an overview of which updates are available for that license.



Click the **Update** button (2) to update the license on your Network License Server.

Note: If the **Bricsys Network License Manager** dialog box was already open, you might need to click the refresh button ( ) to have the **Review updates** button available.

If the update of a license is not automatically detected, retrieve the license key in the **My licenses** section under your account. See the **Where can I find my licenses?** article for more information. Next, enter the key in the **Bricsys Network License Manager** dialog box manually using the **Add a license** button.

For licenses that were offline activated:

- If the license was activated offline, but the Network License Server has an internet connection, the update of the license will be detected automatically. The update process will be as described above.
- If the Network License Server still does not have an internet connection:
    - a You need to revoke the license first. See the **Deactivate a BricsCAD license** article for more information. If no more revokes are available, you need to contact support.

      Remember: Only 2 revokes are possible per license.
    - b Retrieve the license key in the **My licenses** section under your account. See the **Where can I find my licenses?** article for more information.
    - c Activate the license using the offline process. See also the **Offline/manual activation (Network license)** article.

Important: Make sure you permanently delete all copies (including copies of older licenses files) of the BricsCAD Network license files that are currently on your companies' servers (including those of your employees, contractors, etc.) in excess of those that you are entitled to. This means that you need to ensure that even copies of old Network licenses which have been upgraded to newer versions are deleted.

Note: Communicator for BricsCAD® licenses behave the same as BricsCAD licenses.

1 Benefits of the new Bricsys Network License Server

The new Bricsys Network License Server brings the following benefits:

- It detects licenses that are covered by an active maintenance contract and provides the option to upgrade them on the **Bricsys Network License Manager** dialog box.
- It has an improved User Interface that provides a good overview of all licenses deployed on a server.
- It guides users to useful resources, such as Bricsys Help Center, Bricsys Support, RLM diagnostic tools, etc.

1.6.4.2 Upgrading BricsCAD or extending your subscription (Network license)

Install the new Bricsys Network License Server, which contains the Bricsys Network License Manager.
In general, Bricsys Network License Server downloading, deployment and configuration is a duty for the network/system administrator. See the **Setting up a Windows Network License Server** article for more information.

Note:

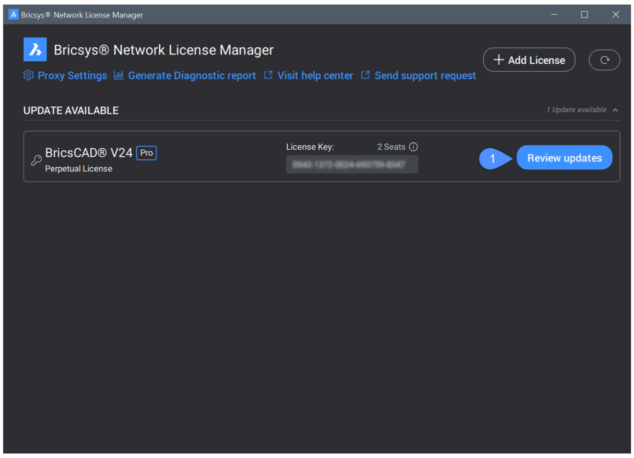
- The new Bricsys Network License Manager uninstalls the old one automatically and all the files from the old **Bricsys Network License Manager** folder are moved to the **Bricsys Network License Server** folder.

  Important:

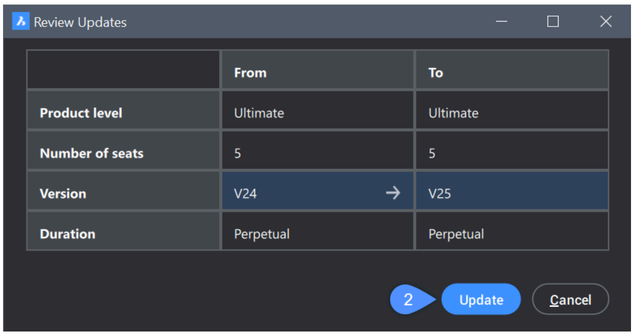
  - Starting with RLM V15.1, a default user with the username *admin* and password *admin* is created if rlm.pw does not already exist in the Bricsys Network License Server folder.
  - If the *admin* password is still the default and/or there are any users without passwords for longer than 10 minutes, the server will shut down automatically.
  - If you created a rlm.pw file with RLM V14.2 or earlier, that PW file is no longer valid. To fix this, you must delete the old PW file and create a new one.
  - See the **Setup RLM login** article for more information.
- The active licenses are kept and displayed in the new **Bricsys Network License Manager** dialog box.

When you have purchased a version upgrade, product upgrade, seats upgrade or have extended your subscription license for Network license, and you are connected to the internet

For network licenses, all updates are detected automatically and are shown under the **Update available** category.



Click the **Review updates** button (1) to open the **Review Updates** dialog box to have an overview of which updates are available for that license.



Click the **Update** button (2) to update the license on your Network License Server.

Note: If the **Bricsys Network License Manager** dialog box was already open, you might need to click the refresh button (⟳) to have the **Review updates** button available.

If the update of a license is not automatically detected, retrieve the license key in the **My licenses** section under your account. See the **Where can I find my licenses?** article for more information. Next, enter the key in the **Bricsys Network License Manager** dialog box manually using the **Add a license** button.

For licenses that were offline activated:

- If the license was activated offline, but the Network License Server has an internet connection, the update of the license will be detected automatically. The update process will be as described above.
- If the Network License Server still does not have an internet connection:
  a. You need to revoke the license first. See the **Deactivate a BricsCAD license** article for more information. If no more revokes are available, you need to contact support.

     Remember: Only 2 revokes are possible per license.
  b. Retrieve the license key in the **My licenses** section under your account. See the **Where can I find my licenses?** article for more information.
  c. Activate the license using the offline process. See also the **Offline/manual activation (Network license)** article.

Important: Make sure you permanently delete all copies (including copies of older licenses files) of the BricsCAD Network license files that are currently on your companies' servers (including those of your employees, contractors, etc.) in excess of those that you are entitled to. This means that you need to ensure that even copies of old Network licenses which have been upgraded to newer versions are deleted.

1 Benefits of the new Bricsys Network License Server

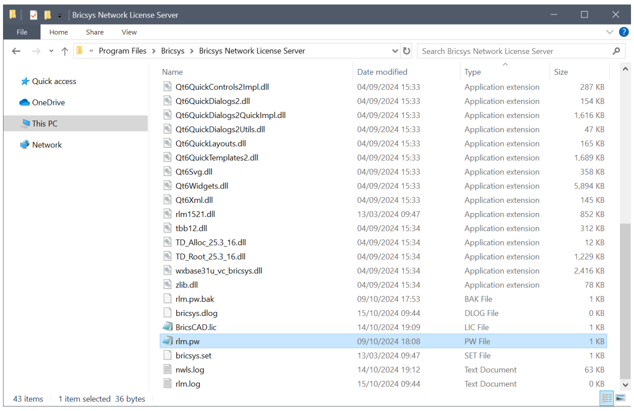
The new Bricsys Network License Server brings the following benefits:

- It detects licenses that are covered by an active maintenance contract and provides the option to upgrade them on the **Bricsys Network License Manager** dialog box.
- It has an improved User Interface that provides a good overview of all licenses deployed on a server.
- It guides users to useful resources, such as Bricsys Help Center, Bricsys Support, RLM diagnostic tools, etc.

1.6.5 Network license administration

1.6.5.1 Setup RLM login

Starting with RLM V15.1, a default user with the username *admin* and password *admin* is created if rlm.pw does not already exist in the Bricsys Network License Server folder (by default *C:\Program Files\Bricsys \Bricsys Network License Server*).



Important: If the *admin* password is still the default and/or there are any users without passwords for longer than 10 minutes, the server will shut down automatically.

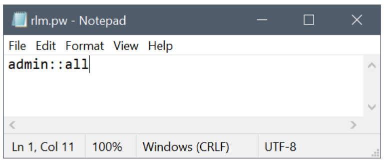
Note:

- When upgrading from the old versions of RLM framework (14.2 and earlier), the old PW files are no longer valid. To fix this, you must delete the old PW file and create a new one.
- It is recommended to protect access to this file so that regular users cannot edit it. Both the RLM password file and the directory which contains it, must be read-write to the RLM process.

1 The RLM password file (rlm.pw) has one line for each user, formatted as follows:

```
1    username::all
```

The username cannot contain a ':' character. The password field is initially left blank. The permissions are generally set to **all**.



The list-of-permissions field is a comma-separated list of the various privileges which you can assign to users:

- **all**: It is a special privilege name. It enables all privileges.
- **edit_meter**: Modifies count for meter counters.
- **edit_options**: Edits options files for ISV servers.
- **edit_rlm_options**: Edits license files and options files for the rlm server.
- **edit_xfer**: Edits server-server license transfer settings for ISV servers.
- **extend_roam**: Allows you to extend roam duration for already-roaming licenses.
- **logfiles**: Enables the functions that change log files - switch, switchr, newlog.
- **remove**: Allows you to remove a license from a running process.
- **reread**: Gives you access to the functions that do reread commands on license servers.
- **shutdown**: Gives you access to the functions which shut down license servers.
- **status**: Allows you to display the status and debug log information from the license servers.

  Note: **edit_meter**, **edit_options**, **edit_rlm_options**, **edit_xfer**, **remove**, and **shutdown** options enable "status" privilege even if not present.

  Note: Check the list-of-permissions at the bottom of this page. A user with no privileges assigned will have access to the ACTIVATE LICENSE, DIAGNOSTICS, RLM MANUAL, SYSTEM INFO, ABOUT, CHANGE PASSWORD, and LOGOUT commands.

  Examples:

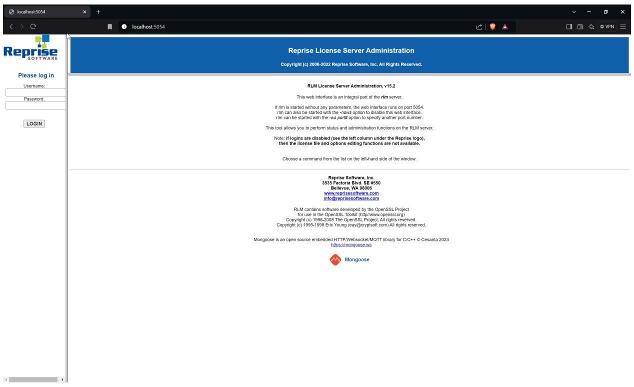
  ```
  user1:$5ukMApW1jixwcrGqRAL091:all
  ```

  ```
  user2::edit_options,edit_rlm_options,reread
  ```

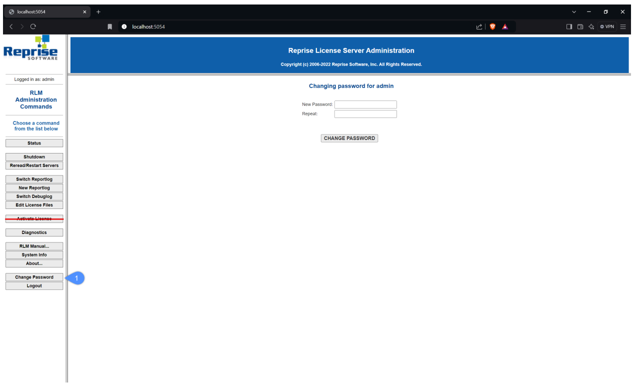
2 Access the license server web portal from the browser by using localhost:5054 in the same system or by using <License_Server_IP>:5054 in other systems.

3 You are prompted for a username and password. You can enter the username and log in without supplying a password since the password field is left empty.

4 ***Note:*** If you do not add a password to the newly created login, the result will be that the RLM will shut down by itself in 10 minutes, and other undesired effects will appear, such as an incomplete Diagnostics file.



5 To set a password or to change it later, select the **Change Password** button (1) once logged in as that user. Enter the new password and click the **CHANGE PASSWORD** button to update it. The new password is written to the rlm.pw file automatically. In the rlm.pw file, the password field is an encrypted hash of the actual password (similar to the Unix password file).



After that, restart the server or simply kill the rlm.exe process and start it again from Windows services. Then, go back to the web UI and log in with the new credentials.

6 Whenever a new user attempts to access the web portal, authentication (username and password) will be requested.

1.6.5.2 Network License Server administration

When the Network License Server (also named RLM License server) is running, you can configure it in the Reprise License Server Administration web interface. See also the **Setup RLM login** article.

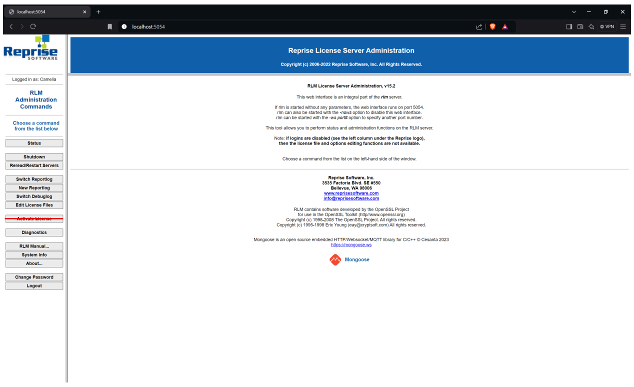
1 Opening the web interface

1 If the Network License Server is not running, click the **Bricsys Network License Server** desktop shortcut (on Windows 10 or higher: right-click > Run as administrator).

The **Bricsys Network License Manager** dialog box opens.

2 Browse to **http://localhost:5054**.

The Reprise License Server Administration web interface opens.



Note:

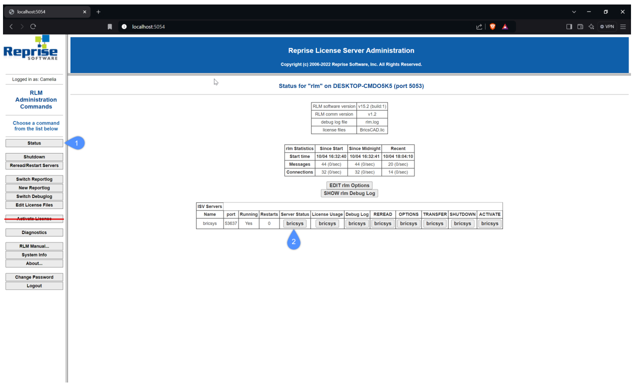
- Do not use the **Activate License** button because it is not related to Bricsys licenses.
- To get an overview of all options, click the **RLM Manual...** button on the left or go to the RLM License Administration Manual.

2 Checking the license and server status

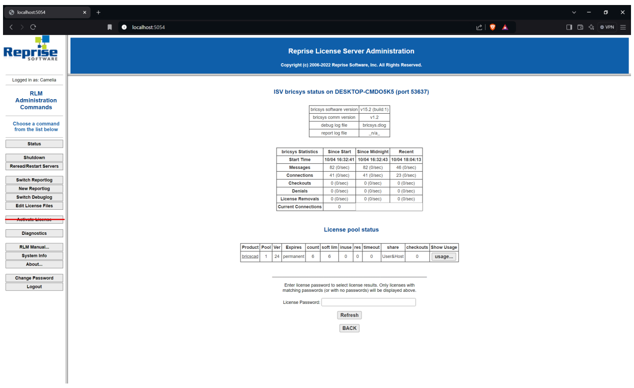
You can check the status of all network licenses on the server.

1 Click the **Status** button (1) on the left.

The **Status for "rlm" on LicenseServer (port 5053)** page opens.



2 Click the **bricsys** button (2) in the **Server Status** column of the ISV Servers table to view the status of the Bricsys Network license.

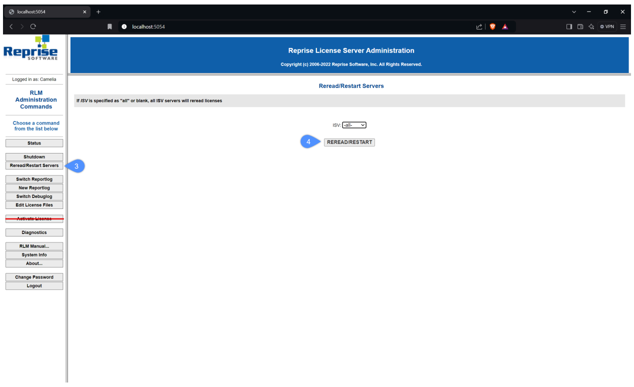


3 Rereading the license / Restarting the server

If you want to restart the server, for example, after modifying the bricsys.opt file, follow the next steps:

1 Click the **Reread/Restart Servers** button (3) on the left.

The **Reread/Restart Servers** page opens.



2 Click the **REREAD/RESTART** button (4).

The server restarts and rereads the Network license.

1.6.5.3 Advanced configurations

Once the Network License Server is set up, BricsCAD may report error -17 when trying to connect to the server.

The presence of firewalls, proxy servers, anti-virus software, and other security measures may be the cause.

1 Anti-virus

If anti-virus software is running on the server and blocking the license server program (**rlm.exe** on Windows, **rlm** on Linux), you need to unblock the RLM service in the anti-virus configuration.

Check your anti-virus configuration on how to unblock services.

2 Firewall

If a firewall is installed between the server and BricsCAD, 2 port numbers need to be opened in the firewall: the port number 5053 and the port number assigned to the Bricsys license server.

This port number is dynamically assigned when the license server starts. To open this port in the firewall, it must be first set in the BricsCAD.lic file on the server:

1 Open the Reprise License Server Administration web interface at **http://localhost:5054**.

2 Click the **Status** button (1) on the left. The **Status for "rlm" on [LicenseServer] (port 5053)** page opens.

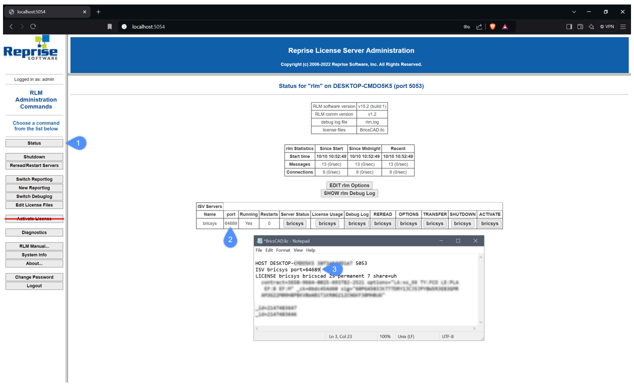
3 Look up the port number in the ISV Servers table. The port number is displayed in the bricsys row in the 2nd column (2).

4 Copy this port number.

5 Open the BricsCAD.lic file in a text editor (for example Notepad, Gedit, etc.). The default location of the

file is *C:\Program Files\Bricsys\Bricsys Network License Server*.

You can fix the port number by modifying the line ISV bricsys into: ISV bricsys port=[port number] (3), where [port number] must be replaced by port number copied from the ISV Servers table.



6 In the firewall, open both numbers 5053 and the port number you added to the BricsCAD.lic file.

7 Shut down and restart the license server program.

Note: If this does not solve the problem, add the license server program (rlm.exe on Windows, rlm on Linux/macOS) as an exception onto the firewall exceptions and restart the license server again.

Remember: The Network License Manager tool, located in the *C:\Program Files\Bricsys\Bricsys Network License Server\NetworkLicenseManager.exe* folder must be able to reach the **license.bricsys.com/actpro** port **80** and **https://api.bricsys.com/** port **443**.

3 Proxy server

If a proxy server exists between BricsCAD and the Network License Server, proxy settings should be configured in BricsCAD.

See more information in the **Proxy settings on a client computer**, **Configuring the proxy settings permanently for network license use** section.

1.6.5.4 License pooling

1 About

When having multiple license levels present, the Bricsys Network License Server will allocate the seats to BricsCAD machines in the order of the LIC file names. If you want to control in detail this action, please edit the options file: **bricsys.opt**.

2 Editing the options

You can edit the options in 2 ways:

- In the **bricsys.opt** file, located in the Bricsys Network License Server installation folder.
- On the web interface (**Reprise License Server Administration**).
  - Browse to *http://localhost:5054* on the Network License Server.
  - Click **Status** in the menu on the left side.

- Click **bricsys** in the OPTIONS column for the **bricsys** ISV row.

Once edited, **Reread/Restart** the license server.

3 Commonly used options

Groups can be used to define a list of usernames, hostnames, or IP-addresses.

The name can later be used to apply rules for that group.

		Syntax	Example
GROUP	Defines a group of usernames.	GROUP name list-of-usernames	GROUP engineers userA userB userC
HOST_GROUP	Defines a group of hostnames.	HOST_GROUP name list-of-hostnames	HOST_GROUP firstFloor machineA machineB machineC
INTERNET_GROUP	Defines a group of IP-addresses.	INTERNET_GROUP name list-of-ip-addresses	INTERNET_GROUP networkGhent 192.168.1.* 192.168.2.* ***Note:*** The wildcard character (*) can be used in IP addresses.
RESERVE	Reserve a number of seats to a specific (group of) user(s).	RESERVE num product user|host|group|host_group|internet|internet_group|project who [id=nnn]	RESERVE 20 bricscad GROUP engineers id=nnn
MAX	Limit the number of seats available to a specific (group of) user(s).	MAX num product user|host|group|host_group|internet|internet_group|project who [id=nnn]	MAX 5 bricscad HOST_GROUP firstFloor id=nnn
INCLUDE	Only allow licenses for a product to a specific (group of) user(s). Anyone not specified by the INCLUDE line is not allowed to use the product.	INCLUDE product userhost|group|host_group|internet|internet_group|project who [id=nnn]	INCLUDE bricscad GROUP bricscadusers id=nnn

		Syntax	Example
EXCLUDE	Disallow licenses for a product to a specific (group of) user(s).	EXCLUDE product user|host|group|host_group|internet|internet_group|project who [id=nnn]	EXCLUDE bricscad INTERNET_GROUP networkGhent 192.168.1.* id=nnn ***Note:*** The wildcard character (*) can be used in IP addresses.

Important: If the number of RESERVED seats for multiple GROUPs equals all seats of the license, then only the first GROUP can access the licenses, and the rest of GROUPs will get the **error -22: All licenses in use**.

More options can be found in the RLM end user manual, under the **The ISV Options File** chapter.

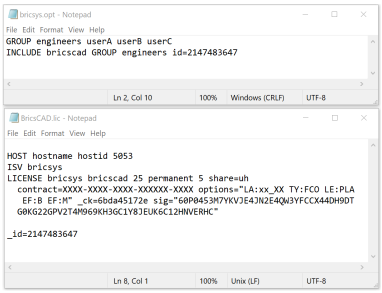
4 License file identifiers

Note:

- After activation of a new network license, an '_id=nnn(*)' field is auto-generated, and it is added at the end of the license file.
- ***Note:*** (*) nnn is the number of the generated ID.
- After upgrading an existing network license, its original IDentifier number remains unchanged.

This '**_id**' is used to identify each license file, and it will be used for license pooling options.

When specifying an [id=nnn] parameter for an added OPTION, that OPTION will only have effect on the pooling of the seats from the corresponding license.


bricsys.opt - Notepad
File Edit Format View Help

```
GROUP engineers userA userB userC
INCLUDE bricscad GROUP engineers id=2147483647
```

Ln 2, Col 10 100% Windows (CRLF) UTF-8

BricsCAD.lic - Notepad
File Edit Format View Help

```
HOST hostname hostid 5053
ISV bricsys
LICENSE bricsys bricscad 25 permanent 5 share=uh
  contract=XXXX-XXXX-XXXX-XXXXXX-XXXX options="LA:xx_XX TY:FCO LE:PLA
   EF:B EF:M" _ck=6bda45172e sig="60P0453M7YKVJE4JN2E4QW3YFCCX44DH9DT
  G0KG22GPV2T4M969KH3GC1Y8JEUK6C12HNVERHC"

_id=2147483647
```

Ln 8, Col 1 100% Unix (LF) UTF-8

5 License pooling by RLM_PROJECT

The following applies to cases where multiple levels of network licenses are activated on the same RLM server. Consequently, the users are able to decide what license level to acquire when activating BricsCAD.

The user can obtain a specific BricsCAD license level, based on the RLM_PROJECT environment variable which should match the ISV server options. The license seats will be reserved on the ISV server for the specific PROJECT users by an IDentifier.

Note: Please be aware that the following actions are to be executed by an IT department person.

1. RLM server actions

Having two license files with different IDentifiers (Lite _id=*xxxxxxxx* and Pro _id=*yyyyyyyy*), add RESERVE options (in the **bricsys.opt** file) for all the licenses seats:

```
RESERVE 5 bricscad PROJECT bc_lite id=xxxxxxxx
RESERVE 5 bricscad PROJECT bc_pro id=yyyyyyyy
```

2. BricsCAD user actions

1 Create a script that *SETs* a temporary environment variable RLM_PROJECT on the user machine, which will allow BricsCAD to negotiate with the RLM server the PROJECT license pool.

2 Executing a specific script will allow BricsCAD to pull the desired license level.

It can be ***.CMD** (command prompt) files:

-
```
@echo off
set RLM_PROJECT=bc_lite
call "C:\Program Files\Bricsys\BricsCAD V25 en_US\bricscad.exe"
```

-
```
@echo off
set RLM_PROJECT=bc_pro
call "C:\Program Files\Bricsys\BricsCAD V25 en_US\bricscad.exe"
```

or it can be ***.PS1** (powershell) files:

-
```
$env:RLM_PROJECT = "bc_lite"
Start-Process -FilePath "C:\Program Files\Bricsys\BricsCAD V25 en_US
\bricscad.exe"
```

-
```
$env:RLM_PROJECT = "bc_pro"
Start-Process -FilePath "C:\Program Files\Bricsys\BricsCAD V25 en_US
\bricscad.exe"
```

Since all license seats are RESERVED in the RLM options, BricsCAD will not be able to acquire a license when the RLM_PROJECT variable does not exist, or when it has a value which does not match the ISV server options.

Raising the available seats above 5 or lowering the RESERVE count below 5 would let some licenses be available freely.

The RLM_PROJECT also works with the other (EXCLUDE/INCLUDE/…) options so a lot of configurations are possible.

1.6.5.5 Network license usage report

There are several possibilities to check the network license usage:

- From the RLM web UI:
- Browse to http://localhost:5054/ and go to **Status** > **License Usage** > **bricsys**.
- From the **bricsys.dlog** file which contains the IN/OUT information. This file is in the RLM folder: *C:\Program Files\Bricsys\Bricsys Network License Server*.
- REPORTLOG option. This option is described on pages 79 and 106 in the RLM License Administration manual.

  It should be added to the **bricsys.opt** file like this:

```
REPORTLOG +D:\your_path\report_name.log
```

Next, perform a Reread/Restart Servers.

The + sign means the new data is appended to the file, otherwise the file will be overwritten.

- RLM License Administration Bundle:

  https://reprisesoftware.com/admin/software-licensing-downloads.php
- JTB FlexReport:

  https://jtbworld.com/jtb-flexreport

1.6.5.6 Migrating a Network license to another server

Do the following:

1. Revoke the license. See the **Revoke a BricsCAD license (limited operation)** article for more information.
2. Activate the license on your new server. See the **Online activation (Network license)** article for more information.
3. Shut down the license manager on the old server and delete the license file on the old server.

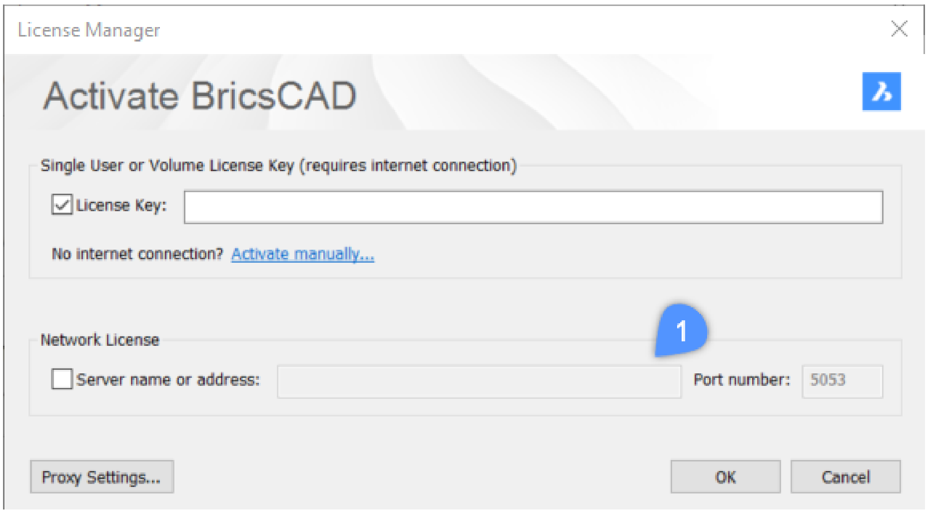
Note: The revoke possibility is limited to two per key. Contact support if you cannot revoke the license.

1.6.6 Using a Network license in BricsCAD

1.6.6.1 Using a Network license on a client computer

Once the network license is activated on the license server, it can be used on client computers.

1. Do one of the following:
    - If you start BricsCAD for the first time, click the **Activate** button in the **BricsCAD Launcher** dialog box.
    - If you replace or upgrade an existing license, launch the LICENSEMANAGER command, click **Manage License** and click **Modify**.
2. Check the **Network License** option in the **License Manager** > **Activate BricsCAD** dialog box.
3. Type the host name or IP address of the license server in the **Server name or address** field (1).



4 (Optional) Type the port number in the **Port Number** field.

5 If your computer is connected to the license server through a proxy server, click the **Proxy Settings** button to configure the proxy settings for a network license.

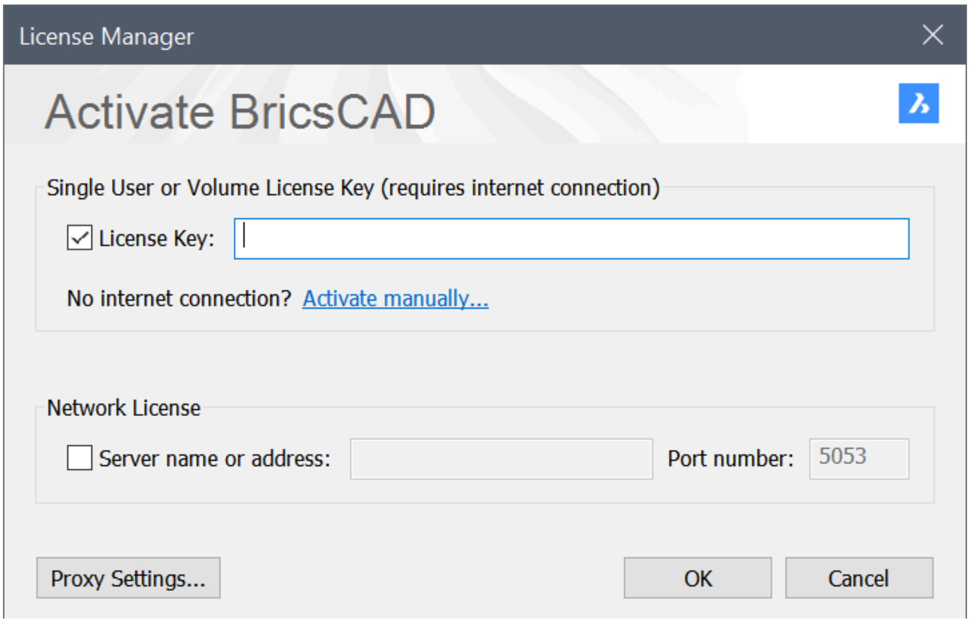
Note: **Proxy Settings** are also configurable from the **Settings** dialog box.

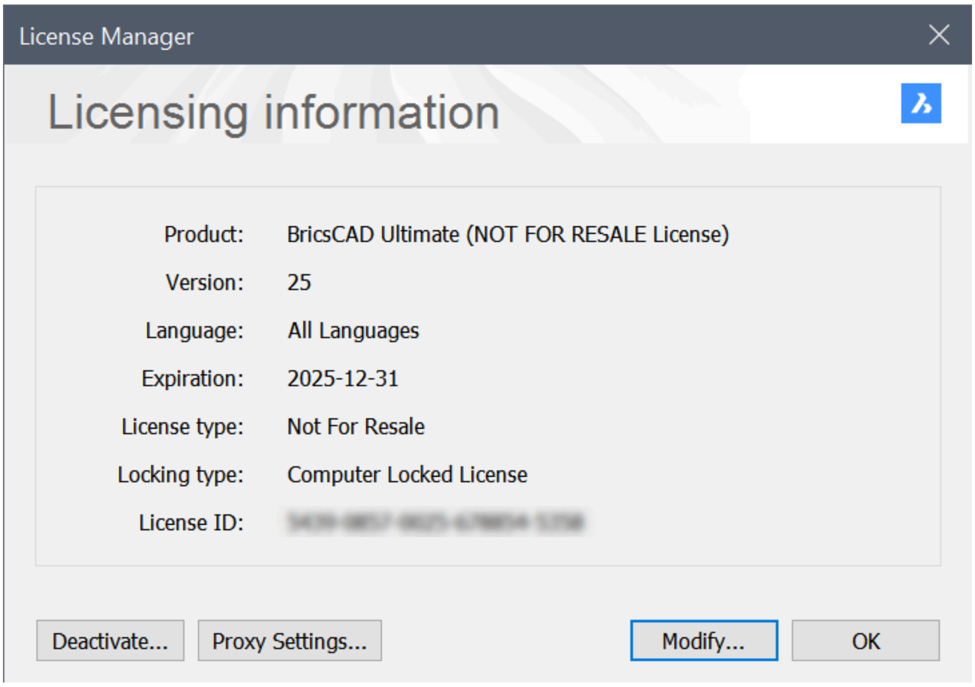
6 Click the **OK** button.

1.6.6.2 Proxy settings on a client computer

1 Configuring the proxy settings

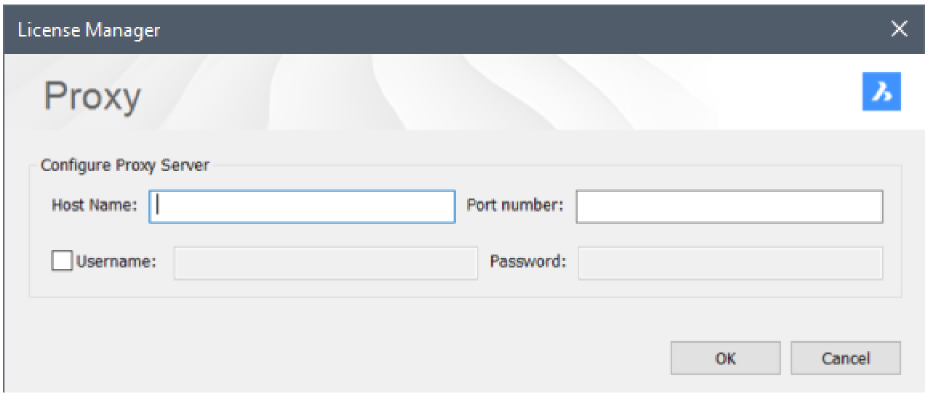
1 Click the **Proxy Settings...** button in the **License Manager** > **Activate BricsCAD** or the **Licensing Information** dialog box.





The **License Manager** > **Proxy** dialog box opens.

2 Complete the necessary fields in the **License Manager** > **Proxy** dialog box.



3 Click the **OK** button to continue.

Note:

- The proxy settings configuration remains active for the current BricsCAD session only. If you need to reactivate or deactivate a license in a future session, you will have to reconfigure the proxy settings.
- The proxy server is also configurable from the **Proxy settings** section in the **Settings** dialog box.

2 Configuring the proxy settings permanently for network license use

If you use a network license, BricsCAD needs to connect to the license server each time a session is opened. Therefore, the proxy settings must be configured permanently.

Proceed as follows on the computer where BricsCAD is running:

1 Create the **HTTP_PROXY** variable: HTTP_PROXY=[LicenseServer]:[PortNumber].

2 (Optional) If the proxy server supports basic authentication, create the **HTTP_PROXY_CREDENTIALS** variable: HTTP_PROXY_CREDENTIALS=[UserName]:[Password].

1.6.6.3 Roam (check-out) a license

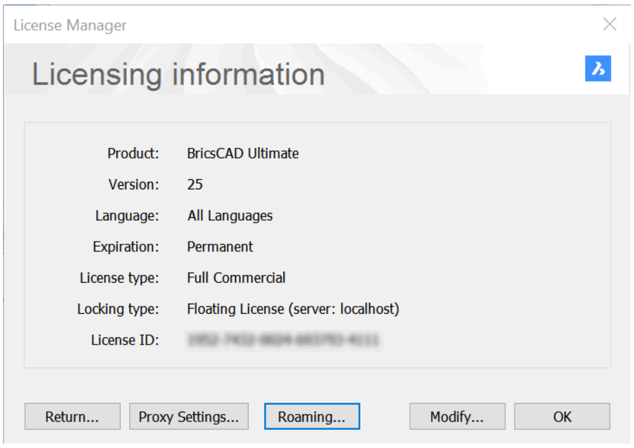
With a network license, you can set a floating license to roam to a system which will subsequently be disconnected from the network. The resulting license can be used for the number of days specified when the license was set to roam. After this period, the license is checked back in automatically. If needed, you can return the roamed license to the license pool early.

The license file(s) on the Network License Server contain(s) a parameter *max_roam_count*. This is set to the total number of seats on a network license by default.

1 Roaming a Network license on a client computer

A network license can be roamed to use BricsCAD without connection to the license server (work outside of the office, at home, etc.).

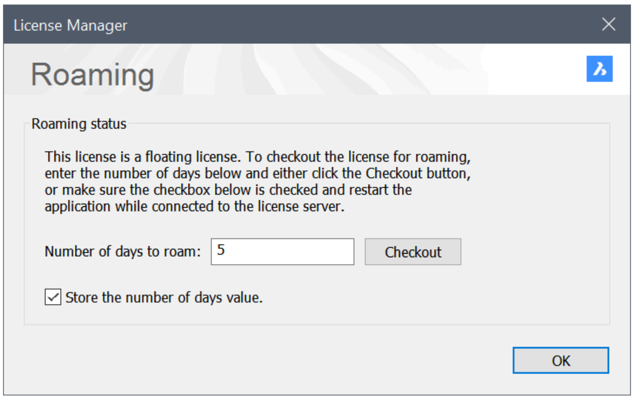
When a network license is used on a client computer, the **Roaming...** button is displayed in the **License Manager** > **Licensing information** dialog box.



1 If your computer is connected to the license server through a proxy server, click the **Proxy Settings** button to configure the proxy settings for a network license.

2 Click the **Roaming...** button.

The **License Manager** > **Roaming** dialog box opens.



3 Enter the number of days you want to check out the license for roaming and click the **Checkout** button.

4 (Optional) Tick the checkbox if you want to automatically refresh the roaming period when your computer is connected to the server.

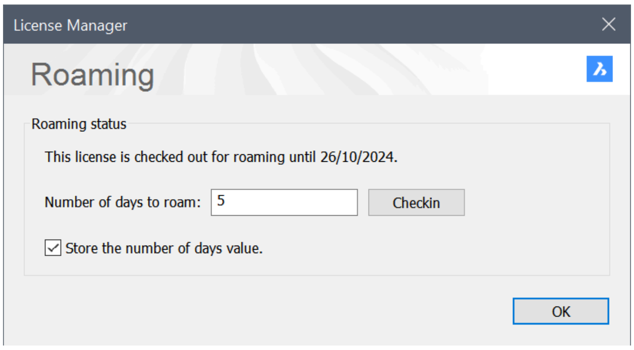
5 Restart BricsCAD to apply the changes.

2 Returning a roamed Network license to the server

If you are connected to the license server, you can return a roamed license to the server.

1 Click the **Roaming...** button in the **License Manager** > **Licensing information** dialog box.

The **License Manager** > **Roaming** dialog box opens.



2 Click the **Checkin** button.

3 Restart BricsCAD to apply the changes.

1.7 Managing BricsCAD licenses with multiple Bricsys accounts

You can share licenses with one or more other Bricsys accounts to allow them to view, activate and revoke your license(s).

Do the following:

1 Log in to the Bricsys website.

2 Click your user name and select **Manage licenses** in the drop-down menu.

3 Hover over the license you want to share and click the Pencil icon ( ).

The **License details** page displays.

4 Under **Managers**, click **Add Manager**.

5 Type the login email address* of the new manager, then click the plus icon.

6 The new manager is added.

An email is sent to inform the new manager.

The **X** icon after the name allows to remove the manager.

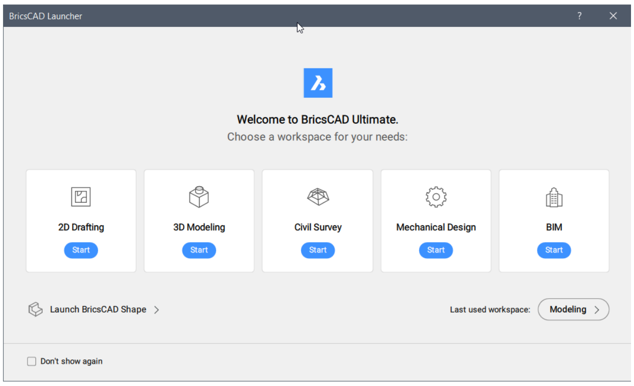
* Managers must have a Bricsys account.

2. Getting started with BricsCAD

2.1 Launching BricsCAD

2.1.1 BricsCAD Launcher

If the GETSTARTED system variable is ON, the **BricsCAD Launcher** dialog box displays when you launch BricsCAD. Here you can choose the Workspace for the current session. The available workspaces depend on the license level.



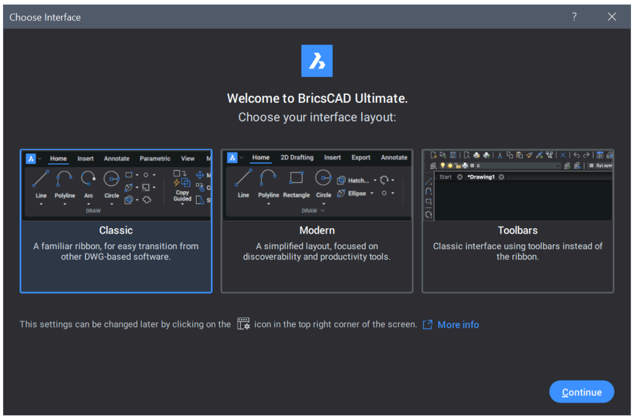
- **2D Drafting**: launches the Drafting workspace. BricsCAD Lite license or higher is required.
- **3D Modeling**: launches the Modeling workspace. BricsCAD Pro license or higher is required.
- **Civil Survey**: launches the Civil workspace. BricsCAD Pro license or higher is required.
- **Mechanical Design**: launches the Mechanical workspace. BricsCAD Mechanical or BricsCAD Ultimate license is required.
- **BIM**: launches the BIM workspace. BricsCAD BIM or BricsCAD Ultimate license is required.
- **Launch BricsCAD Shape**: launches BricsCAD Shape. No license is required.
- **Last used workspace**: launches the last used workspace.
- **Don't show again**: sets the GETSTARTED system variable to 0. To reset the display of the **BricsCAD Launcher** dialog box, type **getstarted** in the Command line and choose **1 for ON**.

  Note: The BricsCAD Launcher cannot be disabled in Trial versions.

Note: The free solid modeling tool, BricsCAD Shape, is built into the single BricsCAD installer. When the BricsCAD Ultimate 30-day trial expires, it reverts to a perpetual BricsCAD Shape license, which can be converted to a full commercial license by purchasing a license key. You do not need to uninstall BricsCAD first.

2.1.2 Choose Interface

On a clean installation of BricsCAD, after selecting the workspace, the **Choose Interface** dialog box displays to choose your interface layout.



You can choose between:

- **Classic**: A familiar ribbon, for easy transition from other DWG-based software.

  Note: The **Classic** interface is the default one.

- **Modern**: A simplified layout, focused on discoverability and productivity tools.
- **Toolbars**: Classic interface using toolbars instead of the ribbon.

Remember: The **Choose Interface** dialog box only displays on a clean installation of BricsCAD. Later on, you can use the **Interface Settings** dialog box to modify the interface.

2.2 BricsCAD and AutoCAD® feature comparison

The following tables compare BricsCAD features with AutoCAD® features.

Drawings							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Native DWG file format (thru AutoCAD® 2025)	√	√	√	√	√	√	√
DWG Password Protection	√	√	√	√	√	-	-
eTransmit	√	√	√	√	√	√	√
Sheet Set Manager	√	√	√	√	√	√	√
Geographic Location Settings	√	√	√	√	√	√	√

Drawings							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Bing and Esri Maps ***Note:*** BricsCAD provides Bing and Esri Maps through the GEOMAP command.	√	√	√	√	√	-	-
Drawing Compare	√	√	√	√	√	√	√
Cloud Collaboration	√	√	√	√	√	√	√

Interface							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Dark and Light Color Themes	√	√	√	√	√	√	√
Industry standard commands, aliases, and variables	√	√	√	√	√	√	√
Clickable Command Line with Auto Complete	√	√	√	√	√	√	√
Start page	√	√	√	√	√	√	√
File (Drawing) tabs	√	√	√	√	√	√	√
Model and Layout tabs	√	√	√	√	√	√	√
Layout Manager	√	√	√	√	√	-	-
Menu Bar	√	√	√	√	√	√	√

Interface							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Toolbars	√	√	√	√	√	√	√
Ribbon	√	√	√	√	√	√	√
Tool Palettes	√	√	√	√	√	√	√
Contextual Right-click Menus	√	√	√	√	√	√	√
Contextual Quad Menu	√	√	√	√	√	-	-
Status Bar	√	√	√	√	√	√	√
Workspaces	√	√	√	√	√	√	√
Searchable Settings (SysVar) Dialog Box	√	√	√	√	√	-	-
Lockable Interface	√	√	√	√	√	√	√
Clean Screen Mode	√	√	√	√	√	√	√
Drawing Explorer	√	√	√	√	√	-	-
Customizable User Interface (CUI)	√	√	√	√	√	√	√
Structure panel	√	√	√	√	√	-	-

Selection, snap & track							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Selection Filtering (QSELECT)	√	√	√	√	√	√	√
Add Selected	√	√	√	√	√	√	√
Select Similar	√	√	√	√	√	√	√
Selection Cycling	√	√	√	√	√	√	√
Entity Snaps (OSNAPs)	√	√	√	√	√	√	√
Polar Tracking	√	√	√	√	√	√	√
Z-intersection 3D snap	√	√	√	√	√	-	-
Nearest Distance	√	√	√	√	√	-	-
Adaptive Grid Snap	√	√	√	√	√	-	-
Real-time Boundary Detection	√	√	√	√	√	-	-
2D Dynamic UCS	√	√	√	√	√	-	-
3D Dynamic UCS	√	√	√	√	√	-	√

Drawing tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Line, Polyline, 3D Polyline, Spline, Ray, Xline	√	√	√	√	√	√	√

Drawing tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Multi-lines and Multi-line Styles	√	√	√	√	√	-	√
Circle, Arc, Ellipse, Elliptical Arc	√	√	√	√	√	√	√
Polygon, Rectangle, Region, Donut, Point	√	√	√	√	√	√	√
Helix	√	√	√	√	√	-	√
2D Parameters & Constraints	√	√	√	√	√	-	√
2D Auto-constrain	√	√	√	√	√	1	1
Calculator	√	√	√	√	√	√	√

Annotation tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Dimensions, Dimension Styles	√	√	√	√	√	√	√
Quick Dimension	√	√	√	√	√	√	√
Multi-leaders, Multi-leader Styles	√	√	√	√	√	√	√
Single and Multi-line text, Text Styles, Text Editor	√	√	√	√	√	√	√
Hatches, Gradients, Hatch/ Gradient Editor	√	√	√	√	√	√	√

Annotation tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Wipeouts	√	√	√	√	√	√	√
Revision Clouds	√	√	√	√	√	√	√
Attributes, Attribute Editor	√	√	√	√	√	√	√
Tables, Table Styles	√	√	√	√	√	√	√
Fields	√	√	√	√	√	√	√
Data Extraction	√	√	√	√	√	-	√
Data Linking	√	√	√	√	√	√	√
Annotation Scaling	√	√	√	√	√	√	√
Spell Check	√	√	√	√	√	√	√
AutoCAD® Mechanical Compatibility: Read, Display, Edit, Create	-	-	-	2	2	-	-

Editing tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Move, Copy, Rotate, Scale, Stretch, Mirror, Align	√	√	√	√	√	√	√

Editing tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Trim, Extend, Lengthen, Break, Join	√	√	√	√	√	√	√
Fillet, Chamfer	√	√	√	√	√	√	√
CopyGuided	√	√	√	√	√	-	-
MoveGuided	√	√	√	√	√	-	-
Grip Editing	√	√	√	√	√	√	√
Properties panel	√	√	√	√	√	√	√
Manipulator (3D Gizmo)	√	√	√	√	√	-	√
Draw Order	√	√	√	√	√	√	√
Undo Per Entity (Property History)	√	√	√	√	√	-	-
Optimize	√	√	√	√	√	-	-
Simplify	√	√	√	√	√	-	-
Drawing Health Management	√	√	√	√	√	-	-

Reference tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Blocks, Block Editor	√	√	√	√	√	√	√

Reference tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Dynamic Blocks	3	3	3	3	3	√	√
Camera-facing Blocks	√	√	√	√	√	-	-
Blockify	√	√	√	√	√	4	4
Associative Arrays	√	√	√	√	√	√	√
DWG (XREF) Attach, Clip, Bind, & Edit	√	√	√	√	√	√	√
PDF Attach, & Clip	√	√	√	√	√	√	√
Image Attach, Clip, & Adjust, with Transparency	√	√	√	√	√	√	√
Point Cloud Color-mapping	-	√	√	√	√	-	√

Visualization tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Layers & Layer States	√	√	√	√	√	√	√
Entity Transparency	√	√	√	√	√	√	√
Visual Styles	√	√	√	√	√	-	√
LookFrom (ViewCube)	√	√	√	√	√	-	√

Visualization tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Match Perspective	√	√	√	√	√	-	-
Lights	-	√	√	√	√	-	√
Sun Properties	-	√	√	√	√	-	√
Materials	-	√	√	√	√	-	√
Rendering	-	√	√	√	√	-	√
Animation Editor	-	√	√	√	√	-	-
Walk-through 3D Navigation	-	√	√	√	√	-	√
ANIPATH command	-	√	√	√	√	-	√
Built-in Datasmith Export	-	√	√	√	√	-	-
Built-in Direct Link with Twinmotion/Unreal	-	√	√	√	√	-	-

Modeling tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
ACIS-based Solid Modeling	-	√	√	√	√	-	√
Solid and Surface Modeling	-	√	√	√	√	-	√
Solid Primitives	-	√	√	√	√	-	√

Modeling tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Mesh Primitives	-	√	√	√	√	-	√
Polysolids	-	√	√	√	√	-	√
Union, Subtract, Intersect, Interfere	-	√	√	√	√	-	√
Extrude, Revolve, Loft, Sweep, Thicken	-	√	√	√	√	-	√
Twist	-	√	√	√	√	-	-
Fillet, Chamfer	-	√	√	√	√	-	√
Slice	-	√	√	√	√	-	√
Multi-slice	-	√	√	√	√	-	-
T-Connect, L-Connect, & Connect with Nearest Tools	-	√	√	√	√	-	-
Extract Edges	-	√	√	√	√	-	√
Direct (history-less) Modeling	-	√	√	√	√	-	√
Deformable (Free-form) Modeling	-	√	√	√	√	-	√
Audit, Simplify	√	√	√	√	√	-	√
Solidify	√	√	√	√	√	-	-
Stitch	-	√	√	√	√	-	√
3D Parameters & Constraints	-	√	√	√	√	-	√
3D Automatic Parameterization	-	√	√	√	√	-	-

Modeling tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
3D Blockify and Parametric Blockify	-	√	√	√	√	-	-
3D Parametric Design Tables	-	√	√	√	√	-	-
Section Planes	-	√	√	√	√	-	√
Drawing Views: Base, Projected, Section, Detail and Break Views	-	√	√	√	√	-	√
3D Threads	-	√	√	√	√	-	-
Point Cloud Align	-	√	√	√	√	-	-
Point Cloud Bubble Viewer (static and kinematic scans)	√	√	√	√	√	-	-

Civil and Survey tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
TIN Surface Modeling	-	√	√	√	√	-	-
Civil Points	-	√	√	√	√	-	-
TIN Volume Surface	-	√	√	√	√	-	-
Grading	-	√	√	√	√	-	-
Alignments	-	√	√	√	√	-	-

Civil and Survey tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Strings	-	√	√	√	√	-	-
Profiles (Vertical Alignments)	-	√	√	√	√	-	-
Corridors	-	√	√	√	√	-	-
Autodesk® Civil 3D Object Enabler	√	√	√	√	√	-	√
Boundary Trim	-	√	√	√	√	-	-
GIS Data	-	√	√	√	√	-	-
Labels	-	√	√	√	√	-	-

Building tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
CAD-based BIM features	-	-	√	-	√	-	-
IFC Import/Export	-	-	√	-	√	-	-
IDS Import	-	-	√	-	√	-	-
RVT/RFA Import	-	-	√	-	√	-	-
RVT Export	-	-	√	-	√	-	-
Structural Modeling Toolset	-	-	√	-	√	-	-

Building tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Project Database	-	-	√	-	√	-	-
Model federation reference points (Survey, Project, and Site Location)	-	-	√	-	√	-	-
Project Browser	-	-	√	-	√	-	-
Library Database	-	-	√	-	√	-	-
Site, Building and Stories	-	-	√	-	√	-	-
Windows and Doors	-	-	√	-	√	-	-
Fully Parametric BIM Component Creation	-	-	√	-	√	-	-
Bill of Materials (BOM) Manager	-	-	√	√	√	-	-
3D Spaces	-	-	√	-	√	-	-
Wall	-	-	√	-	√	-	-
Roof	-	-	√	-	√	-	-
Quickdraw	-	-	√	-	√	-	-
Quickbuilding	-	-	√	-	√	-	-
Point Cloud Classifier	-	√	√	√	√	-	-
Point Cloud Floor Detection	-	-	√	-	√	-	-
Point Cloud Projection	-	-	√	-	√	-	-
Point Cloud Geometry Detection	-	-	√	-	√	-	-

Building tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Point Cloud FitPlanar (geometry detection)	-	-	√	-	√	-	-
Point Cloud Room Detection	-	-	√	-	√	-	-
Point Cloud FitRooms	-	-	√	-	√	-	-
Point Cloud Dollhouse	√	√	√	√	√	-	-
POINTCLOUDGAPFILLING	√	√	√	√	√	-	-
POINTCLOUDSHOWBUBBLES	√	√	√	√	√	-	-
POINTCLOUDEYEDOMELIGHTING	√	√	√	√	√	-	-
Construction Detail Management	-	-	√	-	√	-	-
Propagate	-	-	√	-	√	-	-
Custom BIM Properties	-	-	√	-	√	-	-
Graphical Override	-	-	√	-	√	-	-
Drawing Customizations	-	-	√	-	√	-	-
BIM Dimensions	-	-	√	-	√	-	-
Selection Dimensions	-	-	√	-	√	-	-
Analytical Models	-	-	√	-	√	-	-
Python Scripts (Beta)	-	-	√	-	√	-	-
BCF issue management	-	-	√	-	√	-	-

Manufacturing tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
AutoCAD® Mechanical 2021 DWG file format	-	-	-	√	√	-	√
Support for 2D Mechanical Drawing Standards (ISO, DIN, JIS, ANSI)	-	-	-	√	√	-	√
Parts Reference and Part Lists	-	-	-	√	√	-	√
Balloons associated with Part References	-	-	-	√	√	-	√
Power dimensions	-	-	-	√	√	-	√
Welding Symbols	-	-	-	√	√	-	√
Surface Texture	-	-	-	√	√	-	√
Feature Control Frame	-	-	-	√	√	-	√
Datum ID	-	-	-	√	√	-	√
Edge Symbol	-	-	-	√	√	-	√
Taper and Slope Symbols	-	-	-	√	√	-	√
Leader Notes	-	-	-	√	√	-	√
Rescale Annotations	-	-	-	√	√	-	√
Mechanical Browser	-	√	√	√	√	-	-
Parts Library	√	√	√	√	√	-	-
Piping	-	√	√	√	√	-	-

Manufacturing tools							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Assembly Modeling	-	-	-	√	√	-	-
Assembly Inspect	-	-	-	√	√	-	-
Exploded Views	-	-	-	√	√	-	-
Kinematic Analysis	-	-	-	√	√	-	-
Bill of Materials (BOM)	-	-	-	√	√	-	-
Balloons associated with BOM and Drawing Views	-	-	-	√	√	-	-
Sheet Metal Design	-	-	-	√	√	-	-
3D Solids to Sheet Metal Parts Recognition and Conversion	-	-	-	√	√	-	-
Corner Relief for Adjacent Edge Flanges	-	-	-	√	√	-	-
Lofted Bends	-	-	-	√	√	-	-
Other features: Junctions, Miters, Tabs, Form Features	-	-	-	√	√	-	-
Feature Coloring	-	-	-	√	√	-	-
Diagnostic & Repair for Sheet Metal Features	-	-	-	√	√	-	-
Batch Processing of Sheet Metal Features	-	-	-	√	√	-	-

Import, export, output							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Print/Plot	√	√	√	√	√	√	√
Print as JPG	√	√	√	√	√	√	√
Print as TIFF	√	√	√	√	√	√	√
Print as PNG	√	√	√	√	√	√	√
Print as BMP	√	√	√	√	√	√	√
Publish/Batch Plot	√	√	√	√	√	√	√
Create PDF Bookmarks	√	√	√	√	√	√	√
CTB & STB Files	√	√	√	√	√	√	√
PC3 Printer Configuration Files	√	√	√	√	√	√	√
Named Page Setups	√	√	√	√	√	√	√
PDF Import, Export	√	√	√	√	√	√	√
DAE Import, Export	√	√	√	√	√	-	-
IFC Import, Export	-	-	√	-	√	-	√
3DM Import, Export	-	√	√	√	√	-	-
LandXML Import, Export	-	√	√	√	√	-	-
Import Autodesk® Civil 3D Data	-	√	√	√	√	-	-
RFA Import	-	-	√	-	√	-	-

Import, export, output							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
SKP Import	-	√	√	√	√	-	-
DGN Import	√	√	√	√	√	√	√
RVT Import, Export	-	-	√	-	√	-	-
DWF Export	√	√	√	√	√	√	√
SVG Export	√	√	√	√	√	-	-
FBX Export	√	√	√	√	√	-	-
STL Export	√	√	√	√	√	-	√
Point Cloud Export (available formats below)	-	√	√	√	√	-	5
LAZ	-	√	√	√	√	-	5
HSPC	-	√	√	√	√	-	5
PTS	-	√	√	√	√	-	5
Point Cloud Import (available formats below)	-	√	√	√	√	-	5
e57	-	√	√	√	√	-	5
HSPC	-	√	√	√	√	-	-
LAS	-	√	√	√	√	-	5
LAZ	-	√	√	√	√	-	5
LGSX	-	√	√	√	√	-	-

Import, export, output							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
HSPC.PACK	-	√	√	√	√	-	-
PTS	-	√	√	√	√	-	5
PTX	-	√	√	√	√	-	5
RCP	-	√	√	√	√	-	√
RCS	-	√	√	√	√	-	√
RSP	-	√	√	√	√	-	5
RDBX	-	√	√	√	√	-	5
ZFS	-	√	√	√	√	-	5
Datasmith Export	-	√	√	√	√	-	-
Additional file formats accessible via Communicator for BricsCAD® (available formats below)	-	√	√	√	√	-	+ / -
CATIA V4 4.1.9 – 4.2.4	-	√	√	√	√	-	√
CATIA V5 V5R8 – V5–6 R2021	-	√	√	√	√	-	√
IGES up to 5.3	-	√	√	√	√	-	√
Inventor V11 – V2022	-	√	√	√	√	-	√
JT 8.x, 9.x, 10, 10.2, 10.3, and 10.5	-	√	√	√	√	-	√
NX 1 – NX 195311 – NX 1953	-	√	√	√	√	-	√

Import, export, output							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Parasolid 9.0.x – 33.0.x	-	√	√	√	√	-	√
Pro/E / Creo 16 – Creo 8.0	-	√	√	√	√	-	√
Solid Edge V18 – SE 2021	-	√	√	√	√	-	-
SolidWorks 2003 – 2021	-	√	√	√	√	-	√
STEP AP203, AP214, AP242	-	√	√	√	√	-	√
STL All (Graphical data only)	-	√	√	√	√	-	-
VDA-FS 1.0 – 2.0	-	√	√	√	√	-	-
XCGM R2012 – 2021 1.0	-	√	√	√	√	-	-

Programming							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
ActiveX	√	√	√	√	√	√	√
Extended Entity Data Editor	√	√	√	√	√	-	√
Script Recorder	√	√	√	√	√	-	√
Full LISP Support (vl, vlr, vla and vlax)	√	√	√	√	√	√	√

Programming							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Solutions Development System (SDS/ADS)	√	√	√	√	√	-	√
Dialog Control Language (DCL)	√	√	√	√	√	√	√
COM API	-	√	√	√	√	√	√
VBA Project Support	-	√	√	√	√	-	√
.NET Framework API	-	√	√	√	√	-	-
.NET Core API	-	-	-	-	-	-	√
BRX (ARX-compatible API)	-	√	√	√	√	-	√
TX (Teigha eXtensions)	√	√	√	√	√	-	-
CUI Menu File Support, with Diesel Expressions	√	√	√	√	√	√	√

Licensing							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Cross-platform Licenses (Windows, Linux, macOS)	√	√	√	√	√	-	-
Perpetual Licenses	√	√	√	√	√	-	-

Licensing							
	BricsCAD					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Perpetual + (Optional) Maintenance (6)	√	√	√	√	√	-	-
Subscriptions (7)	√	√	√	√	√	√	√
Network Multi-user	√	√	√	√	√	-	-
Volume (Perpetual)	√	√	√	√	√	-	-
Academic	√	√	√	√	√	√	√

(1) You can automatically apply geometric constraints but you cannot automatically apply a combination of geometric and dimensional constraints.

(2) You can create ACM symbols using the following commands: AMSURFSYM, AMEDGESYM, AMWELDSYM, AMSIMPLEWELD, AMNOTE, AMFCFRAME, AMDATUMID, AMPARTLIST, AMPARTREF, AMRESCALE.

(3) You can manipulate Dynamic block insertions but you cannot create Dynamic block definitions. You can convert some Dynamic blocks to Parametric blocks that can be further edited. You can also create new Parametric block definitions.

(4) AutoCAD® has the Smart Blocks functionality.

(5) In the Autodesk® product suite, Recap is needed to import non-native point cloud formats and convert to AutoCAD®'s native RCS/RCP. For LAS/LAZ/PTS/PTX, Map 3D and/or Civil 3D may be sufficient.

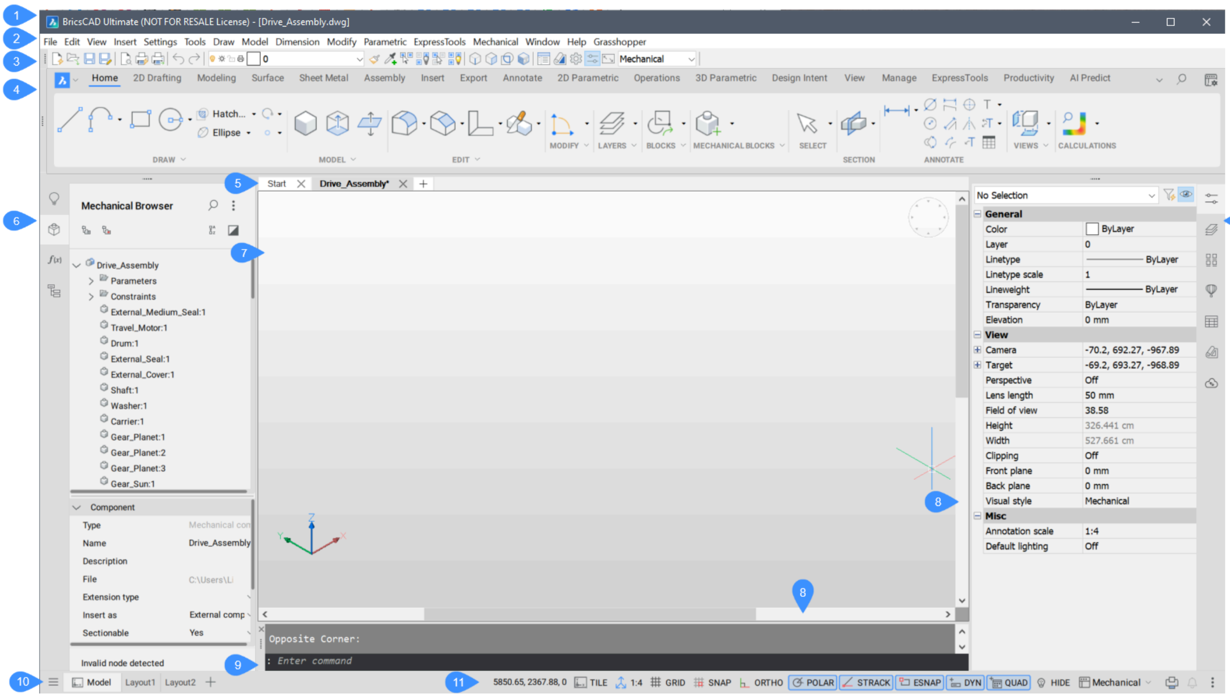
(6) For **Perpetual** license, Maintenance is included for every new license in the first year. From the second year onwards, you are free to renew your Maintenance.

(7) For **Subscriptions**, Maintenance is always included during the subscription period.

2.3 Exploring the interface

2.3.1 The BricsCAD application window

The components of the BricsCAD application window are:



1 License level

2 Menu Bar

3 Toolbars

4 Ribbon

5 Document tabs

6 Panels (left and right)

7 Drawing area

8 Scroll bars

9 Command line

10 Model and Layout Tabs

11 Status Bar

The layout of the BricsCAD application window is controlled by the current Workspace.

License level

Displays the license level used by BricsCAD.

Menu Bar

The **Menu Bar** sits at the top of the BricsCAD application window.

Right click a toolbar, ribbon or panel, then choose **Menu Bar** in the context menu to toggle the display of the menu bar On/Off.

The status of the **Menu Bar** is saved through the MENUBAR system variable.

Toolbars

To display or hide the toolbar, do one of the following:

- Launch the TOOLBAR command.

- Right-click a toolbar, ribbon, or panel, choose **Toolbars** from the context menu, and select the toolbars you want.

Ribbon

To display or hide the ribbon, do one of the following:

- Execute the RIBBON command or the RIBBONCLOSE command.
- Right click a toolbar, ribbon, or panel, choose **Panels**, and select **Ribbon** from the context menu.

Document tabs

Document tabs sit at the top of the graphical screen by default. They allow you to easily switch between open drawings and to access the Start page.

The display of the document tabs is controlled through the SHOWDOCTABS system variable.

Panels (left and right)

Displays the active Panels. Click an icon to open the associated Panel.

To display a Panel:

- Launch the specific command to open the panel.
- Right click a toolbar, ribbon, or panel, choose **Panels**, and select the panel you want to open.

Drawing area

Displays the drawing area.

Scroll bars

Scroll bars are off by default.

To toggle scroll bars On/Off:

- Hold down the Shift key and press the **F4** function key.
- Launch the SCROLLBAR command.

Command line

Run the COMMANDLINE command to display the Command line.

Double click the left side of the status bar to toggle the display of the docked command line.

Model and Layout Tabs

At the bottom of the BricsCAD application window are the Model and one or more Layout tabs which allow you to easily switch between model space and paper space.

Status Bar

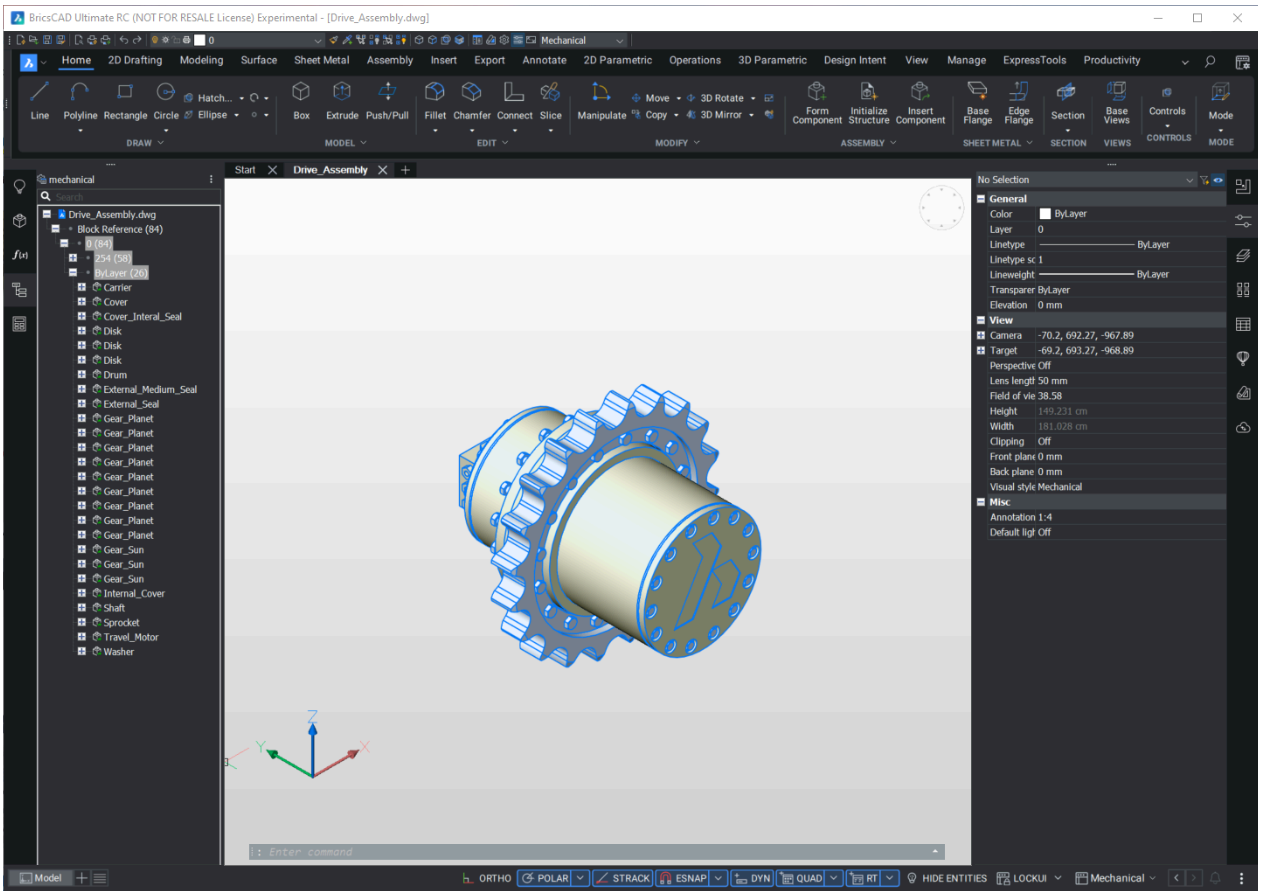
The **Status Bar** sits at the bottom of the BricsCAD application window, in the line with the Model and layout tabs.

Right click a toolbar, ribbon, or panel, then choose **Status Bar** in the context menu to toggle the display of the **Status Bar** On/Off.

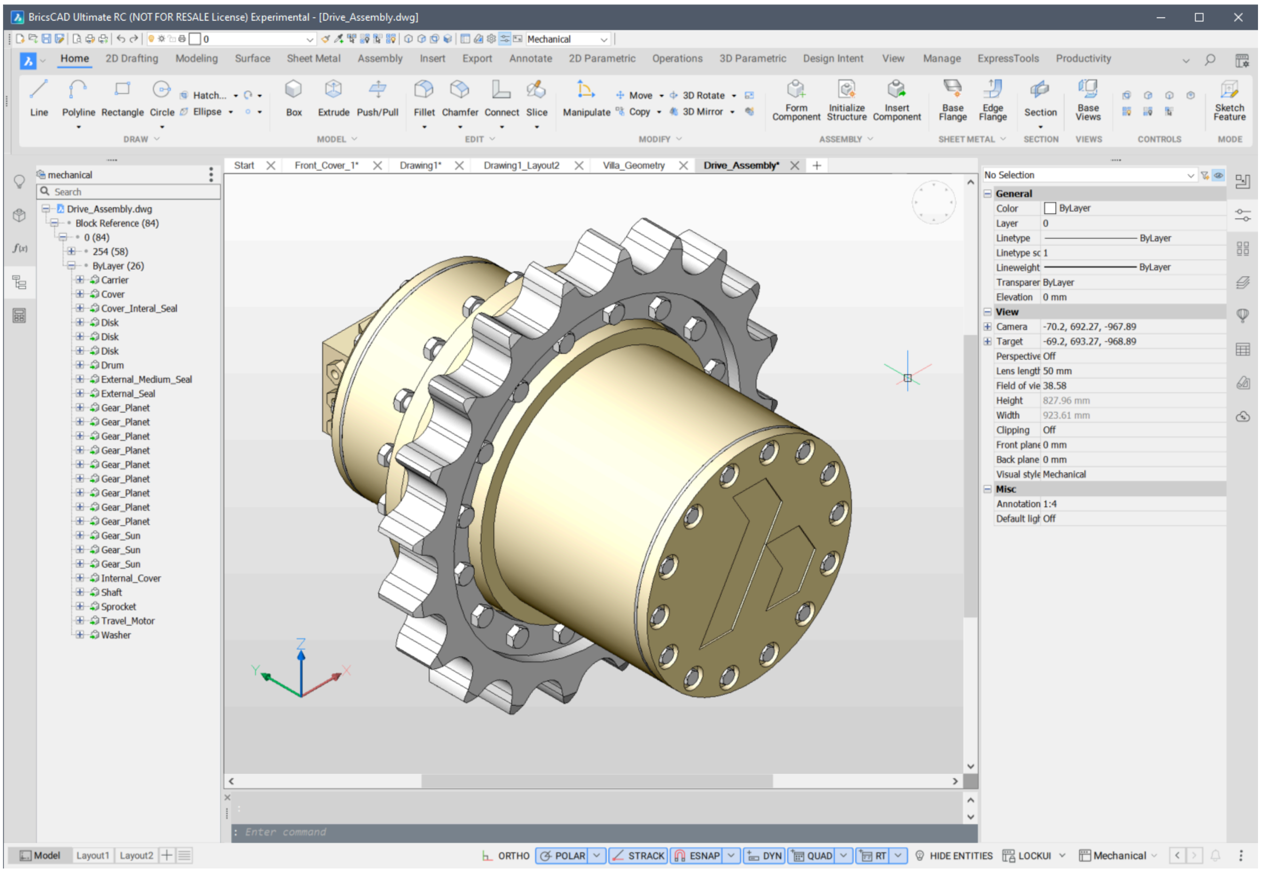
2.3.1.1 The color theme

The COLORTHEME system variable sets the color theme for the ribbon, panels, dialog boxes, etc. You can choose between a dark or a light color theme.

Dark color theme (COLORTHEME = 0):



Light color theme (COLORTHEME = 1):



To set the color theme, do one of the following:

- Type COLORTHEME in the Command line and choose 1 for light or 0 for dark.
- Right-click on a toolbar or ribbon panel and select **Dark Interface** in the context menu.
- Open the **Interface Settings** dialog box and choose a **Theme**.

2.3.1.2 Hiding a selection of user interface elements

The CLEANSCREENON command allows you to hide a selection of user interface elements simultaneously.

The CLEANSCREENOPTIONS system variable defines which user interface elements are hidden.

The **Ctrl+0** keyboard shortcut toggles the cleanscreen state.

On AZERTY-keyboards:

- Do not use the numeric keypad.
- Do not press the Shift-key when hitting the number key.

2.3.2 The Start page

The **Start** page opens after the **BricsCAD Launcher** dialog box, when the GETSTARTED system variable is ON, or at the beginning of a BricsCAD session.

Also, the **Start** page can be accessed by clicking the **Start** tab in the **Document Tabs**, located on top of the drawing area.

The **Start** page has the sections: **Home**, **Cloud**, **Learn**, **Apps**, and **Insights**.

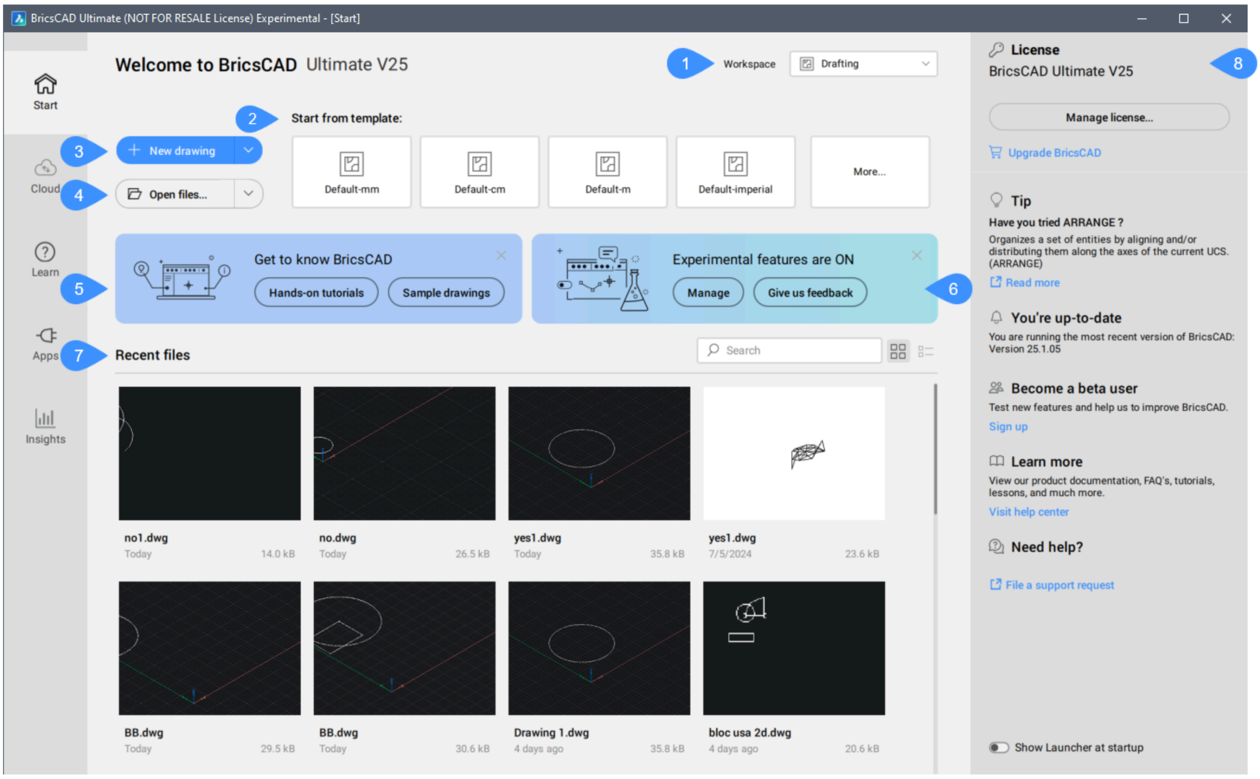
Note: The **Cloud** tab is active only when **Experimental Mode** is enabled.

The **Start** page is active by default (STARTUP == 3). If you accidentally closed the **Start** tab from the **Document Tabs**, you can re-access the **Start** page with the **Ctrl+Home** keyboard shortcut or with the GOTOSTART command.

Note:

- The SHOWDOCTABS system variable needs to be **ON** to display the **Document Tabs** (On by default).
- When opening a DWG file via double click from the operating system's file browser, the **Start** tab is loaded in **Document Tabs** if the STARTUP system variable is set to 2 or 3.

2.3.2.1 Start tab



1 Workspace

2 Start from template

3 New drawing

4 Open files

5 Get to know BricsCAD

6 Experimental new features

7 Recent files

8 Right-side tab

Workspace
Allows you to choose the workspace from the drop-down menu.

Start from template
Displays the last used templates. Click the **More...** button to open the **Create New Drawing From Template** dialog box. You can select a template from the list or you can browse for a custom template to create a new drawing. Each workspace has some predefined templates for new drawings.

New drawing
Creates a new drawing using the current drawing template.

Note: Using the BIM workspace, you can choose between creating a new project or a new drawing.

Click the right-side arrow to display additional options:

Create new sheet set
Opens the **Create New Sheet Set** dialog box.

Start new wizard
Opens the **Create New Drawing** dialog box.

Open files
Opens the **Open file** dialog box to browse for an existing drawing.
Click the right-side arrow to display additional options:

Open files (read-only)
Opens the file selected from the **Open file (read-only)** dialog box only for reading.

Open a sheet set
Opens the **Select a Sheet Set file** dialog box to browse for a sheet set file.

Recover
Opens the **Open Drawing** dialog box to choose a file to recover.

Get to know BricsCAD
Provides step-by-step guidance for using BricsCAD in the selected workspace. The TUTORIALSOPEN command launches.

Note: The TUTORIALSONSTARTPAGE system variable controls the display of this option.

Experimental new features
Provides access to the **Experimental Features** dialog box by pressing the **Discover** or **Manage** button.

Note:

- The EXPERIMENTALONSTARTPAGE system variable controls the display of this option in the **Start** tab. The new value will be considered only after restarting the program.
- The **Manage** and **Give us feedback** buttons are available when Experimental Features are enabled.
- The **Give us feedback** opens a web page where you can send us your feedback about Experimental Features.

Recent files
Displays a list of recently opened files. Click the thumbnail image of the file to easily open it.

Note:

- The number of available recent files is controlled by the value of the RECENTFILES system variable.
- You can arrange the file in **Grid view** or **Table view** mode and search for a specific file.

Right-side tab
Groups information about BricsCAD.

License
Displays the license level which runs on the current session of BricsCAD.

Note:

- Click the **Manage license...** button to open the **Bricsys License Manager** dialog box.
- Click the **Upgrade BricsCAD** button to open the Bricsys store page.
- If you run a Trial version of BricsCAD, there are options to **Enter license** serial number or to **Buy BricsCAD**.

Tip
Displays tips for commands.

Note: Press the **Read more** button to open the relevant Help Center article.

You're up-to-date
Informs you that you are running the most recent version of BricsCAD.

Update available
Notifies you that a newer version of BricsCAD is available.

Note: Click the **Download latest version** button to open the **Download BricsCAD** page.

You're currently offline
You are not connected to the internet and BricsCAD can not check for updates.

Become a beta user
Allows you to become a beta tester. Click the **Sign up** button to open the **Bricsys Beta Tester Program** page, where you can join the program.

Learn more
Click the **Visit help center** button to open the **Bricsys Help Center** website.

Need help?
By pressing the **Fill a support request** button, the website page where you can fill a support request opens.

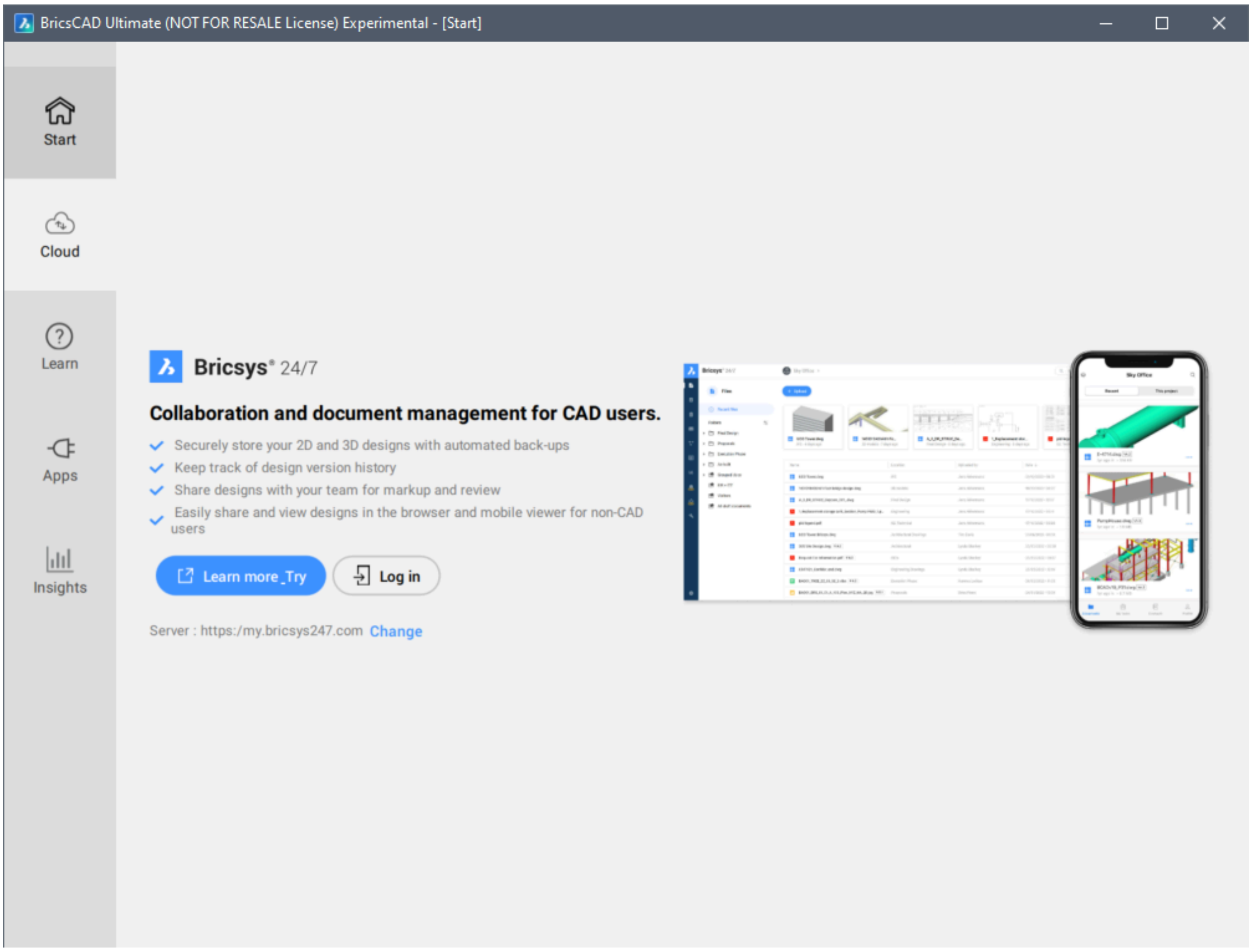
Show launcher at startup
When ticked, the **BricsCAD Launcher** dialog box opens at startup.

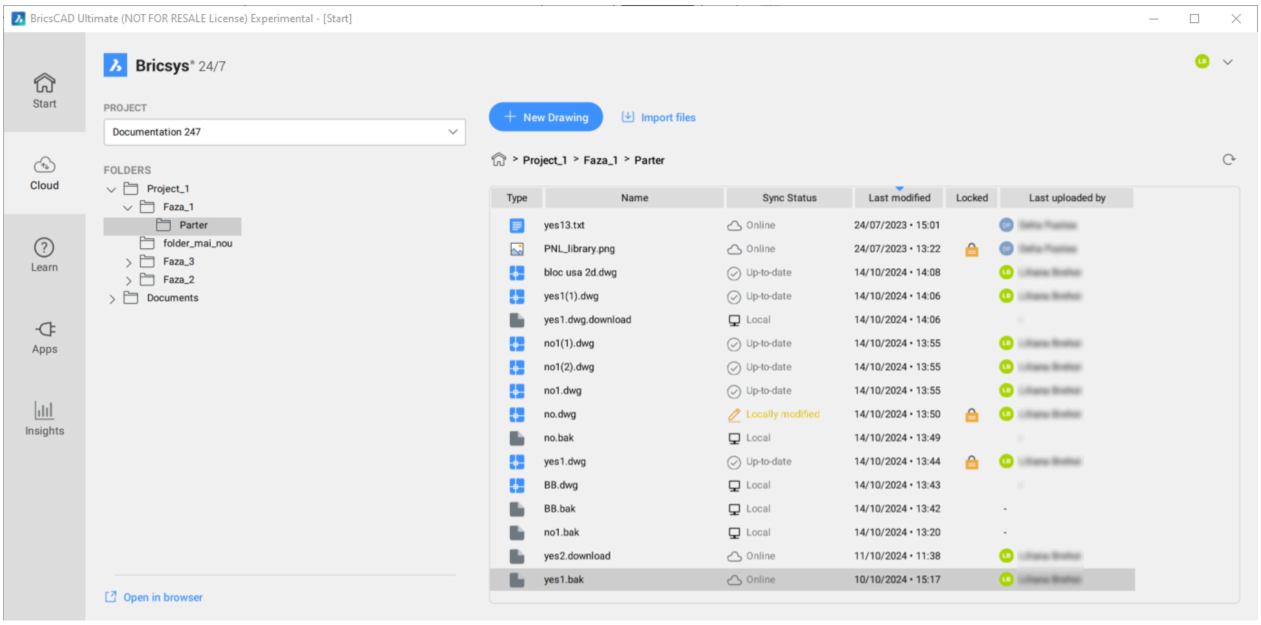
2.3.2.2 Cloud tab (Experimental)

EXP.

The **Cloud** tab provides collaboration and document management for CAD users.

Important: Cloud is an experimental feature. The **Cloud** tab is disabled by default. You have to enable the **Experimental mode** from the **Experimental Features** dialog box.

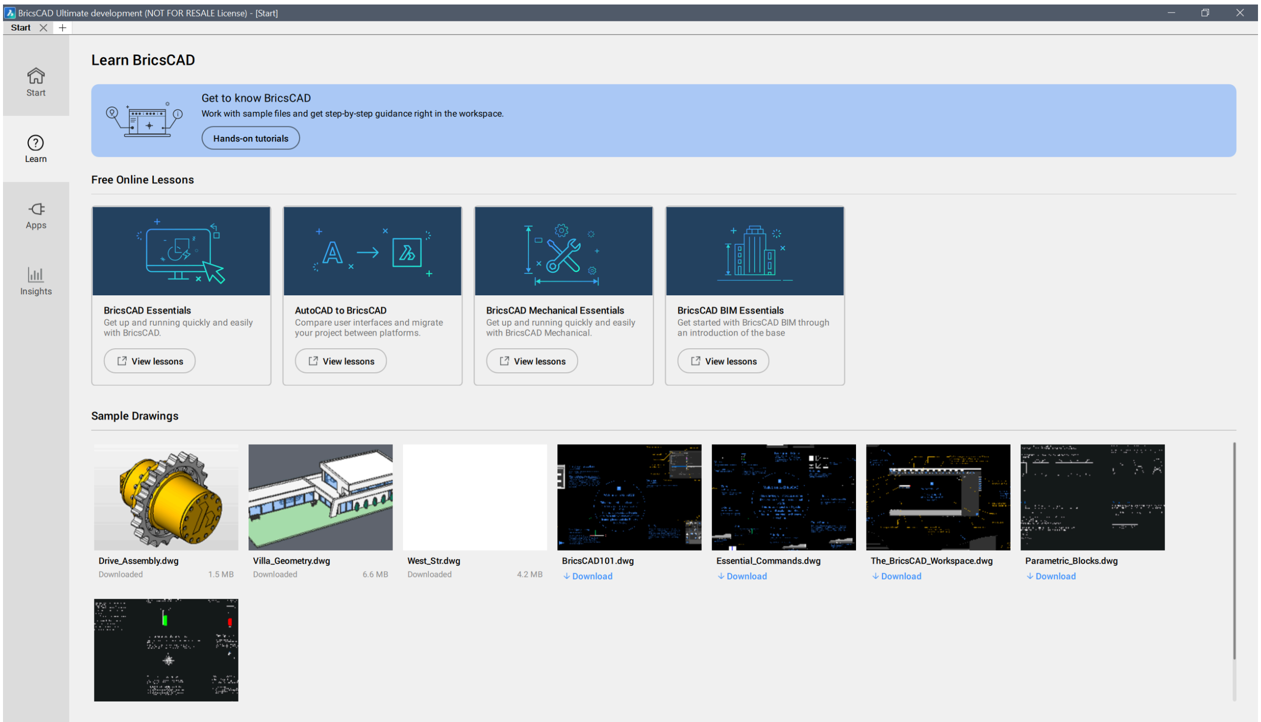




See more information about the **Cloud** feature in the **Cloud** panel article.

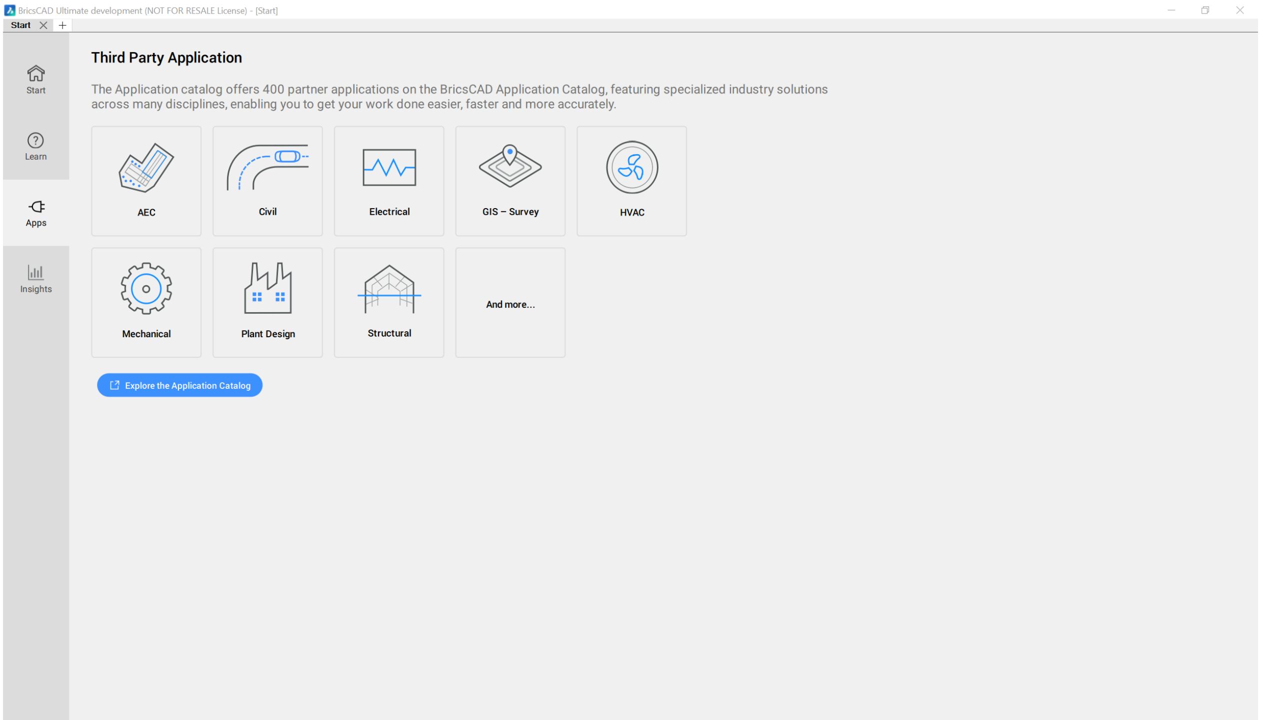
2.3.2.3 Learn tab

Offers free video lessons to learn BricsCAD and sample drawings for use.



2.3.2.4 Apps tab

The Application catalog offers 400 partner applications on the BricsCAD Application Catalog, featuring specialized industry solutions across many disciplines, enabling you to get your work done easier, faster, and more accurately.



Tip:

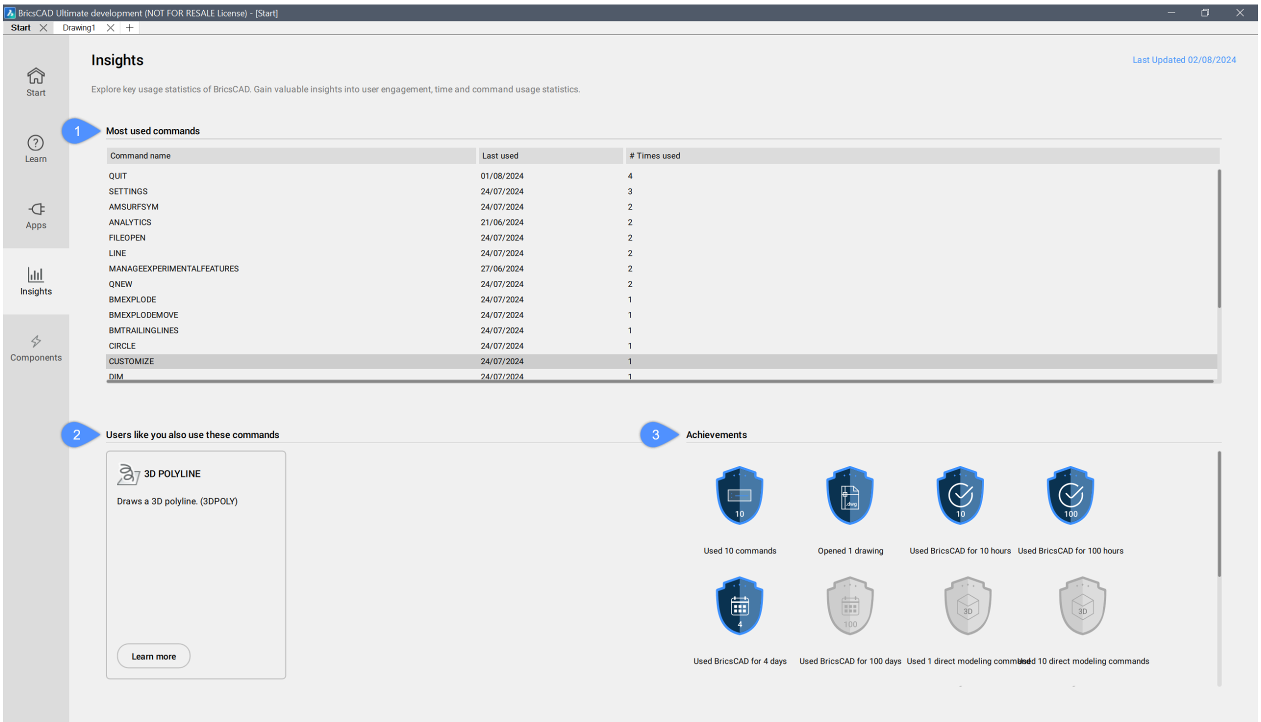
- When you click a category, the BricsCAD Application Catalog opens and filters applications based on the selected category.
- Click the **Explore the Application Catalog** to open the full BricsCAD Application Catalog.

2.3.2.5 Insights tab

Allows you to see summary statistics on the data that you have shared with the Bricsys servers.

The **Insights** tab contains an overview of your most used commands, commands that might be interesting for you, and achievements.

Note: You need to join the BricsCAD Analytics Program to get insights. See more information in the **BricsCAD Analytics Program** article.



Most used commands

Lists the name of the most used commands, when it was last used, and how many times you used the command.

Users like you also use these commands

Lists commands based on how users similar to you use the application.
Click the **Learn more** button to open the relevant Help Center article.

Achievements

The achievements have two states: blue and gray.
Blue: means the achievement has been completed.
Gray: means the achievement has not been completed yet.

Note: The **Insights** tab does not provide live updates. It is updated several times a day on the cloud side.

2.3.3 Document tabs

For each currently open drawing, a document tab is located on top of the drawing area.

Document Tabs allow you to:

- Easily switch between all open drawings.
- Create a new drawing: click the **+** button on the right of the tabs.
- Close a drawing: click the **X** in the document tab.

Note:

- The SHOWDOCTABS system variable needs to be **ON** to display the **Document Tabs** (On by default).
- When opening a DWG file via double click from the operating system's file browser, the **Start** tab is loaded in **Document Tabs** if the STARTUP system variable is set to 2 or 3.

2.3.4 Choosing the workspace

Default workspaces are:

- **Drafting**: requires a BricsCAD Lite or higher license level
- **Modeling**: requires a BricsCAD Pro or higher license level
- **Civil Survey**: requires a BricsCAD Pro or higher license level
- **Mechanical**: requires a BricsCAD Mechanical or BricsCAD Ultimate license level
- **BIM**: requires a BricsCAD BIM or BricsCAD Ultimate license level
- **Complete**: requires a BricsCAD Ultimate license level

The current workspace configures the overall layout of the BricsCAD application window.

A workspace defines the content of:

- Menus
- Toolbars
- Panels
- The ribbon
- The Quad basics

You can set the current workspace in:

- The BricsCAD Launcher.
- The Start page.
- The Current Workspace field in the Status bar.
- The Workspaces toolbar.
- The WORKSPACE command.

2.3.5 The drawing area

When you start a drawing session, your initial working area is called **model space**. Model space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or on a user coordinate system (UCS). BricsCAD provides an additional work area called **paper space**. Each layout represents a sheet of paper. In BricsCAD, there is no limited number of layouts that can be created in a single drawing.

Use the **Model** tab to view and work in model space. Click one of the **Layout** tabs at the bottom of the drawing area to switch to paper space.

2.3.5.1 Model space

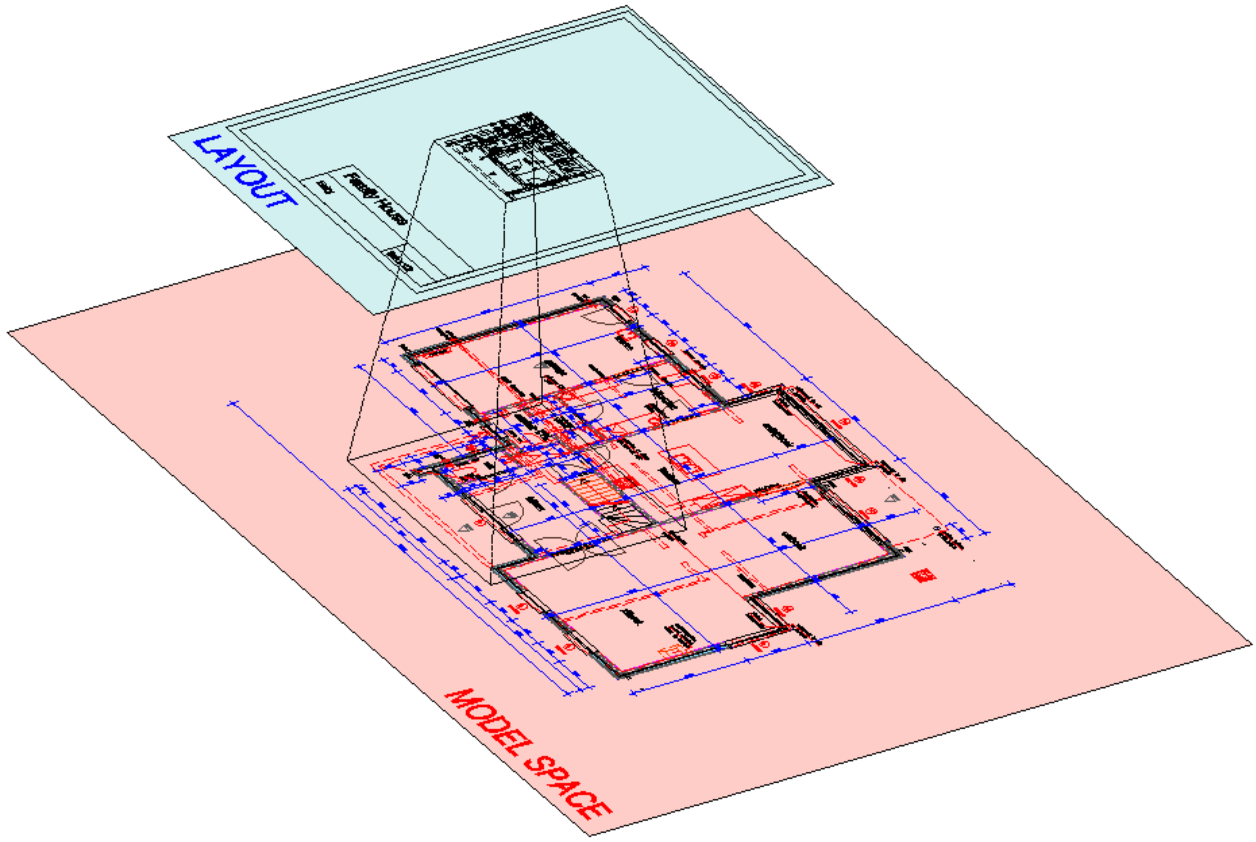
Model space consists of a single view that fills the screen. If needed, you can create additional views called viewports, which can show different views of your drawing or 3D model. You can manage drawing viewports with the VPORTS command.

Viewports are displayed as tiles. You can work in only one of these viewports at once, but all viewports are updated simultaneously. Click in a viewport to make it the current viewport.

2.3.5.2 Paper space

Paper space represents the paper layout of your drawing. In this work area, you can create and arrange different views of your drawing like the way you arrange drawings on a sheet of paper. In paper space you can also add keynotes, annotations, borders, title blocks and other print-related entities which you do not want to view in model space.

Each drawing has at least one **Layout** in which you can have one or more layout viewports. You can consider a layout viewport as a view window on your drawing in model space:



You can place layout viewports anywhere in a paper space layout. Each layout viewport has its own scale and layer visibility. You can print all viewports in the same layout simultaneously.

You can manage layout viewports with the MVIEW command.

Paper space offers multiple advantages when printing your drawing:

- You can create multiple viewports to print multiple views. These views can each display a part of your drawing at the same or at different scales.
- You can create multiple layouts to print the same drawing with different print settings, such as pen widths, printer configuration files, line weight settings, drawing scale, etc.
- You can add print-related entities that are not essential to the model itself, such as keynotes, annotations, title blocks, etc.

2.3.6 Look From widget

The Look From widget allows you to control the display of your drawing. You can select a number of preset views, such as orthographic views and isometric views.

By default, the **Look From** widget displays in the top right corner of the graphic screen.

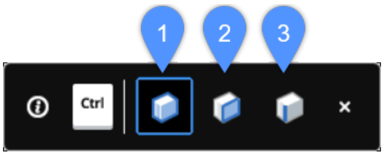
The behavior of the **Look From** widget can be controlled by a number of system variables and user preferences.

2.3.7 Hotkey Assistant widget

2.3.7.1 Description

The Hotkey Assistant widget can be toggled On/Off using the HOTKEYASSISTANT system variable. The options within the widget depend on the commands for which the widget is available.

For example, for the SELECT command the following options are available in the Hotkey Assistant:

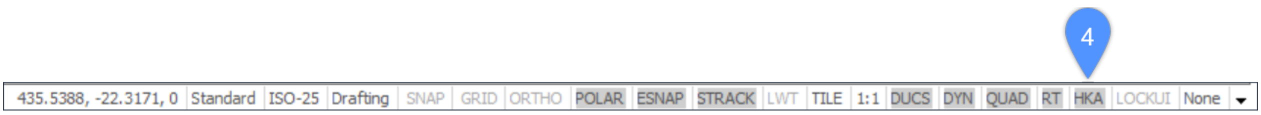


1 Select entity

2 Select face

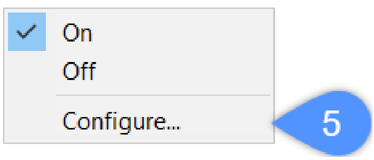
3 Select edge

Note: You can also right-click the **HKA** button (4) in the Status Bar to toggle On/Off the Hotkey Assistant widget.

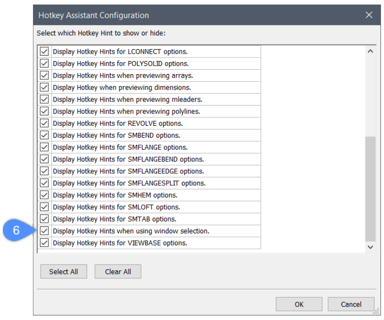


2.3.7.2 Deactivating/reactivating the widget for certain commands/actions

If the widget has been deactivated for a specific command or action (selecting, dragging, editing), you can reactivate it via the **Hotkey Assistant Configuration** dialog box. You can use the same dialog box to deactivate the widget for these commands/actions. To open the **Hotkey Assistant Configuration** dialog box, right-click the **HKA** button in the Status Bar and click the **Configure...** option (5).



Make sure that the Hotkey Hint options are enabled for the desired commands (6).



2.3.8 The Command line

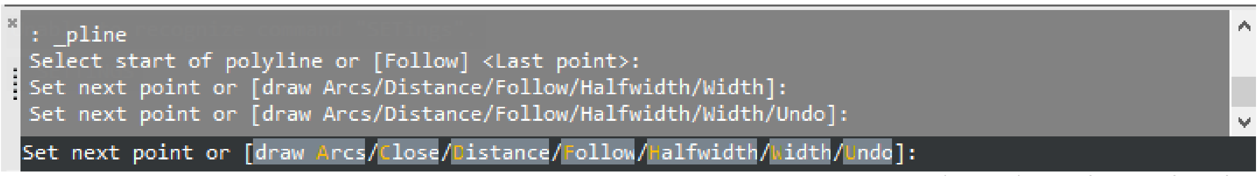
To display the Command line, do one of the following:

- Launch theCOMMANDLINEcommand.
- Use the Shift + F2 keyboard shortcut.
- Move the cursor over a docked toolbar, the ribbon or a panel icon, then right-click and choose**Panels** > **Command line** in the context menu.
- Double-click the far left side of the**Status** bar.

2.3.8.1 Setting the Command line state

The Command line can either be docked or floating.

By default, the Command line is docked at the bottom of the BricsCAD application window:



To modify the height of the Command line:

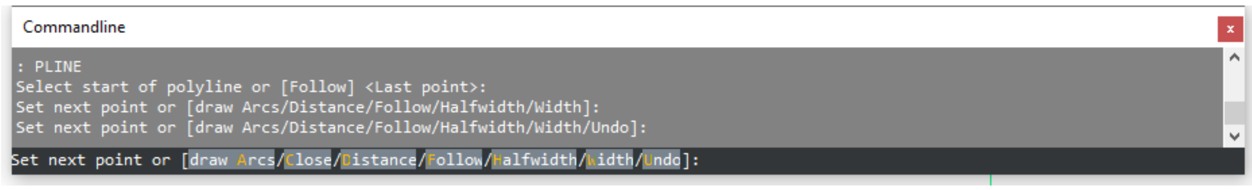
1 Place the cursor over the top edge of the Command line.

2 Press and hold the left mouse button when a double arrow appears.

3 Move the mouse.

Click the**Close**(x) button to close the Command line.

Move the cursor over the dotted left hand side, then press and hold the left mouse button to make the Command line floating:



When the Command line is floating, set the value of theCMDLINEUSEMINIFRAME (command line use new floating frame) system variable = 1 to minimize the Command line to a single line. When executing a command, a number of prompt lines are displayed temporarily and transparently on the graphic screen:

```
Select start of polyline or [Follow] <Last point>:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width/Undo]:
Set next point or [draw Arcs/Close/Distance/Follow/Halfwidth/Width/Undo]:
```

The number of prompt lines is set by the CLIPROMPTLINES system variable.

Click the up arrow at the right hand side to display the **Prompt History** window.

When you move the Command line mini-frame near the bottom of the graphic screen, it will stick there.

Note:

- You can browse the command history with the up arrow and down arrow keys of the keyboard. Press Enter to restart a command.
- When the Command line is closed, command options and keyboard entries are displayed in the **Status** bar.
- Press the F2 function key to toggle the**Command History**window.

2.3.8.2 Launching a command in the Command line

If the **Enable** flag of theAUTOCOMPLETEMODEsystem variable is set, command entries are completed automatically as you type.

All commands and/or system variables and/or user preferences that contain the current character string are displayed in the**Suggestion**list.

Do one of the following:

- Press Enter to choose the highlighted command.
- Use the down or up arrow key to choose a command, then press Enter.
- Click a command in the list.
- Complete the command name, then press Enter.

2.3.8.3 Accessing the input history

Use the Up Arrow and Down Arrow keys at the Command prompt or at an input prompt to access and reuse the history of recent commands input.

Note: Markers for recent input of point locations are displayed in the drawing area. The markers are controlled by SNAPMARKERSIZE, SNAPMARKERCOLOR, and SNAPMARKERTHICKNESS system variables.

Remember: The input history can include the last 12 items from the current command (from the current and previous calls) and the last 12 items from other previous commands.

2.3.8.4 Setting the Command line properties

1 Open the **Settings** dialog box.

2 Go to **Program Options** > **User Preferences** > **Command Line**.

2.3.9 The Status Bar

To display the **Status Bar**, do one of the following:

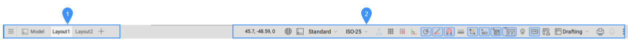
- Move the cursor over a docked toolbar, the ribbon or a panel icon, then right-click and choose **Status Bar** in the context menu.
- Press the **Shift + F3** keyboard shortcut.

The **Status Bar** closes if it was open, and vice versa.

Note: By default, the new **Status Bar** is enabled.

2.3.9.1 The new Status Bar

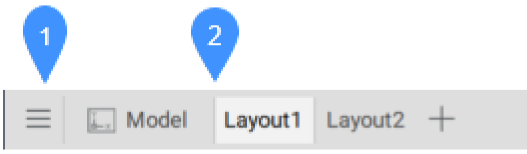
By default, the new **Status Bar** is enabled (USENEWSTATUSBAR system variable value is On). It displays the Model and Layout tabs (1) and the **Status Bar** section (2) in one line.



1 Model and Layouts tabs

2 Status Bar section

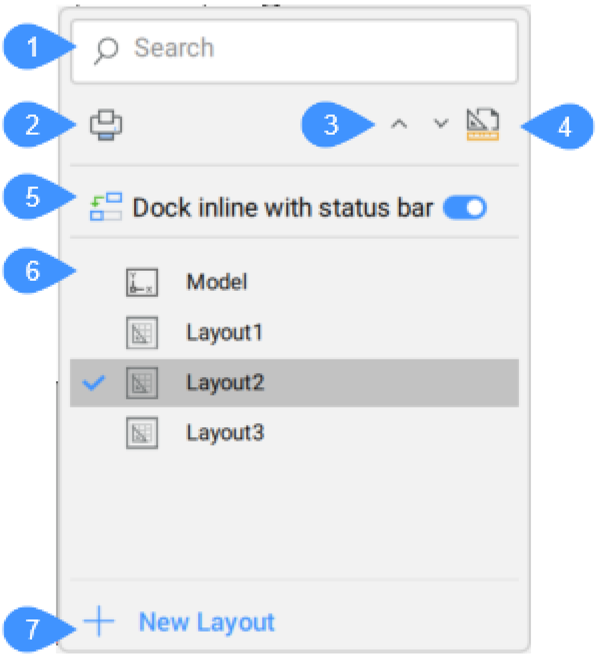
Model and Layouts tabs
Displays the Model and Layout tabs.



1 Manage Model and Layouts menu

2 Model and Layout tabs

Manage model and layouts menu
Opens the manage model and layouts menu that allows you to create, copy and delete layouts.



1 Search

2 Publish

3 Move Up and down buttons

4 Page setup

5 Dock inline with status bar button

6 Model and layouts list

7 New Layout button

Search

Searches for a layout name and displays only the searched layout. This is useful when a drawing contains many layouts.

Publish

Opens the **Publish** dialog box.

Move Up and down buttons

Moves the selected layout up or down in the list.

Page setup

Opens the **Page Setup** dialog box.

Dock inline with status bar button

If enabled, the Model and Layout tabs are inline with the **Status bar**, to leave more space for the drawing area. If disabled, the Model and Layout tabs are placed above the **Status bar**.

Model and layouts list

Lists the Model and available Layouts.

A context menu opens when you double-click a layout name, with the following options: **Rename Layout**, **Copy Layout**, **Publish**, and **Delete**.

New Layout button

Adds a new layout and adds it at the end of the list.

Note: The LAYOUTMANAGER command opens the **Layout Manager** dialog box that allows you to create, copy and delete layouts.

Model and Layout tabs

Displays the Model and Layout tabs. Right-click a layout to open a context menu with the following options:

New Layout

Creates a new layout and adds it at the end of the tabs list.

From template

Creates a new layout from a template file chosen from the **Select Template From File** dialog box.

Delete

Deletes the layout.

Rename

Renames the layout.

Copy

Makes a copy of the layout and adds it after the layout in the tabs list.

Move left

Moves the layout to the left in the tabs list.

Move right

Moves the layout to the right in the tabs list.

Page setup

Opens the **Page Setup** dialog box.

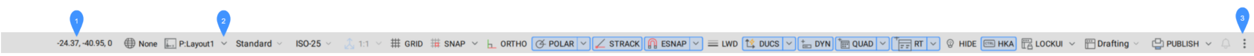
Print

Opens the **Print** dialog box.

Export

Exports the layout to a specified drawing chosen from the **Export Layout to drawing** dialog box.

Status Bar section



1 Visible field

2 Drop-down button

3 Status Bar menu

Click a field in the **Status Bar** (1) to toggle it ON/OFF. Right-click a field or click the drop-down button (2) to display a context menu or settings for that field. Hover over a field to display the name and its description. For the narrow BricsCAD windows, fields are displayed as icons.

Visible fields

Displays the visible fields of the **Status Bar**. Click the field to toggle it ON/OFF. Right-click some of the fields to open a context menu with the **Setting** or **Customize** options.

Drop-down button

Displays a pop-up list of options for the field.

Status Bar menu

Displays the **Configure Status Bar** list of available fields for the **Status Bar**. Click an item in the list to toggle the display of the corresponding field in the **Status Bar**. The position of a field in the Configure Status Bar list corresponds to the position of the field in the **Status Bar**. Customization and settings options are available for each field in the **Status Bar**, either from the drop-down button or by right-clicking.

2.3.9.2 Available fields in the Status Bar

Cursor Coordinate Values

Displays the coordinates of the current cursor position (depending on the value of the COORDS system variable).

Click the field in the **Status Bar** to open a context menu. It allows you to change the linear units, and coordinate settings.

GIS Coordinate System

Displays the name of the current GIS coordinate system.

Paper/Model

Allows you to switch between Model Space and Paper Space or viewports or layouts.

- When working in **Model Space**, the field reads:

  Tile (= Model space with tiled viewports).

  Click to switch to the most recently opened paper space layout.
- When working in a layout, the field reads:
- **M:Layout**: Model space with floating viewports

  or

  P:Layout: Paper space

  Click to toggle between paper space and model space with floating viewports.

Current Layer

Displays the name of the current layer.

- Click to open a pop-up list with all available layers.
- Click to select the current layer in the pop-up list.
- Click the buttons to toggle each layer property.

Current Color

Displays the current color.

- Click to open a pop-up list with colors options.
- Click to select the current color in the pop-up list.
- Click the **Select color** option to open the **Color** dialog box.

Current Linetype

Displays the name of the current linetype.

- Click to open a pop-up list with the linetypes options.
- Click to select the current linetype in the pop-up list.

- Click the **Load** option to open the **Load Linetypes** dialog box.
- Click the **Properties** option to open the **Drawing Explorer** > **Linetypes** dialog box.

Current Text Style
Displays the name of the current text style.

- Click to open a pop-up list with the text styles options.
- Click to select the current text style in the pop-up list.
- Click the **Properties** option to open the **Drawing Explorer** > **Text Styles** dialog box.

Current Dimension Style
Displays the name of the current dimension style.

- Click to open a pop-up list with the dimension styles options.
- Then, click to select the current dimension style in the pop-up list.
- Also, click the **Properties** option to open the **Drawing Explorer** > **Dimension Styles** dialog box.

Grid
Indicates the status of the GRIDMODE system variable.

- Click to toggle the display of grid in the current viewport.
- Right-click and choose **Settings** in the context menu to edit the **Snap and Grid** settings.

Snap
Indicates the status of the SNAPMODE system variable.

- Click the field to toggle SNAPMODE **On**/**Off**.
- Click the drop-down button and click to select the current SNAPTYPE system variable option in the pop-up list.
- Click the drop-down button and choose **Settings** in the pop-up list to edit the **Snap and Grid** settings.

Dynamic Input (DYN)
Indicates the status of the DYNMODE system variable.

- Click to toggle the display of Dynamic Dimensions **On**/**Off**.

Ortho
Indicates the status of the ORTHOMODE system variable. Turning on **Ortho** automatically disables **Polar**.

- Click to toggle ORTHOMODE **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the ORTHOMODE system variable.

Polar
Toggles the **Polar Tracking** option of the AUTOSNAP system variable.

- Click to toggle Polar Tracking **On**/**Off**. Turning on **Polar** automatically disables **Ortho**.
- Click the drop-down button and click to select the current POLARANG system variable options in the pop-up list.
- Click the drop-down button and choose **Settings** in the pop-up list to edit the **Polar Tracking** settings.

STrack
Indicates the status of the **Entity Snap Tracking** option of the AUTOSNAP system variable.

- Click to toggle Entity Snap Tracking **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Snap Tracking** settings.

ESnap

Indicates the status of the **Entity Snap** option of the OSMODE system variable.

- Click to toggle Entity Snaps **On**/**Off**.
- Click the drop-down button to open the **Entity Snapping Settings** pop-up list and click to enable the Entity Snaps options.
- Choose **Settings** in the pop-up list to edit the **2D and 3D Entity Snaps** settings.

Lineweight (LWT)

Indicates the status of the LWDISPLAY system variable.

- Click to toggle the display of line weights **On**/**Off**.

Selection Cycling

Indicates the status of the SELECTIONCYCLING system variable.

- Click to toggle the display of Selection Cycling **On**/**Off**.
- Click the drop-down button and click to select the current SELECTIONCYCLING system variable options in the pop-up list.
- Click the drop-down button and choose **Settings** in the pop-up list to edit the **Selection cycling** settings.

Dynamic UCS (DUCS)

Indicates the status of the UCSDETECT system variable.

- Click to toggle the UCSDETECT system variable **On**/**Off**.
- Click the drop-down button and click to select the supported entity types in the pop-up list:
-
    - **DUCS on Faces of 3D Solids**: 3D solid faces and regions.
    - **DUCS on Other Entities**: 2D linear entities.
- Click the drop-down button and choose **Settings** in the pop-up list to edit the **UCS detect** settings.

AutoScale

Indicates the status of the ANNOAUTOSCALE system variable. When **On**, annotative entities are synchronized with the current annotation scale. Click the field to toggle ANNOAUTOSCALE **On**/**Off**.

Annotation Scale

Displays the value of the CANNOSCALE (= current annotation scale) system variable. Click the drop-down button to open the pop-up list:

- Select an annotation scale in the list.
- **Custom**: opens the **Edit Scale List** dialog box.
- **Hide Xref scales**: lists annotation scales of the current drawing only.

Annotation monitor

Indicates the status of the ANNOMONITOR system variable.

- Click to toggle the ANNOMONITOR **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Annotation monitor** settings.

Viewport Lock (VPLOCK)

Displays the status of **Display Locked** property of the current viewport or the selected viewport(s). Click the VPLOCK field to toggle the property **On**/**Off**.

The VPLOCK field only appears when one or more paper space viewports are selected or when a viewport is active (Model space with floating viewports).

Viewport Standard Scale
Allows you to conveniently set the scale of the selected layout viewport relative to paper space.

Tablet
Initializes the use of a drawing tablet.
Right-click and choose **Settings** in the context menu to edit the **Tablet mode** settings.

Quad
Indicates the status of the QUADDISPLAY system variable.

- Click to toggle the display of the Quad **On**/**Off**.
- Click the drop-down button and click to select an option in the pop-up list.
-
  - **Show Quad on Hover**
  - **Show Quad on Select**
  - **Show Quad on Right Click**
  - **Suppress Quad on Hover When Entities are Selected**
  - **On/Off**
  - **Customize Quad Items...**: Displays the **Customize** > **Quad** dialog box
  - **Settings...**: Displays the **Settings** - **Quad display** dialog box

Rollover tips (RT)
Indicates the value of the ROLLOVERTIPS system variable.

- Click to toggle the display of rollover tips.
- Click the drop-down button and click to select the available rollover tips display options in the pop-up list:
-
  - **Show Rollover when Hovering Entities**
  - **No Rollover tips**
  - **Show Block Parameters**
- Click the drop-down button and choose **Customize...** in the pop-up list to open the **Customize** > **Properties** dialog box. Here you can define which properties are displayed in the **Rollover Tips** for each entity type.

Hide/Isolate Entities
Highlights only when there are hidden objects in the drawing.

- Click the field to open the options:
  - **Isolate**: launches the ISOLATEOBJECTS command.
  - **Hide**: launches the HIDEOBJECTS command.
  - **Show All**: launches the UNISOLATEOBJECTS command.

Hotkey Assistant (HKA)
Indicates the value of the HOTKEYASSISTANT system variable. When switched **On**, an assistant widget at the bottom of the screen reveals options available by pressing the Ctrl key.

- Click to toggle the display of the HKA widget **On**/**Off**.

- Right-click and choose **Configure** option in the context menu to display the **Hotkey Assistant Configuration** dialog box. Select the commands for which you want the HKA widget to display when HOTKEYASSISTANT = ON.

Current Workspace

Displays the name of the current workspace.

- Click to set the value of the WSCURRENT system variable in the Command line.
- Click and choose the current workspace in the pop-up list.
- Choose **Customize** in the pop-up list to open the **Customize** > **Workspaces** dialog box.

Lock User Interface (LOCKUI)

Allows you to lock the location and size of toolbars and dockable panels.

- Click to toggle the lock **On**/**Off**.
- Click the drop-down button to display a pop-up list.
-
  - Select the UI items to lock: toolbars, panels or both.
  - Choose **Settings...** to set the value of the LOCKUI system variable in the **Settings** dialog box.

Publish Monitor

Manages the publishing process. The button is highlighted while publish is in progress.
Click the drop-down button to display the pop-up list:

- **Cancel the job**: cancels the ongoing publish job.
- **Publish**: opens the **Publish** dialog box.
- **Show publish log**: opens the **Publish Output** dialog box.

Show Notifications

Toggles the displaying of the notifications.

2.3.9.3 The old Status bar

Setting the USENEWSTATUSBAR system variable to Off returns the old **Status Bar**.

1 Status field

2 Field List button

Status field

- Displays the status of the software.
- When the cursor is in a menu or on a toolbar, it gives a brief description of the tool or menu item under the cursor.
- When the command window is closed, it displays the command prompts, tool options and keyboard entry.
- Double-click the status field to open or close the Command line.

Field List button

Click the arrow button (▼) to display a list of available **Status Bar** fields. Click an item in the list to toggle the display of the corresponding field in the **Status Bar**.

Tip: Right-click a field to open a context menu with more options.

2.3.10 Menus

The menu bar is hidden in all workspaces by default.

The content of the menu is defined in the current workspace.

2.3.10.1 Controlling the display of the menu bar

1. Move the cursor over a toolbar or ribbon panel.
2. Right click and choose **Menu Bar** in the context menu.

2.3.11 Toolbars

The availability of a command depends on the license level (**Shape** - **Lite** - **Pro** - **Mechanical** - **BIM** - **Ultimate**) as indicated in the command articles in the Command Reference.

2.3.11.1 To set the icon size

You can choose between small (16 x 16), large (32 x 32) or extra large (64 x 64) icons.

The icon size is saved in the TOOLBUTTONSIZE system variable in the current Workspace.

- Move the cursor over a toolbar, then right-click.
- Choose **Toolbar size**, then select either **Small Icons**, **Large Icons** or **Extra Large Icons**.

2.3.11.2 Toolbar Flyouts

On some tool buttons, a small black arrow at the bottom right corner of the icon indicates that a flyout is available. A flyout holds a group of related tools. Press and hold the left mouse button to expand the flyout and choose one of the flyout tools. The tool that was last chosen remains visible in the collapsed toolbar.

Each of the flyouts can be opened as a separate toolbar.

To tear off a flyout toolbar:

1. Press and hold the left mouse button to drag the visible icon of a flyout toolbar into the drawing area.
2. Release the left mouse button to place the toolbar.

2.3.11.3 Opening a toolbar

1. In **Windows**: Hover over a toolbar or a blank portion of the toolbar area, then right-click.

   In **Linux**/**Mac**: Hover over a blank portion of the menu bar, status bar or toolbar area, then right-click.

   A context menu appears.
2. In the context menu, choose **Toolbars**, then choose the Main CUI file (e.g. BRICSCAD) or one of the Partial CUI files (if any).

   A list of available toolbars in the selected CUI file is displayed.

   Toolbars that are already open, are ticked.

   If the list does not fit on the screen, go to **More** at the bottom of the context menu.

   Some related toolbars are grouped in a flyout menu, e.g. 3D.

3 Click a toolbar on the context menu.

4 (option) To dock a toolbar, drag the toolbar to the top, bottom, left or right edge of the BricsCAD application window.

Note:

- If you select a toolbar that was already open, it will be closed.
- To prevent a toolbar from docking, press and hold the Ctrl key while dragging the toolbar.
- Drag a toolbar by the dotted grip at its left edge (when docked horizontally) or top edge (when docked vertically) to undock.
- To dock a toolbar vertically in **Linux**, the **Position** property of the toolbar must be set to **Left** or **Right**.
- The LOCKUI system variable allows you to lock the location and size of toolbars and dockable panels.
- Click the **LOCKUI** field in the **Status** bar to toggle the locking of the user interface.

2.3.12 The ribbon

The ribbon is a set of tabbed toolbars, filled with tool buttons and other graphical control elements, grouped by functionality.

The content of the ribbon depends on the current workspace.

The Drafting, Modeling, Mechanical and BIM workspaces use the ribbon by default.

Use the RIBBON command and the RIBBONCLOSE command to open or close the ribbon.

Features available in the new ribbon:

- Press and hold the left mouse button to open the drop-down menu on the flyout button.
- You can change the ribbon active tab using the mouse wheel.

Click the **Interface Settings** button (5) located at the top-right of the Ribbon to easily modify the interface via the **Interface Settings** dialog box (see the **Interface Settings dialog box** related article).

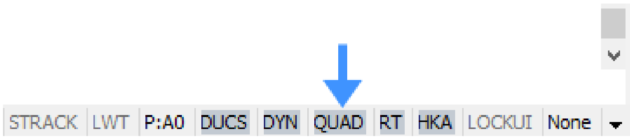
Note: **Classic** is the default layout.

2.3.13 The Quad basics

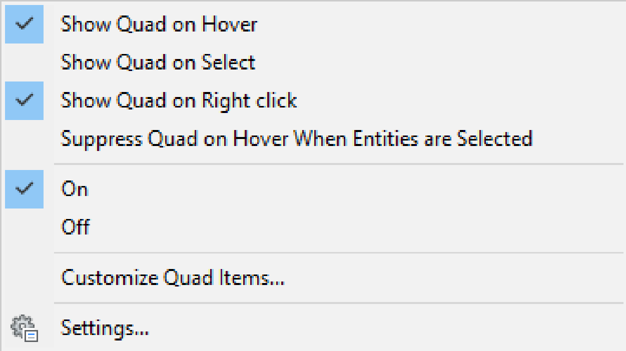
The **Quad** is a complete command system located directly at the mouse cursor.

When you move the cursor across the screen or when a selection set exists, the Quad detects the entity type and presents commands that work on this entity or selection of entities. When no entity is detected or selected, you can use the Quad to create entities, insert blocks, attach underlays, or open the **Settings** and **Drawing Explorer** dialog boxes.

- To toggle the display of the **Quad**, click the Quad field in the **Status Bar** at the bottom of the BricsCAD application window.

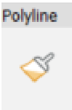


- Right-click the **Quad** field to define the Quad settings in a context menu.



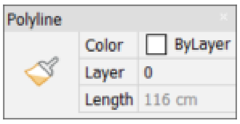
2.3.13.1 Basic procedure to edit entities using the Quad

1 Move the cursor over the entity. The entity highlights and the Quad displays the most recently used commands with this entity type.



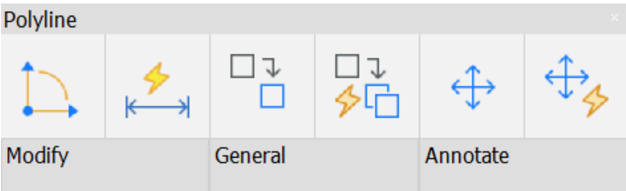
If the ROLLOVERTIPS system variable = ON, the properties of the entity under the cursor display in the Quad.

Note: When the SELECTIONPREVIEW system variable is Off, the **Display the Quad when the cursor hovers over an entity** option of the QUADDISPLAY system variable is ignored, and the Quad is not displayed.

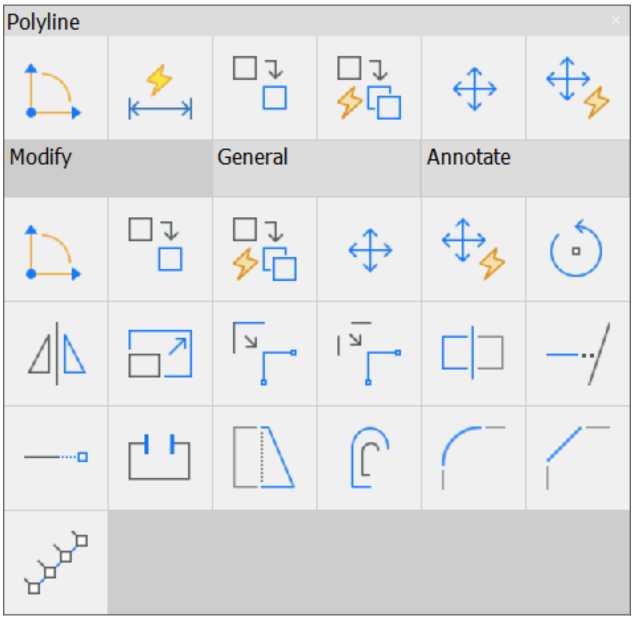


2 Move the cursor to the Quad to display the most recently used commands with this entity type.

A few command category tabs display at the bottom.



3 Move the cursor over the tabs to expand the command group under the cursor.



4 Click an icon to launch the command.

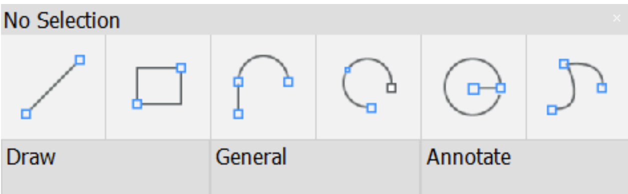
2.3.13.2 Using the No Selection Quad

Make sure the **Display the Quad on right click** option is set.

1 Do one of the following:

- Move the cursor to a void area of the drawing window and right-click.
- When the cursor is on an entity, hold down the right mouse button longer.

The **No Selection** Quad displays, showing the icons of the most recently used commands.



2 Do one of the following:

- Click one of the most recently used commands.
- Move the cursor over a command tab and select a command when it expands.

3 Execute the command.

2.3.14 The Quad cursor menu

The Quad cursor menu, aka the Quad is a context sensitive graphic menu which allows you to easily edit, select, show or hide and create entities. The content of the Quad is defined in the workspace and is fully customizable.

2.3.14.1 Activating the Quad

Activation of the Quad is controlled by the QUADDISPLAY system variable.

To toggle the Quad On/Off, do one of the following:

- Click the QUAD field in the **Status Bar**.
- Press the **F12** function key.

2.3.14.2 Setting the Quad display options

Right-click the **Quad** field in the **Status Bar**, then choose an option in the context menu.

- **Show Quad on Hover**.
- **Show Quad on Select**.
- **Show Quad on Right Click**: displays the **No Selection** Quad which allows you to create entities.
- **Suppress Quad on Hover When Entities are Selected**.

Note: When the SELECTIONPREVIEW system variable is Off, the **Display the Quad when the cursor hovers over an entity** option of the QUADDISPLAY system variable is ignored, and the Quad is not displayed.

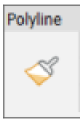
2.3.14.3 Editing the Quad settings

Right-click the **QUAD** field in the **Status Bar**, then choose **Settings** in the context menu.

The **Settings** dialog box opens, showing the **Quad settings** screen.

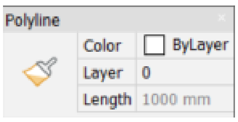
2.3.14.4 Using the Quad

When the cursor hovers over an entity, the entity is highlighted and the Quad displays the most recently used command with this entity type.

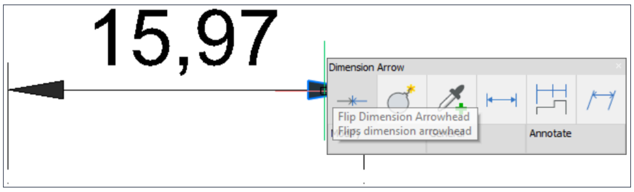


If multiple entities are under the cursor, repeatedly press the Tab key to highlight all entities one by one.

If Rollover Tips (RT) = On, the properties of the entity under the cursor are displayed next to the default command icon in the Quad.

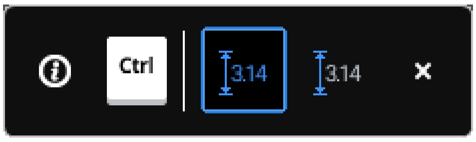


When the cursor hovers over a Dimension entity and press the Ctrl key, the sub entities arrow or text can be highlighted and then selected. The Quad displays the most appropriate commands for the selected sub-entity type. For example, when a Dimension Arrow is highlighted, the Quad will propose the **Flip Dimension Arrowhead** tool.



If Rollover Tips (RT) = On, the Dimension Arrow and Dimension Text properties are displayed next to the default command icon in the Quad.

Note: The Hotkey Assistant widget indicates the current selection mode, **Select Dimension entity** or **Select Dimension subent**:

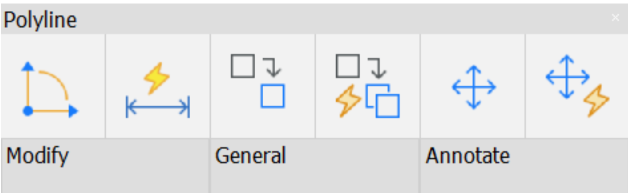


Launching the default command

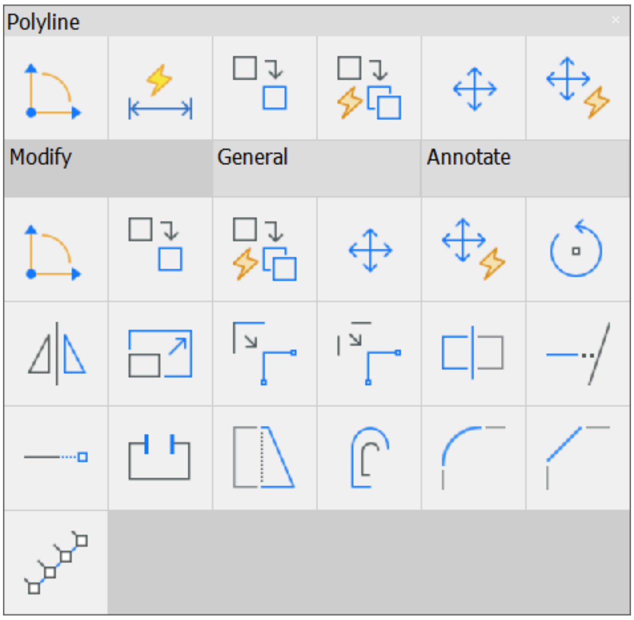
- If QUADCOMMANDLAUNCH = 1, right-click without entering the Quad to launch the command.
- If QUADCOMMANDLAUNCH = 0, click the command button.

Launching a different command

Move the cursor to the default command to expand the Quad: the 6 most recently used commands that apply to the selected entity are displayed. The number of icons that appear is set by the QUADWIDTH system variable (default = 6).



Other commands that apply to the selected entity sit in a number of tabs. Move the cursor over a tab to expand its content.



Click an icon to launch the command.

Note: Click the title bar of the Quad to go back and forth between the Quad commands and the Rollover Tips.

2.3.14.5 Selecting multiple entities

1 Move the cursor over the first entity, then click when the entity is highlighted.

1 The entity is selected and remains highlighted.

2 Move the cursor to another entity, then click when the entity highlights.

3 The entity is added to the selection set and remains highlighted.

4 (option) Repeat step 2 to select more entities.

5 (option) Move the cursor to a selected entity, then press and hold the Shift key and click.

6 The entity is removed from the selection set.

Note:

- When working with solids, the SELECTIONMODES system variable controls which subentities are detected: edges, faces and/or boundaries.
- Holding down the Ctrl key inverts the current selection mode.
- Enable detection of 3D solid edges
- Enable detection of 3D solid faces
- Enable boundary detection
- Repeatedly press the Tab key to highlight all (sub)entities under the cursor one by one.

2.3.14.6 Using the Quad to create entities

If flag 4 (= **Display the Quad on right-click**) of the QUADDISPLAY system variable is set, entity creation commands are available in the Quad (No Selection Quad). Right-click the **Quad** field in the **Status Bar** and check the option in the context menu.

When the cursor is on an entity, hold down the right mouse button longer to display the **No Selection** Quad.

The available entity creation commands in the Quad are:

- The most recently used entity creation commands.
- The number of available commands depends on the QUADWIDTH system variable.
- A number of command group tabs, depending on the current workspace.

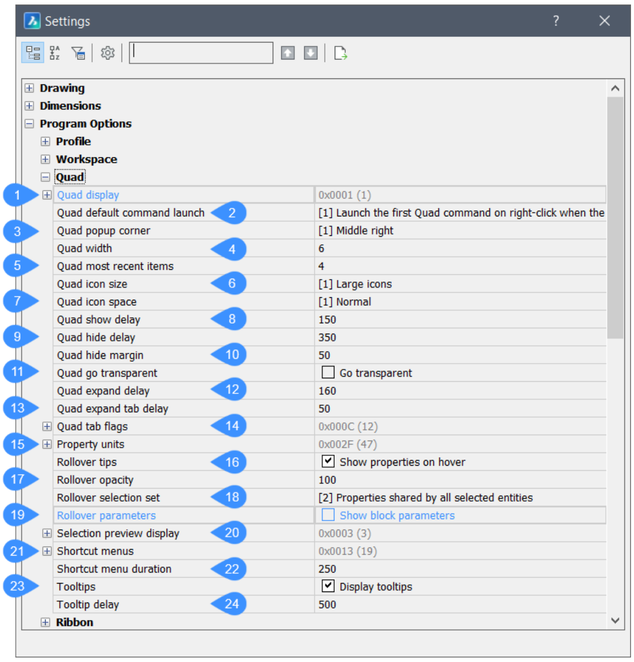
2.3.15 Detailed settings for the Quad cursor

This article contains the detailed setting for Quad Cursor.

To see how to start working with the Quad Cursor, see the article **The Quad cursor menu**.

2.3.15.1 Editing the Quad settings

Right click the QUAD field in the **Status bar**, then choose **Settings** in the context menu. The **Settings** dialog box opens, showing the **Quad** settings.



1 Quad display

2 Quad default command launch

3 Quad pop-up corner

4 Quad width

5 Quad most recent items

6 Quad icon size

7 Quad icon space

8 Quad show delay

9 Quad hide delay

10 Quad hide margin

11 Quad go transparent

12 Quad expand delay

13 Quad expand tab delay

14 Quad tab flags

15 Property units

16 Rollover tips

17 Rollover opacity

18 Rollover selection set

19 Rollover parameters

20 Selection preview display

21 Shortcut menus

22 Shortcut menu duration

23 Tooltips

24 Tooltip delay

2.3.15.2 Quad display

Determines when the Quad displays.

- Negative: Switches the Quad off.
- 1: When the cursor is over an entity. The entity under the cursor highlights.
- 2: When entities are selected.
- 4: On right click when no entities are selected and there is no entity under the cursor.
- 8: Suppress the Quad when hovering over entities that are not in the current selection set.

2.3.15.3 Quad default command launch

Enables launching the default quad command without requiring clicking the button.

- 0: The user must click the button.
- 1: The default command is launched by right clicking on the entity.

2.3.15.4 Quad pop-up corner

Controls the pop-up position of the Quad relative to the cursor position: **Upper right**, **Middle right** or **Lower right**.

2.3.15.5 Quad width

Defines the number of commands in a row. By default, this number is 6.

2.3.15.6 Quad most recent items

Defines the number of the most recent items (entities or operations) that are displayed in the menu.

2.3.15.7 Quad icon size

Sets the size of the buttons in the menu:

- [0] = Small buttons (16 x 16 pixels).
- [1] = Large buttons (32 x 32 pixels).
- [2] = Extra-large buttons (64 x 64 pixels).

2.3.15.8 Quad icon space

Defines the space around the Quad buttons: **narrow**, **normal** or **large**.

2.3.15.9 Quad show delay

Sets the time interval, in milliseconds, between the highlighting of the entity and the Quad display.

2.3.15.10 Quad hide delay

Sets the time interval, in milliseconds, to keep the Quad visible after the cursor is outside the Quad hide margin limit.

2.3.15.11 Quad hide margin

Sets the distance, in pixels, where the cursor can move away from the Quad and keep it visible. If the cursor moves outside this limit, the Quad is hidden.

2.3.15.12 Quad go transparent

Defines whether the Quad becomes transparent when the mouse moves outside the Quad, but within the Quad hide margin.

2.3.15.13 Quad expand delay

Sets the time interval, in milliseconds, to expand the Quad after mouse-enter.

2.3.15.14 Quad expand tab delay

Sets the time interval, in milliseconds, to expand a group tab after mouse-enter.

2.3.15.15 Quad tab flags

Sets some display properties of the Quad tabs.

- 1 - Fixed with tabs: the tabs will be displayed with the same width as the buttons above them
- 2 - Center tab labels: the labels of the tabs will be displayed center aligned
- 4 - Tab borders: the tabs will display a border between them
- 8 - Double tab height: the tabs will have two rows, allowing for the display of longer labels
- 16 - Show 3D Solid mass properties: displays the mass properties for **3d Solid** entities.

2.3.15.16 Property units

Displays the units for entity properties like:

- 1: length properties
- 2: area properties
- 4: volume properties
- 16: dynamic dimensions
- 32: mass properties

The units displayed are those set in the drawing properties. See UNITS command.

2.3.15.17 Rollover tips

Specifies whether entity properties display in the Quad while hovering. See the ROLLOVERTIPS system variable.

2.3.15.18 Rollover opacity

Specifies the degree of opacity when the Quad is in rollover state. Values between 10 (very transparent) and 100 (fully opaque) are accepted.

2.3.15.19 Rollover selection set

Controls the content of Rollover tips when multiple entities are selected.

2.3.15.20 Rollover parameters

Shows block parameters in the rollover tips.

2.3.15.21 Selection preview display

Controls in which cases objects are highlighted when the pick box cursor hovers over them.

- 1: when no commands are active
- 2: when a command prompts for entity selection

Note: When the SELECTIONPREVIEW system variable is Off:

- the **Display the Quad when the cursor hovers over an entity** option of the QUADDISPLAY system variable is ignored and the Quad is not displayed,
- the ROLLOVERTIPS system variable is ignored and entity properties are not displayed (the Quad is not displayed),
- the SELECTIONCYCLING system variable is ignored and no badge or selection dialog box is displayed (the Quad is not displayed).

2.3.15.22 Shortcut menus

Controls the status of the DEFAULT, EDIT and COMMAND shortcut menus. The value is stored as a bit code using the sum of the values of all selected options.

- 1: enable Default shortcut menus.
- 2: enable Edit mode shortcut menus.
- 4: enable Command mode shortcut menus (available whenever a command is active).
- 8: enable Command mode shortcut menus only when command options are currently available from the command line.
- 16: enable display of a shortcut menu when the right button on the pointing device is held down longer.
- 32: repeat command with a short right click when entities are selected, and Quad is not shown.

2.3.15.23 Shortcut menu duration

Specifies how long the right button on a pointing device must be pressed to display a shortcut menu.

2.3.15.24 Tooltips

Toggles the display of tooltips for buttons, tabs and menus.

2.3.15.25 Tooltip delay

Sets the delay time, in milliseconds, to display tooltips. If the value is negative, no tooltips display.

2.3.15.26 The Quad layout

The layout of the Quad cursor menu depends on:

- The entity type under the cursor or the current selection set.
- The current Workspace.
- Geometric events such as intersection or tangency.

When the cursor hovers over an entity, the entity highlights, and the Quad displays the most recently used command with this entity type.

When multiple entities are under the cursor, hit the TAB key to highlight all entities one by one.

Note: The rollover tips display if the SELECTIONPREVIEW system variable is set to 1 or 3, and the ROLLOVERTIPS system variable is set to 1 (ON).

The properties displayed for each entity type are specified in the Properties tab of the Customize dialog box. Select an entity type in the left pane, then check properties in the Rollover pane.

Moving the cursor over the first button Quad will expand the menu. Clicking an icon will launch the associated command. Clicking on the Quad title bar will minimize the Quad and display the Rollover tips.

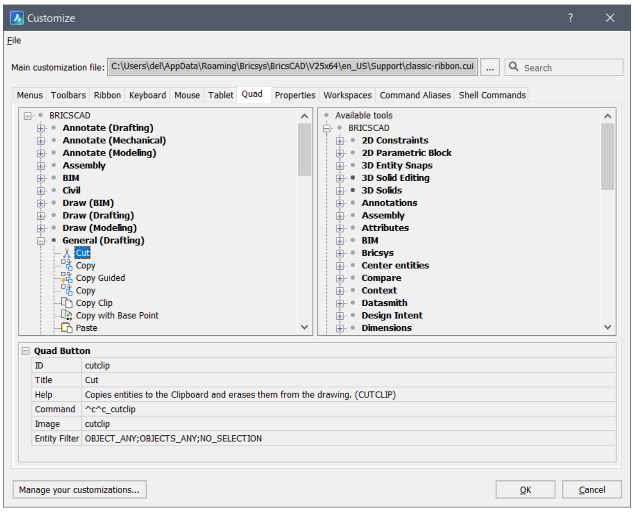
2.3.15.27 Customize Quad Items

To open the **Customize/Quad** dialog box, do one of the following:

- Right click the Quad field in the **Status Bar** and choose **Customize Quad items...** in the context menu.
- Run the CUSTOMIZE command, then select the Quad tab.

The Customize/Quad dialog box has two section tabs that are involved in customizing the Quad:

2.3.15.28 The Quad section



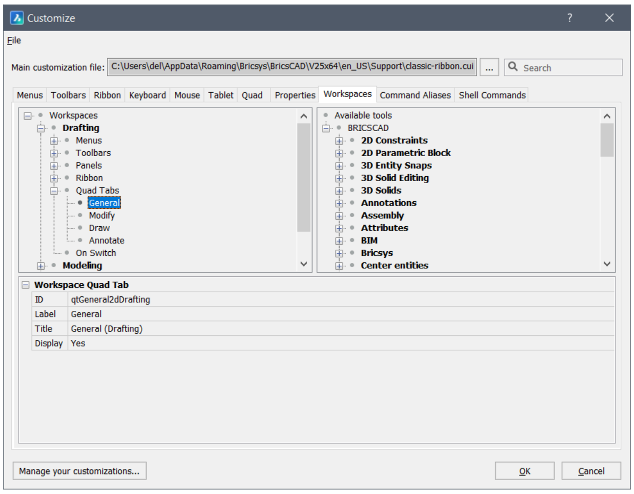
In this section, the name and content of all Quad Tabs can be specified, like the way Ribbon Tabs are customized. Quad Tabs can be created, copied, renamed and deleted.

Manage the content of a Quad Tab by dragging buttons from the right side of the dialog, or by using the right-click context menu to add, rename, move or delete Quad Buttons.

The right side of the dialog contains the full list of all buttons which can be shown in the quad. These include the tools defined in the CUI file, plus all internally defined Quad Buttons, which are listed under the new **Built-in** branch at the bottom. These built-in buttons are like the tools, only they cannot be edited: their name, icon, command string etc. are read-only.

To customize the Quad tool filter see the **Creating Custom Filters in QUAD Cursor** article.

2.3.15.29 The Workspaces section



This section defines which Quad Tabs will be available in the Quad, in which order, and in which workspace. In this interface, you can add/move/remove entire Quad Tabs. The contents of the Quad Tabs is not shown nor editable in this section, as this is done in the Quad section of the dialog. When a new Quad Tab is created in the Quad section, this Quad Tab is automatically added to the Quad in all workspaces.

To edit the Quad tabs of a workspace:

1 Open the **Customize** dialog box.

2 Click the **Workspaces** tab.

3 Expand the workspace you want to edit.

4 Expand the **Quad Tabs** list.

   Note: Tabs that are currently switched off display in gray.

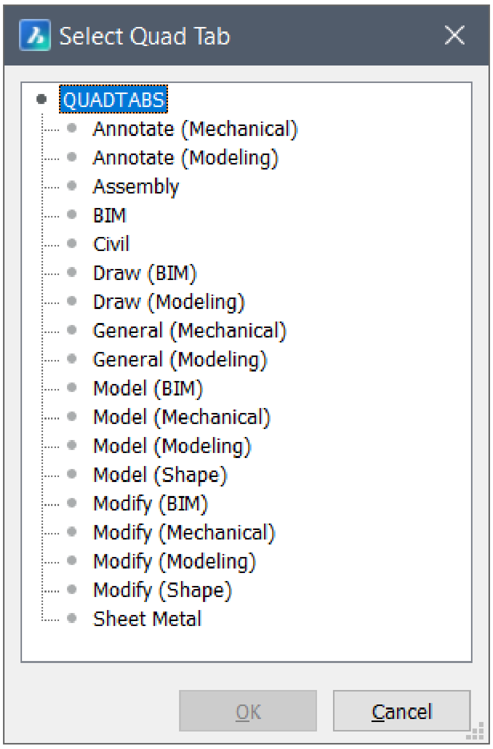
5 To show or hide a tab:

   - Select the tab.
   - Click the **Display** field in the tab properties at the bottom of the dialog box
   - Choose **Yes** or **No**.

6 To add a tab:

   - Right-click the **Quad Tabs** node and choose **Append quad tab** in the context menu to add a tab at the bottom of the tab list.
   - Select a tab node, then right-click and choose **Insert** in the context menu to add a tab above the selected tab.

The **Select Quad Tab** dialog box displays.

1 

On the **Select Quad Tab** dialog box select the tab you want to insert and click the **OK** button.

2 To remove a tab, select the tab, then right-click and choose **Delete** in the context menu.

3 To change the tab order: drag and drop the tabs in the list.

2.3.16 Right-click menus

Context menus or right-click menus display a list of commands next to the cursor when you right-click.

When the Quad is active, hold the right mouse button a bit longer (= long right-click).

The content of a context menu depends on the current entity or selection set.

When the Quad is not active and nothing is selected, a right-click repeats the most recent command. Hold down the right mouse button a bit longer to display the context menu.

2.3.17 The clean screen mode

In clean screen mode, you can maximize the drawing area by dismissing user interface elements, such as the menu bar, the ribbon, toolbars, panels, the Command line, status bar and document tabs.

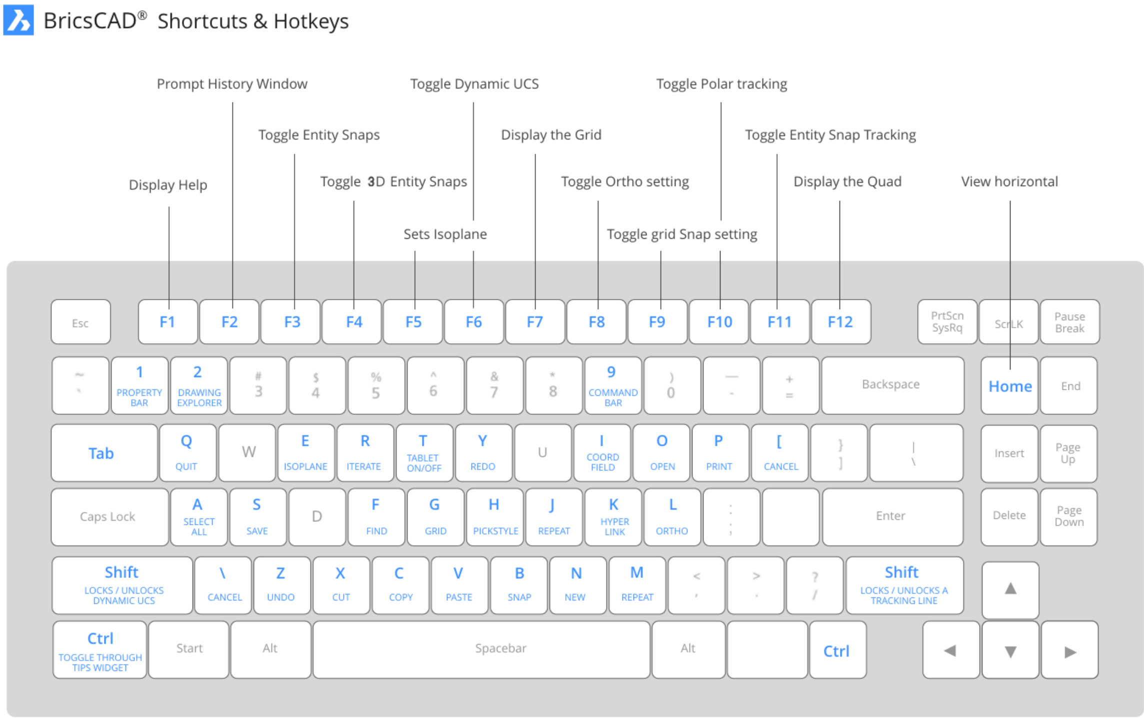
You can still use the Quad to launch commands.

Press and hold the Ctrl key and press the 0 (zero) key to toggle the clean screen mode on/off.

Note: Do not use the numeric keypad.

Note: On AZERTY keyboards: do not press the Shift key when hitting the 0 key.

2.3.18 Keyboard shortcuts



Keyboard shortcuts can be a great performance enhancer, especially when toggling settings.

The following keyboard shortcuts are available:

Keyboard	Description
F1	Opens the BricsCAD Help. Executes the HELP command.
F2	Toggles the display of the **Prompt History** window. See the GRAPHSCR command.
F3	Toggles the entity snaps ON/OFF. See the OSMODE setting.
F4	Toggles the 3D entity snaps ON/OFF. See the 3DOSMODE setting.
F5	Sets the isometric plane; toggles the value of the SNAPISOPAIR setting. Applies only when the SNAPSTYL setting = 1.

Keyboard	Description
F6	Toggles the Dynamic UCS feature.
F7	Toggles the display of the grid (see the GRIDMODE setting).
F8	Toggles the ORTHOMODE setting ON/OFF. Disables bitcode 8 of the AUTOSNAP setting, if set.
F9	Toggles Snap ON/OFF (see the SNAP command).
F10	Toggles the Polar Tracking setting. Sets the ORTHOMODE setting OFF, if ON. Toggles bitcode 8 of the AUTOSNAP setting ON/OFF.
F11	Toggles the Entity Snap Tracking setting. Toggles bitcode 16 of the AUTOSNAP setting ON/OFF.
F12	Toggles the display of the Quad cursor menu. See the QUADDISPLAY setting.
Alt + F8	Launches the VBARUN command.
Alt + F11	Launches the VBAIDE command.
Ctrl + 1	***Note:*** Do not use the numeric keypad. On AZERTY keyboards: do not use the Shift key when pressing the number key. Toggles the Properties panel.
Ctrl + 2	***Note:*** Do not use the numeric keypad. On AZERTY keyboards: do not use the Shift key when pressing the number key. Opens the **Drawing Explorer** dialog box.
Ctrl + 9	***Note:*** Do not use the numeric keypad. On AZERTY keyboards: do not use the Shift key when pressing the number key. Toggles the display of the Command line. Executes the COMMANDLINE command or the COMMANDLINEHIDE command.

Keyboard	Description
Ctrl + 0	***Note:*** Do not use the numeric keypad. On AZERTY keyboards: do not use the Shift key when pressing the number key. Toggles the display of user interface entities as specified by the CLEANSCREENOPTIONS setting. Executes the CLEANSCREENON command or the CLEANSCREENOFF command.
Ctrl + A	Selects all entities.
Ctrl + B	Toggles Snap ON/OFF (see the SNAP command).
Ctrl + C	Copies the selection to the clipboard (COPYCLIP command).
Ctrl + Shift + C	Copies the selection along with a base point (COPYBASE command).
Ctrl + E	Sets the isometric plane; iterates the value of the SNAPISOPAIR setting. Applies only when the SNAPSTYL setting = 1.
Ctrl + F	Launches the FIND command.
Ctrl + G	Toggles the display of the grid (GRIDMODE setting).
Ctrl + H	Toggles the value of the PICKSTYLE setting, which controls the selection of groups and associative hatches.
Ctrl + I	Sets the readout of the coordinate field in the Status bar (COORDS setting).
Ctrl + J	Repeats the previously issued command.
Ctrl + K	Attach a hyperlink to an object or modify an existing hyperlink (HYPERLINK command).
Ctrl + L	Toggles the ORTHO setting.
Ctrl + M	Repeats the previously issued command.
Ctrl + N	Creates a new drawing (NEW command).
Ctrl + O	Opens an existing drawing (OPEN command).
Ctrl + P	Prints the current drawing (PRINT command).

Keyboard	Description
Ctrl + Shift + P	Toggles the display of the Properties panel (PROPERTIES command and PROPERTIESCLOSE command).
Ctrl + Q	Quits the application; prompts to save changes (QUIT command).
Ctrl + R	Iterates through viewports.
Ctrl + S	Saves the current drawing (QSAVE command).
Ctrl + T	Turns the tablet on/off (TABLET command).
Ctrl + V	Pastes the contents of the Clipboard. (PASTECLIP command).
Ctrl + Alt + V	Pastes the contents of the Clipboard in a specified format (PASTESPEC command).
Ctrl + Shift + V	Pastes the contents of the Clipboard as a block (PASTEBLOCK command).
Ctrl + X	Cuts the selection and puts it on the clipboard (CUTCLIP command).
Ctrl + Y	Redoes the last action undone (REDO command).
Ctrl + Z	Undoes the last action (UNDO command).
Ctrl + [	Cancels the running command.
Ctrl + \	Cancels the running command.
Ctrl + Home	Creates or opens the Start page (GOTOSTART command).
Shift + F2	Toggles the display of the Command line (COMMANDLINE command and COMMANDLINEHIDE command).
Shift + F3	Turns the Status bar on/off (STATBAR command).
Shift + F4	Turns the Scroll Bars on/off (SCROLLBAR command).
Shift + F8	Launches the Visual Basic Project Manager (VBAMAN command).

Keyboard	Description
Shift + F11	Launches the VBA COM Add-In Manager (ADDINMAN command).
PgUp	Moves the view up.
PgDn	Moves the view down.
Shift + Left Arrow	Moves the view to the left.
Shift + Right Arrow	Moves the view to the right.
Shift + Up Arrow	Moves the view up. If (sub)entities are selected, the entities are nudged up (along the z-axis).
Shift + Down Arrow	Moves the view down. If (sub)entities are selected, the entities are nudged down (along the z-axis).
Ctrl + Left Arrow	Nudges the selected (sub)entities to the left (along the x-axis).
Ctrl + Right Arrow	Nudges the selected (sub)entities to the right (along the x-axis).
Ctrl + Up Arrow	Nudges the selected (sub)entities up (along the y-axis).
Ctrl + Down Arrow	Nudges the selected (sub)entities down (along the y-axis).
Tab	Navigates down into the autocomplete command list. ***Note:*** This is a cross-platform shortcut.
Tab + Shift	Navigates up into the autocomplete command list. ***Note:*** This is a cross-platform shortcut.

Keyboard	Description
Ctrl + N	Navigates down into the autocomplete command list. ***Note:*** This is a macOS only shortcut.
Ctrl + P	Navigates up into the autocomplete command list. ***Note:*** This is a macOS only shortcut.

2.3.19 Prompt menus

When executing a command, BricsCAD prompts you in the Command line or the Status bar. Optionally, command options are also available in a prompt menu.

2.3.19.1 Controlling the display of prompt menus

Whether prompt menus are displayed and where they are located is controlled through thePROMPTMENU system variable.

The PROMPTMENUFLAGS system variable controls the content and behavior of prompt menus.

Note: These system variables are saved in the registry and therefore apply to all drawings.

2.3.20 Rollover tips

The Rollover Tips feature displays a selection of entity properties in a pop-up box near the cursor when hovering over an entity or when an entity is selected. When multiple entities are selected, only their shared properties are displayed.

Which properties are available can be defined per entity type in the **Customize / Properties** dialog box.

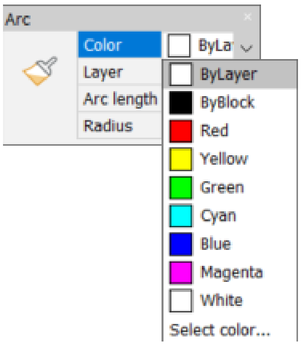
The availability of Rollover Tips is controlled through the ROLLOVERTIPS system variable and the SELECTIONPREVIEW system variable.

Rollover tips are displayed on hover when SELECTIONPREVIEW == 1 or 3 and ROLLOVERTIPS == 1.

Note: When the SELECTIONPREVIEW system variable is Off, the ROLLOVERTIPS system variable is ignored, and entity properties are not displayed when you hover the cursor over entities.

Click the **RT** field in the **Status Bar** to toggle Rollover Tips On/Off.

Select a property to edit it, much like you would do in the **Properties**panel.



When the Quad is On, you can go back and forth between the Quad commands and the Rollover Tips by clicking the title bar of the Quad/Rollover Tips feature.

2.4 Working in BricsCAD

2.4.1 Creating a drawing

There are several ways to create new drawings in BricsCAD:

- In the **Start** tab.
- Using the QNEW command.
- Using the NEW command.
- Using the NEWWIZ command.
- Double clicking a *.dwt file in a Windows Explorer window.

Note: A new drawing is always created as a copy of a template drawing. The default template is saved through the BASEFILE system variable.

2.4.2 Template drawings

A template drawing is a drawing file with file extension DWT.

It is used to create a new drawing that is a copy of the template drawing. Any drawing with extension DWT is considered a template drawing. The BricsCAD template drawings are located in the template folder(s). Use the TEMPLATEFOLDER command to open the folder. If you save your customized template drawings in the template folder, they are available in the Start page to create new drawings. Using properly prepared templates is essential for a productive use of BricsCAD.

A template drawing does not contain entities in model space, but contains settings according to the specific drawing and project requirements:

- Units: imperial or metric and units display.
- Layers.
- Dimension and leader styles.
- Text styles.
- Table styles.
- Multiple layouts, each with title block, paper size, printer, and plot style.
- Standard blocks for notes, tables, symbols, and other entities.

2.4.3 Drawing file format compatibility in BricsCAD

The tables below list the AutoCAD® drawing file (.dwg) and drawing exchange file (.dxf) versions that BricsCAD can OPEN and SAVE directly.

The **Version** column lists the drawing file schemas used in various AutoCAD® releases. For example, DWG 2013 is used in AutoCAD 2013 (R27) through AutoCAD 2017 (R31).

The **Internal version** is the information reported in the first six positions in the .dwg file header. See the **Pro Tip** below in the **Notes** section for more information.

DWG & DXF file versions compatible with BricsCAD *V25*:

Version	Internal version
DWG 2021 (R32)	AC1032
DWG 2018 (R32)	AC1032
DWG 2013 (R27)	AC1027
DWG 2010 (R24)	AC1024
DWG 2007 (R21)	AC1021
DWG 2004 (R18)	AC1018
DWG 2000 (R15)	AC1015
DWG R14 (R14)	AC1014
DWG R13 (R13)	AC1012
DWG R11/12 (R12)	AC1009

Version	Internal version
Binary DXF 2018 (R32)	AC1032
Binary DXF 2013 (R27)	AC1027

Version	Internal version
Binary DXF 2010 (R24)	AC1024
Binary DXF 2007 (R21)	AC1021
Binary DXF 2004 (R18)	AC1018
Binary DXF 2000 (R15)	AC1015
Binary DXF Release 14 (R14)	AC1014
Binary DXF Release 13 (R13)	AC1012
Binary DXF Release 11/12 (R12)	AC1009
Binary DXF Release 10 (R10)	AC1006

Version	Internal version
ASCII DXF 2018 (R32)	AC1032
ASCII DXF 2013 (R27)	AC1027
ASCII DXF 2010 (R24)	AC1024
ASCII DXF 2007 (R21)	AC1021
ASCII DXF 2004 (R18)	AC1018

Version	Internal version
ASCII DXF 2000 (R15)	AC1015
ASCII DXF Release 14 (R14)	AC1014
ASCII DXF Release 13 (R13)	AC1012
ASCII DXF Release 11/12 (R12)	AC1009
ASCII DXF Release 10 (R10)	AC1006
ASCII DXF Release 9 (R9)	AC1004

Note:

- The OPEN command opens a DWG or DXF file, directly into a new Document tab in BricsCAD.
- The IMPORT command imports the geometry from an (external) DWG or DXF file into the current drawing in your BricsCAD editing session. This is effectively an external "INSERT" operation; only geometry is loaded into the current drawing.
- **Pro Tip:** Even though DWG files are binary, and not general human-readable, you can open a drawing file in an ASCII text editor (like MS Notepad) and see the **Internal version** number (for example: AC1024) as the first six characters in the file. This can be helpful in determining file compatibility, or when working with files from an outside source.

2.4.4 Entity creation settings

New entities are created on the current layer, using the current entity creation settings.

You can define entity creation settings in the **General** section of the **Properties** panel.

By default, all entity creation settings are set **ByLayer**. This means that the corresponding setting of the current layer is used.

Layer properties can be set in the Layer section in the **Drawing Explorer** dialog box.

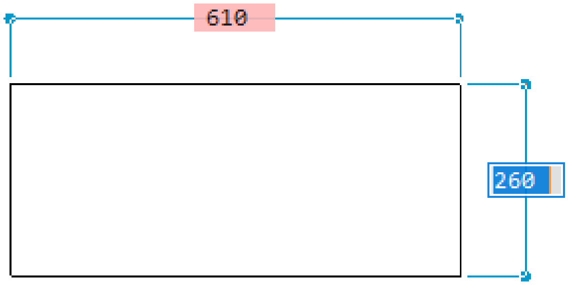
2.4.5 How to draw accurately

New entities are created in the XY-plane of the current coordinate system. By default, this is the World Coordinate System (WCS). All dimensions in a DWG file are expressed in drawing units (DU). The INSUNITS system variable (Insertion Units) sets the value of 1 DU.

For new drawings, INSUNITS system variable is defined in the template drawing that you selected to create the drawing. BricsCAD provides a series of template drawings for decimal and imperial units.

To draw accurately, BricsCAD provides several drawing aids:

- 2D entity snaps: you can use key points on existing 2D entities, such as endpoint, midpoint or center and the intersection. You can also draw perpendicularly, parallel, or tangentially to an entity.
- 3D entity snaps: you can use key points on 3D entities, such as solids.
- Dynamic dimensions. When dynamic dimensions are switched **ON**, you can enter distances and angles in dynamic dimension fields. When multiple dynamic dimension fields display, for example when drawing a rectangle, press the Tab key to switch to the next field. When you enter a value and press the Tab key, the field turns red, which indicates that the value is fixed.



- Polar tracking forces the drawing direction to specified angles with respect to the axes (X, Y, Z) of the current coordinate system.
- Entity snap tracking combines entity snaps and polar tracking.
- Coordinate input allows you to enter XYZ coordinates to specify points.
- Snap and grid. A grid is a set of evenly spaced lines or dots that serve as a visual distance reference. The cursor snaps to the grid points, which allows you to draw fast at fixed intervals.
- Drawing limits allows you to limit the drawing area, which prevents entity creation outside the drawing limits.
- Ortho mode forces the cursor to move parallel to either the X- or Y-axis of the current coordinate system.
- User coordinate systems. You can define named user coordinate systems (UCS) located anywhere in three-dimensional space. You can restore a named UCS at any time.
- Direct distance entry allows you to enter lengths, radii, diameters, and movement vectors.
- Nearest distance allows you to dynamically view and modify the distance between 2 selected entities or sub entities.
- The Manipulator is a versatile grip tool to swiftly move, rotate, scale and mirror selected entities.

2.4.6 Executing commands

In BricsCAD you can launch commands in the following ways:

- Type the command in the Command Line.

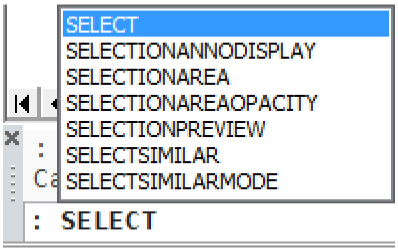
- Choose the command in a menu.
- Click the command tool button in a toolbar.
- Click the command icon in the Ribbon.
- Choose the command in the Quad.

2.4.6.1 Typing a command in the Command line

If you know the command name, you can enter it in the Command line. When the command line is closed, keyboard entry and command prompts display in the status bar.

If the **Enable** flag of the AUTOCOMPLETEMODE system variable is set, command names are automatically completed when you type.

All commands and/or system variables and/or user preferences that contain the current character string display in the suggestion list.



Do one of the following:

- Press Enter if the currently highlighted command displays.
- Click a command in the suggestion list.

You can launch some commands with a command alias. For example, you can launch the LINE command by entering L in the command prompt.

2.4.6.2 Choosing the command options

Many BricsCAD commands provide multiple options. These options appear in the Command line or status bar (when the Command line is closed) and optionally also in a prompt menu.

Do one of the following:

- Press Enter to choose the default option. The default option is located at the end of the command prompt between arrow brackets (<>).
- Click the option in the Command line.
- Click the option in the prompt menu.
- Type the capitalized letter of the option and press Enter or right-click.

2.4.7 Using experimental features

The Experimental features (EXP) is a module where new features (commands, panels, or dialog boxes) that are experimentally introduced in BricsCAD are centralized and easy to use under Experimental mode.

The Experimental mode is disabled by default, but can be enabled in the following ways:

- Enable it in the **Experimental new features** module on the **Start** page.
- Set the EXPERIMENTALMODE system variable to ON.
- Run the MANAGEEXPERIMENTALFEATURES command and press **Enable** button in the **Experimental Features** dialog box.

Enabling the Experimental mode gives you access to early features and provides you with the opportunity to help the BricsCAD team to define the direction of these features.

Important: You may experience bugs or performance issues in BricsCAD when the **Experimental mode** is enabled. We encourage you to report them. Experimental features can change or be removed from future versions.

Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

When the **Experimental mode** is disabled, the experimental features are available in BricsCAD, but do not work. When trying to access one of them, a message is displayed. For example, if you run the WORKPLANEACTIVATE command, the following message is displayed in the Command line: *Work Plane is an experimental feature that is currently not enabled. Use the MANAGEEXPERIMENTALFEATURES command to enable experimental features.*

2.4.8 View control

In BricsCAD you can control the display of your drawing in various ways:

- By using the mouse and keyboard
- By using View commands
- By using the Look From widget

The most frequently used view manipulations are:

Action	Mouse	Command	Look From
View Extents	Double-click the scroll wheel	**Zoom** > **Extents**	Clicking a trigger zone rotates the view and zooms to the extents of the drawing.
Zoom	Scroll wheel	**Zoom** > **InZoom** > **Out**	
Pan	Hold down the scroll wheel ***Note:*** When the MBUTTONPAN system variable = ON	Pan	

Action	Mouse	Command	Look From
Restore Plan View (Top View)		**Plan** > **Current UCS**	Click the center trigger zone

2.4.9 Selecting entities

Manipulation commands, such as MOVE, COPY and MIRROR prompt you to select one or more entities. If the PICKFIRST variable is ON, you can also compose the selection set before launching the command.

BricsCAD provides several selection methods. You can combine selection methods to compose a selection set.

2.4.9.1 Selecting entities one by one

1 Move the cursor over the entity.

1 The entity highlights.

The Quad appears and displays the most recently used command and a selection of the entity properties.

2 Do one of the following:

- Click to select the entity.
- Repeatedly press the Tab key to highlight obscured entities and click when the desired entity highlights.

3 Repeat the previous steps to select more entities or launch a command.

2.4.9.2 Selecting multiple entities

BricsCAD uses window selection methods to select multiple entities:

1 Click to define the corner of the selection rectangle.

2 Do one of the following:

- Move the cursor to the right. A blue rectangle is displayed. The **circle inside the rectangle** icon indicates that all entities completely inside the selection rectangle will be selected.
- Move the cursor to the left. A green rectangle is displayed. The **circle overlapping the rectangle** icon indicates that all entities that are either inside or that overlap the selection rectangle will be selected.

3 Click to define the opposite corner of the selection rectangle.

2.4.9.3 Removing entities from a selection set

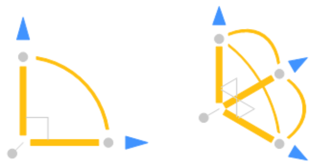
Hold down the Shift key when executing a selection method to remove entities from the current selection set.

Press the Esc key to clear the current selection set.

2.4.10 Using the Manipulator

The Manipulator is a versatile grip tool to swiftly move, rotate, scale and mirror selected entities.

If you hold down the Ctrl key during one of these actions, the selected entities are copied.

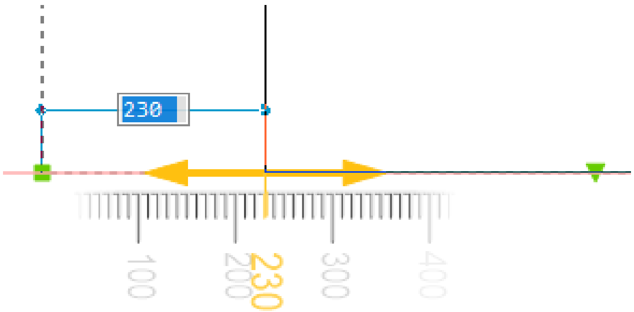


By default, the Manipulator displays when selecting an entity by a long click (keeping the left mouse button down a bit longer).

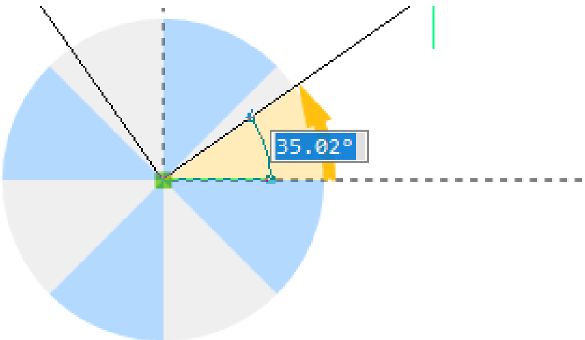
The anchor point of the Manipulator is located at the closest grip point, with respect to the selection point.

Place the cursor over one of the trigger zones, to start an action:

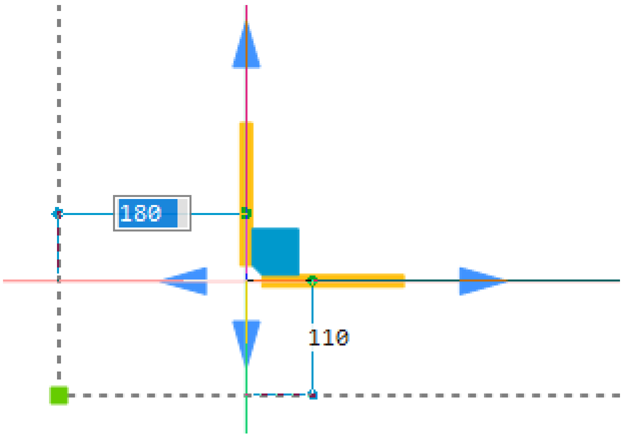
- Axes, to move along the axis:



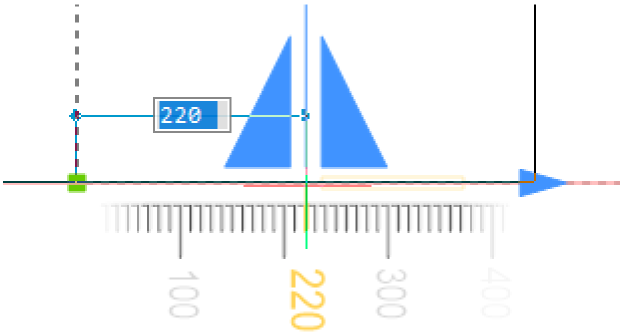
- Arcs, to rotate:



- Plane swatches (= squares at the anchor point), to move freely in the selected plane (XY, XZ or YZ):



- Arrow heads, to mirror:



Note: The step size of the ruler depends on the zoom factor. Zoom in to decrease the step size, zoom out to increase.

2.4.11 Working with the Structure panel

The **Structure** panel displays a structured tree view of the entire drawing content and enables you to easily work with it (filter, group and sort elements to easily select it and modify their properties).

BricsCAD provides six **Structure Tree Configurations** (**CST** files):

- **Default**: displays a layer-based structure.
- **Mechanical**: displays an entity-based structure.
- **BIM**: displays a BIM entity-based structure.
- **BIM - Compositions**: displays a BIM composition-based structure.
- **BIM - Profiles**: displays a BIM profile-based structure.
- **BIM - Types**: displays a BIM type-based structure.

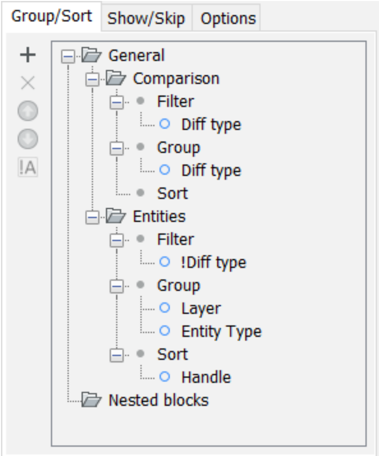
2.4.11.1 Common use cases

Delete all entities of a specific layer (default CST file)

1 Open the **Structure** panel ( ).

2 Click the drop-down menu button (⋮) and select the **default** configuration.

Note: To inspect the selected configuration click the **Configure structure tree** option in the drop-down menu.



3 In the Structure Tree expand the **Entities** node. All entities in the drawing are listed grouped first by the **Layer** property.

4 In the Structure Tree select the node with the desired layer name. All entities belonging to this layer are automatically selected in the Model space.

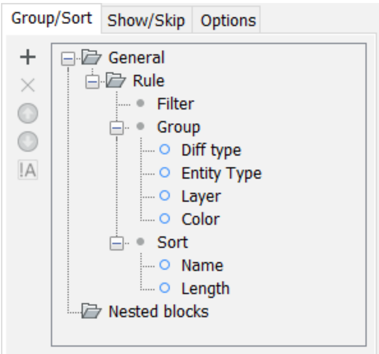
5 Press the **Delete** key. All layer's entities have been removed.

Delete all the lines in a drawing (mechanical CST file)

1 Open the **Structure** panel ( ).

2 Click the drop-down menu button (⋮) and select the **mechanical** configuration.

Note: To inspect the selected configuration click the **Configure structure tree** option in the drop-down menu.



3 In the Structure Tree expand the **Entities** node. All entities in the drawing are listed grouped first by the **Entity** property.

4 In the Structure Tree select the **Lines** node. All the lines in the drawing are automatically selected in the Model space.

5 Press the **Delete** key. All the lines in the drawing have been removed.

2.4.11.2 How to work with rules

The principle of the **Structure** panel is comparing values of entity properties. The rules for the comparison process are stored in **Structure Tree Configuration** files (CST). The entities are then grouped in a structure tree based on the comparison's results. Grouping entities in the structure tree allows you to easily select entities with the same property value, check whether everything is as expected, or make changes to their properties.

You can decide how the entities are displayed in the structure tree by selecting a **Structure Tree Configuration** or creating a new one via the **Configure Structure Tree** dialog box. This dialog box lets you add rules defined by filters with results organized by grouping and sorting criteria. You can choose any entity type property BricsCAD recognizes (CAD, BIM, Mechanical, etc.) and custom properties as filtering, grouping, or sorting criteria.

Note: The position of the rules, filters, grouping and sorting criteria in the **Structure Tree Configuration** is essential, as the structure is hierarchical. Use the **Move property up** () and **Move property down** () buttons in the **Configure Structure Tree** dialog box to change the position of the selected rule/property as needed.

Rules

Each rule is marked by a folder in the **Configure Structure Tree** dialog box.

All entities in the drawing try to fit under existing rules in top-down order by checking each rule's filters. Entities that do not fall under a specific rule's filters pass to the following rule in the structure to be analyzed by its filters.

When at least one entity falls under a rule, the **Structure** panel shows a corresponding node in the Structure Tree containing the rule's filtered entities. The Structure Tree displays these entities in the node in the order dictated by the rule's grouping and sorting criteria.

Filter

Filters control whether an entity will be listed under a certain rule. If the filter does not apply to the entity the entity is passed on to the next rule in the tree.

Note: Filtering is done by checking whether an entity has a certain property, not a property value. For example, if you filter on dimension style (select **Dim style** property as a filter), all entities with a dimension style will be included, meaning all dimensions. You cannot filter on **Dim style** is equal to ISO-25.

Filters are additive. To have new entities listed under the same rule, select the rule's **Filter** node, click the **Add rule or property button** (+) and select a different entity property (for example, if you have a **Dim style** property filter and add a **Radius** property filter, all dimensions, as well as circles and arcs, will be listed under that rule).

Note: In this case, the order in which you add the new criteria in the **Configure Structure Tree** dialog box does not matter.

Click the **Toggle negation** button on a selected filter to exclude the entities from being listed under the rule. The "!" sign appears in front of the filter's name.

If a rule has no filters, all entities other rules above it have not yet filtered fall under this rule. This is a catchall method for grouping and sorting the remaining unfiltered entities. Rules placed directly under a no-filter rule have no entities, as none is left to filter.

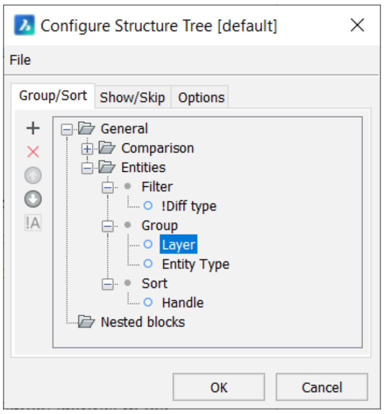
Group

A group defines how the entities filtered within a rule are displayed in the Structure Tree. The sequence of the items in the group deals with the nesting of Structure Tree's branches.

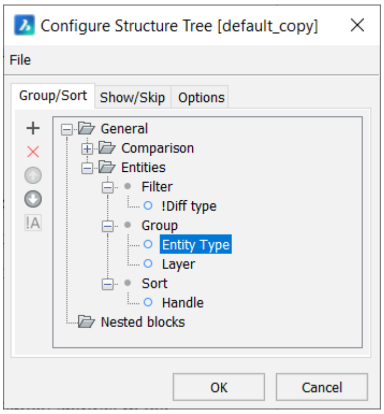
To add a property to a **Group** node, select the desired node, then click the **Add rule or property button** (+).

Two commonly used properties for grouping are **Layer** and **Entity Type**. To display entities in the Structure Tree:

- grouped by layer and sub-grouped by entity type, move the **Layer** property above the **Entity Type** property in the configuration tree.



- grouped by entity type and sub-grouped by layer, move the **Entity Type** property above the **Layer** property in the configuration tree.



When no entity properties are added as grouping criteria, the entities falling under the filter are shown in the structure tree as a flat list.

To delete a grouping property, select it and click the **Delete rule or property** button (×).

Sort

Sorting defines how entities are listed in the deepest branch.

When applying sorting, the entities within a group are labeled with the value of their sorting property and displayed in a flat list (no sub-nodes).

If more properties are added as sorting criteria, the first listed property that applies to a node's entities is the one used for labeling.

Predefined rules

There are two predefined rules in the **Structure** panel's **[default]** CST configuration, **Comparison** and **Entities**.

Comparison rule

This rule can have entities fall under it only when there is a DWG Compare session active.

If no entity has the property **Diff type**, then no entity is shown in this folder. In this case the **Comparison** node is not shown in the **Structure** panel, as there is no entity to display under this rule.

For more information, see the **DWGCOMPARE command** article.

Entities rule

By default, this rule has no filters; all entities that were not filtered by previous rules are shown here.

The entity properties used as grouping criteria are **Layer** and **Entity type**. Entities are grouped first by their layer property, then sub-grouped by their entity type.

2.4.11.3 Procedure: filter all dimension entities in a drawing

To have all dimensions in a drawing displayed under one node in the Structure Tree, do the following:

1 Right-click the name of your drawing in the **Structure** panel, then choose **Configure** in the context menu to open the **Configure Structure Tree** dialog box.

2 Go to **File** and choose **Save as...** option. Give a name to the new configuration file (for example, '**dim_filter**') and save it. The default folder for CST files is the **Support** folder: *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support*

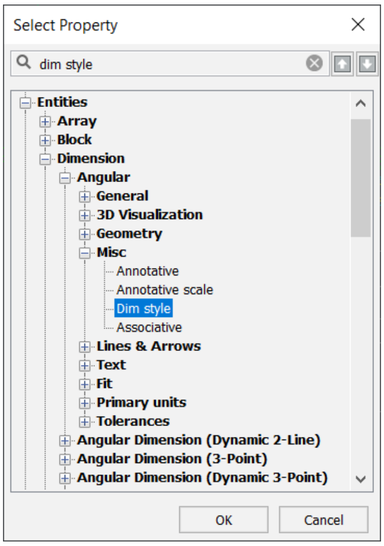
3 Go to **Group/Sort** tab and click the **Add rule or property** button. A new **Rule** is added at the end of the structure.

4 Click the new rule's name and rename it '**Dimensions**'.

   Note: Renaming the new rule with a distinct name is essential to have an actual filtering effect in the panel's Structure Tree.

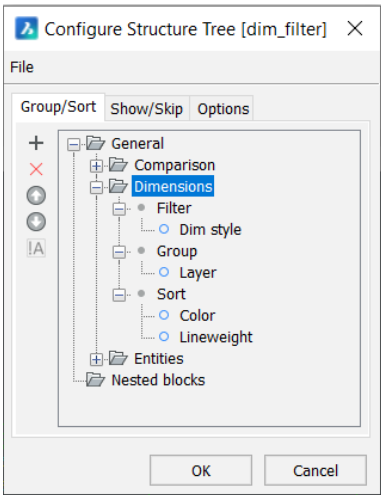
5 Click **Dimensions** rule's **Filter** node and click the **Add rule or property** button to display the **Select property** dialog box.

6 In the **Select property** dialog box's search field type 'dym style'. Use the **Find next** button () or press **Enter** to jump to the **Angular** dimension **Dym style** property.



7 Click **OK** to add the **Dym style** dimension property to the rule.

8 In the **Configure Structure Tree** dialog box select the **Dimensions** rule. Click the **Move property up** button until the **Dimensions** rule is placed above the **Entities** rule.



The Structure Tree's **Dimensions** node in the **Structure** panel now displays all the dimension entities in the drawing.

2.4.12 Recording and playing actions in BricsCAD

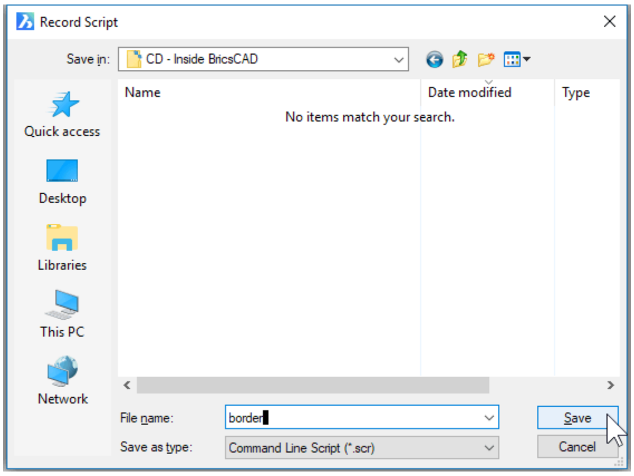
2.4.12.1 Commands

RECSCRIPT, STOPSCRIPT, SCRIPT

2.4.12.2 Recording actions in BricsCAD

1 In a new drawing, launch the RECSCRIPT command or from the **Tools** menu select **Record Script...**.

The **Record Script** dialog box opens.



2 Enter a file name for the script. For this tutorial, enter **border** and then click **Save**. The border.scr script file is created.

3 Notice that the **Record Script** dialog box is closed and the Command line prompt is available, and you can enter the commands you like.

For example:

a Launch the RECTANG command.

a
```
: RECTANG
Select first corner of rectangle or [Chamfer/Fillet/Rotated/Square/
Elevation/Thickness/Width of line/Area/Dimensions]: 0,0
Other corner of rectangle: 36,24
```

b Launch the ZOOM command.

c
```
: ZOOM
Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/
Scale (nx/nxp)/Window/OBject] <Scale (nX/nXP)>: E
```

Note: You can use the mouse to pick points in the drawing during commands while recording. BricsCAD records the pick points as x, y coordinates.

4 Launch the STOPSCRIPT command whenever you want to stop the recording.

2.4.12.3 Playing a script file

You can run the script with the SCRIPT command as follows:

1 Start a new drawing with the NEW command so you can see the effect of the script.

2 Enter the SCRIPT command.

3 The **Run Script** dialog box opens. Choose **border.scr** and click **Open**.

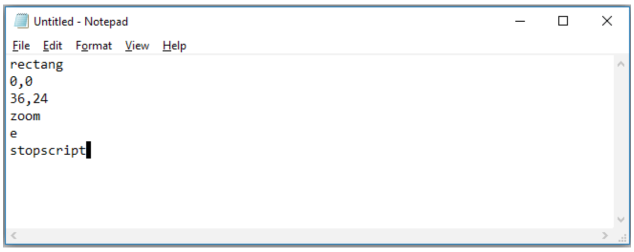
The script instantly draws the rectangle and then zooms the drawing to the extents of the newly drawn border.

2.4.12.4 Writing Scripts by Hand

While BricsCAD has commands for creating and running scripts, it has no command for editing them. If you want to change the coordinates used by the RECTANG command, you have to edit the script file with a text editor, such as Notepad in Windows, Text Edit in Linux, or TextEdit in macOS.

1 Open the border.scr file in a text editor.

1 Notice the commands and options that you entered during the script recording session:



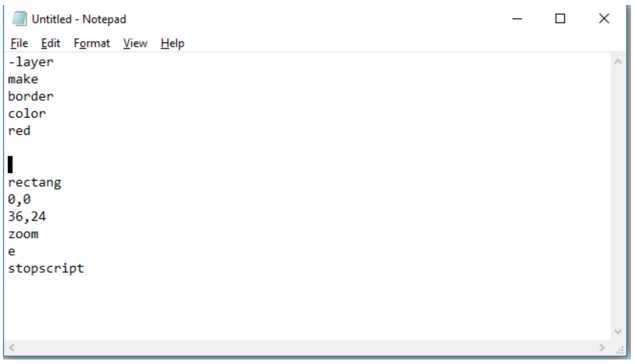
Change the commands as below:

a Change the size of the border to 18×24. Edit the "36,24" text, replacing it with "18,24".

b Add the command for placing the rectangle on a layer named "Border" and colored red:

c i Place the cursor in front of "rectang", then press **Enter** to make an empty line.
ii Enter the following text:
iii -layer
make
border
color
red
? <-- One blank line
? <-- A second blank line

a Make sure you include two blank lines. These act like pressing **Enter** during commands. The file should now look like this:



4 Save the file.

5 Return to BricsCAD and start a new drawing.

6 Use the SCRIPT command to test that the border.scr file is operating correctly. You should see a red rectangle.

2.4.12.5 Script Commands and Modifiers

There are four commands that relate specifically to scripts. In fact, these commands are of absolutely no use for any other purpose. In addition, BricsCAD has the RECSCRIPT command for recording scripts, as described earlier in this chapter.

The four basic commands are:

SCRIPT

The SCRIPT command performs double-duty: (1) it loads a script file and (2) immediately begins running it.

Note: Remember to turn off (set to 0) the FILEDIA system variable so that the prompts appear at the Command line, instead of the dialog box.

RSCRIPT

Short for "repeat script", this command reruns whatever script is currently loaded in BricsCAD. A great way to create infinite loops.

RESUME

This command resumes a paused script file. Pause a script file by pressing the **Backspace** key.

DELAY

To create a pause in a script file without human intervention, use the DELAY command along with a number. The number specifies the pause in milliseconds, where 1,000 milliseconds equal one second.

Note: The minimum delay is 1 millisecond. The maximum is 32767 milliseconds, which is just under 33 seconds.

The DELAY command is used in a script file to wait while a slide file is displayed or to slow down the script file enough for humans to watch the process.

2.4.12.6 SPECIAL CHARACTERS

In addition to the script-specific commands, there are some special characters and keys.

Enter -- (space)

The most important special characters are invisible: both the space and the carriage return (or end-of-line) are the equivalent to when you press the Space or Enter keys. In fact, both are interchangeable. But the tricky part is that they are invisible. Sometimes, writing a script requires a lot of blank spaces because the command requires pressing the ENTER key several times in a row. The ATTEDIT command is an excellent example:

```
; Edit the attributes one at a time:
attedit    1,2
```

How many spaces are there between attedit and the coordinates **1,2**?

For this reason, it is better to place one script item per line, like this:

```
; Edit the attributes one at a time:
attedit




1,2
```

Now it is easier to count the four spaces, since there is one per blank line.

Comment -- ;

The semicolon lets you insert comments in a script file.

Note: BricsCAD ignores anything following the semicolon.

Transparent -- '

Scripts can be run *transparently* during a command. Simply prefix the SCRIPT command with an apostrophe to run a script while another command is active, like this:

```
: LINE
Start of line: 'script
Script to run: filename
```

All four of BricsCAD's script-specific commands are transparent, even DELAY, which lets you create a delay during the operation of a command.

Pause -- Backspace

Represents the key mentioned earlier for pausing a script file.

Stop -- esc

Stops a script file dead in its tracks. Use the RSCRIPT command to start it up again from the beginning.

3. Managing drawings

3.1 Creating a drawing

A new drawing is always created as a copy of a template drawing. The default template is saved through the BASEFILE system variable.

In BricsCAD you can create new drawings:

- In the **Start** page.
- Using the QNEW command.
- Using the NEW command.
- Using the NEWWIZ command.
- Double-clicking a *.dwt file in a Windows Explorer window.

3.1.1 Specifying the template folder

1 Open the **Settings** dialog box.

2 Under **Program options** > **Files** > **Templates**, select **Template path** (TEMPLATEPATH system variable).

3 Click the **Browse** button.

4 On the **Choose a directory:** dialog box, select a folder, then click the **OK** button.

3.1.2 Setting the default template drawing

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Files** > **Templates**, select **Template** (BASEFILE system variable).

3 Click the **Browse** button.

4 On the **Choose a file** dialog box, select a drawing file (*.dwg) or drawing template (*.dwt), then click the **Open** button.

3.1.3 Using the Start page

1 Click the **Start** page.

Launch the GOTOSTART command to open the **Start** page if necessary.

The name of the current drawing template is displayed when hovering over the **New drawing** button.

2 Do the following:

3
- (Optional) Select a new template to start from.
- ***Note:*** All template drawings in the **Templates** folder matching the current workspace (Drafting, Modeling, Mechanical, Civil Survey or BIM), are listed here. Choose **More...** to see the full content of the **Templates** folder.
- Click the **New drawing** button.

3.1.4 Creating a drawing using the default template drawing

First method:

Click the + button at the right hand side of the document tabs.

Second method:

Launch the QNEW command.

Third method:

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog, choose **Start from Default Template**, then click the **Finish** button.

Note: If the default template drawing is not specified, the **Create New Drawing** wizard is launched depending on the current value of the STARTUP system variable, otherwise the **Select Template** dialog box appears.

3.1.5 Creating a drawing using the NEWWIZ command

1 Launch the NEWWIZ command.

2 In the **Create New Drawing** dialog box, choose **Start from Scratch**, then click the **Next** button.

3 Choose between either **Imperial** or **Metric** default settings.

Click the **Finish** button to create the drawing.

Note: Using the above procedure, it depends on the status of the BASEFILE and PSTYLEPOLICY variables whether **CTB** (Color table) or **STB** (Named plot style table) will be used to set up plotter configurations in the newly created drawing.

BASEFILE	PSTYLEPOLICY	CTB or STB?
not specified	OFF	STB
not specified	ON	CTB
specified	ON or OFF	as defined in the template drawing

3.1.6 Setting the Plot style policy

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Plot and publish**, select **Plot style policy** (PSTYLEPOLICY system variable).

3 Do one of the following:

4 - Choose **Associate an entity's plot style with its color** for Color dependent (CTB) plot styles.
- Choose **No association between color and plot style** for Named (STB) plot styles.

3.1.7 Creating a drawing using a template drawing

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog box choose **Start from Template**, then click the **Next** button.

3 Do one of the following:

4
- Choose a template drawing in the **Select a Template** list.
  By default this is the content of the **Templates** folder.
- Click the **Browse...** button to select a template drawing.
  You can choose between ***.dwg** and ***.dwt** file types.

3.1.8 Creating a drawing using a wizard

1. Launch the NEWWIZ command.
2. On the **Create New Drawing** dialog box, choose **Use a Wizard**, then click the **Next** button.
3. Choose between either **Imperial** or **Metric** default settings, then click the **Next** button.
4. Select the **Unit of measurement** and the display **Precision**, then click the **Next** button.
5. Select the **Angle of measurement** and the display **Precision**, then click the **Next** button.
6. Choose between **Color Dependent** plot styles (CTB) or **Named** plot styles (STB), then click the **Next** button.
7. Select the default **Color** and **Linetype**.
8. Define the **Grid**, **Snap**, **UCS icon** and **Marker Blips** settings.
9. Click the **Finish** button to create the drawing.

   (Optional) Choose **Save As...** in the **File** menu to save the drawing as a new template in the **Templates** folder.

3.2 Working in the Drawing Explorer dialog box

3.2.1 Adding a drawing folder

1. In the **Drawings** > **Folders** sub-window of the **Drawing Explorer** main window, click the **Add local folder...** button.

   The **Choose a folder** dialog box appears.
2. Do one of the following:
   - To add an existing folder: select the folder you want to add.
   - To create a new folder: click the **New folder** button.
3. Click the **Select Folder** button.

   The folder is added.

Note:

- Drawings in the subfolders of the selected folder can be opened too.
- The parent folder(s) of the selected folder are greyed out in the drawing folder tree. Drawings in such folders cannot be opened.

3.2.2 Opening a drawing

To open a drawing in one of your favorite folders:

1 In the **Drawings** > **Folders** sub-window, open the folder of the drawing.

2 Select a drawing.

3 A preview is shown in the **Preview** sub-window.

4 Do one of the following:

- Double-click the drawing.
- Right-click and choose **Open** in the context menu.

3.2.3 Logging on to Bricsys 24/7

1 In the **Drawings** panel, select **Folders**.

2 Click **Logon to Bricsys 24/7...**.

3 Fill out your username and password.

4 When logged in, all of your projects on Bricsys 24/7 are listed.

3.2.4 Opening a drawing from a Bricsys 24/7 project

1 Expand the project, then select the drawing folder.

2 Select the drawing.

3 A preview is shown in the **Preview** sub-window.

4 Right-click the drawing, then choose **Download** in the context menu.

A **Save File As** dialog box appears.

5 Select a folder and optionally rename the drawing, then click the **Save** button.

The drawing is downloaded and opened in the current BricsCAD session.

3.2.5 Copying definitions between drawings

1 Open both the source and the target drawing.

2 Launch the **Drawing Explorer** dialog box.

3 In the source drawing, select the category you want to copy items from, e.g. **Blocks**.

4 (option) Adjust the view option of the **Details** panel, e.g. **Icon View** button ().

5 Select the definition(s) you want to copy, e.g. a series of blocks.

6 Press and hold the **Ctrl** key to select multiple items.

7 Do one of the following:

- Press and hold the left mouse button to drag the selection set to the **Open Drawings** section of the Drawing Explorer.

  Release the left mouse button when on the target drawing name.

  The selected items are copied to the target drawing.

- Right-click and choose **Copy** in the context menu.

  Select the same definition category in the target drawing, then right-click and choose **Paste** in the context menu.

Note:

- You will be prompted to overwrite items of the same name.
- Drag-and-drop is not supported for some definition categories, e.g. MultiLeader Styles, Visual styles, Render Presets and Page Setups.

3.2.6 Binding definitions from Xrefs to the current drawing

The Drawing Explorer allows you to bind individual table records (layers, linetypes, text styles, dimension styles, blocks) from Xrefs to the current drawing.

1 Open the drawing with the Xref attached.

2 Select the desired category, then the Xref definition(s) you want to bind.

3 Right-click the definition(s), then choose **Bind** in the context menu.

3.3 Drawing properties

The DWGPROPS command opens the **Drawing Properties** dialog box, showing the general information and user defined properties stored within a drawing.

The **Drawing Properties** dialog box includes:

- General information: drawing data
- Summary: editable default properties
- Statistics: history data
- Sheetset: info about the associated sheetset
- User defined custom properties

Drawing properties can be referenced in fields.

3.3.1 Opening the Drawing Properties dialog

Do one of the following:

- Choose **Drawing Properties...** in the **File** menu.
- Execute the DWGPROPS command.

3.3.2 Defining default properties

1 On the **Drawing Properties** dialog box, click the **Summary** tab page.

2 Fill out the property fields.

3 Click the **OK** button.

3.3.3 Creating custom properties

1 On the **Drawing Properties** dialog box, click the **Custom** tab page.

2 Click the **Add** button.

The **Custom Property** dialog box appears.

3 Type a name in the **Name** field.

The custom property name is listed under **Document** in the **Field** dialog box.

4 Type a value in the **Value** field.

The value is displayed where the custom property is inserted as a field.

3.4 Drawing security options

The SECURITYOPTIONS command allows you to protect your drawings with a password. Password protected drawings cannot be opened without entering the password first.

3.4.1 Setting the password

1 Do one of the following:
- Choose **Security Options...** in the **File | Drawing Utilities** menu.
- Execute the SECURITYOPTIONS command.

The **Security options** dialog box opens.

2 Type the password in the **Enter Password** field.

3 Choose a crypt provider from the **Crypt Provider** list.

4 Set the **Key Length** to one of the values from the list, depending on the chosen Crypt Provider.

5 Click the **OK** button or press Enter.

The **Security options** dialog box closes.

The **Password** dialog opens.

6 Type the password in the **Confirm Password** field.

7 Click the **OK** button or press Enter.

The password is defined.

3.4.2 Removing the password

1 Do one of the following:
- Choose **Security Options...** in the **File | Drawing Utilities** menu.
- Execute the SECURITYOPTIONS command.

2 The **Security options** dialog box opens.

3 Empty the **Enter Password** field.

4 Click the **OK** button or press Enter.

A message box appears to confirm that the password is removed.

5 Click the **OK** button or press Enter to close the message box.

3.5 Opening and saving drawings

The way BricsCAD opens and saves drawings is controlled through a series of system variables.

Name	Title	Description
	Display full path in title	Controls whether the full path of a drawing displays in the title bar, or the file name only.
	Drawings path	Specifies an additional folder in the file dialog for the OPEN and SAVEAS commands. On the Windows platform: the fifth folder in the **Open File** dialog's **Places** bar.
-	First, Second, Third and Fourth folder	Sets the folders 1 through 4 in the **Places** bar of the non-standard **Open File** dialog (Windows platform only).
	Thumbnail preview image size	Sets the size of the thumbnail image in the Preview on file dialogs.
	Index Control	Determines whether layer and/or spatial indexes are created and saved. • The layer index lists all entities in the drawing and what layer they are on. • The spatial index stores the location of all the entities in 3D space. The indexes are used to load Xrefs if demand loading is enabled. If a layer index exists, entities on layers which are Off or Frozen are not loaded. If a spatial index exists, entities in clipped Xrefs which are outside the clipping boundary are not loaded. The demand loading of Xrefs is controlled through the XLOADCTL system variable). Creating indexes slows down the saving of a drawing, but increases the speed of handling Xrefs. Therefore create indexes only for drawings that will be used as an external reference. 0 = No indexes.

Name	Title	Description
	Xref load control	Controls demand loading of entities in external references. If demand loading is turned on you will experience a performance increase: • When clipping external references which are saved with a spatial index. Entities in the external reference within the clipped area are loaded only. If the clipping area is modified, more or less entities are loaded from the reference drawing. • When freezing layers in external references which are saved with a layer index, entities in the external reference on thawed layers are loaded only. If xref-dependent layers are thawed more entities are loaded.
	Multi-Threading flags	Bitcode 1024: Delayed XREF loading. Delays the loading of Xrefs to idle time.
	Incremental save backup	Controls the creation of backup files (BAK). Improves the speed of incremental saves if switched off, especially for large drawings.
	Save percent	This system variable controls how often BricsCAD performs a full save. A full save reorganizes all the data in the drawing database and creates a compact file. A partial save stores all changes at the end of the drawing data base. A full save takes longer to be executed than a partial save.
	Recent File List	Specifies the number of files shown in the **Recent Files** flyout of the **File** menu and on the **Start** tab.
	Proxy graphics	Determines whether images of proxy entities(1) are saved in the drawing. If switched off, a bounding box displays instead.
	Proxy notice	Displays a notice when you open a drawing containing proxy entities (1) created by an application that is not present.
	Proxy show	Controls how proxy entities (1) display in a drawing.

Name	Title	Description
	Proxy web search	Determines whether BricsCAD checks for object enablers.
	Save format	Sets the default dwg version to save a drawing.
	Save file path	Specifies the path and folder where automatic saves (.SV$) and temporary files are stored. The SAVEFILEFOLDER command opens the SAVEFILEPATH location in the system file browser.
	Save time interval	Sets the interval in minutes for automatic saves. If set to zero, automatic saves are turned off. Automatic saves are created with a .SV$ extension in the folder defined by the SAVEFILEPATH system variable.
	Enable macros	Defines whether macros are enabled when loading a VBA-project.
	Enable VB applications.	Use registry settings to make existing VB applications run with BricsCAD.
	Ignore all but first bit of DBMOD(2) for autosave	If this user preference is switched on, no auto-save file will be created for files that are viewed only.
	Ignore all but first bit of DBMOD(2) for close	If this user preference is switched on, you can view a file, then close it without getting a request to save the file.

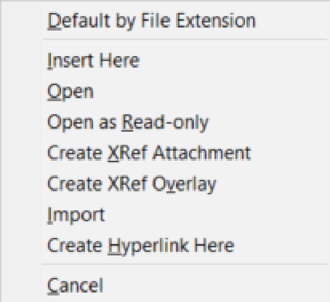
(1) Proxy entities are custom entities which are created by an add-on application. If the add-on is currently not loaded or not installed, such entities cannot be displayed properly.

(2) DBMOD: Stores the status of the drawing modification as a bit-code as the sum of the following values: 1 = Object database modified, 4 = Database variable modified, 8 = Window modified, 16 = View modified, 32 = Field modified.

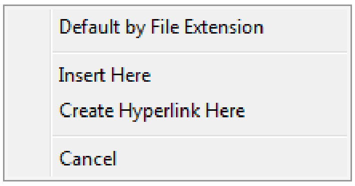
3.5.1 Using drag-and-drop to open a drawing

You can open a drawing by dragging it from a **File** dialog box onto the BricsCAD application window. Select the file, then hold down the left mouse button and move the cursor to the BricsCAD application window and release the left mouse button to load the drawing. When you drag a file holding down the right mouse button, a context menu displays:

When a drawing file is selected:



When another file type is selected:



Choose **Default by File Extension** to:

- Open a drawing file.
- Attach a PDF.
- Attach a raster image file.
- Insert a file as an OLE object (e.g. an Excel spreadsheet).

Choose **Insert Here** to:

- Insert a drawing file as a block in the current drawing.
- Attach a PDF.
- Attach a raster image file.
- Insert the file as an OLE object.

Choose **Open** to open a drawing file.

Choose **Open as Read-only** to open a drawing file and preserve the original from being overwritten. You can only save such drawing under a different name.

Choose **Create Xref** or **Create Xref Overlay** to attach a drawing file as an external reference to the current drawing.

- **Attachment**: if the current drawing is attached as an xref to another drawing, an attachment becomes a nested xref.
- **Overlay**: an overlay is ignored when the drawing to which it is attached is then attached as an xref to another drawing.

Choose **Import** to import the geometry in the current drawing.

Choose **Create Hyperlink Here** to create an hyperlink to the selected file.

Note:

- Drawing file extensions are: DWG, DXF or DWT

- When multiple files are selected, the files are processed one after the other.

3.5.2 Opening drawings embedded as OLE objects in other applications

You can open a drawing file embedded as OLE object in other applications by double-clicking on that object from within the application. The drawing will open in BricsCAD in a tab with the name of the original file. You can modify the drawing and save it.

Opening drawings from older versions of BricsCAD which are embedded as OLE objects in other applications will automatically open in the latest installed version of BricsCAD.

When the COMACADCOMPATIBILITY system variable is set to **On**, AutoCAD® drawing files that are embedded as OLE objects in other applications will open in BricsCAD.

3.5.3 Replacing unknown fonts

When you open a drawing with unknown fonts, a notifying balloon, **Missed font(s)**, displays in the right bottom corner. Click **Specify font replacement** to open the **Font Substitution** dialog box where the missing fonts appears. This dialog box helps you to choose the fonts for the ones that are not found in the default.fmp file.

Note: This usually happens when Big Fonts are used because they contain characters for multiple languages.

3.6 Getting drawing information

You can retrieve two types of information from a drawing:

- General information about the drawing.
- Information about the entities in the drawing.

3.6.1 General drawing information

The DWGPROPS command opens the **Drawing Properties** dialog box, showing the general information and user defined properties stored with a drawing.

The SETVAR command displays or changes the values of system variables (short for "set variables").

The STATUS command reports the status of the drawing: drawing name, the status of the most important system variables and the total number of entities in the drawing.

The TIME command reports on time in the drawing: current time, creation time, last update, total editing time and user elapsed time.

3.6.2 Information about entities

The STATUS command reports the total number of entities in the drawing.

Using **Quick Select**, you can see the number of entities for each category.

The AREA command finds the area and perimeter (or length) of 2D entities.

The MASSPROP command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties").

The ID command reports the X, Y, Z coordinates of picked points.

The LIST command lists the properties of selected entities.

3.7 Comparing drawings

The DWGCOMPARE command compares a selected drawing with the current drawing during a comparison session.

The ENDCOMPARE command concludes a comparison session.

The KEEPME command allows you to merge the compared drawings visually.

3.7.1 Comparison system variables

Name	Title	Description
CMPCLRMISS	Color of missing entities	Controls the color of missing entities. The default color is Red.
CMPCLRMOD1	Color of modified entities	Controls the color of modified entities. The default color is index color 253.
CMPCLRMOD2	Color of modified entities in the second drawing	Controls the color of modified entities in the second drawing. The default color is Yellow.
CMPCLRNEW	Color of new entities	Controls the color of new entities. The default color is Green.
CMPDIFFLIMIT	Maximum number of entities	Controls the limit for entities to compare. The comparison will be aborted after reaching the difference limit. Values between 1 and 10,000,000 are accepted.
CMPFADECTL	Fade	Controls the fade level for unmodified entities. The default value is 80.
CMPLOG	Log control	Toggles the creation of a log report (cmplog). By default, it is set Off.

3.7.2 Executing a comparison session

1 Launch the DWGCOMPARE command.

1 You are prompted:

CMPDIFFLIMIT = 10000000.

Select a file to compare with or set the comparison limit [Select/Limit] <Select>:

2 (Optional) Choose the **Limit** option to set a new value for entities comparison.

3 Choose **Select** to display the **Select the file to compare with** dialog box.

4 Select the drawing to be compared with the current drawing, then click the **Open** button or double-click the drawing name.

The **Compare** panel displays to compare the drawings. See also the **Compare panel** article.

The **Compare** toolbar also appears.

5 Click the **End compare** button in the **Compare** panel to conclude the comparison session.

3.7.3 Analyzing the comparison in the Structure panel

1 While running the DWGCOMPARE command, load the **Default.cst** configuration in the **Structure** panel.

2 Expand the **Comparison** node.

Four subnodes are shown: **Missing**, **Modified (Original)**, **New** and **Modified (Compared)**.

The number of entities in each subnode is indicated between brackets.

3 Do the following:

- Click a subnode to select the corresponding entities in the drawing.
- Expand a subnode, then click an entity to highlight the corresponding entity in the drawing.
- Press and hold the Ctrl key to select multiple entities.
- Right-click, then choose **Zoom** in the context menu to zoom onto the selected entity or entities.

Note: When selecting entities in the **Structure** panel, entities are either selected or highlighted in the drawing, or tree selection is ignored. (See the **Structure panel** article).

3.8 CAD Standards

3.8.1 About

Associates one or more standards (DWS) files to drawing DWG files.

There are two methods to check the compatibility of drawing DWG files with the standards DWS files and validate or ignore the mapping of entities properties.

1 Using the CAD Standards commands inside BricsCAD.

2 Using the **standards_app.exe** stand alone application.

3.8.2 Using the CAD Standards commands inside BricsCAD

The CAD Standards commands are listed in the **CAD STANDARDS** ribbon panel.

The steps for checking and validation are as follows:

1 Launch the STANDARDS command to associate standard DWS files with the working DWG drawing file and choose which plug-ins will be checked.

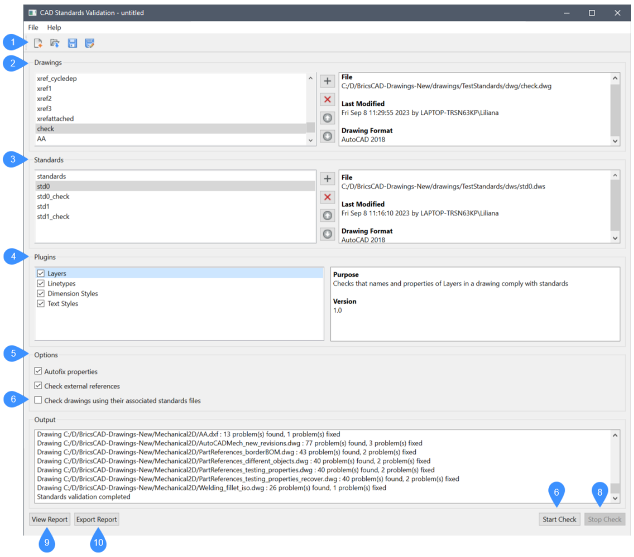
2 Launch the CHECKSTANDARDS command to check for compatibility between desired plug-ins from the current drawing and the standards drawing and validate or ignore the recommended matching.

3 The LAYTRANS command will be used to associate layers in the current drawing with layers in other drawings or standard drawings.

3.8.3 Using the standards_app.exe stand alone application

The **standards_app.exe** is a stand alone application stored in the BricsCAD installation folder (by default: *C:\Program Files\Bricsys\BricsCAD V25x64*).

Run the **standards_app.exe** application to open the **CAD Standards Validation** dialog box. The application will check the compatibility between drawing (DWG) files and the standards (DWS) files according to the chosen plug-ins.



1 File menu

2 Drawings

3 Standards

4 Plugins

5 Options

6 Output

7 Start Check

8 Stop Check

9 View Report

10 Export report

3.8.4 File menu

Displays management options for CHX files. A CHX file lists the drawing (DWG) and standard (DWS) files and options for checking them. It can be used from ACAD or created from scratch.

New
Creates a CHX file from scratch.

Open
Opens the **CAD Standards Validation** dialog box to choose an existing CHX file.

Save
Saves the CAD Standards Validation configuration into a CHX file.

Save as
Saves the CAD Standards Validation configuration to a different CHX file.

3.8.5 Drawings

Lists the drawing files (DWG) added for checking.

With right-side buttons you are able to:

- +: add a new drawing to the list.
- : remove the selected file.
- : move up in the list the selected file.
- : move down in the list the selected file.

3.8.6 Standards

Lists the drawing standards (DWS) file used for standard checking.

Note:

- You can create your drawing standard file by saving a DWG file in which you define layers, dimension styles, line types, and text styles as a DWS file.
- The right-side buttons work the same as for drawing (DWG) files.

3.8.7 Plugins

Lists the plug-ins that will be used for checking operation.

A plug-in refers to a property or set of entity properties in standard files. Only checked plug-ins will be considered for checking operations.

The available plug-ins are:

Layers
Checks that names and properties of Layers in a drawing comply with standards.

Linetypes
Checks that names and properties of Linetypes in a drawing comply with standards.

Dimension Styles
Checks that names and properties of Dimension Styles in a drawing comply with standards.

Text Styles
Checks that names and properties of Text Styles in a drawing comply with standards.

3.8.8 Options

Lists the checking options:

- **Autofix properties**: automatically applies the standard properties to the target named object.
- **Check external references**: includes external references in checking.
- **Check drawings using their associated standards files**: checks the drawing files that contains their associated standards file.

3.8.9 Output

Displays a report of the standards checking.

3.8.10 Start Check

Runs the checking.

3.8.11 Stop Check

Stops the checking.

3.8.12 View Report

Opens the report into the **StandardsApp** readable window.

3.8.13 Export report

Exports the checking result in a report (LOG) file.

3.9 Repairing a drawing

TheAUDIT command analyzes the integrity of the current drawing and optionally tries to fix errors.

TheRECOVER command repairs a damaged drawing, then opens the drawing.

3.9.1 Analyzing the integrity of the current drawing

1 Launch the AUDIT command.

Prompts you:Fix any errors detected? Yes/<No>

2 Do one of the following:

- ChooseYesin the context menu or typeY, then press Enter to analyze the integrity of the current drawing and fix the errors detected.
- ChooseNoin the context menu or typeN, then press Enter to analyze the integrity of the current drawing.

The drawing is analyzed.

3 (option) Press F2 to open the**Prompt History**window in order to read the audit report:

- # objects audited
- Total errors found during audit #, fixed #.

Note:

- Set the AUDITCTL system variable to 1, to let AUDIT create an ASCII file describing the detected problems and the action taken. This report, with the .adt file extension, is placed in the same directory as the current drawing .
- Use the RECOVER command to repair the errors that AUDIT cannot fix.

3.9.2 Repairing a drawing

1 Do one of the following:
- Choose **Recover** in the **File** > **Drawing Utilities** menu.
- Click the BricsCAD logo in front of the ribbon tabs, then choose **Drawing Utilities** > **Recover**.
- Launch the RECOVER command.

The **Open Drawing** dialog window displays.

2 Select the drawing, then press the **Open** button on the **Open Drawing** dialog window.

The drawing is processed and - if the repair succeeds - opened.

3 (option) Press F2 to open the **Prompt History** window in order to read the recover report.

3.10 Projects

Projects are especially useful to manage Xrefs and images when exchanging drawings. If external references (Xrefs) and images are not found in the saved path, the project search paths are used to find the external references and images.

Projects are saved in the registry and contain one or more search paths.

3.10.1 Accessing the Project settings

1 Launch the SETTINGS command.

1 The **Settings** dialog box opens.

2 Go to **Program Options** > **Files** > **Projects**.

3.10.2 Creating projects

1 Edit the PROJECTSEARCHPATHS system variable.

2 Click the **Browse** button at the right hand side of the **Project search paths** settings field.

The **Project Settings** dialog box appears.

3 To create the first project:
- Click the **Add Project or Path** button on the **Project Settings** dialog box.

  A new project is added.
- Type a name to replace the **NewProject1** default name.

4 To create additional projects:
- Click **Projects** (on top of the projects tree) or select an existing project.
- Click the **Add Project or Path** button.

  A new project is added.

- Type a name to replace the **NewProject1** default name.

3.10.3 Adding search paths to a project

1 Edit the PROJECTSEARCHPATHS system variable.

2 Select the project in the **Projects** tree.

3 Click the **Add Project or Path** button.

A path is added to the project.

4 Type a name for the new path.

5 Click the **Browse** button.

The **Choose a Path** dialog box appears.

6 Browse to a folder, then click the **Select Folder** button.

7 Repeat the previous steps to add more search paths to a project.

3.10.4 Changing the order of the search paths

1 Edit the PROJECTSEARCHPATHS system variable.

2 Expand the project in the **Projects** tree.

3 Select a search path, then click the up or down buttons to change the order of the search paths.

3.10.5 Assigning a project to the current drawing

1 Edit the PROJECTNAME system variable.

2 Type a project name in the settings field.

3.11 File Types Dictionary

The file types supported in BricsCAD are:

CAD Formats	Extension	Description
FBX binary file	FBX	Mudbox format used to send 3D models to animation and rendering software; binary format creates compact files maintained by Autodesk®; see the FBXEXPORT command.
FBX ASCII file	FBX	Mudbox format used to send 3D models to animation and rendering software; ASCII format creates human-readable files.
AutoCAD® Drawing	DWG	CAD drawing format created by Autodesk®; to export drawings in an earlier version of DWG, use the SAVEAS command.

CAD Formats	Extension	Description
AutoCAD® ASCII DXF	DXF	Drawing interchange format created by Autodesk®; to create binary DXF files, use the DXFOUT command.
Drawing Template	DWT	Templates on which new drawings are based.
3D DWF	DWF or DWFx	CAD viewing format for 3D models created by Autodesk®.
Binary DWF	DWF	Compressed version of DWF; creates more compact files.
Zipped ASCII Encoded 2D Stream DWF v6.0	DWF	ASCII version of DWF compressed into a ZIP file.
Compressed DWF v5.5	DWF	Compressed ASCII version of an older version of DWF; creates more compact files.
Binary DWF v5.5	DWF	Compressed version of an older version of DWF; creates more compact files.
ASCII DWF v5.5	DWF	ASCII version of an older version of DWF.
IFC file	IFC	Saves a BIM model as an IFC file to use with an IFC-certified applications data. **BIM properties** are included in the IFC export. The Industry Foundation Classes (IFC) data model is intended to describe building and construction industry. The **IFC4 Reference View file** is a simplified version that allows exchanging IFC models with tools that do not support all the IFC4 features.
Revit® Project	RVT	The RVT file extension is associated with Revit®, a Building Information Modeling (BIM) software developed by Autodesk®.
Unreal Engine	UDATASMITH	Exports BIM data to the Unreal Engine file type, which can be imported in the **Twinmotion 3D architectural visualization** program.
MicroStation DGN File	DGN	Microstation design files.

CAD Formats	Extension	Description
SketchUp file	SKP	SketchUp model files are imported as meshes. If desired, the CONVTOSOLID and CONVTOMESH commands provide easy conversion between meshes and solids.
Standards File	DWS	Drawing standards file, used by the STANDARDS and CHECKSTANDARDS commands.
ESRI Shapefile	SHP	ESRI Shape file with support for GIS data types.
Keyhole Markup Language	KML, KMZ	XML based format files used to define geographical data for mapping applications like Google Earth and Google Maps.

Vector Formats	Extension	Description
Windows Meta File*	WMF	Vector format created by Microsoft; used by the Clipboard; see the WMFOUT command.
Enhanced Windows Meta File*	EMF	Updated version of WMF.
Scalable Vector Graphics	SVG	Used by Web pages to display vector graphics. Scalable Vector Graphics (SVG) is a family of specifications of an XML-based file format for describing two-dimensional vector graphics, both static and dynamic (for example, interactive or animated). The SVG specification is an open standard that has been under development by the World Wide Web Consortium (W3C) since 1999.
Lithography	STL	3D format used by 3D printers. STL (STereoLithography) is a file format native to the stereolithography CAD software created by 3D Systems. STL is widely used for rapid prototyping and computer-aided manufacturing. STL files describe only the surface geometry of a three dimensional object without any representation of color, texture or other common CAD model attributes. The STL format specifies both ASCII and binary representations. Binary files are more common, since they are more compact. See also the STLOUT command.

Vector Formats	Extension	Description
Collada file	DAE	3D format used by rendering and animation software. COLLADA (from collaborative design activity) is an interchange file format for interactive 3D applications. It is managed by the nonprofit technology consortium, the Khronos Group, and has been adopted by ISO as a publicly available specification, ISO/PAS 17506.
Slide File	SLD	Proprietary graphics format created by Autodesk® for saving screen grabs of viewports.
3D PDF**	PDF	Portable Document Format (PDF) is a file format used to represent documents, independently from application software, hardware, and operating system.

HTML Format	Extension	Description
HTML	HTM	Hyper Text Markup Language format files are opened in a Web browser. The VMLOUT command exports drawings to VML (short for **vector markup language**) embedded in a HTML file.
Wavefront Object file	OBJ	A text file format which contains 3D graphic information such as vertices, normals, polygonal faces and 3D coordinates. Optionally, an OBJ file can refer to a material (MTL) file (which is also a text file) and the latter can refer to image or texture (JPG/PNG/BMP) files.

Raster Format	Extension	Description
Bitmap*	BMP	Raster format created by Microsoft; used by the Clipboard.

(*) The PICTUREEXPORTSCALE system variable sets the view size when exporting to picture formats (WMF, EMF, BMP).

(**) If the EXPORT3DPDFWRITER system variable is set to 1 - Internal 3D PDF writer, it is possible to export Mechanical Structure or exploded-view Animation.

The file types supported in Communicator for BricsCAD® are:

Format	Extension	Description
Standard for Exchange of Product data	STE, STP, STEP, STPZ	STEP (Standard for the Exchange of Product Data) is an ISO standard exchange format used for representing three-dimensional data in a format that can be recognized by multiple programs.
Initial Graphics Exchange Specification	IGS, IGES	A file format which defines a vendor neutral data format that allows the digital exchange of information among Computer-aided design (CAD) systems.
CATIA V4	model	CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systèmes.
CATIA V5/V6	CATPart, CATProduct, CGR	V5/V6 of the Catia CAD program from Dassault.
Parasolid	XMT_TXT, XMT_BIN X_T, X_B	Parasolid is a geometric modeling kernel originally developed by ShapeData, now owned by Siemens PLM Software (formerly UGS Corp.), that can be licensed by other companies for use in their 3D computer graphics software products.
VDA-FS	VDA	VDA-FS is a CAD data exchange format for the transfer of surface models from one CAD system to another. Its name is an abbreviation of **Verband der Automobilindustrie - Flächenschnittstelle**, which translates to the **automotive industry association - surface data interface**. The standard was specified by the German organization VDA.
Autodesk Inventor®	IAM, IPT	Autodesk Inventor®, developed by U.S.-based software company Autodesk®, is 3D mechanical CAD design software for creating 3D digital prototypes used in the design, visualization and simulation of products. Inventor uses specific file formats for parts (IPT) and assemblies (IAM).

Format	Extension	Description
Siemens	JT	JT is a 3D data format developed by Siemens PLM Software (formerly UGS Corp.) and is used for product visualization, collaboration, and CAD data exchange. It can contain any combination of approximate (faceted) data, boundary representation surfaces (NURBS), Product and Manufacturing Information (PMI), and Metadata (textual attributes) either exported from the native CAD system or inserted by a product data management (PDM) system.
Solid Edge	PAR, PSM	Solid Edge is a 3D CAD parametric feature solid modeling software. which provides solid modeling, assembly modeling and drafting functionality.
NX	PRT	PRT is a part or model file format used by PTC Pro/Engineer, which is a CAD program that creates design parts and assembly processes.
Creo Elements Pro Engineer	ASM, PRT	Parametric, integrated 3D CAD/CAM/CAE solution created by Parametric Technology Corporation (PTC).
Solidworks	SLDASM, SLDPRT	SolidWorks is a 3D mechanical CAD program that runs on Microsoft Windows and is being developed by Dassault Systèmes. SLDPRT: part file, SLDASM: assembly file The IMPORTSOLIDWORKSROTATEYZ system variable controls if geometry imported from SolidWorks files should be automatically rotated to map Y axis to Z axis.
3D PDF*	PDF	Portable Document Format (PDF) is a file format used to represent documents, independently from application software, hardware, and operating system.
ACIS**	ASAB, ASAT, SAB, SAT	The 3D ACIS Modeler (ACIS) is a geometric modeling kernel developed by Spatial Corporation.
XCGM	XCGM	Computer Graphics Metafile (CGM) File Format Family.

Note: (*) If the EXPORT3DPDFWRITER system variable is set to 0 - Communicator for BricsCAD® 3D PDF writer, mechanical structure or exploded views are not supported.

Note: (**) EXPORT stores a scalefactor in the SAT and SAB files relative to mm, and IMPORT uses it to scale.

4. Drawing accurately

4.1 Orthogonal mode

When creating or manipulating entities, Orthogonal mode restricts the movement of the cursor to be parallel to the X-axis or the Y-axis of the currentcoordinate system.

Lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. When drawing, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is nearest to the cursor.

Note:

- When you enable the Isometric snap style, cursor movement is restricted to orthogonal equivalents within the current isometric plane.
- You can rotate the Orthogonal Mode axes using the SNAPANG system variable.
- Holding down the Shift key reverses the Orthogonal Mode setting.
- Orthogonal Mode is overruled by entity snaps.
- Turning on Ortho Mode automatically disables Polar Tracking and vice versa.

4.1.1 Toggling Orthogonal mode

Do one of the following:

- Click theORTHOfield in theStatus Bar.
- Press theF8function key.
- Hold down the Shift key.
- Launch the ORTHO command.

4.2 Unit settings

The UNITS command sets units of linear and angular measurements through the **Settings** dialog box. The -UNITS command sets units of linear and angular measurements in the Command line.

The INSUNITS (Insertion Units) system variable defines a drawing units value for automatic scaling when inserting blocks or attaching Xrefs.

Any length, area or volume property can be displayed using the unit(s) preferred by the user.

The unit defined by INSUNITS is used as the base unit for any requested unit conversions. If INSUNITS = 0 (undefined), the PROPUNITS mechanism is disabled because the base drawing unit is undefined.

Note:

- When LUNITS = 3 (Engineering) or 4 (Architectural), property values will be formatted in imperial notation.
- Automatic unit conversion during length input can be used when bitcode 1 of PROPUNITS (Format length properties) is set. E.g. when INSUNITS is set to "mm", entering "2m" in a length property field will result in entering [2000.0000 mm]. Entering "1ft" will give [304.8000 mm].

4.2.1 System variables

Name	Title	Description
INSUNITS	Insertion Units	Defines the value of 1 Drawing Unit (DU), which is used for automatic scaling when inserting or attaching blocks, images or Xrefs. When both INSUNITS and PROPUNITS are ON; **Length**, **Area**, **Volume** and/or **Inertia** properties will be formatted with a dedicated unit.
MEASUREMENT	Measurement	Sets drawing units as **Imperial** or **Metric**. The MEASUREMENT system variable controls whether ANSI or ISO hatch pattern and linetype files are used.
PROPUNITS	Property Units	Defines which property values are automatically formatted with a dedicated unit when INSUNITS is not zero. The value is stored as a bitcode, using the sum of the values of all selected options.
LUNITS	Linear Units Type	Sets the linear unit type: **Scientific**, **Decimal**, **Engineering**, **Architectural** or **Fractional**.
LUPREC	Linear Unit Precision	Sets the number of decimal places displayed for linear units.
AREAPREC	Area Precision	Sets the number of decimal places displayed for areas when the **Area** bit of PROPUNITS is ON. If set negative, the value of LUPREC is used.
VOLUMEPREC	Volume Precision	Sets the number of decimal places displayed for volumes when the **Volume** bit of PROPUNITS is ON. If set negative, the value of LUPREC is used.
LENGTHUNITS	Length Units	Defines a list of units used for displaying lengths when the **Length** bit of PROPUNITS is ON. If empty, all lengths are displayed in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.
AREAUNITS	Area Units	Defines a list of units used for displaying areas when the **Area** bit of PROPUNITS is ON. If empty, all lengths are displayed in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.

Name	Title	Description
VOLUMEUNITS	Volume Units	Defines a list of units used for displaying volumes when the **Volume** bit of PROPUNITS is ON. If empty, all lengths are displayed in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.
UNITMODE	Unit Mode	Specifies how Imperial units are displayed.

4.2.2 Entering angles

The AUNITS system variable defines the angular unit type.

The ANGDIR system variable sets the positive angle direction, counter clockwise (default) or clockwise. Angles are measured with respect to direction of the positive X-axis of the current coordinate system.

You can enter angles in any of 5 formats:

- Decimal degrees : 37.456 (AUNITS = 0)
- Degrees / Minutes / Seconds : 37°27'22" or 37d27'22" (AUNITS = 1)
- Gradians : 41.6178g (AUNITS = 2)
- Radians : 0.6537r (AUNITS = 3)
- Surveyor's Units : N52°32'38"E or N52d32'38"E (AUNITS = 4)
- Surveyor's units are measured from the North (N) or the South (S) to the East (E) or to the West (W).

4.3 Coordinate reference system

A Coordinate System is a set of mathematical rules for specifying how coordinates are to be assigned to points.

The coordinate system is unrelated to the Earth. A Coordinate Reference System (CRS) is a coordinate system related to the Earth through a Datum. A Geodetic Datum is a model of the Earth that specifies a reference surface (Ellipsoid or Spheroid).

A projected coordinate reference system is the result of the application of a map projection to a geographic coordinate reference system. A map projection is a type of coordinate conversion. It uses an identified method with specific formulas and a set of parameters specific to that coordinate conversion method.

Coordinates may be changed from one CRS to another through the coordinate operation. Two types of coordinate operation may be distinguished:

- **coordinate conversion**, where no change of datum is involved and the parameters are chosen and thus error free.
- **coordinate transformation**, where the target CRS is based on a different datum to the source CRS. Transformation parameters are empirically determined and thus subject to measurement errors. (Geodetic, Seven parameters, Molodenksy...)

Geographic datasets are defined in **geodatabase.xml**that is placed in *<User folder>\AppData\Roaming \Bricsys\BricsCAD\<RELEASE>\en_US\Support* for Windows. The content of this file is composed of the 5 sections:

1 CoordinateReferenceSystems

2 Datums

3 Ellipsoids

4 Transformations

5 ProjectionCodes and Methods

4.3.1 CoordinateReferenceSystems

This section is the main XML node where all Coordinate Reference Systems are defined. It has many sub-nodes called**CRS**, each XML node must define a unique projected coordinate reference system and the necessary projected parameters.

The supported CRS projection types are described in the**ProjectionCodes**and**Methods**sections.

CRSnode attributes:

Follow Table 1 of OGP Geomatics Guidance Note No 7, part 2

Attribute name	Description	Units
epsg	Unique EPSG database id.	Integer
name	Human-readable name of CRS, Example: "WGS 84 -- WGS84 - World Geodetic System 1984, used in GPS".	String
codeSpace	Owner of CRS. It is optional.	String
x,y	Orientation of projected direction. Possible values: • "Westing" • "Easting" • "Southing" • "Northing" Directions MUST be defined.	String
xAxis, yAxis	Orientation of geographic direction. Possible values: • "Latitude" • "Longitude"	String
Greenwich	Greenwich Meridian relation.	Degree

Attribute name	Description	Units
proj	Projection method of CRS. Possible values: • "LL", "LatLon", "LonLat" is a Geodetic Latitude-Longitude. • "MercA", "Mercator_1SP" is a Mercator (variant A). • "MercB", "Mercator_2SP" is a Mercator (variant B). • "MercC" is Mercator (variant C). • "MercSp" is a Mercator (Spherical). • "MercPv" is a Popular Visualisation Pseudo Mercator. • "TMerc" is a Transverse Mercator. • "TMercSO" is a Transverse Mercator South Orientated. • "LCC_1SP" is a Lambert Conic Conformal 1SP. • "LCC_2SP" is a Lambert Conic Conformal 2SP. • "LCC_2SP_BE" is a Lambert Conic Conformal 2SP Belgium.	String
AngleSG	Angle from Rectified to Skew Grid.	Degree
Azimuth	Azimuth of initial line.	String
FE	Easting at false origin, False easting.	Float-Point value in CRS units
EC	Easting at projection center.	Float-Point value in CRS units
FN	Northing at false origin, False northing.	Float-Point value in CRS units
FC	Northing at projection center.	Float-Point value in CRS units
LonI	The longitude of the western limit of the first zone of a Transverse Mercator zoned grid system.	Degree
Lat	Latitude of natural origin, Latitude of standard parallel. It depends on Projection method.	Degree
Lat1	Latitude of 1st standard parallel.	Degree

Attribute name	Description	Units
Lat2	Latitude of 2nd standard parallel.	Degree
LatF	Latitude of false origin.	Degree
LatC	Latitude of projection center.	Degree
Lon	Longitude of natural origin, Longitude of origin.	Degree
LonF	Longitude of false origin.	Degree
LonC	Longitude of projection center.	Degree
SF	Scale factor at natural origin.	Float-Point value, unity
SFIL	Scale factor on initial line.	Float-Point value, unity
SFPSP	Scale factor on pseudo standard parallel.	Float-Point value, unity
W	Zone width in longitude.	Degree
zone	Zoned Grid System.	Degree
units	Units of conversion result from geographic coordinates to projected. Example: "Meter", unit "Degree" means no conversion, Geographic Coordinate System.	String

Example:common attributes for all map projections based on EPSG:31468.

```
<CRS epsg="31468" codeSpace="OGP" name="DHDN / 3-degree Gauss-Kruger zone 4"
 y="Easting" x="Northing"
```

```
Greenwich="0" proj="TMerc" Lon="12" Lat="0" SF="1" FE="4500000" FN="0" zone="4"
 units="Meter">
```

For each node**CRS**there should be at least one sub-node**Datum**that refers via**epsg**or**alias**attributes to sub-node**Datum**in the**Datums**section. The sub-node**Datum**MUST have the**id**attribute of a unique CRS name. At least one of the epsg code and the alias of Datum must be valid.

Attribute name	Description	Units
epsg	Unique EPSG database id. Example: "4326".	Integer

Attribute name	Description	Units
alias	Unique Datum name. Example: "WGS 84".	String
id	Unique CRS name: combination of Projected method and Datum. Example: "WORLD-MERCATOR". It related to CS-MAP, AutoCAD® names.	String
pjcode	Indicate code for the coordinate reference system projection method types. Corresponded to AutoCAD® projection codes. For example, 3 - Transverse Mercator, 44 - UTM, 45 - Transverse Mercator of Snyder formulation and etc.	Integer

Example:there are unique CRS definitions according to Datums insideCRSnode:

```
<Datum epsg="6314" alias="DHDN/3" id="DHDN/3.Gauss3d-4" pjcode="3" />
<Datum epsg="6314" alias="DHDN/2" id="DHDN/2.Gauss3d-4" pjcode="3" />
<Datum epsg="6314" alias="DHDN" id="DHDN.Gauss3d-4" pjcode="3" />
```

4.3.2 Datums

This section is a main node where all Datums are stored. Datum is a combination of the Earth model (Ellipsoid or Spheroid) and the transformation method to model WGS84. A**Datum**node specifies common parameters and contains at least one inner sub-nodes**Datum**exactly one sub-node**Ellipsoid**. Each inner sub-node**Datum**should define a unique name within the**id**attribute and may specify transformation parameters to WGS84 model via sub-node**Transformation**that refers via**epsg**or**alias**attributes to a sub-node**Transformation**in the**Transformations**section.

Attribute name	Description	Units
alias	Unique id. Example: "DHDN/3". It is related to CS-MAP, AutoCAD® names.	String
epsg	Unique EPSG database id. Example: "6314".	Integer
name	Human-readable name of Datum. Example: "Deutsches Hauptdreiecksnetz".	String
codeSpace	Owner of CRS. It is optional.	String

Example:

```
<Datums>
<Datum epsg="6314" codeSpace="OGP" name="Deutsches Hauptdreiecksnetz ">
```

```
<Datum id="DHDN/3">
<Transformation epsg="1777" alias="DHDN/3_to_WGS84" />
</Datum>
<Datum id="DHDN/2" name="Deprecated - Replaced by DHDN/3">
<Transformation epsg="1777" alias="DHDN/2_to_WGS84" />
</Datum>
<Datum id="DHDN" name="Deprecated - Replaced by DHDN/2">
<Transformation epsg="1673" alias="DHDN_to_WGS84" />
</Datum>
<Ellipsoid epsg="7004" alias="BESSEL" />
</Datum>
â€¦
</Datums>
```

4.3.3 Ellipsoids

The Ellipsoids section is the main section where models of the Earth are defined. Each**Ellipsoid**node should define an identification by a unique name (**alias**) and a semi-major axis (**a**) and one of the following attributes at least: flattening (**f**), semi-minor axis (**b**) or eccentricity (**e**).

Attribute name	Description	Units
alias	Unique id. Example: "WGS84". It related to CS-MAP, AutoCAD® names.	String
epsg	Unique EPSG database id. Example: "7030".	Integer
name	Human-readable name of Datum, Example: "WGS 84".	String
codeSpace	Owner of CRS. It is optional.	String
a	Length of the semi-major axis of the ellipsoid, the radius of the equator.	Float-Point value, strongly in Meter
b	Length of the semi-minor axis of the ellipsoid, the distance along the ellipsoid axis between equator and pole.	Float-Point value, strongly in Meter
f	Flattening	Float-Point value, unity
e	Eccentricity	Float-Point value, unity

Example:

```
<Ellipsoid epsg="7008" alias="CLRK66" name="Clarke 1866, Benoit Ratio"
 a="6378206.4000000004"
```

```
b="6356583.7999999998" f="294.9786982139" e="0.0822718542" />
```

4.3.4 Transformations

This section is a main node where all transformation methods are defined. Supported Transformation methods are described in the **ProjectionCodes** and **Methods** sections.

Attribute name	Description	Units
alias	Unique id. Example: "DHDN_to_WGS84". It related to CS-MAP, AutoCAD® names.	String
epsg	Unique EPSG database id. Example: "1673".	Integer
codeSpace	Owner of Transformation. It is optional.	String
source	Source Datum.	String
target	Target Datum.	String
use	Transformation method. Supported transformations using geocentric methods: • Geocentric translations • Four/Six/Seven Parameter Transformation • Similarity Transformation • Position Vector transformation • Coordinate Frame rotation	String
method	Methods of building of rotation matrix, if applicable, "PVT" is Position Vector Transformation, "CFR" is Coordinate Frame Rotation.	String
tX	X-axis translation.	Meter
tY	Y-axis translation.	Meter
tZ	Z-axis translation.	Meter
rX	X-axis rotation.	Degree
rY	Y-axis rotation.	Degree
rZ	Z-axis rotation.	Degree
dS	Scale difference.	Unity

Attribute name	Description	Units
xp	Coordinate 1 of evaluation point.	Meter
yp	Coordinate 2 of evaluation point.	Meter
zp	Coordinate 3 of evaluation point.	Meter
dtX	Rate of change of X-axis translation.	Unity
dtY	Rate of change of Y-axis translation.	Unity
dtZ	Rate of change of Z-axis translation.	Unity
drX	Rate of change of X-axis rotation.	Unity
drY	Rate of change of Y-axis rotation.	Unity
drZ	Rate of change of Z-axis rotation.	Unity
ddS	Rate of change of scale difference.	Unity
t0	The reference epoch for time-dependent parameters.	Float point value
fallback	Alias of transformation from target datum to WGS84 datum.	String

Example:

```
<Transformation epsg="1679" alias="Pulkovo42/2_to_WGS84" src="Pulkovo42/2"
 trd="WGS84" use="Param7"

method="CFR" tx="-40.595" ty="-18.55" tz="-69.339" ds="-4.299" rx="-2.508"
 ry="-1.832" rz="2.611"

accuracy="9" />
```

4.3.5 ProjectionCodes and Methods

These sections describe the **ProjectionCodes** and **Methods** that are mapped to AutoCAD® definitions of projections and transformations types. They are used to store the definitions in an XML file of coordinate reference system definition inside the **AcDbGeoData** object that represents a geographic location.

4.4 2D constraints

Parametric constraints are used to maintain relationships and control dimensions of 2D geometry.

There are two types of constraints:

- **Geometric** constraints control relationships between entities.
- **Dimensional** constraints are used to control the dimensions of an entity, such as length, angle, radius or diameter.

Note:

- It is recommended to first apply geometric constraints, then dimensional constraints.
- The PARAMETERCOPYMODE system variable controls how constraints and related parameters are processed when constrained entities are copied.
- The PARAMETRIZE2D command automatically creates geometric and dimensional parameters.

A dimensional constraint can use a value or a formula, with other dimension constraints or parameters. Dimensional constraints can be edited in the **Properties** panel, the **Parameters Manager** panel, and the **Mechanical Browser** panel.

You can set **dimensional constraints** (dynamic or annotative) as **reference parameters** from:

- The **Properties** panel by changing for the selected dimensional constraint/s the value of the **Reference** property from **No** to **Yes**.
- The **Mechanical Browser** panel by changing the value of the **Geometry-driven** field from **Off** to **On**.

Note: **Reference parameters** no longer constrain geometry. They display measurements and update when the geometry is modified. Thus, you can no longer manually edit the **Expression** field value. To do this, set back the reference parameter as a dimensional constraint.

The reference parameters are displayed within brackets in model space, paper space, and in the **Mechanical Browser** panel. All reference parameters in the drawing are listed in the **Parameters Manager** under the **Reference Parameters** category.

When using constraints, a drawing can be in one of three states:

- **Underconstrained**: some entities are constrained. It might be impossible to apply some changes using modification commands and procedures.

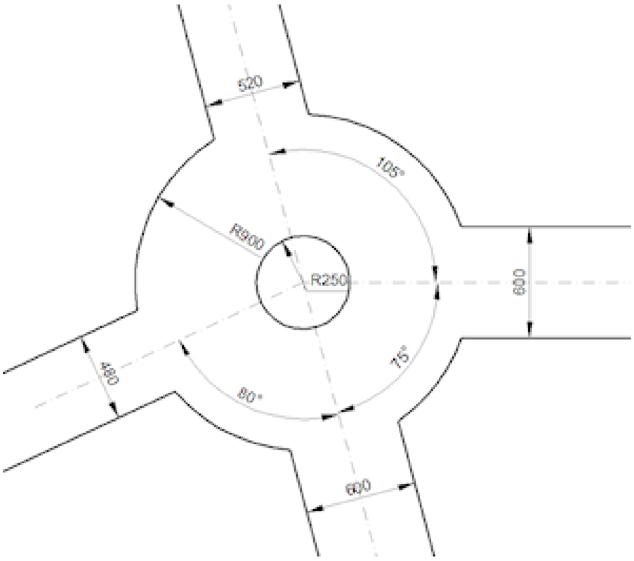
  Note: For underconstrained entities created with BricsCAD V23, you must set the 2DCONSTRAINTFLAGS system variable value to 1.

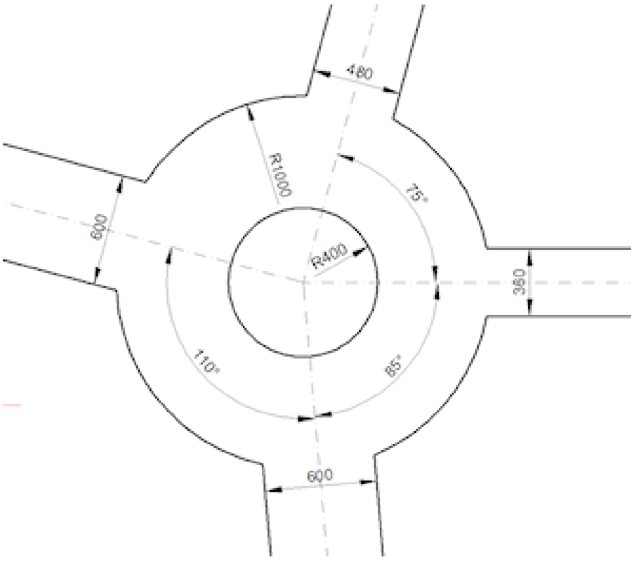
- **Fully constrained**: all possible and relevant geometric and dimensional constraints are applied to the design geometry. The drawing can be modified only by changing the values of the dimensional constraints.
- **Overconstrained**: one or more constraints violate or contradicts other constraints.

Parametric constraints allow to:

- Maintain geometric design intent.
- Make multiple versions of a design by applying different values to dimensional constrains.

Two versions of the same fully constrained drawing representing a roundabout:





4.4.1 Commands and toolbars

2D parametric constraints tools are located on the **2D Constraints** toolbar and the **Parametric** ribbon tab.

Tools to control the display of constraint bars and geometric constraints are located on two flyout toolbars: **Dimensional Constraint** and **Geometric Constraint**.

Command	Description
	Allows to set dimensional constraints to be displayed or hidden.
	Applies geometric relationships between entities and/or valid constraint points on entities.

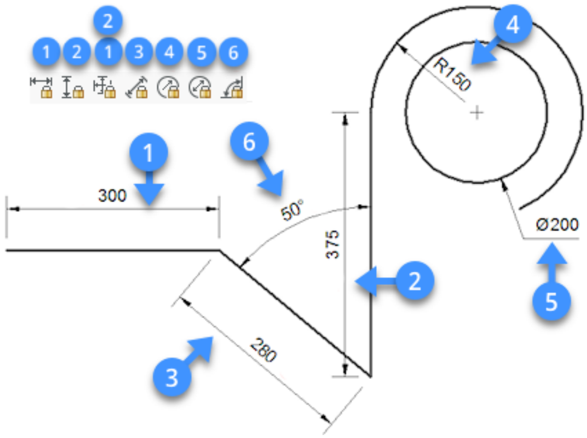
Command	Description
	Constrains points and entities at a fixed position.
	Applies a coincident geometric constraint to two points or constraints a point to an entity.
	Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
	Forces lines or polyline segments to be collinear.
	Forces two lines or linear polyline segments to be parallel to each other.
	Constrains two lines or polyline segments to lie perpendicularly to each other.
	Constrains two entities to maintain a point of tangency to each other or their extensions. The following entities are accepted: lines, polyline segments, circles, arcs, ellipses or elliptical arcs.
	Constrains lines, linear polyline segments or pairs of points to lie parallel to the X-axis of the current coordinate system.
	Constrains lines, linear polyline segments or pairs of points to lie parallel to the Y-axis of the current coordinate system.
	Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
	Constrains two entities or points to lie symmetrically with respect to a selected line.
	Constrains arcs and circles to the same radius, or lines and polyline segments to the same length.

Command	Description
	Controls the display of the constraint bar of entities. Constraint bars are always hidden when opening a drawing.
	Applies a dimensional constraint to an entity or between constraint points on entities. Converts associative dimensions to dynamic dimensions.
	Constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
	Constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.
	Constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
	Constrains the distance between two points or the length of a line or polyline segment.
	Constrains the radius of a circle, an arc or an arc polyline segment.
	Constrains the diameter of a circle, an arc or an arc polyline segment.
	Constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
	Converts an associative dimension to a dimensional constraint.
	Removes all dimensional and geometric constraints from an entity.

4.5 Using dimensional constraints

Dimensional constraints control the dimensions of 2D entities.

- Horizontal distance (1) (DCLINEAR and DCHORIZONTAL commands).
- Vertical distance (2) (DCLINEAR and DCVERTICAL commands).
- Distance between points or the length of lines and polyline segments (3) (DCALIGNED command).
- Radius of arcs, circles and arc polyline segments (4) (DCRADIUS command).
- Diameter of arcs, circles and arc polyline segments (5) (DCDIAMETER command).
- Angle between entities or between points and entities (6) (DCANGULAR command).



Note: Dimensional constraints are created using the current dimension style.

4.5.1 Editing dimensional constraints

Dimensional constraints can be edited:

- In the **Properties** panel.
- In the **Mechanical Browser** panel.
- In the **Parameters Manager** panel.
- Using the DDEDIT command.

You can set **dimensional constraints** (dynamic or annotative) as **reference parameters** from:

- The **Properties** panel by changing for the selected dimensional constraint/s the value of the **Reference** property from **No** to **Yes**.
- The **Mechanical Browser** panel by changing the value of the **Geometry-driven** field from **Off** to **On**.

Note: **Reference parameters** no longer constrain geometry. They display measurements and update when the geometry is modified. Thus, you can no longer manually edit the **Expression** field value. To do this, set back the reference parameter as a dimensional constraint.

The reference parameters are displayed within brackets in model space, paper space, and in the **Mechanical Browser** panel. All reference parameters in the drawing are listed in the **Parameters Manager** under the **Reference Parameters** category.

Editing dimensional constraints in the Properties panel

1 Select the dimensional constraint.

1 The properties of the entity display in the **Properties** panel.

2 Type a new value or expression in the **Expression** field, then press **Enter**.

The readout of the **Value** field changes accordingly.

3 (Optional) Edit the **Name** field.

4 (Optional) Edit the **Description** field.

5 (Optional) Select the **Reference** field, then click the down-arrow button and choose an option. A new parameter gets the **Off** option by default.

6 (Optional) Edit the **Text rotation** field.

7 Press the **Esc** key to unselect.

Note: Select multiple dimensional constraints simultaneously to set their value equally.

Editing dimensional constraints in the Mechanical browser panel

1 If not already open, open the **Mechanical Browser** panel.

2 Select a name of the dimensional constraint in the Parameters list.

3 The properties of the selected parameter are displayed at the bottom of the **Mechanical browser** panel.

4 (Optional) Select the **Name** field, then type a new name.

5 (Optional) Select the **Expression** field, then type a new value or expression.

6 (Optional) Select the **Geometry-driven** field, then click the down-arrow button and choose an option. A new parameter gets the **Off** option by default. If set to **On**, it disables the **Expression** field. The value of a parameter value is obtained from the model and can be changed with direct modeling operations.

7 (Optional) Select the **Description** field, then type a new description.

8 (Optional) Set the **Exposed** field, which specifies whether the parameter is exposed when the drawing is inserted in another drawing using the BMINSERT command.

The options are:

- **Auto**: the parameter is exposed only if it does not depend on other parameters.
- **ON**: the parameter is always exposed.
- **OFF**: the parameter is never exposed.

9 (Optional) Set the **Units** property: linear, planar or volume.

Select the field, then click the down-arrow button and choose an option.

Using the DDEDIT command

1 Do one of the following:

- Double click the dimensional constraint.
- Launch the DDEDIT command.

You are prompted: _DDEDIT

The **Edit Text** dialog box displays.

2 Type a new value in the **Text** field, then press Enter or click the **OK** button.

3 (Optional) Click another dimensional constraint and repeat the previous step.

4 Do one of the following:
   - Keep editing more dimensional constraints.
   - Press Enter or right click to stop.

     All edited dimensional constraints are updated simultaneously.

4.5.2 Setting the visibility of dimensional constraints

Dimensional constraints are hidden when a drawing is closed and reopened.

Tools to control the display of dimensional constraints are located on:

- the **Dimensional Constraint** toolbar.
- the **2D Parametric** ribbon tab.
- the **Show/Hide 2D Constraints** fly-out menu of the **Parametric** menu.

These tools execute the DCDISPLAY command using a specific option.

The options are:

- **Show/Hide**: prompts you to select the entities.
- **Show All**: show all dimensional constraints in the drawing.
- **Hide All**: hide all dimensional constraints in the drawing.

Note:

- The DYNCONSTRAINTMODE system variable controls the visibility of hidden dimensional constraints when selecting entities:
   - ON: display dimensional constraints when entities are selected.
   - OFF: dimensional constraints remain hidden when entities are selected.
- When hovering over an entity with a dimensional constraint, the blue constraint glyph ( ) displays if the SELECTIONPREVIEW system variable is set to 1 or 3.

4.5.3 Using expressions to define dimensional constraints

Dimensional constraints are named automatically upon creation.

The naming rules are as follows:

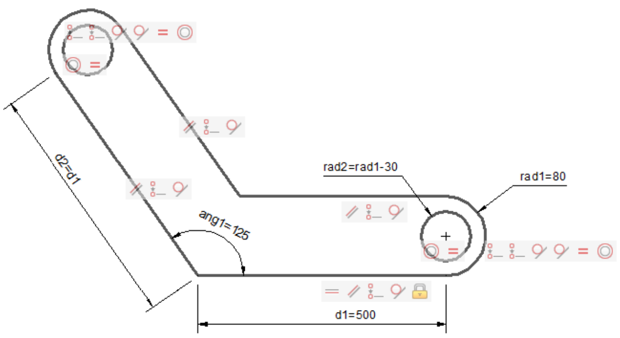
- distance: d1, d2, d3, ...
- radius: rad1, rad2, rad3, ...
- diameter: dia1, dia2, dia3, ..
- angular: ang1, ang2, ang3, ..

The name of a constraint can be used as a parameter in a mathematical expression to define another dimensional constraint.

The dimensions of the bracket below are controlled through the **rad1** radius, **d1** distance and **ang1** angular dimensional constraints.

The radius of the hole (**rad2**) and the length of the second leg (**d2**) are defined by expressions.

Geometric constraints force the width of the second leg and the hole in it to be equal to the first leg.



Using operators in expressions

The following operators can be used in expressions:

Operator	Description
+	Addition
-	Subtraction or Negative
*	Multiplication
/	Division
^	Exponentiation
%	Modulo or Remainder operator The expression "5 % 2" would evaluate to 1, because 5 divided by 2 leaves a quotient of 2 and a remainder of 1.

Expressions are evaluated according to the standard mathematical rules of precedence:

1 Expressions within brackets; innermost sets first.

2 Standard operations order:
   - a exponent
   - b multiplication and division
   - c addition and subtraction

3 Operators of equal precedence from left to right.

Using functions in expressions

The following functions can be used in expressions:

Function	Syntax
Sine	sin(expression)
Cosine	cos(expression)
Tangent	tan(expression)
Arc sine	asin(expression)
Arc cosine	acos(expression)
Arc tangent	atan(expression)
Hyperbolic sine	sinh(expression)
Hyperbolic cosine	cosh(expression)
Hyperbolic tangent	tanh(expression)
Arc hyperbolic sine	asinh(expression)
Arc hyperbolic cosine	acosh(expression)
Arc hyperbolic tangent	atanh(expression)
Square root	sqrt(expression)
Signum function (-1,0,1)	sign(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Round down	floor(expression)
Round up	ceil(expression)
Absolute value	abs(expression)

Function	Syntax
Largest element in array	max(expression1;expression2)*
Smallest element in array	min(expression1;expression2)*
Degrees to radians	d2r(expression)
Radians to degrees	r2d(expression)
Logarithm, base e	ln(expression)
Logarithm, base 10	log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Power function	pow(expression1;expression2)*
Random decimal between 0 and 1	Random

* Use the list separator character as defined on your system: , (comma) or ; (semicolon).

Creating a new parameter

1 Right click **Parameters** in the **Mechanical Browser** panel.

A context menu displays.

2 Select **Add new parameter** in the context menu.

A new parameter is created.

3 Edit the properties of the new parameter.

4 See the **Editing dimensional constraints in the Mechanical browser panel** procedure above.

Note:

- The constants **Pi=3.14...** and **e=2.72...** can be used in expressions. The constants names are not allowed to be used as a parameter or constraint name.
- The CLEANUNUSEDVARIABLES command purges parameters that are not used in constraint expressions or linked to dimensions.

4.6 Using geometric constraints

Geometric constraints:

- Specify a geometric relation between 2 entities (coincident, concentric, collinear, parallel, perpendicular, tangent, smooth, symmetric, equal).

- Specify a fixed angle (horizontal, vertical).
- Specify a fixed location (fix).

When a geometric constrained is applied to an entity:

- The position of the entity is adjusted according to the applied constraint.
- An icon displays next to the entity, to indicate the applied constraint. If multiple geometric constraints are applied, the icons are joined in a constraint bar.

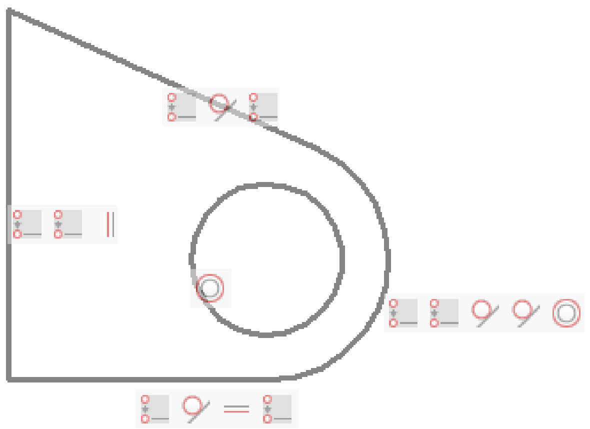
4.6.1 Creating geometric constraints

Tools to create dimensional constraints are located in:

- The **2D Constraints** toolbar.
- The **Parametric ribbon** tab.
- The **2D Constraints** fly-out menu of the **Parametric** menu.

In the image below:

- The endpoints of the three lines and the arc are joined by coincident constraints.
- The midpoint of the circle and the arc are made concentric.
- Two lines are forced to be tangent to the arc.
- One line has a vertical constraint (= parallel to the Y-axis of the current coordinate system), one line has a horizontal constraint (= parallel to the X-axis of the current coordinate system).



4.6.2 Displaying constraint bars

Constraint bars are hidden when a drawing is closed and reopened.

Tools to control the display of constraint bars are located on:

- The **Geometric Constraint** toolbar.
- The **Parametric** ribbon tab.
- The **Show/Hide 2D Constraints** fly-out menu of the **Parametric** menu.

These tools execute the CONSTRAINTBAR command using a specific option.

The options are:

- **Show/Hide**: prompts you to select the entities.
- **Show All**: show all dimensional constraints in the drawing.
- **Hide All**: hide all dimensional constraints in the drawing.

Note:

- The value of theCONSTRAINTBARDISPLAYsystem variable controls the visibility of geometric constraints:
  - 1: Display constraint bars when geometric constraints are added.
  - 2: Display hidden dimensional constraints when the constrained entities are selected.
- When hovering over an entity with a geometric constraint, the blue constraint glyph ( ) displays if theSELECTIONPREVIEWsystem variable is set to 1 or 3 or when the Quad is active.

4.6.3 Controlling the position of a constraint bar

By default constraint glyph bars are created close to the midpoint of the entity and are kept at that relative position when the entity position changes. You can drag the constraint bar to a different location. This new relative position is then maintained until the**Reset**option of theCONSTRAINTBARcommand restores the default position of the constraint bar.

Relocating a constraint bar

1 Place the cursor on the constraint bar.

2 Press and hold the left mouse button to move the constraint bar.

3 Release the left mouse button at the desired location.

Restoring the default position of constraint bars

1 Do one of the following:

- Click the**Show/Hide Geometric Constraint**tool.
- Launch the CONSTRAINTBAR command.

Prompts you:Select Entities.

2 Select the entities, then right click or press Enter.

BricsCAD reports the number of selected entities.

Prompts you: Select option for constraints [Show/Hide/Reset] <Show>:

3 Do one of the following:

- Choose**Reset**in the context menu.
- Type**R**in the Command line, then press Enter.

4.6.4 Working with constraint bars

Controlling a constraint

1 Move the cursor over a constraint icon.

2 A tooltip displays, indicating the constraint type.

3 The associated entity (or entities) highlights.

4 The corresponding icon on the constraint bar of the associated entity highlights.

Deleting a constraint

1 Move the cursor over the constraint icon in the constraint bar.

2 Right click, then click **Delete**.

3 The constraint is deleted and the icon is removed from the constraint bar and from the constraint bar of the associated entity.

Hiding the constraint bar of an entity

Move the cursor to the constraint bar, then click the **Close** (x) button.

4.6.5 Deleting constraints

To delete all constraints from a selection set:

1 Do one of the following:

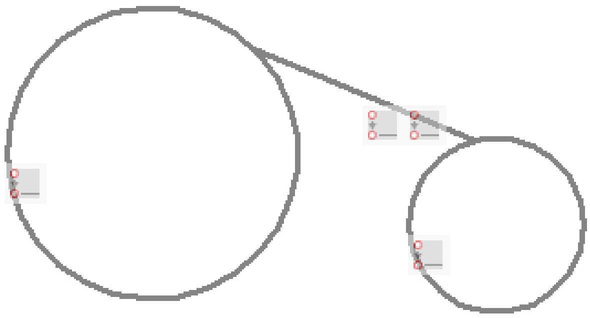
- Click the **Delete 2D Constraints** tool.
- Launch the DELCONSTRAINT command.

You are prompted: Select entities to delete all constraints:

2 Select the entities you want to delete the constraints from.

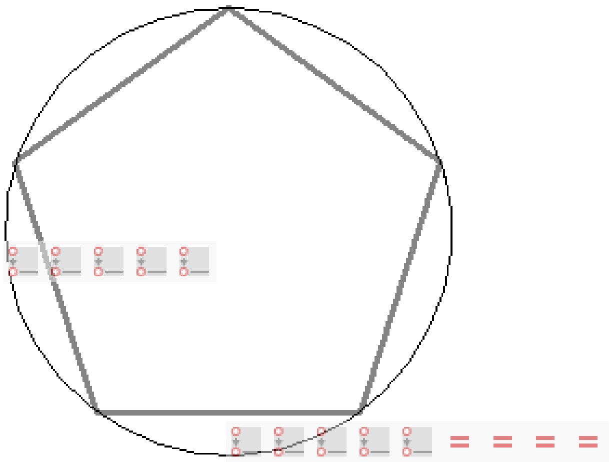
3 Right click to stop selecting entities and delete all constraints from the selection set.

4.6.6 Some examples

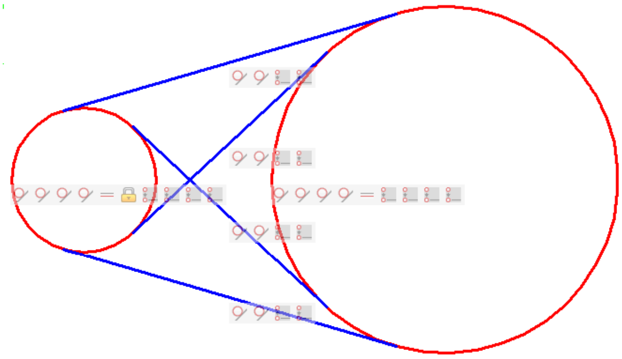


Coincident constraints between the endpoint of a line and two circles.

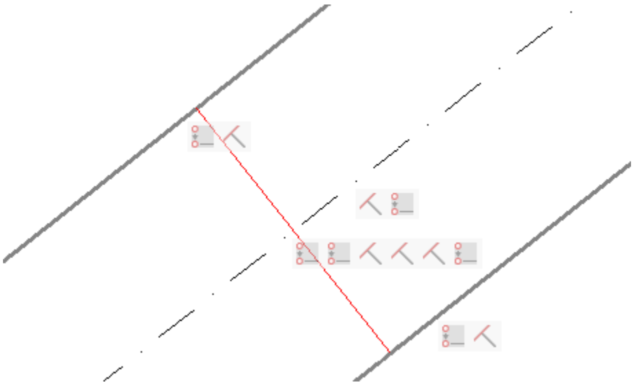
If the endpoints or center points of entities already coincide, the **Autoconstrain** option of the GCCOINCIDENT command automatically applies coincident constraints to such points.



By applying coincident constraints between the pentagon vertices and the circle, and equal constraints between one side and the four other sides, the circle radius defines the size of the pentagon.



Coincident constraints between the endpoints of the tangent lines and the circles prevent the tangent lines to extend beyond the tangent points.



Coincident constraints are used to:

- Force the endpoints of the red line to lie on the bold lines.
- Force the midpoint of the red line to lie on the dash-dot line.

The bold lines and the dash-dot line have a perpendicular constraint with the red line. As a result the dash-dot line always lies centered between the two bold lines.

4.7 Dynamic dimensions

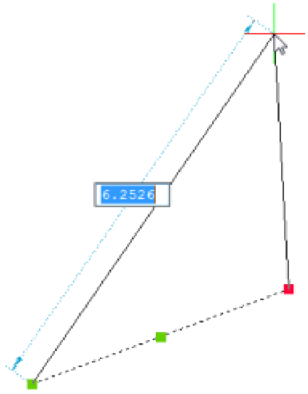
Dynamic dimensions provide an interface at the cursor position to specify the length and angle of the linear entity being created or grip-edited. As such, dynamic dimensions help you to focus on the drawing area.

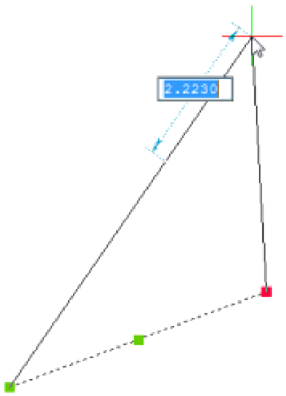
The behavior and appearance of dynamic dimensions is controlled through a series of system variables and user preferences.

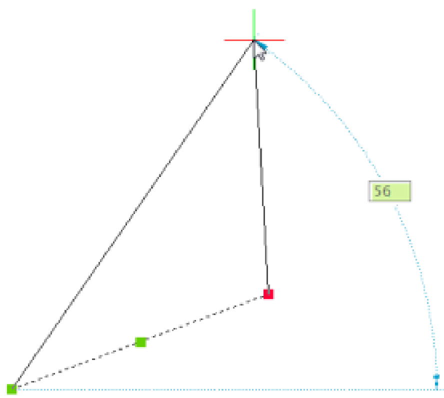
4.7.1 Defining the Dynamic Dimensions Settings

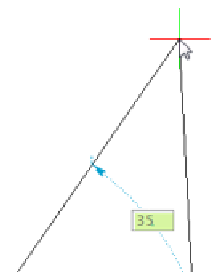
Right click the **DYN** field in the Status Bar, then choose **Settings** in the context menu. The **Dynamic input** settings in the **Settings** dialog box display.

Name	Title	Description
DYNMODE	Dynamic input mode	Controls whether and when dynamic dimensions display: • Editable dynamic dimensions: when creating or manipulating entities. • Tracking dynamic dimensions: when using entity snap tracking.

<table>
<tr><th>Name</th><th>Title</th><th>Description</th></tr>
<tr><td>DYNDIGRIP</td><td>Show dynamic dimensions</td><td>
Defines which dynamic dimensions display when creating or grip editing entities, depending on the value of the DYNDIVIS system variable
Resulting length: total length of the line or polyline segment


Extended length: incremental length of the line


Absolute angle: angle relative to the X-axis of the current UCS


Relative angle: angle relative to the original angle of the linear entity


</td></tr>
</table>

Name	Title	Description
DYNDIVIS	Dynamic dimension visibility	Defines the number of dynamic dimensions that display when creating or grip editing entities.
DYNPICOORDS	Default mode for dynamic coordinates input	Specifies the default mode in which coordinates are entered during dynamic input: Relative or Absolute.

Note:

- The appearance of dynamic dimensions (color, linetype, position and transparency) is controlled by a set of user preferences: DynDimColorHover, DynDimColorHot, DynDimDistance, DynDimLinetype, DynInputTransparency.
- If bitcode 16 (= format dynamic dimensions) of the PROPUNITS system variable is set, dynamic dimensions are formatted using a dedicated unit.
- Automatic unit conversion during length input can be used when bitcode 1 of PROPUNITS (Format length properties) is set. For example, when INSUNITS is set to **mm**, entering **2m** in a length property field will result in entering [2000.0000 mm]. Entering **1ft** gives [304.8000 mm]. If no unit is specified in the entry field, the unit as specified by INSUNITS is used.

4.7.2 Using dynamic dimensions when creating entities

1 Launch the command to create a 2D entity (e.g. LINE and DYNDIVIS = 1 or 2).

2 Specify the first point, then move the cursor to display the dynamic dimensions.

The **Length** dynamic dimension highlights.

3 Do one of the following:

- Type the desired length of the line, then press the TAB key to jump to the **Angle** dynamic dimension.

  The **Length** dynamic dimension is locked and displays in red and the **Angle** dynamic dimension is selected.
- Type the desired length of the line, then click to create the line at the angle defined by the cursor position.
- Press he TAB key to jump to the **Angle** dynamic dimension.

4 Do one of the following:

- Type the desired angle, then press the Enter key to create the line.
- Type the desired angle, then press the TAB key to return to the **Length** dynamic dimension.
- Press the Enter key to create the line at the angle defined by the cursor position.
- Press the TAB key to return to the **Length** dynamic dimension.

Note:

- If DYNDIVIS = 0, only one dynamic dimension displays. Press the TAB key to display the next field.
- If you type a value in the **Length** field, and add a comma (,) the content of the **Length** field is copied to the Command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates with respect to the previous point.

4.7.3 Using dynamic dimensions to grip-edit entities

1 Select the entity or entities.

All grips on the selected entities display.

2 Click the grip you want to edit, then move the cursor.

Depending on the value of the DYNDIVIS system variable one, two or all dynamic dimensions display simultaneously .

The first dynamic dimension highlights.

3 Do one of the following:

- Type the desired value for the currently highlighted dynamic dimension, then press the TAB key to jump to the next dynamic dimension.

  The dynamic dimension is locked and displays in red.
- Type the desired value for the currently highlighted dynamic dimension, then press Enter to stop.
- Press the TAB key to jump to the next dynamic dimension.

4 Repeat step 3 or press Enter to stop.

4.7.4 Using dynamic dimensions to measure entities

The following procedure applies to the 2DWireframe visual style only.

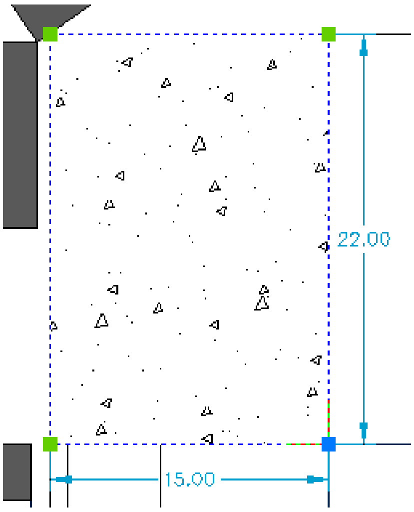
1 Select the entity or entities.

All grips on the selected entities display.

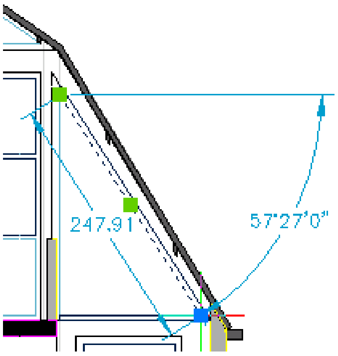
2 Hover the cursor over a grip.

The grip highlights and all dimensions related to the selected grip display.

Dynamic dimensions used to measure a rectangle



Dynamic dimensions used to measure a line



3 (option) Click to start editing the selected grip.

4.7.5 Using dynamic dimensions with tracking points

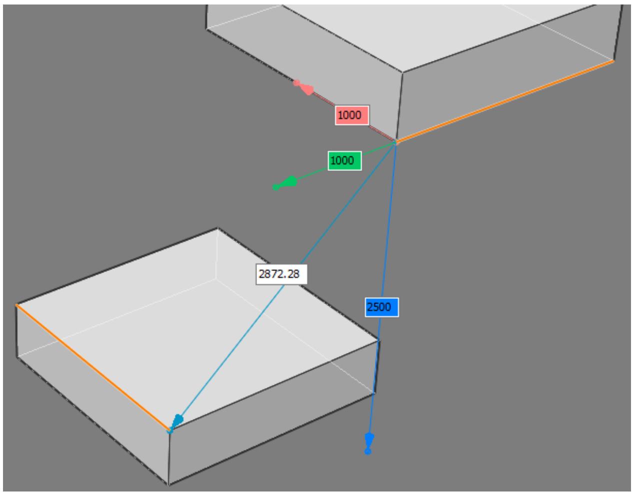
If the **Tracking dynamic dimensions** option of the DYNMODE system variable is set, dynamic dimensions display when entity snap tracking points or temporary tracking points are acquired.

4.8 Nearest distance

The Nearest Distance feature allows you to dynamically view and modify the distance between two selected entities or subentities. In addition, the aligned nearest distances along the X-, Y-, and Z-axes of the WCS are displayed.

Double clicking a Nearest Distance field:

- Toggles the display of the X-, Y- and Z-axis aligned distance fields.
- Zooms out to fully display the selected entities if necessary.



TheNEARESTDISTANCE system variable controls the display of the Nearest Distance feature.

4.8.1 Using Nearest Distance

1 Select the first entity or subentity*.

2 Select the second entity or subentity*.

The nearest distance displays.

The arrowhead points to the second entity.

3 (option) Double click a distance field to zoom out to fully display the selected (sub)entities and/or toggle the display of the aligned nearest distances along the X-, Y- and Z-axes.

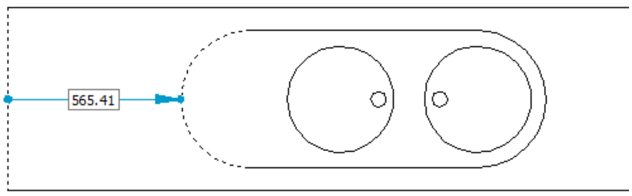
4 (option) Click a distance field and type a value.

The second entity is moved.

* Press and hold the Ctrl-key to select a subentity.

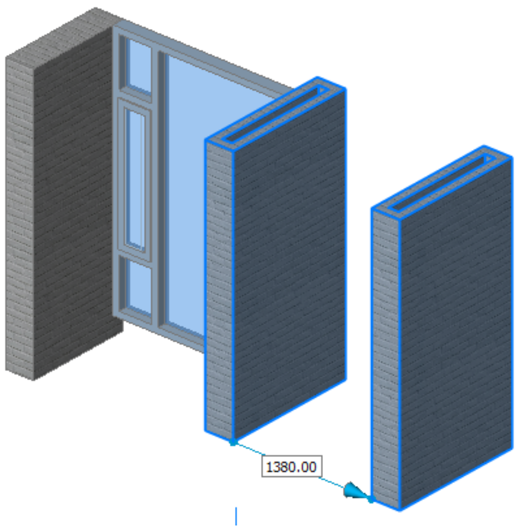
You can use the Nearest Distance feature to:

- Measure the distance between two entities or subentities.
- Adjust the distance between a (sub)entity and a block.

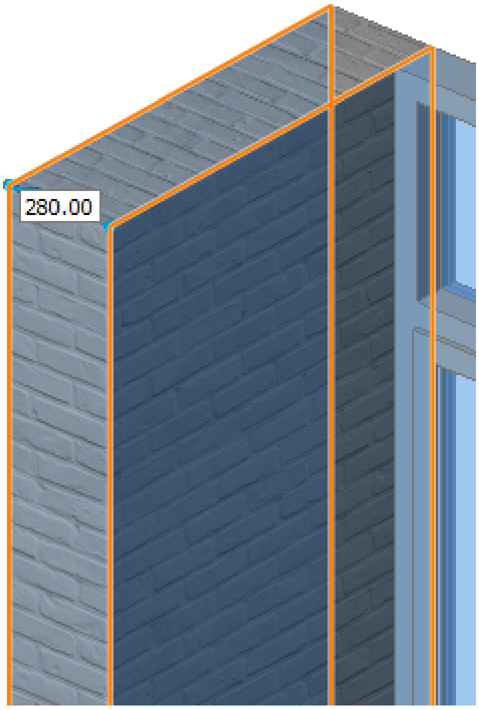


- Modify the dimensions of a rectangle.

- Adjust the distance between 3D solids.



- Edit the width of a 3D solid.



- Measure and edit the height of 3D solid.

4.9 Coordinate input

When you create entities in a drawing, they are located in relation to the underlying Cartesian coordinate system of the drawing. A drawing has a fixed coordinate system called the **World Coordinate System** (WCS).

You can also define arbitrary coordinate systems which are called **User Coordinate Systems** (UCS). They can be located anywhere in the WCS and oriented in any direction.

To specify points and distances using the keyboard, you can use the following formats:

- Cartesian coordinates: x,y,z
- Cylindrical coordinates: R<alpha,z
- Spherical coordinates: R<alpha<beta

4.9.1 Relative coordinates

When you type the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called **Relative Coordinates**.

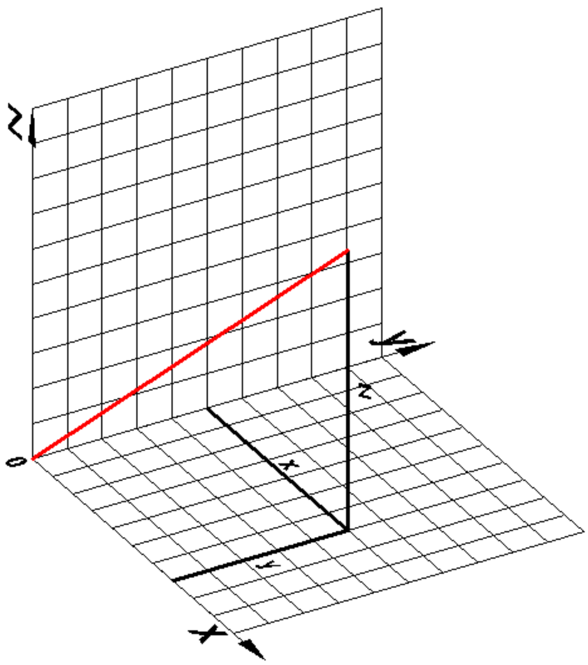
When Dynamic Dimensions are active and you type a value in the **Length** field and add a comma (,), the content of the **Length** field is copied to the Command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates with respect to the previous point.

4.9.2 Absolute coordinates

When a user-defined coordinate system is active, you can enter absolute coordinates (World coordinates) if you add an asterisk (*) in front. For example, *0,0 refers to the origin of the WCS (World Coordinate System).

4.9.3 Working with Cartesian coordinates

The Cartesian coordinate system uses three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis respectively, the y-axis and the z-axis define vertical planes. A point is defined by its distances to the yz-, xz- and xy- planes. These distances are called the xyz-coordinates of a point.



- Type the x-, y- and z- coordinates separated by commas to enter the absolute Cartesian coordinates of a point.
- When the z-coordinate is omitted, the point is placed in the xy-plane (Z = 0).
- Type the @-character in front of the entry (@x,y) to specify the coordinates with respect to the previous point. This technique is called **Relative Cartesian coordinates**.

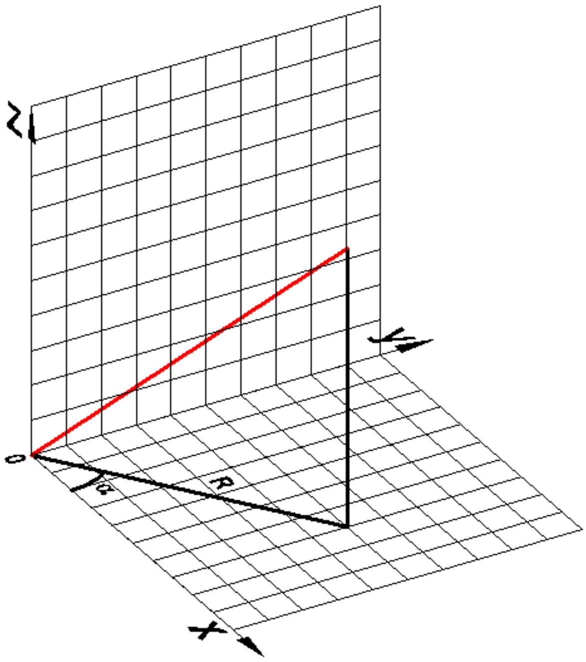
4.9.4 Working with cylindrical coordinates

A cylindrical coordinate system uses three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis respectively, the y-axis and the z-axis define vertical planes.

A point is defined using the following format: R<alpha, z.

- R = distance to the origin in the xy-plane.

- <alpha = the angle between R and the x-axis (positive angles are measured counter clockwise)
- z = the height above the xy-plane.
- When the z-coordinate is omitted, cylindrical coordinates are referred to as polar coordinates.
- Type the @-character in front of the entry to calculate the coordinates with respect to the previous point.

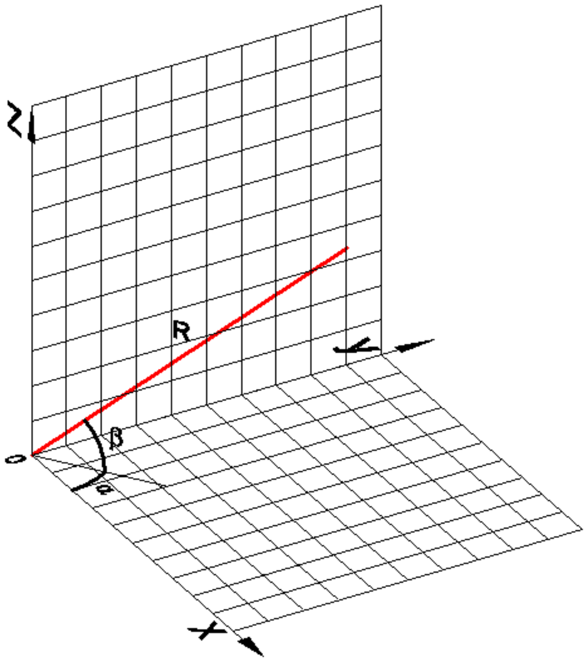


4.9.5 Working with spherical coordinates

The spherical coordinate system uses three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis respectively, the y-axis and the z-axis define vertical planes.

A point is defined using the following format: R<alpha<beta

- R = distance from the origin
- <alpha = angle in the xy-plane (positive angles are measured counter clockwise)
- <beta = angle measured from the xy-plane (positive angles are measured counter clockwise, above the xy-plane)

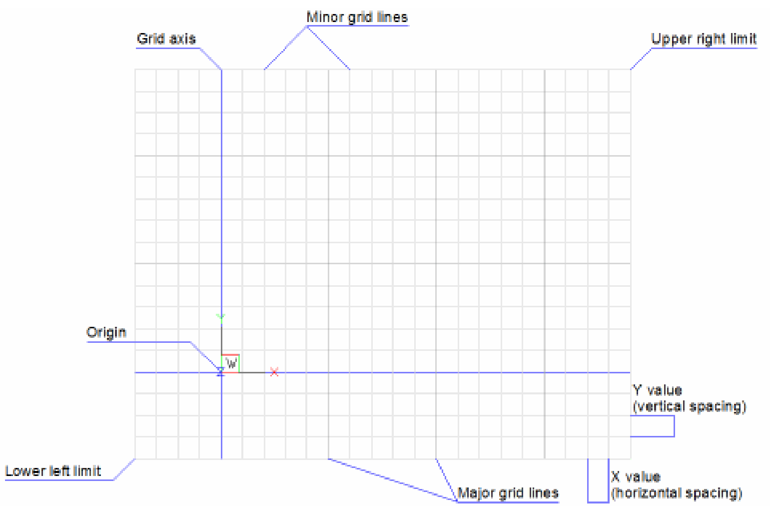


4.10 Snap and grid

Grid and snap help you to draw fast and accurately.

A grid is a set of evenly spaced lines or dots that serve as a visual distance reference.

The GRIDSTYLE system variable controls whether grid lines or dots display. The GRIDDISPLAY system variable controls the display of the grid. The grid can also indicate how far the drawing limits extend.



The snap feature creates a set of evenly spaced, invisible magnetic points, which make the crosshairs move in even increments. Both grid and snap are like the intersection points of the lines on a piece of grid paper. Grid points are for visual reference only and they do not print. Snap constrains the points that you can pick with the mouse.

- Click the GRID tool () on the Settings toolbar or the GRID field in the Status Bar, or press Ctrl+G to toggle grid display on/off.
- Click the SNAP tool () on the Settings toolbar or the SNAP field in the Status Bar, or press Ctrl+B to toggle snap on/off. The setting is saved through the SNAPMODE system variable.
- Both SNAP and GRID can be set differently in each viewport.
- The Snap option of the GRID command equals the grid spacing to the current snap spacing.
- Grid lines or dots do not print.

4.10.1 Displaying the Snap and grid settings

Do one of the following:

- In the Status Bar, right click on the SNAP or GRID field, then choose Settings in the context menu.
- In the **Settings** dialog box go to: **Drawing** > **Drafting** > **Coordinate Input** > **Snap/Grid**.

4.10.2 Synchronizing snap and grid spacing

1. Launch the GRID command.

   Prompts you:

   Grid is on. Enter grid spacing or [grid OFf/Snap/Aspect] <10>:
2. Type **s** , then press Enter.

   The grid spacing matches the snap spacing.
3. (Option) Use the **Aspect** option of the SNAP command to adjust the spacing.

Note: The grid spacing remains in sync with the snap spacing until you set the grid spacing differently using the GRID command or in the **Settings** dialog box.

4.10.3 Using drawing limits

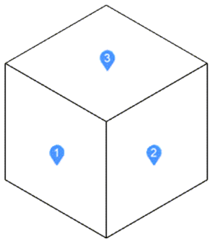
- Execute the LIMITS command to set the drawing limits.
- Click the **Drawing Limits** item () in the Settings menu to toggle the LIMCHECK system variable on/off.

  If LIMCHECK = ON, it is not possible to draw outside the drawing limits.
- The **Display beyond LIMITS area** option of the GRIDDISPLAY system variable defines whether or not the grid displays beyond the drawing limits.

4.10.4 Using isometric snap

Isometric grid allows to create two-dimensional isometric drawings, which simulate a three-dimensional view on a two-dimensional plane. Do not confuse isometric drawings with three-dimensional drawings.

The isometric option always uses one of three preset planes, which are denoted as **Left** (1), **Right** (2) and **Top** (3). You cannot alter the arrangement of these planes. The three isometric axes are 30 degrees, 90 degrees, and 150 degrees.



Do the following:

- Set SNAPSTYL = 1 (isometric Snap).
- Set GRIDSTYLE = 1 (dotted grid).
- Set SNAPISOPAIR: 0 = Left, 1 = Top, 2 = Right.

  Press the F5 function key to cycle the isometric planes.
- Set ORTHOMODE = On to constrain the movement of the crosshairs to the current isometric plane.
- Make sure SNAPANG = 0.

4.10.5 Setting the snap angle

You can rotate a dotted grid using the SNAPANG system variable (if SNAPANG is different from 0, grid lines do not display).

1 Display the **Snap/Grid** settings in the Settings dialog.

2 Select SNAPANG system variable.

3 Type a new value in the setting field.

4 Close the Settings dialog.

Note: You can also set the Snap Angle by typing **snapang** in the Command line. When in a command, type **'snapang** (with an apostrophe in front), to set the Snap Angle variable transparently (= without interrupting the running command).

4.11 Adaptive grid snap

The adaptive grid snap feature lets you create, edit,nudgeandmanipulateentities using concise values without keyboard entry.

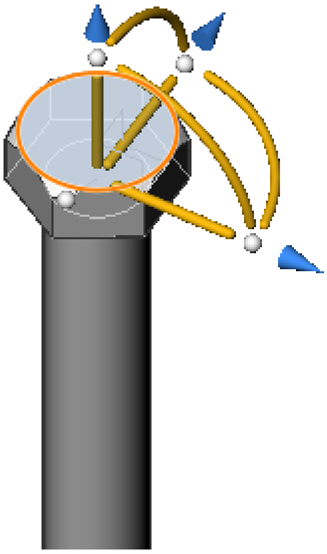
The snap step size is adapted according to the current screen zoom factor. The adaptive grid snap is active ifSNAPTYPE = 2.

The ADAPTIVEGRIDSTEPSIZE system variable controls the smallest available step size. The default value is 4 pixels.

4.11.1 Using adaptive grid snap with the Manipulator

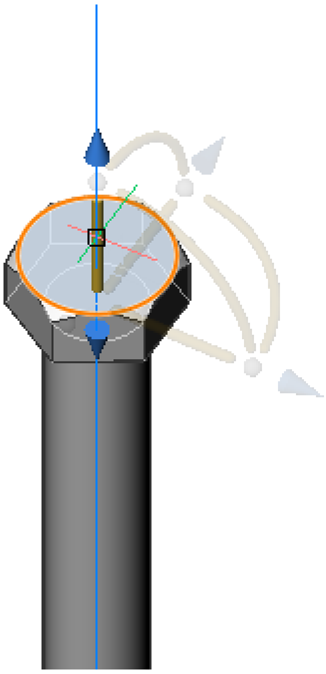
1 Move the cursor over an entity or subentity, then hold down the left mouse button a little bit longer to select the entity (= long left click).

1 The Manipulator widget displays:



2 Move the cursor over the axis in the direction you want to move the entity. The selected axis highlights; the other parts of the Manipulator are faded.

3



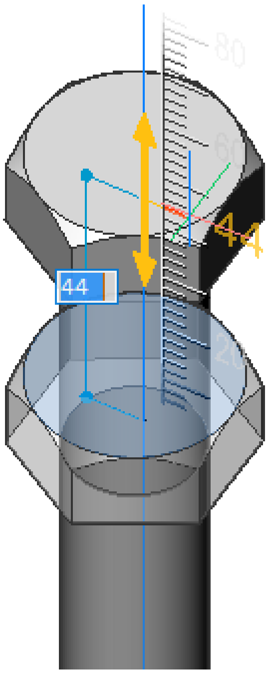
4 Click to select the axis, then move the cursor.

5 The Ruler displays.

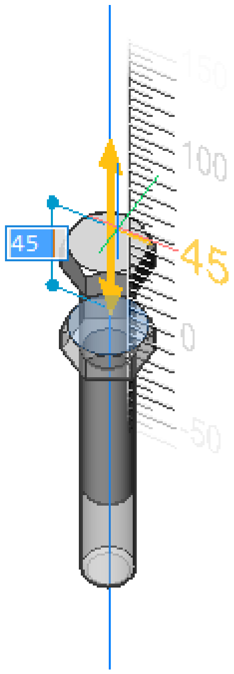
The current displacement distance displays in a dynamic field.

The grid snap resolution automatically adapts according to the zoom factor.

Ruler divisions at 2 mm:



Ruler divisions at 5 mm:



6 (Optional) Zoom in or out to adjust the dynamic grid.

7 Do one of the following:

- Click to accept the current value in the Ruler.
- Type a value in the dynamic field, then press Enter.

4.12 Nudge

The Nudge drawing aid allows you to move selected entities or sub-entities in small increments along the X, Y, or Z axis using the keyboard.

Note: Entities from a locked layer cannot be nudged.

4.12.1 Using Nudge

1 Select one or more entities or subentities (for example polyline segments or 3D solid faces).

2 Do the following to move the selection:

- Along the X axis of the current coordinate system:

  Hold down the Ctrl key, then use the left/right arrow keys.

  The Command line reads: NUDGE: offset along the X axis of the UCS: xx*

- Along the Y axis of the current coordinate system:

  Hold down the Ctrl key, then use the up/down arrow keys.

  The Command line reads: NUDGE: offset along the Y axis of the UCS: xx*

- Along the Z-axis of the current coordinate system:

  Hold down the Shift key, then use the up/down arrow keys.

  The Command line reads: NUDGE: offset along the Z axis of the UCS: xx*

* The step size used by the NUDGE functionality is based on theadaptive grid snapresolution. Zoom in/out to decrease/increase the step size.

4.13 2D entity snaps

Entity snaps enable you to quickly select exact geometric points on existing entities without having to know the exact coordinates of those points.

With entity snaps, you can select the end point of a line or arc, the center point of a circle, the intersection of any two entities, or any other geometrically significant position. You can also use entity snaps to draw entities that are tangent or perpendicular to an existing entity. You can use entity snaps any time you need to specify a point.

You can work with entity snaps in one of two ways:

- Enable a running entity snap that remains in effect until you turn it off by choosing an entity snap when no other command is active.
- Enable a one-time entity snap for a single selection by choosing an entity snap when another command is active. You can also use a one-time entity snap to override a running entity snap.

When using entity snaps, the program recognizes only visible entities or visible portions of entities. You cannot snap to entities on layers that have been turned off or to the blank portions of dashed lines.

If the APBOX system variable is on, the entity Snap Aperture Box is added to the crosshairs when one or more entity snaps are active.

When you move the cross hairs, the program snaps to the snap point closest to the center of the Snap Aperture Box. The **Snap Marker** indicates the current snap point.

The DRAGSNAP system variable controls the snap behavior while dragging, providing an increased WYSIWIG experience. DRAGSNAP controls whether rubber band dynamics are displayed at the current cursor location or at the current entity snap location. DRAGSNAP applies to all modify commands that display dynamics, such as COPY, PASTECLIP, PASTEBLOCK, MOVE, ROTATE, MIRROR, SCALE and STRETCH.

- DRAGSNAP = 0 (Default): dragged entities display at the cursor location.
- DRAGSNAP = 1: dragged entities display at the current entity snap location.

The OSOPTIONS system variable suppresses entity snaps on certain entity types. The value is stored as a bitcode using the sum of the values of the selected options.

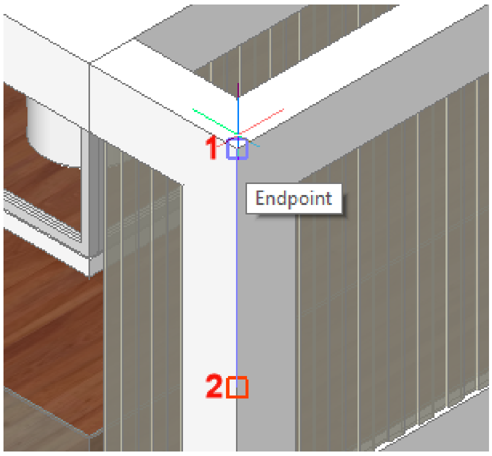
- 1: Entity snapping is disabled for hatch entities.
- 2: Entity snapping is disabled for geometry with negative Z values when using a dynamic UCS.
- 4: Entity snapping is disabled for endpoints of dimension extension lines.

Note: Press the TAB key, without moving the mouse, to cycle through all possible entity snaps available at the current cursor position.

4.13.1 Defining the 2D Entity Snap system variables

Right click the ESNAP field in the Status bar then choose settings in the context menu. The **Settings** dialog box displays with the 2D and 3D entity snaps settings expanded.

4.13.2 2D Entity Snap options

Icon	Description
	Sets the OSNAPZ system variable to Ignore entity snap elevation, replacing the Z coordinate of the snapped point with the current value of the ELEVATION system variable.  The 3d snap marker (1) displays the identified 3d point, the snap marker (2) displays at the current Elevation height.
	Disables the Ignore hatches option of the OSOPTIONS system variable, thus allowing to snap to hatches.
	Disables the Entity snap to negative-z option of the OSOPTIONS system variable, thus allowing entity snaps with a negative z-value when a Dynamic UCS is active.
	Disables the Ignore endpoints of dimension extension lines option of the OSOPTIONS system variable, thus allowing entity snaps to dimension extension lines.

4.13.3 Entity Snap Modes

Name	Icon	Marker	Context menu shortcut*	Transparent shortcut**	Description
TK				tk	Use temporary tracking points. Prompts you to identify temporary points; press Enter to accept the last temporary point.
MTP			2	mtp	Snaps to a point in the middle between two points. Prompts you to identify two points.
ENDPOINT			E	end	Snaps to the nearest endpoint of an entity or polyline segment.
MIDPOINT			M	mid	Snaps to the midpoint of an entity or polyline segment.
NEAREST			N	nea	Snaps to the nearest point on an entity.
CENTER			C	cen	Snaps to the center point of an arc, circle, polygon, ellipse or elliptical arc.
GEOMETRIC CENTER			G	gce	Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids.
PERPENDICULAR			P	per	Snaps to the perpendicular point of another entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline or edge of a plane to form a perpendicular alignment with that entity or with an extension of that entity.
PARALLEL			L	par	Displays a tracking line parallel to the selected entity.
TANGENT			T	tan	Snaps to the point on an arc, ellipse, spline or circle that, when connected to the previous point, forms a line tangent to that entity.

Name	Icon	Marker	Context menu shortcut*	Transparent shortcut**	Description
QUADRANT			Q	qua	Snaps to the closest quadrant of an arc, circle, ellipse, or elliptical arc.
INSERTION			I	ins	Snaps to the insertion point of an attribute, block or text entity.
NODE			O	poi nod	Snaps to a point entity.
INTERSECTION			R	int	Snaps to the intersection of any combination of entities.
APPARENT INTERSECTION			A	app	Snaps to the apparent intersection in the current view of two entities that do not intersect in three-dimensional space. Snaps also to the intersection of any combination of entities.
EXTENSION			X	ext	Snaps to the extension of an entity or to the intersection of the extension of two entities.
NONE			Y	non	Turns off all entity snap modes.

* Type this letter to quickly select the entity snap mode in the right click context menu.

** Type the transparent shortcut in the Command line when a command prompts for a point.

Other running entity snaps are temporarily disabled.

4.13.4 Setting the Entity snaps

Do one of the following:

- Press and hold the Shift key, then right click and select the Entity Snap mode in the context menu.

  The icons of the currently active Entity Snap modes are outlined.
- Click the buttons on the Entity Snaps toolbar.

  The buttons of the currently active entity snap modes are pressed.

When no command is active, the above procedures toggle the Entity Snap modes on/off.

When a command is active, the above procedures set a 'one shot' override of the current Entity Snap modes.

Alternatively, you can type a single letter shortcut as indicated in the table below.

To toggle the Entity Snaps on/off:

- Click the ESNAP field in the Status Bar.
- Press the **F3** function key.

4.13.5 Working with multiple Entity Snap modes

1 Move the cursor to the entity you want to snap.

One of the active Entity Snap modes markers display.

2 Do not move the mouse and press the TAB key.

The entity the snap point is on highlights.

3 Do one of the following:

- Click to accept the snap point.
- Press the TAB key.

  The next possible Entity Snap mode marker displays.

  The entity the snap point is on highlights.

4 (option) Repeat step 3 until the right snap point is found.

4.13.6 Snapping to the extension of two entities

1 If not already on, turn on the **Extension** Entity Snap mode.

2 Launch an entity creation command, for example LINE.

3 Move the cursor over the endpoint of the first entity (1).

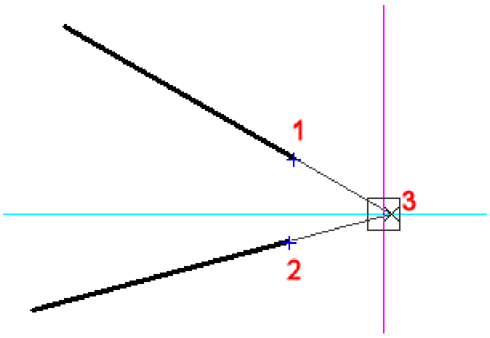
A small red cross indicates the entity is marked for extension.

4 Move the cursor over the endpoint of the second entity (2).

A small red cross indicates the entity is marked for extension.

5 Move the cursor near the intersection of the extensions of the two entities.

An X indicates the intersection (3).



6 Click to accept the snap point.

Note: You can snap to the extension of lines, polylines, arcs and elliptical arcs.

4.13.7 Using the Middle of 2 points snap mode

1 When prompted to specify a point, do one of the following:

- Activate the **Snap to the middle of 2 points** snap mode ( ).
- Type **mtp** or **m2p** in the Command line and press Enter.

You are prompted: First point of mid:

2 Specify the first point.

You are prompted: Second point of mid:

3 Specify the second point.

The point in the middle of the two points is snapped.

4.13.8 Using the Parallel entity snap mode

1 Make sure Entity Snap Tracking is active.

2 Launch an entity create command to create a linear entity (for example Line).

3 Specify the first point of the entity.

The line displays dynamically.

4 Press and hold the Shift key, then right click and choose **Snap to parallel** ( ) in the context menu.

5 Hover the cursor over the entity you want to draw parallel to.

The **Parallel Entity Snap** marker ( ) displays on the selected entity.

Wait until a small cross (+) indicates the entity is recognized.

6 Move the cursor in order to place the dynamics of the line more or less parallel to the selected entity.

A tracking line displays from the start point of the line.

7 Do one of the following to specify the second point while the tracking line displays:

- Click a point.
- Enter the desired length and press Enter.
- Use a second entity snap tracking point, then move the cursor to the intersection of the parallel tracking line and the second entity snap tracking line.

4.14 3D entity snaps

In addition to the2D entity snaps, a series of 3D entity snap modes apply to points on 3D entities such as solids, surfaces, splines and meshes.

The current 3D entity snap modes are saved through the 3DOSMODE system variable as the sum of the bitcodes of the selected snap modes.

Name	Icon	Marker	Description
Center			Snaps to the center of 3D faces.
Intersection			Snaps to the intersections of linear entities, edges, and polar or entity snap tracking lines with faces.
Knot			Snaps to knots (fit points) onsplines.
Midpoint			Snaps to the midpoint of a face edge.
Nearest			Snaps to a point on the face of a 3D entity that is nearest to the cursor.
None			Disables all 3D snap modes.
Perpendicular			Snaps to a point perpendicular to a face on a 3D entity.
Vertex			Snaps to the closest vertex of a 3D entity and the control vertices ofsplines.
Point Cloud			Snaps to point cloud points.

4.14.1 Setting the 3D Entity Snap modes

- Click the buttons on the**3D Entity Snaps**toolbar.

  The buttons of the currently active entity snap modes are pressed.
- When no command is running, clicking a tool button toggles the snap mode on/off.
- When executing a command, clicking a tool button sets the snap mode for the next point, temporarily disabling all other selected snap modes.
- Run the3DOSNAPor-3DOSNAP commands.
- Shift + Right click, hover over**3d Entity Snap**in the context menu and click the 3D entity snap option on the flyout menu.

4.15 Polar tracking

Polar tracking (POLAR) allows you to draw at exact angles:

- At fixed intervals, starting from 3 o'clock (East).
- At specific additional angles.

A polar tracking path line displays from the origin point when you move the cursor close to one of the polar tracking angles. The tracking path lines display as long as theSnap Aperture box overlaps the polar tracking line.

In 3D views an additional tracking line displays parallel to the Z-axis of the current UCS.

Snap track lock: to lock a snap track path hit the Shift key. To unlock a track path, hit Shift again: it works as a toggle. Locked tracking paths display in thecolor defined by the SnapMarkerColoruser preference.

- The display of the tracking path line is controlled through the TRACKPATH system variable.
- The color of the tracking path line is controlled through the AutoTrackingVecColor user preference.
- Tracking paths parallel to the X-, Y- or Z-axis of the current coordinate system (WCS or UCS) display in the color of the axis they are parallel with (defined by COLORX , COLORY or COLORZ variables).
- Turning onPOLARautomatically disablesORTHO (Orthogonal Mode) and vice versa.
- The ANGBASE system variable defines the base angle of the tracking lines with respect to the current coordinate system.

4.15.1 Defining the tracking settings

1 Do one of the following:
  - Right click on thePOLARor the STRACK field in theStatus Bar, then right click and select**Settings**in the context menu.
  - Open the**Settings**dialog and expand the**Coordinate input**subcategory under**Drafting**in the**Drawing**settings class.

    Expand the**Snap Tracking**settings group.

2 Adjust the settings:
  - AUTOSNAP system variable: controls the behavior of Polar Tracking and whether polar tracking and/or entity snap tracking is active.
  - POLARANG system variable: sets the polar angle increment in degrees; default = 90.
  - POLARADDANG system variable: sets additional angles; separate angles by semicolons.
  - POLARMODE system variable: controls the behavior of polar and entity snap tracking.
  - • Relative: the angles are measured with respect to the direction of the snapped entity; otherwise angles are measured according to the current coordinate system (WCS or UCS).
  - • Use polar angle settings for entity snap tracking also.
  - • Whether additional angles (POLARADDANG) are used or not.
  - • Use the Shift key to acquire entity snapping points.
  - TRACKPATH system variable: controls whether and how the track paths display: full screen or partially.

4.15.2 Locking a tracking line

Press the Shift key while a snap track marker line displays to lock a tracking path, any (snap)point that is picked will be projected perpendicularly onto the tracking path. Press the Shift key again to unlock.

A locked tracking path changes to the color defined by theSnapMarkerColoruser preference.

4.15.3 Using temporary tracking points

When**Polar Tracking** is ON, temporary tracking points can be used to pick points at locations where no existing geometry is available to snap directly to, while you know the desired offset distance(s) from snap points on existing geometry. It is recommended to havedynamic dimensions(DYN) active as well.

1 Whenever a command prompts for a point, do one of the following:
   - Click the middle mouse button (mouse wheel).
   - Activate the**Temporary tracking points**entity snap mode ( ).
   - Type**TK**in the Command line and press Enter.

     The command is suspended temporarily.

     You are prompted: Specify first temporary tracking point:

2 Enter a point, for example use an entity snap to specify a point.

   You are prompted: Specify next temporary tracking point (press ENTER to end).

3 Do one of the following:
   - Enter a point.
   - Press Enter to accept the last temporary tracking.

     The command resumes.

4.16 Entity snap tracking

Entity Snap Tracking(STRACK) assists you in drawing entities with respect to other entities.

WhenEntity Snap Trackingis active, BricsCAD displays temporary tracking lines from entity snap points.At least oneEntity Snapmode must be on if you want to useEntity Snap Tracking.

Click theSTRACKfield in theStatus barto enable/disableEntity Snap Tracking.

Entity Snap Trackinglines are parallel to the X- and Y-axis of the current UCS.

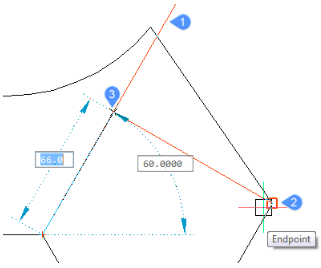
If bitcode 2 of the POLARMODE system variable is checked, the polar angles used inPolar Tracking(interval and additional angles) are also used inEntity Snap Tracking.

Alignment points are acquired when you hover over a point, using entity snaps. Acquired points display a small plus sign (+). To remove an acquired point, hover over the point again. The small plus sign (+) is then removed. If bitcode 8 of the POLARMODE is checked, press and hold the Shift key to acquire entity tracking points. Alignments relative to an acquired point are displayed as you move the cursor over their tracking paths. You can then define points either on the alignments or at the intersection of two alignments.

Note: The ANGBASE system variable defines the base angle of the tracking lines with respect to the current coordinate system.

4.16.1 Locking a tracking line

Press the Shift key when a tracking line displays to lock the tracking line. Locked tracking lines display in the color defined by theSnapMarkerColoruser preference (1). Entity snaps (2) or specified points are projected perpendicularly onto a locked tracking line (3). Press the Shift key again to unlock a tracking line.



More options with locked tracking lines are:

- If the Intersection entity snap is active, it is possible to snap to the intersections of a locked tracking line and entities in the drawing.
- Acquire additional tracking lines and snap to the locations where these intersect with the locked tracking line.

4.16.2 Using Entity Snap Tracking

1 Launch a drawing command, for example LINE.

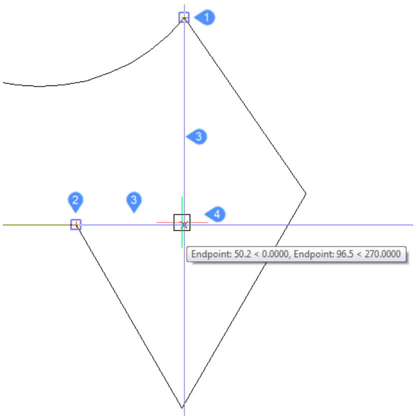
Prompts you: Start of line or [Follow] <Last point>:

2 Hover the cursor over the first tracking point (1).

A small plus sign (+) displays to indicate the tracking point is active.

3 Hover the cursor over the second tracking point (2).

A small plus sign (+) displays to indicate the tracking point is active.



4 Move the cursor close to position 4.
  - Tracking lines display from both tracking points (3).
  - A cross marker (x) indicates the intersection of the tracking lines.
  - Snap markers display at the tracking points.

- An Entity Snap Tracking tooltip at the intersection of the tracking lines shows the angles and distances from the tracking points.

5 Click to accept theEntity Snap Trackingpoint.

4.17 User coordinate systems

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. The ELEVATION system variable specifies the distance from the XY-plane of the current coordinate system. If ELEVATION = 0 (default), entities are created in the XY-plane of the current coordinate system.

Every drawing has a fixed coordinate system called the World Coordinate System (WCS).

You can define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS). You can create as many UCS as you want, saving or redefining them to help you construct entities in the three-dimensional space.

When you create a new drawing, the WCS is the current coordinate system by default, which is indicated by the letter W in the UCS icon. When you display a drawing in plan view, you see the coordinate system icon from the top, with the Z-axis directed straight toward you. When you display a three-dimensional drawing in a view other than plan view, the coordinate system icon changes to reflect the new viewpoint.

Note:

- You cannot delete or modify the WCS.
- When the UCSORTHO system variable is On, an orthographic UCS is restored automatically when the related orthographic view is restored by the -VIEW command or selected on the Look From control.

  An orthographic UCS has its origin at the WCS origin or the UCS referred to by the UCSBASE system variable.

  Orthographic views are: top, bottom, left, right, front, back.
- When multiple viewports exist, the UCSVP (UCS viewports) system variable controls whether the UCS in a viewport is fixed (UCSVP = On) or changes to match the UCS in the current viewport (UCSVP = Off).
- When the UCSFOLLOW system variable is On, the plan view, zoomed to extents, is generated automatically whenever the coordinate system changes.
- The PLAN command restores the Plan view (Top view) of the current coordinate system (UCS or WCS).

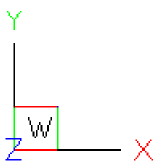
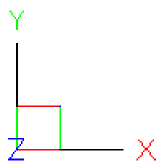
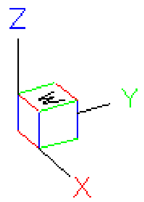
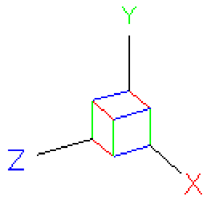
4.17.1 Controlling the display of the coordinate system icon

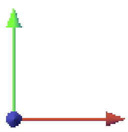
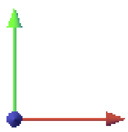
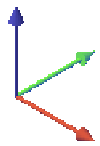
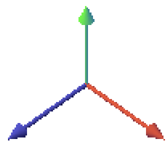
The UCSICON and UCSICONPOS system variables control the display and location of the UCS icon:

- UCSICON:
  - **Show icon**: Controls whether the icon shows or not.
  - **At origin** : Controls the location of the UCS icon: if On, the icon indicates the origin of the current coordinate system (UCS or WCS). However, if the origin is not within the viewport

borders, the UCS icon moves to the corner of the viewport, as defined by the UCSICONPOS system variable. When the **At Origin** option is not checked, the icon always displays in the corner of the viewport defined by UCSICONPOS system variable.

- UCSICONPOS : Defines the corner of the viewport for the non-origin location of the UCS icon.

2D Wireframe visual style			
 WCS Top View	 UCS Top View	 WCS Isometric View	 UCS Isometric View

Rendered Visual Styles			
			

To help you keep your bearings in the current coordinate system, BricsCAD displays a coordinate system icon (UCS icon). The visible portions of the axes are the positive directions.

The color of the three axes indicate the orientation of the three-dimensional space:

- X-axis: red
- Y-axis: green
- Z-axis: blue

The color of the axes is controlled by the COLORX, COLORY and COLORZ system variables.

4.17.2 Defining a UCS

1 Type **UCS** at the command prompt and press Enter

You are prompted:

Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:

2 Specify the origin point of the UCS.

You are prompted: Point of positive X axis <current point>:

3 Specify a point to define the positive X-axis.

The command window reads: Point in X-Y plane with positive Y value <current point>:

4 Specify a point to define the positive Y-axis.

The UCS is defined.

4.17.3 Defining and saving a User Coordinate System

1 Launch the EXPUCS command.

The **Drawing Explorer** - Coordinate Systems dialog box displays.

2 Click the New tool button on the Drawing Explorer dialog.

The BricsCAD Explore dialog closes.

You are prompted: Z Axis/3 point/Entity/Origin/View/X/Y/Z/Current <Current>

3 Press 3 point option (define the UCS using 3 points).

4 Specify the origin point of the UCS.

You are prompted: New origin <0,0,0>:

5 Specify a point to define the positive X - axis.

You are prompted: Point of positive X axis <1,0,0>:

6 Specify a point to define the positive Y- axis.

You are prompted: Point on the XY-plane with positive Y value <0,1,0>:

The UCS is defined.

The Drawing Explorer dialog reopens.

7 Click in the UCS Name field of the newly defined UCS to replace the <NewUCS> default name.

8 Close the Drawing Explorer dialog.

4.17.4 Restoring a saved UCS

1 Launch the EXPUCS command.

The **Drawing Explorer** - Coordinate Systems dialog box displays.

2 Click the blank tile in front of the UCS name to make it current.

3 Close the **Drawing Explorer** dialog box.

4.17.5 Restoring the WCS

Type **UCS** in the Command line, then press Enter twice (= choosing the default option of the UCS command).

4.18 Dynamic UCS

With the Dynamic UCS feature enabled, the coordinate system is aligned automatically with the entity under the cursor.

TheUCSDETECTsystem variable controls the supported entity types:

1 Faces of 3d Solids, surfaces, subdivision meshes and regions (the default).

2 Other entity types: point, line, polyline, 2D polyline, 3D polyline, ray, xline, arc, circle, ellipse, spline, text,

mtext, solid, 3Dface, trace, block insert, viewport, mline, leader, mleader, hatch, helix, camera, light, section, shape, pdf underlay and image.

Right click the DUCS field in the status bar to select the supported entity types.

Entities can be selected inside blocks, except curved faces of ACIS entities in non-uniformly scaled blocks.

These features are controlled by user preferences.

- RedHilite_DUCSLocked_Face_Colorcontrols the highlight color for the face to which the Dynamic UCS has been locked by pressing the Shift key.
- RedHilite_DUCSLocked_Face_Alphacontrols the highlight transparency for the face to which the Dynamic UCS has been locked by pressing the Shift key.

Note: It is recommended to set UCSFOLLOW = Off when using the dynamic UCS feature.

4.18.1 Toggling the dynamic UCS

Do one of the following:

- Press theF6function key.
- Click the**DUCS**field in thestatus bar.

4.18.2 Using the Dynamic UCS

1 With the Dynamic UCS feature active, launch an entity creation command, for example LINE.

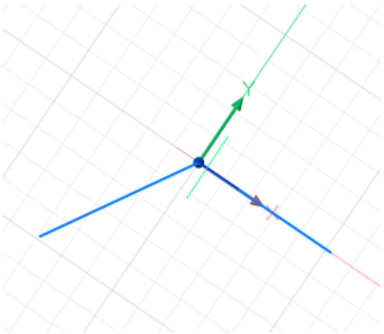
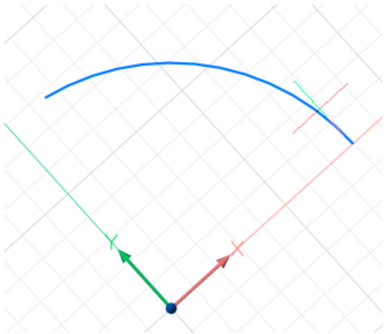
2 Move the cursor over an entity or 3D solid face.

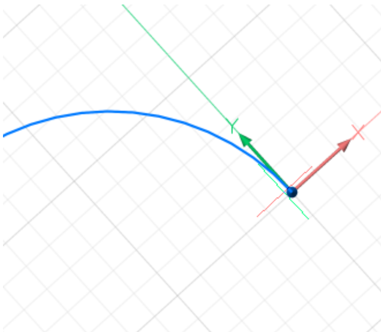
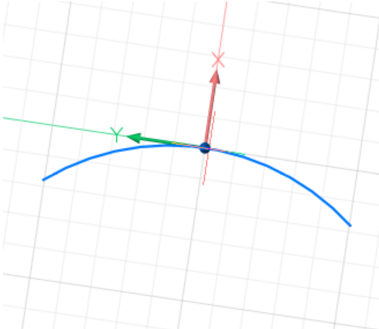
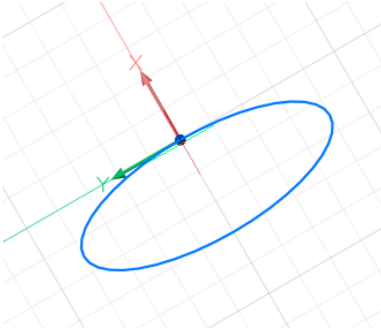
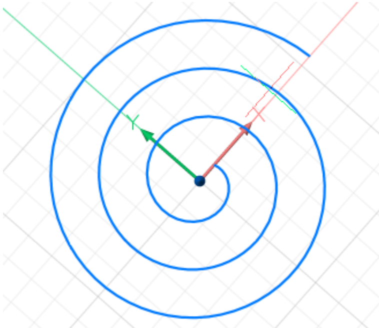
The entity or face highlights.

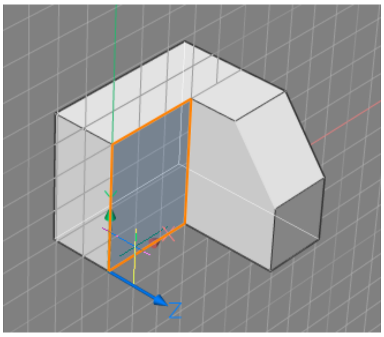
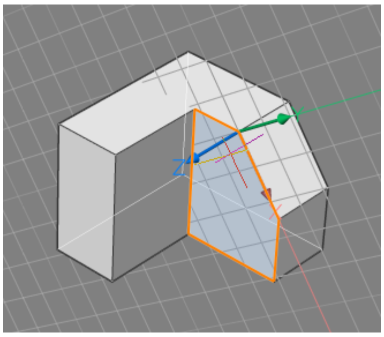
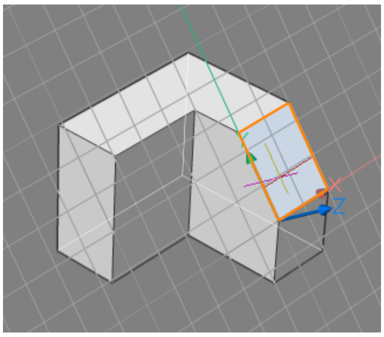
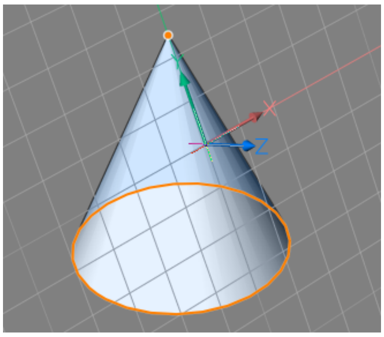
If GRIDMODE is On, the grid is aligned with the hovered entity or face.

3 IfUCSICON= 3 (On at Origin), the UCS icon displays.

4

Dynamic UCS on other entities	
	

Dynamic UCS on other entities	
	
	

Dynamic UCS on 3D solid faces	
	
	

The orientation of dynamic UCS depends on how the cursor touches the hovered entity or face This provides users a way to manipulate the orientation of the DUCS. On 3D solid faces, the origin of the DUCS is set to the starting point of the solid edge and the X-axis is chosen co-directional to the tangent at the starting point of the edge. On curved faces, the X-axis is tangent to the face, the Y-axis lies in the face and the Z-axis is perpendicular to the face.

5 (option) Tap the Shift key to lock the DUCS, which allows you to start drawing outside the hovered entity or face. Tap the Shift key again to unlock the DUCS.

6 Execute the entity creation command.

Note: When bitcode 2 of the OSOPTIONS system variable is selected, entity snap ignores negative Z-values in the current UCS.

7 When the command concludes, the current coordinate system (WCS or UCS) is restored.

4.19 Measuring

The DIST command reports the distance and angle between two points.

The AREA command finds the area and perimeter (or length) of 2D entities.

The MASSPROP command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for **mass properties**).

The ID command reports the x,y,z coordinates of picked points.

The LIST command lists the properties of selected entities.

Note: BricsCAD reports the results in the Command line or in the Prompt History window when the Command line is closed. Depending on the number of prompt history lines in the Command line, it can be necessary to press the F2 function key to open the Prompt History window. Press the F2 function key again to close the Prompt History window.

4.19.1 Measuring distances

To measure a distance between 2 points:

1 Launch the DIST command.

Prompts you: Starting point for distance.

2 Identify the first point.

Prompts you: Enter second point or [Multiple points]:

3 Hover over the second point using an appropriate entity snap mode.

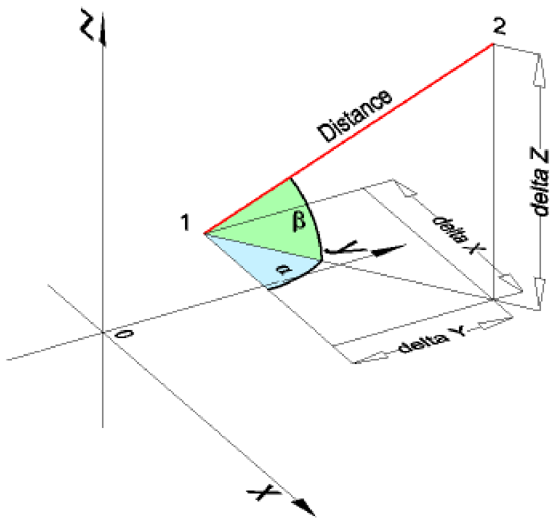
When Dynamic Input (DYN) is On, the distance displays in the dynamic dimension field.

4 Click to confirm the second point.

BricsCAD prints the results in the Command line:

Distance, Angle in XY Plane (α), Angle from XY Plane (β)

Delta X, Delta Y, Delta Z



The values are in drawing units and with respect to the current coordinate system.

To measure a distance between multiple points:

1 Launch the DIST command.

Prompts you: Starting point for distance.

2 Identify the first point.

Prompts you: Enter second point or [Multiple points]:

3 Choose the Multiple points option.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

4 Identify the second point.

BricsCAD displays the distance between the two points.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

5 Identify the next point.

BricsCAD displays the total distance between the points.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

6 Keep adding points, then press Enter or right click to conclude.

4.19.2 Measuring lengths

To measure the length of single linear entity, click the entity then read the **Length** field in the **Properties** panel.

To combine the lengths of a series of linear entities:

1 Launch the AREA command.

Prompts you:

Select first point [find area of one Entity/Add areas together/Subtract areas]:

2 Choose the **Add areas together** option.

Prompts you: Draw first point to add to current area or [add Entities to area/Subtract from area]:

3 Choose the **ad Entities** option.

Prompts you: Select entities to add to area:

4 Select the first linear entity.

BricsCAD reports in the Command line:

Area = xx, Length = xx

Total length = xx

Total area = xx

Prompts you: Select entities to add to area:

5 Select the second linear entity.

BricsCAD reports in the Command line:

Area = xx, Length = xx

Total length = xx

Total area = xx

Prompts you: Select entities to add to area:

6 Keep selecting linear entities or press Enter or right click to stop.

Note: You can combine the lengths of lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

4.19.3 Measuring areas

To measure the area and length of single closed entity, click the entity then read the **Area** and **Length** fields in the **Properties** panel.

To measure areas and perimeters by specifying points in a drawing:

1 Launch the AREA command.

Prompts you:

Select first point [find area of one Entity/Add areas together/Subtract areas]:

2 Click the first point.

Prompts you: Next point.

3 Click the second point.

Prompts you: Next point.

A dashed outline displays dynamically.

4 Keep adding points, then press Enter or right click.

BricsCAD reports in the command line:

Area = xx, Perimeter = xx

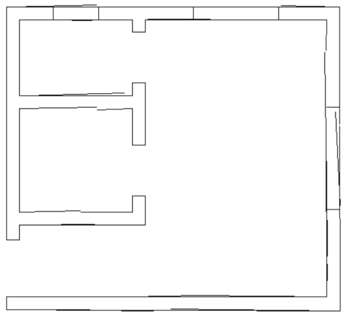
4.20 Workflow for OPTIMIZE, OVERKILL and SIMPLIFY commands

4.20.1 Context

The operation of exporting data from some 3D software and importing it to 2D many times might result in entities that are angled, overlapped and having gaps between them.

4.20.2 Example

Here is a simple room plan as it was exported from a 3D and imported in 2D drawing. Due to incompatibilities of the software programs involved in the export / import operations, there are some lines that are angled, broken or overlapped.



Using a workflow that includes the commands OPTIMIZE, OVERKILL and SIMPLIFY, the plan can be improved.

4.20.3 Optimizing the lines

The command OPTIMIZE is used to straighten the lines.

Depending on the complexity of the input drawing and the quality of the import operation, there might be required to use this command many times.

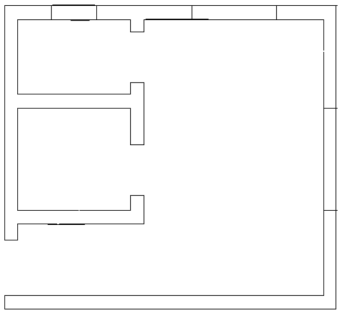
In this case, there are many angled lines, overlapped lines and some gaps between them.

For the first run of the OPTIMIZE command, the parameters are:

- Selection: all the lines that are shown in the image.
- Close gaps: not checked.
- Reference angles: horizontal and vertical.
- Angle tolerance: 6 degrees.
- Distance tolerance: 50 mm.

See the command OPTIMIZE for more information.

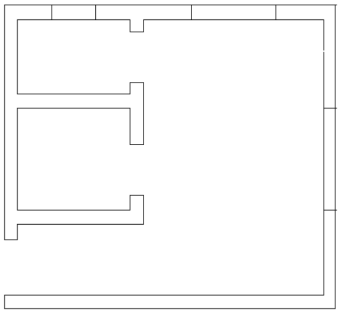
As shown in the image below, the result is better than the original drawing.



To further improve the quality of the geometry, the command could be run again.

For the second run of the OPTIMIZE command, the parameters are:

- Selection: all the lines that are shown in the image.
- Close gaps: checked.
- Reference angles: horizontal and vertical.
- Angle tolerance: 1 degree.
- Distance tolerance: 50 mm.



The drawing has improved considerably and also the gaps have been closed.

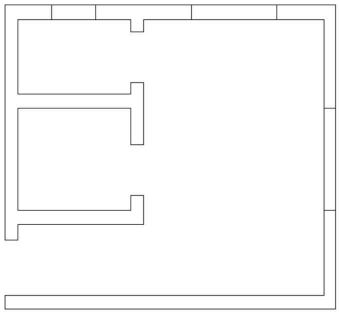
The next step is to further clean up the drawing.

4.20.4 Deleting duplicates and joining contiguous entities

The OVERKILL command allows to:

- Delete duplicated and overlapped entities.
- Straighten the angled entities
- Join partly overlapping or contiguous entities.

Running the OVERKILL command with a tolerance of 1 mm will achieve all the tasks above.

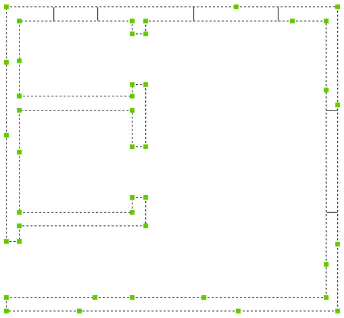


At this stage, the drawing has been cleaned up. There are no more angled or duplicated or overlapped lines. Also, the gaps have been close.

4.20.5 Simplifying the drawing

After joining the lines representing the walls into a single polyline, this one will have too many vertices. See the image below.

To remove the unnecessary vertices, the SIMPLIFY command is used.



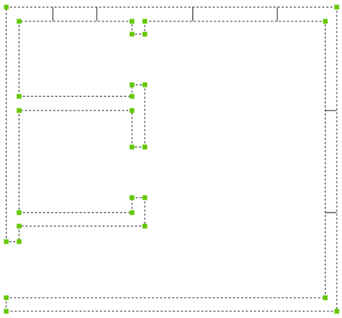
The steps to applying this command are:

- Start the command.
- Choose the **Simplify** method.
- Choose the **Max angle and distance** option.

The parameters for the command are:

- Maximum distance between non-collinear vertices to straighten: **1**
- Maximum change (angle) in a direction between each two consecutive segments to straighten: **1**
- Straighten polyline arc segments? **No**

As shown, all the unnecessary vertices have been removed.

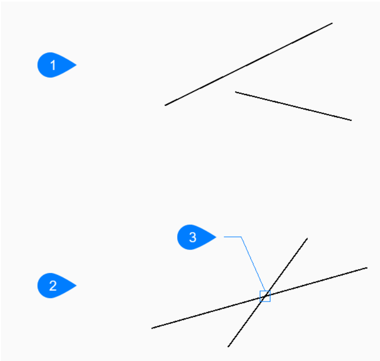


Note: The parameters specified in this article are strictly related to this example. There are some factors that highly influence the result of this workflow. Mainly, the complexity of the drawing and the number and amplitude of the irregularities. To obtain a satisfactory result, there might be needed further experimentation with various values for the parameters of each step.

4.21 Entity snaps overview

4.21.1 Apparent intersection entity snap

The Apparent intersection entity snap enables the cursor to snap to the intersections of entities, even when they only appear to intersect in 3D space.



1 In one 3D viewpoint, the lines do not intersect.

2 In another 3D viewpoint, the lines appear intersect.

3 Apparent intersection.

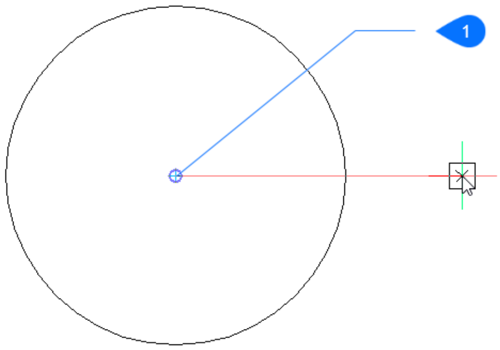
4.21.2 Center entity snap

The Center entity snap enables the cursor to snap to the center point of circular or elliptical entities.

Including:

- Circles
- Arcs
- Polyline arcs
- Ellipses
- Elliptical arcs

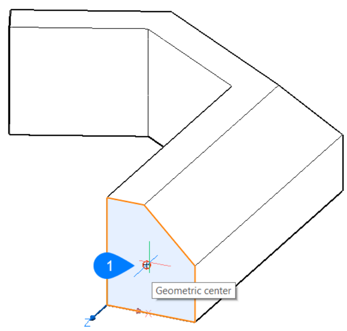
- Circular 3D faces
- Elliptical 3D faces



1 Center of circle.

4.21.3 Center of a Face 3D entity snap

The Center of a Face 3D entity snap enables the cursor to snap to the geometric center of a 3D face.



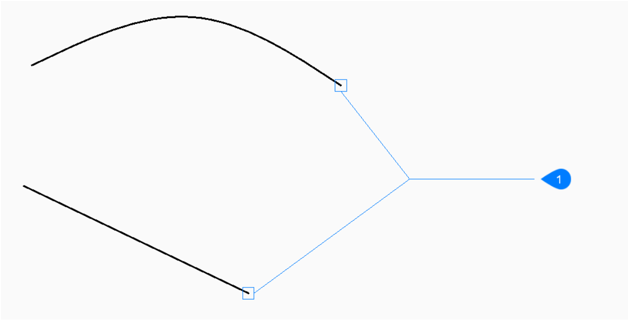
1 Center of a Face 3D entity snap.

4.21.4 Endpoint entity snap

The Endpoint entity snap enables the cursor to snap to the endpoints of entities.

Including:

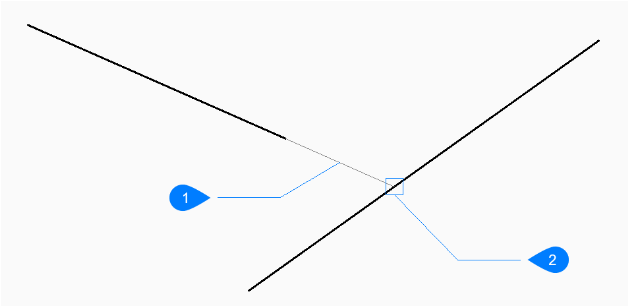
- Lines
- Arcs
- Polyline segments
- Open splines
- 3D edges



1 Endpoint entity snap.

4.21.5 Extension entity snap

The Extension entity snap enables the cursor to snap to the intersection of one or two extended entities.



1 Extension

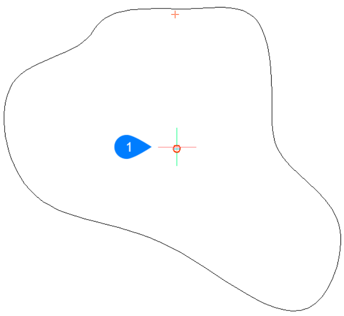
2 Extension entity snap.

4.21.6 Geometric center entity snap

The Geometric center entity snap enables the cursor to snap to the centroid of closed entities.

Including:

- Polylines
- Splines
- Regions
- Planar 3D faces



1 Geometric center entity snap.

4.21.7 Insertion entity snap

The Insertion entity snap enables the cursor to snap to the insertion points of entities.

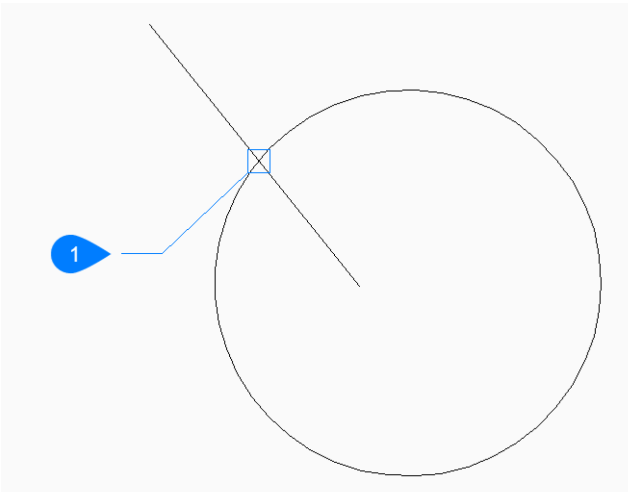
Including:

- Text
- Mtext
- Tables
- Attribute text
- Blocks
- External References
- PDF References
- Raster images

1 Insertion entity snap.

4.21.8 Intersection entity snap

The Intersection entity snap enables the cursor to snap to the intersection of entities that physically intersect each other.



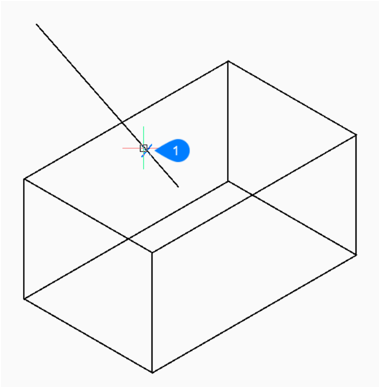
1 Intersection entity snap.

4.21.9 Intersection 3D entity snap

The Intersection 3D entity snap enables the cursor to snap to the intersection of faces with linear entities.

Including:

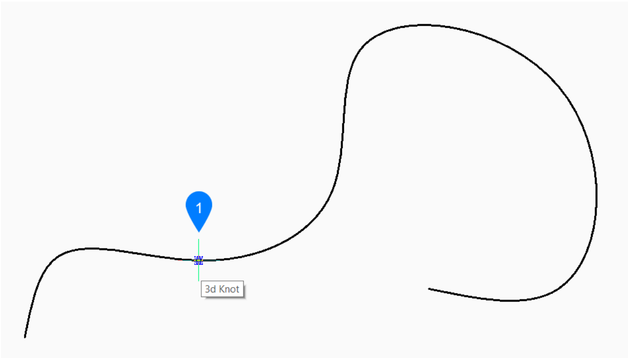
- Lines
- Polyline segments
- 3D edges
- Polar tracking lines
- Entity snap tracking lines



1 3D intersection entity snap.

4.21.10 Knot 3D entity snap

The Knot 3D entity snap enables the cursor to snap to knots on splines.



1 Knot 3D entity snap.

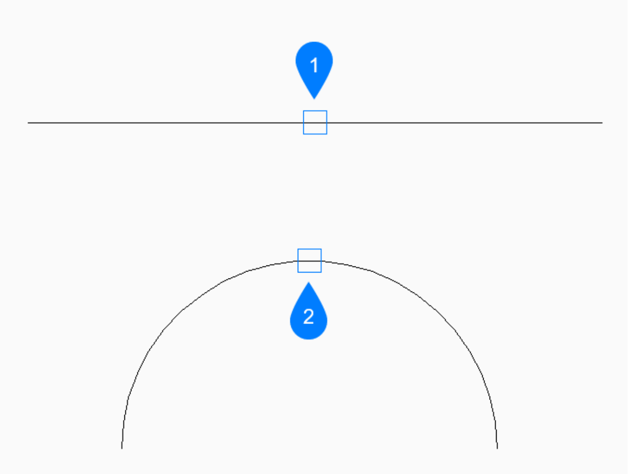
4.21.11 Midpoint entity snap

The Midpoint entity snap enables the cursor to snap to the midpoint of entities.

Including:

- Lines
- Polyline segments
- Arcs
- Elliptical Arcs
- Splines

- 3D edges

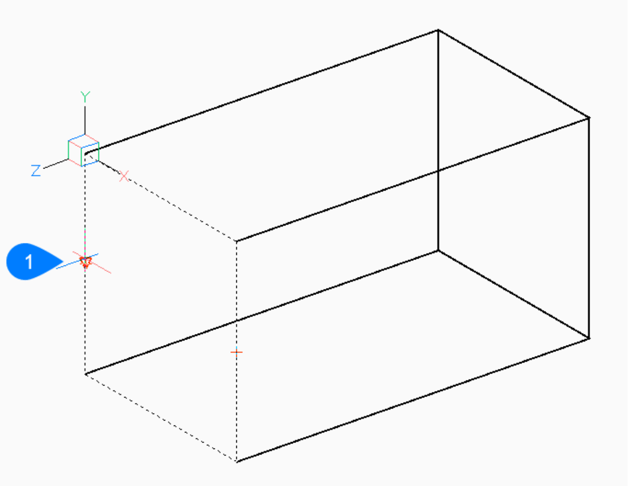


1 Midpoint (of line).

2 Midpoint (of arc).

4.21.12 Midpoint 3D entity snap

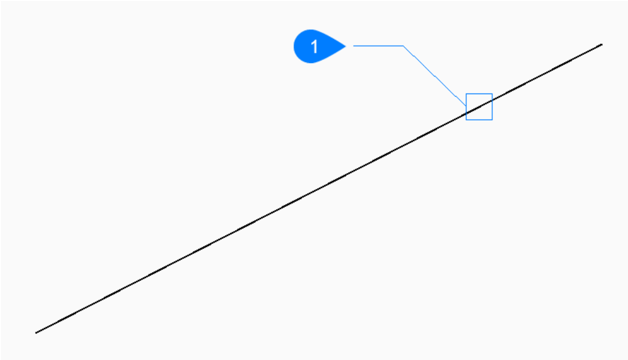
The Midpoint 3D entity snap enables the cursor to snap to the midpoint of 3D edges.



1 Midpoint 3D entity snap.

4.21.13 Nearest entity snap

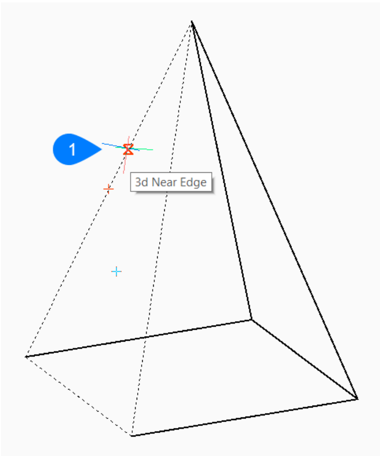
The Nearest entity snap enables the cursor to snap to 2D or 3D geometry nearest to the cursor.



1 Nearest entity snap.

4.21.14 Nearest 3D entity snap

The Nearest 3D entity snap enables the cursor to snap to a point on a 3D entity that is nearest to the cursor.



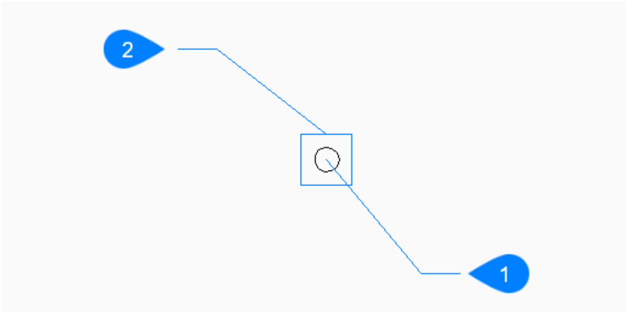
1 Nearest 3D entity snap.

4.21.15 Node entity snap

The Node entity snap enables the cursor to snap to point entities.

Including:

- Mtext

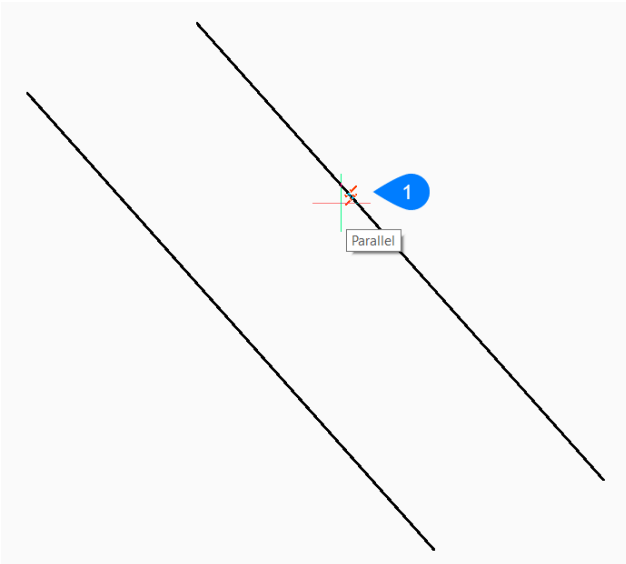


1 Point.

2 Node entity snap.

4.21.16 Parallel entity snap

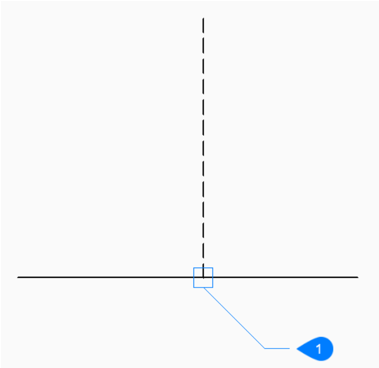
The Parallel entity snap enables the cursor to snap to a point parallel to an existing entity.



1 Parallel entity snap.

4.21.17 Perpendicular entity snap

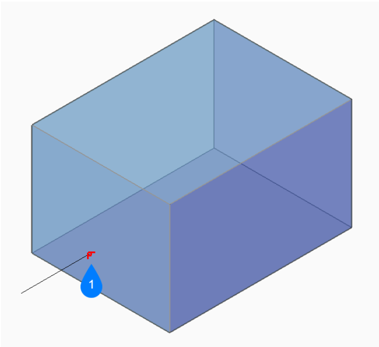
The Perpendicular entity snap enables the cursor to snap to a point perpendicular to an existing entity.



1 Perpendicular (to entity).

4.21.18 Perpendicular on a face 3D entity snap

The Perpendicular on a face 3D entity snap enables the cursor to snap to a point perpendicular to a 3D face.

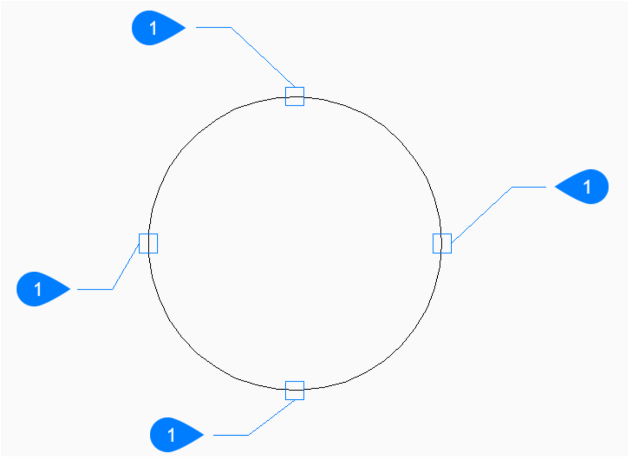


1 Perpendicular to a face 3D entity snap.

4.21.19 Quadrant entity snap

The Quadrant entity snap enables the cursor to snap to the quadrant points of circular or elliptical entities. Including:

- Circles
- Arcs
- Polyline arcs
- Ellipses
- Elliptical arcs
- Circular 3D faces
- Elliptical 3D faces



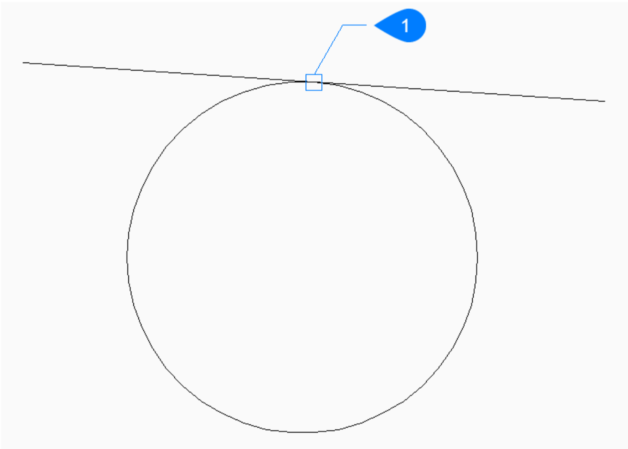
1 Quadrant.

4.21.20 Tangent entity snap

The Tangent entity snap enables the cursor to snap to the tangent points of curved entities.

Including:

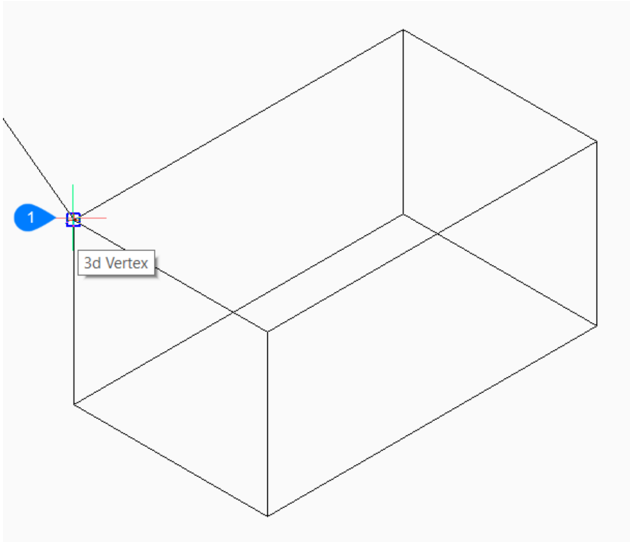
- Circles
- Arcs
- Polyline arcs
- Ellipses
- Elliptical arcs
- Splines
- Circular 3D faces
- Elliptical 3D faces



1 Point of tangency.

4.21.21 Vertex 3D entity snap

The Vertex 3D entity snap enables the cursor to snap to the closest vertex of a 3D entity.



1 Vertex 3D entity snap.

5. Viewing your drawing

5.1 View commands overview

BricsCAD provides various tools and commands to control the display and view orientation of your drawing.

These tools can be found in the **View** menu, toolbar or ribbon tab.

Icon	Tool Name	Command	Description
	Redraw	REDRAW	Refreshes the screen display in the current viewport.
	Redraw All	REDRAWALL	Refreshes the screen display in all open viewports.
	Regenerate	REGEN	Recalculates the screen display in the current viewport.
	Regenerate All	REGENALL	Recalculates the screen display in all open viewports.
	Update Fields	UPDATEFIELD	Updates the selected fields.
	Real Time Pan	RTPAN	Pans the view dynamically.
	Pan	PAN	Pans the view.
	Real Time Zoom	RTZOOM	Zooms in/out dynamically.
	Zoom In	ZOOM + *I(*)*	Zooms in on the center of the window by a factor of 2.
	Zoom Out	ZOOM + *O(*)*	Zooms out from the center of the window by a factor of 1/2.
	Zoom Extents	ZOOM + *E(*)*	Displays all the entities in the drawing (referred to as the drawing extents).
	Zoom Window	ZOOM + *W(*)*	You are prompted to pick two corners of a box on the existing view in order to enlarge that area to fill the display.
	Zoom Previous	ZOOM + *P(*)*	Restores the displayed view prior to the current one.

Icon	Tool Name	Command	Description
	Zoom All	ZOOM + *A(*)*	Displays the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).
	Zoom Left	ZOOM + *L(*)*	Pick the lower left corner of the next view, then specify the magnification factor or the view height.
	Zoom Center	ZOOM + *C(*)*	Pick the center of the next view, then specify the magnification factor or the view height.
	Zoom Right	ZOOM + *R(*)*	Pick the upper right corner of the next view, then specify the magnification factor or the view height.
	Look Around	RTLOOK	Look around in a 3D scene.
	Look Up and Down	RTUPDOWN	Move up / down and left / right in a 3D scene.
	Walk	RTWALK	Walk left / right and forward / backward in a 3D scene.
	Real-Time Constrained Sphere	RTROT	Rotates the view dynamically in 3D. Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the world Z-axis. Moving the mouse vertically (parallel to the screen y-axis) rotates the screen x-axis.
	Real-Time Sphere Center	RTROTCTR	Rotates the view dynamically in 3D about a user-defined center point. Behaves like RTROT.
	Real-Time Free Sphere	RTROTF	Rotates the view dynamically in 3D. Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the screen Y-axis. Moving the mouse vertically (parallel to the screen y-axis) rotates the view about the screen X-axis.
	Real-Time X	RTROTX	Rotates the view dynamically about the screen X-axis.
	Real-Time Y	RTROTY	Rotates the view dynamically about the screen Y-axis.
	Real-Time Z	RTROTZ	Rotates the view dynamically about the screen Z-axis.

Icon	Tool Name	Command	Description
	Set View Point	DDVPOINT	Opens the **Preset Viewpoints** dialog.
	Plan View	PLAN	Restores the plan view with respect to the WCS, the current UCS or a saved UCS.
	Define View	DVIEW	Defines parallel and visual perspective views.
	Save/Restore View	VIEW	Saves and restores named views.
	Viewports	VPORTS	Manages viewports in model space.
	Paper Space Views	MVIEW	Creates viewports in paper space.

(*) <command> <option>: type **zoom** followed by Enter, then type the command option letter, followed by Enter

5.2 Layer utilities

The layer utility commands help you to easily manipulate the **On/Off**, **Thaw/Freeze** and **Unlocked/Locked** layer properties by selecting entities. You can then save the result in a Layer State.

The layer utility commands sit on the **Layers** toolbar and on the **Home / Layers** ribbon panel in the **Drafting** and **Modeling** workspaces.

Command	Icon	Description
LAYON		Turns on all layers that have been turned off.
LAYOFF		Turns off layers associated with selected entities.
LAYFRZ		Freezes layers associated with selected entities.
LAYTHW		Thaws all frozen layers.
LAYLCK		Locks layers associated with selected entities.
LAYULK		Unlocks layers associated with selected entities.

Command	Icon	Description
LAYISO		Isolates layers associated with selected entities. If no entity on the current layer is selected, the layer of the entity that is selected first temporarily becomes the current layer until the LAYUNISO command is executed.
LAYUNISO		Restores layers isolated by the LAYISO command.

5.3 Hide and isolate entities

Command	Icon	Description
HIDEOBJECTS		Hides the selected entities.
ISOLATEOBJECTS		Displays the selected entities only.
UNISOLATEOBJECTS		Brings back all previously hidden entities.

The tools to launch these commands can be found on:

- the **Hide/Show Entities** and **Access 3D** toolbars.
- the **Select** Quad tab.
- The **Home / Layers** ribbon panel in the **Drafting** and **Modeling** workspaces.

Show, Hide and Isolate entities is also available in the right-click context menu in the **Structure** panel.

Note: The OBJECTISOLATIONMODE system variable controls whether hiding and isolating entities is saved or not.

5.4 Redrawing and regenerating a drawing

The screen display of a drawing is really a simplified version of the drawing database. From time to time, it is necessary to synchronize the screen display and the drawing database.

The following commands are available:

- The REGEN command (short for "regenerate") regenerates the current viewport.
- The REGENALL command (short for "regenerate all") regenerates all viewports.
- The REGENAUTO command controls when BricsCAD automatically regenerates the drawing.
- The REDRAW command redraws the current viewport to clean it up.
- The REDRAWALL command redraws all viewports to clean them up.
- The REGENMODE system variable controls whether BricsCAD regenerates the drawing automatically. You can change the status of REGENMODE through the REGENAUTO command. When REGENMODE = ON, BricsCAD regenerates the display automatically, but in a few cases a forced regeneration of the drawing might still be necessary.

Take care not to confuse the REGEN command with the REDRAW command, which simply repaints the screen, without attempting to synchronize the screen display with the drawing database.

5.5 Panning

The PAN and RTPAN commands move the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space. The only change is the portion of the drawing being displayed.

5.5.1 Using the PAN command

1 Launch the PAN or RTPAN command.

1 The shape of the cursor changes to a hand.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.
- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

5.6 Zooming

The **ZOOM** command visually changes the size of the drawing within the current viewport.

The RT**ZOOM** command zooms the drawing in real-time.

You can change the magnification of your drawing at any time by zooming. The cursor changes to a magnifying glass when aN RTZOOM tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail. Changing the magnification of the drawing affects only the way the drawing is displayed; it has no effect on the dimensions of the entities in your drawing.

5.6.1 Real-time zooming

1 Launch the RTZOOM command.

1 The shape of the cursor changes to a magnifying glass.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.

- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

5.6.2 Zooming to extents

Do one of the following:

- Double-click the middle mouse button (scroll wheel).
- Launch the ZOOM command, then type E and press Enter.

5.6.3 Dynamic zooming

Pans and zooms using a rectangular view box, representing the current viewport. The command first zooms out to display the entire drawing. You can then control the size of the view box dynamically and position the view box on the part of the drawing you want to display in the current viewport.

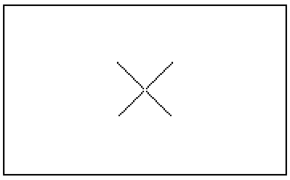
1 Launch the ZOOM command.

2 Choose the **Dynamic** option.

The drawing zooms out to extents.

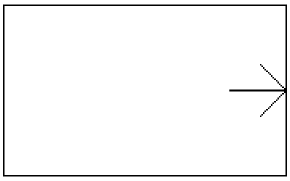
A dashed rectangle indicates the area of the previous view.

A view box of the same size as the dashed rectangle is attached to the cursor.



3 (option) To resize the view box:

- Click, then move the cursor to resize.



- Click again to stop resizing.

4 Position the view box over the drawing area you want to enlarge, then right-click.

5 The drawing is zoomed in on the selected area.

5.7 View manipulation using the mouse and keyboard

In BricsCAD you can navigate through a 3D model using the mouse and keyboard shortcuts.

In perspective mode you can walk inside the 3D model. In isometric mode you can only zoom in very close, but not go inside.

Mouse or Keyboard	Action	Result
Mouse wheel	scroll	Zoom in / out. The incremental change in zoom with each mouse-wheel action is controlled through the **Zoom Factor** variable (ZOOMFACTOR system variable). Default = 60. Values between 3 and 100 are accepted. When zooming in, the incremental step decreases gradually, allowing you to focus on a particular detail easily. In a 3D model, the entity under the cursor has the zoom focus. The ZOOMWHEEL system variable controls the zoom direction: 0 or OFF: scrolling forwards zooms in, backwards zooms out. 1 or ON: scrolling forwards zooms out, backwards zooms in.
Middle mouse button or wheel*	press and hold while moving the mouse	Real-time pan (RTPAN command)
Middle mouse button or wheel*	double-click	Zoom extents
Ctrl + Shift + Left mouse button	press and hold while moving the mouse	Real-time zoom in / out (RTZOOM command)
Ctrl + Shift + Right mouse button	press and hold while moving the mouse	Constrained real-time pan (RTPAN command). Release the Shift key for free panning.
Ctrl + Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time sphere (RTROTF command)
Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time constrained sphere (RTROT command)
Ctrl + Right Button	press and hold while moving the mouse	Real-time rotate about the screen Z-axis (RTROTZ command)

Mouse or Keyboard	Action	Result
Walk through navigation		
Alt + Left Button	press and hold while moving the mouse	Walk left / right and forward / backward (RTWALK command)
Alt + Middle Button	press and hold while moving the mouse	Move up / down and left / right (RTUPDOWN command)
Ctrl + Middle Button	press and hold while moving the mouse	Look around (RTLOOK command)
Ctrl + Arrow Keys	press the arrow keys while holding the Ctrl key	Walk left / right and forward / backward (RTWALK command)
Ctrl + Shift + Arrow Keys	press the arrow keys while holding the Alt + Ctrl keys	Look up / down and left / right (RTUPDOWN command)
Home		Sets the view direction horizontal (VIEWHORIZONTAL command)
Alt + Home		Moves target point to center of scene
Alt + plus/minus		Increase / decrease movement speed
Ctrl + plus/minus		Increase / decrease rotation speed

* On the condition that the MBUTTONPAN system variable is set to **On**.

5.8 View rotation

The following command are available for view rotation operations:

- The **RTROT** (Real-Time Constrained Sphere) command rotates 3D drawings in real-time.
- The **RTROTCTR** (Real-time Sphere Center) command freely rotates 3D drawings in real-time about a user-defined center point.
- The **RTROTF** (Real-time Free Sphere) command freely rotates 3D drawings in real-time.

- The **RTROTX, RTROTY** and **RTROTZ** commands rotates 3D drawings in real-time about the x-, y- or z-screen axis.
- The **DDVPOINT** command sets 3D viewpoints or plan view, through a dialog box (short for "dynamic dialog view point").
- The **PLAN** command restores the plan (top) view.
- The **VPOINT** command changes the 3D viewpoint.

Note:

- If the **CONTINUOUSMOTION system** variable is set, the view rotation continues until you press the Esc key to conclude the command.
- View rotation commands should not be used when working in 2D. Use the PLAN command or the LookFrom control to restore top view.

5.8.1 Rotating a view freely

1 Launch the RTROT, RTROTF, RTROTX, RTROTY or RTROTZ command.

1 The shape of the cursor changes according to the selected command.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.
- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

5.8.2 Using Preset Viewpoints

Note: Switch off Perspective mode before starting the command.

1 Launch the DDVPOINT command.

1 The **Set Viewpoint** dialog box appears.

2 Click the **Angle Precision** button to set the display mode for the **Set Viewpoint** dialog box.

The options are: **45°**, **15°** or **5°**.

The **Horizontal** and **Vertical** select panes are divided accordingly.

3 Click an angle in the **Horizontal** and **Vertical** select panes.

The view orientation is updated.

4 (option) Click the **Plan** button to restore the plan (top) view.

5 (option) Repeatedly click the **Previous** button to restore previous viewpoints.

6 To conclude the command:
  - Click the **OK** button to keep the current view.
  - Click the **Cancel** button to restore the original view.

5.9 The LookFrom control

The **LookFrom** control allows you to select a number of preset views such as orthographic views and isometric views. By default, the **LookFrom** control is displayed in the top right corner of the graphic screen.

The NAVVCUBEDISPLAY system variable sets the display of the LookFrom control.

5.9.1 Context menu options

When you right-click the LookFrom control, a context menu appears. The options are:

- **UCS**: sets the current coordinate system.
  - **World**: restores the WCS (World Coordinate System)
  - **New**: runs the UCS command.
  - **UCS Manager...**: runs the EXPUCS command; displays the **Drawing Explorer / Coordinate Systems** dialog box.
- **View**:
  - **New**: runs the -VIEW command.
  - **View Manager...**: runs the VIEW command; displays the **Drawing Explorer / Views** dialog box.
- **Visual Style**: Allows you to set the visual style.
- **Isometric Mode**: Sets the isometric mode.
- **Twist Mode**: Sets the Twist mode.
- **WCS**: LookFrom uses the WCS as reference coordinate system.
- **UCS**: LookFrom uses the current coordinate system (WCS or UCS) as reference coordinate system.
- **Perspective**: Sets the perspective view mode.
- **Location**: Sets the location of the LookFrom control.
- **Dismiss**: Closes the LookFrom control.

  Edit the NAVVCUBEDISPLAY system variable to restore the LookFrom control.

5.9.2 LookFrom control settings

The behavior of the **LookFrom** control depends on a number of system variables and user preferences.

To access the LookFrom settings:

1 Right-click the LookFrom control and choose **LookFrom Settings...** in the context menu.

2 Run the LOOKFROM command and choose the **Settings** option.

5.9.3 Setting the LookFrom mode

The LookFrom control runs in one of two modes:

- Isometric mode for working in 3D models.
- Twist mode for 2D drafting.

To set the LookFrom mode:

1 Right-click the LookFrom control.

2 Select **Isometric Mode** or **Twist Mode** in the context menu.

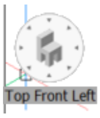
5.9.4 Using the LookFrom control in Isometric mode

1 Move the cursor to the **LookFrom** control.

The **LookFrom** control becomes active.

2 Hover the cursor over a view orientation.

3 The chair in the center of the LookFrom control rotates according to the view orientation.



4 (option) Hold down the Ctrl key to look bottom-up.

5 Click to confirm.

6 The view is rotated.

5.9.5 Using the LookFrom control in Twist mode

In **Twist** mode, the **LookFrom** control rotates the view around the screen Z-axis.

1 Move the cursor to the **LookFrom** control.

The **LookFrom** control becomes active.

2 Move the cursor to the edge of the **LookFrom** control, on the left half for counterclockwise, on the right half for clockwise rotation. 15°, 30°,45°, 60°, 90°, 120°, 135° and 180° angles are available.

3 Click to rotate the view.

4 (option) Without moving the mouse, click again to rotate over the same angle again.

5 (option) Move the cursor to the center of the **LookFrom** control, then click to restore an unrotated view.

5.9.6 View transition options

The following system variables control the behavior of view transitions:

- VTENABLE: determines whether animated view transitions are enabled for zoom/pan and/or for view rotation operations.
- VTDURATION: determines the duration of view transition animations in milliseconds.

- VTFPS: defines the minimum frame rate required to allow animation. That is 7 by default, which means that the redraw time should take less than 143 (=1000/7) milliseconds. If the computer is not capable to redraw the view fast enough, no animation will be available.

5.10 Layer states

Layer states let you save and restore configurations of layer properties. You might want a layer to be displayed in blue sometimes and green at other times, or you need some layers to be off or frozen or locked when editing a specific part of a drawing. After spending a lot of time adjusting all layer settings you can use layer states to save your settings. You can save multiple layer states in a drawing. Layer states can be copied between drawings and you can save a layer state to an external file, which can then be imported in another drawing.

To easily create layer states, the Layer Utility commands might be useful.

The LAYERSTATE command opens the **Drawing Explorer / Layer States** dialog box.

5.10.1 Saving the current layer state

1 Click the **New** button on the **Drawing Explorer / Layer States** dialog box.

A new layer state is created.

2 Type a name for the layer state, replacing the **NewLayerState1** default name.

3 (Optional) When in **Detail View**, type a description in the **Description** field.

4 Select the **Layer Properties to restore**.

Click the tick box to select a layer property, click again to remove a property.

5 Select the **Restore option**.

6 When in paper space, tick the **Apply properties as viewport overrides** option.

Note: In a new layer state, all layer properties are selected to be restored by default.

5.10.2 Editing layer states

1 (Optional) Click the **Overwrite** button to overwrite the layer state with the current layer settings.

2 (Optional) Rename the layer state.

3 (Optional) Edit the layer state description.

4 (Optional) Edit the **Layer Properties to restore**.

Click the tick box to select a layer property, click again to remove a property.

5.10.3 Restoring layer states

1 Do one of the following:

- Click the **Restore** button on the **Drawing Explorer / Layer States** dialog box.
- Right-click, then choose **Restore** in the context menu.

2 Click the **Regen** button, unless the **Regen at each change** button is pressed.

Note:

- The restore option of the layer state applies to layers which are added after the selected layer state was created. The options are:
  - Turn Off.
  - Freeze
  - Do nothing.
- Layer states can be restored using the **Layer State** control, which you can add to a ribbon panel or a toolbar.

5.10.4 Copying layer states between open drawings

1 Open both, the source and the target drawing.

2 Select the source drawing in the **Open Drawings** list on the **Drawing Explorer / Layer States** dialog box.

3 Select the layer state(s) you want to copy.

4 Press and hold the Ctrl key to select multiple layer states.

5 Hold down the left mouse button, then drag-and-drop the selection onto the target drawing in the **Open Drawings** list on the **Drawing Explorer / Layer States** dialog box.

The selected layer states are copied to the target drawing.

5.10.5 Exporting layer states

1 Select the layer state on the **Drawing Explorer / Layer States** dialog box.

2 Click the **Export** button.

The **Export Layer States** dialog box appears.

3 In the **File Name** field, type a name for the layer state file.

4 Select a destination folder.

5 Click the **Save** button.

5.10.6 Importing layer states

1 Click the **Import** button on the **Drawing Explorer / Layer States** dialog box.

The **Import Layer States** dialog box appears.

2 Select the layer state file to import.

3 Click the **Open** button.

The layer state is imported.

5.11 Named views

You can save the view in the current window as a named view. Saved views can be restored at any time.

A Background can be assigned to a named view, which will be used when the view is rendered.

5.11.1 Exploring views

1 Launch the VIEW command.

1 The **Drawing Explorer / Views** dialog box appears.

2 (option) Click the blank tile in front of the **View Name** to restore the view.

The current view is marked.

3 (option) Click the **New** button in the **Drawing Explorer** toolbar to save the current view (launches the -VIEW command).

4 (option) Click the **Delete** button in the **Drawing Explorer** toolbar to delete the selected view.

5 (option) Click the **View Name** , then right-click and choose **Rename** in the context menu.

Type a new name for the saved view.

6 (option) Check the **Perspective** property to make the view a visual perspective view.

If not checked the view is a parallel perspective view.

7 (option) Click the **Background** column of a view to choose a background. The background definition of the view is applied during rendering.

5.11.2 Saving a view

1 Do one of the following:

- Launch the -VIEW command.
- Click the **New** button in the **Drawing Explorer / Views** toolbar.

2 Choose the **Save** option.

-VIEW command workflow:

- **You are prompted**: Save view as:
- Type a name for the view in the Command line, then press Enter.

Drawing Explorer workflow:

- The **Drawing Explorer / Views** dialog box appears.
- Type a name to replace the NewView1 default name.

Note: If you type a name in the Command line that is already used, the current view is saved, replacing the previously saved view. There is no warning when overwriting a saved view.

5.11.3 Restoring a view

In the **Drawing Explorer / Views** dialog box, click the **Current** field of the view.

5.11.4 Editing the view background

1 Select the view in the **Drawing Explorer - Views** dialog box.

2 Select the **Background** property.

3 Click the **Browse** button at the right hand side of the settings field.

The **Background** dialog box appears.

4 Click the **Type** list button, then select the background type.

The options are **None**, **Solid**, **Gradient** and **Image**.

5 Select a background type.

6 The settings for the selected type are displayed in the **Background** dialog box.

7 Specify the background settings.

8 Click the **OK** button on the **Background** dialog box.

9 (option) Click the **Current** field.

Note: The GRADIENTMODE system variable sets the default for gradient backgrounds: 2 or 3 colors.

5.12 Visual styles

A visual style is a collection of settings that control the display of 3D solids and surfaces.

The VISUALSTYLES command opens **Drawing Explorer** > **Visual Styles**.

You can:

- Edit existing visual styles.
- Create new visual styles.
- Delete visual styles.
- Copy and paste visual styles between open drawings.

5.12.1 Setting the current visual style

The current visual style can be set in the **Properties** panel: **View** > **Visual style** and in the right-click context menu of the **LookFrom** widget.

Click the desired visual style in the list.

The default options are:

- **2D Wireframe**: entities are displayed by using lines and curves. It is used for 2D drawings.
- **Conceptual**: entities are displayed by using smooth shading and the Gooch face style.
- **Hidden**: entities are displayed by using wireframe representation and lines representing back faces are hidden.
- **Modeling**: entities are displayed by using smooth shading, Realistic face style, Materials and textures and visible edge.
- **Realistic**: entities are displayed by using smooth shading, Realistic face style and Materials and textures. Edges are not shown.
- **Shaded**: entities are displayed by using smooth shading, Realistic face style and Materials.
- **Shaded with edges**: entities are displayed by using smooth shading, Realistic face style, Materials, and visible edges.
- **Shades of gray**: entities are displayed by using smooth shading, Realistic face style and monochromatic shades of gray.
- **Sketchy**: 2D and 3D entities are displayed with a hand-sketched effect by using the Constant color face style, Line Extensions and textured lines (bpen).

- **Wireframe**: 3D entities are displayed by using lines and curves only.
- **X-Ray**: entities are displayed by using smooth shading, Realistic face style and Materials and textures with partial transparency.
- **BIM (BIM templates only)** : entities are displayed by using smooth shading, Realistic face style, Materials and textures, and Facet Edges (visible edges).
- **Maquette (BIM templates only)** : entities are displayed by using smooth shading, Realistic face style, Materials, and Facet Edges (visible edges).
- **Render (BIM templates only)** : entities are displayed in the background color, by using smooth shading, Realistic face style and Materials and textures. Edges are not shown.
- **See-through (BIM templates only)** : entities are displayed by using smooth shading, Constant color face style with partial transparency.
- **Mechanical (Mechanical templates only)**: entities are displayed by using smooth shading, Realistic face style, Materials and textures and Facet Edges.

In shaded visual styles, faces are illuminated by one distant light source which follows the viewpoint as you move around the model. This default lighting illuminates all faces in the model so the user can identify them. Only when all other lights are turned off, the default lighting is available.

5.13 Model space and paper space

When you start a drawing session, your initial working area is called **Model Space**. Model Space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the **Model** tab.

In general, model space consists of a single view that fills the screen. If needed, you can create additional views, called viewports, which can show different views of your drawing or 3D model. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. Click in a viewport to make it the current viewport. You can print the current viewport only. The VPORTS command lets you manage your viewports in model space.

BricsCAD provides an additional work area, called **Paper Space**. Paper space represents a paper layout of your drawing. In this work area, you can create and arrange different views of your drawing similar to the way you arrange drawings on a sheet of paper. In paper space you can also add keynotes, annotations, borders, title blocks, and other print-related entities, which you don't want to see in model space.

Each drawing has at least one **Layout** in which you can have one or more **Layout Viewports**. Such layout viewports are to be considered as a view window on the entities in model space. Layout viewports can be placed anywhere in a paper space layout. Each viewport has its own scale and layer visibility. All viewports in the same layout can be printed simultaneously.

The MVIEW command lets you manage your viewports in paper space.

Although not necessary to print your drawing, paper space offers a lot of advantages:

- Create multiple layouts to print the same drawing with different print settings, such as pen widths, printer configuration files, lineweight settings, drawing scale, and more.

- Add print-related entities that are not essential to the model itself, such as keynotes, annotations, title blocks, etc.
- For a single layout, create multiple layout viewports to print multiple views of your drawing at different scales.

5.13.1 Switching between model space and paper space

- To switch between Model Space and Paper Space layouts, use the **Model** and **Layout** tabs at the bottom of the graphic window.
- When working in a layout, you can access model space through a paperspace viewport:
  - Double-click inside a viewport to access model space. Double-click outside a viewport to go back to paper space.
  - Click the **Paper/Model** field in the **Status** bar to toggle between the workspaces. This method has to be used when it is not possible to click outside the viewport.
  - Use the **MSPACE** and **PSPACE** commands. These commands are available in the right-click context menu of the **Paper/Model** field in the **Status** bar.

5.13.2 Using the layout tabs

- The name of the active tab is displayed in bold.
- Click the navigation buttons at the left hand side to switch.
- Drag layout tabs to change their order.
- Click the button after the last **Layout** tab to open the **Manage Layouts Menu**.
- Right-click a layout tab to display a context menu.
  - **New Layout**: Create a new layout.
  - **From Template**: Imports a layout from another drawing.
  - **Delete**: Deletes the layout.
  - **Rename**: Renames the layout.
  - **Copy**: Inserts a copy of the layout.
  - **Move Right / Move Left**: Moves the layout.
  - **Export**: Exports the layout to a new drawing.

5.14 Model space viewports

The display in model space (Model Space, with tiled viewports) can be divided into multiple viewports, each of which can contain a different view of the current drawing. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. When no command is active, click in a viewport to make it the current viewport. You can print the current viewport only.

The coordinate system (WCS or UCS), grid display and snap can be set per viewport.

The VPORTS command creates and manages model space viewports.

5.14.1 Working with multiple viewports

- Click in a viewport to make it current.
- The border of the current viewport is highlighted. You can resize the viewports by dragging their borders.
- The cursor is shown in the current viewport only.
- You can start a command in one viewport, and finish the operation in a different viewport.

5.14.2 Joining adjacent viewports

You can join viewports which share an edge of equal length.

1 Launch the **VPORTS**command.

2 Select the **Join** command option.

You are prompted: Select inside dominant viewport <Current>:

3 Click in the dominant viewport or press Enter to choose the current viewport.

4 The border of the selected viewport is highlighted.

You are prompted: Select inside viewport to join.

5 Click the viewport to join again to confirm.

6 The two selected viewports are joined.

5.15 Paper space viewports

The **ALIGNSPACE** command adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space.

In a layout, you can create multiple viewports, each of which displays a unique view of the entities created in model space. Each layout viewport functions as a window into your model space drawing. You can control the view, scale, and content of each layout viewport separately.

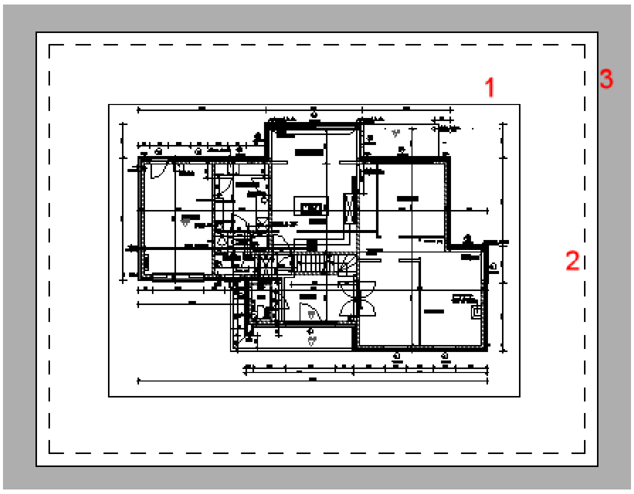
A layout viewport is created as a separate entity that you can copy, delete, move, scale, and stretch as you would any other drawing entity. You can snap to the viewport borders using entity snap. When you are working in model space in layout, click any layout viewport to make it the current viewport, and then add or modify model space entities in that viewport. Any changes you make in one layout viewport are immediately visible in the other viewports (if the other layout viewports are displaying that portion of the drawing). Zooming or panning in the current viewport affects only that viewport.

Each viewport has its own layer visibility settings. You can also turn off the display of the content of a viewport.

To preserve the scaling of a viewport, you can lock the display. It is no longer possible to zoom or pan in a locked viewport.

Apart from viewports, you can add print-related entities in a paper space layout that are not essential to the model itself, such as keynotes, annotations, title blocks, etc. Such entities are part of a specific paper space layout and do not appear in other layouts or in model space.

5.15.1 Displaying the paper sheet and the printable area



1 Viewport.

2 Printable area.

3 The display of the printable area is controlled through the DISPPAPERMARGINS system variable.

4 Paper sheet.

5 The display of the paper sheet is controlled through the DISPPAPERBKG system variable.

5.16 Creating paper space viewports

The MVIEW and VPORT commands create one or more viewports in layout tabs.

- A new viewport shows all entities on the layers which are currently visible in model space.
- It is recommended to use a dedicated layer for viewport entities. If you don't want the viewport outlines to be printed, set the **Do Not Plot** property of the viewport layer.

5.16.1 Creating an array of layout viewports

The MVSETUP command (short for "multiple viewports setup") creates multiple paper space viewports, and then aligns, rotates and scales them.

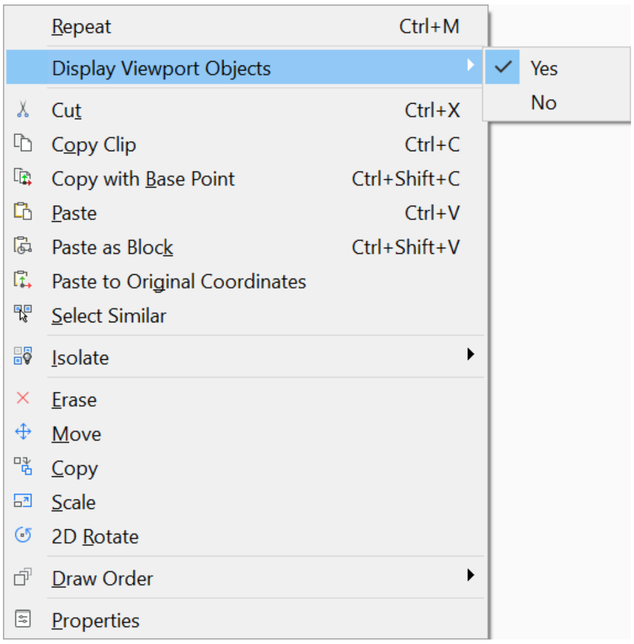
5.16.2 Defining the viewport properties

1 Select the viewport.

1 The viewport properties appear in the **Properties** panel.

2 Click the settings field of a property to modify.

Name	Description
Center	XYZ coordinates of the center point of the viewport with respect to the bottom left corner of the printable area of the layout.

Name	Description
Height (*)	Height of the viewport in **drawing units**.
Width (*)	Width of the viewport in **drawing units**.
On	Controls the display of the content of the viewport. ***Note:*** You can also control the display of the selected viewport from its context menu: 1 Select a viewport. 2 Right-click to open the context menu. 3 Expand the **Display Viewport Objects** option. 4 Choose between **Yes** or **No**. 
Clipped	Indicates whether the viewport is clipped or not.
Display locked	Locks the scaling of the viewport content to preserve the scaling factor. The VPLOCK field in the **Status** bar displays the Display Locked status of the selected viewport(s). Click the field to toggle the **Display Locked** property of the selection.

Name	Description
Scale	The SCALELISTEDIT command allows you to edit the available scales in the current drawing. **Annotation scale**: Sets the annotation scale for the viewport. Editing the Annotation scale automatically sets the Standard scale. If the ANNOALLVISIBLE system variable is OFF, annotative entities which do not support the annotation scale are not displayed. **Standard scale**: By default, the Standard scale matches the Annotation scale, but you can adjust it manually. The **Standard scale** field reads '**Custom**' if the value in the **Custom scale** field is not available in the drawings scale list. **Custom scale**: Displays the viewport scale factor. Allows you to set the viewport scale to a scale which is not available in the drawings scale list.
UCS per viewport	If **Yes**, lets you define a UCS for this viewport.
Visual Style	Sets the visual style for the viewport. This option only applies to 3D models.
Shade Plot	Allows you to adjust the plotted shading from the visual style. Default is '**As Displayed**'.
Default Lighting	Allows you to switch off the lights in the model and use the default lighting.

5.17 Editing paper space viewports

5.17.1 Setting the layer properties in a paper space viewport

The layer visibility can be controlled per paper space viewport. When working in a paper space layout, a number of VP* (Viewport) properties are available, which affect the current viewport only and can be adjusted from the corresponding default properties:

- VP Freeze
- VP Color
- VP Linetype
- VP Lineweight
- VP Transparency
- VP Plot Style

1. Double-click inside the paper space viewport.
2. Launch the EXPLAYERS command.

3 The **Drawing Explorer / Layers** dialog box appears.

4 (option) Click the icon in the **VP Freeze** column to toggle the display of a layer.

5 (option) Click the icon in the **VP Color** column, then select a color in the **Color** dialog box.

6 (option) Click the **VP Linetype** column, then select a linetype in the drop-down list.

7 (option) Click the **VP Lineweight** column, then select a lineweight in the drop-down list.

8 (option) Click the **VP Transparency** column, then select a transparency value in the drop-down list.

9 (option) Click the **VP Plot Style** column, then select a plot style in the **Select Plot Style** dialog box. This option applies to STB (Style Tables) drawings only.

Note:

- When the **Indicate layers in use** option in the **View** menu of the **Drawing Explorer** dialog box is set, the icon in the **Current** column indicates whether VP property overrides exist.
- Use the **Viewports / Vpfreeze** option of the LAYOFF and LAYFRZ commands to freeze layers in a viewport by clicking entities.

5.17.2 Clipping a viewport

The **VPCLIP** command (short for "viewport clipping") clips viewports in layouts.

Only circles and closed polylines can be used to create a clipped viewport. Therefore, use the **Create polyline representations of an ellipse** option of the PELLIPS system variable when creating an elliptical viewport.

5.17.3 Removing the clipping boundary of a viewport

The **Delete** option of the VPCLIP command removes the clipping boundary.

5.17.4 Rotating the display in a paper space viewport

The display of a paper space viewport can be rotated only if the display of the viewport is not locked.

The **Align** > **Rotate** option of the MVSETUP command defines the angle of the content of a viewport relative to the X-axis of the world coordinate system (WCS).

If the VPROTATEASSOC system variable is ON, the ROTATE and MANIPULATE commands rotate the content of a viewport over a specified angle. The viewport is clipped by a rectangle of the dimensions of the original viewport. Use the **Delete** option of the VPCLIP command to remove the clipping.

In order to undo the rotation:

- Set the rotation angle = 0 (zero) in the **Align** > **Rotate** option of the MVSETUP command.
- Double-click inside the viewport, then launch the PLAN command and press Enter twice (= set plan view of the WCS).

5.17.5 Aligning viewports

1 Launch the MVSETUP command.

2 Choose **Align** in the prompt menu.

You are prompted: Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:

3 (option) Choose the **Horizontal** alignment option.

You are prompted: Specify basepoint:

- 1. Pick a point in the reference viewport.
- 2. **You are prompted**: Specify point in viewport to be panned:
- 3. Pick a point in the viewport to be aligned.
- 4. The display in the viewport is moved vertically to align the two points horizontally.

4 (option) Choose the **Vertical alignment option**.

You are prompted: Specify basepoint:

- 1. Pick a point in the reference viewport.
- 2. **You are prompted**: Specify point in viewport to be panned:
- 3. Pick a point in the viewport to be aligned.
- 4. The display in the viewport is moved horizontally to align the two points vertically.

5.17.6 Aligning model space and paper space points

The ALIGNSPACE command adjusts viewport angle, zoom factor, and pan position based on alignment points specified in model space and paper space.

Aligning 1 point

1 Launch the ALIGNSPACE command.

1 **You are prompted**: First alignment point in Model space:

2 Click inside a layout viewport, then pick a point.

3 **You are prompted**: Second alignment point in Model space or <Return> for none:

4 Right-click or press Enter.

5 **You are prompted**: Alignment point in Paper space:

6 Pick a point in the layout.

7 **You are prompted**: Activate the viewport which will be used for alignment.

8 Click inside the viewport you want to align.

9 Right-click or press Enter to confirm.

10 The display in the selected viewport is moved to make the two points coincident.

Aligning 2 points

1 Launch the ALIGNSPACE command.

1 **You are prompted**: First alignment point in Model space:

2 Click inside a layout viewport, then pick a point.

3 **You are prompted**: Second alignment point in Model space or <Return> for none:

4 Pick a second point in the layout viewport.

5 **You are prompted**: First alignment point in Paper space:

6 Pick a point in the layout.

7 **You are prompted**: Second alignment point in Paper space:

8 Pick a second point in the layout.

9 **Prompts you:** Activate the viewport which will be used for alignment.

10 Click inside the viewport you want to align.

11 Right-click or press Enter to confirm.

12 The display in the selected viewport is moved, rotated and/or scaled to make the points two by two coincident.

5.18 Layouts

In BricsCAD, there is no limited number of layouts that can be created in a single drawing. Each layout represents a sheet of paper. For each layout you can specify the paper size, print area, print scale, lineweight scale, pen mappings, and add viewports, dimensions, a title block, and other geometry specific to the layout. The entities you add to a layout in paper space do not appear in model space.

Each layout requires at least one layout viewport. If the CREATEVIEWPORTS system variable is ON, a viewport is added automatically in a new layout. If this variable is OFF, newly created viewports are empty. To add new layouts, you can either define them yourself or you can import layouts from another drawing: template file (DWT), drawing file (DWG) or drawing interchange file (DXF).

Viewports can display all or part of the drawing's model space entities.

The CACHELAYOUT system variable controls whether the content of layouts is cached. The extra memory required for caching is made up for by improvements to the memory requirements for displaying entities, the net result being that the memory needed to display drawings has been reduced.

5.18.1 Creating a new layout

1 Launch the LAYOUT command, then select the **New** option.

2 **You are prompted**: Enter a layout name : <Layout1>:

3 Type a unique name for the layout, then press Enter.

4 The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_), or any combination.

5 Right-click or press Enter to accept the default name (for example, Layout1).

6 The new layout tab is added.

Note: Right-click a layout tab, then choose **New Layout** in the context menu or click the plus (+) button after the last layout tab to add a new layout using a default name.

5.18.2 Copying layouts

1 Right-click the layout tab you want to copy.

1 A context menu appears.

2 Choose **Copy** in the context menu.

You are prompted: Enter new layout name:

3 Type a new unique name in the Command line, then press Enter.

4 A copy of the selected layout is added.

5.18.3 Arranging the layout tabs

Do one of the following:

- Drag the layout tab to the desired position.
- Right-click the layout tab, then choose a **Move Left** or **Move Right** option in the context menu.

5.18.4 Importing layouts

1 Right-click the **Model** tab or one of the layout tabs.

A context menu appears.

2 Choose **From Template...** in the context menu.

The **Select Template From File** dialog box opens.

3 Browse to the drawing that you want to import layouts from.

4 Click the **Open** button on the **Select Template From File** window.

The **Insert Layout(s)** window opens.

5 Select the layout(s) you want to import.

6 Hold down the Ctrl key to select multiple layouts.

7 Click the **OK** button.

The selected layouts are imported.

5.18.5 Exporting layouts

1 Right-click one of the **Layout** tabs.

A context menu appears.

2 Choose **Export** in the context menu.

The **Export layout to drawing** dialog box opens.

3 Browse the location where you want to export the layout.

4 Click the **Save** button.

5 Choose if you want to open the created file.

6 The selected layout is exported.

5.18.6 Renaming a layout

1 Double-click the layout tab you want to rename.

1 The **Rename layout** dialog box opens.

2 Type a new unique name in the **Name** field.

3 Click the **Save** button.

5.18.7 Deleting a layout

1 Right-click the layout tab you want to delete.

1 A context menu appears.

2 Choose **Delete** in the context menu.

3 An **Alert** box appears.

4 Click the **OK** button to confirm.

5.18.8 Using the Layout Manager

1 Launch the LAYOUTMANAGER command.

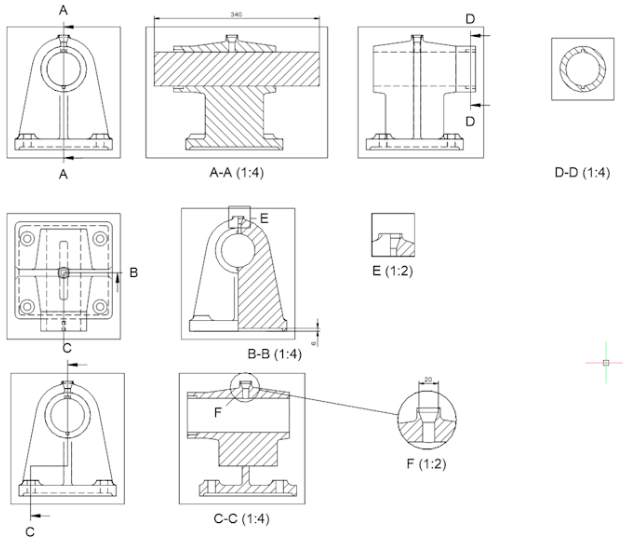
2 In the **Layout Manager** dialog box you can:

- Create layouts.
- Copy layouts.
- Delete layouts.
- Change the order of the layout tabs.
- Publish layouts.

5.19 Generated drawing views

The Generated Drawing Views functionality allows you to automatically generate associative orthographic and standard isometric views of a 3D solid model. All drawing views are placed in a paper space layout, they are not visible in model space.

Analytical hidden line removal (HLR) procedures are used to create the drawing views using standard 2D entities, mostly lines and arcs. Much like the result of the FLATSHOT command. Drawing views created in V14.1 will be automatically converted into HLR geometry by the VIEWUPDATE command.



The VIEWBASE command generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.

The VIEWSECTION command creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.

The VIEWSECTIONSTYLE command allows you to modify the appearance of the section views created by the VIEWSECTION command.

The VIEWPROJ command generates additional projected views from an existing generated drawing view.

The VIEWDETAIL command creates a detail view of a portion of a standard generated drawing, at a larger scale.

The VIEWDETAILSTYLE command allows you to modify the appearance of the detail views created by the VIEWDETAIL command.

The VIEWUPDATE command updates a selection of drawing views obtained by VIEWBASE and VIEWSECTION when VIEWUPDATEAUTO = 0.

The VIEWEXPORT command exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing. This command can be used in Paper Space only.

The VIEWEDIT command allows you to change the scale and the hidden line visibility of drawing views.

Tools to generate drawing views of your 3D solid model are available on:

- The Model - Drawing Views menu.
- The Section Types and Detail Types toolbars.
- The **Views** panel of the Annotate Ribbon tab (Mechanical Workspace).

5.19.1 Standard drawing views

Standard views include multiview orthographic and isometric projections.

In a technical drawing, a multiview orthographic projection is an illustration technique in which up to six images of an object are generated, with each projection plane parallel to one of the coordinate axes of the object.

Isometric projection is a method to represent three-dimensional objects in two dimensions in technical and engineering drawings. It is an axonometric projection in which the three coordinate axes appear equally foreshortened and the angles between any two of them are 120 degrees.

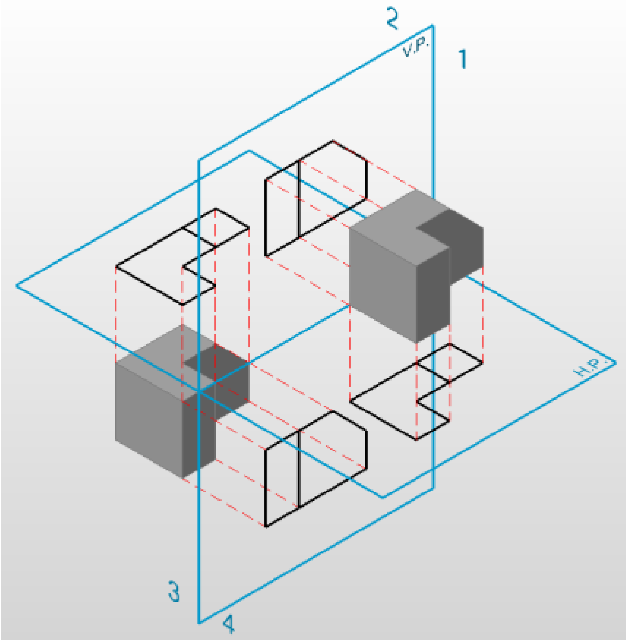
To generate the standard views of your 3D solid model, run the ViewBase command from Model Space.

You can choose between first angle (European) projection or third angle (American) projection.

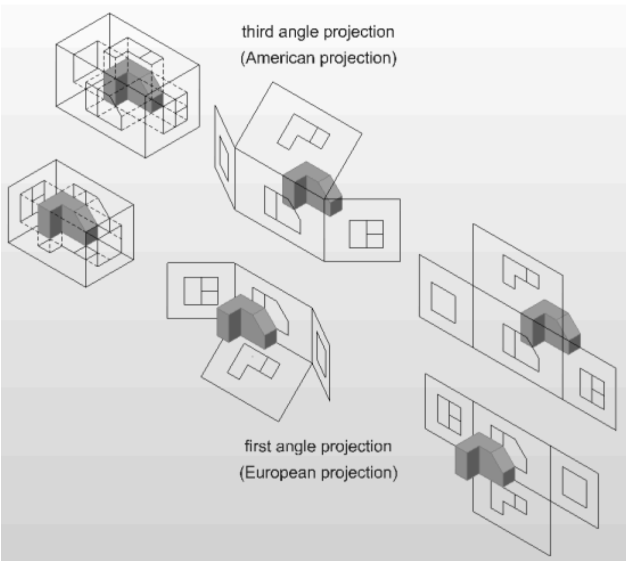
Orthographic projection (or orthogonal projection) is a means of representing a three-dimensional object in two dimensions. It is a parallel projection, where all the projection lines are orthogonal to the projection plane, resulting in every plane of the scene appearing in affine transformation on the viewing surface. It is further divided into multiview orthographic projections and axonometric projections.

Two projection planes, one horizontal (H.P.) and one vertical (V.P.) divide 3D space in four quarters.

- In the first angle projection system, the model is placed in the first quarter space (first angle).
- In the third angle projection system, the model is placed in the third quarter (third angle).



With multiview orthographic projections, up to six views of a 3D model are produced, with each projection plane parallel to one of the coordinate axes of the model. The views are positioned relative to each other, according to either of two schemes: first-angle or third-angle projection. In each, the appearances of views may be thought of as being projected onto the 6 faces of a box around the model. When unfolding the box, all views are organized in a 2D drawing.



The projection type is set by the **Projection Type** option of the VIEWBASE command or by editing the PROJECTIONTYPE system variable:

- 0 = First angle projection
- 1 = Third angle projection

The value of the PROJECTIONTYPE system variable is saved in the drawing.

Generated drawing view lines are created on default layers:

- BM_Isometric_Hidden;
- BM_Isometric_Visible;
- BM_Ortho_Hidden;
- BM_Ortho_Visible.

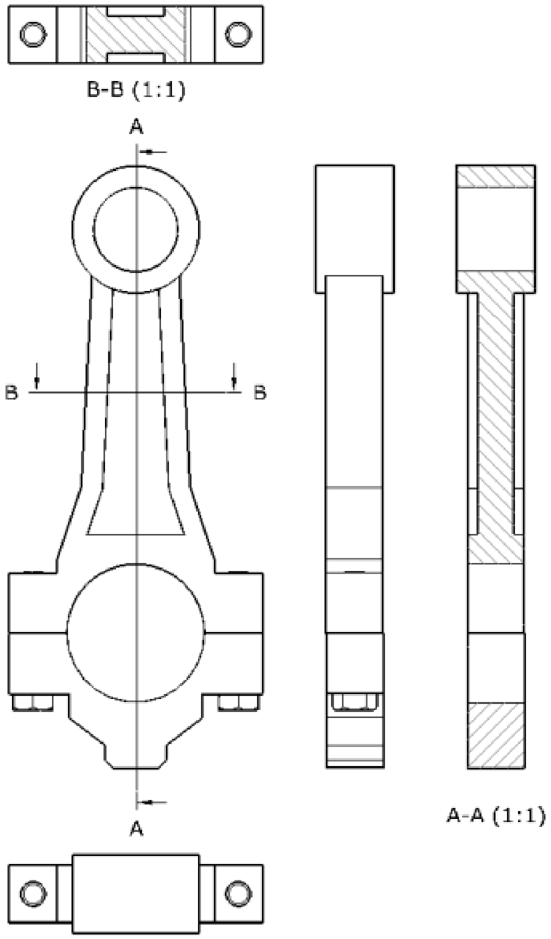
The display of the BM_Isometric_Hidden layer is turned off by default.

Additional projected views can be added using the VIEWPROJ command. A projected view inherits the scale, display settings and alignment from the parent view.

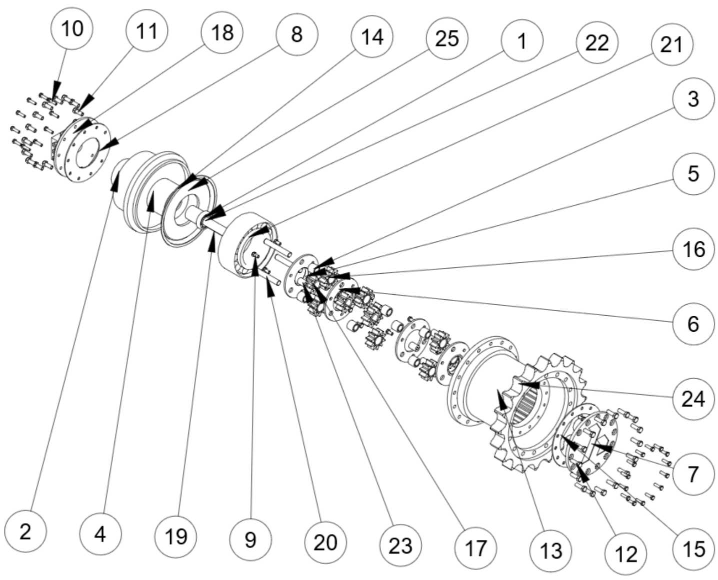
5.19.2 Generating section views

A section is the intersection of a 3D solid model with a section plane. In technical drawings, the internal parts of the 3D model are hatched in cross sections.

To generate section views of a 3D solid model, run the VIEWSECTION command from the paper space layout where the standard drawing views are located.



5.19.3 Special views



Bill of Materials Drive_Assembly		
No.	Component	Quantity
1	BBRG-03	2
2	BBRG-03	1
3	BBRG-60	9
4	Body	1
5	Carrier_1	2
6	Carrier_2	2
7	Cover	1
8	Cover_Interal_Seal	1
9	DIN 933	12
10	DIN 933	24
11	DIN 933	16
12	DIN 933	16
13	Drum	1
14	External_Medium_Seal	1
15	External_Seal	1
16	Gear_Planet	9
17	Gear_Sun	3
18	Internal_Cover	1
19	Key	1
20	Pin	3
21	RLR BRG-32	1
22	Shaft	1
23	Sleeve	6
24	Sprocket	1
25	Washer	1

It is possible to generate drawing views for a selected exploded representation. Normally, an orthogonal projection is used for that; however, BricsCAD allows you to create any standard projection for exploded representations. Sections and detailed views are also supported.

To create a view for an exploded representation, run the VIEWBASE command with the **Special** views option, select the **Exploded view** option and then select the required exploded representation from the dialog box.

Generated views for exploded representation can be associative with the original model, so annotations and balloons can be placed. To do that, the views should be created or updated with GENERATEASSOCVIEWS = ON.

5.19.4 Updating drawing views

By default BricsCAD checks whether the source 3D solid model or the associated exploded representation was modified and automatically recalculates the drawing when:

- Opening a paper space layout containing out-of-date views if the VIEWUPDATEAUTO system variable is ON.
- Running the VIEWUPDATE command.
- Running the BMUPDATE command.

Note:

- Hidden line removal calculation might take some time for complex 3D models. In such cases, it is recommended to set VIEWUPDATEAUTO = OFF. The viewport border of out-of-date drawing views turns red to indicate an update is needed. When executing the ViewUpdate command, BricsCAD prompts you to either select the drawing views to be updated or update all drawing views.

- If GENERATEASSOCVIEWS = ON, associative dimensions will be updated according to the modified 3D geometry.

5.19.5 Exporting views to model space

The associativity mechanism of drawing views in a paper space layout allows for limited control by the user. Such views can be moved or scaled, but you cannot edit the geometry. In order to get the full control to the drawing views geometry, the VIEWEXPORT command allows you to move or copy drawing views to model space or to a separate drawing. Exported drawing views lose their associativity with the 3D model and become standard blocks, which can be exploded, edited, erased, etc.

5.19.6 Customizing drawing views

The first time the VIEWBASE command is launched, four new layers are created:

- BM_Ortho_Hidden: hidden lines in drawing views;
- BM_Ortho_Visible: visible lines in drawing views;
- BM_Tangent_Hidden: hidden tangent lines in drawing views;
- BM_Tangent_Visible: visible tangent lines in drawing views.

Use the VIEWEDIT command to change the visibility of these layers for all views or for a selection of views only.

- Edit the properties of these layers, such as linetype, lineweight or color, to modify the display of the drawing views.

Use VIEWSECTIONSTYLE command to adjust different properties of section views, such as:

- Style, color, and height of identifier text.
- Start and end symbols, their colors, size and direction of arrows scale, pattern, color, and angle of hatches.
- Color, type, and weight of a line representing a cutting plane.

Use the VIEWDETAILSTYLE command to adjust different properties of details views, such as:

- Style, color, height of identified text, and the type of identifier.
- Symbol, and its color and size
- Color, type, and weight of detail view boundary

6. 2D Drafting

6.1 Creating entities

In a drawing you can create a variety of different entity types. Drawing entities can be very simple such as lines, circles, arcs, points and rays or complex such as polylines, splines and planes. Apart from 2D entities you can also create 3D surface entities or 3D solids in BricsCAD.

To further complete your drawing, BricsCAD provides a series of tools to add text, hatching and dimensions.

6.1.1 General procedure to create new entities

New entities are created using the current entity creation settings on the current layer in the XY-plane of the current coordinate system, by default this is the World Coordinate System (WCS).

1 (option) Set the current layer.

2 (option) Adjust the entity creation properties in the **General** section on the **Properties** panel.

3 Launch the command.

4 Execute the command according to the command prompts.

6.2 Entity creation settings

The default settings and preferences to create drawing entities are specified by system variables.

You can view and modify these variables:

- In the Settings dialog box. Expand **Drafting** > **Entity Creation**. At the top of the list, general properties such as color, linetype, and layer are displayed. Below that, settings for entity-specific properties are displayed.
- In the **General** node of the **Properties** panel when nothing is selected.

The **General** settings are:

- **Color**: Sets the entity color. Default = ByLayer.
- **Layer**: Sets the current layer.
- **Linetype**: Sets the current linetype. Default = ByLayer.
- **Linetype scale**: Sets the current linetype scale. Default = 1.
- **Lineweight**: Sets the current lineweight. Default = ByLayer.
- **Transparency**: Sets the current transparency. Default = ByLayer.
- **Elevation**: Specifies the height (Z-value) above the XY-plane of the current coordinate system (WCS or UCS). Default = 0.

Note: Most entity creation settings are saved in the drawing, so they only apply to the current drawing. If you want to adjust these settings for all your future drawings, edit your default template drawing(s) and adjust the settings there. Template drawings are best saved in the folder saved through the TEMPLATEPATH setting. The BASEFILE system variable specifies the path and name of the default template file.

6.3 Working with layers

You can create and manage layers in the **Layers** section of the **Drawing explorer** dialog box (EXPLAYERS command) or in the **Layers** panel (LAYERSPANELOPEN command).

The **Drawing Explorer** dialog box cannot be opened while you use other commands, while the Layers panel can. You can move and resize it as desired, or even move it to a second monitor.

6.3.1 Current layer

New entities are drawn on the current layer. To draw new entities on a different layer, you must first make that layer the current layer.

- The current layer is saved through the CLAYER system variable.
- The CENTERLAYER* system variable controls the layer where centerlines and centermarks created by the DIMCENTER command are placed on.
- The DIMLAYER* system variable controls the layer for dimension entities created by the dimensioning commands.
- Entities created by the INTERFERE command are created on the layer which is specified by the INTERFERELAYER system variable.

* The default value is <Use Current>

To set the current layer, do one of the following:

- With no entity selected, select **Layer** on the Properties panel, then choose the new current layer in the drop down list.
- On the **Entity Properties** toolbar, click the down arrow next to the **Layer** field, then choose a layer in the drop down list.
- On the **Layers** panel on the Ribbon, click the down arrow and choose a layer in the drop down list.
- Right click on the **Layer** field in the Status Bar, then choose a layer in the context menu.

Note:

- It is possible to turn the display of the current layer off ( ). As a result, everything you draw will be invisible until the display of the current layer is turned on ( ) again. A warning displays if you try to turn off the current layer.
- It is not possible the freeze ( ) the current layer.

6.3.2 Layer states

LAYERSTATE command.

Layer states are named content stored in a drawing.

Layer states allow you to save and restore configurations of layer properties and states. You might want a layer to display in blue sometimes and green at other times, or you need some layers to be off or frozen or locked when editing a specific part of a drawing. After spending a lot of time adjusting all layer settings, you can use layer states to save your settings. You can save multiple layer states in each drawing. You can copy layer states between drawings and you can save a layer state to an external file, which can then be imported in another drawing.

6.4 Entity linetype

You can use different linetypes to differentiate the purpose of one line from another. A linetype consists of a repeating pattern of dots, dashes, or blank spaces. Linetypes determine the appearance of entities both on the screen and when printed.

By default, every drawing has at least three linetypes: Continuous, Bylayer and ByBlock. Your drawing can also contain an unlimited number of named linetypes.

Linetype definitions are saved in the drawing. New linetypes are either imported from other drawings or loaded from a linetype file (*.lin). Which linetype source file is used by default is controlled by the MEASUREMENT system variable:

- MEASUREMENT = ON uses **Iso.lin**: Linetype definitions are expressed in mm.
- MEASUREMENT = OFF: **Default.lin**: Linetype definitions are expressed in inch.

New entities are drawn using the current linetype set by the CELTYPE system variable.

The options to set the Linetype are:

- ByLayer: Entities adopt the linetype of their layer. This allows you to change the linetype of all such entities by adjusting the linetype of the layer.
- ByBlock: Entities drawn as continuous lines until you include them into a block. The entities then inherit the linetype setting of the block entity when you insert the block into a drawing.
- Named linetype: Continuous or one of the linetypes saved in the drawing.

Linetype scale

Non-continuous linetypes consist of lines, gaps and dots put together in a variety of patterns. In complex linetypes also text or shapes can be included.

In the linetype definition the length of the lines and gaps are either defined in inch (default.lin) or mm (iso.lin). If the drawing units are different from inch or mm, linetypes need to be scaled. The linetype scaling is controlled by the LTSCALE system variable, which must be set with respect to the drawing units (DU):

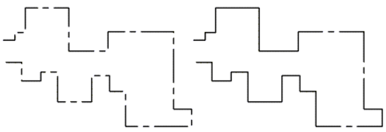
Drawing Unit	LTSCALE
Imperial (MEASUREMENT = OFF)	
Inch	1
Foot	1/12
Metric (MEASUREMENT = ON)	

Drawing Unit	LTSCALE
mm	1
cm	0.1
m	0.001

BricsCAD will always put a dash at the start and end of a line, polyline, arc or spline. The linetype pattern is also nicely centered, so that both ends of the entity look the same.

Polyline Generation

Linetypes are normally generated from vertex to vertex. Polylines of which the vertices are very close together might be rendered as a continuous line, if the linetype pattern does not fit between two subsequent vertices. The PLINEGEN system variable addresses this problem: when set to 1 or ON, the linetype is drawn from one end of the polyline to the other end, instead of from vertex to vertex.



The same polyline drawn with PLINEGEN on (left) and off (right).

Paper space linetype scale

The PSLTSCALE system variable controls the linetype scaling in paperspace layouts.

If ON, linetype scaling is independent from the viewport scale.

In general, PSLTSCALE is best set ON. Notice that this variable can be set per layout.

Model space linetype scale

The MSLTSCALE system variable controls the linetype scaling in model space.

If ON, the current annotation scale (CANNOSCALE system variable) is applied to linetypes in model space.

MSLTSCALE is best set OFF. It might be useful to switch it ON when printing from model space.

6.4.1 Setting the current linetype

Do one of the following:

- On the **Properties** panel, select **Linetype** .

  Make sure that no entity is selected.

  Choose a linetype in the drop down list.
- On the **Entity Properties** toolbar, click the down arrow next to the **Linetype** field, then choose a linetype in the drop down list.
- Right click on the **Linetype** field in the Status Bar, then choose a linetype in the context menu.

6.4.2 Adding a new linetype

1 Do one of the following:

- Launch the LINETYPE command, then choose the **New** option.
- Select **Linetype** in the **Properties** panel, then click the down arrow and choose **Load...**

The **Load linetypes** dialog box displays.

2 (option) Click the **File...** button to load a different linetype source file.

3 Select one or more linetype(s) in the **Load linetypes** dialog box.

Hold down the Ctrl-key to add or remove linetypes.

4 Click the **OK** button

6.4.3 Setting the entity linetype scale

1 On the **Properties** panel, select **Linetype scale**.

Make sure no entity is selected.

2 Type a new value in the Linetype scale field.

The value of the Linetype Scale is saved through the CELTSCALE system variable. This variable is best set to 1, except for special linetypes, such as BATTING or GAS_LINE, which most likely need to be scaled.

Note: The Entity Linetype Scale applies to new entities only.

6.5 Entity color

The color determines how an entity displays on screen and how it prints. Entities are created in the current color, as defined by the CECOLOR system variable.

You can choose between 255 **index colors** or define a **true color**.

6.5.1 Index colors

Index color is the specification of the color of a pixel on a display screen using a an 8-bit color value, allowing up to 256 possible colors.

Each of the Index Colors has a unique number from 1 to 255. Seven of the index colors can also be referred to by name: red (1), yellow (2), green (3), cyan (4), blue (5), magenta (6) and white/black (7). Index color 7 displays white on a black screen background and black on a white screen background. Index color 7 always prints in black.

The two additional color properties are ByLayer and ByBlock. These color properties cause an entity to adopt the color either of the layer or of the block in which it is a member. ByLayer is color number 256, and ByBlock is color number 0. In all commands where you would use a color, you can indicate ByLayer and ByBlock as well as by numbers 256 and 0, respectively.

The options to set the Color are:

- ByLayer: Entities adopt the color of their layer. This allows you to change the color of all such entities by adjusting the color of the layer.

- ByBlock: Entities are drawn in index color 7 (black or white, depending on the screen background color). When included in a block definition, such entities adopt the color of the block when you insert the block into a drawing.
- Index Color
- True Color

Note: Whether entities using an index color will print in this color, depends on the **Color Table** (CTB) or **Style Table** (STB) that is used for printing. Only if the **Color** setting in the CTB or STB definition file is set to **Use Object Color**, the printed color matches the entity color.

6.5.2 True colors

True color is the specification of the color of a pixel on a display screen using a 24-bit value, which allows up to 16 777 216 possible colors.

The number of bits used to define a pixel's color shade is its bit-depth. True color is also known as 24-bit color. Some new color display systems offer a 32-bit color mode. The extra byte, called the alpha channel, is used for control and special effects information.

True colors use a **RGB** (**R**ed, **G**reen, **Bl**ue) color definition. Each of these parameters has a range from 0 to 255. The RGB definition for black is (0,0,0), the RGB definition for white is (255,255,255).

6.5.3 Setting the current entity color

1 Do one of the following:

- Launch the COLOR command.
- Click the **Select Color** tool button ( ) on the Settings toolbar.
- In the **Properties** panel, select **Color**, then choose **Select Color** in the drop down list.
- Double click the **Color Field** in the Status bar.

The **Color** dialog box displays.

2 (option) Click the **Index Color** tab, then do one of the following:

- Click one of the colored tiles.
- Type the color number in the **Color Number** field.

3 (option) Click the **True Color** tab, then do one of the following:

- Click the in **Color Selection** panel.
- Set the color parameters in the **Color Parameter** fields.

4 Click the **OK** button to confirm.

6.6 Lineweight

Lineweights determine how thick or thin entities appear on the screen and / or when printed.

The lineweight can be set ByLayer, ByBlock, Default or to a specific value.

Lineweights are defined in millimeters or inches, depending on theLWUNITS system variable.

You cannot assign lineweights to planes, points, TrueType fonts, and raster images.

The options to set the lineweight are:

- ByLayer: The lineweight is determined by the Lineweight property of the current layer. Changing the Lineweight property of a layer will affect all entities that have their lineweight set ByLayer on this layer.
- ByBlock: Entities created using the lineweight ByBlock, are get the Default lineweight until they are included in a block definition. The entities then inherit the lineweight of the block when you insert the block into a drawing.
- Default: The default lineweight is saved through the LWDEFAULT system variable. Changing the Default lineweight affects all the entities in the entire drawing that have the lineweight set to Default.

6.6.1 System variables

The CELWEIGHT system variable defines the lineweight for new entities.

The LWDEFAULT system variable sets the value of the Default lineweight.

The LWDISPLAY system variable controls the display of lineweights. Click the LWT field in the Status bar to toggle the lineweight display On/Off.

The LWDISPLAYSCALE system variable controls the display scale of lineweights in the Model tab. Values between 0 and 1 are accepted.

6.6.2 Printing lineweights

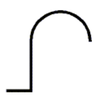
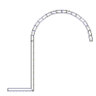
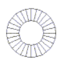
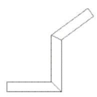
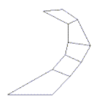
Entities using anindex color will use lineweights when printed only if the**Color Table**(CTB) or**Plot Style**(STB) is defined as **Use object lineweight**. Otherwise the lineweight is defined by the entity color or plot style.

Entities using atrue coloralways use the object line weight when printed.

6.7 Fill mode

TheFILLcommand toggles the display of filled and hatched areas and specifies whether wide polylines, traces, solids, hatches (including solid-fill) and wide polylines are filled in. It toggles the value of the FILLMODE system variable.

When FILLMODEis Off, all filled entities display and print as outlines and reduces the time it takes to display or print a drawing.

	Polyline	Donut	Trace	Plane (Solid)
Fill On				
Fill Off				

6.7.1 Setting Fill Mode

Do one of the following:

- Run the FILL command.
- Set the value of the FILLMODE system variable.

Run the REGEN or REGENALL command to regenerate the viewport(s).

6.8 Transparency

The Transparency property allows to control the transparency of drawing entities. You can set the transparency for a selection of entities or for all entities on the same layer. The default transparency for new entities is controlled by the CETRANSPARENCY system variable. The default transparency for Hatches and Gradient Fills is controlled by the HPTRANSPARENCY system variable.

Transparency can be used to enhance the display of your drawings, for example to de-emphasize less important details or entities that are used for reference only.

The options to set the Transparency are:

- ByLayer: apply the value of the transparency property of the layer.
- ByBlock: entities created using the transparency ByBlock inherit the transparency of the block when you insert the block into a drawing.
- Value: values between 0 (fully opaque) and 90 are accepted.

  The transparency level is limited to 90 % to avoid confusion with layers that are frozen or turned off.

6.8.1 Printing transparencies

Transparencies will only be printed if the Plot Transparencies option is checked on the Print or **Page Setup** dialog boxes.

6.9 Editing entity properties

Entity properties can be edited either in the **Properties** panel, or by using the **Rollover Tips** feature.

6.9.1 Editing in the Properties panel

BricsCAD offers a **Properties** panel like the **Properties** palette in AutoCAD®. You can access it with the PROPERTIES command (PR alias) as well as other methods such as double-clicking on an entity.

The properties displayed in the panel depend on the entity type(s) that you have selected. When multiple entities are selected on the shared properties display, the value of shared properties that vary between the selection set, display ***varies***.

As different from AutoCAD®, the **Properties** panel in BricsCAD displays the **Handle** (read-only) of the selected entity, which can be useful to store and recall for programing purposes.

6.9.2 Editing with Rollover Tips

When Rollover Tips = ON (**RT** field in the **Status** bar), a selection of entity properties display at the cursor. Click a property to edit it.

When the Quad is on, click the **Quad** title bar to switch between Rollover Tips and the Quad cursor menu.

Right-click the **RT** field in the **Status** bar and choose **Customize...** in the context menu to open the **Customize / Properties** dialog box. Here you can define which properties display in the **Rollover Tips** for each entity type.

6.10 Drawing linear entities

Commands for drawing linear entities are:

	Draws polylines with x,y,z coordinates.
	Draws arcs.
	Creates a polyline that forms a boundary around closed areas.
	Draws circles.
	Draws wide circles from polylines.
	Draws ellipses and elliptical arcs.
	Draws 2D spirals or a 3D helices.
	Draws line segments.
	Draws multilines.
	Draws connected lines and arcs.
	Draws regular polygons out of polylines.
	Draws semi-infinite lines.
	Draws rectangles and squares.
	Draws revision clouds.
	Draws sketched (freehand) lines.
	Draws open and closed splines in 2D and 3D.
	Draws wide line segments.

	Draws infinite lines.

6.11 Drawing lines

A line consists of two points: a start point and an endpoint. Using theLINEcommand you can draw single lines or a series of connected lines, but each line is considered a separate Line entity.

The Line tool is available on:

- The **Draw** menu.
- The **Draw** toolbar.
- The **Draw** ribbon panel.
- The **Draw** tab of the No Selection Quad.

6.11.1 General procedure

1 Launch the LINE command.

Prompts you: Start of line or [Follow] <Last point>:

2 Specify the start point.

The line displays dynamically.

Prompts you: Set end point or [Angle/Length/Follow/Undo]:

3 Specify the end point.

The line is created. A new line displays dynamically.

Prompts you: Set end point or [Angle/Length/Follow/Undo]:

4 Specify the end point.

The line is created. A new line displays dynamically.

Prompts you: Set end point or [Angle/Length/Follow/Close/Undo]:

5 Do one of the following:

- Keep specifying end points for each new line segment.
- Choose the **Close** option to connect to the start point of the first line.
- Press Enter or right click to stop.

6.11.2 Specifying the angle and the length

The options are:

- Use the Angle and Length command options.
- Use dynamic dimensions (DYNDIVIS = 1 or 2).

Press the TAB key to switch between the Angle and Length fields.

6.11.3 Specifying the angle to another linear entity

Right click the DUCS field in the Status bar, then choose DUCS on other entities to set the **Enable for 2D Entities** option of the UCSDETECT system variable.

1. Launch the LINE command.
2. Hover over the reference line or linear polyline segment.

   A temporary UCS aligns with line or polyline segment.

   The UCS icon displays at the nearest midpoint or endpoint of the entity under the cursor.
3. Press the Shift-key to lock the temporary UCS.
4. Specify the start point of the line.

   Angles are now relative to the selected line or linear polyline segment.

Note: The WCS is restored when you conclude the LINE command.

6.12 Drawing rays

ARayis a line that starts at a point and extends to infinity. Because rays extend to infinity, they are not calculated as part of the drawing extents.

The default method for drawing a ray is to select the start point of the ray, and then specify its direction.

As they are often used as references for creating other entities Rays andInfinite Lines(XLINE) are sometimes referred to as construction lines.

The Ray tool is available on:

- The Draw menu.
- The Draw toolbar.
- The **Draw** ribbon panel.

6.12.1 General procedure

1. Launch the RAY command.

   Prompts you: Infinite ray: Start of ray or [Horizontal/Vertical/Angle/Bisect/Parallel]:
2. Specify the start point of the ray.

   The ray displays dynamically.
3. Specify the direction of the ray.

   The next ray displays dynamically.
4. (option) Keep specifying directions to draw a series of rays, starting from the same point.
5. Right click to stop drawing rays.

6.13 Drawing infinite lines

AnInfinite Line(XLINE) Is a line through a given point, oriented at a specified angle and extending to infinity in both directions. Because infinite lines extend to infinity, they are not calculated as part of the drawing extents.

The default method for drawing an infinite line is to select a point and then specify its direction.

As they are often used as references for creating other entities, Rays andInfinite Lines are sometimes referred to as construction lines.

The Infinite Line tool is available on:

- The **Draw** menu.
- The **Draw** toolbar.
- The **Draw** ribbon panel.

6.13.1 General procedure

1 Launch the XLINE command.

Prompts you:

Infinite line: Set point along line or [Horizontal/Vertical/Angle/Bisect/Parallel]:

2 Specify the start point of the infinite line.

The infinite line displays dynamically.

3 Specify the direction of the infinite line.

The next infinite line displays dynamically.

4 (option) Keep specifying directions to draw a series of infinite lines, starting from the same point.

5 Right click to stop.

6.14 Drawing polylines

A polyline is an open or closed sequence of connected line and/or arc segments, which are treated as a single entity. Each segment of a polyline can have a width that is either constant or tapers over the length of the segment.

The PLINE command interactively creates open and closed polylines entering points.

The BOUNDARY and -BOUNDARY commands create closed polylines from an enclosed area, defined by other entities.

The JOIN command creates polylines from a series of connected lines, polylines and arcs.

The **Polyline** tool is available on:

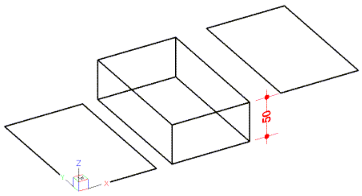
- The **Draw** menu.
- The **Draw** toolbar.
- The **Draw** ribbon panel.

- The **Draw** tab of the No Selection Quad.

The **Thickness** property specifies the height of a polyline. If Thickness = 0 (zero) there is no visual difference between polylines and a series of lines and/or arcs.

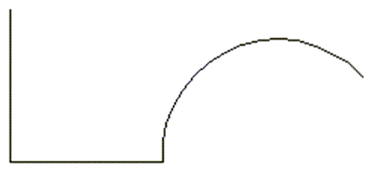
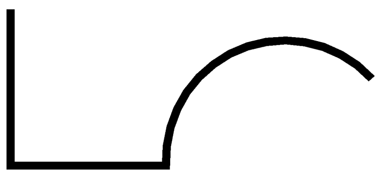
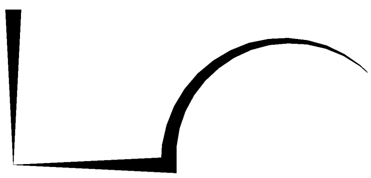
The default thickness is saved through the THICKNESS system variable.

The **Elevation** property specifies the distance between the plane of the polyline and the XY-plane of the WCS (World coordinate system).



THICKNESS = 0 (left), **THICKNESS** = 50 (middle), **ELEVATION** = 50 (right)

When a polyline is edited, you can modify the entire polyline or change individual segments and you can add or remove vertices.

		
Zero width polyline	Constant width polyline	Tapered width polyline

Note:

- When FILLMODE = 0, only the outline of all filled entities, such as wide polylines and planes, displays.
- Depending on the value of the GRIPS system variable polylines have grips on their endpoints and vertices (GRIPS = 1) or endpoints, vertices and vertex midpoints (GRIPS=2)
- Polylines have a direction, which can be modified by the **Reverse Direction** option of the PEDIT (polyline edit) command.

6.14.1 General procedure

1. Launch the PLINE command.

   Prompts you: Select start of polyline or [Follow] <Last point>:

2. Specify a point.

   Prompts you: Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]:

3. Specify a point.

   Prompts you: Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width/Undo]:

4. Specify a point.

Prompts you:

Set next point or [draw Arcs/Close/Distance/Follow/Halfwidth/Width/Undo]:

5 Keep specifying points.

6 Do one of the following:

- Choose the **Close** option to connect to the start point and create a closed polyline.
- Choose the **Undo** option to remove the previous vertex.
- Right click or press Enter to stop.

6.14.2 Creating boundary polylines

1 Launch the BOUNDARY command.

The **Boundary** dialog box displays.

2 Click the **Pick points in boundaries** button.

The dialog box closes.

3 Click in the area where you want to create the boundary polyline.

The boundary polyline highlights.

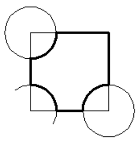
4 (option) Keep picking points to create more boundary polylines.

5 Right click to accept the polyline(s).

The **Boundary** dialog box opens again.

6 Click the **OK** button.

The polyline(s) is (are) created.



6.15 Drawing multilines

Multilines are made of two or more parallel lines, according to the style created by the MLSTYLE command.

The Multiline tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel

6.15.1 General procedure

1 Launch the MLINE command.

1 Current Settings: Justification = Top, Scale = 20.00, Style = Standard

You are prompted: Specify start point or [Justification/Scale/STyle]:

2 Specify a point.

3 **You are prompted**: Set end point or [Angle/Length/Undo]:

4 Specify a point.

5 **You are prompted**: Set end point or [Angle/Length/Follow/Undo]:

6 Specify a point.

7 **You are prompted**: Set end point or [Angle/Length/Follow/Close/Undo]:

8 Do one of the following:

- Specify more points.
- Choose the **Close** option to create a closed loop.
- Right click to conclude.

Note:

Multilines are created using the current multiline style as defined by the CMLSTYLE system variable.

The CMLJUST system variable sets the justification of the multiline relative to the specified points. The options are:

- **Top**: At the top line of the multiline style.
- **Zero**: At the zero offset of the multiline style definition.
- **Bottom**: At the bottom line of the multiline style.

The CMLSCALE system variable sets the scale of the multiline.

The properties of a multiline can be edited in the Properties panel.

6.16 Drawing splines

A spline is an open or closed smooth curve defined by a set of points. You can use splines to draw curved lines, which cannot be drawn as a polyline.

The Spline tool is available on:

- The **Draw** menu.
- The **Draw** toolbar.
- The **Draw** ribbon panel.
- The **Draw** tab of the No Selection Quad.

6.16.1 General procedure

1 Launch the SPLINE command.

Prompts you: First point of spline:

2 Specify a point.

Prompts you: Second point:

3 Specify a point.

Prompts you: Set next point or [Close/Fit tolerance/Undo]:

4 (option) Repeat step 3 to define more points.

5 Right click to stop adding more points.

Prompts you: Select starting tangent point:

A line that is tangent to the spline displays dynamically from the start point of the spline.

6 Click to define the starting curve of the spline.

Prompts you: Select ending tangent point:

A line that is tangent to the spline displays dynamically from the end point of the spline.

7 Click to create the spline.

6.17 Drawing freehand sketches

A freehand sketch consists of a series of straight line segments, created either as individual line entities or as a polyline.

A freehand sketch line is not added to the drawing until you "write" the sketch into your drawing.

The SKPOLY system variable defines whether a chain(s) of connected lines or a polyline(s) is created.

The SKETCHINC system variable sets the length or increment of each line or polyline segment. The smaller the increment, the more accurate your sketch is, but a small value increases the file size.

The Sketch tool is available on:

- The **Draw** menu.
- The **Draw** toolbar.
- The **Draw** ribbon panel.

6.17.1 General procedure

1 Launch the SKETCH command.

1 Prompts you: Length of sketch line segments <1>:

2 Do one of the following:

  - Press Enter to accept the current length.
  - Type a new value in the Command line.
  - Click two points to define the length graphically.

The crosshairs are replaced by thePenciltool ()

Prompts you: Press ENTER or click to toggle [Erase/Connect/save and eXit/Quit no save/Record (save)]:

3 Click to start sketching: as you move the cursor the sketched line displays in light green.

4 (Optional) Click to stop sketching, then click again to resume.

5 (Optional) Choose the**Erase**option.

The cursor changes to the**Erase**tool ().

Move the cursor to the end of a sketch line to start erasing.

Click to stop erasing and resume sketching.

6 (Optional) Choose the**Record (save)**option to save the sketching line and keep sketching.

7 Choose the**save and eXit**option to save the sketch lines and exit the sketch command.

6.18 Drawing circles

The default method for drawing a circle is to specify a center point and a radius.

Other methods to draw circles can be found in the Circle submenu of the Draw menu and in the Circles toolbar.

The AI_CIRCTAN command draws circles tangent to three linear entities.

The Circle tool is available on:

- The **Draw** menu.
- The **Draw** toolbar.
- The **Draw** ribbon panel.
- The **Draw** tab of the No Selection Quad.

6.18.1 General procedure

1 Launch the CIRCLE command.

Prompts you:

Select center of circle or [2 Point/3 Point/Tangent-Tangent-Radius/turn Arc into circle/Multiple circles]:

2 Specify a point.

Prompts you: Set Radius or [Diameter]:

3 Do one of the following:

- Press Enter to accept the current radius.
- Type a new radius and press Enter.
- Click to define the radius graphically.

6.18.2 Drawing a circle tangent to three entities

1 Launch the AI_CIRCTAN command.

Prompts you: Enter Tangent spec:

2 Click a linear entity.

Lines, arcs, circles, polyline segments, xlines and rays are accepted.

Prompts you: Enter second Tangent spec:

3 Click the second linear entity.

Prompts you: Enter third Tangent spec:

4 Click the third linear entity.

A circle tangent to the three selected entities is created.

6.18.3 Editing a circle

1 Select the circle.

The properties of the circle display in the **Properties** panel.

2 Type a new value in the **Center**, **Radius**, **Diameter**, **Circumference** or **Area** field, then press Enter.

3 Press the Esc key to stop editing the circle.

6.19 Drawing arcs

The default method for drawing circular arcs is to specify three points: the start point, a second point and the endpoint.

Other methods to draw arcs can be found in the Arc submenu of the Draw menu and in the Arcs toolbar.

The Arc tool is available on:

- The **Draw** menu.
- The **Draw** toolbar.
- The **Draw** ribbon panel.
- The **Draw** tab of the No Selection Quad.

6.19.1 General procedure

1 Launch the ARC command.

Prompts you: Enter start of arc or [Center/Follow last point] <Follow last point>:

2 Specify a point.

Prompts you: Enter second point or [Angle/Center/Direction/End/Radius]:

3 Specify a point.

Prompts you: End point:

4 Specify a point.

6.19.2 Editing an arc

1 Select the arc.

The properties of the arc display in the **Properties** panel.

2 Type a new value in the **Center** , **Radius** , **Start angle** or **End angle**, then press Enter.

3 Press the Esc key to stop editing the arc.

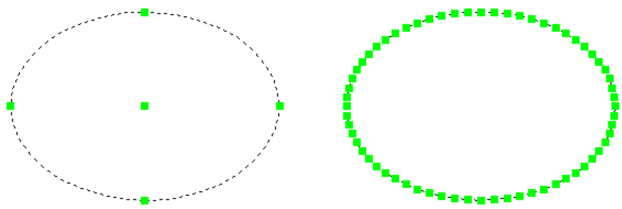
6.20 Drawing ellipses

The default method for drawing an ellipse is to specify the endpoints of one axis of the ellipse and then specify a distance representing half the length of the second axis. The endpoints of the first axis determine the orientation of the ellipse. The longer axis of the ellipse is called the major axis, and the shorter one is the minor axis. The order in which you define the axes does not matter. The program determines the major and minor axes based on their relative lengths. Half the major axis is called the major radius, half the minor axis is the minor radius. The ratio (minor radius divided by the major radius) defines the eccentricity of the ellipse. A circle is an ellipse with an eccentricity of 1.

Other methods to draw ellipses can be found in the Ellipse submenu of the **Draw** menu and in the Ellipses toolbar.

The PELLIPSE system variable controls whether real ellipses or a polyline representation is created.

The difference between a real ellipse and a polyline representation of an ellipse is visible when you select the ellipse. A real ellipse has five handles: center point and the endpoints of the axes. A polyline representation of an ellipse is a closed polyline composed of arc segments.



real ellipse (left) and the polyline representation of an ellipse (right)

The Ellipse tool is available on:

- The **Draw** menu.
- The **Draw** toolbar.
- The **Draw** ribbon panel.
- The **Draw** tab of the No Selection Quad.

6.20.1 General procedure to create an ellipse

1 Launch the ELLIPSE command.

Prompts you: First end of ellipse axis or [Arc/Center]:

2 Specify a point:

Prompts you: Second end of axis:

3 Specify a point:

Prompts you: Set Other axis or [Rotation]:

4 Specify a point.

6.20.2 General procedure to create an elliptical arc

An elliptical arc is a portion of an ellipse.

1 Launch the ELLIPSE command.

Prompts you: First end of ellipse axis or [Arc/Center]:

2 Select the Arc option.

Prompts you: First end of ellipse axis or [Center]:

3 Specify a point:

Prompts you: Second end of axis:

4 Specify a point:

Prompts you: Set Other axis or [Rotation]:

5 Specify a point.

Prompts you: Set start angle of arc or [Parameter]:

6 Do one of the following:

- Type a value in the dynamic field.
- Click to specify the arc graphically.

Prompts you: [Parameter/Included angle] <End angle>:

7 Click a point.

6.20.3 Editing an ellipse or an elliptical arc

1 Select the ellipse or elliptical arc..

The properties of the ellipse display in the **Properties** panel.

2 Type a new value in the **Center**, **Major radius**, **Minor radius**, **Radius ratio**, **Start angle** or **End angle** field, then press Enter.

3 Press the Esc key to stop editing.

Note: Editing the Start angle or End angle fields of an ellipse turns the ellipse into an elliptical arc. Setting both properties to 0 (zero) turns an elliptical arc into an ellipse.

6.21 Drawing rectangles

A rectangle is created as a closed, four sided polyline.

A rectangle is drawn by specifying two opposite corners. Dynamic dimensions allow you to specify the width and height of the rectangle.

Unless the **Rotated** option is chosen, the sides of a rectangle are always parallel to the x-axis and y-axis of the current coordinate system (WCS or UCS).

The Rectangle tool is available on:

- the **Draw** menu

- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad**

6.21.1 General procedure

1 Launch the RECTANG command.

1 **Prompts you**:

Select first corner of rectangle or [Chamfer/Fillet/Rotated/Square/Elevation/Thickness/Width of line/Area/Dimensions]:

2 Specify a point.

3 The rectangle displays dynamically.

Prompts you: Other corner of rectangle:

4 Do one of the following

- Specify a point.
- Type a value in the dynamic dimension fields.
- Press the TAB key to switch. Press Enter to confirm.

Note:

Because a rectangle is created as a closed polyline, rectangles have a direction. The direction of a closed polyline is positive if it is drawn counter-clockwise and negative if it is drawn in a clockwise direction. Rectangles have a counter-clockwise direction if both the X- and Y-coordinate of the second point are higher or lower than the coordinates of the first point (= movement of the cursor is positive or negative along both X- and Y-axis). Rectangles have a clock-wise direction if the X- or Y-coordinate is higher or lower than the X- or Y-coordinate of the first point (= movement of the cursor is positive along the X-axis and negative along the Y-axis or vice versa).

The Reverse Direction option of the PEDIT (Edit Polyline) command changes the direction of a polyline.

6.22 Drawing polygons

Polygons are closed polylines comprised of a minimum of three and a maximum of 1,024 equal-length sides.

You can draw polygons using the following methods:

- **Center - Vertex**: first define the center point, then the vertex (= radius of the circumscribed circle method).
- **Center - Side**: first define the center point, then the midpoint of a side (= radius of the inscribed circle method).
- **Edge**: define the length of the side of the polygon.

The Polygon tool is available on:

- the **Draw** menu
- the **Draw** toolbar

- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad**

6.22.1 General procedure

1 Launch the POLYGON command.

1 **Prompts you**: Polygon: Set number of sides or [Width of line/Multiple polygons] <4>:

2 Do one of the following:

- Press Enter or right click to accept the current number of sides.
- Type a value, then press Enter.

Prompts you: Specify by: Set center of polygon or [specify by Edge]:

3 Specify a point.

4 **Prompts you**: Specify by: Select midpoint of side or [specify by Vertex]:

5 Specify a point.

Note:

- The option that you choose in step 3 (midpoint of side or specify by Vertex) is remembered the next time.
- The number of sides is set by the POLYSIDES system variable, but the value is not saved between sessions. The first time you launch the POLYGON command in a BricsCAD session POLYSIDES = 4.

6.23 Drawing traces

Traces are 4-sided filled entities, created with two parallel sides. When you create a series of traces, they are L-connected to each other.

The Trace tool is available on

- the **Draw** menu.
- the **Draw** ribbon panel

The default trace width is saved through the TRACEWID system variable.

6.23.1 General procedure

1 Launch the TRACE command.

1 **Prompts you:** Width of trace <current width>:

2 Do one of the following:

- Right click to accept the current width.
- Type a value, then press Enter.
- Define a new current width graphically by clicking two points.

Prompts you: Start of trace:

3 Specify the start point of the trace.

4 **Prompts you:** Next point:

5 Specify the end point of the first trace.

6 A line indicates the direction of the first trace.

The direction of the second trace displays dynamically.

7 Do one of the following:

- Right click to create the trace.
- Specify the endpoint of the next trace.

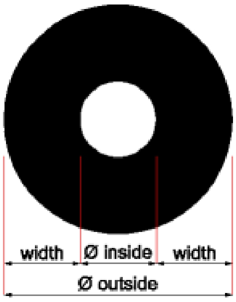
8 Do one of the following:

- Repeat step 5 to keep adding traces.
- Right click to stop.

Note: When the FILLMODE system variable is turned off, all filled entities, such as traces, display and print as outlines.

6.24 Drawing donuts

Donuts are solid, filled circles or rings created as closed, 2-segment, wide polylines.



The default method to draw donuts is to specify its inside and outside diameters, and then specify its center. If the inside diameter is zero, a filled circle is created.

The Donut tool is available on:

- the **Draw** menu
- the **Draw** ribbon panel

6.24.1 General procedure

1 Launch the DONUT command.

1 **Prompts you**: Inside diameter of donut [2 Point/3 Point/Tangent Tangent Radius] < current inside diameter >:

2 Do one of the following:

- Right click to accept the current inside diameter.
- Type a new value, then press Enter.
- Define a new inside diameter graphically by clicking two points.

Prompts you: Outside diameter of donut <current outside diameter>:

3 Do one of the following:

- Right click to accept the current outside diameter.
- Type a new value, then press Enter.
- Define a new outside diameter graphically by clicking two points.

Prompts you: Center of donut:

Specify a point.

Prompts you: Center of donut:

4 Do one of the following:

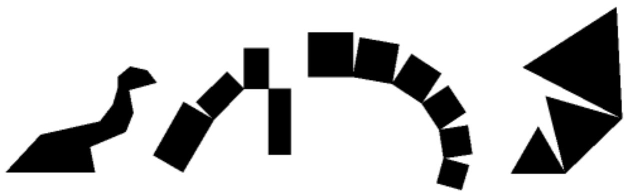
- Keep creating donuts.
- Right click to stop.

Note:

- When the FILLMODE system variable is turned off, all filled entities, such as donuts, display and print as outlines.
- The current inside and outside diameter value is saved through the DONUTID and DONUTOD system variables. These values are not saved between sessions. The first time you launch the DONUT command in a BricsCAD session DONUTID=0.5 and DONUTOD=1.

6.25 Drawing solids

2D Solids are rectangular, square, triangular or quadrilateral areas filled with a solid color.



Left to right: quadrilateral, rectangular, square and triangular solids

The Solid tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel

6.25.1 General procedure

1 Launch the SOLID command.

1 **Prompts you**: Set first point of plane or [Rectangle/Square/Triangle]:

2 Specify a point.

3 **Prompts you**: Second point:

4 Specify a point.

5 **Prompts you**: Third point of plane:

6 Specify a point.

7 **Prompts you**: Fourth point:

8 Specify a point.

9 The solid is created.

Prompts you: Third point of plane:

10 Do one of the following:

- Keep specifying points to create more solids.
- Right click to stop.

Note: When the FILLMODE system variable is turned off, all filled entities, such as solids, display and print as outlines.

6.26 Drawing revision clouds

A Revision Cloud is an open or closed polyline composed of arcs. Revisions clouds are used to for reviewing or redlining purposes to indicate the parts of the drawing that need to be adjusted or annotated.

Revision Clouds: 1. Normal 2. Calligraphy 3. Reversed calligraphy (based on a rectangle).

Note: The **Reverse** option is available only in **Entity** mode or when creating an open revision cloud using the **Freehand** mode.

REVCLOUDCREATEMODE: sets the default mode.

REVCLOUDMINARCLENGTH: sets the minimum arc length.

REVCLOUDMAXARCLENGTH: sets the maximum arc length.

REVCLOUDARCSTYLE: sets the default style (normal or calligraphy).

REVCLOUDGRIPS: when On, grip editing a revision cloud also affects adjacent segments.

The Revision Cloud tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad**

6.26.1 General procedure

1 Launch the REVCLOUD command.

1 **Prompts you**:

Specify first point or [Arc length/Entity/Rectangular/Polygonal/Freehand/Style] <default mode>:

2 Specify a point to start creating a freehand mode revision cloud.

3 As you move the cursor, the revision cloud is being created.

4 Move the cursor to the start point to finish.

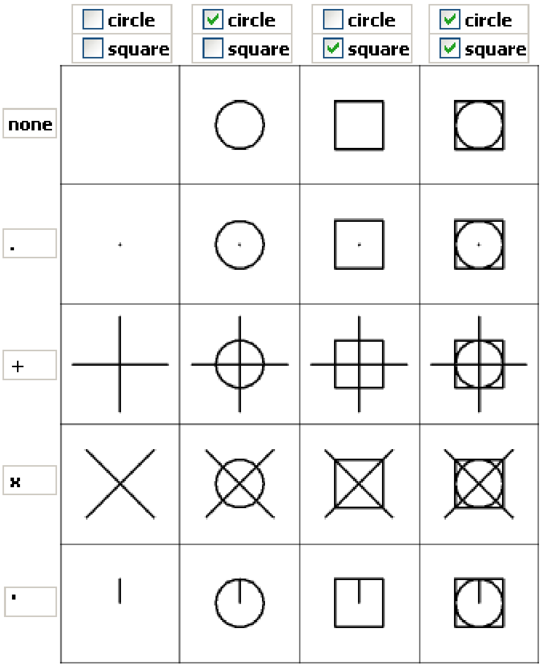
5 BricsCAD reports in the command line: Revision cloud finished.

Note:

- If you right click at the first prompt you start creating a revision cloud in the default mode (= current value of the REVCLOUDCREATEMODE system variable).
- If you right click before the cursor is at the start point, an open revision cloud is created.

6.27 Drawing points

Since points are dimensionless, a single pixel should be the (nearly) correct representation on the screen. But this is hardly visible, especially in a complex drawing. Therefore you can choose between a number of possible display styles, set by the PDMODE (Point Display Mode) system variable:



The size of the point representation style is controlled through the PDSIZE (Point Display Size) system variable. When PDSIZE=0 the point display size is 5% of the screen size.

Note: If the PDSIZE=0 (zero), use the REGEN command to resize the display of the points to 5% of the screen size after zooming in or out.

The Point tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel

6.27.1 General procedure

1 Launch the POINT command.

1 **Prompts you**: Location of point or [point Settings.../Multiple points]:

2 Specify a point.

6.28 Drawing wipeouts

A wipeout is a polygonal area that hides the underlying entities using the current background color. Most commonly wipeouts are used to mask part of a drawing for clarity. For example, you may want to add text to a complicated part of a drawing. A wipeout could be used then to mask an area behind the text to increase the readability of the text.

The **Frame** option of the WIPEOUT command sets the value of the WIPEOUTFRAME system variable, which controls whether the wipe out outlines display or not, and whether they are printed or not.

Note: Wipeouts hide existing entities only. Use the **Bring to front** option of the DRAWORDER command to hide all entities.

The Wipeout tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad**

6.28.1 General procedure

1 Launch the WIPEOUT command.

1 **Prompts you:** Specify start point or [Polyline/Frames]:

2 Specify a point.

3 **Prompts you**: Specify next point:

4 Specify a point.

5 **Prompts you**: Specify next point or [Undo]:

6 Specify a point.

7 **Prompts you**: Specify next point or [Undo/Close]:

8 Do one of the following:

  - Keep specifying points.
  - Right click or choose **Close** to stop.

6.28.2 To create a wipeout from a polyline

1 Launch the WIPEOUT command.

1 **Prompts you:** Specify start point or [Polyline/Frames]:

2 Choose the **Polyline** option.

Prompts you: Select a closed polyline [selection options (?)]:

3 Select a closed polyline, composed of line segments only.

4 **Prompts you**: Delete selected polyline? [Yes/No] <No>:

5 Do one of the following:

- Press Enter to keep the polyline.
- Choose **Yes** to delete the polyline.

6.29 Drawing helixes

A helix is a curve turning about an axis on the surface of a cylinder (cylindrical helix) or cone (conical helix) while rising at a constant upward angle from a base.

A spiral is a planar curve that winds around a fixed center point at a continuously increasing distance from that point.

The HELIX command can create both 3D helixes and 2D spirals.

The Helix tool is available on:

- the **Draw** menu
- the **Polylines** toolbar
- the **Draw** ribbon panel

6.29.1 General procedure to draw a spiral

1 Launch the HELIX command.

1 Number of turns = 3 Twist = CCW

You are prompted: Specify center point of base:

2 Specify a point.

3 Specify base radius or [Diameter] <1>:

4 Do one of the following:

- Specify a point.
- Type a value, then press Enter.

You are prompted: Specify top radius or [Diameter] <base radius>:

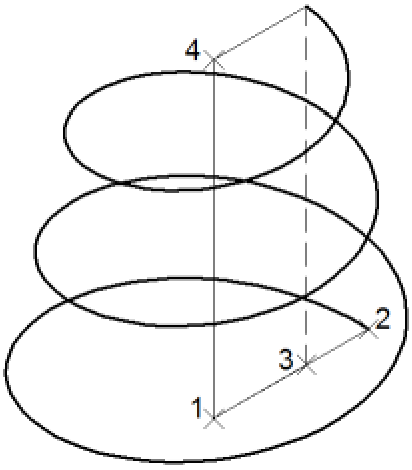
5 Do one of the following:

- Specify a point.
- Type a value, then press Enter.

You are prompted: Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1>:

6 Type 0 (zero), then press Enter

6.29.2 General procedure to draw a helix



1 Launch the HELIX command.

1 Number of turns = 3 Twist = CCW

You are prompted: Specify center point of base:

2 Specify a point (1).

3 **You are prompted**: Specify base radius or [Diameter] <1>:

4 Do one of the following (2):

- Specify a point.
- Type a value, then press Enter.

5 Do one of the following (3):

- Press Enter to set the top radius equal to the base radius, thus creating a cylindrical helix.
- Specify a point.
- Type a value, then press Enter.

You are prompted: Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1>:

6 Do one of the following (4):

- Specify a point.
- Type a value, then press Enter.

6.29.3 Editing a helix

1 Select the helix.

1 The helix properties display in the **Geometry** category in the **Properties** panel.

2 Edit the properties:

- **Constrain**: constrains either the height, turn height or number of turns. The unconstrained properties are adjusted automatically.
- **Height**: Overall height of the helix. height = turn height * turns.

- **Turns**: Number of turns.
- **Turn height**: Height of one turn.
- **Base radius**: start radius.
- **Top radius**: end radius.
- **Twist**: direction CW (Clockwise) or CCW (Counterclockwise).

Note:

- Set the **Top radius** = **Base radius** for cylindrical helixes.
- Set **Height** or **Turn height** to zero for spirals.
- The EXPLODE command turns a helix into a spline.

6.30 Drawing 3D polylines

A 3D polyline is composed of linear segments in 3D space.

The 3DPOLY tool is available on:

- the **Draw** menu
- the **Polylines** toolbar
- the **Draw** ribbon panel

6.30.1 General procedure

1 Launch the 3DPOLY command.

1 **You are prompted**: Select start of polyline or [Follow] <Last point>:

2 Specify a point.

3 **You are prompted**: Next point:

4 Specify a point.

5 **You are prompted**: Next point or [Close/Undo]:

6 Do one of the following:

  - Choose the **Close** option to connect to the start point and create planar triangular 3D polyline.
  - Specify a point.

  You are prompted: Set next point or [Close/Undo]:

7 Do one of the following:

  - Choose the **Close** option to connect to the start point.
  - Specify a point.

  You are prompted: Set next point or [Close/Undo]:

8 Keep specifying points or press **Enter** to stop.

Note:

- 3D polylines with assigned linetypes draw with an uninterrupted pattern. To make linetypes visible for 3D polylines, set LINETYPE3DPLINE system variable to 1.
- PLINEGEN system variable does not affect 3D polylines, and no other setting can change how the linetype pattern for a 3D polyline is drawn.

6.30.2 Editing a 3D polyline

1 Select the 3D polyline.

1 The properties display in the **Properties** panel.

2 Under **Misc**:

  - **Fit/Smooth**: select **None**, **Cubic** or **Quadratic**.

    The **Cubic** and **Quadratic** options create a spline representation of the 3D polyline.
  - **Closed**: When set to **No**, removes the closing segment.

6.31 Creating regions

The REGION command converts closed entities Into a solid entity.

You can create regions from closed entities, such as polylines, polygons, circles, ellipses, closed splines and donuts.

Creating regions typically has no visible effect on a drawing using a wireframe visual style. However, if the original entity had a width or lineweight, that information is lost when you create the region.

In rendered visuals styles, region entities are displayed using a solid fill.

6.31.1 To create regions

1 Launch the REGION command.

1 **You are prompted**: Select entities or boundaries [selection options (?)]:

2 Select one or more entities.

3 Entities in set: x

You are prompted: Select entities or boundaries [selection options (?)]:

4 Select more entities or press Enter to stop selection.

5 BricsCAD reports: x region(s) created.

6.32 Creating hatches and gradient fills

The HATCH command fills an enclosed area in your drawing with a pattern.

The GRADIENT command creates solid fills in nine different patterns and one or two colors.

The area can either be a single entity such as a circle or a closed polyline or an area enclosed by a selection of entities.

To create the hatch you can click inside the closed perimeter of a boundary or you can select entities.

Note:

- Depending on the pattern, hatchings are memory intensive and might take a considerable amount of time to draw and display. To improve performance, add hatching as one of the last steps when you create a drawing, or insert hatches on a separate layer that you can freeze as you continue to work on your drawing. Or use the FILLMODE system variable to switch off the display of hatches, gradient fills and filled entities.
- The maximum number of dashes is controlled by the MAXHATCH settings variable. When the number of dashes is higher than the current MAXHATCH value, the hatch cannot be created. A warning box displays offering possible choices for subsequent actions. By default the maximum number of dashes is 100 000.
- The maximum number of points on a contour (outline) that a hatch entity can contain and still be rendered is controlled by the HPMAXCONTOURPOINTS settings variable. Hatches do not render if the number of points exceeds the specified value. Values between 0 and 10000000 are accepted. The default value of this variable is 100000. Setting to 0 disables the check, i.e. the variable is not used.

The Hatch and Gradient tools are available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad** (Hatch only)

6.32.1 General procedure

1 Launch the HATCH or GRADIENT command.

1 The **Hatch and Gradient** dialog box displays.

2 Do one of the following:

3 (option) Click the **Hatch** tab to create a hatch, then define the hatch properties.

4 (option) Click the **Gradient** tab to create a gradient fill, then define the gradient fill properties.

5 Define the properties for the hatch or gradient fill.

6 Choose the boundary for the hatch or gradient fill.

7 Set the options.

8 Click the **OK** button to create the hatch or gradient fill.

Hatches and gradient fills are created on the layer defined by the HPLAYER system variable. Default = current layer.

6.32.2 To define hatch properties

1 Select the **Hatch** tab on the **Hatch and Gradient** dialog box.

2 (option) Click the **Inherit properties** button.

The **Hatch and Gradient** dialog box closes temporarily.

You are prompted: Select hatch entity to copy properties from:

Select a hatch entity.

The **Hatch and Gradient** dialog reopens, displaying the properties of the selected hatch entity.

3 (option) Select the **Pattern Type**:

- **User Defined** - uses the pattern defined in this dialog box with the **Angle**, **Spacing**, and **Cross Hatch** options.
- **Predefined** - uses patterns defined in:
- • iso.pat (metric units -MEASUREMENT= ON)
- • default.pat (imperial units - MEASUREMENT = OFF).
- **Custom** - uses patterns defined in *.pat files (one pattern definition per file).

  BricsCAD searches for *.pat files in the folders that are defined in the SRCHPATH (Support File Search Path) system variable.

4 If the **Pattern Type** is **Predefined** or **Custom**:

- Double click the **Swatch** field,
- Select a **Pattern Name** in the **Hatch pattern** palette, then double click the pattern or click the **OK** Button.
- Type a scale in the **Scale** settings field or choose the scale from the drop down list.
- Type an angle in the **Angle** settings field or choose the angle from the drop down list.

5 If the **Pattern Type** is **User Defined**:

- Specify the **Pattern Spacing** in drawing units.
- Optionally set the **Cross Hatch** option.

6 Define the **Hatch Origin**.

The options are:

- **Use current origin** : The origin as defined by the HPORIGIN system variable is reused.
- **Specified origin** : Click the **Pick new origin** button, then pick a point in the drawing. The new origin is stored in the HPORIGIN system variable.

7 Set the **Color** for the hatch pattern.

8 Set the **Background color**.

6.32.3 To define gradient fill properties

1 Select the **Gradient** tab on the **Hatch and Gradient** dialog box.

2 (option) Click the **One color** radio button to create a 'color to white' or 'color to black' gradient fill.

- Click the colored tile to open the **Color** dialog box, to pick a different color.
- The color slider sets the transition from color to white or black.
- Press and hold the left mouse button to move the color slider.

Move the slider to the far right to create a 'color to white' gradient fill.

Move the slider to the far left to create a 'color to black' gradient fill.

3 (option) Click the **Two color** radio button to create 'two color' gradient fill.
   - Click the colored tiles to open the **Color** dialog box to pick the colors.
   - Select one of the 9 gradient patterns.
   - The options are: linear, cylindrical, inverted cylindrical, spherical, hemispherical, curved, inverted spherical, inverted hemispherical and inverted curved.

4 (option) **Centered**: If centered, the gradient fill is created symmetrical; if not centered, the gradient fill is built up from the left of the boundary.

5 **Angle**: Specifies the angle of the gradient fill, relative to the current UCS.

6.32.4 To define the boundary

1 Select a **Boundary retention** option.

2 Select an **Island** detection option.

This applies to enclosed areas inside hatch boundaries.

3 Set the **Boundary tolerance**. This value defines the maximum gap size in the boundary. When it is set to 0 (zero), BricsCAD sets the tolerance with respect to the current view size. Zoom out to increase the tolerance.

This value is saved through the HPGAPTOL system variable.

4 (option) Click the **Select** button to select a **Boundary set**.

The **Hatch and Gradient** dialog box temporarily closes to let you select entities.

If a selection set was active when you launch the HATCH or GRADIENT command, this selection set will be used as the **Boundary Set**. In this case, the **Hatch and Gradient** dialog box closes and immediately reopens when you click the **Select** button.

5 Do one of the following:
   - Click the **Pick Points** button.

     The **Hatch and Gradient** dialog box closes.

     You are prompted: Select a point to define a boundary or hatch area:
   - Specify a point in each of the areas you want to hatch.
   - **You are prompted**: Select a point to define a boundary or hatch area:
   - Right click or press Enter to stop adding areas.
   - The **Hatch and Gradient** dialog displays again.
   - Click the **Select Boundary Entities** button.

     The **Hatch and Gradient** dialog box closes.

     You are prompted: Select entities:
   - Select the entities you want to hatch.
   - Right click or press Enter to stop selecting entities.

- The **Hatch and Gradient** dialog box displays again.
- Click the **Remove Boundary Entities** button.

  The **Hatch and Gradient** dialog box closes.

  You are prompted: Select entities:
- Select the current boundary entities to be removed from the selection.
- Right click or press Enter to stop selecting entities.
- The **Hatch and Gradient** dialog box displays again.

6.32.5 Set the options

- The **Annotative** option creates annotative hatches.
- Select the **Associative** option to associate the hatch with its boundary.
- Select the **Create Separate Hatches** option. When multiple boundaries are detected or multiple entities are selected, separate hatches are created, instead of a single hatch.
- Choose a **Draw order** option.

  This option is reset to **Send behind boundary** when the drawing is closed and reopened.
- **Layer**: choose an existing layer or **Use Current** (default).
- **Transparency**: sets the transparency; values between 0 and 90 are accepted.

6.33 Editing hatches and gradient fills

Hatches and gradient fills can be edited:

- In the **Properties** panel.
- Using the HATCHEDIT command, in a dialog box.
- Using tools in the Edit command group in the Quad when a hatch or gradient fill entity is selected.

6.33.1 To edit a hatch or gradient fill in the properties panel

1 Select the hatch or gradient fill in the drawing.

1 The properties of the selected entity display in the **Properties** panel.

2 Select the property you want to edit in the **Properties** bar.

The selected property field is activated.

3 Edit the selected property.

4 Press the Esc key to stop editing the hatch.

Note:

- When multiple hatches or gradient fills are selected properties which are different display as ***varies***. If you edit such property, the changes apply to entire selection.
- Linetype, Linetype scale and Lineweight properties apply to **User Defined** pattern types only.

6.33.2 To edit a hatch or gradient fill in a dialog box

The Edit Hatch tool is available on:

- the **Modify** menu
- the **Modify** toolbar
- the **Modify** ribbon panel

1 Do one of the following:
   - Launch the HATCHEDIT command.
   - Select a hatch or gradient entity.
   - Double click a hatch or gradient fill entity.

   The **Hatch Edit** dialog box displays.

2 Edit the properties and settings.

3 The changes are applied in real time.

4 Click the **OK** button to accept the changes.

6.33.3 To edit a hatch or gradient fill using the Quad

1 Pause the cursor on the hatch or gradient fill.

1 The Quad displays.

2 Move the cursor to the Quad.

3 (option) In the **Edit** tab, click the **Add Loop** button.

   You are prompted: specify internal point or [Select entities/Undo]:
   - Click inside a closed area,
   - or

     Type **S** , then press Enter to select entities.
   - Right click or press Enter to conclude.

4 (option) In the **Edit** tab, click the **Remove Loop** button.

   The boundaries of the selected hatch highlight.

   You are prompted: select entities:
   - Click the boundaries you want to remove.
   - Right click or press Enter to conclude.

5 (option) In the **Edit** tab, click the **Send to back** button.

   All hatches in the drawing are sent behind other entities.

6 (option) In the **Edit** tab, click the **Match** button.

   You are prompted: select entities to copy properties to or [Settings/selection options (?)]:
   - Select the hatch entities to copy the properties to.
   - Right click or press Enter to conclude.

7 (option) In the **Draw** tab, click the **Generate Boundary** button.

   The boundary of the hatch is generated.

6.34 Hatch and gradient system variables

System variables can be set using the **Settings** dialog box.

Variable Name	Variable Title	Description
MEASUREINIT	Measurement initial	Sets drawing units as Imperial or Metric for new created drawings (also controls which hatch pattern and linetype files are used: ANSI for Imperial and ISO for Metric units).
MEASUREMENT	Measurement	Sets drawing units as Imperial or Metric for the current drawing. Measurement controls whether ANSI or ISO hatch pattern and linetype files are used.
OSOPTIONS	Entity snap options	Suppresses entity snaps on certain entity types. The value is stored as a bitcode using the sum of the values of all selected options.
PICKSTYLE	Pick style	Controls the selection of groups and associative hatches. The value of PICKSTYLE is stored as a bitcode using the sum of the values of all selected options. ***Note:*** Ctrl+H toggles PICKSTYLE 0/1 if 0 or 1 and 2/3 if 2 or 3
PREVIEWFILTER	Selection preview filter	Controls which entity types are excluded from selection previewing. The value is stored as a bitcode using the sum of the values of all selected options.
FILLMODE	Fill mode	Determines whether multilines, traces, solids, hatches (including solid-fill), and wide polylines are filled in.
GFANG	Gradient fill angle	Determines the angle of a gradient fill.
GFCLR1	Gradient fill primary color	The first color of a gradient fill.
GFCLR2	Gradient fill secondary color	The second color of a gradient fill.

Variable Name	Variable Title	Description
GFCLRLUM	Gradient fill tint level	Controls the tint intensity in a one-color gradient fill.
GFCLRSTATE	Number of colors for a gradient fill	Defines number of colors (one or two) for a gradient fill.
GFNAME	Gradient fill name	Determines the pattern of a gradient fill.
GFSHIFT	Gradient fill shift	Specifies whether the pattern in a gradient fill is centered or is shifted up and to the left.
HPANG	Hatch pattern angle	Stores the hatch pattern angle.
HPANNOTATIVE	Hatch pattern annotative	Controls whether a new hatch pattern is annotative.
HPASSOC	Hatch pattern associativity	Determines whether new hatch patterns and gradient fills are associative or not. Associative hatches and gradient fills are updated automatically when their boundaries change.
HPBACKGROUNDCOLOR	Hatch background default color	The background color of a hatch. Enter '.' for none.
HPBOUNDRETAIN	Hatch pattern boundary retain	Determines whether HATCH/BHATCH commands will create boundary entities.
HPBOUND	Hatch pattern boundary	Determines the entity type (region or polyline) created by BHATCH and BOUNDARY.
HPCOLOR	Hatch default color	The foreground color of a hatch. Enter '.' to use the current color, as defined in CECOLOR.
HPDOUBLE	Hatch pattern doubling	Controls whether user-defined patterns create a single hatch or a cross hatch.
HPDRAWORDER	Hatch pattern draw order	Sets the draw order of hatches and gradient fills. The Draw Order setting from the Hatch and Gradient dialog is saved here.

Variable Name	Variable Title	Description
HPGAPTOL	Hatch pattern gap tolerance	Defines a tolerance to be used when entities are not completely closed when generating a boundary for BHATCH or BOUNDARY. The Tolerance setting from the Hatch and Gradient dialog is saved here. The default value is 0: the tolerance is set by the application, based on the current view size. When zoomed in closely, boundary detection will fail; when zooming out further so the contour 'looks' closed, the boundary will become detected. Values greater than 0 define the maximum gap in drawing units. Values between 0 and 5000 are accepted.
HPISLANDDETECTION	Hatch pattern island detection	Specifies islands processing within the hatch boundary.
HPLAYER	Default layer for new hatches	Stores the default layer for new hatches.
HPLINETYPE	Hatch pattern linetype	Controls the display of non-continuous linetypes in hatch objects. When off (0, default), linetypes will be displayed continuous, ignoring the assigned linetype. When on (1), non-continuous linetypes will be applied. (Not recommended, decreases performance. You can choose a predefined hatch pattern with a non-continuous linetype instead).
HPMAXAREAS	Fill mode for sparse hatches	Controls how sparse hatches are filled.
HPMAXCONTOURPOINTS	Maximum number of points on a hatch contour	Specifies the maximum number of points on a contour (outline) that a hatch entity can contain and still be rendered. Hatches do not render if the number of points exceeds the specified value. Values between 0 and 10000000 are accepted. The default value of this variable is 100000. Setting to 0 disables the check, i.e. variable is not used.
HPNAME	Hatch pattern name	Stores the default hatch pattern name.

Variable Name	Variable Title	Description
HPOBJWARNING	Hatch pattern object warning	Specifies how many hatch boundary entities can be selected before a warning message appears. Values between 1 and 100000000 are accepted. The default value of this variable is 10000.
HPORIGIN	Hatch pattern origin	Stores the origin point for new hatches, relative to the current UCS.
HPSCALE	Hatch pattern scale	Stores the hatch pattern scale factor.
HPSEPARATE	Hatch pattern separate	Determines whether the HATCH command creates a single hatch pattern object or separate objects when several hatch boundaries are selected.
HPSPACE	Hatch pattern spacing	Sets the hatch pattern line spacing for user-defined hatch patterns.
HPTRANSPARENCY	Default transparency for new hatches	Stores the default transparency level for new hatches. Following values are accepted: '.' (for 'use current', the default), ByLayer , ByBlock, 0 (fully opaque) and 1-90 (transparency percentage).
MAXHATCH	Maximum hatch dashes	Specifies the maximum number of dashes in a hatch pattern. Hatches of which the number of dashes exceeds the maximum number of dashes cannot be created. Values between 100 and 10000000 are accepted. The default value of this variable is 100000.
MIRRHATCH	Hatch pattern mirroring	Determines whether the MIRROR command mirrors hatch patterns.
SRCHPATH	Support file search path	Specifies the folder(s) to search for text fonts, customization files, plugins, drawings to insert, linetypes, and hatch patterns that are not in the current folder. Search paths are separated by semicolons (;).

Variable Name	Variable Title	Description
PdfImportVectorGeometry	Vector geometry	Ignore or Import vector geometry. PDF geometric data types include linear paths, Beziér curves, and solid-filled areas, which are imported as polylines, and 2D solids or solid-filled hatches. Within a tolerance, curves that resemble arcs, circles, and ellipses are interpolated as such. Patterned hatches are imported as many separate objects.
PdfImportSolidFills	Solid fills	Ignore or import solid-filled areas. If these filled areas were originally exported into PDF format from AutoCAD®, the solid areas would include solid-filled hatches, 2D solids, wipeout objects, wide polylines, and triangular arrowheads. ***Note:*** Note: Solid-filled hatches are assigned a 50% transparency so that objects on top or underneath can be easily seen
PdfImportConvertSolidsToHatches	Convert solid fills to hatches	Converts 2D solid objects into solid-filled hatches.

7. 3D Modeling

7.1 Direct modeling overview

Direct modeling is much like sculpting on-screen: just like a sculptor starts with a lump of clay or a piece of stone or wood, you start with a base volume, which is then further detailed by adding, removing and deforming geometry. All operations are executed dynamically, and you decide whether to define dimensions intuitively or by specifying values. Unlike history-based modeling, the order in which features are created is not important. All operations are performed in interactive mode: the intermediate result of an operation shows dynamically, according to user input defined by mouse movement.

A key feature of direct editing capabilities provided by BricsCAD consists in preserving the design intent. 3D dimensional constraints (**Distance**, **Radius**, and **Angle**) imposed for faces and edges give you full parametric control over the solid model. Different configurations and scales of the model can be obtained simply by changing the parameter of a dimensional constraint. Geometric constraints (**Coincident**, **Tangent**, **Parallel** and others) define how these changes influence the geometry of the model. However, there is no need to specify all geometric constraints manually since most of them can be recognized automatically by setting **Design Intent Recognition** options.

Another feature of **Direct Editing** operations is that they can be easily applied with the Quad cursor menu. This tool provides a set of operations based on the currently selected or previewed entity or selection set. At the same time the Quad provides the other basic solid editing operations such as copy, union and subtract.

Direct modeling operations apply to all ACIS solid geometry, whether created in BricsCAD or imported.

When the separately licensed Communicator for BricsCAD® is installed, the IMPORT command supports all major 3D model formats, both industry standard and proprietary formats.

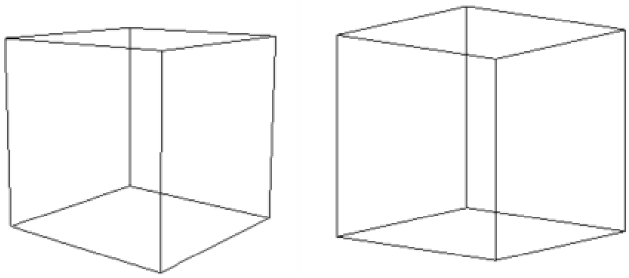
Depending on the value of the DELOBJ (Delete Entity) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities (see table below).

Value	Result
0	Retain defining entities
1	Delete profile entities; retain paths and guide curves
2	Delete all defining entities
-1	Prompt to delete profile entities; retain paths and guide curves
-2	Prompt to delete all defining entities

7.2 Navigate in 3D

As you navigate around and through your 3D model, the location of the model in space remains constant. It is your current view (viewpoint), of the model that is changing. You can see the current view properties in the **Properties** panel when nothing is selected. The **Camera** (X,Y,Z) represents your location as you view the model. Pretend you are viewing it through a camera lens. The **Target** (X,Y,Z) is where the camera lens is pointed. As you navigate around the model, the **Camera** and **Target** coordinates update accordingly. Those coordinates are read-only in the **Properties** panel. There are other **View** properties that you can edit. **Perspective** is the most important control to be aware of for navigation purposes.

You can set **Perspective** to **On** or **Off** for the current view. When set to **On**, the view displays the visual perspective mode (left). When set to **Off**, it displays the parallel perspective mode (right).



7.2.1 To control the Perspective property

Do one of the following:

- Select the **View / Perspective** property in the **Properties** panel.
- Right click the **Look From** widget, then choose **Perspective** in the context menu.

7.2.2 Mouse and keyboard navigation options

One of the easiest and most common methods for navigating around a 3D model is with the mouse and keyboard. Simply press the Shift key and middle mouse button as you move the mouse to orbit the model. The cursor changes to an orbit icon indicating you're in orbit mode.

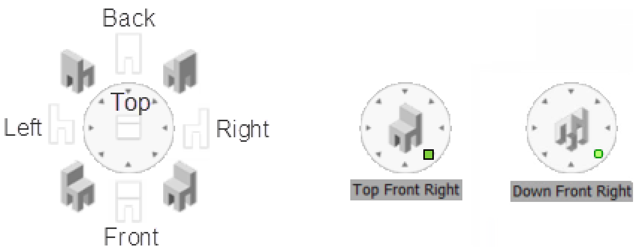
There are many other mouse and keyboard navigation methods that you're probably familiar with from your 2D CAD experience. Most of those methods are also useful for 3D modeling.

- Zoom in/out – Roll the mouse wheel. You can zoom all the way into a model and walk around inside it when in perspective mode. In isometric mode you can only zoom in very close, but not go inside.
- Zoom Extents – Double-click the mouse wheel.
- Pan – Press the mouse wheel or middle mouse button.

See the 'View Manipulation using the Mouse and Keyboard' article for a comprehensive list.

7.2.3 The LookFrom navigation tool

The **LookFrom** tool is displayed in the upper right corner of the **Drawing** area. Click on different places on the **LookFrom** tool to display the view from standard viewpoints. By default, it includes **Top**, **Left**, **Right**, **Front**, **Back**, **Top Back Left**, **Top Back Right**, **Top Front Left**, and **Top Front Right**. As you pass the cursor over a viewpoint, the relevant icon appears at the center and a tooltip displays below. You can press the Ctrl key to access similar options for **Bottom** viewpoints.



A right-click menu offers access to additional controls and settings.

Choose **LookFrom Settings...** in the context menu to open the **Settings** dialog box and edit the **LookFrom** related system variables.

7.3 Direct modeling commands

BricsCAD direct modeling commands can create both solid and surface geometry. To distinguish direct modeling commands from other commands, all direct modeling commands have a prefix **DM**.

Direct modeling commands are available in Pro, BIM, Mechanical and Ultimate license levels. Commands that create geometry are also available in BricsCAD Shape, along with basic modification commands such as DMPUSHPULL, DMCHAMFER and DMFILLET.

7.3.1 Creation commands

EXTRUDE: creates 3D solids or surfaces by extruding open or closed 2D entities, faces of 3D solids, regions or closed boundaries. Optionally adds volume to solids or removes volume from solids.

REVOLVE: creates 3D solids or surfaces by revolution of open or closed 2D entities or regions about an axis.

DMTHICKEN: creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, faces of 3D solids, regions and wire entities.

DMCOPYFACES: copies a set of 3D solid faces forming a feature (e.g. hole, rib or protrusion) to a new position on the same or another 3D solid.

DMSTITCH: converts a set of region and surface entities that bound a watertight area to a 3d solid.

7.3.2 Modification commands

DMCHAMFER: creates equal distance and variable chamfers between adjacent faces sharing a sharp edge.

DMDEFORMCURVE: deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.

DMDEFORMMOVE: deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.

DMDEFORMPOINT: deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction. The selected faces are deformed as smooth as possible.

DMFILLET: creates a smooth fillet between adjacent faces sharing a sharp edge.

DMPUSHPULL: adds or removes volume from a solid by moving/offsetting a face.

DMROTATE: rotates solids, faces, edges, or vertices of a solid around an axis.

DMTWIST: modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

7.3.3 Helper commands:

DMAUDIT: fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces).

DMAUDITALL: fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces) in drawings inserted as external references.

DMGROUP: creates group features, which are named persistent collections of faces and edges of 3D solids and surfaces.

DMSELECT: selects edges and faces of 3D solids or surfaces based on their geometric properties.

DMSELECTEDGES: selects the edges of faces and solids.

DMSIMPLIFY: simplifies the geometry and topology of 3d solid entities. The command removes unnecessary edges and vertices, merges seam edges, and replaces the geometry of faces and edges by analytic surfaces and curves, if possible within the user-specified tolerance.

The direct modeling commands are available in:

- The **Model** tab on the Quad cursor menu when the cursor is over an entity or subentity or when a selection set exists.
- The **Direct Modeling** menu.
- The **Direct Modeling**, **Fillet**, **Chamfer** and **Deform** toolbars.
- The **Home / Direct Modeling** ribbon panel (**Modeling** workspace).

Icon	Command	Applies to
		Any face of a 3D solid or surface.
		Cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface. Linear and circular edges of a 3D solid or surface. Whole 3D solids, surfaces, and other entities.
		Planar, cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface. Whole 3D solids, surfaces, and other entities.

Icon	Command	Applies to
		3D solids, surfaces, and their faces.
		Surfaces.
		3D solids and surfaces.
		Planar faces of 3D solids and surfaces. This tool launches the UCS command with the Face option selected. ***Note:*** When the Dynamic UCS feature is ON, the UCS aligns automatically to the solid face under the cursor and the UCS-icon changes accordingly.
	Solid mode	Closed 2D entities and detected closed boundaries. If the **Select detected boundaries** option of the SELECTIONMODES system variable is switched ON, boundaries enclosed by 2D linear entities in the XY-plane of the current UCS are detected automatically. Planar faces of 3D solids and surfaces. Closed 2D entities and detected closed boundaries on a planar face of a solid.
	Solid mode	Closed 2D entities and detected closed boundaries.* Planar faces of solids and surfaces. Closed 2D entities and detected closed boundaries on the planar face of a solid.
	Extract Faces	Surfaces and faces of 3D solids.
		A set of 3D solid faces.
		Faces of 3D solids and surfaces. Surfaces. Wire entities; lines, polylines, circles, ellipses, arcs, helices, and splines.
	Constant radius mode	Sharp edges of a 3D solid or surface.
	Variable radius mode	Sharp edges of a 3D solid or surface.

Icon	Command	Applies to
		Sharp edges of a 3D solid or surface.
	Advanced Mode	Sharp edges of a 3D solid or surface.
		Faces of a solid or a surface. Whole 3D solids and surfaces. Free edges of a surface (a surface edge is called **free** if it has only one adjacent face). Edges between coplanar faces.
	Surface mode	2D linear entities and edges of 3D solids and surfaces.
	Surface mode	2D linear entities and edges of 3D solids and surfaces.
		Faces of a 3D solid or a surface.
		Edges of a 3D solid or a surface.
		Edges of a 3D solid or a surface.
		3D solids, surfaces or regions.

7.4 Direct modeling operations

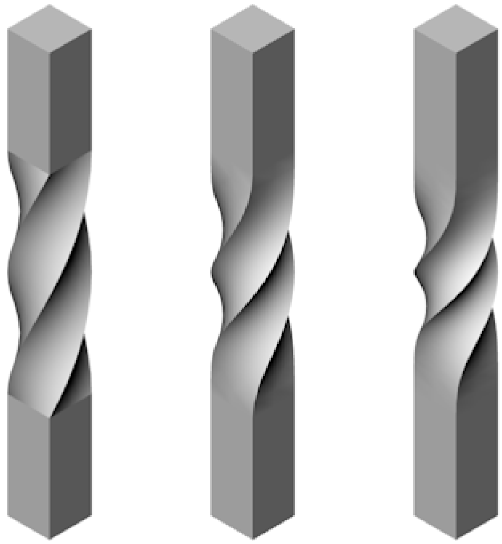
7.4.1 Controlling the coordinate system

Since entities are created in the XY-plane of the current coordinate system (UCS), it is necessary to align the UCS with the 3D solid face. You have two options here:

- The **Align UCS** tool ( ) launches the UCS command with the **Face** option selected.
- Click the **DUCS** field in the **Status Bar** to toggle the dynamic UCS feature ON and OFF. When ON, the UCS aligns automatically to the solid face under the cursor when executing a command and the UCS-icon changes accordingly. Press the Tab key to select an obscured face. Press the Shift key to temporarily lock the UCS, which allows you to work outside the hovered entity or face. Press the Shift key again to unlock it.

7.4.2 Selecting geometry

- Hit the Tab key to select obscured geometry.
- The SELECTIONMODES system variable controls which subentities highlight: edges, faces, and/or closed boundaries. Hold down the Ctrl key to overrule SELECTIONMODES, and select the subentity under the cursor.

- Window selection: to select the faces or edges of 3D solids and surfaces, press the CTRL key to cycle the options. The icon next to the cursor indicates what will be selected.
- The DMSELECT command allows you to select edges and faces based on their geometrical and topological properties:
  - Edges of a particular length.
  - Edges that form a concave/convex network.
  - Edge loops on the border of a set of connected faces.
  - Faces of a particular area.
  - Smoothly connected faces (G1 - tangent faces, or G2 - curvature continuity).
  - 

    G0: No continuity (left), G1: Tangent Faces (middle), and G2: Curvature continuity (right)
  - Fillets of a specific radius.
  - Cylindrical protrusions of a specific radius.
  - Protrusion or depression features; pockets, extrusions, or ribs.

The value of a particular length, area, or radius can be taken from a selected edge/face.

7.4.3 Moving faces to control dimensions

A dynamic dimension displays with the DMPUSHPULL command.

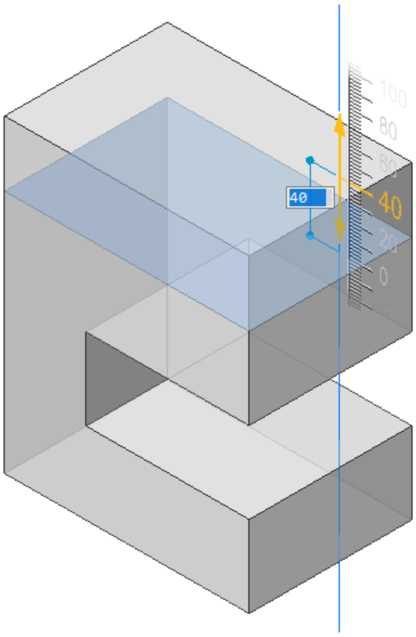
Make sure the **Quad** and **Dynamic Dimensions** are switched ON.

1 Hover the cursor over the face you want to move.

1 The face highlights.

2 Choose **Push/Pull** ( ) in the **Model** tab of the Quad.

The selected face moves with the cursor and a dynamic dimension field displays the relative distance along with a ruler.

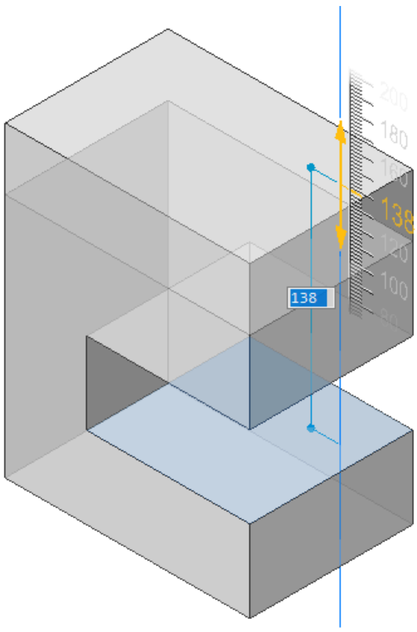


When the cursor is on the ruler, the face moves with fixed intervals. Zoom in or out to increase or decrease the interval.

When the cursor is outside the ruler, the face moves freely.

3 (Option) repeatedly hit the Tab key to see the distance with respect to an internal face, parallel to the selected face (left) or the bottom face (right).

4



Press and hold the Shift key and hit the Tab key to reverse the display order of reference faces.

5 Do one of the following.

- Type a value in the dynamic dimension field, then hit Enter.
- Using the ruler: click when the desired value displays.

7.4.4 Making holes

Method 1:

1 Create the subtractor solid.

2 Use the SUBTRACT command.

3 Depending on the value of the DELETETOOL system variable, the subtractor solid is either deleted or retained.

Method 2:

1 Create the outline of the opening on the 3D solid face.

2 Place the cursor over the outline, then choose **Solid Extrude** () (DMEXTRUDE command) in the **Model** tab of Quad.

3 Move the cursor inside the solid to make the opening.

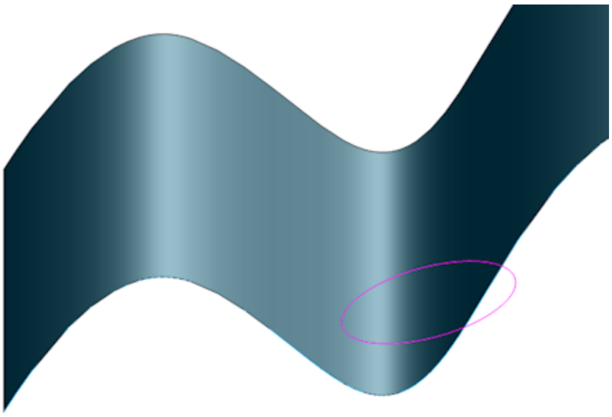
4 Do one of the following:

- Specify a point.
- Type a value.

To make a hole in a surface

1 Create a 2D closed entity, e.g. an ellipse.

1

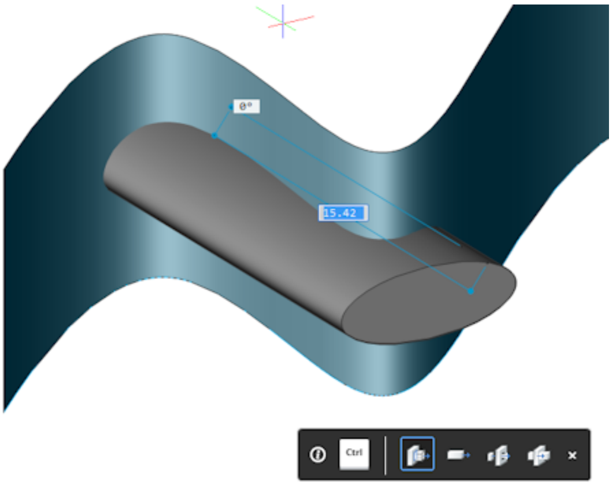


2 Place the cursor over the 2D closed entity, then choose **Solid Extrude** () (DMEXTRUDE command) in the **Model** tab of Quad.

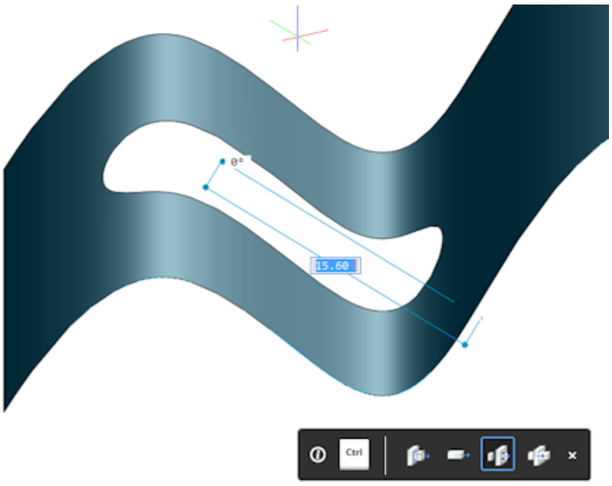
3 Extrude the 2D entity in the direction of the surface.

4 The default extrusion direction is perpendicular to the plane of the 2D entity.

The **Hotkey Assistant** (HKA) appears at the bottom of the screen.



5 Hit the CTRL key twice to select the **Subtract** option of the DMEXTRUDE command.



6 Click to create the opening.

7.4.5 Deleting features

The **Feature** option of the DMSELECT command allows to easily select all faces of a hole, depression or protrusion.

In the **Mechanical** workspace, the **Protrusion** and **Depression** tools are available in the **Select** panel of the **Modeling** ribbon tab and the **Select** tab of the Quad.

The tools sit in the **Sheet Metal** toolbox. See **Available Tools** in the **Customize** dialog box.

To delete a depression (hole or pocket) or a protrusion feature (extrusion or rib)

1 Click the **Protrusion** ( ) or **Depression** ( ) tool button in the **Select** panel of the **Modeling** ribbon tab or from the **Select** command group of the Quad.

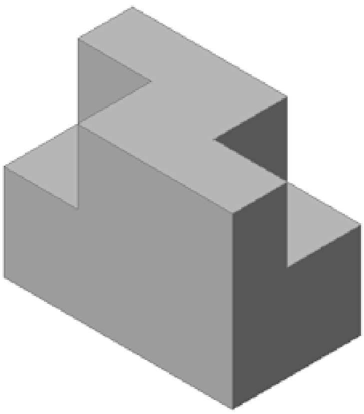
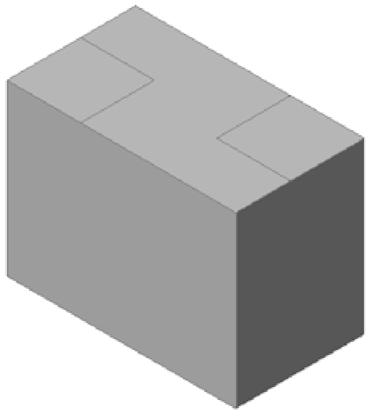
2 Select one or more faces of the feature.

3 All adjacent faces of the feature are detected.

4 Click the **Erase** ( ) tool button.

7.4.6 Removing edges between coplanar faces

When two or more faces are made coplanar, e.g. with the DMPUSHPULL command, it might be necessary to remove the shared edges using the DMSIMPLIFY command.

	
Two depressed faces	Two depressed faces moved to the top face, to create three coplanar faces.

1 +Launch the DMSIMPLIFY command.

1 **You are prompted**: Select entities to simplify or [Entire model/SEttings/selection options (?)] <Entire model>:

2 Select the solid, then press Enter.

3 The edges are removed.

7.4.7 Copying a Feature

The DMCOPYFACES command allows you to copy one or more features from a 3D solid (depressions - holes, grooves, and pockets - or protrusions - extrusions and ribs), to a new location on the same or a different 3D solid.

Make sure dynamic UCS is enabled.

1 Select all faces of the entire feature(s).

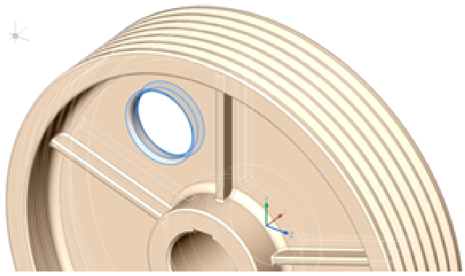
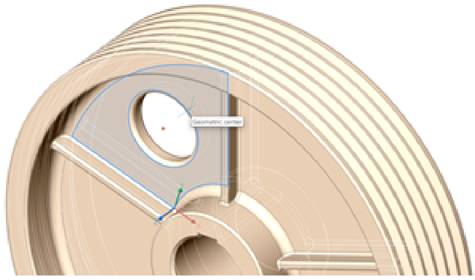
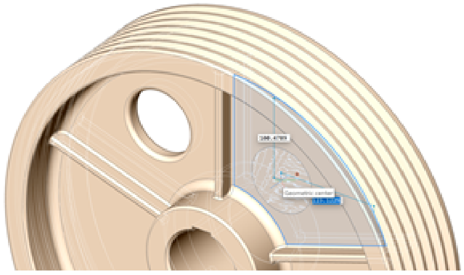
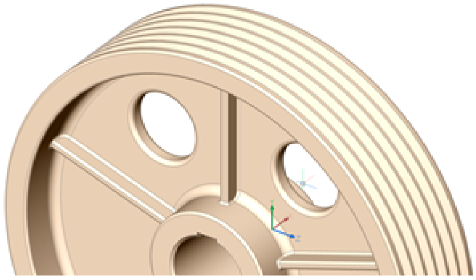
2 Click the **Copy Faces** ( ) tool:

- on the ribbon: **Modeling | Edit | Copy Faces**.
- on the **Direct Modeling** toolbar.
- in the menu bar: **Model | Direct Modeling | Copy Faces**
- in the **Model** tab on the Quad

3 Specify the base point.

4 **You are prompted:** Select an insertion point or [Rotate/mUltiple/MOde]:

5 (Optional):
   - Use the **Rotate** option to rotate the faces around the Z axis of the UCS.
   - Use the **Multiple** option to insert the multiple copies of a feature.
6 Specify the insertion point.
7 The feature is copied from the base point to the insertion point with dynamic UCS.

	
Select the faces of the entire feature	Specify base point
	
Specify the insertion point	The feature is copied

7.4.8 Deformable modeling

Use the tools on the **Deform** toolbar, to deform a set of adjacent faces on a 3D solid or a surface, to create complex aesthetic shapes.

The faces are always deformed if:

- The external boundary (= edges, which are adjacent to only one of the selected faces) remains unchanged.
- The internal boundaries (= edges, which are adjacent to two of the selected faces) are deformed together with the faces.
- The initial continuity between deformed faces (G1 – tangent faces, or G2 – curvature continuity) is kept when deformed.

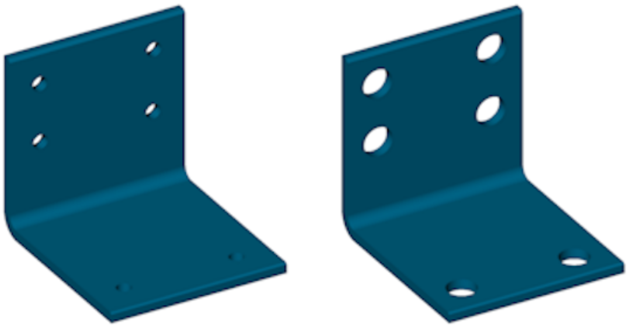
For a twist deformation, it is possible to specify the continuity between the deformed and the fixed portion of the entities.

Deformation can be defined in several ways:

- DMTWIST command: Twist the whole entity or a portion.
- DMDEFORMPOINT command: Move a point on a face in any direction in 3D space.
- DMDEFORMMOVE command: Move or rotate one or more edges.
- DMDEFORMCURVE command: Replace an edge with a given 3D curve.

7.5 Design intent recognition

Direct modeling operations allow you to change geometry of 3D solid models easily: when a single face of a solid is modified with a particular operation other faces are modified accordingly in order to keep design intent of the model and its topological and geometrical validity. For example, consider a model with a number of holes of the same radius. When the radius of a cylindrical face is modified with a push/pull operation or by changing the value of a radial constraint other faces that share the same diameter are recognized by BricsCAD and modified automatically to keep the radii synchronized.



Apart from the recognition of equal radii, BricsCAD automatically recognizes other geometrical relations between surfaces of a solid and preserves them during direct modeling operations.

To define the design intent recognition, click the buttons on the **Design Intent** toolbar. A pressed button indicates the feature is selected. An unpressed button indicates the feature is not active.

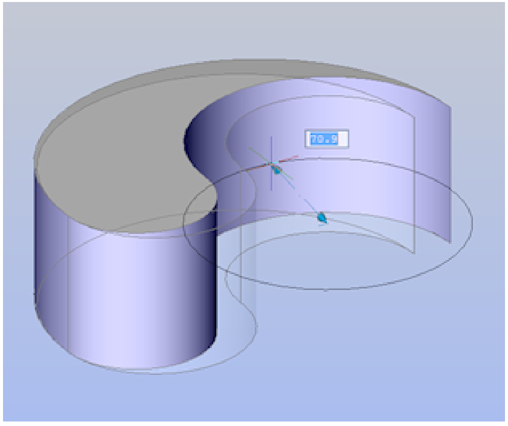
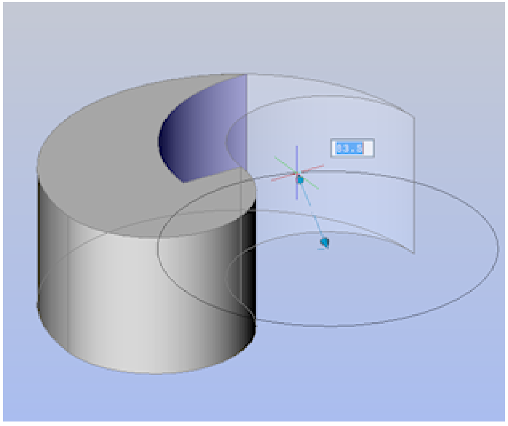
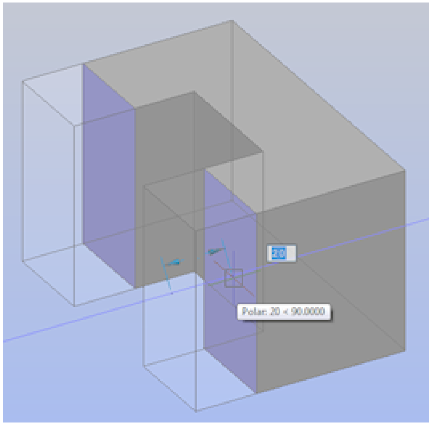
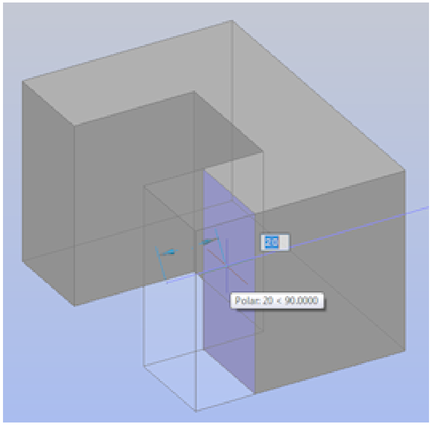
- tangent surfaces (planes, cylinders and cones) ()
- coincident planes ()
- parallel planes ()
- perpendicular planes ()
- cylinders perpendicular to planes ()
- coaxial surfaces (cylinders and cones) ()
- surfaces of equal radius (cylinders and spheres) ()
- vertices between 4 or more faces ()
- edges between coincident faces ()
- switch off/on all ()

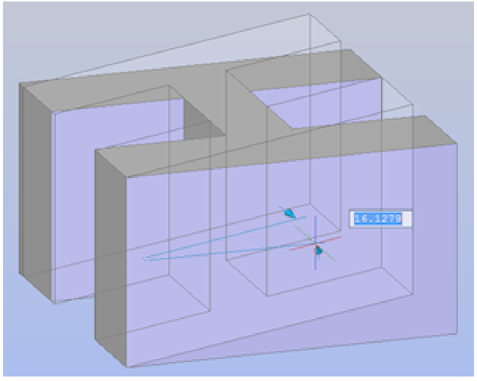
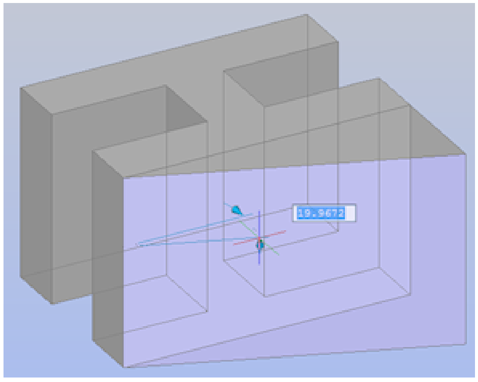
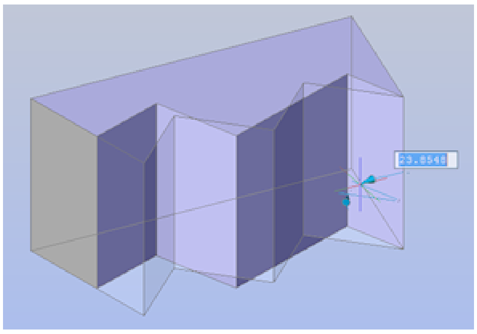
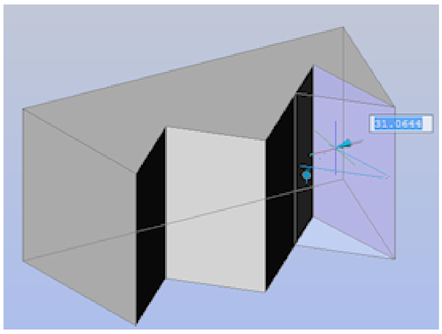
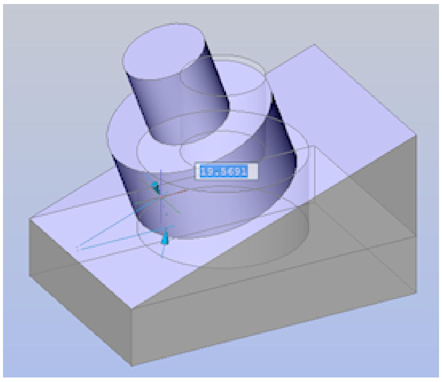
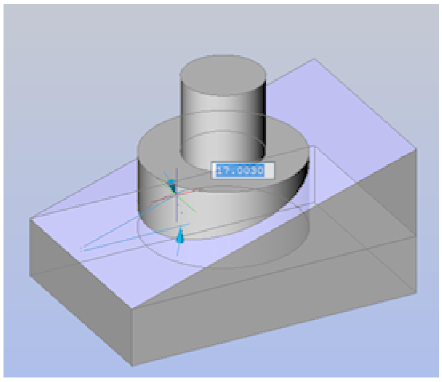
The design intent options can also be set by editing the DMRECOGNIZE system variable in the **Settings** dialog box.

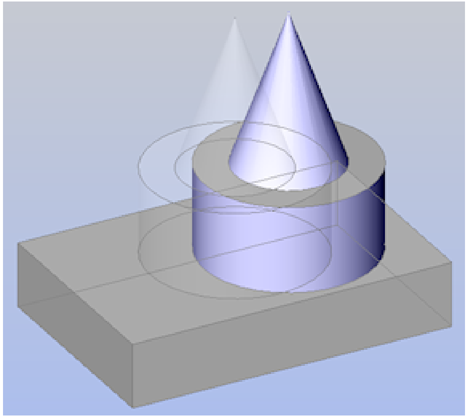
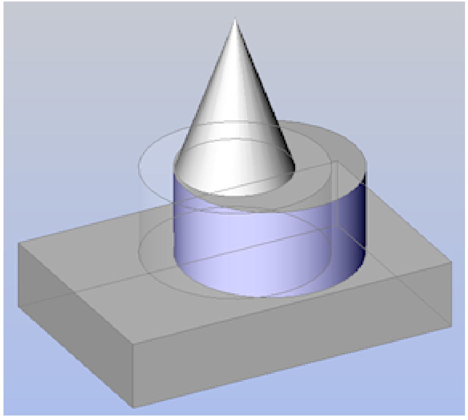
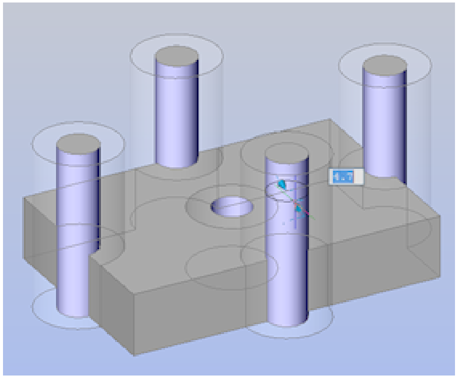
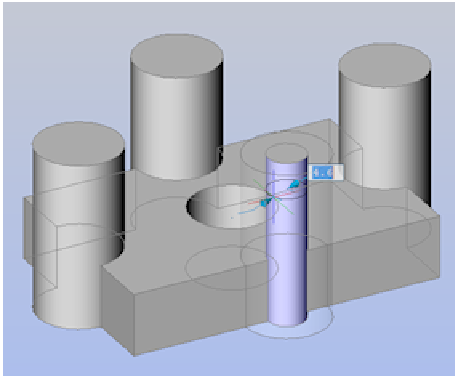
Click the **Options** tool button () on the **3D Constraints** toolbar to set the **Automatic 3D geometry constraints recognition**. The state of tool buttons on the **Design Intent** toolbar (pressed or unpressed) is adjusted accordingly.

Note: 3D Constraints which are explicitly defined by the user are taken into account when solid model is being modified with direct modeling commands. These constraints have a higher priority than automatically recognized ones and allow the user to better control the behavior of solid model modifications.

The following examples illustrate the difference in the behavior whether automatic 3D geometry constraints are recognized or not.

	
tangent surfaces recognized ()	tangent surfaces not recognized ()
	
coincident planes recognized ()	coincident planes not recognized ()

	
parallel planes recognized ()	parallel planes not recognized ()
	
perpendicular planes recognized ()	perpendicular planes not recognized ()
	
perpendicular cylinders recognized ()	perpendicular cylinders not recognized ()

	
coaxial surfaces recognized ()	coaxial surfaces not recognized ()
	
equal radius recognized ()	equal radius not recognized ()

7.6 Working with sections

The SECTION command creates section planes of 3D solids; the results are region entities.

The SECTIONPLANE command creates a section entity that creates sections of 3D solids, surfaces, and meshes.

The LIVESECTION command toggles the **Live Section** property of a section plane.

The SECTIONPLANESETTINGS command defines the properties of a section plane entity in the **Drawing Explorer - Section Planes** dialog box.

The SECTIONPLANETOBLOCK command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

Note: When using a BIM license level:

- The BIMSECTION command creates **BIM section** entities, which are Section entities with extended properties, such as type (plan, section, elevation) and initial scale of the 2D section result.
- The BIMIFY command can assign BIM properties to a Section entity.

7.6.1 Difference between Live Section and Clip Display

In both statuses all drawing content is sectioned and clipped-away parts are no longer displayed.

Live Section: entities in a drawing are temporarily modified, e.g. clipped solids are replaced by temporary solids with a different shape.

Clip Display: the display of entities is clipped. Entities are not replaced by temporary versions. **Clip Display** allows you to continue using all modeling and editing operations on all drawing content. As opposed to the **Live Section** status, the **Clip Display** status can be set for multiple section entities simultaneously.

7.6.2 To align a section plane to a face

1 Launch SECTIONPLANE command.

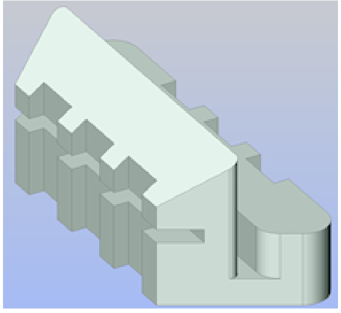
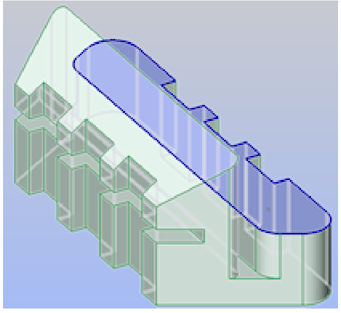
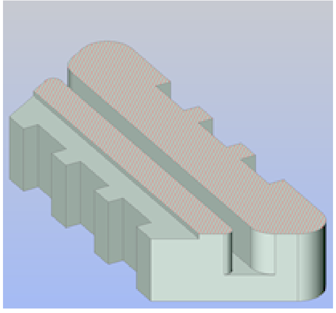
1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Select Face** option.

You are prompted: Select entity:

3 Select the face of a solid you want to align the section plane to.

4

		
Solid	Select face	Section plane aligned to face

5 The section plane is aligned to the selected face.

6 The **Clip Display** property of the section entity is switched **On** automatically.

7 (Optional) Edit the properties of the section plane in the **Properties** panel.

7.6.3 To create a vertical section plane

Note: The **WCS** must be the current coordinate system.

1 Launch SECTIONPLANE command.

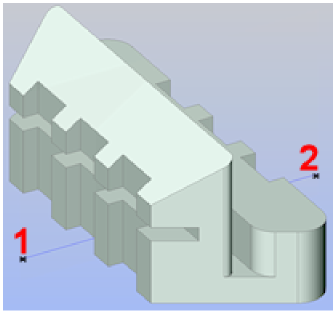
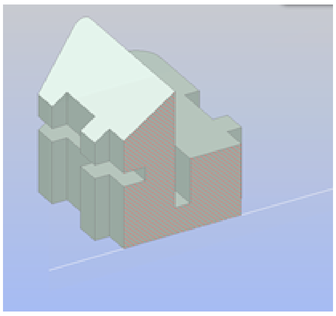
1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Specify two points.

3 The section plane is a vertical plane created through these points.

The part of the solid at the right hand side (with respect to the draw order of the points) of the section plane is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

	
Specify two points	Vertical section plane

4 (Optional) Edit the properties of the section plane in the **Properties** panel.

7.6.4 To create a horizontal section plane

1 Choose **Front** in the **Look From** widget.

2 Align the **UCS** to the front view:

Use the **View** option of the UCS command.

Note: If UCSORTHO = ON and NAVVCUBEORIENT = 0, the **UCS** aligns with the view automatically.

3 Launch SECTIONPLANE command.

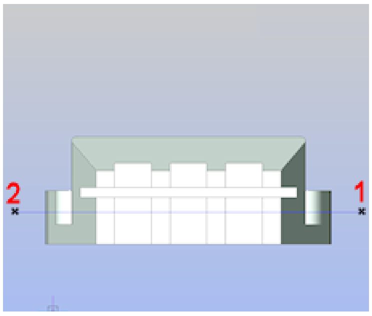
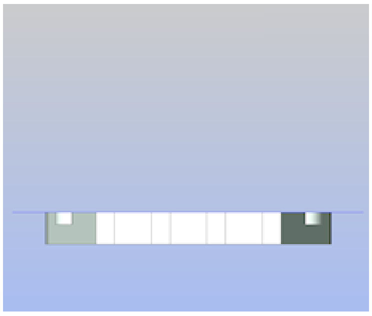
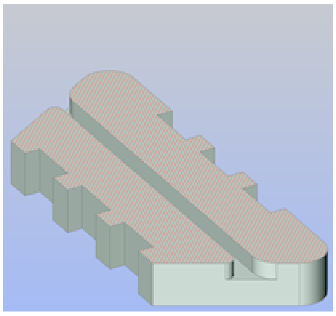
4 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

5 Specify two points.

6 The section plane is created through these points and perpendicular to the XY-plane of the current coordinate system.

7 The part of the solid at the right hand side (with respect to the draw order of the points) of the section plane is cut away.

8 The **Clip Display** property of the section entity is switched **On** automatically.

Specify two points	The part of the solid at the right hand side of the section plane is cut away	Horizontal section plane

9 (Optional) Edit the properties of the section plane.

7.6.5 To create a vertical jogged section plane

Note: The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Draw** option.

You are prompted: Specify start point:

3 Specify the first point (1).

4 **You are prompted**: Specify next point:

5 Specify the second point (2):

6 **You are prompted**: Specify next point or ENTER to finish:

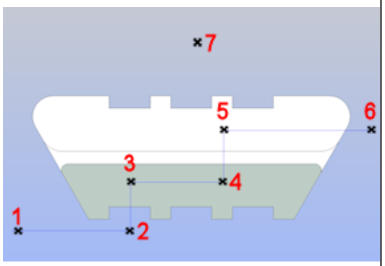
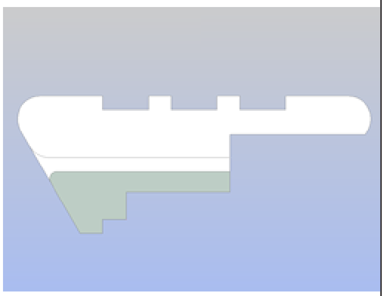
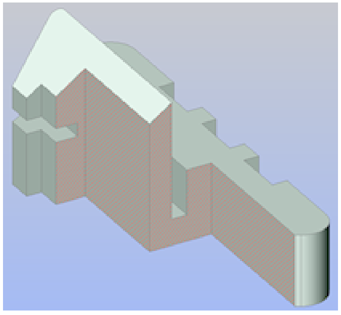
7 Specify the other points (3 - 6), then right click or press **Enter** to stop.

You are prompted: Specify the direction of the section entity:

8 Click a point (7) to specify the direction of the section entity.

9 The part of the solid opposite to the direction of the section entity is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

		
Specify points	The part of the solid opposite to the direction of the section entity is cut away	Vertical jogged section plane

10 (Optional) Edit the properties of the section plane in the **Properties** panel.

7.6.6 To create a horizontal jogged section plane

1 Choose **Front** in the **Look From** widget.

2 Align the **UCS** to the front view:

Use the **View** option of the UCS command.

Note: If UCSORTHO = ON and NAVVCUBEORIENT = 0, the **UCS** aligns with the view automatically.

3 Launch SECTIONPLANE command.

4 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

5 Choose the **Draw** option.

You are prompted: Specify start point:

6 Specify the first point (1).

7 **You are prompted**: Specify next point:

8 Specify the second point (2).

9 **You are prompted**: Specify next point or ENTER to finish:

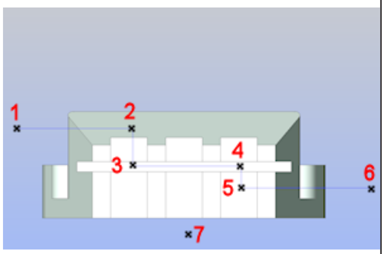
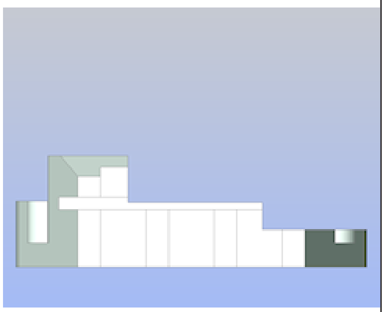
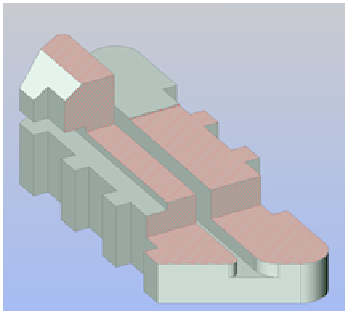
10 Specify the other points (3 - 6) , the right click or press **Enter** to stop.

You are prompted: Specify the direction of the section entity:

11 Click a point (7) to specify the direction of the section entity.

12 The part of the solid opposite to the direction of the section entity is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

		
Specify points	The part of the solid opposite to the direction of the section entity is cut away	Horizontal jogged section plane

13 (Optional) Edit the properties of the section plane in the **Properties** panel.

7.6.7 To create an orthographic section plane

Note: The **WCS** must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Orthographic** option.

You are prompted: Align section to [Front/bAck/Top/Bottom/Left/Right] <Top>:

3 Pick the orthographic section orientation of your choice in the prompt menu or type the corresponding option in the Command line.

4 Orthographic section planes are created through the center of the solid, parallel to the XY-plane (**Top** and **Bottom**), YZ-plane (**Left** and **Right**) or XZ-plane (**Front** and **Back**) of the current coordinate system.

The **Clip Display** property of the section entity is switched **On** automatically.

5 (Optional) Edit the properties of the section plane in the **Properties** panel.

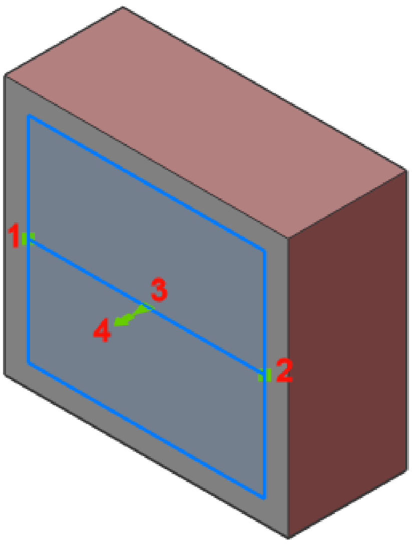
7.6.8 To modify the size and position of a section plane

1 Select the section plane entity in the drawing.

1 Depending on the current state of the section plane entity a number of grips display.

2 (Optional) Switch on the **Clip Display** property of the section plane in the **Properties** panel to see the result of the modifications dynamically.

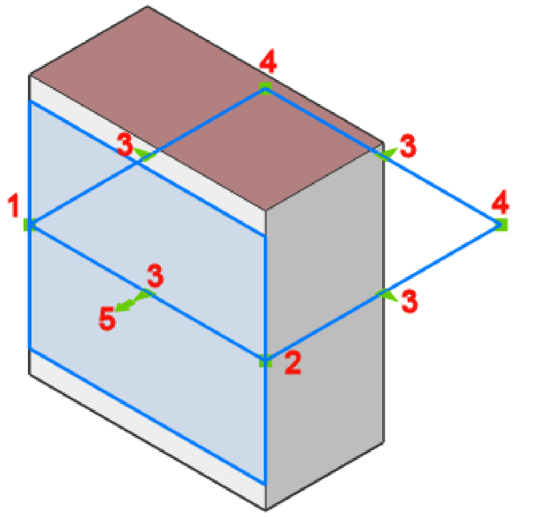
3 (Optional) Set the **State** property to **Plane**:



Four grips display:

- 1 () = origin point: moves the position of the section plane in the XY-plane.
- 2 () = through point: redefines the through point of the section plane.
  The section plane is rotated around the origin point (1).
- 3 () = midpoint: moves the section plane parallel in a direction perpendicular to the section plane.
- 4 (): click to flip the view direction of the section plane.

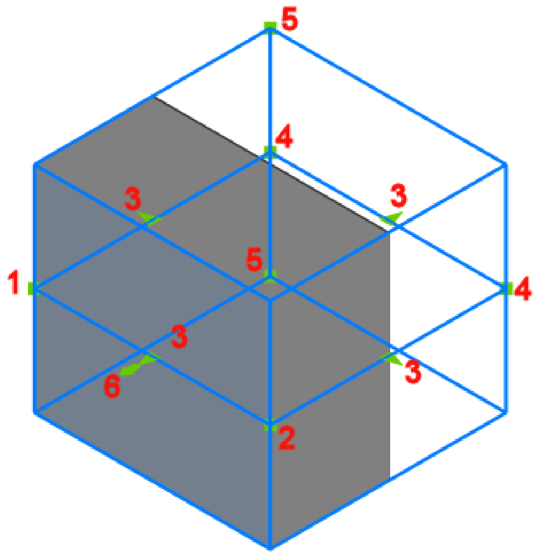
4 (Optional) Set the **State** property to **Boundary**:



Ten grips display.

- 1 (■) = origin point: moves the position of the section plane in the XY-plane.
- 2 (■) = through point: redefines the through point of the section plane.
  The section plane boundary is stretched and rotated around the origin point (1).
- 3 (▲): stretches the section plane boundary parallel.
- 4 (■): modifies the shape of the section plane boundary.
- 5 (⬆)

5 (Optional) Set the **State** property to **Volume**:



Twelve grips display.

- 1 (■) = origin point: moves the position of the section plane in the XY-plane.

- 2 () = through point: redefines the through point of the section plane.
  The section plane volume is stretched and rotated around the origin point (1).
- 3 (): stretches the section plane volume parallel.
- 4 (): modifies the shape of the section plane volume.
- 5 (): modifies the height of the section plane volume.
  Polar Tracking must be on to move grip 5.
- 6 (): click to flip the view direction of the section plane.

7.6.9 To save a section

The SECTIONPLANETOBLOCK command allows to:

- Insert a section in the drawing as a 2D section / elevation block or a 3D cutaway block.
- Replace a section block (e.g. when the section entity has been modified).
- Save a section to a new drawing.

7.7 Creating solids using the EXTRUDE command

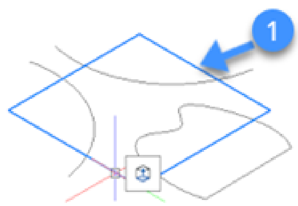
7.7.1 About

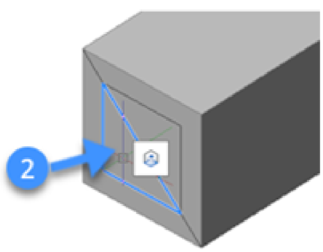
The EXTRUDE command allows you to create 3D solid(s) by extruding closed 2D entities, faces of 3D solid(s), regions or closed boundaries. The EXTRUDE command enables you to partially cut the solid model by removing the volume from the main solid and it can also create new solid(s) by adding volume to the main solid model.

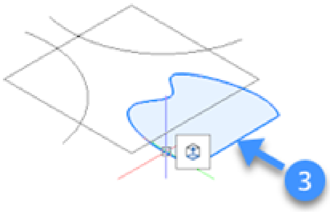
For more information about this command, visit the EXTRUDE Command reference article.

The following entities/subentities can be extruded:

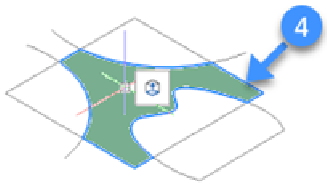
Closed 2D entities (1) (2) and regions (3):

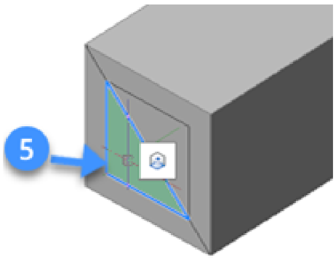






Closed boundaries in the XY-plane of the current coordinate system (4) or on the face of the solid (5)





When the **Enable Boundary Detection** option on the **Selection modes** toolbar is active, closed boundaries are automatically recognized.

Note: Only closed polylines, circles, ellipses and splines are accepted as a closed entity.

7.7.2 Using EXTRUDE in solid mode

These are the general steps to add and remove volume from the solid(s) using the **Extrude** tool:

1. Hover over an entity or sub-entity in the drawing area.
2. Select **Solid Extrude** in the **Model** tab from the Quad.
3. You are prompted:

   Specify height of extrusion or [Direction/Path/Taper angle/Auto/Subtract/Create/Unite/Both sides/ALign/set Limit] <123.45>:
4. The **Extrude Hotkey Assistant** appears at the bottom of the screen with a set of extrusion options.

   Note: The default mode is indicated by a blue frame.

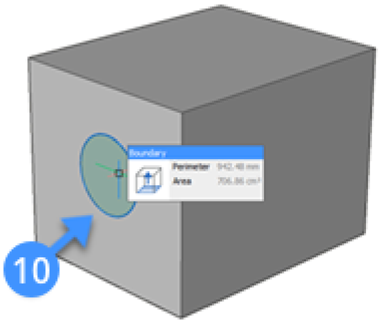
- **Auto** (6): the result of the Auto option depends on the extrusion direction of the 3D solid and the value of the EXTRUDEOUTSIDE, EXTRUDEINSIDE, and INTERSECTEDENTITIES system variables found in the **Extrude mode** group.

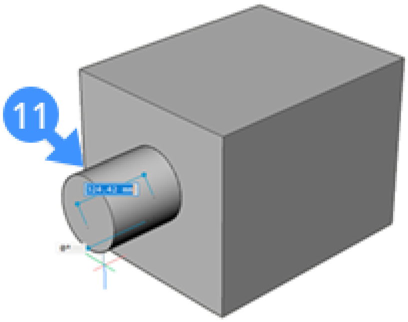
Note: The value of these variables could be 0 (= Do not modify), 1 (Subtract), and 2 (Unite).

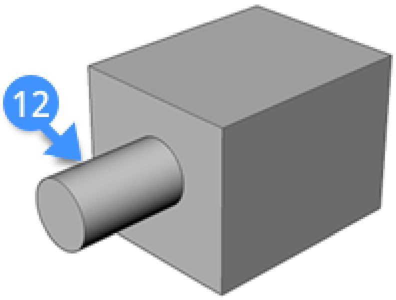
Note: The default value of these variables depends on the Workspace:

- **0** for Draft and Modeling.
- **1** for Mechanical and BIM.

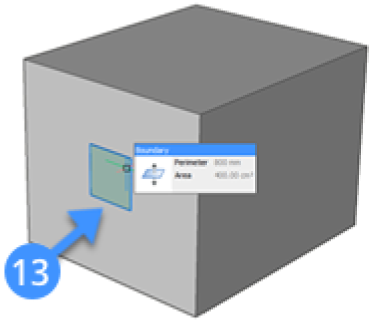
-

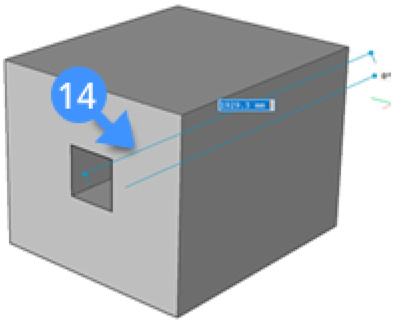


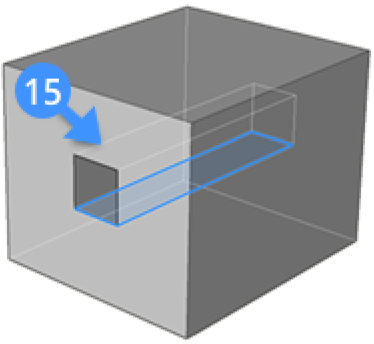




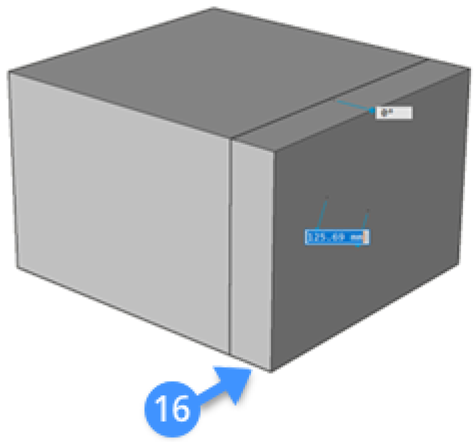
The following illustration shows the subtracted entity from the main solid for Mechanical and BIM workspace (the defaults values of all four Extrude Mode system variables = 1) When you highlight an entity (13) and extrude it inwards (14), a volume is subtracted (15) from the main solid:





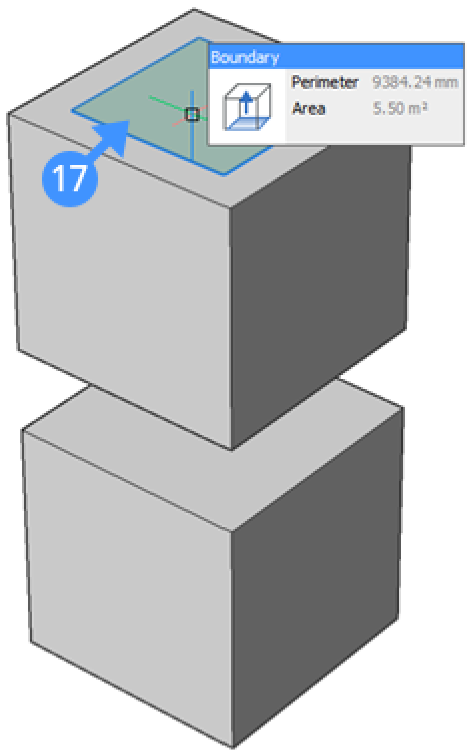


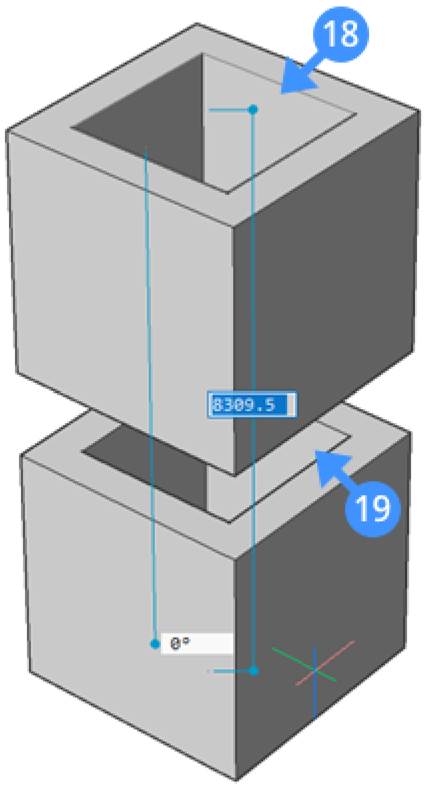
- **Create** (7): regardless of the extrusion direction, a new volume is created (16) from the existing solid:



- **Subtract** (8): the 3D solid is subtracted from each interfering existing solid.

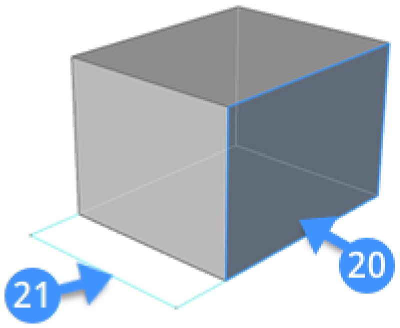
  The following illustration shows the subtracted solids (18,19) using the highlighted boundary (17):

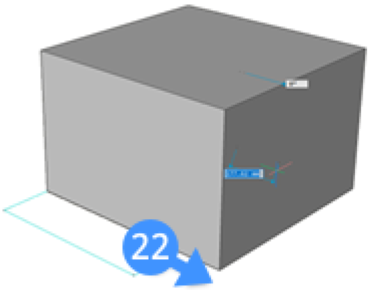




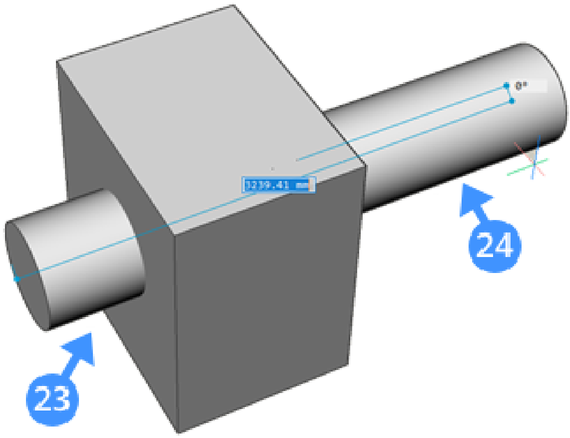
- **Unite** (9): the new volume is unified with each interfering existing solid.

  The following illustration shows the situation before and after when using the extrusion tool unite option. The solid surface is highlighted (20) and the distance between the left and right face is shown with the aligned line (21). The new volume is unified (22) with the first solid:





- **Both sides**: new symmetric volumes are created (23), (24) in both directions from the main solid:



To create symmetric volumes from the existing solid, type **B** in the Command line, which corresponds to **Both** sides option of the EXTRUDE command.

- Select one of the available options to execute the extrusion function on your solid.
- Next, move the cursor in the desired direction to specify the extrusion direction. The selected entity or sub-entity is extruded dynamically.
- Define the height of extrusion by specifying a point in the drawing, entering a value in the dynamic dimension field or choose the limit option.
- Press Enter or right-click to accept the extrusion from your existing solid.

7.7.3 EXTRUDE in Surface Mode

The EXTRUDE command allows you to create 3D surface(s) by extruding 2D entities such as lines or polylines.

For more information about this command, visit the EXTRUDE Command reference article.

7.7.4 Using the extrude tool in surface mode

These are the general steps to create 3D surface(s) using the EXTRUDE command in the Surface mode:

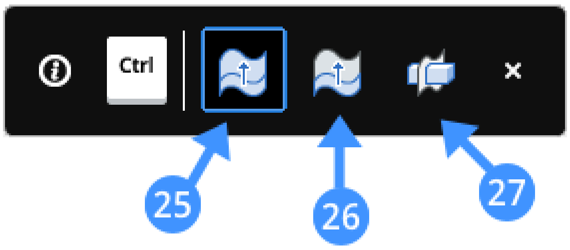
1 Move the cursor over an entity or sub-entity in the drawing area.

2 Select **Surface extrude** in the **Model** tab from the Quad.

3 You are prompted:

Specify height of extrusion or [Direction/Path/Taper angle/Auto/Create/Slice/Both sides/ALign] <123.45>:

4 The **Extrude Surface Hotkey Assistant** appears at the bottom of the screen with a set of surface extrusion options.



Note: The default mode is indicated by a blue frame.

- **Auto** (25): when multiple touching entities or subentities are selected, the result depends on the value of the UNITESURFACES system variable found in the **Extrude mode** group. When UNITESURFACES is ON, touching surfaces are united. When UNITESURFACES is OFF, each entity/subentity creates a separate surface.

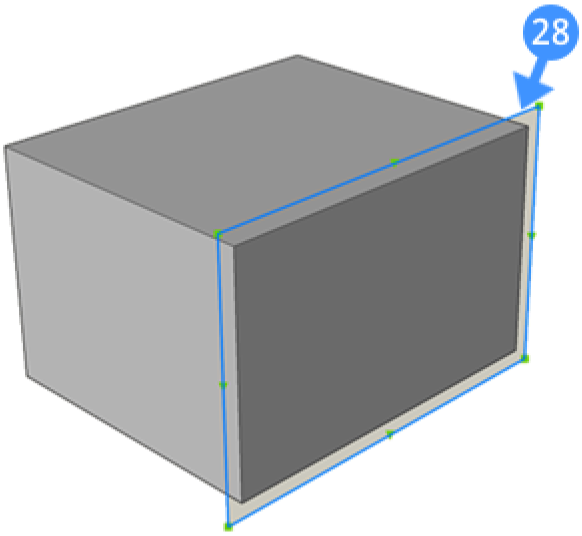
  Note: The default value of this variable depends on the Workspace:

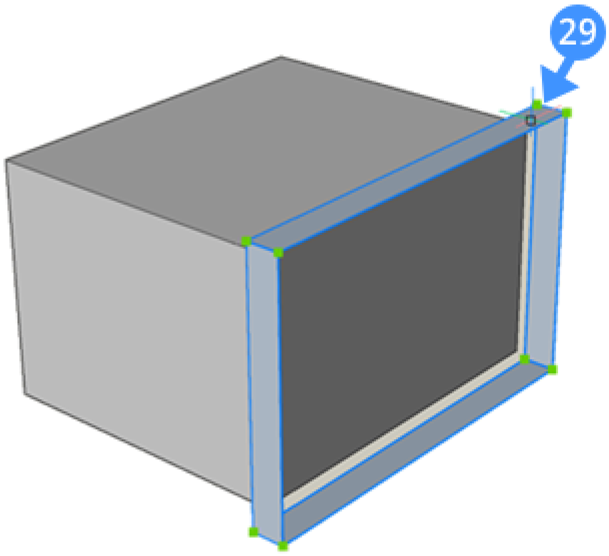
- **0** for Draft and Modeling
- **1** for Mechanical and BIM
- **Create** (26): create new surfaces.
- **Slice** (27): cut through solids with the extruded surface. This option is similar to the SLICE command.

5 Select one of the options and move the mouse to indicate the extrusion direction, entering a value in the dynamic dimension field.

6 Press Enter or right-click to accept the extrusion for the surface.

7 The following illustrations show the created surface (29) by extruding the selected 2D entity (28) in the drawing:





7.8 Sketch based features

Sketch Based Feature is a BricsCAD feature that allows you to edit a 2D entity used to create a 3D solid and automatically update the 3D solid.

The EXTRUDE, LOFT, SWEEP, and REVOLVE commands and their options **Subtract** and **Unite** create sketch features when the CREATESKETCHFEATURE system variable is ON. The sketch features are available in the **Mechanical Browser** panel.

Note: An automatic conversion algorithm runs when opening a drawing that contains old sketch features.

If something goes wrong, you must decide whether to continue the conversion.

- If yes, the algorithm will do its best to convert the features, guessing the geometry of arbitrary features. However, the converted Boolean features (UNITE or SUBTRACT) will not be properly renamed, even though the algorithm treated them as Booleans.
- If not, it dissolves the old sketch features and keeps only the 3D features.

Important: Sketch features created when the CREATESKETCHFEATURE system variable is ON are not compatible with other CAD programs. When you manually SAVE these drawings, a warning message appears in the Command line and in a Warning balloon at the bottom right side of the drawing: "*This drawing contains sketch features that are not compatible with other CAD programs. These features might break if you open the drawing using another CAD program!*".

7.8.1 Create a 3D solid

1 Turn ON the CREATESKETCHFEATURE system variable.

2 Draw a 2D entity, such as a closed polyline.

3 Optionally, add parametrics to the Sketch entity.

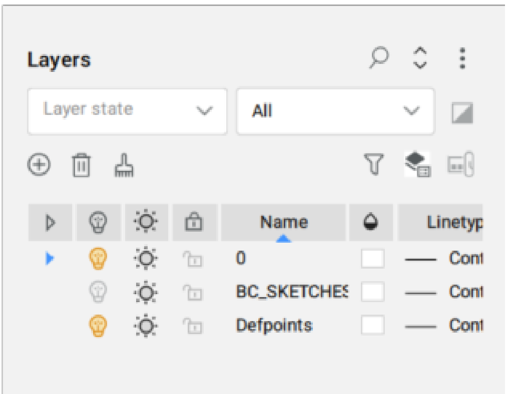
4 Launch the EXTRUDE, LOFT, SWEEP, or REVOLVE command to create the 3D Solid features, which are visible in the **Mechanical Browser** panel.

Sketch features can be copied/pasted into the same or a different drawing. The SKETCHFEATURECOPYMODE system variable will dictate the behavior of the copied sketch features.

When the SKETCHFEATURECOPYMODE system variable is ON, copies of sketch features will be independent of their source (new blocks of the sketches/paths/guide curves/etc. will be created). When it is OFF, copies of sketch features will share their sketches (paths, guide curves, etc.) with their source.

7.8.2 Edit the 3D solid by editing the corresponding sketch entity

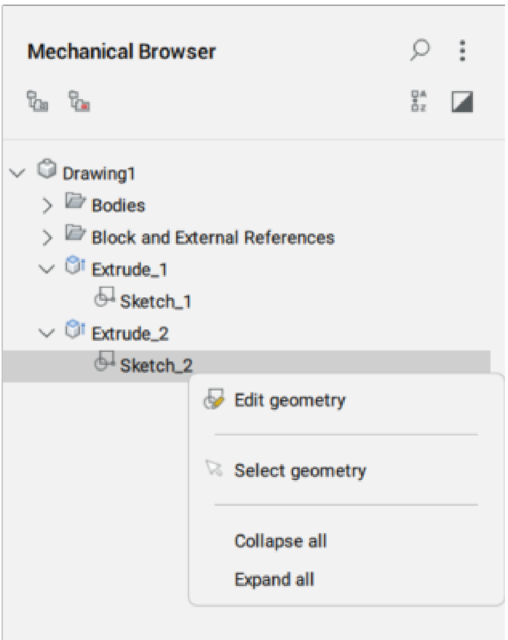
A Sketch feature is created from the 2D entity as a block and moved to a new layer: **BC_SKETCHES**. This layer is turned Off by default.



All Sketch features will be visible in the **Mechanical Browser** panel.

Method 1

To edit the Sketch feature, right-click the entity in the **Mechanical Browser** panel and select **Edit geometry**.



After save the changes for the Sketch feature (BCLOSE command), the 3D solid is automatically updated.

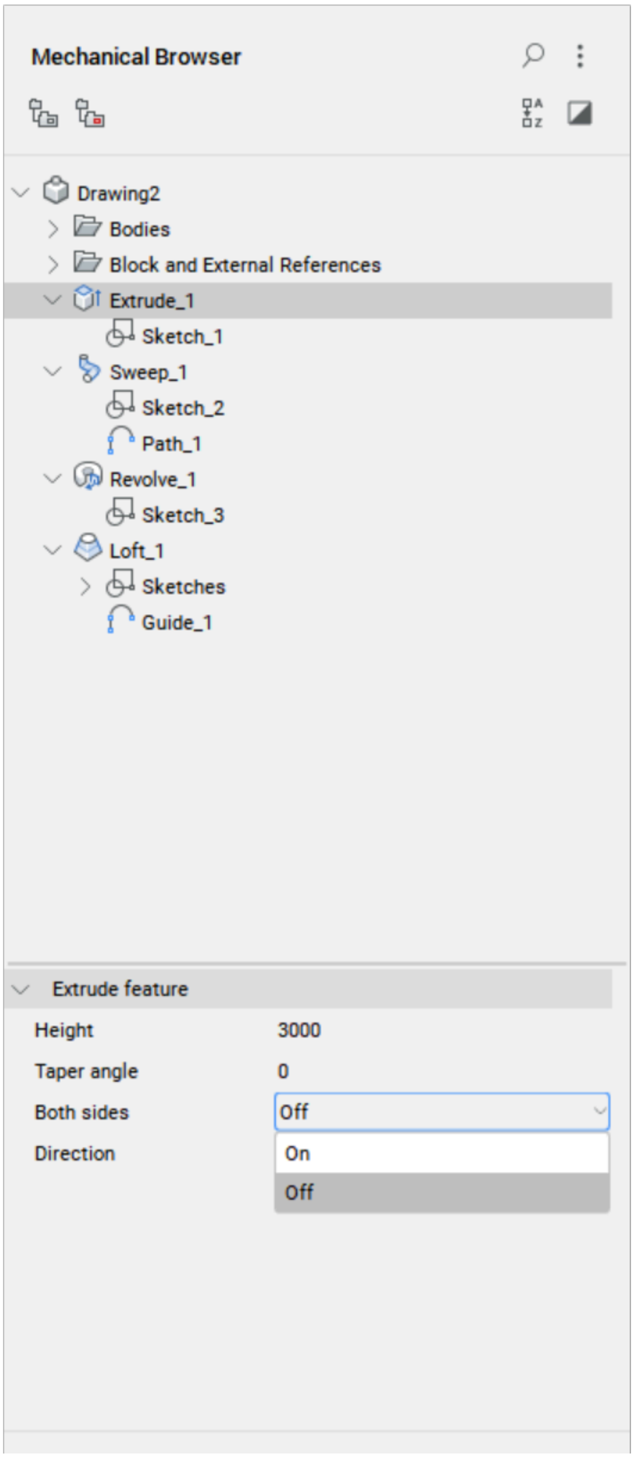
Method 2

To edit the Sketch feature you can also use the BEDIT command that opens the **Create or Edit Block Definition** dialog box from where you select the Sketch feature to open it in Block Editor.

To modify a Sketch Based Feature (3D solid), there are two methods:

Method 1

Click the Solid in the **Mechanical Browser** and adjust the properties in the bottom of the **Mechanical Browser** panel.



Method 2

Launch the UPDATESKETCH command and specify in the Command line the name of the feature, the name of the parameter, and its new value.

7.8.3 Removing all created sketch entities

Launch the REMOVESKETCHFEATURE or DISSOLVESKETCHFEATURE commands to remove only the sketch features. The REMOVESKETCHFEATURE command does an additional purge after the DISSOLVESKETCHFEATURE command.

Both commands keep the contents of the scene untouched.

Note: You can also dissolve the sketch features of a 3D solid by choosing the **Dissolve** option from its context menu in the **Mechanical Browser** panel.

7.8.4 Deleting all features

Launch the DELETESKETCHFEATURE command to delete the sketch features and their corresponding 3D solid.

Note: You can also delete a sketch based feature (3D Solid) by choosing the **Delete** option from its context menu in the **Mechanical Browser** panel.

7.9 Working with Work Planes (Experimental)

7.9.1 Disclaimer

EXP.

Work Plane is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

7.9.2 About

This first experimental version of Work Plane allows you to create XY, YZ, and XZ work planes, as well as work planes that coincide with a face and work planes offset from a face or another work plane with the WORKPLANECREATE command. Sketches (2D entities) can be linked or unlinked with a work plane using the WORKPLANELINK and WORKPLANEUNLINK commands. The WORKPLANESHOW and WORKPLANEHIDE commands control the visibility of the work planes in the Model Space. The Work Planes are visible in the **Mechanical Browser** panel. When the CREATESKETCHFEATURE system variable is ON, linked solids and Work Planes will update with parameter changes.

7.9.3 Create a Work Plane

1 Launch the WORKPLANECREATE command.

2 Choose one option from All/XY/YZ/XZ/Face/Offset.

3 The Work Plane(s) are visible in the Model Space and listed in the **Mechanical Browser** panel.

7.9.4 Activate a Work Plane

There are two ways to activate a Work Plane where you draw a sketch:

1. Launch the WORKPLANEACTIVATE command.
2. Open the context menu of a Work Plane in the **Mechanical Browser** panel and choose the **Create sketch** option.

Note: To activate another Work Plane, you must deactivate the previously activated one.

7.9.5 Deactivate a Work Plane

There are two ways to deactivate a Work Plane:

1. Launch the WORKPLANEACTIVATE command and choose **Deactivate** option.
2. Open the context menu of an activated Work Plane in the **Mechanical Browser** panel and choose the **Exit sketch** option.

7.9.6 The visibility of a Work Plane

There are two ways to make a Work Plane visible in Model Space:

1. Launch the WORKPLANESHOW command.
2. Open the context menu of a Work Plane in the **Mechanical Browser** panel and choose the **Show** option.

There are two ways to hide a Work Plane from the Model Space:

1. Launch the WORKPLANEHIDE command.
2. Open the context menu of a Work Plane in the **Mechanical Browser** panel and choose the **Hide** option.

7.9.7 Link or unlink sketches to a Work Plane

Launch the WORKPLANELINK or WORKPLANEUNLINK commands to link or unlink the sketches and solids with the Work Plane.

8. Civil tools

8.1 Civil commands

The Civil commands are listed in the table below:

Name	Title	Description
	Add Surface Labels	Adds spot elevation labels, slope labels or contour labels on a surface.
	Alignment by PI	Creates horizontal, vertical and 3D alignments.
	3D Alignment	Creates a 3D alignment from horizontal and vertical alignments.
	Alignment by Element - Curve	Creates an unconstrained, horizontal, curve alignment.
	Edit Alignment	Edits a horizontal or vertical alignment.
	Alignment by Element - Between Elements	Creates a horizontal alignment constrained on both sides by other elements.
	Alignment by Element - To/From Element	Creates a horizontal alignment, constrained on one side by another element.
	Alignment by Element - Line	Creates an unconstrained, horizontal, line alignment.
	Vertical Alignment	Creates a vertical alignment.
	Vertical Alignment View	Creates a vertical alignment view for a horizontal or 3D alignment.
	Import Civil 3D	Imports objects from a Civil 3D drawing.
	Export DWG	Exports civil objects as native DWG entities.
	Close the **Civil Explorer** panel.	Closes the **Civil Explorer** panel to hide it from the current workspace.
	Open the **Civil Explorer** panel.	Opens the **Civil Explorer** panel to display it in the current workspace.

Name	Title	Description
	Civil Points	Creates civil points (civil entities) represented by symbols and labels.
	Civil Point Attributes	Adds or removes user-defined attributes, to civil points.
	Civil Point Editor	Edits Civil points.
	Export Civil Points	Exports Civil points to an external file.
	Civil Points Groups	Creates civil point groups.
	Corridor	Creates a corridor from a corridor template.
	Edit Corridor	Adds or removes a corridor region.
	Extract from Corridor	Extracts 3D solids, meshes, TIN surfaces, 3D polylines or outer boundaries, from a corridor.
	Corridor Template	Creates a corridor template.
	Corridor Template Element	Creates a corridor template element from closed polylines.
	Edit Corridor Template Element	Adds or removes components to corridor template elements.
	GIS Export	Exports spatial and attribute data in an ESRI SHP file.
	GIS Import	Imports spatial and attribute data from an ESRI SHP file.
	Grading	Creates a graded surface on a TIN surface.
	Balance Grading Volumes	Balances grading cut and fill volumes.
	Grading Edit	Splits or merges gradings.
	Export to IFC	Exports the model to an IFC file.

Name	Title	Description
	Export LandXML	Exports Civil entities to a LandXML file.
	Import LandXML	Creates TIN surfaces or alignments from a LandXML file.
	Boundary Trim	Trims objects to a boundary: closed polylines, circles, closed splines or ellipses.
	String	Creates a Civil String entity.
	Add curves to a String	Adds curves to a Civil String.
	Add elevation points for a String	Adds elevation points for a Civil String.
	Add PI (Points of intersections) to a String	Adds PI to a Civil String.
	Delete elevation points from a String	Deletes elevation points from a Civil String.
	Delete PI from a String	Delete PI from a Civil String.
	String from object	Creates a Civil String entity from an existing object.
	TIN Surface	Creates a topographical (TIN) surface.
	Assign Image to TIN Surface	Assigns a raster image to a TIN surface, as a material.
	Edit TIN Surface	Adds or removes points, breaklines or boundaries on a TIN surface.
	Extract from TIN Surface	Extracts meshes, solids, points, faces, contours or borders from a TIN surface.
	Merge TIN Surfaces	Combines two or more TIN surfaces into a single TIN surface.
	Modify TIN Surface	Deforms or smooths a TIN surface.

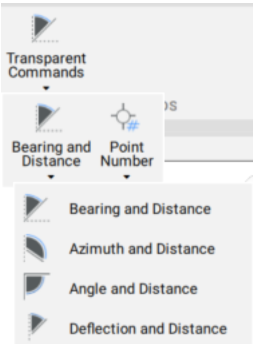
Name	Title	Description
	Project to TIN Surface	Projects points (points, blocks and text) or linear entities (lines, polylines and circles) to a TIN surface.
	TIN Volume Surface	Creates a TIN volume surface between a base and comparison TIN surface, or an elevation.
	Water Drop	Creates a water drop path on a TIN Surface.

8.2 Transparent Commands in BricsCAD

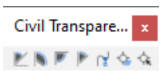
Use Civil Transparent Commands to enter a value based on known information (a point, distance, or radius).

You can run the transparent commands within another command (for example, PLINE, 3DPOLY, ALIGNMENT) by using the following methods:

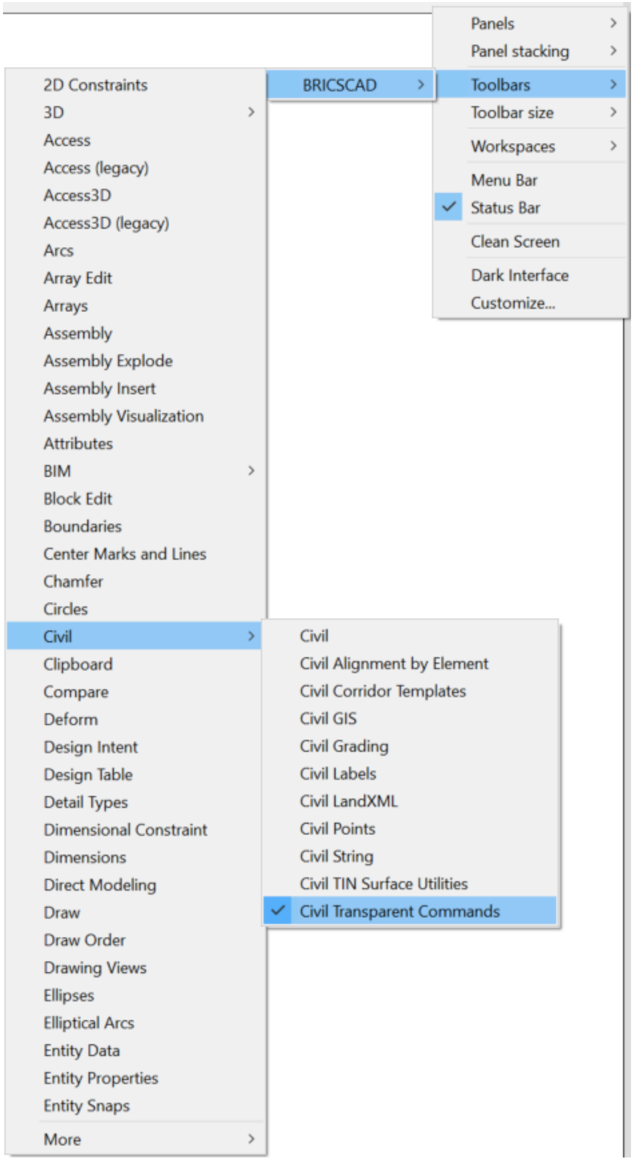
- Click the command in the **Civil Transparent Commands** ribbon panel in the **Home** tab of the Ribbon.



- Enter the command at the Command line.
- Activate the **Civil Transparent Commands** toolbar and select the command.



To activate the **Civil Transparent Commands** toolbar, right-click the ribbon, go to **Toolbars**, select **BRICSCAD**, and under **Civil**, select **Civil Transparent Commands**.



8.2.1 Bearing and distance

Alias: 'BD

Specifies a point location at a bearing and distance from a known point. The output is a coordinate of a new point.

Options within the command

Specify quadrant
Allows you to select the quadrant needed.
Quadrants: NE = 1, SE = 2, SW = 3, NW = 4

Specify bearing
Specifies the angle measured inside the selected quadrant. The measurement is very precise (degrees, minutes, seconds).

You can specify the angle dynamically on the drawing or by typing it in the Command line. (NE: 0, 90: SE: 90, 0: SW: 0, 90: NW: 90, 0 - As an example, for NE, 0 represents the N axis and the starting axis for measurement. The ending axis is represented by 90 degrees and in our example, that is the E axis.)

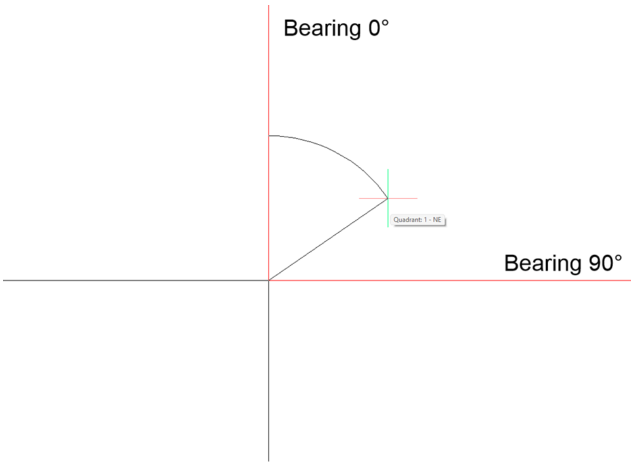
Or, you can enter the angle in decimal numbers and it will be converted to degrees, minutes and seconds (the first two decimal digits represent minutes and the second two represent seconds).

Note: The valid values are:

- for degrees: 0-90
- for minutes: 0-59
- for seconds: 0-59

Specify distance

Specifies the offset from the origin of the quadrant at the previously specified angle. You can specify the distance dynamically on the drawing or by typing it in the Command line.



Example: Run the PLINE command:

```
: PLINE
Select start of polyline or [Follow] <Last point>:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width/Undo]: 'BD

>>Specify quadrant [1 for North East, 2 for South East, 3 for South West, 4 for
 North West]: 1
>>Specify bearing: 40d30'20"
>>Specify distance: 123
Resuming PLINE command
```

Note: For larger scales, a new coordinate system is created to reduce errors.

8.2.2 Azimuth and distance

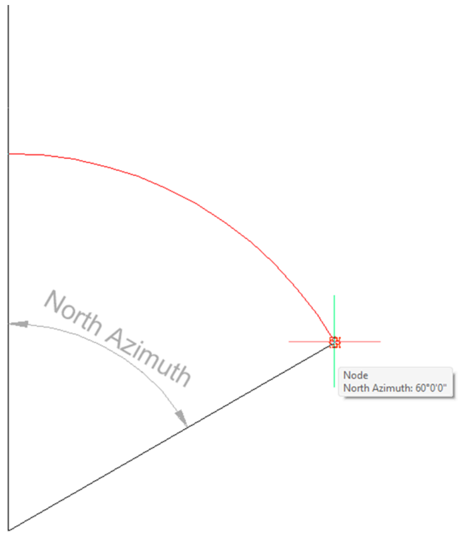
Alias: 'ZD

Specifies a point location at an azimuth and distance from a known point.

Options within the command

Specify azimuth
Allows you to select an angle relative to the line previously created by the command. This can be done dynamically in the drawing or by typing the exact angle in the Command line (degrees, minutes, seconds, or in decimal numbers).

Specify distance
Allows you to specify the offset from the starting point. This can be done dynamically or by typing it in the Command line.



Example: Run the PLINE command:

```
: pl
Select start of polyline or [Follow] <Last point>: 0,0
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]: '_ZD
Specify azimuth: 60
Specify distance: 50
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]:
```

8.2.3 Station and offset

Alias: '_SO

Specifies a point location at a station and an offset from an alignment.

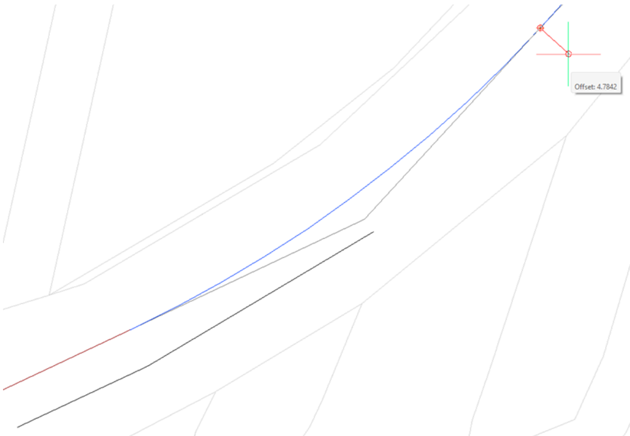
Note: This option is used when you are working with Horizontal Alignment elements.

Options within the command

Select alignment
Allows you to select the Horizontal Alignment needed to be used.

Specify station
Allows you to select any point on the alignment as starting point.

Specify offset
Creates a supporting line, perpendicular on the station. This allows you to specify the offset on the supporting line. You can select the distance graphically or by typing it in the Command line.



Example: Run the PLINE command:

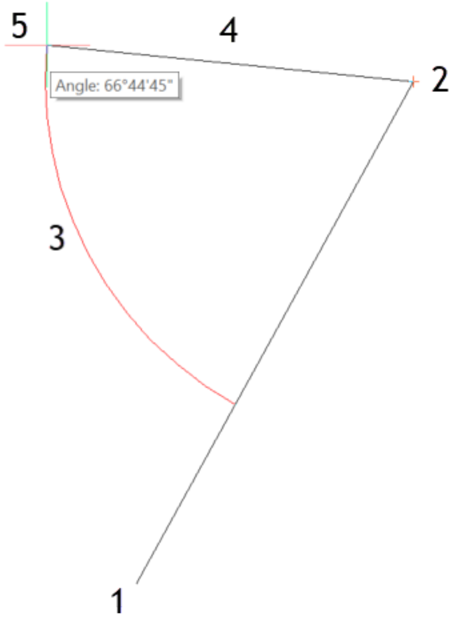
```
: PLINE
Select start of polyline or [Follow] <Last point>:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width/Undo]: '_SO


>>Select alignment:
>>Specify station:
>>Specify offset:
Resuming PLINE command
```

8.2.4 Angle and Distance

Alias: 'ANGD

Specifies a point location (5) at an angle (3) and a distance (4) from a known point (2) and direction (1-2).



Example: Run the PLINE command:

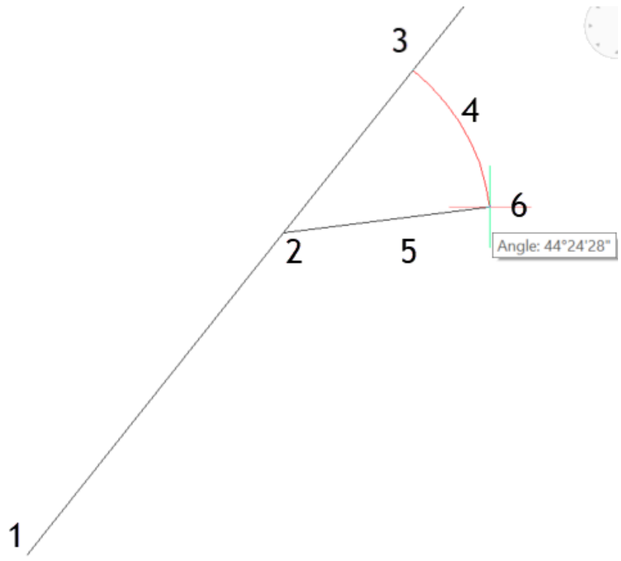
```
: PLINE
Select start of polyline or [Follow] <Last point>:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width/Undo]: 'ANGD
Specify angle or [Clockwise]: 70
Specify distance: 200

Resuming PLINE command
```

8.2.5 Deflection and Distance

Alias: ‘DD

Specifies a point location (6) at an angle (4) and distance (5) from a known point (2) and previous direction (1-3).




```
: PLINE
Select start of polyline or [Follow] <Last point>:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]:
Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width/Undo]: 'DD
Specify angle or [Clockwise]: 70
Specify distance: 200

Resuming PLINE command
```

8.2.6 Point number

Alias: '_PN

Specifies a point location using a point number. You can enter multiple point numbers or number ranges separated by commas.

Note: This command is used when you are working with Civil Points.

Example: Run the 3DPOLY command:

```
: 3DPOLY
Start of polyline: '_PN
>>Enter point number: 1-4
Resuming 3DPOLY command
```

8.2.7 Point object (selected in drawing graphically)

Alias: '_PO

Specifies a point location by picking an existing Civil Point.

Example: Run the 3DPOLY command:

```
: 3DPOLY
Start of polyline: '_PO
>>Select point object:
Resuming 3DPOLY command
>>Select point object:
Resuming 3DPOLY command
```

```
>>Select point object:
Resuming 3DPOLY command
```

8.3 TIN surfaces

8.3.1 Create a TIN Surface

8.3.1.1 Commands

TIN, TINASSIGNIMAGE, TINPROJECT

8.3.1.2 Overview

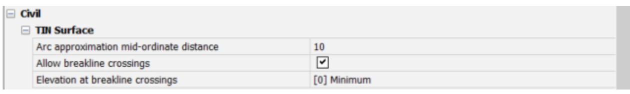
In BricsCAD, TIN (Triangulated Irregular Network) Surface is created using the TIN command.

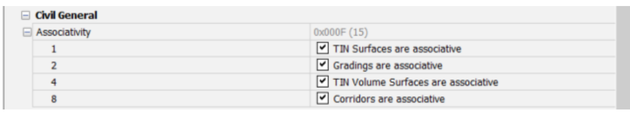
A TIN surface uses two types of entry data: points and breaklines (polylines).

- Points contain X, Y coordinates and Z values. All points are used to make a connection with the two nearest neighbors to create triangles. The surface triangulation is based on the Delaunay algorithm, which ensures that no point is inside the circumcircle of any triangle.
- Breaklines represent linear infrastructure features like curbs, retaining walls, etc. These lines also define the edges of triangles. Breaklines can be created from linear entities such as line, polyline, arc or circle.
- Boundaries are the features to define bounded 3D surfaces. The surface boundaries can be created with closed polylines. Defining outer boundaries on a large 3D surface improves the performance of a TIN Surface plan.

Creating a TIN Surface is a process that needs to be followed to map a particular area in detail. On BIM models, these surfaces are used to represent the site. The TIN surface can be generated from large data sets such us Point Clouds, from drawing entities, from 3D Faces, from custom ASCII point files, from Civil 3D surfaces.

8.3.1.3 Setting the TIN Surface preferences





Arc approximation mid-ordinate distance

> ***Note:*** Mid-ordinate distance is the maximal distance between arc and chord (straight) segment which is used for arc approximation.

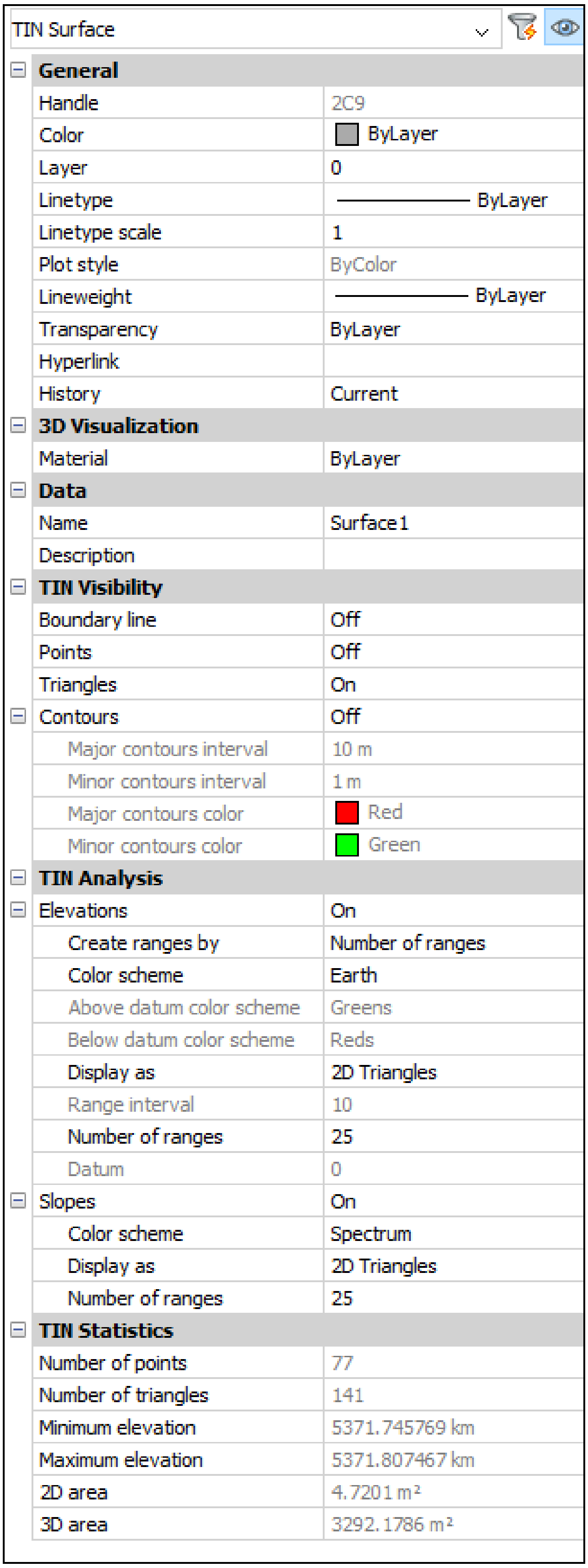
Defines how many points along the curve you want to add. When you have polylines with curves, this parameter allows you to tessellate the arcs in the polyline. The default value of this parameter is variable in BricsCAD predefined templates. For instance, the distance value is set to 0.01 in BIM-m, 1 in BIM-cm, 10 inBIM-mm, and 0.4 in BIM imperial templates.

Allow breakline crossings
Defines if crossings breaklines are allowed. By default, it is set to **On**. If breakline crossings are allowed, intersections between breakline segments are calculated and added as points to the TIN Surface.

Elevation at breakline crossings
Defines how elevation at crossing breaklines should be determined. There are three options (e.g., minimal, maximal, and average) for defining breaklines elevation at the intersection.

Associativity
Enable/disable grading, surface and corridor associativities.
By default, the system variable is set to 15.
0: Neither TIN surface, grading nor corridor associativity is enabled.
1: Enables TIN surface associativity.
2: Enables grading associativity.
3: Both TIN surface and grading associativities are enabled.
4: Enables TIN volume surface associativity.
5: Both TIN surface and TIN volume surface associativities are enabled.
6: Both grading and TIN volume surface associativities are enabled.
7: TIN Surface, Grading and TIN volume surface associativities are enabled.
8: Enables corridor associativity.
9: Both TIN surface and corridor associativities are enabled.
10: Both grading and corridor associativities are enabled.
11: TIN surface, grading and corridor associativities are enabled.
12: Both TIN volume surface and corridor associativities are enabled.
13: TIN surface, TIN volume surface and corridor associativities are enabled.
14: Grading, TIN volume surface and corridor associativities are enabled.
15: All TIN surface, grading, TIN volume surface and corridor associativities are enabled.

8.3.1.4 Properties of a TIN Surface


TIN Surface
General
Handle 2C9
Color ByLayer
Layer 0
Linetype ByLayer
Linetype scale 1
Plot style ByColor
Lineweight ByLayer
Transparency ByLayer
Hyperlink
History Current
3D Visualization
Material ByLayer
Data
Name Surface1
Description
TIN Visibility
Boundary line Off
Points Off
Triangles On
Contours Off
Major contours interval 10 m
Minor contours interval 1 m
Major contours color Red
Minor contours color Green
TIN Analysis
Elevations On
Create ranges by Number of ranges
Color scheme Earth
Above datum color scheme Greens
Below datum color scheme Reds
Display as 2D Triangles
Range interval 10
Number of ranges 25
Datum 0
Slopes On
Color scheme Spectrum
Display as 2D Triangles
Number of ranges 25
TIN Statistics
Number of points 77
Number of triangles 141
Minimum elevation 5371.745769 km
Maximum elevation 5371.807467 km
2D area 4.7201 m²
3D area 3292.1786 m²

TIN Visibility
Defines the visibility of the TIN surface elements.

Boundary line
Changes visibility of the outer boundary on a TIN Surface.

Points
Changes visibility of points on a TIN Surface.

Triangles
Changes visibility of triangles on a TIN Surface.

Contours
Defines the visibility of contours on a TIN Surface.

Major contours interval
Sets the major contours interval.

Minor contours interval
Sets the minor contours interval.

Major contours color
Defines the color of the major contour.

Minor contours color
Defines the color of the minor contour.

TIN Analysis
Defines the parameters for displaying elevations and slopes analyses.

Elevations
If ON, this option defines the parameters for displaying elevations analyses.

Create ranges by
Sets the range type (Number of ranges, Range interval, Range interval with datum).

Color scheme
Sets a color scheme for the elevation analysis.

Above datum color scheme
Sets the above datum color scheme.

Below datum color scheme
Sets the below datum color scheme.

Display as
Sets the type of entities that will be used for displaying elevation analysis (Triangles, Contours, Points, 2D Triangles, 2D Contours)

Range interval
Sets the range interval.

Number of ranges
Sets the number of ranges.

Datum
Sets the datum.

Slopes
If ON, this option defines the parameters for displaying slopes analyses.

Color scheme
Sets the color scheme for slopes.

Display as
Sets the type of entities that will be used for displaying slopes analysis (Triangles, 2D Triangles).

Number of ranges
Sets the number of ranges.

TIN Statistics
Displays the TIN Statistics.

The number of points
Displays the number of TIN Surface points.

The number of triangles
Displays the number of TIN Surface triangles.

Minimum elevation
Displays the minimum elevation value found in the TIN surface.

Maximum elevation
Displays the maximum elevation value found in the TIN surface.

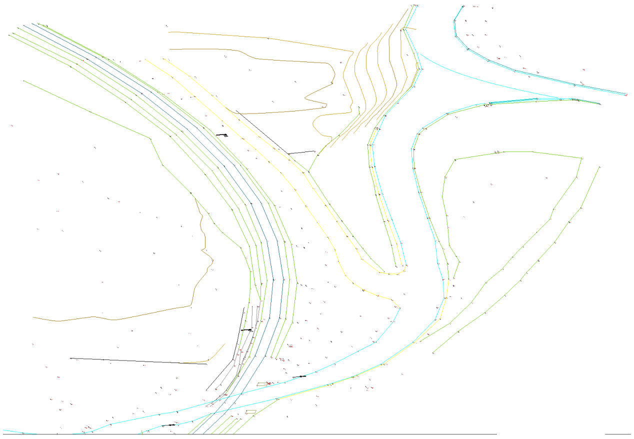
2D area
Displays the 2D area of the TIN surface.

3D area
Displays the 3D area of the TIN surface.

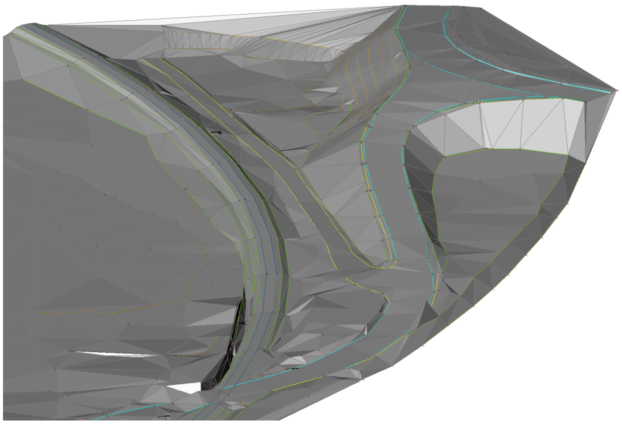
8.3.1.5 Procedure: Creating a TIN Surface from drawing entities

1 Open the drawing that contains drawing entities, such as CAD points, blocks and polylines.

1



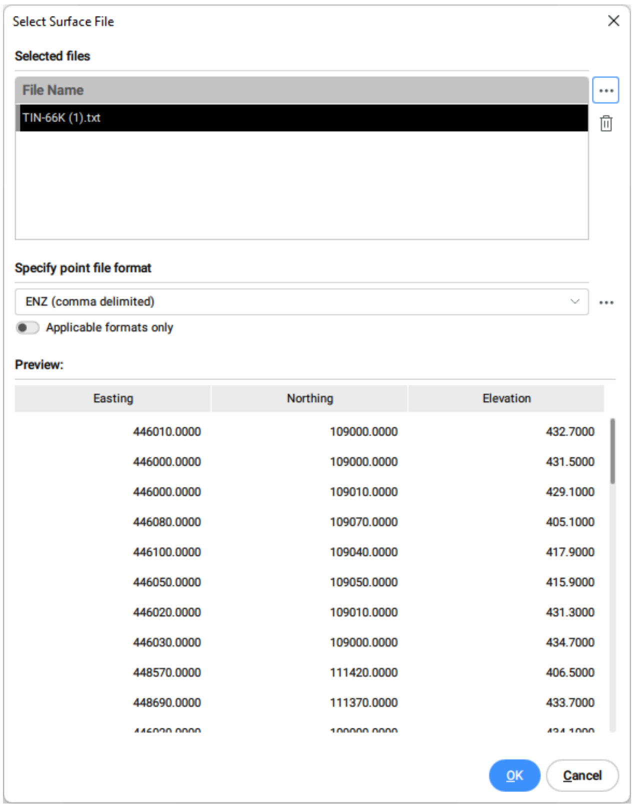
2 Launch the TIN command.
3 Select all entities in the drawing.
4 Press ENTER
5 Select an option in command line to apply selected linear entities as Breaklines.



Note: TIN Surface maintains a live connection to linear drawing entities when they are added as Breaklines. When editing such drawing entities, the TIN surface is automatically rebuilt. If linear drawing entities are added to a TIN Surface as edges or points, this connection is not maintained. The same applies to CAD blocks and points.

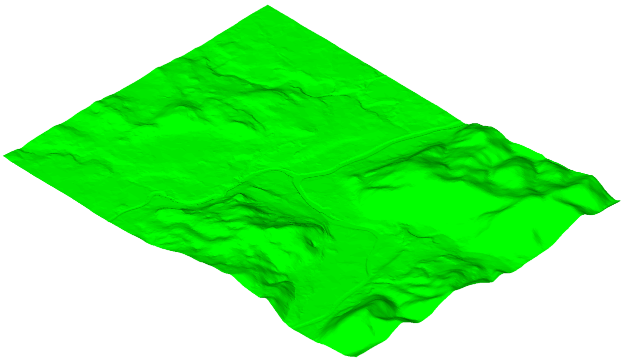
8.3.1.6 Procedure: Creating a TIN Surface from a point file

1 Launch the TIN command.
2 Select **Import from file** in the Command line.
3 In the **Create Surface From Point File** dialog box, select the point data file that you want to open, or choose multiple files to import all at once.
4 Specify point file format from the drop-down list.
5 ***Note:*** Click the browse button in the **Specify point file format** section to manage surface file formats.
6 Toggle the **Applicable formats only** to automatically filter appropriate point file formats, according to the selected input point file(s).
7 Click **OK** to create TIN Surface from a point file.



8 Select **No** in the Command line to create TIN Surface without simplifying points.

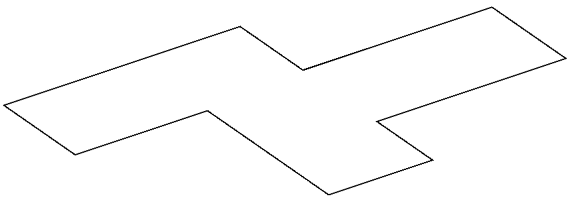
Note: Check the description of the TIN Simplify algorithm for detailed explanation of the parameters.



8.3.1.7 Procedure: Creating a TIN Surface from a point file using Clip polygon

1 Use existing or create a new closed polyline to create a TIN Surface inside that area.

1



2 Launch the TIN command.

3 Select the **cLip polygon** option in the Command line.

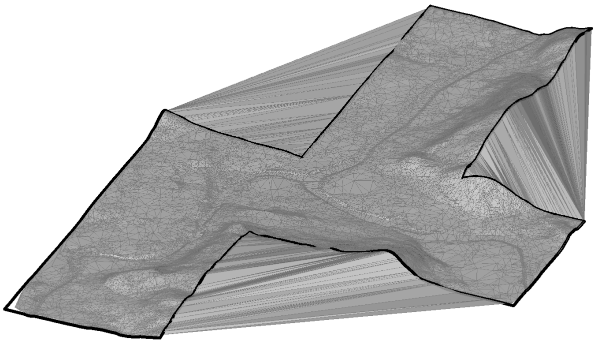
4 Select a closed polyline in a drawing to be used as a clipping polygon.

5 Select the **Import from file** option in the Command line to create TIN surface from a point file inside the selected clipping polygon.

Note: Select any other option: **select (drawing) entities**, **create from Faces**, **create from Point Cloud**, **create from Point Groups**.

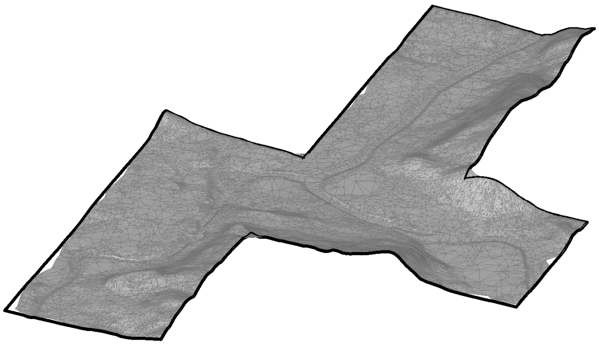
6 Follow the procedure for creating a TIN Surface from a point file, which is described in this article.

7



Note: The TIN surface algorithm connects triangles outside the boundary of the concave clipping polygon. In such areas, long and narrow triangles are created, which can be removed with the TINEDIT command.

8 Launch the TINEDIT command and select the **Remove Outer Edges** option. Then specify the maximum triangle length. Triangles that are longer than specified value are removed.



8.3.1.8 Procedure: Creating a TIN Surface by placing points

1 Open a drawing.

2 Launch the TIN command.

3 Select the **place Points** option in the Command line.

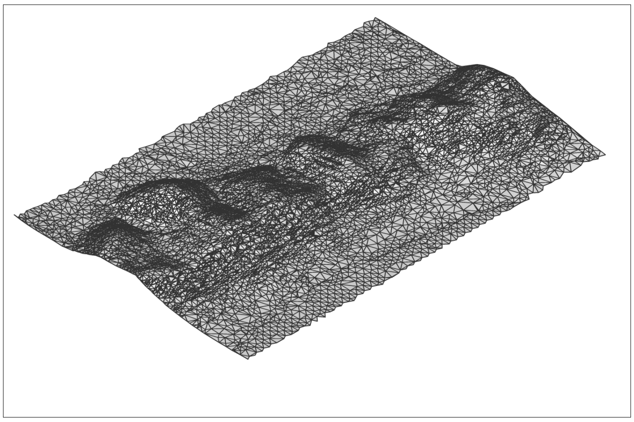
4 Pick a point where you want to start creating a TIN surface.

5 Specify the elevation value of each of those points, then press ENTER to accept.

Note: At least three points are needed to create a TIN surface.

8.3.1.9 Procedure: Creating a TIN Surface from Faces

1 Open a drawing file that contains 3D faces.

1 

2 Launch the TIN command.

3 Select the **create from Faces** option in the Command line.

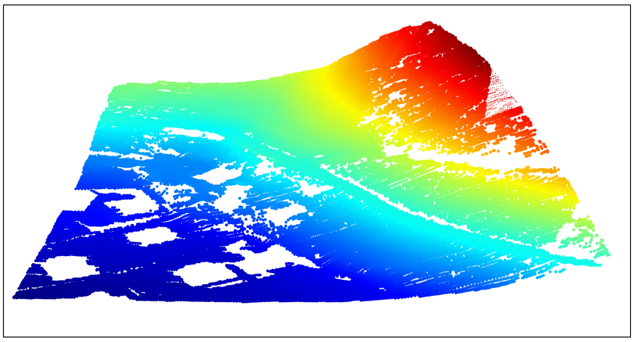
4 Select desired 3D faces.

5 Select an option whether to apply 3D Faces as edges (and visibility) or not.

6 ***Note:*** If this option is selected, then edges and vertices of the 3D Faces are used to create the TIN surface. Otherwise, only the vertices of the 3D faces are used to create the TIN surface. As a result, the TIN Surface does not follow the edges of the 3D Faces.

8.3.1.10 Procedure: Creating a TIN Surface from a Point Cloud

1 Open the drawing file that contains a point cloud.

1 

2 Launch the TIN command.

3 Select the **create from point Cloud** option in the Command line.

4 Select a point cloud in the drawing.

5 Press ENTER.

6 Select **No** to create TIN Surface without using the simplification method.

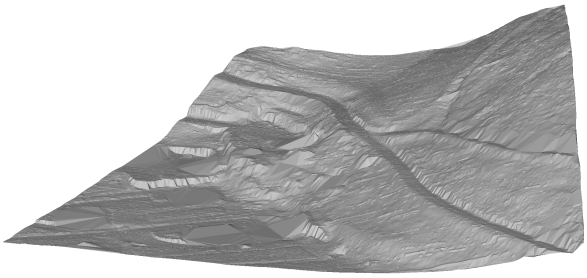


Image shows original TIN Surface created from a Point Cloud with 87350 points.

8.3.1.11 Procedure: Use TIN Simplify and TIN Densify options when creating TIN Surface from a Point Cloud

This section describes the procedure of creating a TIN Surface from a Point Cloud, where we also use the TIN Simplify and TIN Densify options.

Follow steps 1 to 5 from the procedure described above to create a TIN surface from a Point Cloud. Then, follow the procedure below to use TIN Simplify and TIN Densify options.

1. Select **Yes** when command line offers the option to simplify TIN surface.
2. Enter radius for simplification step.
3. Enter Elevation difference.
4. ***Note:*** Check the description of the TIN Simplify algorithm for detailed explanation of the parameters.
5. Select the **Create tin surface** option in the Command line.

   Or select the **Simplify again** option to edit the simplification parameters.

   Note: The Command line reports the number of points before and after simplification so that the user can make a better decision for the next steps.

```
Number of points before simplification: 78350
Number of removed points: 76788
Number of points after simplification: 1562
```

6. Select **Yes** to Densify TIN Surface.

   Or select **No** to create simplified TIN Surface.

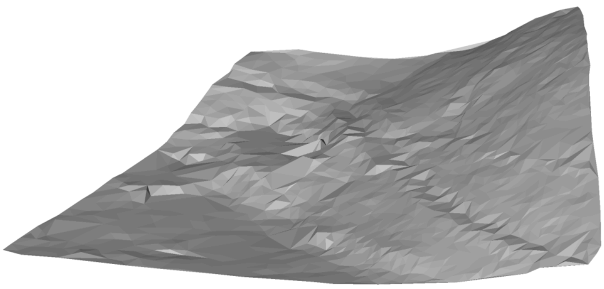


Image Shows simplified TIN Surface with 1562 points.

7 Enter elevation difference.

8 Select one of the options to specify the area for TIN Densify process: select existing polygons in a drawing, draw a new polygon or press ENTER to use entire TIN Surface.

9 ***Note:*** Check the description of the TIN Densify algorithm for detailed explanation of the parameters.

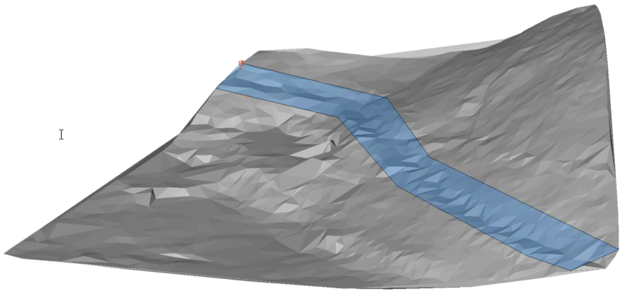


Image shows a polygon that defines an area for TIN densification.

Note: The final result represents a simplified TIN surface to which individual points within the specified polygon are returned. By using TIN Densify option on a specific area on the entire TIN Surface, we achieve the goal that in this area the elevation difference between the original and the final TIN surface is never greater than the specified value of the "elevation difference" in step 6.

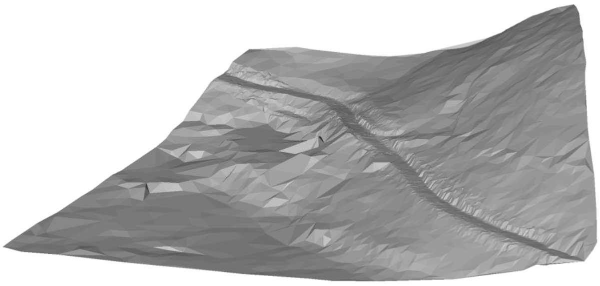


Image shows a final TIN Surface with 6536 points. Original TIN Surface consists of 87350 points.

Note: You can edit the TIN Simplify and TIN Densify parameters at any time in Civil Explorer. Select the appropriate TIN Surface, right-click on the **Add points** definition and select the **Edit** option from the context menu. The **Surface definition** dialog box opens in which you can edit all the necessary parameters.

8.3.1.12 Procedure: Creating a TIN Surface from a Civil 3D surface

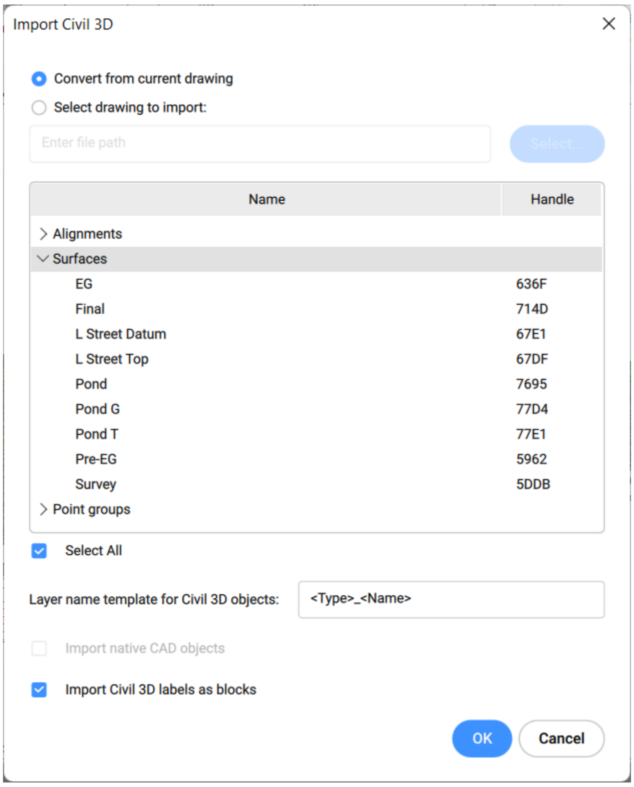
1 Open a drawing with a Civil 3D surface.

2 Launch the CIVIL3DIMPORT command.

3 In the **Import Civil 3D** dialog box, select **Convert from current drawing**.

4 ***Note:*** Select AutoCAD® Civil 3D Surfaces that will be converted to BricsCAD Civil TIN Surfaces.

Note: In the **Import Civil 3D** dialog box, you can also select other Civil 3D objects, such as Point Groups, Alignments and Profiles, that will be converted to BricsCAD Civil elements.



5 Specify the name template for Civil 3D objects.

6 Tick the option **Import Civil 3D labels as blocks**.

7 Click the OK button.

8 The **Create from Civil 3D surface** definition is created.

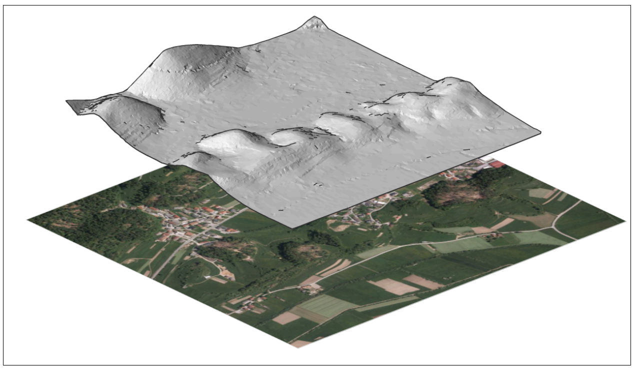
8.3.1.13 About TIN Assign Image

1 Open the drawing file that contains a TIN surface and an image.

2 Launch the TINASSIGNIMAGE command.

3 Select the TIN surface.

4



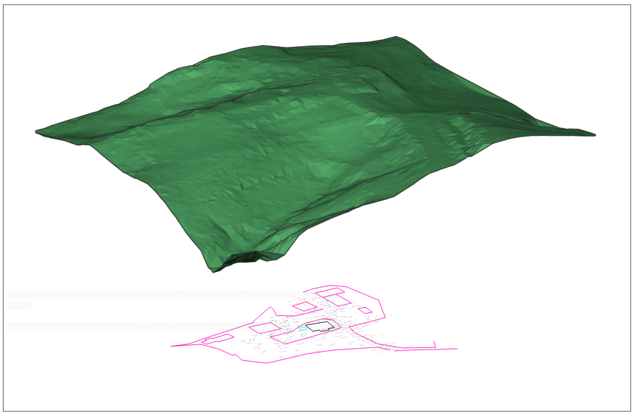
5 Select the raster image.

6 

Note: If the image is not shown on the TIN surface, change the Visual Style to Modeling or Realistic and set TIN surface triangles to ON.

8.3.1.14 About TIN Project

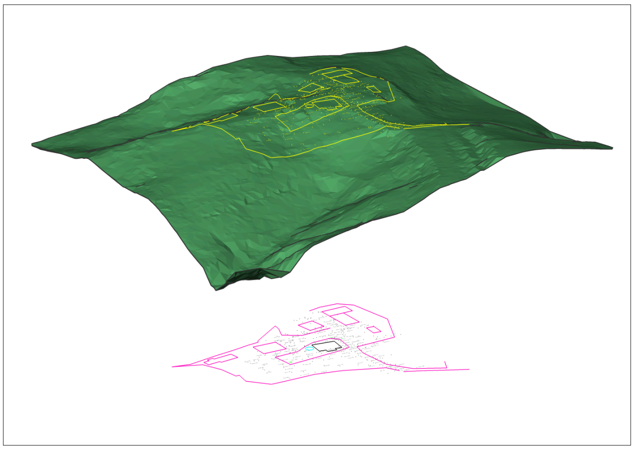
1 Open the drawing file that contains the TIN surface and the entities (points, blocks, text, lines, polylines, circles…).

1 

2 Launch the TINPROJECT command.

3 Select the entities to project.

4 Select if the original entities are to be erased.

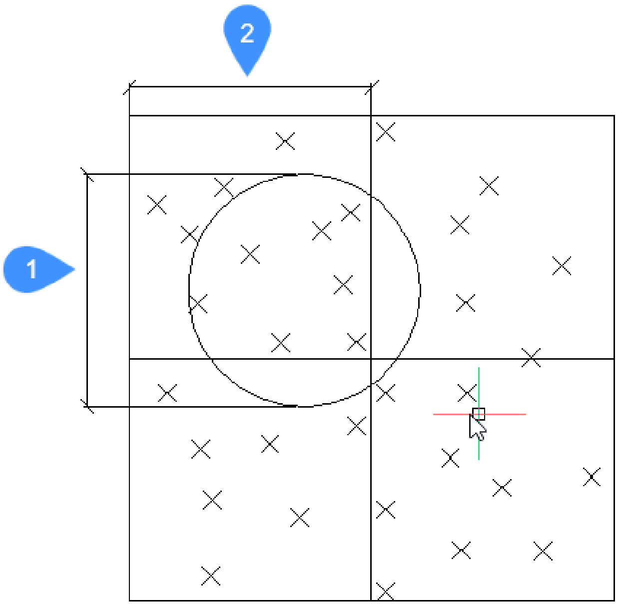
5 

8.3.2 TIN Simplify

The algorithm divides the points into square cells so that each point lies in an individual cell. In the first step, the first point is taken from the first cell, according to the order of the points in the input point file or point cloud.

Around the first point, the fulfillment of condition Δh (**Elevation difference**) is checked for all adjacent points. All points within a given radius, including those in adjacent cells, are considered as adjacent points (see figure below).

When all points with a higher value of Δh than the user-defined one are excluded, the described procedure continues for the next point in the first cell. The process then continues for all points in the adjacent cell in the x-coordinate direction and so on to the last cell.



1 Radius for simplification step

2 Radius for simplification step

The algorithm automatically proposes the value of the **Radius for simplification step** parameter. It does this by calculating the **Bounding Box** based on the given input points. It then takes the larger of the dimensions Δx and Δy (length and width). 0.5% of the obtained value is calculated, which is rounded off and displayed in the command line as the proposed value for the user.

Larger cell size (i.e. the **Radius for simplification step** parameter) brings lower accuracy of the final TIN surface. For greater accuracy of the simplified TIN network, it makes more sense to enter a smaller cell size.

8.3.3 TIN Densify

After TIN Surface is already being simplified, the user specifies the maximum elevation difference between the simplified TIN Surface and the excluded input points on a specified area in a TIN Surface. All input points that have a greater elevation difference than specified are added back to the simplified TIN Surface.

8.3.3.1 Procedure: How to densify a TIN surface

1 Simplify TIN Surface created from a point file or a point cloud

1

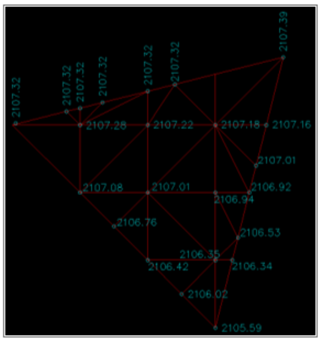


Figure 1: Original TIN Surface

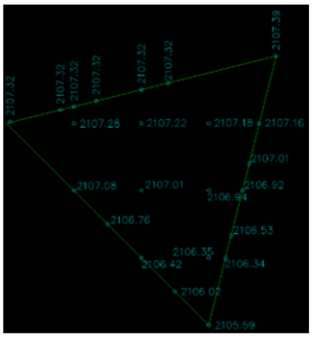


Figure 2: Simplified TIN Surface*

*Radius for a simplification step = 3, elevation difference = 0.5

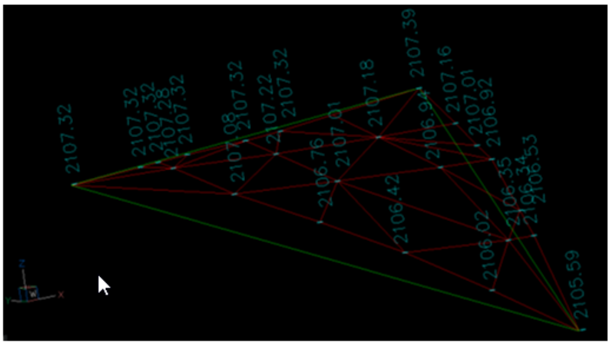


Figure 3: Original TIN presented in red, and Simplified TIN presented in green, shown in perspective 3D view

2 **Densify TIN Surface**

User has an option to select "TIN Densify" option, after TIN Surface is already being simplified.

The user specifies the maximum elevation difference between the simplified TIN Surface and the excluded input points on a specified area in a TIN Surface. All input points that have a greater elevation difference than specified, are added back to the simplified TIN Surface.

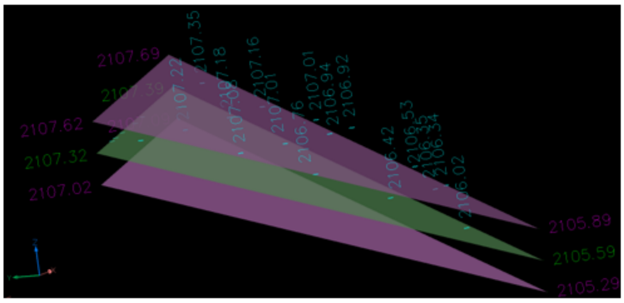


Figure 4: Simplified TIN, presented in green and "Elevation difference = 0.3", presented in magenta

In the first step algorithm takes a random point from all the points that were excluded in the TIN simplification process. Then it is checked whether the "Maximum Elevation Difference" condition is met for the selected point. If the condition is met, the point is added to a simplified TIN Surface, which is consequently updated. For the next excluded point, the elevation difference is checked on the updated TIN Surface. The procedure is repeated until the mentioned condition is checked on all excluded points.

The TIN Density algorithm thus increases the density of points and consequently the accuracy of the TIN surface, either in the selected area or in the entire TIN Surface.

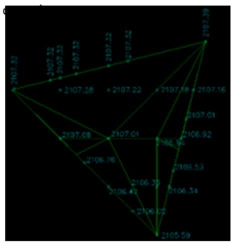


Figure 5: Densified TIN in a 2D plan view

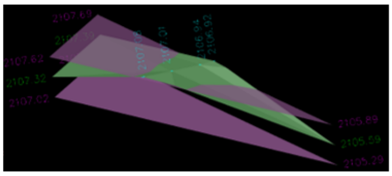


Figure 6: Densified TIN Surface in a perspective 3D view – showing points, excluded in a step 1, that are added back to the simplified TIN Surface in a step 2, according to a specified "Elevation difference" parameter.

8.3.4 TIN Surface Edit

8.3.4.1 Commands

TINEDIT

8.3.4.2 Overview

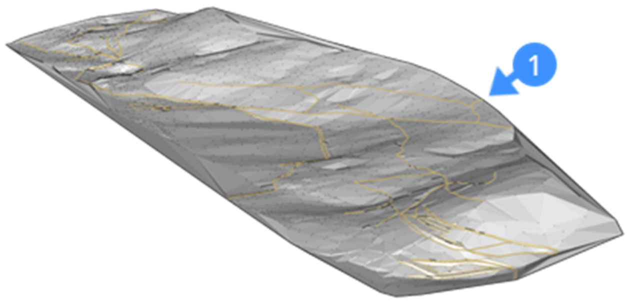
In BricsCAD, points, breaklines and boundaries on a triangulated irregular network (TIN) surface can be added or removed using the TINEDIT command.

The following entities represent features on a TIN surface:

- Points contain x y coordinates, and z values. All points are used to make a connection with the two nearest neighbors to create triangles. The surface triangulation is based on the Delaunay algorithm, which ensures that no point is inside the circumcircle of any triangle.
- Breaklines represent linear infrastructure features like curbs, retaining walls, etc. These lines also define the edges of triangles. Breaklines can be created from linear entities such as line, polyline, arc, circle.
- Boundaries are the features to define bounded 3D surfaces. The surface boundaries can be created with closed polylines. Defining outer boundaries on a large 3D surface improves the performance of a TIN Surface plan.

8.3.4.3 Adding points to a toposurface

1 Open a TIN surface with surface points (1)

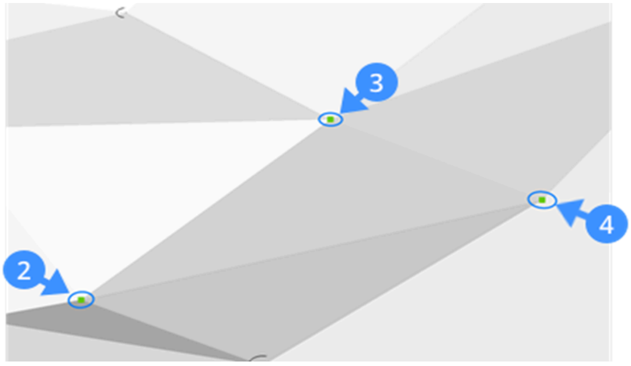


2 Select the TIN surface and launch the TINEDIT command.

3 Click on an area to add points to the surface.

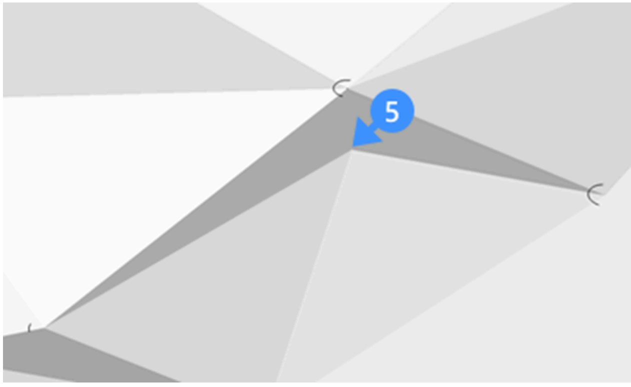
Note:

- The value is calculated as an interpolated elevation for selected point inside triangle (2,3,4).
- If ESNAP is ON, you can check the elevation of a point in the cursor coordinate.



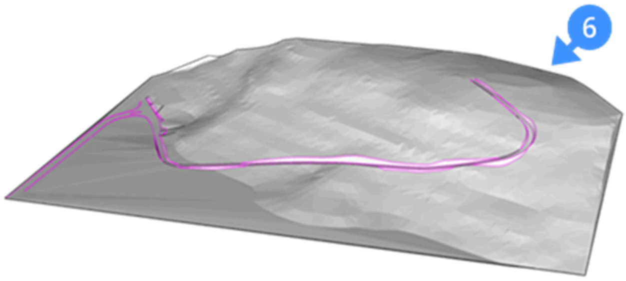
4 To specify the elevation of the newly added point, type an elevation value in the Command line, and press Enter.

5 Keep adding points to the surface, or press Enter to exit the command. The new surface point (5) is added from entities, such as polylines vertices.



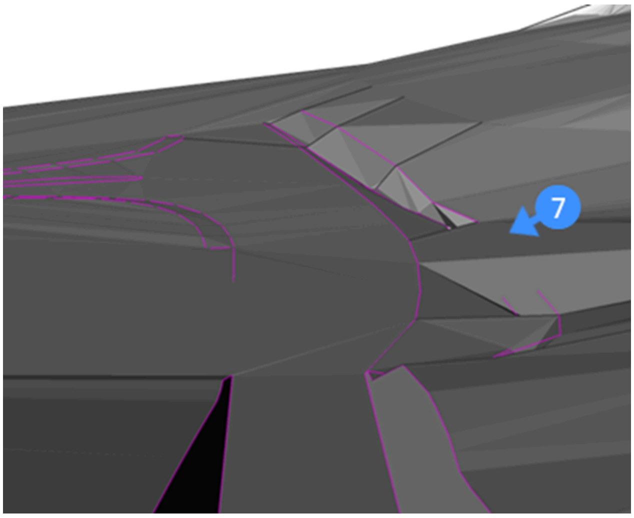
8.3.4.4 Adding breaklines to a toposurface

1 Open a TIN surface (6).



2 Select the TIN surface and launch the TINEDIT command.

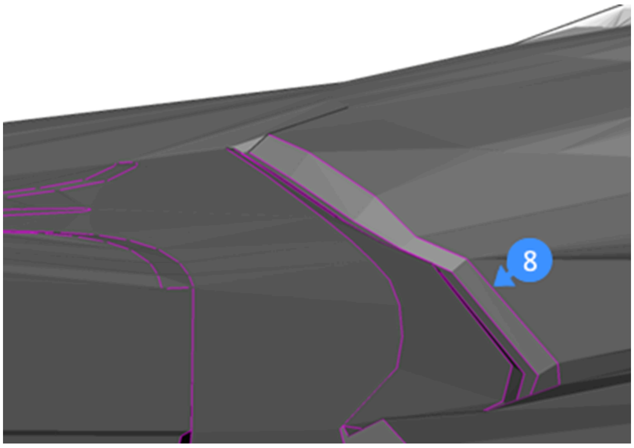
3 Select **Add Breaklines** or type **AB** in the Command line.



Note: The retaining wall breaklines before adding breaklines to the surface (7).

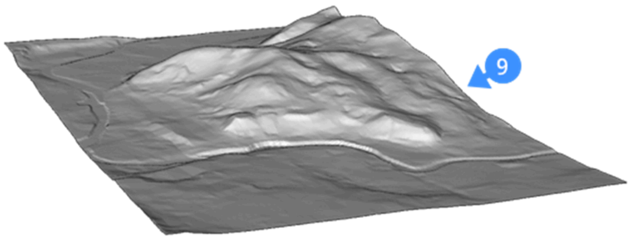
4 Select the breaklines you want to add. This will modify the surface according to the selected breakline (8).

Note: If the breakline is moved, the surface will be updated automatically.



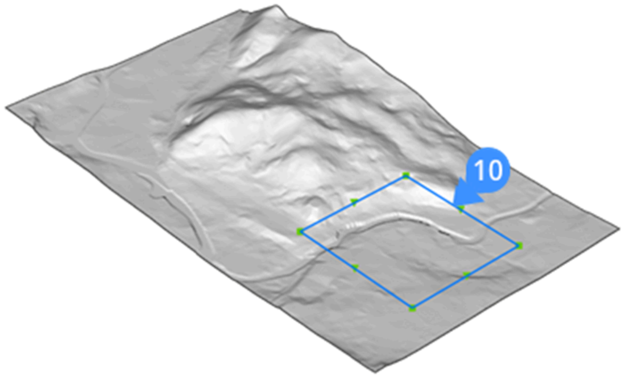
8.3.4.5 Adding boundaries to a toposurface

1 Open a TIN surface where you want to create a boundary (9).



2 Create a closed boundary on the location where you want to clip the surface (10).

Note: Make sure that Entity Snap Tracking (ESNAP) is active.



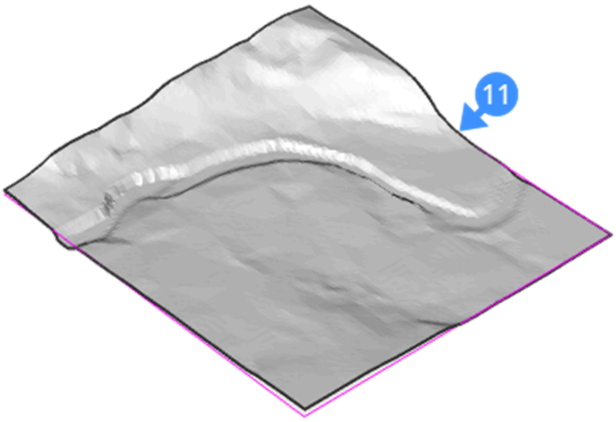
3 Select the TIN surface and launch the TINEDIT command.

4 Select **Add Boundaries** in the Command line.

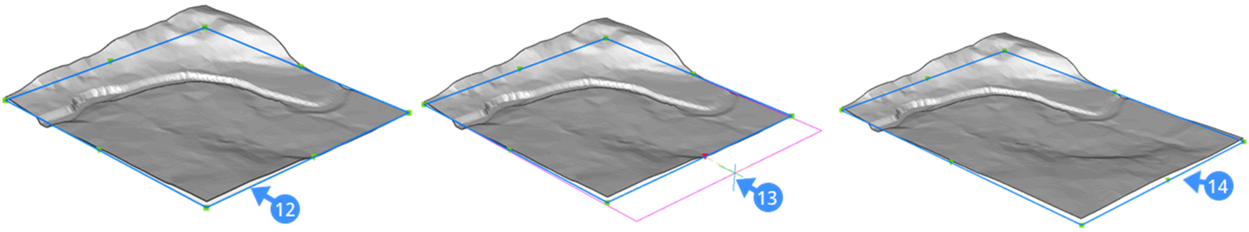
5 Click on the closed boundary to use in the surface boundary definition and press Enter.

Note:

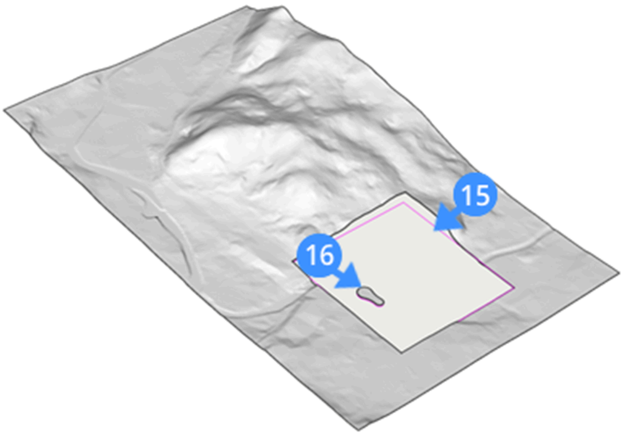
- By default, the outer option is selected, which allows creating an outer surface boundary (11).



- Select the bounding entity (12) (e.g. polyline) to resize the surface boundary. Moving the grip (13) of the entity changes the surface dynamically (14).



- Select the hide option to exclude a part of the surface from the TIN when calculating the total area.
- Select the show option to reveal the hidden surface in the TIN.



- The hidden part according to the closed boundary (15) is shown, the surface of the hidden part revealed (16) using the show option.

8.3.5 TIN Surface Direct Modeling

8.3.5.1 Commands

TINEXTRACT, TINMODIFY, TINMERGE

8.3.5.2 Overview

In BricsCAD, topographical surfaces created as a TIN Surface, can be adjusted using different tools.

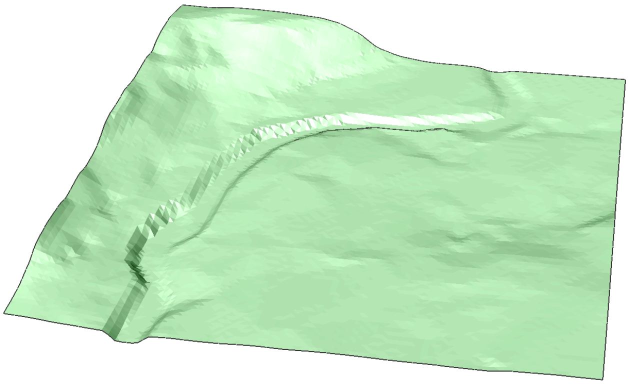
8.3.5.3 About TINEXTRACT

The TINEXTRACT command is used to generate a solid, mesh, border, intersection lines, faces and points from a TIN surface. You can exchange these entities in other applications, to further analyze the surfaces. The solids or meshes can be created in different ways: as a vertical offset of the TIN Surface, as a model between two TIN Surfaces or as a model between a fixed elevation plane and the TIN Surface.

8.3.5.4 Procedure: Creating a solid from a TIN surface using option Elevation

1 Select a TIN Surface and type TINEXTRACT in the Command line.

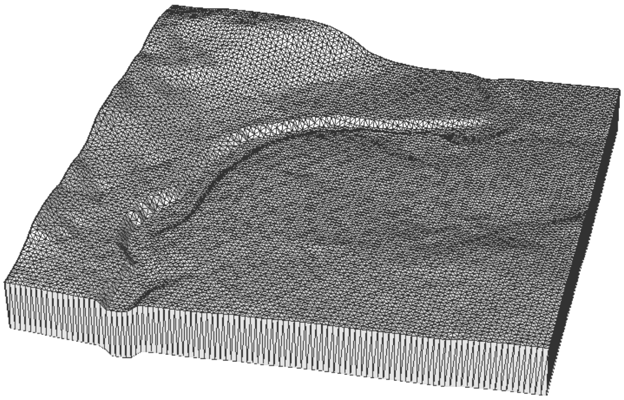
1



2 The **Extract From TIN Surface** dialog box opens. Set the **Solids/Meshes** option to On.

3 Tick the **Elevation** button and set a value for height.

4 Press the **Extract** button.



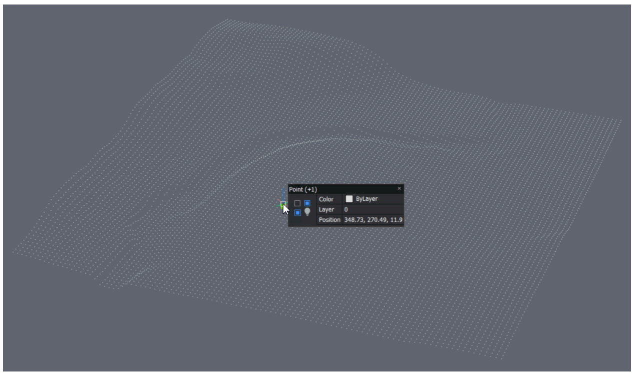
8.3.5.5 Procedure: Creating Points from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 The **Extract From TIN Surface** dialog box opens.

3 Tick the **Points** option and, optionally, select the desired layer.

4 Press the **Extract** button.



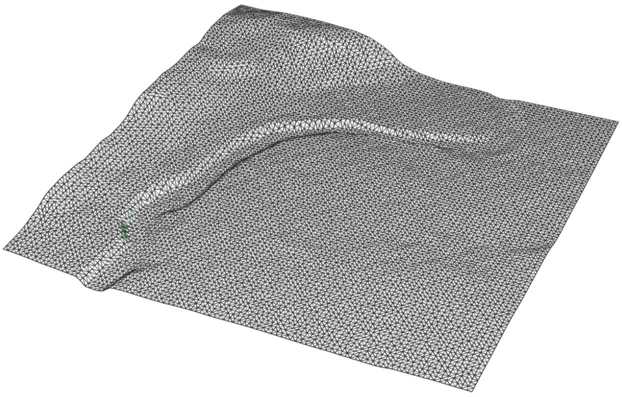
8.3.5.6 Procedure: Creating Faces from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 The **Extract From TIN Surface** dialog box opens.

3 Tick the **Triangles** option and, optionally, select the desired layer.

4 Press the **Extract** button.



8.3.5.7 Procedure: Creating a Contours from a TIN surface

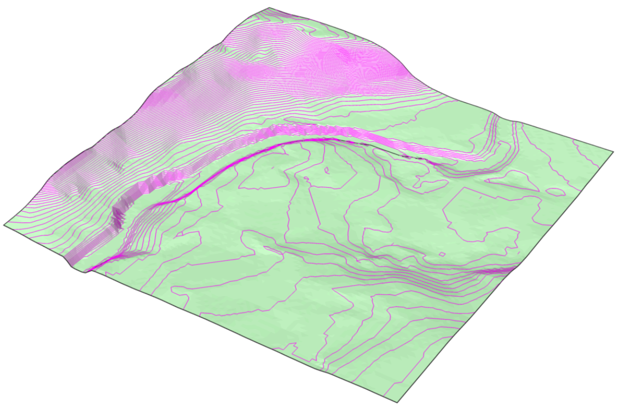
1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 The **Extract From TIN Surface** dialog box opens.

3 Tick the **Major contours** and **Minor contours** options and, optionally, select the desired layers.

4 Set the **Interval** values for both major and minor contours.

5 Press the **Extract** button.



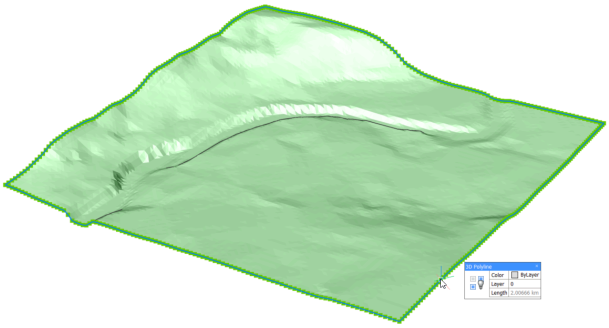
8.3.5.8 Procedure: Creating a Border from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 The **Extract From TIN Surface** dialog box opens.

3 Tick the **Borders** option and, optionally, select the desired layer.

4 Press the **Extract** button.



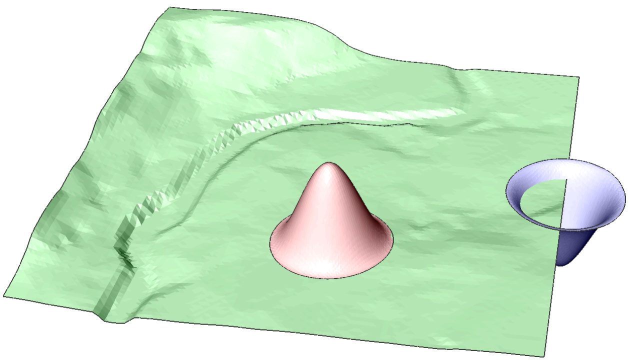
8.3.5.9 About TIN Merge

Different TIN Surfaces can be merged into one by using the TINMERGE command. A merge is executed between a base surface and one or more surfaces to merge with.

8.3.5.10 Merging TIN Surfaces

1 Type TINMERGE in the Command line.

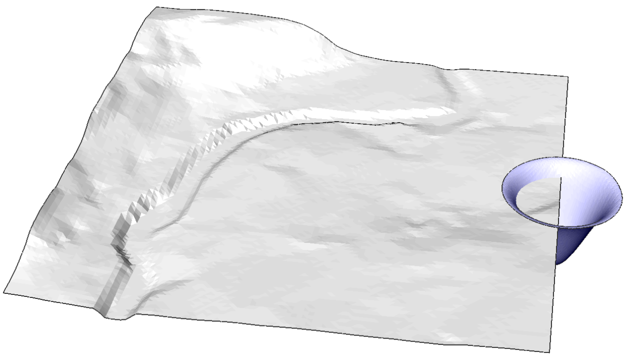
1



2 Choose a base surface. In this example, we choose the green surface as the base surface.

3 Choose to keep or delete the original surfaces.

4 ***Note:*** Selecting a different baseline, results in a different merge. It is important to remember that the surface to merge replaces the part of the base surface that it covers.



The first example shows the merge of the green surface as the base surface and the blue and red surface.

The second surface shows the merge of the blue surface as the base surface and the green surface.

The third surface shows the merge of the red surface as the base surface and the green surface.

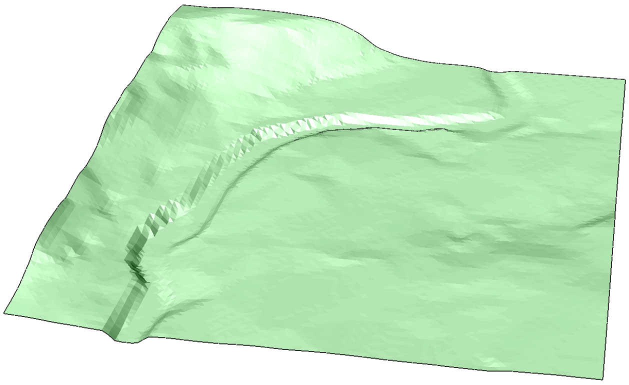
8.3.5.11 About TIN Modify

A TIN Surface can be modified by deforming or smoothening the surface. By deforming the surface, the TIN Surface is adjusted following the contours of an area defined by the user. By smoothening the surface, a selected area is leveled which is done by interpolation of this predefined area.

8.3.5.12 Procedure: Adding deformations on a TIN Surface using Radius

1 Select a TIN Surface and type TINMODIFY in the Command line.

1



2 Select the **Deform** option.

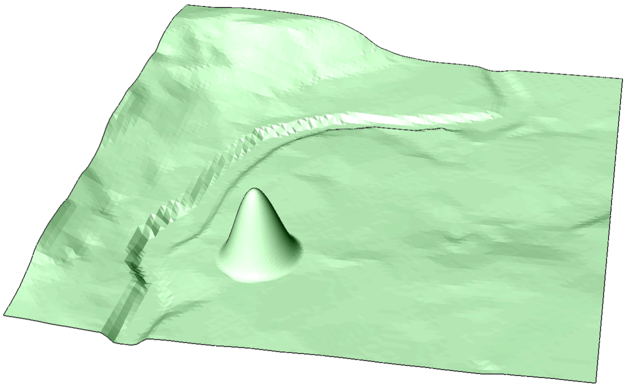
3 Choose a deformation radius.

4 Click the area to apply the deformation.

5 Choose the elevation height.

6 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface.

7



Note: By adding the modification as a new element, the structure of the original TIN Surface isn't modified but added as a new TIN surface.

8 Click other areas to add new deformation or press Enter to end the command.

8.3.5.13 Procedure: Adding deformations on a TIN Surface using specify Contour

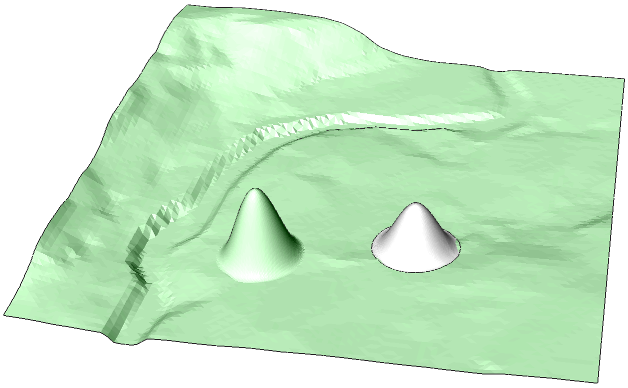
1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

3 Select the **specify Contour** option.

4 Pick a contour on the TIN Surface.

5

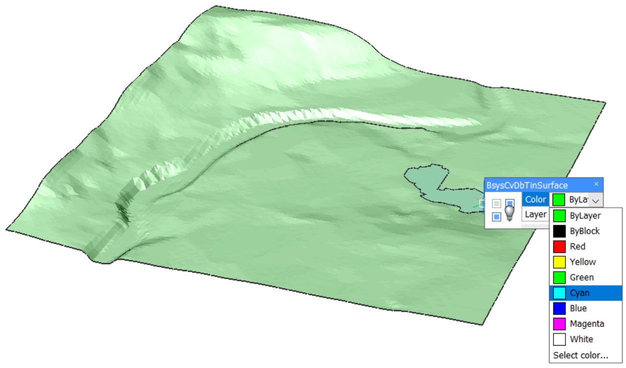


6 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a new element for the adjustment.

7 Click other contours to add new deformation or press Enter to end the command.

8 You have now created a new element on your TIN Surface. You can change its properties as you like. For example: change the color of the element.

9



8.3.5.14 Procedure: Adding deformations on a TIN Surface using select Entity

1 Select a TIN Surface and type TINMODIFY in the Command line.

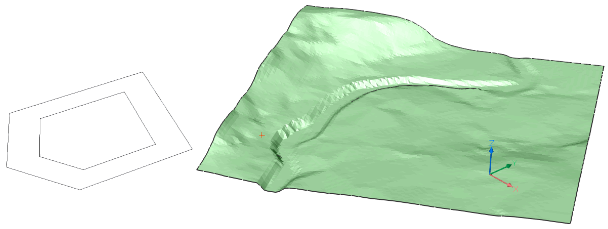
2 Select the **Deform** option.

3 Type E and hit Enter to choose **Select Entity**.

4 Select a closed entity (for example, Polyline).

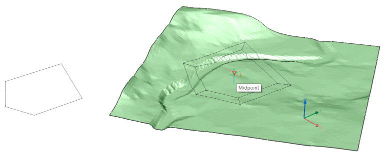
5 Choose an offset distance.

6



7 Click the area to apply the deformation.

8



9 Choose the elevation height.

10

11 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a New element for your adjustment.

12 Click other areas to add new deformation or press Enter to end the command. You have now added a deformation using **select Entity**.

8.3.5.15 Procedure: Adding deformations on a TIN Surface using draw Polygon

1 Select a TIN Surface and type TINMODIFY in the Command line.

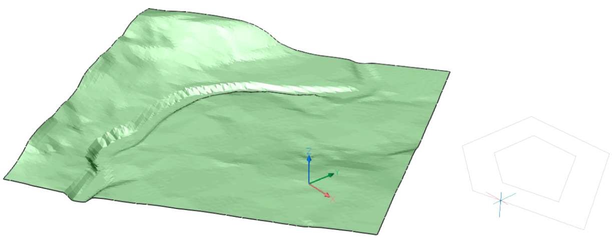
2 Select the **Deform** option.

3 Select **draw Polygon**.

4 Draw a polygon, hit Enter to close the polygon.

5 Choose an offset distance.

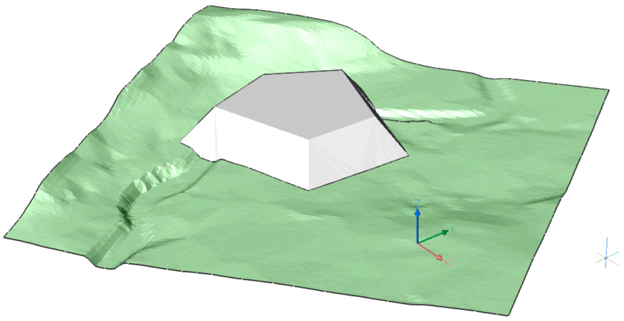
6



7 Select the area where to apply the deformation.

8 Choose the elevation height.

9



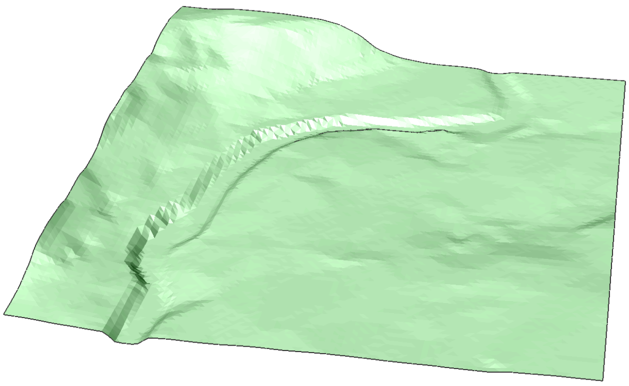
10 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a new element for your adjustment.

11 Click other areas to add new deformation or press Enter to end the command. You have now added a deformation using draw polygon.

8.3.5.16 Procedure: Smoothening a TIN Surface

1 Select a TIN Surface and type TINMODIFY in the Command line.

1

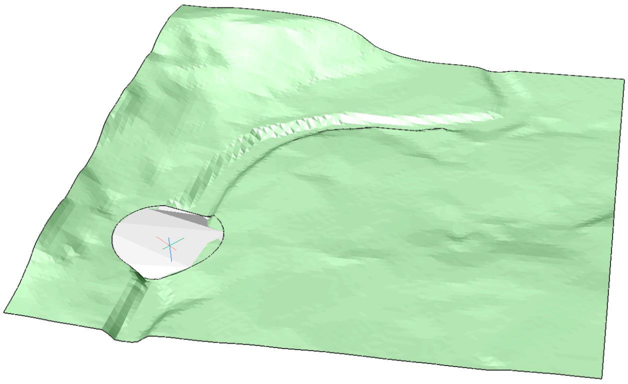


2 Select the **Smoothen** option.

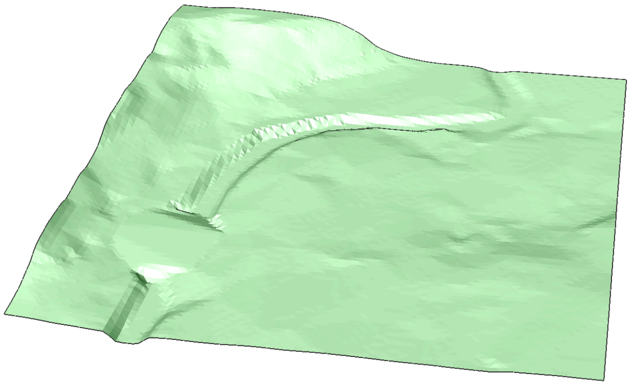
3 Choose a radius.

4 ***Note:*** The procedure for **select Entity** or **draw Polygon** is analogue to **Adding deformations using select Entity** and **Adding deformations using draw Polygon** starting from step 3.

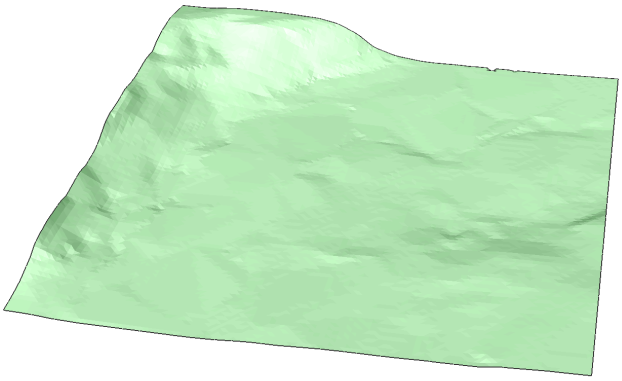
5 A preview of the intervention is shown.

6 

7 Click to execute the command.

8 

9 Click other areas to smoothen or press Enter to end the command.

10 

8.3.6 Analyzing TIN Surfaces

8.3.6.1 Commands

TINVOLUME, TINWATERDROP

8.3.6.2 Overview

In BricsCAD, topographical surfaces created as a TIN Surface can be analyzed using different tools.

8.3.6.3 About TIN Volume

A TIN volume surface provides an exact difference between the base and comparison surfaces. Therefore, the Z-value of any point in the volume surface is precisely the difference between the Z of the comparison surface at that point and the base surface at that point. A TIN Volume Surface is drawn as a custom entity with additional fill/cut volume properties.

You can use TIN Volume to calculate or analyze earthworks:

- Calculate fill/cut volumes inside bounding area.
- Draw zero contours which divide fill and cut areas.
- Visualize the mass distribution.

8.3.6.4 Creating a TIN Volume

In the following procedures, a TIN Volume is created. Choose whether the volume is associative or not. If associativity is turned on, the TIN Volume Surface is automatically updated if base or comparison surfaces are changed. To disable or enable the associativity, follow these steps:

1 Type SETTINGS in the Command line and press Enter. The **Settings** dialog box opens.

2 Go to **Civil** > **Civil General** > **Associativity**.

3 Tick or untick **TIN Volume Surfaces are associative**.

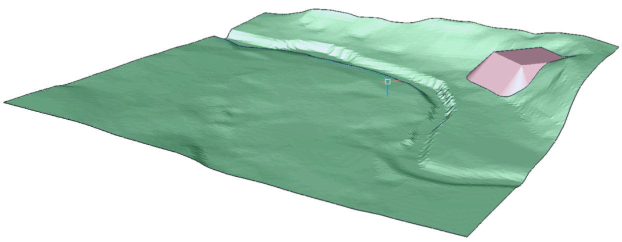
Civil General	
Associativity	0x000F (15)
1	☑ TIN Surfaces are associative
2	☑ Gradings are associative
4	☑ TIN Volume Surfaces are associative
8	☑ Corridors are associative

8.3.6.5 Creating a TIN Volume using a comparison TIN Surface

1 Launch the TINVOLUME command.

2 Choose a base surface. In this example, the green TIN surface is chosen as the base surface.

3



4 Choose a comparison surface. In this example, the red grading is chosen as the comparison surface.

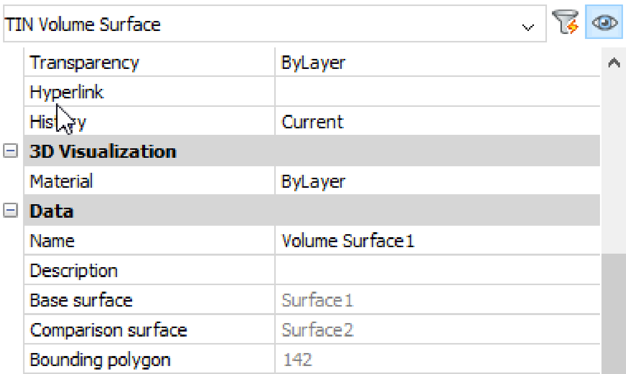
5 Select a bounding area as the area to apply the operation or press Enter. The outer boundary of the surface will be used as a boundary.

6 The **TIN Statistics** category appears under the **TIN Surface Properties** panel and displays the **Cut volume** and **Fill volume** inside bounding area:

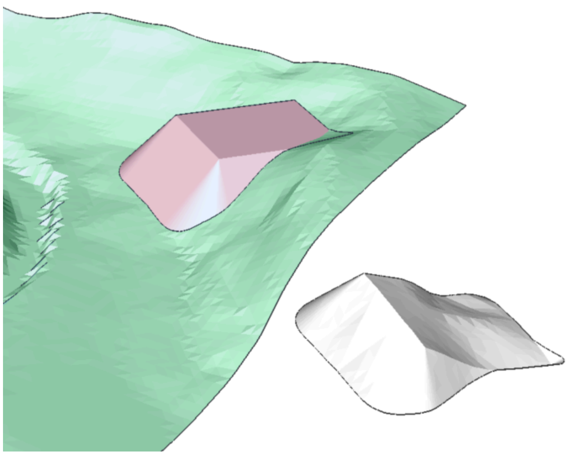
TIN Volume Statistics	
Cut volume	442.08 m³
Fill volume	34732.76 m³

The **Tin Volume Surface Properties** panel under the Data category contains information about:

- Name
- Description
- Base surface (read-only)
- Comparison surface (read-only)
- Bounding polygon (read-only)


TIN Volume Surface

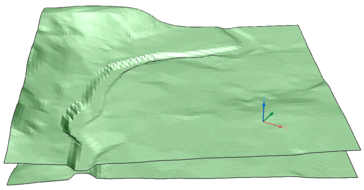
Transparency	ByLayer
Hyperlink	
History	Current
3D Visualization	
Material	ByLayer
Data	
Name	Volume Surface1
Description	
Base surface	Surface1
Comparison surface	Surface2
Bounding polygon	142



8.3.6.6 Creating a TIN Volume using Elevation

1. Launch the TINVOLUME command.
2. Choose a base surface.
3. Select **Elevation** in the Command line.
4. Choose the elevation height.
5. Select a bounding area as the area to apply the operation or press ENTER. The outer boundary of the surface will be used as a boundary.

6



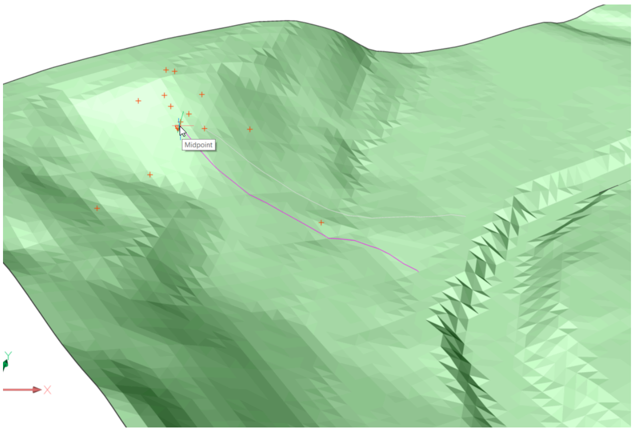
8.3.6.7 About TIN Water Drop

The TINWATERDROP command creates water drop paths in real-time by moving the cursor over a TIN Surface. A water path is created as a 3D polyline in the current position if you click the mouse button.

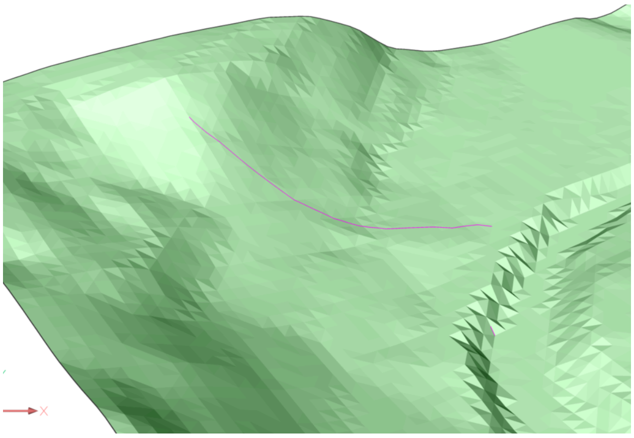
8.3.6.8 Procedure: Creating a water drop path

1. Launch the TINWATERDROP command.
2. Select the TIN surface.
3. Move the cursor over the surface, a line representing the drop path is shown in real-time.

   Note: A colored water drop path is more clearly visible. To color the water drop path, change the current layer to another color, e.g. red.



4. Left click the mouse to create a 3D polyline of the water drop path.
5.

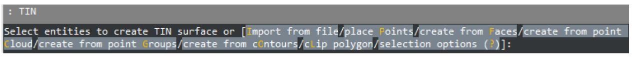


8.3.7 TIN Definitions

8.3.7.1 TIN Definitions in TIN command

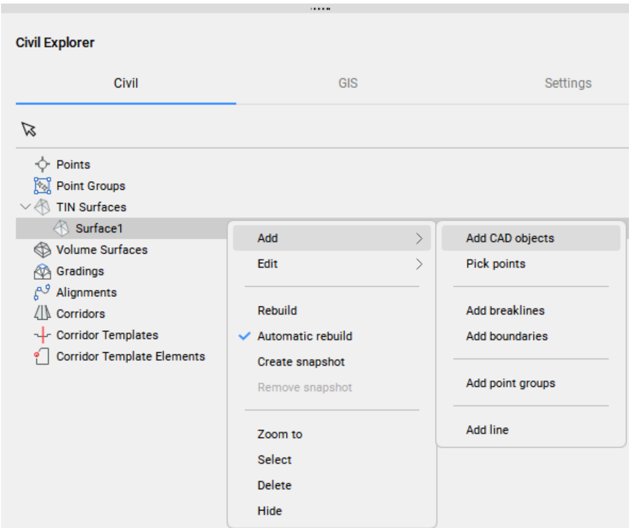
1 Overview

This article describes the TIN Definitions/operations that are created with the TIN Surface creation methods available in the TIN command. It also describes the dialog of each definition, where you can modify parameters.



Some of the operations, such as **Create from faces**, **Add point clouds**, and **Create from contours** are only used to create a new TIN Surface.

While other operations, such as **Add drawing (CAD) objects**, **Add point**, **Add point groups** can also be used to add data to existing TIN Surface. For easy access, these operations are also available in the **Civil Explorer** panel (see figure below).



Note: Some TIN operations listed in the **Add** group in **Civil Explorer** are available in the TINEDIT command.

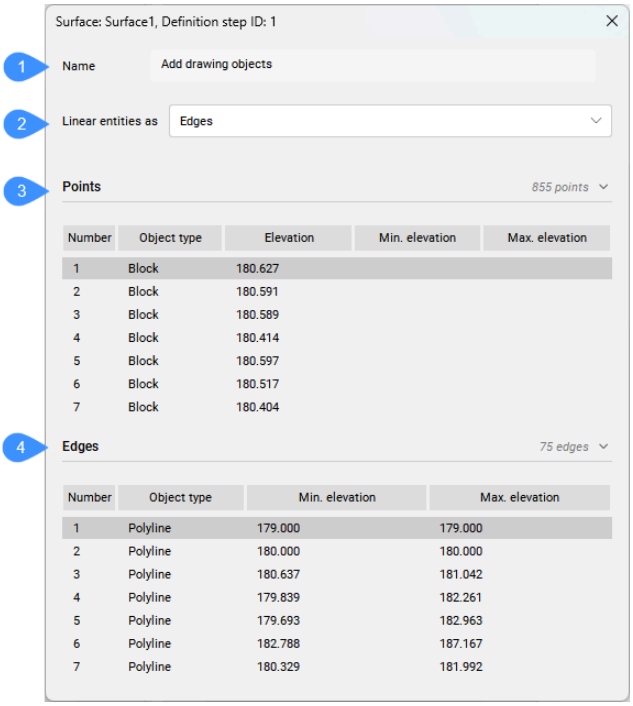
2 Add drawing objects

The **Add drawing objects** definition creates a new TIN Surface or modifies an existing one from selected drawing/CAD objects.

To create the **Add drawing object** definition:

- Use the **Select entities to create TIN surface** option in the TIN command.
- Right-click an existing TIN Surface in the **Civil Explorer** and select **Add CAD objects** from the **Add** group.

When you double-click the existing **Add drawing objects** definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Linear entities as

3 Points

4 Edges

Name

Displays the TIN Surface definition name: **Add drawing objects**.

Linear entities as

Allows you to control how linear entities are applied to the TIN surface. You can choose between points and edges.

Points

Lists the points added to the TIN Surface as drawing objects. In addition to CAD points, also blocks, texts and Civil points are added to a TIN Surface as points. If the **Points** option is selected for polylines added to the TIN surface as drawing objects, the vertices of the polylines are also added to the TIN surface as points.

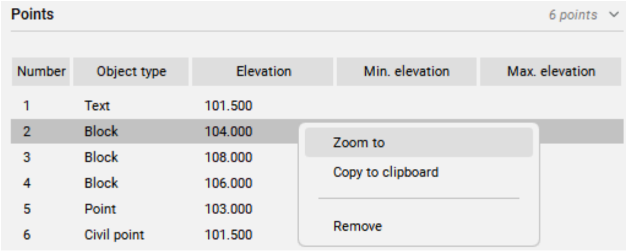
Note: The entities that could be added with **Add Points** option are points, Civil points, lines, or polylines.

Edges

Lists the linear entities that are applied to TIN Surface as triangles edges.

Note: Right-click the object in the dialog box to open a context menu:

- **Zoom to**: zooms to the object in the drawing.
- **Copy to clipboard**: copies the geometry information.
- **Remove**: removes the object from the TIN Surface definition.



Min. and Max. elevation: the min. and max. elevation of the 3D Polyline. If the min. and max. elevations match, then the object is 2D Polyline / Polyline.

3 Add point files

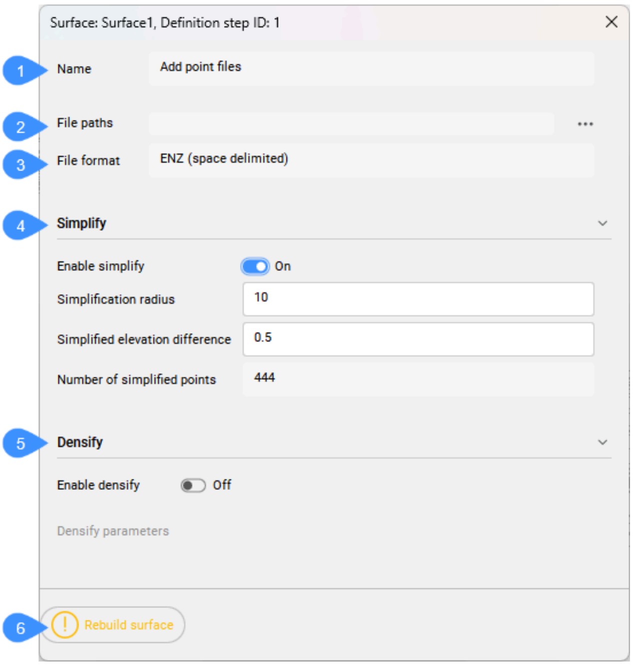
The **Add point files** definition creates a TIN Surface from a selected point file.

It allows the simplification of points from the source point file while maintaining high accuracy in user-defined areas of the TIN Surface where more accurate terrain data is required.

To create the **Add point files** definition:

- select the **Import from file** option in the TIN command.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 File paths

3 File format

4 Simplify

5 Densify

6 Rebuild surface

Name
Displays the TIN Surface definition name: **Add point files**.

File paths
Displays the file path. Press the browse button to open the **Create Surface From Point File** dialog box.

File format
Displays the file format used for importing point file.

Simplify
Simplifies the TIN surface.

Enable simplify
When enabled, it allows you to change the simplification parameters.

Simplification radius
Specifies the cell size within which all points with an elevation difference greater than the specified maximum are eliminated/simplified.

> ***Note:*** Larger cell size (that is the **Radius for simplification step** parameter) brings lower accuracy of the final TIN Surface. For greater accuracy of the simplified TIN Surface, it makes more sense to enter a smaller cell size.

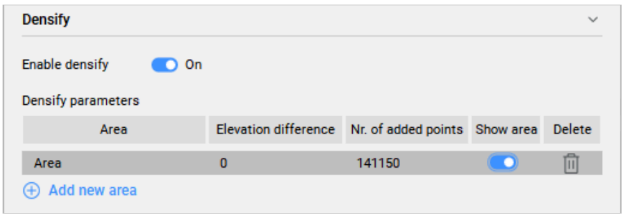
Simplified elevation difference
Specifies the maximum elevation difference between adjacent points on the TIN Surface. All points with a higher elevation difference are excluded from the TIN Surface.

Number of simplified points
Displays the number of excluded/simplified points.

Densify
Densifies the TIN surface.

Enable densify
When enabled, it displays the file format used for importing point file.

Densify parameters:



Elevation difference
Specified maximum elevation difference between the simplified TIN Surface and the excluded input points on a specified area in a TIN Surface.

Number of added points
Displays the number of points that have an elevation difference greater than the specified maximum value, and are consequently added back to the simplified TIN Surface.

Show area
If enabled, it highlights the specified densification area in a drawing.

Delete
Enables to delete the densification area.

Rebuild surface
Rebuilds the surface.

Note: This button appears when editing the Simplify and Densify parameters.

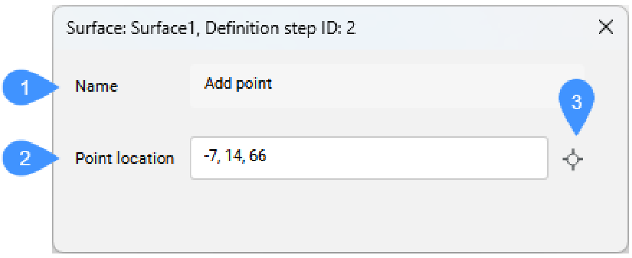
4 Add point

The **Add point** definition creates a new TIN Surface or modifies an existing one based on specified locations and elevations of the TIN points.

To create the **Add point** definition:

- Select the **place Points** option in the TIN command.
- Right-click an existing TIN Surface in the **Civil Explorer** and select **Pick points** option from the **Add** group.

When you double-click the **Add point** definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Point location

3 Pick point

Name
Displays the TIN Surface definition name: **Add point**.

Point location
Displays the point coordinates.

Pick point
Allows you to pick a new location and set a new point elevation.

5 Create from faces

The **Create from faces** definition creates a new TIN Surface from the selected 3D Face elements.

To create the **Create from faces** definition:

- select the **create from Faces** option in the TIN command.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.

Name
Displays the surface definition name: **Create from faces**.

6 Add point clouds

The **Create from Point Cloud** definition creates a TIN Surface from a selected Point Cloud.

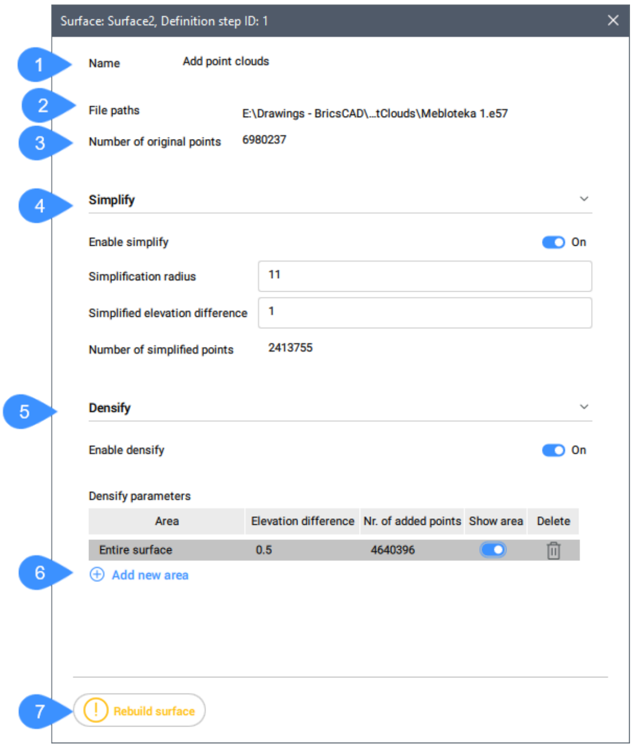
It allows the simplification of points from the source Point Cloud while maintaining high precision in user-defined areas of the TIN Surface where more accurate terrain data is required.

The **Remove outer edges** and **Create snapshot** definitions are automatically added to the TIN Surface after this operation.

To create the **Add point clouds** definition:

- select the **create from point Cloud** option in the TIN command.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 File paths

3 Number of original points

4 Simplify
5 Densify
6 Add new area
7 Rebuild surface

Name
Displays the TIN Surface definition name: **Add point clouds**.

File paths
Displays the input file path.

Number of original points
Displays the number of original points in the point clouds.

Simplify
Simplifies the TIN surface.

Enable simplify
When enabled, it allows you to change the simplification parameters.

Simplification radius
Specifies the cell size within which all points with an elevation difference greater than the specified maximum are eliminated/simplified.

> ***Note:*** Larger cell size (that is the **Radius for simplification step** parameter) brings lower accuracy of the final TIN Surface. For greater accuracy of the simplified TIN Surface, it makes more sense to enter a smaller cell size.

Simplified elevation difference
Specifies the maximum elevation difference between adjacent points on the TIN Surface. All points with a higher elevation difference are excluded from the TIN Surface.

Number of simplified points
Displays the number of excluded/simplified points.

Densify
Densifies the surface.

Enable densify
When enabled, it displays the file format used for importing point clouds file.

Densify parameters:

Elevation difference
Specified the maximum elevation difference between the simplified TIN Surface and the excluded input points on a specified area in a TIN Surface.

Number of added points
Displays the number of points that have an elevation difference greater than the specified maximum value, and are consequently added back to the simplified TIN Surface.

Show area
If enabled, it highlights the specified densification area in a drawing.

Delete
Allows you to delete the densification area.

Add new area
Adds a new densification area. It requires to enter the elevation difference in the Command line and then select an existing or draw a new polygon in the drawing.

Rebuild surface
Rebuilds the TIN surface.

Note: This button appears when editing the Simplify and Densify parameters.

7 Add point groups

The **Add point groups** definition adds **Civil points** from a specified **Point Groups** to the existing TIN Surface or creates a new TIN Surface from specified **Point Groups**.

To create the **Add point groups** definition:

- Select the **create from point Groups** option in the TIN command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Add point groups** from the **Add** group.

The **Remove outer edges** definition is automatically added to the TIN Surface after this operation.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.

Name
Displays the surface definition name: **Add point groups**.

Point groups
Displays a list of Point groups used to create TIN Surface.

8 Create from Contours

The **Create from Contours** definition creates a TIN Surface from selected line entities. The density of points on contours taken for triangulation can be managed by specifying **weeding** and **supplementing** factors. When the input data includes arcs, the accuracy of TIN point along arcs depends on the **Mid-ordinate distance** parameter.

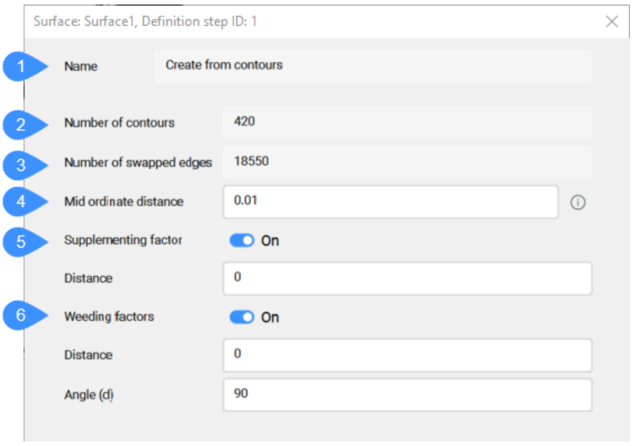
This method automatically minimizes flat triangles by swapping TIN edges.

To create the **Create from Contours** definition:

- select the **create from cOntours** option in the TIN command.

The **Remove outer edges** definition is automatically added to the TIN Surface after this operation.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Number of contours

3 Number of swapped edges

4 Mid-ordinate distance

5 Supplementing factor

6 Weeding factor

Name
Displays the TIN Surface definition name: **Create from contours**.

Number of contours
Displays the number of the contours.

Number of swapped edges
Displays the number of the swapped edges.

Mid-ordinate distance
Adds supplementary TIN points along the arc according to the mid-ordinate distance which is used for arc approximation.

Supplementing factor
Toggles the using of supplementing factor.

Distance
Sets the supplementing distance.

Weeding factor
Toggles the using of weeding factor.

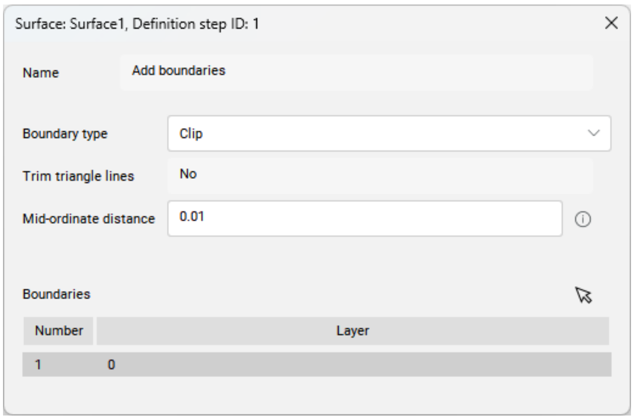
Distance
Sets the weeding distance.

Angle
Sets the weeding angle.

9 Clip polygon

If you select the **cLip polygon** option in the TIN command, a new **Clip Boundary** is added to the TIN Surface as an **Add boundaries** definition.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.

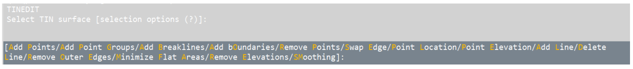


Note: See the description of the **Add boundaries** definition for a detailed explanation of the parameters in this dialog box.

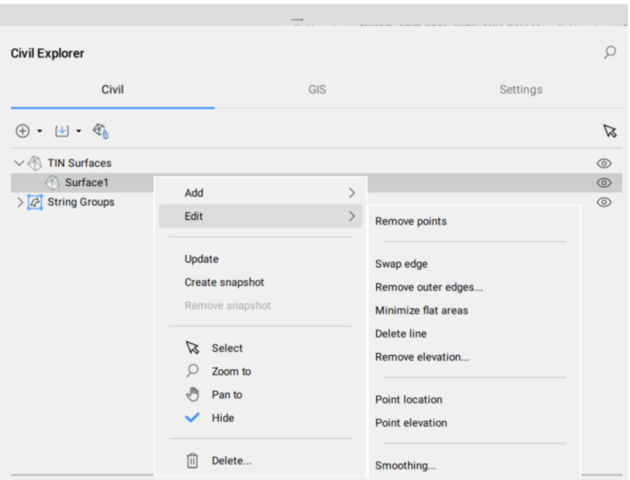
8.3.7.2 TIN Definitions for editing

1 Overview

This article describes the TIN Definitions/operations available in the TINEDIT command or created from Civil 3D surfaces. It also describes the dialog box of each definition, where you can modify parameters.



These operations are also available in the **Civil Explorer** panel (see figure below).



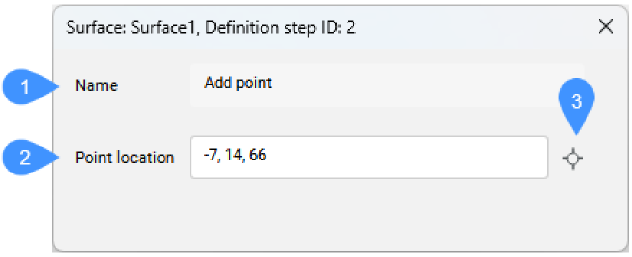
2 Add point

The **Add point** definition creates a new TIN Surface or modifies an existing one based on specified locations and elevations of the TIN points.

To create the **Add point** definition:

- Select the **Add Points** option in the TINEDIT command.
- Right-click an existing TIN Surface in the **Civil Explorer** and select **Pick points** option from the **Add** group.

When you double-click the **Add point** definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Point location

3 Pick point

Name
Displays the TIN Surface definition name: **Add point**.

Point location
Displays the point coordinates.

Pick point
Allows you to pick a new location and set a new point elevation.

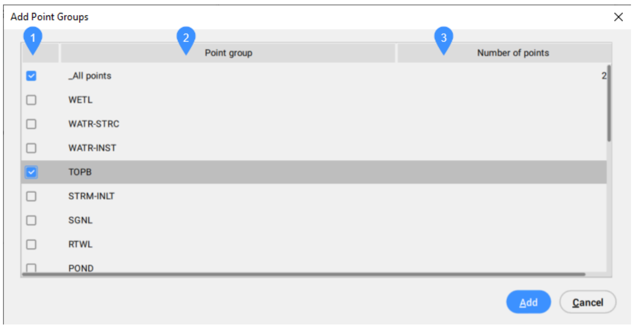
3 Add point groups

The **Add point groups** definition adds **Civil points** from a specified **Point Groups** to the existing TIN Surface or creates a new TIN Surface from specified **Point Groups**.

To create the **Add point groups** definition:

- Select the **Add Point Groups** option in the TINEDIT command.
- Right-click an existing TIN Surface in the **Civil Explorer** panel, select **Add point groups** from the **Add** group.

The following dialog box opens:



1 Select box

2 Point group

3 Number of points

Select box
Allows you to select or deselect a point group to be included.

Point group
Lists all available point groups.

Number of points
Displays the number of points of the point group.

The **Remove outer edges** definition is automatically added to the TIN Surface after this operation.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.

Name
Displays the surface definition name: **Add point groups**.

Point groups
Displays a list of Point groups used to create TIN Surface.

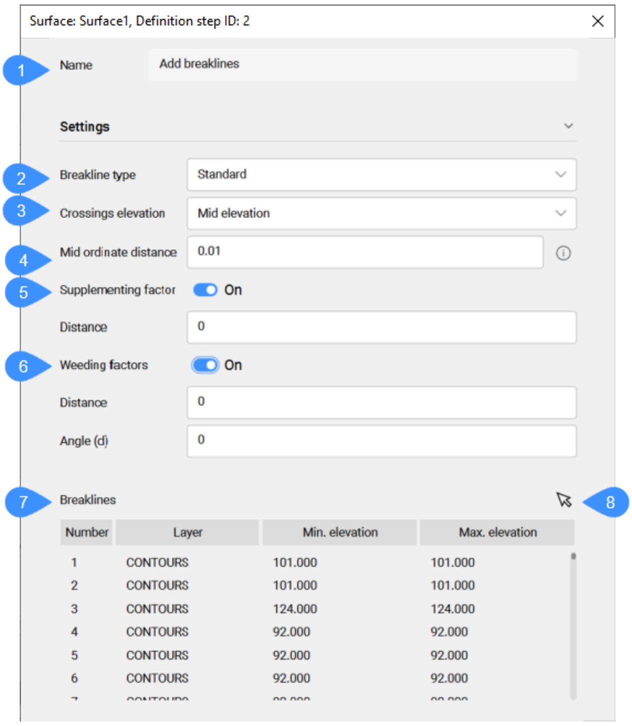
4 Add breaklines

The **Add breaklines** definition adds selected linear CAD objects to the TIN Surface as breaklines.

To create the **Add breaklines** definition:

- select the **Add Breaklines** option in the TINEDIT command.
- Create TIN Surface from **linear CAD entities/objects** and select an option to apply them as **Breaklines**.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Add breaklines** from the **Add** group.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Breaklines type

3 Crossing elevation

4 Mid-ordinate distance

5 Supplementing factor

6 Weeding factor

7 Breaklines

8 Pick button

Name

Displays the TIN Surface definition name: **Add breaklines**.

Breaklines type

Displays the selected breakline type. Select the appropriate breakline type from the drop-down list.

Standard breakline

Adds breaklines to the TIN Surface, re-builds the TIN Surface so that triangles follow the breaklines edges and elevations.

Projected breakline

Projects breakline to the TIN Surface and creates new triangles.

Crossing elevation
Determines which elevation is taken for triangulation at the intersection of two breaklines, added in the same **Add breakline** definition.

Not allowed
The intersection is treated as one breakline.

Min elevation
The elevation of lower breakline is taken.

Max elevation
The elevation of upper breakline is taken.

Mid elevation
The middle elevation between the crossing breaklines is taken.

Mid-ordinate distance
Adds supplementary TIN points along the arc according to the mid-ordinate distance which is used for arc approximation.

Supplementing factor
Toggles the using of supplementing factor.

Distance
Sets the supplementing distance.

Note: Additional vertices are added to breakline, if the distance between vertices on the breakline is greater than the specified supplementing distance.

Weeding factor
Toggles the using of weeding factor.

Distance
Sets the weeding distance.

Angle
Sets the weeding angle.

Note: Existing breakline vertices are not used to create a TIN Surface, if the distance and angle between them is less than the specified weeding parameters.

Note: The **Supplementing** and **Weeding factor** are disabled for the **Projected breaklines**.

Breaklines
Lists the breaklines included in the selected **Add breaklines** definition.
The table include sequence **Number**, **Layers**, **Min. and Max elevation** of the **Breaklines**.
Right-click an individual breakline in the table offers the option to **Zoom to**, **Select**, **Remove**, or **Copy geometry to clipboard**.

Pick button
Picks breakline in the drawing to select it in the table.

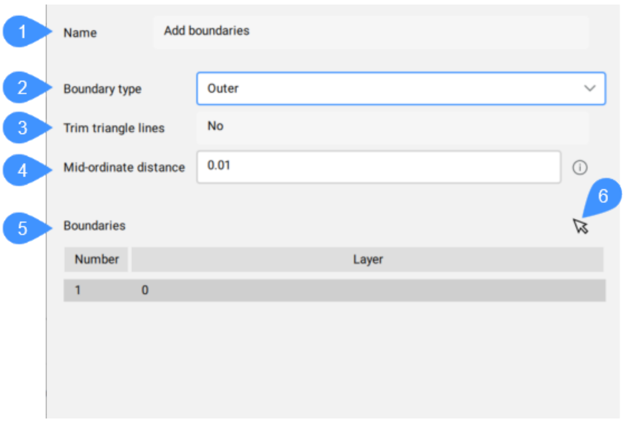
5 Add boundaries

The **Add boundaries** definition adds selected linear CAD objects to the TIN Surface as boundaries.

To create the **Add boundaries** definition:

- Select the **Add boundaries** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Add** boundaries from the **Add** group.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Boundary type

3 Trim triangle lines

4 Mid-ordinate distance

5 Boundaries

6 Pick button

Name

Displays the TIN Surface definition name: **Add boundaries**.

Boundary type

Displays the selected boundary type. Select the appropriate boundary type from the drop-down list.

Show

Displays triangles inside the boundary polygon.

Hide

Hides triangles inside the boundary polygon.

Outer

Defines the outer boundary of the TIN Surface. All the triangles outside the outer boundary are hidden.

Clip

Does not change the TIN Surface boundary, but affects all elements (definitions), added to the TIN Surface after the Clip boundary. Only elements located inside the Clip boundary affects triangulation.

Note: An example of such elements, that affects the triangluation inside the Clip boundary are: Point Clouds, CAD points, Civil points, 3D Faces, Breaklines, Contours.

Trim triangle lines
Displays whether or not this parameter is used when adding a boundary to the TIN Surface.

Mid-ordinate distance
Adds supplementary TIN points along the arc according to the mid-ordinate distance which is used for arc approximation.

Boundaries
Lists the boundaries added in a single definition and their layers. The context menu offers the possibility to **Zoom to**, **Select** or **Remove** a boundary.

Note: Multiple boundaries of the same type can be added in one step.

Pick button
Picks boundary in the drawing to select it in the table.

6 Delete points

The **Remove point** definition removes points/vertices of triangles from the TIN Surface. You can delete individual points or select the option **Multiple** in the Command line to remove multiple points at once.

To create the **Remove point** definition:

- Select the **Remove Points** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Remove points** option from the **Edit** group.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.

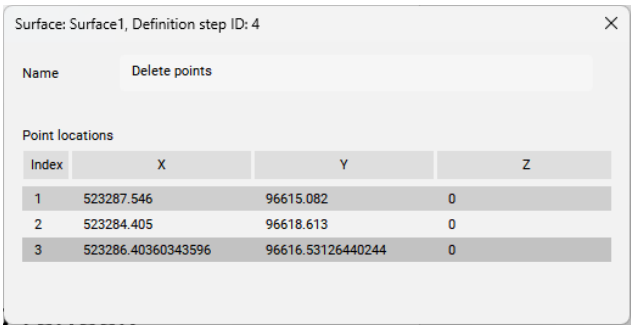
1 Name

2 Point location

Name
Displays the TIN Surface definition name: **Delete point**.

Point location
Displays the coordinates of the deleted TIN point.

Note: If the option **Multiple** is selected in the TINEDIT command (check the screenshot of the Command line above), then the dialog box displays coordinates of all deleted TIN points.



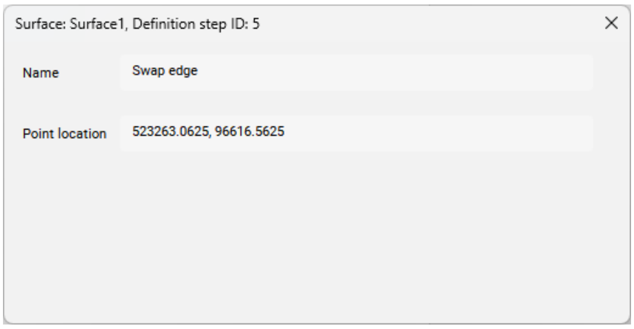
7 Swap edge

The **Swap edge** definition swaps the selected triangle edge.

To create the **Swap edge** definition:

- Select the **Swap edge** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Swap edge** option from the **Edit** group.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Point location

Name
Displays the TIN Surface definition name: **Swap edge**.

Point location
Displays the XY coordinates of the swapped edge.

8 Move point

This definition modifies the location of a single or multiple TIN points in one step.

To create the **Move point** definition:

- Select the **Point Location** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Point Location** option from the **Edit** group.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.

1 Name

2 From position

3 To position

Name
Displays the TIN Surface definition name: **Move point**.

From position
Displays the original coordinates of the TIN point before it was moved.

To position
Displays the new coordinates of the TIN point after it has been moved to a new location.

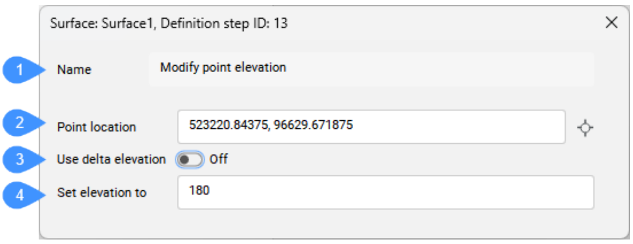
9 Modify point elevation

This definition modifies the elevation of a single or multiple TIN points in one step.

To create the **Modify point elevation** definition:

- Select the **Point Elevation** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Point elevation** option from the **Edit** group.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Point location

3 Use delta elevation

4 Set elevation to

Name
Displays the TIN Surface definition name: **Modify point elevation**.

Point location
Displays point location of the point of which elevation was modified.

Use delta elevation
Enables to specify the delta elevation (elevation difference).

Set elevation to
Sets the new elevation for the TIN point or delta elevation (elevation difference) if the **Use delta elevation** option is enabled.

10Add line

The **Add line** definition adds new TIN Edge between two existing TIN points.

To create the **Add line** definition:

- Select the **Add Line** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Add line** from the **Add** group.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.

Name
Displays the TIN Surface definition name: **Add line**.

11Delete edges

The **Delete edges** definition deletes TIN Edge inside the specified selection area.

To create the **Delete edges** definition:

- Select the **Delete Line** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Delete line** from the **Edit** group.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.

Name
Displays the TIN Surface definition name: **Delete edges**.

12Remove outer edges

The **Remove point** definition removes triangles on the outer boundary of the TIN Surface, that have:

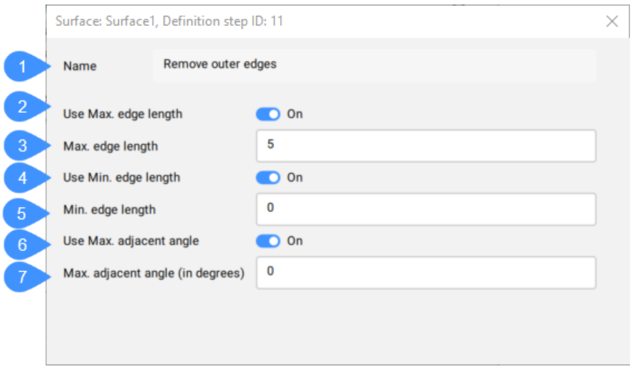
- greater edge length
- shorter edge length
- greater angle than specified

To create the **Remove outer edges** definition:

- Select the **Remove Outer Edges** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Remove outer edges** from the **Edit** group.

The **Remove outer edges** definition is automatically added to the TIN Surface after this operation.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Use Max. edge length

3 Max. edge length

4 Use Min. edge length

5 Min. edge length

6 Use Max. adjacent angle

7 Max. adjacent angle (in degrees)

Name

Displays the surface definition name: **Remove outer edges**.

Use Max. edge length

Toggles the using of maximum edge length.

Max. edge length

Sets the value for maximum edge length.

Note: Triangles with an edge exceeding the specified maximum edge length are deleted from the TIN Surface.

Use Min. edge length
Toggles the using of minimum edge length.

Min. edge length
Sets the value for minimum edge length.

Note: Triangles with an edge exceeding the specified minimum edge length are deleted from the TIN Surface.

Use Max. adjacent angle
Toggles the using of maximum adjacent angle.

Max. adjacent angle (in degrees)
Sets the value for maximum adjacent angle.

Note: Triangles with an inner angle exceeding a specified max. angle are deleted from the TIN Surface.

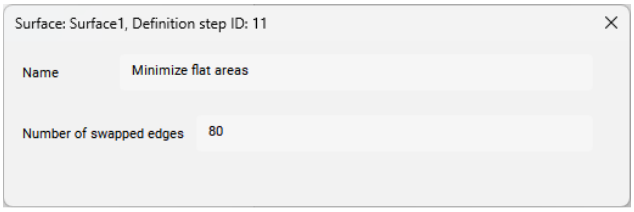
13Minimize flat areas

The **Minimize flat areas** definition finds flat triangles and then swaps the adjacent edges of those triangles so that their slope is non-zero.

To create the **Minimize flat areas** definition:

- Select the **Minimize flat areas** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Minimize flat areas** from the **Edit** group.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



Name
Displays the TIN Surface definition name: **Minimize flat areas**.

Number of swapped edges
Displays the number of swapped edges on the TIN Surface.

14Remove elevation

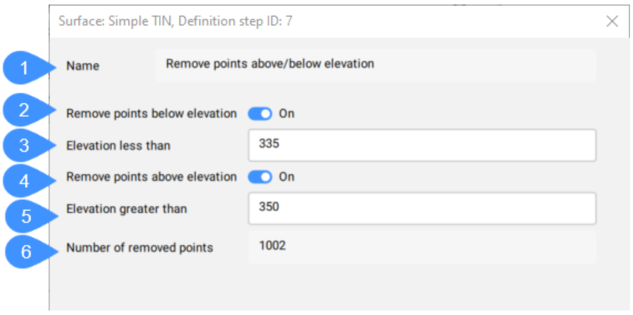
This definition removes TIN points below/above the specified elevation and displays the number of points removed.

To create the **Remove points above/below elevation** definition:

- Select the **Remove elevations** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Remove elevation** from the **Edit** group.

The **Remove outer edges** definition is automatically added to the TIN Surface after this operation.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Name

2 Remove points below elevation

3 Elevation less than

4 Remove points above elevation

5 Elevation greater than

6 Number of removed points

Name
Displays the surface definition name: **Remove points above/below elevation**.

Remove points below elevation
Toggles the option to remove points below a specified elevation.

Elevation less than
Sets the elevation below which points are removed.

Remove points above elevation
Toggles the option to remove points above a specified elevation.

Elevation greater than
Sets the elevation above which points are removed.

Number of removed points
Displays the number of removed points.

15Smoothing

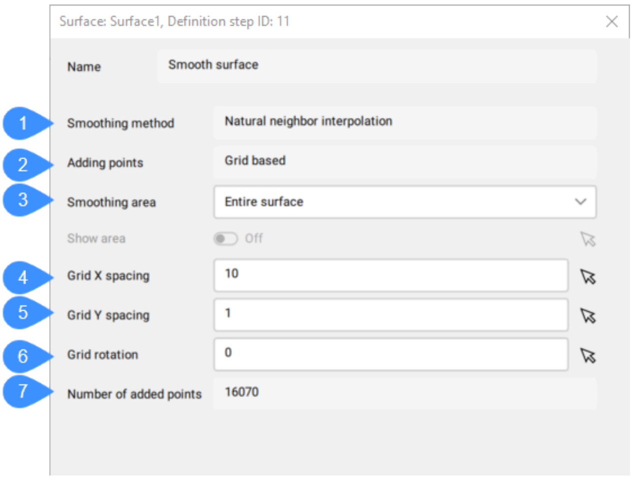
This definition adds points to the TIN surface for smoothing it.

To create the **Smoothing** definition:

- Select the **Smoothing** option in the TINEDIT command.
- Right-click an existing TIN Surface in **Civil Explorer** panel and select **Smoothing** from the **Edit** group.

The **Smoothing** definition is automatically added to the TIN Surface after this operation.

When you double-click this definition in the **Definitions** tab of the **Civil Explorer** panel, the following dialog box opens.



1 Smoothing method

2 Adding points

3 Smoothing area

4 Grid X spacing

5 Grid Y spacing

6 Grid rotation

7 Number of added points

Smoothing method

Displays the method for smoothing the TIN surface.

The **Natural neighbor interpolation** algorithm finds the closest subset of input samples to a query point and applies weights to them based on proportionate areas to interpolate a value.

Adding points

Displays the method for adding points to the TIN surface.

Smoothing area

Defines the area for smoothing. You can choose to smooth the **Entire surface** or a **Selected area**. To select the area, click the right side arrow to define the area in the Model space.

Grid X spacing

Sets the grid X spacing value. You can also define it by clicking the distance in the Model space.

Grid Y spacing

Sets the grid Y spacing value. You can also define it by clicking the distance in the Model space.

Grid rotation

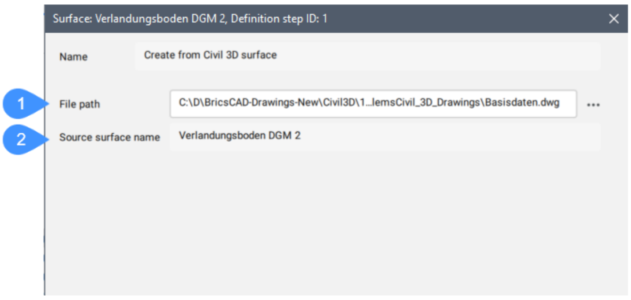
Sets the rotation value. You can also define it by clicking the desired angle in the Model space.

Number of added points

Displays the number of points that will be added to the TIN surface.

16Create from Civil 3D surface

The **Create from Civil 3D surface** definition edits a TIN Surface created from a Civil 3D surface with the CIVIL3DIMPORT command.



1 File path

2 Source surface name

File path
Displays the path of the imported 3D Civil entities file.

Note: You are enabled to change the file path.

Source surface name
Displays the source surface name.

8.4 Civil points

8.4.1 Working with Civil points

8.4.1.1 Overview

This article describes the procedures for importing points from an ASCII file and inserting an individual Civil point in a drawing.

It also describes few basic BricsCAD commands used for Civil points editing.

8.4.1.2 Importing Civil points from the ASCII File

1 Launch the CIVILPOINT command.

2 Select **Import points from file** in the Command line.

3 In the **Import Points From Point File** dialog box, select the point data file that you want to open, or choose multiple files to import all at once.

4 Specify point file format. You can edit or create new point file format in the **Manage Point File Formats** dialog box.

5 Set the **Add points to point group** button to On and choose a **Point group** name.

6 Set the symbol and label styles other than the default ones.

7 Click **Import** button to import the Civil points.

Note: If no filter is specified or no point group is selected, the points are placed in the **_All points** group by default.

8.4.1.3 Specifying Civil points locations in a drawing

1 Launch the CIVILPOINT command.

2 Specify a point location in a drawing.

3 Enter a point description.

4 Enter a point elevation.

5 Specify the location of another point or press Enter to finish the command.

Note: If there is no filter specified, the point is placed to the **_All points** group by default.

8.4.1.4 Importing Cogo points from Civil 3D

1 Open new empty drawing or a Civil 3D drawing with Cogo points (AECC_COGO_POINTS).

2 Launch CIVIL3DIMPORT command.

3 The **Import Civil 3D** dialog box opens. Choose whether to convert Cogo Points from current drawing or select the option to convert Cogo points from an external Civil 3D drawing.

4 Define Layer name template for Civil 3D objects.

5 Select whether you also want to import native CAD objects from the external Civil 3D drawing.

6 ***Note:*** If you select the option to convert Civil 3D objects from the current drawing, this option is frozen.

7 Click **Ok** to import the Civil points.

Note: The **Import Civil 3D labels as blocks** option does not affect the conversion of Civil 3D Cogo points to BricsCAD Civil points, because the Cogo points are converted to Civil points along with the labels

8.4.1.5 Importing Civil points from LandXML file

1 Launch LANDXMLIMPORT command.

2 Select XML file with Civil points that you want to open.

3 Click **Ok** to import Civil points.

8.4.1.6 Using the BricsCAD base commands to edit Civil points

1 **MOVE command:**

Select the Civil point, click on its insertion point and move the mouse to change the point position in a drawing. You can also use the MOVE command to move multiple points simultaneously.

You can also edit the points position in the **Properties** panel.

2 **COPY and PASTE commands:**

Use the COPY and COPYCLIP commands to copy selected points. The Easting and Northing coordinates of the pasted points are determined from the locations of these points in the drawing.

The elevation of new points is equal to 0. You can also edit the points position and elevation in the **Properties** panel.

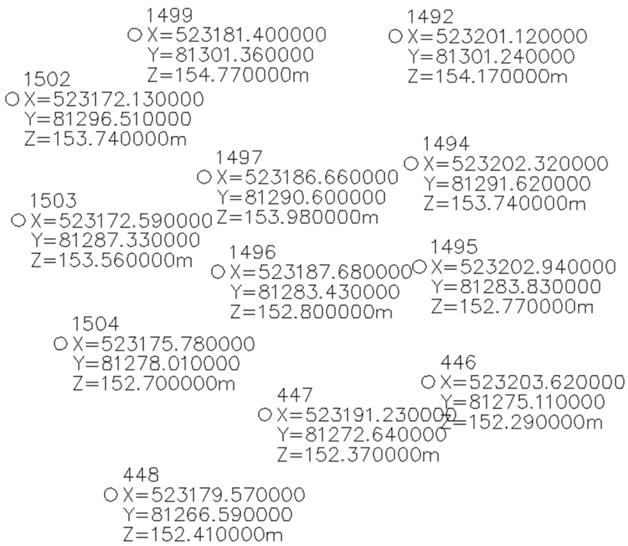
The COPYCLIP command copies all the selected Civil points to the clipboard. You can use the COPYCLIP and PASTECLIP commands to transfer Civil points from one drawing to another. Use the PASTEORIG command to place points to the same coordinates in a new drawing.

> ***Note:*** Using COPYCLIP and PASTECLIP to copy Civil points from one drawing to another will create a copy of selected Civil Points with assigned **Symbol and Label Styles**. More information about Symbol and Label Styles is available in articles **Creating user-defined attributes for Civil points** and **Working with Symbols and Labels**.

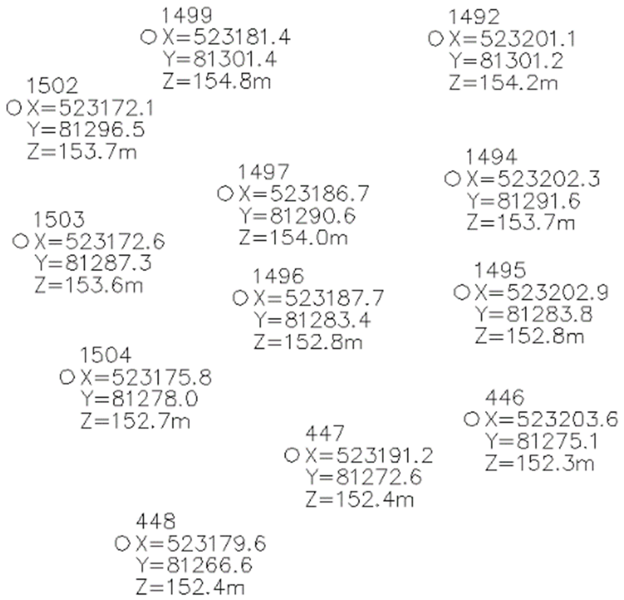
3 **LUPREC system variable:**

Use the LUPREC system variable to set the precision of the displayed coordinates for Civil points in the drawing. Type **0** in the Command line to the display coordinates without decimals, and **8** to display the maximum number of decimals (8). Use all other integers between 0 and 8 to set the appropriate precision.

LUPREC = 6



LUPREC = 1



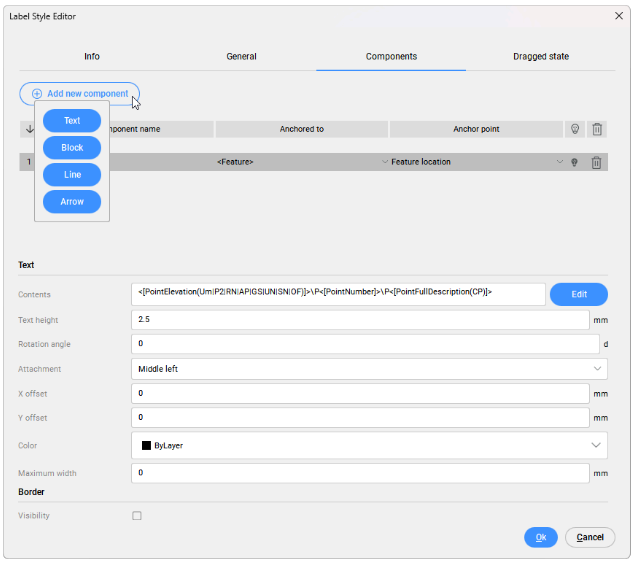
Note: Launch the REGEN command each time the LUPREC variable is changed to refresh the displayed precision of the Civil points coordinates.

8.4.2 Working with Civil Point Symbols and Labels

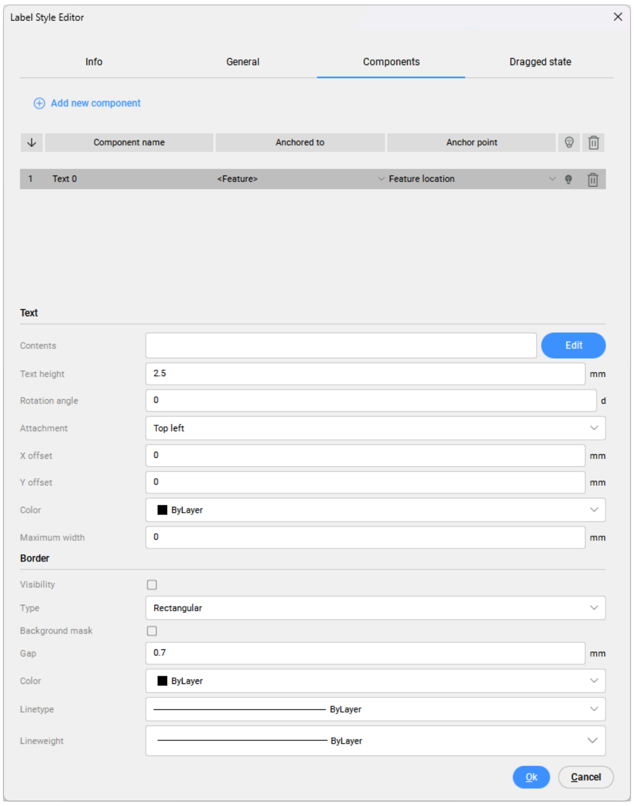
8.4.2.1 Creating a user-defined Label Style

Note: All the existing Civil point attributes can be displayed in the drawing. New attributes are added with the CIVILPOINTATTRIBUTES command. The procedure for adding a new attribute is described in the **Creating user-defined attributes for Civil points** article.

1. Open a new drawing with Civil points or Create/Import new civil points with CIVILPOINT command.
2. To create a new Label Style, open the **Civil Explorer** panel, select **Settings** tab, and under Civil Points right-click on Label Styles and select **Create** option. You can also Edit, Copy or Delete an existing label style.
3. When creating a new or editing an existing label style, the **Label Style Editor** dialog box will open.
4. Under **Info** tab write the Name and Description of the new/edited label style.
5. Under **General** tab, define general label properties including text style, label visibility, layer, orientation reference, forced insertion, plan readable, readability bias and decide if you want to flip anchors with text.
6. Under **Components** tab, add new Text, Block, Line and/or Arrow components of the Label Style.



7 Click **Add new component** button and select Text to display properties in the Civil point Label style. Define Text and Border style of the component.

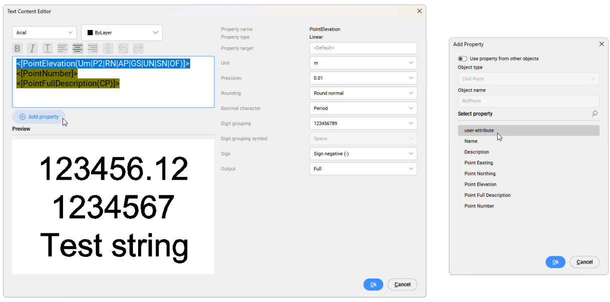


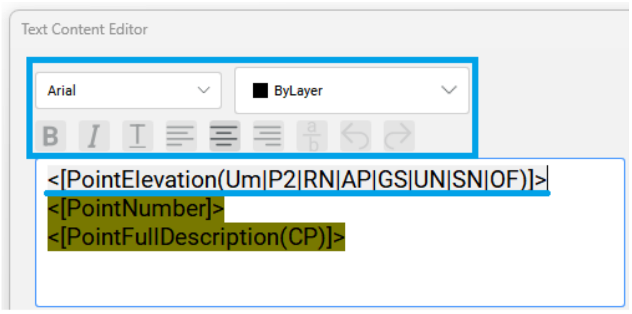
To edit the text content of the label, click the **Edit** button to open the **Text Content Editor** dialog box.

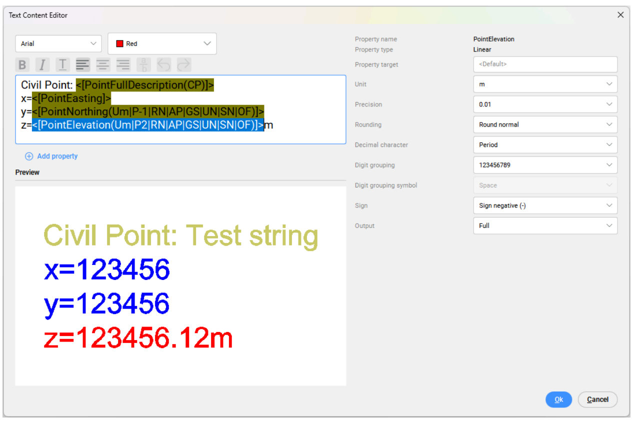
- The following attributes are used in the example: Point Full Description, Point Easting, Point Northing, and Point Elevation.
- To add new property in **Text Content Editor** dialog box, click **Add property** button.
- In **Add property** dialog box you can choose to display Civil Point attributes in Civil Point Label

style as well as properties from other Civil object (Surface, Horizontal Alignment, Vertical Alignment).

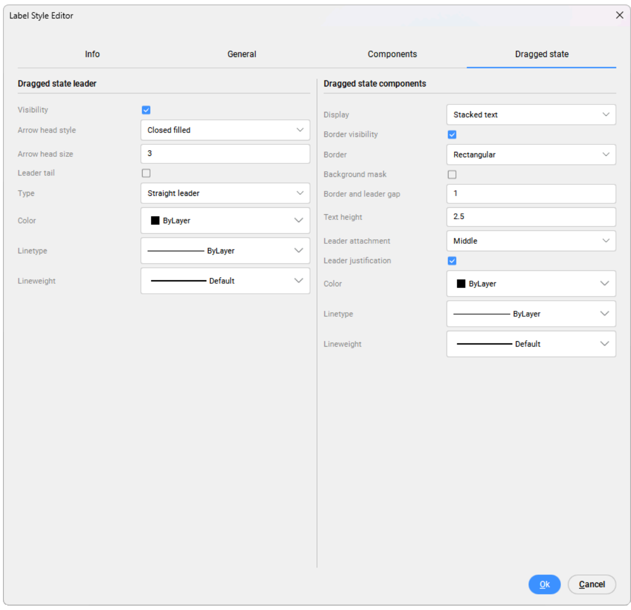
Note: Toggle **Use property from other objects** to On and select the property you want to display in the Civil point Label style.



- Click the desired attribute in the text editor field and set font, color, style and position of text.
- 

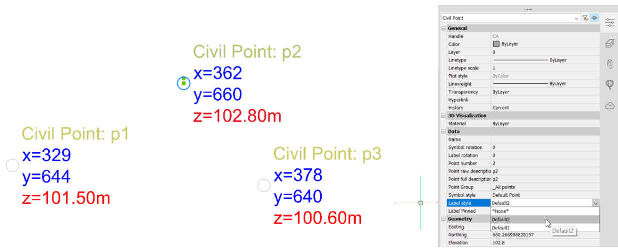
- By clicking on the individual property in the text editor field, you can specify additional settings for that property on the right side of the dialog box.
- The order of the properties can be rearranged by copying and pasting each property line in the desired place.
- Insert new text as prefix or suffix. To add a prefix, type the z = before the < character. To add a suffix, type m after the > character. For example, write z=<[PointElevation(Um|P2|RN|AP|GS|UN|SN|OF)]>m to do display points elevation.
- 

8 In **Label Style Editor** dialog box select **Dragged state** tab to define dragged state leader and dragged state components.



9 Confirm changes by selection OK in **Label Style Editor** dialog box.

10 Assign Label style to selected Civil points. Select the desired Civil points in the drawing and choose the appropriate Labels style from the drop-down list in the **Properties** panel.



8.4.2.2 Creating a user-defined Symbol Style

1 To create a new Symbol Style, open the **Civil Explorer** panel, select **Settings** tab, and under **General** settings right-click on Symbol Styles and select **Create** option. You can also Edit, Copy or Delete an existing symbol style.

2 When creating a new or editing an existing label style, the **Symbol Style Editor** dialog box opens.

3 Under **Info** tab write the Name and Description of the new/edited symbol style.

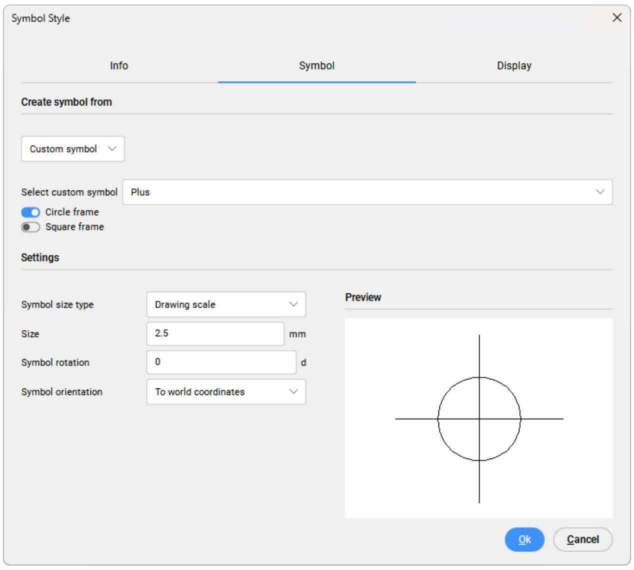
4 Under **Symbol** tab, select if symbol will be defined as Point object, Custom Symbol or as Block and define its settings.

- **Point object**: Uses a point object (CAD point entity) as a basis for the Symbol Style. You can define Symbol rotation (degrees) and Symbol orientation (based on object, view, or to world coordinates).
- **Custom symbol**: Allows you to create a symbol style from one of the custom symbols (None, Dot, Plus, X Mark, or Line). The custom symbol can be inside a circle and/or square frame. There are additional settings to define symbol size type, size, symbol rotation and orientation.
- **Block**: Allows you to select from a list of all point symbols (markers) that are defined by a

block and present in the drawing. There are additional settings to define symbol size type (Absolute scale, Drawing scale, Fixed scale, Absolute units or Relative to screen). The process of creating and saving blocks is described in **Creating blocks** article. More information about editing blocks is available in **Editing blocks** article.

5 **Define symbol style Settings**.

Different symbol types have different settings available.



- The **Symbol rotation** parameter specifies the point symbol rotation angle.
- The **Symbol orientation** parameter controls the orientation of the symbol on user-defined coordinate system (based on Object, View, or To World Coordinates).
- The **Symbol size** type parameter determines how the point symbol is scaled. The following methods are available:
- **Drawing units**: the size of the point symbol is determined by the parameter Symbol size.
- **Fixed scale**: the actual dimensions of the **Symbol block** multiplied by the value of the Symbol size are taken.
- **Absolute units**: the point symbol size is specified as the product of the **Symbol Size** parameter and the **Custom Scale** parameter on the **Viewport**.
- **Relative to screen**: the size of the symbol is determined as a user-specified percentage of the drawing screen size and its maximum size.

Note: The options **Symbol size** type and **Size** are not available when starting from a Point object. Set the point style with the PDMODE variable.

6 Under **Display** tab, specify the display of your symbol component. Specify Layer settings for Plan and Model view.

7 Confirm changes by clicking OK in **Symbol Style** dialog box.

8 **Assign Symbol style** to selected Civil points. Select the desired Civil points in the drawing and choose the appropriate Symbol style from the drop-down list in the Civil Point Properties panel.

8.4.3 Working with Point File Formats

8.4.3.1 Overview

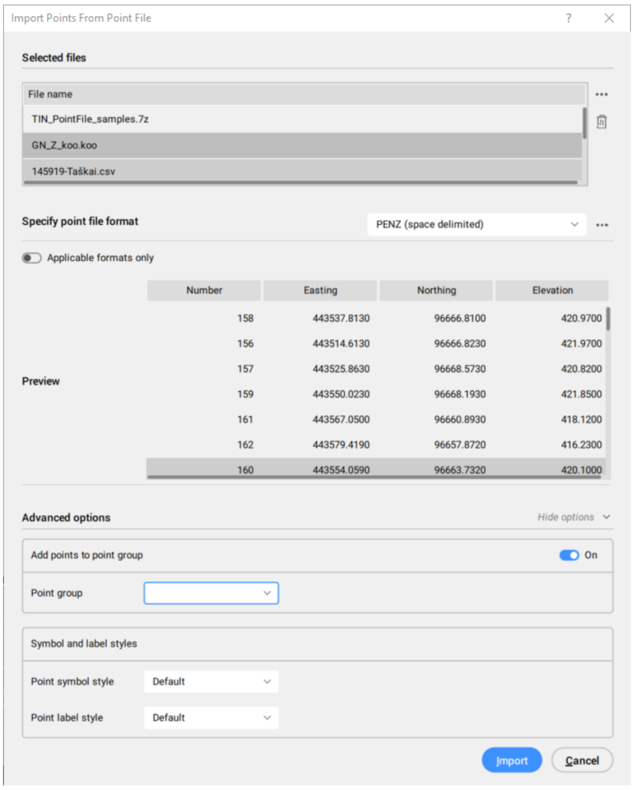
This article describes the procedures for importing Civil points from ASCII file and how to manage point file formats.

8.4.3.2 Importing Civil Points from file

1 Launch the CIVILPOINT command.

2 In the Command line type **I** or select the **Import points from file** option.

3 In the **Import Points From Point File** dialog box, choose the point data file that you want to import, or select multiple files to import all at once.



4 Under **Specify point file format**, select the appropriate point file format of your ASCII file or click the three dots from the right side to open the **Manage Point File Formats** dialog box.

5 The Preview window will automatically update once the right point file format is selected.

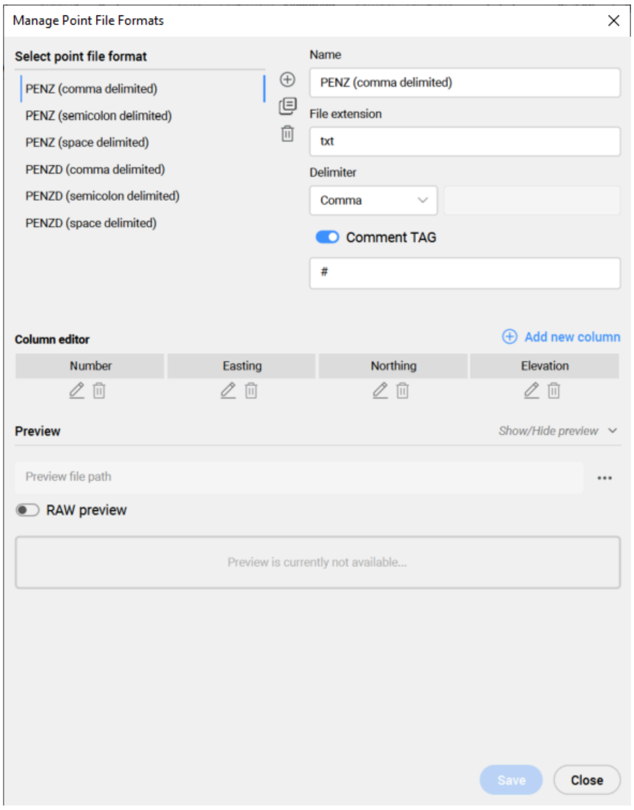
6 Set the **Add points to point group** button to On and choose a **Point group** name.

7 Set the symbol and label styles other than the default ones.

8 Click **Import** button to import Civil points.

Note: If no filter is specified or no point group is selected, the points are placed in the **_All points** group by default.

8.4.3.3 Manage Point File Formats



The **Manage Point File Formats** dialog box allows you to create, edit, and delete point file formats.

There are two ways to access the **Manage Point File Format** dialog box:

- Through the process of importing Civil points from the ASCII file, described in the section above. In **Import Points From Point File** dialog box, click the three dots from the right side of the **Specify point file format** section to open the **Manage Point File Formats** dialog box.
- Go to **Settings** tab of the **Civil Explorer** panel and double click (or right-click and select **Edit**) on **Point File Formats** section.

8.4.3.4 Delete Point File Format

In the **Manage Point File Formats** dialog box select the file format you wish to delete and click on the delete icon to remove the selected point file format.

You can also **Delete** point file formats directly from **Civil Explorer** panel. In the **Settings** tab of the **Civil Explorer** panel right-click the Point File Format you wish to remove, and select **Delete**.

8.4.3.5 Create a user-defined point file format

To create a user-defined point file format, you can:

- Add new point file format.
- Duplicate point file format.
- Edit existing point file format.

For the new point file format you have to specify the following:

1 Specify the point file format **Name**.

   Note: A warning message appears when the name of the new point file format is empty or already exists.

2 Specify the **File extension** of the ASCII file.

3 Select the **Delimiter** used in your ASCII file. You can choose between space, tabulator, comma, dot, semicolon, or define a custom delimiter.

4 Toggle to enable the **Comment TAG**. The comment tag can be set in the box bellow.

5 Define columns in the **Column Editor**:
   - **Rename** the column by selecting the edit icon. In the **Column Editor** dialog box you can choose between **Predefined column names** (<skip>, Easting, Northing, Elevation, Name, Number, Raw Description, Full Description), or use a **User defined column name**.
   - **Remove** column by selecting the bin icon.
   - Press the **Add new column** button to open the **Column Editor** dialog box, where you can choose the **Predefined column names** or use the **User defined column name**.

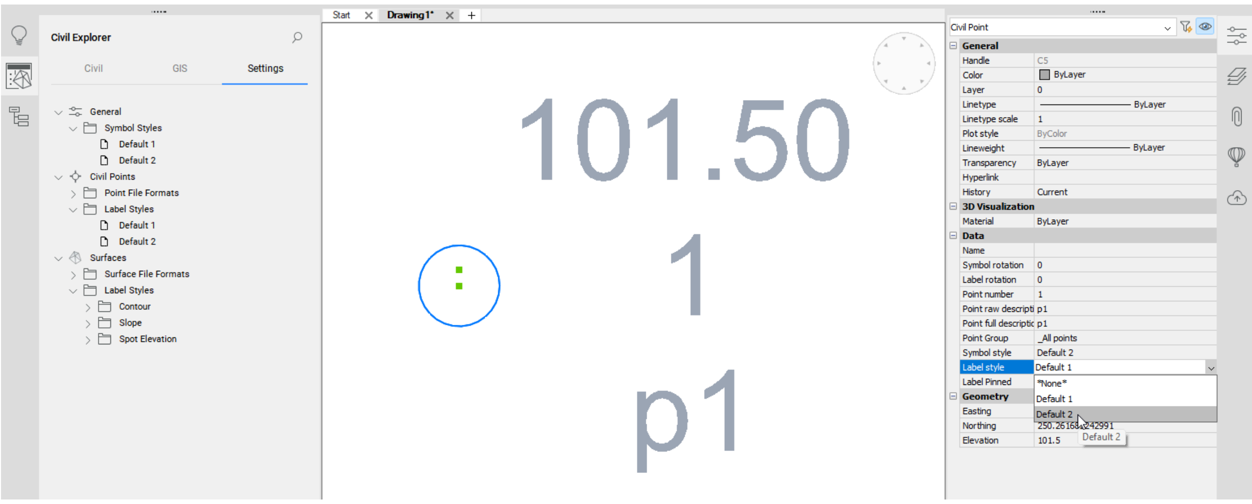
6 It is possible to use the **Preview** option. In the Preview file path, open the ASCII file. You can view the data in its RAW/original state or as arranged in columns based on the settings in the Column Editor.

7 Select **Save** and **Close**.

8.4.4 Understanding Civil point Symbols and Labels

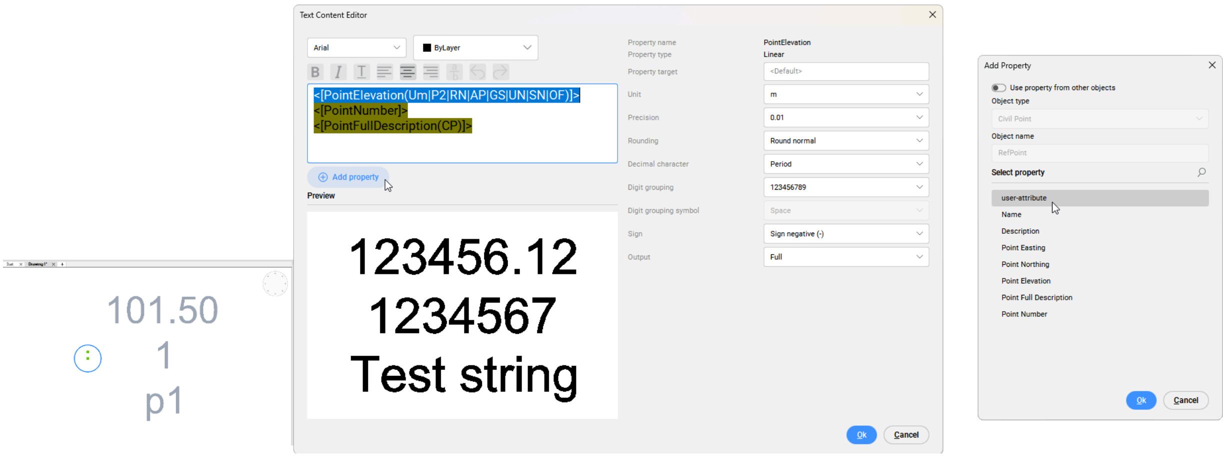
8.4.4.1 About Civil point Symbols and Labels

You can assign a unique label and symbol style to each Civil point in the drawing. The Civil Point **Label Style** and Civil Point **Symbol Style** are specified in the properties. You can define Label and Symbol Style in the **Settings** tab of **Civil Explorer** panel.



The predefined label style specified by the **Label Styles**, shown on the picture below, includes the following attributes:

- **Point Elevation**, defined by text: <[PointElevation(Um|P2|RN|AP|GS|UN|SN|OF)]>
- **Point Number**, defined by text: <[PointNumber]>
- **Point Full Description**, defined by text: <[PointFullDescription(CP)]>



The properties of attributes can be edited on the right side of **Text Content Editor** dialog box, or by editing abbreviations in the brackets. For example, in case of Point Elevation:

Um – **U**nits: **m**eter

P2 – **P**recision: **2** decimals

RN – **R**ounding: **N**ormal

AP – Decimal Character (**A**rithmetic point): **P**eriod

GS – Digit **G**rouping Symbol: **S**pace

UN – Digit Grouping/**U**ngrouping: 123456789

SN – **S**ign: Sign **N**egative (-)

OF – **O**utput: **F**ull

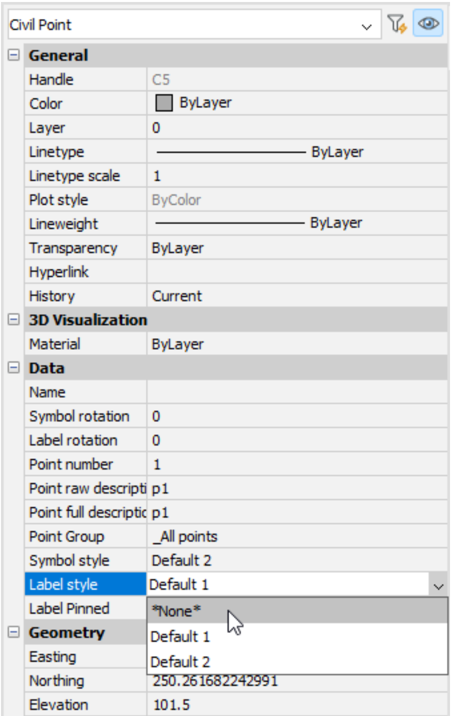
The changes made in attribute settings are automatically updated in the text editor and vice versa.

The process of creating and editing new Label and Symbol Styles are described in the article **Working with Civil Point Symbols and Labels**.

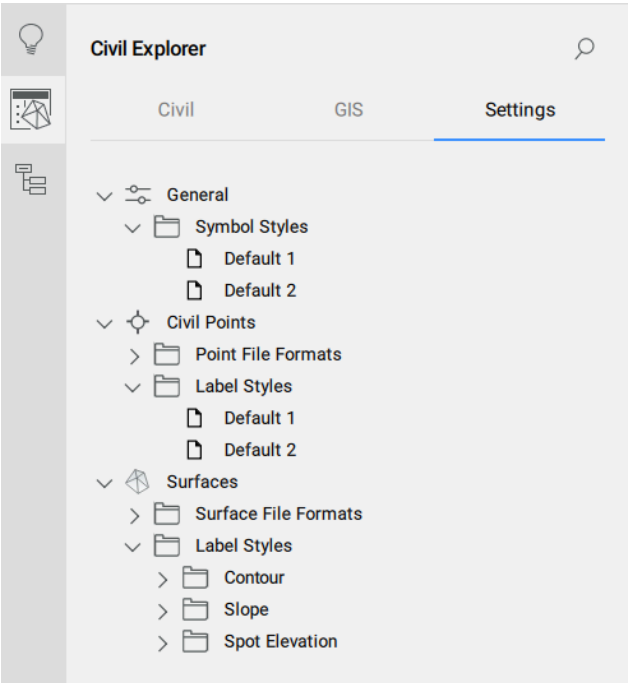
The process of creating a new user-defined attributes for Civil points is described in the article **Creating user-defined attributes for Civil points**. The content of **Text Content Editor** and **Add property** dialog boxes are described in the articles **Text Content Editor dialog box** and **Add property dialog box.**

Note: In the **Add property** dialog box you can select Civil Point properties as well as use properties from another object (Surface, Horizontal Alignment and Vertical Alignment).

For each point or point group, you can select the appropriate label or symbol style from the Symbol Style and Label Style drop-down list. You can set the Civil point so that it does not contain any labels or symbols. You can do this by selecting the **None** option the drop-down list of the Symbol and Labels style properties.



You can specify various settings for the selected Symbol and Label Styles in the **Civil Explorer** panel.



8.4.4.2 Symbols Styles

List of available Symbol Styles can be seen in Civil point properties panel and in the **Settings** section of **Civil Explorer**. In the **Settings** section of **Civil Explorer**, right-click a Symbol style to **Create**, **Edit**, **Copy** or

Delete symbol styles. Selecting **Create**, **Edit** or **Copy** options will open the **Symbol Style** dialog box ,which allows you to define a new symbol style.

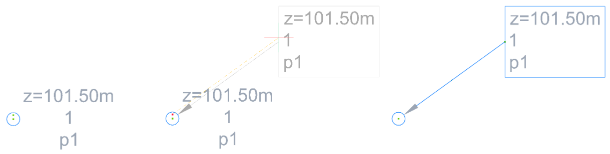
8.4.4.3 Label Styles

List of available Label Styles can be seen in Civil point properties panel and in the **Settings** tab of **Civil Explorer** panel. In the **Settings** tab of **Civil Explorer** panel, right-click a **Label style** to **Create**, **Edit**, **Copy** or **Delete** label styles. Selecting **Create**, **Edit** or **Copy** options will open the **Label Style Editor** dialog box which allows you to define a new label style in general and dragged state. To define attributes shown in the Label, open **Text Content Editor** dialog box and select **Add property** button. In **Add property** dialog box you can select Civil Point properties as well as use properties from another object (Surface, Horizontal Alignment, Vertical Alignment).

8.4.5 Understanding Civil point leader and components in dragged state

8.4.5.1 Overview

Civil points contain two grips. The primary grip is placed at the insertion point, while the alternative grip is placed vertically above the insertion point and it is always attached to Label style. You can use the alternative grip to drag the point label away from the insertion point and to create a **Leader**.

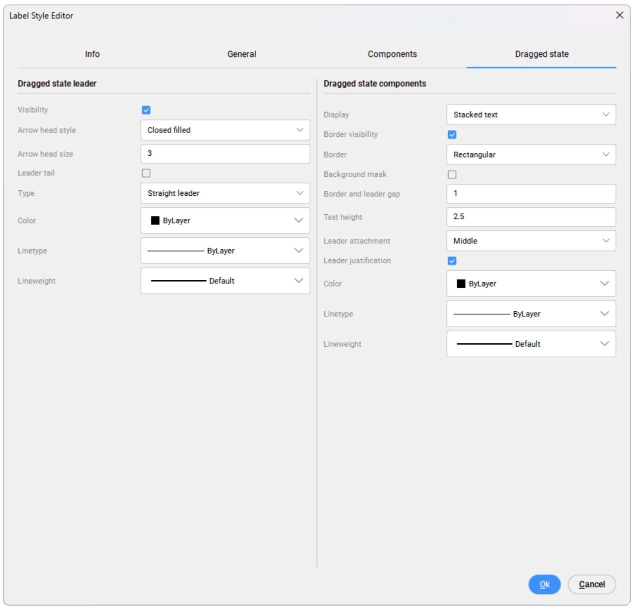


Civil point leader style is defined inside **Label Styles** and can be edited in the **Dragged state** tab in **Label Style Editor** dialog box. The **Label Style Editor** dialog box can be opened from the **Settings** tab of the **Civil Explorer** panel.

Note: The process of creating and editing new Label and Symbol Styles are described in the article **Working with Civil Point Symbol and Labels**.

8.4.5.2 Dragged state settings

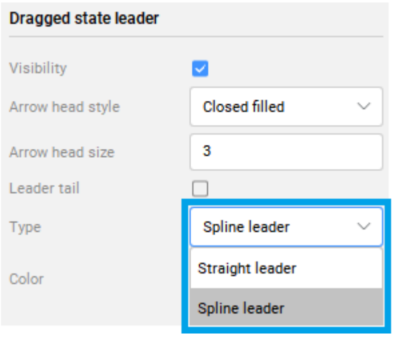
In the **Label Style Editor** dialog box, you can define the style of the leader and label components when they are in a dragged state by navigating to the **Dragged State** tab.



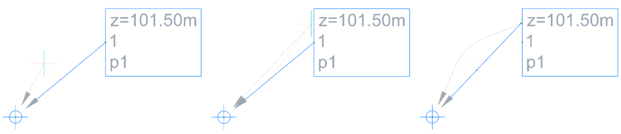
Dragged state leader

The visibility of the leader can be toggled ON or OFF. You can choose between different types of arrow heads, that can be selected from the **Arrow head style** drop-down list. The arrow head size can be defined in the **Arrow head size** field. The **Leader tail** can also be toggled ON or OFF. When the leader tail is turned on, its length is equal to the arrow head size.

You can define the leader type as either a **straight line** or a **spline**. To create a spline curve, change the leader type on the Dragged state tab in the **Label Style Editor** dialog box to: **Spline Leader**.



Then click on the **+** sign in the middle of the leader and drag its end to the desired endpoint. Selecting and dragging the **+** sections of the leader will create curves, while selecting **-** will remove sections of the spline curve.



The color of the leader can be selected from a **Color** drop-down list. **Linetype** of the leader can be defined by layer, by block or from the list of linetype styles. The lineweight can be defined from the **Lineweight** drop-down list.

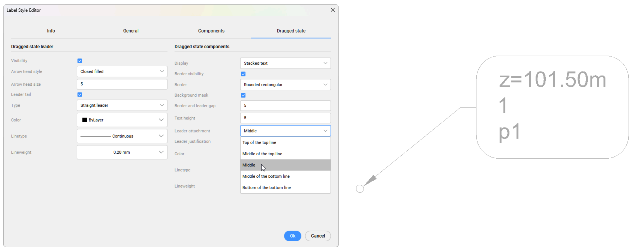
Note: All the changes are dynamically displayed in the drawing.

Dragged state components

Dragged state components can be displayed as **Stacked text** or **As composed**.

When components are displayed as **Stacked text**, it means their style will be defined under a **Dragged State** tab. When they are displayed **As composed**, it means the style is the same as it is defined under the **Components** tab. If you want the label style to be different in dragged state, select **Display** option **Stacked text** and define the rest of its properties.

Border visibility can be toggled ON or OFF. You can choose between three different **Border** shapes: **Rectangular**, **Rounded rectangular** or **Circle**. To improve the visibility of the label, you can turn on the **Background mask**. The border and leader gap of the label can be defined in the **Border and leader gap** field. The text height is defined in **Text height** field.



The position of the label components is based on the Leader attachment parameter. To change the position of attachment, select one of the options form a drop-down list of **Leader attachment position** Toggle **Leader justification** to change components alignment with the movement of the leader. The color of the components and the border can be selected from a **Color** drop-down list. **Linetype** of the border can be defined by layer, by block or from the list of linetype styles. The lineweight of the border can be defined from the **Lineweight** drop-down list.

Note: All the changes are dynamically displayed in the drawing.

8.4.6 Creating user-defined attributes for Civil points

8.4.6.1 Creating a user-defined attribute

1 Launch the CIVILPOINTATTRIBUTES command.

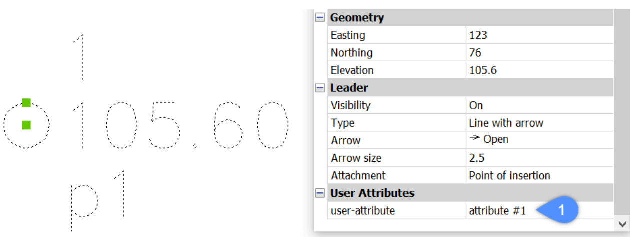
2 Select the Civil points in the drawing to which you want to add a user attribute.

3 Select the **Add attribute** option in the Command line.

4 Enter the attribute key (attribute name).

5 Enter the attribute value.

Note: The new user attribute is listed in the Civil point **Properties** panel (1). You can also edit the attribute value here.



8.4.6.2 Displaying a user-defined attribute on selected Civil points in a drawing

This section describes the procedure to add a new user attribute (attribute name: user-attribute) to an existing Civil point Label Style. You can add the user attribute to a new user Label Style in the same way.

Note: Steps of creating a user-defined Label style or editing an existing one is described in the article **Working with Symbols and Labels**. The content of the Label Style Editor is described in the article **Label Style Editor** dialog box.

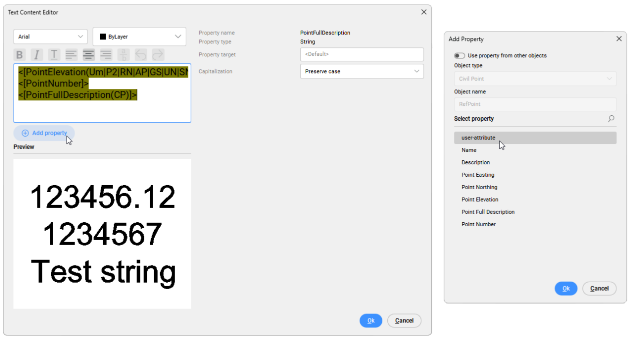
To add a user-defined attribute on selected Civil points:

1 Go to **Civil Explorer** panel and select **Settings** tab.

2 Under Civil Points, extend **Label Styles** and select the style you want to edit. Right-click the style and select **Edit** option.

3 In **Label Style Editor** dialog box go to **Components** tab and select **Edit** button to open **Text Content Editor** dialog box.

4 In **Text Content Editor** dialog box select **Add property** button. Select a property you wish to add and select **Ok**.



Note: In **Add property** dialog box you can select Civil Point properties as well as use properties from another object (Surface, Horizontal Alignment and Vertical Alignment). For more details, see the **Text Content Editor** dialog box article.

8.4.6.3 Displaying Properties from other Civil objects in Civil points

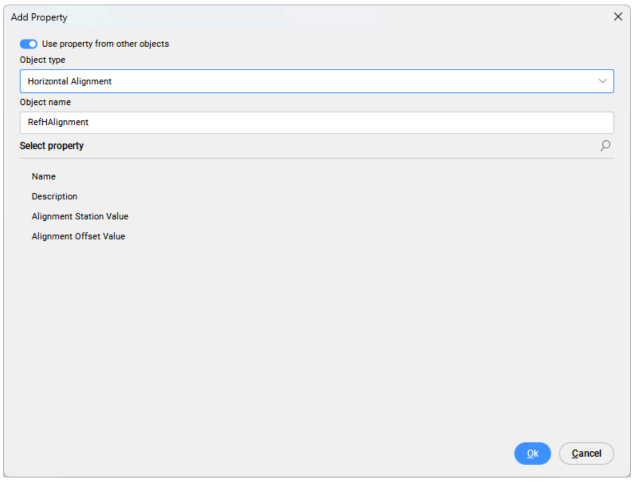
1 In the **Add Property** dialog box, toggle ON the **Use property from other objects** option.

2 Select the Civil object type.

3 Type the object name.

4 Select the property of the selected Civil object to be displayed in the Civil points labels.

5 Click OK button in the **Add Property** dialog box.



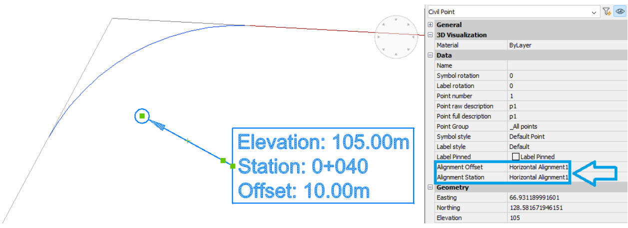
6 Click the **Ok** button in the **Text Content Editor** dialog box.

7 Click the **Ok** button in the **Label Style Editor** dialog box.

8 Select Civil points in a drawing and open **Properties** panel.

9 In the **Properties** panel, select the appropriate Civil objects to read the property values and display them in a Civil point Label style.

The example below shows the display of the **Station** and the **Offset** of the Civil point from the Alignment named Horizontal Alignment1.



8.5 Working with gradings

8.5.1 Overview

In BricsCAD, graded surfaces are created using the GRADING command.

Grading is a set of processes to maintain a base level for construction works or TIN surface design by defining a slope or adjusting the offset of a grading. For instance, creating parking areas, recreation reserves or road and railway designs.

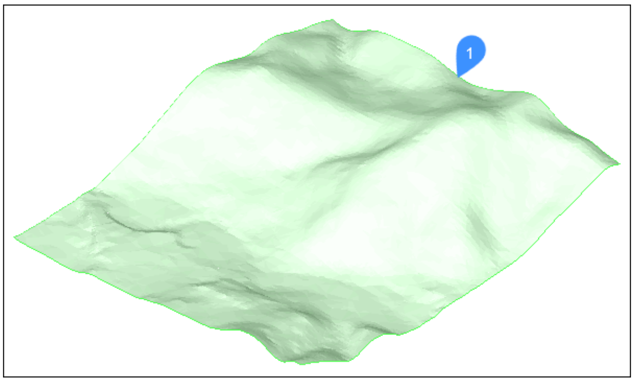
Two projection methods **Slope to surface** and **Offset to slope** are supported:

- **Slope to surface** needs an input entity (for example, polyline, line, arc, circle), a surface and slope to grade the surface.
- **Offset to slope** needs an input entity, offset and slope.

You can edit graded surfaces with the GRADINGEDIT command.

8.5.2 Grading a surface to create a pond

1 Open a drawing that has a topographical surface (1).

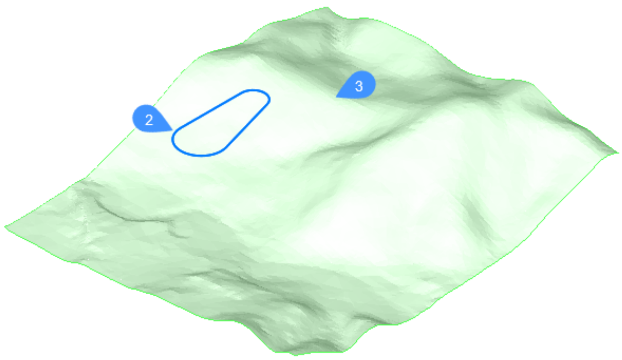
1 

2 Launch the GRADING command.

3 Select the input entity (2) that defines the footprint area of your pond. Different types of opened and closed entities are accepted as input entities, such as lines, polylines, 2D/3D polylines, arcs, 3D Alignments, and gradings.

4 ***Note:*** Before selecting the target surface, you can change to **slOpe/offset** projection method which requires an input entity, offset and slope.

5 Select the target surface (3).

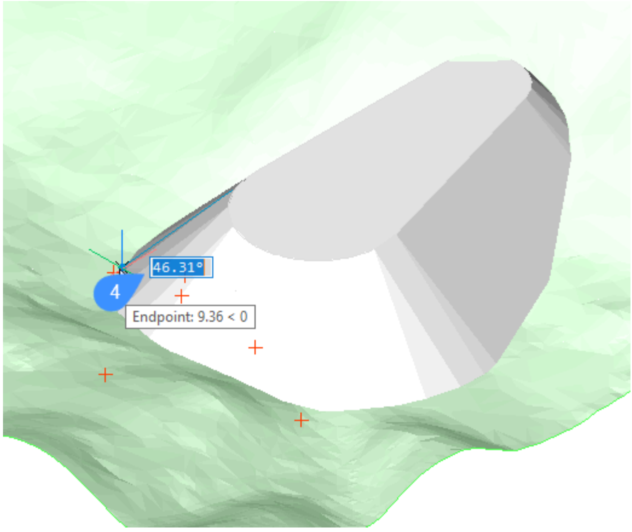
6 

7 Select a part of the entity by picking the start and end points, or the **Entire length** of the entity.

8 Define the slope of the grading.

9 ***Note:*** The grading is attached to the input entity, and it is connected with the target surface (TIN surface). Moving the mouse in/out dynamically increases/decreases the slope of the grading.

Note: If DYNMODE variable is set to 3, you can also input the slope angle directly from keyboard (4).



10 Press ENTER to accept the slope of the grading.

8.5.3 Displaying the properties of a graded surface

The properties are displayed in the **Properties** panel after selecting the grading.


Grading
General
3D Visualization
Data
Name | Grading
Description
Grading type | Slope to surface
Input Entity | 14C
Target surface | Surface2
Grading transition | Off
Regions | 1
Cut slope | 47.76
Fill slope | 47.76
Region start | 0
Region end | To the end
TIN Visibility
Boundary line | Off
Points | Off
Triangles | On
Contours | Off
Major contours interval | 10 m
Minor contours interval | 1 m
Major contours color | Red
Minor contours color | Green
TIN Analysis
Elevations | Off
Create ranges by | Number of ranges
Color scheme | Spectrum
Above datum color scheme | Greens
Below datum color scheme | Reds
Display as | 2D Triangles
Range interval | 10
Number of ranges | 25
Datum | 0
Slopes | Off
Color scheme | Spectrum
Display as | 2D Triangles
Number of ranges | 25
TIN Statistics
Number of points | 1195
Number of triangles | 1368
Minimum elevation | -21.53 m
Maximum elevation | 88.94 m
2D area | 89368.21 m²
3D area | 0.12 km²
Grading settings
Side | Right
Infill | On
Boundary | Off
Grading statistics
Fill volume | 4729059.50 m³
Cut volume | 1402.53 m³
Net volume | 4727656.96 m³

Grading type
Displays which projection method is used to grade the surface (**Slope to surface** or **Slope and offset**).

Input entity
Displays the handle number of the input entity.

The input entity can be changed by pressing the three dots button () on the right side of the **Input entity** option and selecting another input entity on the screen.

Target surface
Displays the name of the target surface.

The target surface of a grading can be changed by pressing the three dots button () on the right side of the **Target surface** option and selecting another target surface on the screen.

Grading transition
Displays the grading transition status.

Cut slope
Displays the cut slope (start and end).

Fill Slope (start and end)
Displays the cut slope.

Region start
Displays the start of the region.

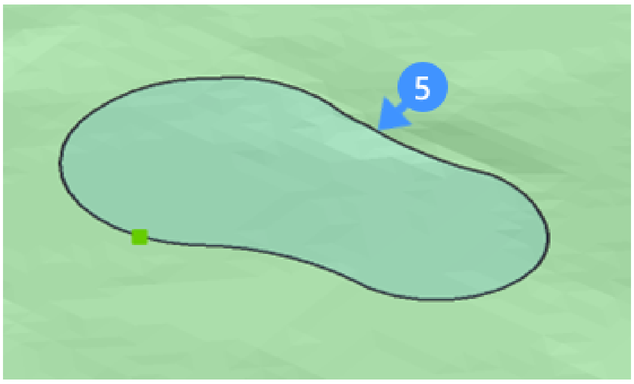
Region end
Displays the end of the region.

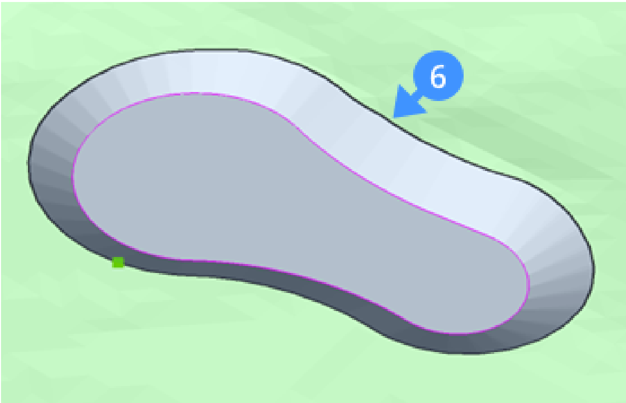
As the grading is derived from the TIN surface, you have access to TIN Visibility, TIN Analysis and TIN Statistics properties. You can also calculate the TIN Volume Surface and extract the TIN surface.

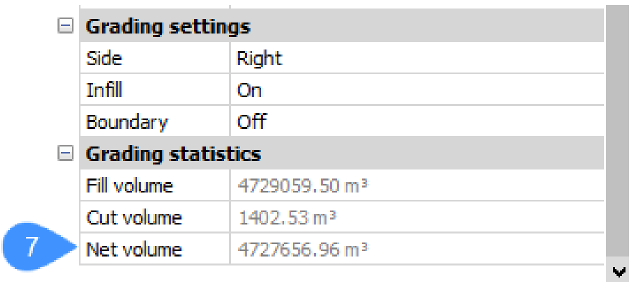
Side
Controls on which side of the input entity the grading is drawn (left or right).

Infill
The internal area of entities is filled if this option is turned on. Valid for closed entities only (circle, closed polyline).

Boundary
Controls the boundary of the grading. When the grading boundary is on, that boundary is added as a **hide** boundary to the target surface.
The following images show when the grading boundary of the pond is off (5) and on (6). Based on the grading boundary, the net volume for the body of the pond is calculated and can be found in the properties panel (7).







Grading settings	
Side	Right
Infill	On
Boundary	Off
Grading statistics	
Fill volume	4729059.50 m³
Cut volume	1402.53 m³
Net volume	4727656.96 m³

Fill volume
Displays the fill volume for a body/shape limited with the target surface and graded surface.

Cut volume
Displays the cut volume for a body/shape limited with the target surface and graded surface.

Net volume
Displays the net volume for a body/shape.

8.5.4 Grading a surface to create a platform

1 Open a drawing that has a topographical surface (8).

1

2 Launch the GRADING command.

3 Select the entity that defines the footprint area of your platform (9).

Note: Before selecting the target surface, you can change to the **slOpe/offset** projection method which requires an input entity, offset and slope.

4 Select the target surface (10) to create a graded area.

5

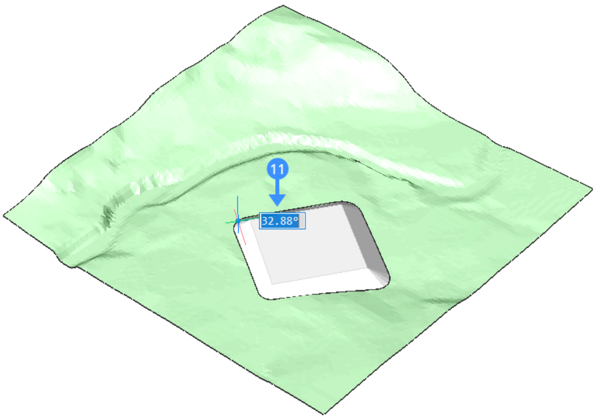
6 Select the **Entire length** of the input entity.

7 Define the slope of the grading.

8 ***Note:***

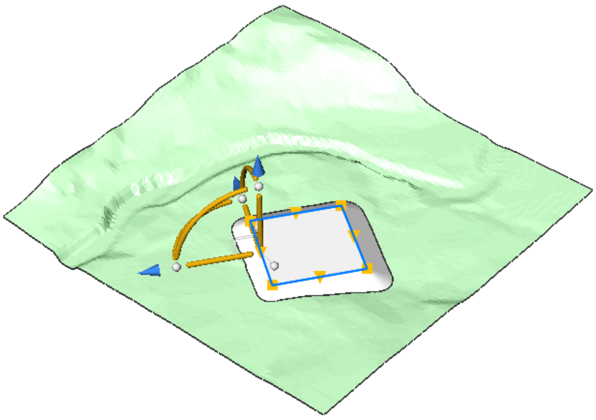
- The grading is attached to the input entity, and it is connected with the target surface. Moving the mouse in/out dynamically increases/decreases the slope of the grading.
- If DYNMODE variable is set to 3, you can also input the slope angle directly from keyboard (11).

-

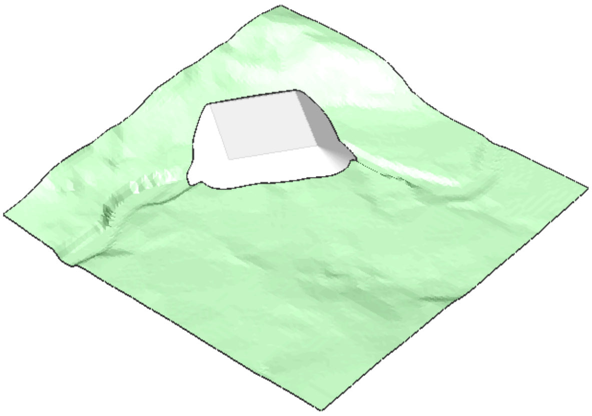


9 Press ENTER to accept the slope of the grading.

Note: You can easily move the platform that you created. Do this by using the manipulator on the entity from which you created the platform. You can move both horizontally and vertically.



The grading will automatically be adapted to its surroundings.



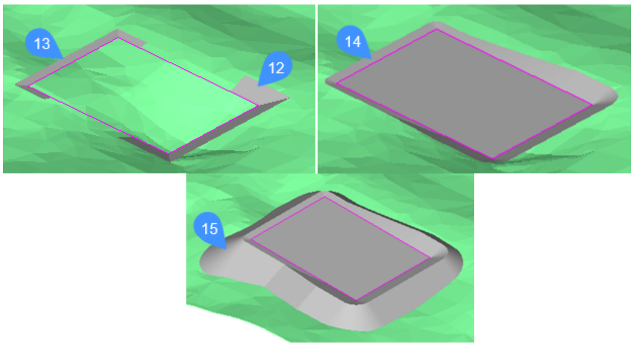
8.5.5 Editing a grading surface

With the GRADINGEDIT command, you can:

- Merge two gradings into one grading with several regions.
- Perform an (automatic) calculation for the transitions between gradings.
- Split a grading into multiple regions, so you can edit the slopes for each region separately.

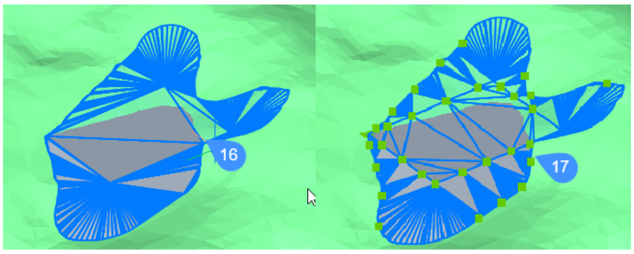
Merge steps:

- Merge Grading1 (12) and Grading 2 (13).
- Connect the grips to close the grading. The resulted grading is (14).
- On the basis of the resulting grading (14) create another grading, that is sloped to the TIN surface, and turn OFF its Infill parameter. The resulted grading is (15).



12: Grading 1
13: Grading 2
14: Merged Grading with connected grips
15: The resulted Grading with Infill parameter set to OFF

Split



16: Grading before splitting.
17: Grading after splitting. Each region can be edited by moving the grips.

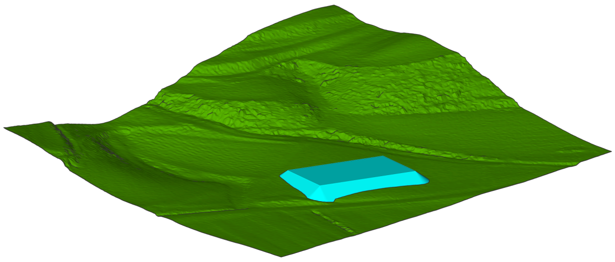
8.5.6 About balance grading volumes

The GRADINGBALANCE command allows you to balance cut and fill volumes within the specified tolerance. This is achieved by raising/lowering the elevation of the grading input entity and the result is grading that has a net volume around zero, depending on the set tolerance.

8.5.7 Adjusting the elevation of a grading to balance cut and fill volumes

1 Open a drawing that contains a TIN surface and a grading representing a building pad.

1

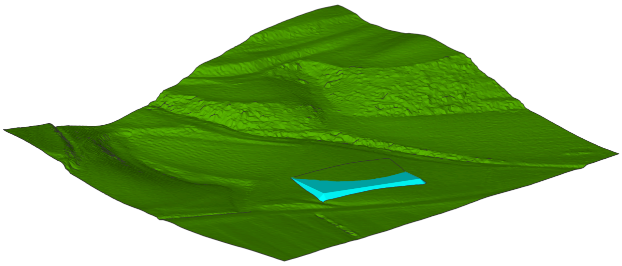


2 Launch the GRADINGBALANCE command.

3 Select the grading.

4 Enter the tolerance in percent (1-100). 1 means that the difference between the cut and fill volume will be less than one percent. 100 means that the difference between the cut and fill volume will be less than 100 percent.

5



In the **Properties** panel, you can see that the cut and fill volumes are now balanced.

Grading statistics	
Fill volume	670.77
Cut volume	673.19
Net volume	-2.42

8.6 Corridors

8.6.1 Modeling a Corridor

8.6.1.1 Commands

CORRIDOR, CORRIDOREDIT, CORRIDOREXTRACT

8.6.1.2 Overview

In BricsCAD, Corridors that follow a certain 3D alignment are created using the Corridor tool.

8.6.1.3 About Corridor

A Corridor is a BricsCAD Civil drawing object used for modeling three-dimensional linear objects such as roads, railway, retaining walls, and bridges. It principally consists of a 3D alignment of the linear object and its typical cross-section – which is called Corridor Template in BricsCAD.

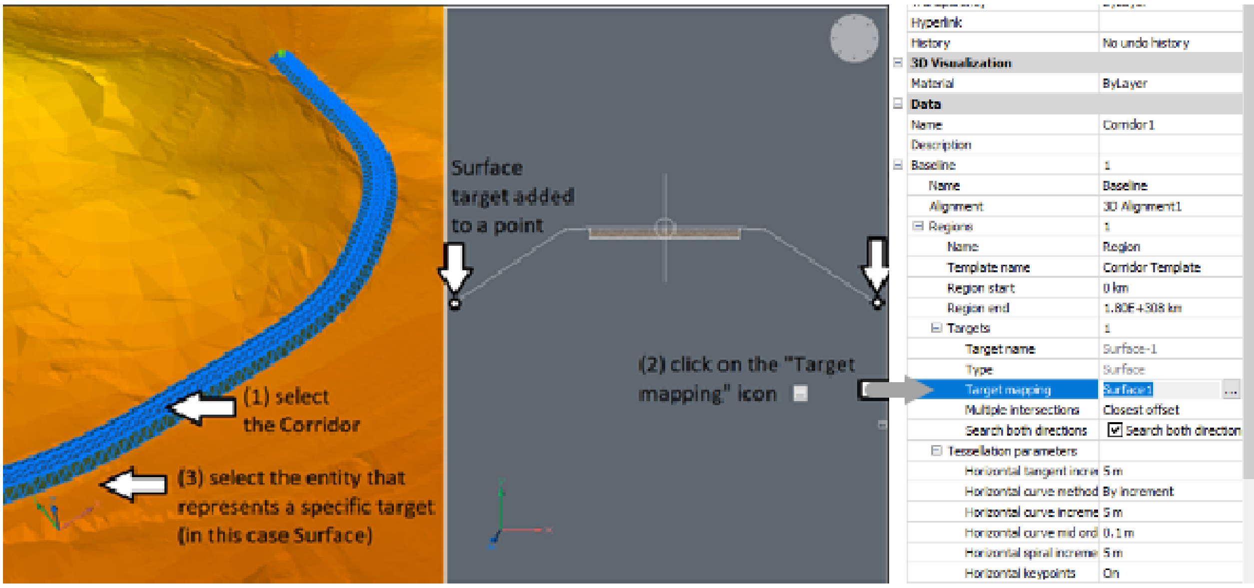
The CORRIDOR command enables you to create a Corridor, based on a Corridor Template, that follows a 3D alignment on a certain topographical surface.

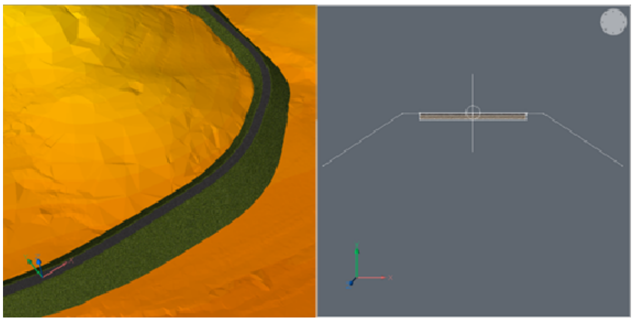
8.6.1.4 About Corridor Targets

When the geometry of an individual corridor template element must follow a specific surface, horizontal or vertical object, a target is created and added to the corresponding point on the corridor template element. The following target types can be defined: Surface, Horizontal and Vertical targets.

8.6.1.5 About Target Mapping

Once created, the targets must be mapped from the Corridor Template Element point to the Corridor model in the drawing. We do this by selecting an entity in the drawing that represents the individual target of our corridor. This process is called Target Mapping.





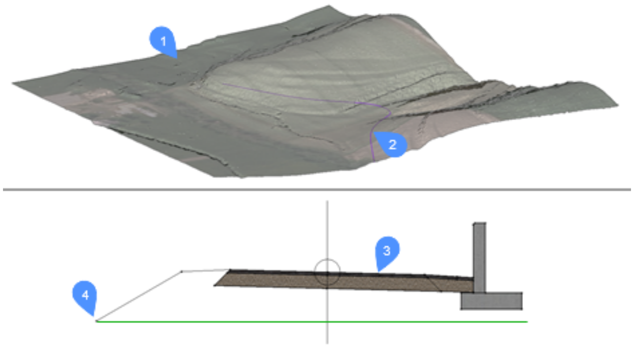
8.6.1.6 About Corridor Extract

The CORRIDOREXTRACT command is used to generate meshes, solids, polylines or outer boundaries from an existing Corridor.

8.6.1.7 Creating a road Corridor

1 Open the drawing file that contains the TIN Surface (1), 3D Alignment (2), Corridor Template (3) and Surface target added to a Template Element Point (4).

1 ***Note:***In the example shown in the figure, the Surface Target (4) is added to the Point of the Template Element, which represents the end point of the embankment.



2 Launch the CORRIDOR command in the Command line.

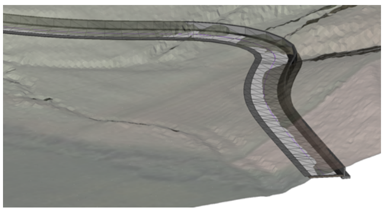
3 ***Note:*** You can find the Civil commands also in the Civil Toolbar, in the Menu and on the Ribbon.

4 Select 3D alignment (2) used as a corridor baseline.

5 Select the corridor template (3).

6 Enter region start station. Press ENTER to start from the beginning or type any station where you want your corridor to start.

7 Enter the region end station. Press Enter to create a corridor to the end of the selected alignment or type any station where you want your corridor to end.

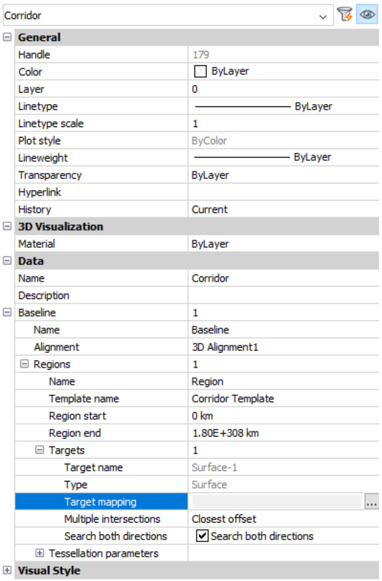


8.6.1.8 Mapping the Target from the Corridor template to the Corridor model (Target Mapping)

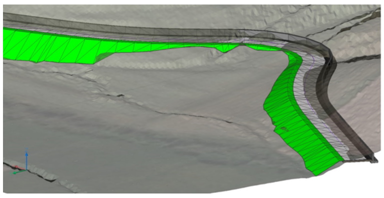
The described procedure continues after the previous procedure of creating a new Corridor.

1 Select the Corridor in a drawing.

2 Click the icon ... in the **Target mapping** field.



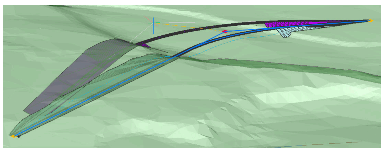
3 Select the entity in a drawing that represents the specific target, which is the TIN Surface in this example.

4 

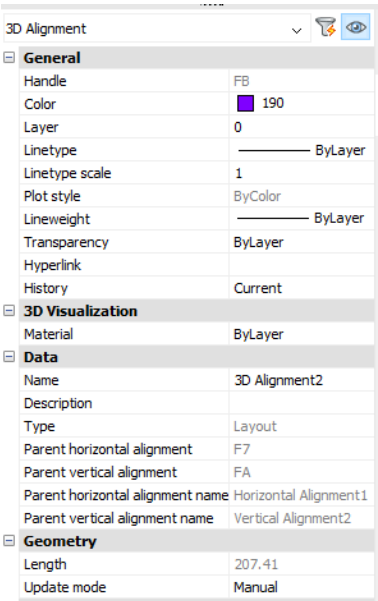
8.6.1.9 Editing the Corridor's horizontal and vertical position

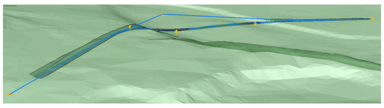
1 Open the drawing file that contains the corridor you want edit.

2 Select the 3D alignment that the corridor has as a reference and move the grip points to edit the corridor horizontally.

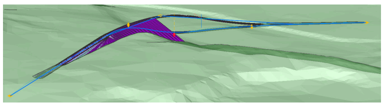


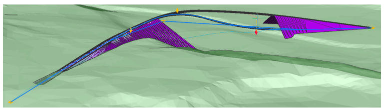
3 To edit the corridor vertically, select the 3D alignment that the corridor has as a reference and open the **Properties** panel. Set the **Update mode** to **Manual**. Extra grip points will appear.



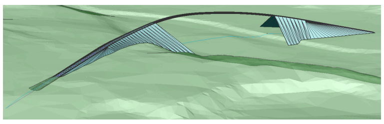


4 Move these grip points to edit the corridor vertically.





5 Press Enter to finish your edited corridor.



8.6.1.10 Splitting a Corridor region

1 Open the drawing file that contains the Corridor you want to edit.

2 Launch the CORRIDOREDIT command.

3 Select the Corridor model in a drawing.

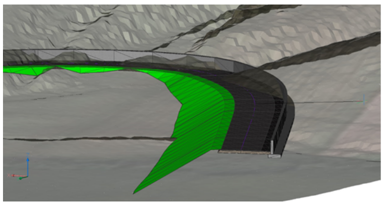
4 Select the option **Split Region** in the Command line.

5 Select the desired Corridor region you want to split.

Note: Corridor region can be selected by Index, by Name or by selection in a drawing.

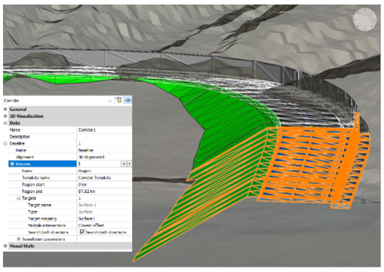
6 Select or enter station at which the region will be split.

7 *Note:* Specify all stations where you want to split the region.



8 Select or enter the station at which the region will be split.

Note: The original region is now split into two regions at the specified station.

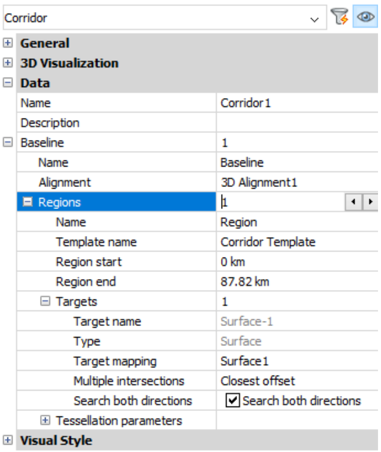


8.6.1.11 Replacing the Template element of a selected Corridor region

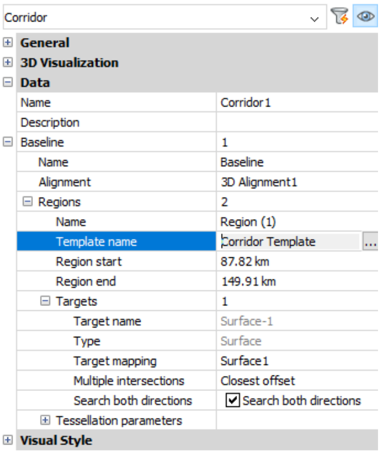
1 Open the drawing file that contains Template Elements and a Corridor you want to edit.

2 Open the **Properties** panel and click the Corridor model in a drawing.

3 Select the appropriate Region in the **Properties** panel for which you want to replace the Template Element.

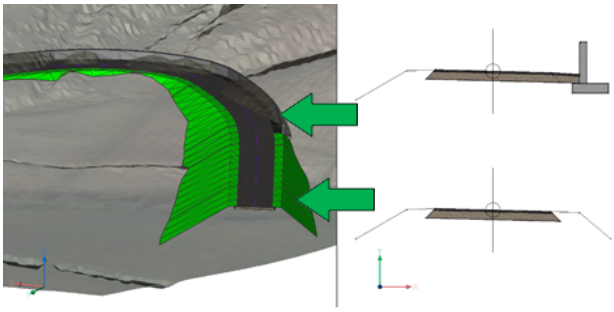


4 Click the button ... in the **Template name** field.



5 Select a new Corridor Template in a drawing.

6



8.6.1.12 Extracting a TIN Surface from a Corridor

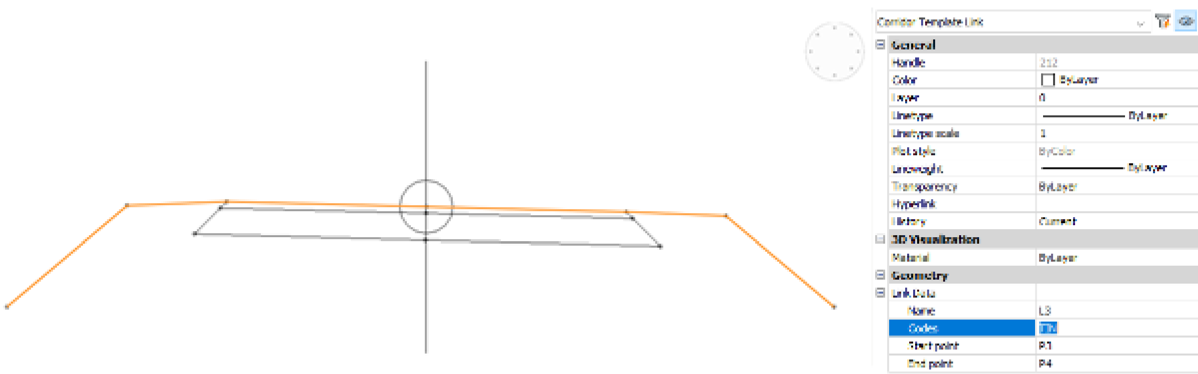
1 Open the drawing file that contains a TIN Surface, 3D Alignment, Template Element and a Corridor model.

2 Select the Corridor Template Element.

3 Select the Template Element Link you want to use for extracting TIN Surface.

4 ***Note:*** Hold the Ctrl button and select all required Template Element Links.

5 Enter the code name (i.e. TIN) in the **Codes** field, which is located under the **Link Data**.



6 Launch the CORRIDOREXTRACT command.

7 Select the option TIN from the command line.

8 Enter Links to Extract: **TIN**.

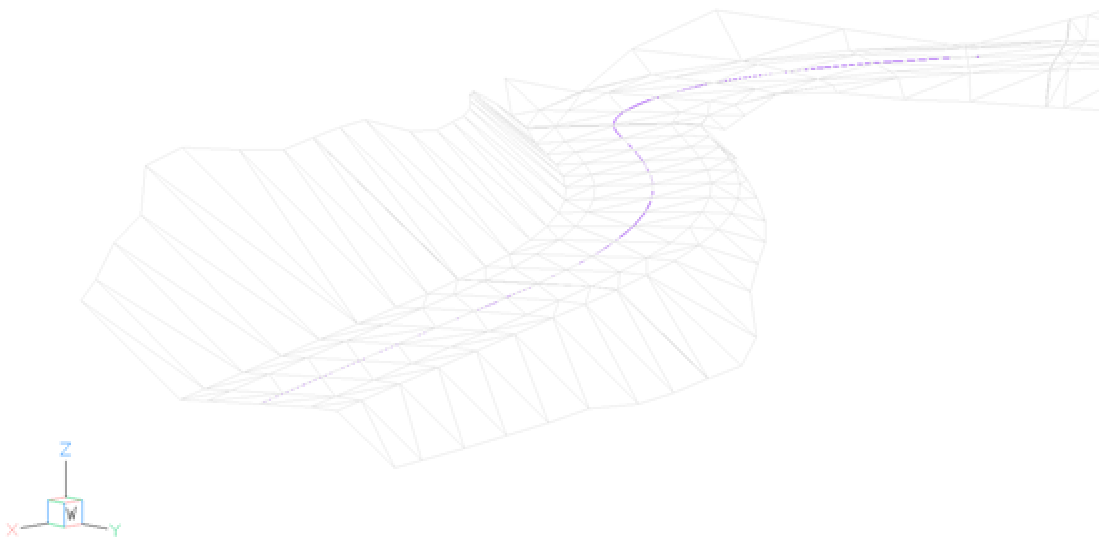
Note: Type **?** to display all the available link codes.

9 Add as Breaklines: **Yes**.

10 Enter another link to Extract or press Enter.

11 Use the outer edge as a surface boundary: **Yes**.

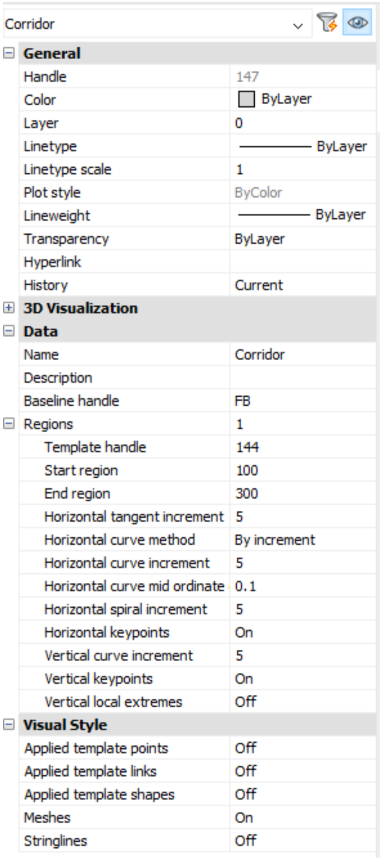
12 Create dynamic link: **Yes**.



8.6.1.13 Displaying the properties of a Corridor

1 Select the Corridor.

2 The properties of the Corridor are displayed in the **Properties** panel.



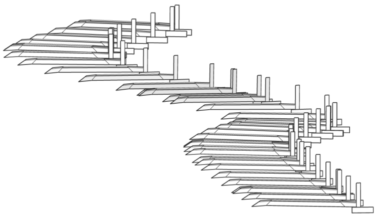
Corridor	
General	
Handle	147
Color	ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
History	Current
3D Visualization	
Data	
Name	Corridor
Description	
Baseline handle	FB
Regions	1
Template handle	144
Start region	100
End region	300
Horizontal tangent increment	5
Horizontal curve method	By increment
Horizontal curve increment	5
Horizontal curve mid ordinate	0.1
Horizontal spiral increment	5
Horizontal keypoints	On
Vertical curve increment	5
Vertical keypoints	On
Vertical local extremes	Off
Visual Style	
Applied template points	Off
Applied template links	Off
Applied template shapes	Off
Meshes	On
Stringlines	Off

Applied template points

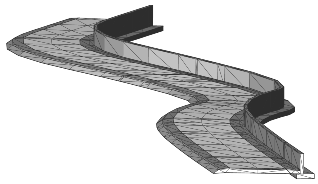
Represents the corridor with template element points.

Applied template links
Represents the corridor with template element links.

Applied template shapes
Represents the corridor with template element shapes.



Meshes
Represents the corridor with meshes.



Stringlines
Represents the corridor with lines connecting the template element points along the corridor.

8.6.2 Creating Corridor Template Elements

8.6.2.1 Commands

CORRIDORTEMPLATE, CORRIDORTEMPLATEELEMENT, CORRIDORTEMPLATEELEMENTEDIT

8.6.2.2 Overview

In BricsCAD, Corridors that follow a certain 3D alignment are created using the Corridor tool.

8.6.2.3 About the Corridor Template

In BricsCAD, a Corridor Template entity represents the framework to append a collection of template element objects. The CORRIDORTEMPLATE command enables you to create a corridor template entity and append one or more template elements to it.

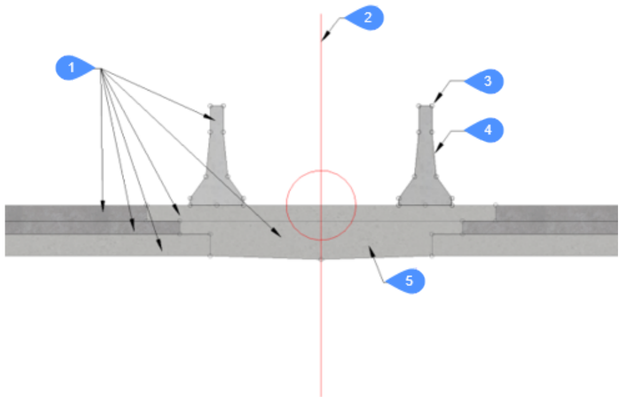
8.6.2.4 About the Corridor Template Element

In BricsCAD, a Corridor Template Element entity defines the geometry of a component used in a corridor section. The CORRIDORTEMPLATEELEMENT command enables you to create a corridor template element from a polyline.

8.6.2.5 About Points, Links and Shapes

A Corridor Template Element consists of the following components: points, links between points and shapes, that are created from a single or multiple links forming a polygon.

Shapes determine the geometry of the cross-sectional elements. They can be created from a single template element link or from multiple links forming a closed polygon. We assign materials to the shapes for a realistic representation of the corridor model.



1 Corridor Template Elements

2 Corridor Template

3 Point

4 Link

5 Shape with material assigned

8.6.2.6 About Corridor Targets

When the geometry of an individual corridor template element must follow a specific surface, a horizontal or a vertical object, a target is created and added to the corresponding point on the corridor template element. The following target types can be defined: Surface, Horizontal and Vertical targets.

8.6.2.7 Procedure: Step 1 - Convert Polyline to a Corridor template element

A simple polyline with 2 vertices and a (closed) polyline with several vertices can be used as an input for conversion into a Template Element. In both cases, the procedure described below should be used.

1 Draw a polyline, representing a roadway asphalt layer (in across-section view).

2 Launch the CORRIDORTEMPLATEELEMENT command.

3 Select the polyline.

4 Pick the template element base point.

5 Select the template element orientation (left, right, none).

Note:

A new template element is created, consisting of two points connected with a link. The position of a second point depends on the selected point type. By default, the Delta X and Delta Y type is used. This means that the position of the second point is determined by the delta X and delta Y offset from the Origin.

The grip is placed at the specified base point of the template element.

The created template element is overlapped with the existing polyline, that can be erased now.

1. P1 -Origin (Start point)
2. P2 - End point

8.6.2.8 Step 2 - Adding points to a Corridor template element

Continue from Step 1.

1 Launch the CORRIDORTEMPLATEELEMENTEDIT command.

2 Select the Template Element.

3 Select the option **Add point** in the command line.

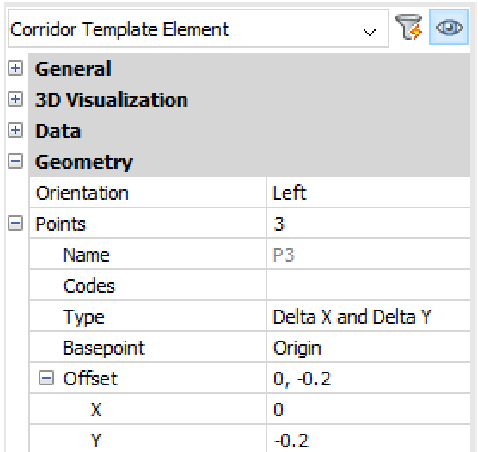
4 Specify positions for two new points, as shown on the figure below.

5 

6 Select a Corridor Template Element in a drawing and open **Properties** panel. Click in the **Points** field and click the arrow button to navigate to the point P3.

7 Keep the basepoint of the point P3 set to Origin (template element basepoint), which in this example overlaps with the point P1.

8



9 Define the offset of the point P3 from the Origin, for example X=0, Y=-0.2.

10 Navigate to the point P4 in the **Corridor Template Element Properties** panel.

11 Set the basepoint to Point P2.

12 Define the offset of the point P4 from its basepoint P2, for example X=0, Y=-0.2.

8.6.2.9 Step 3 – Editing Template Element Point Type

The value of parameter **Type** can be changed by selecting a different way of determining the distance from the selected Basepoint. The procedure is described below.

Continue from Step 2.

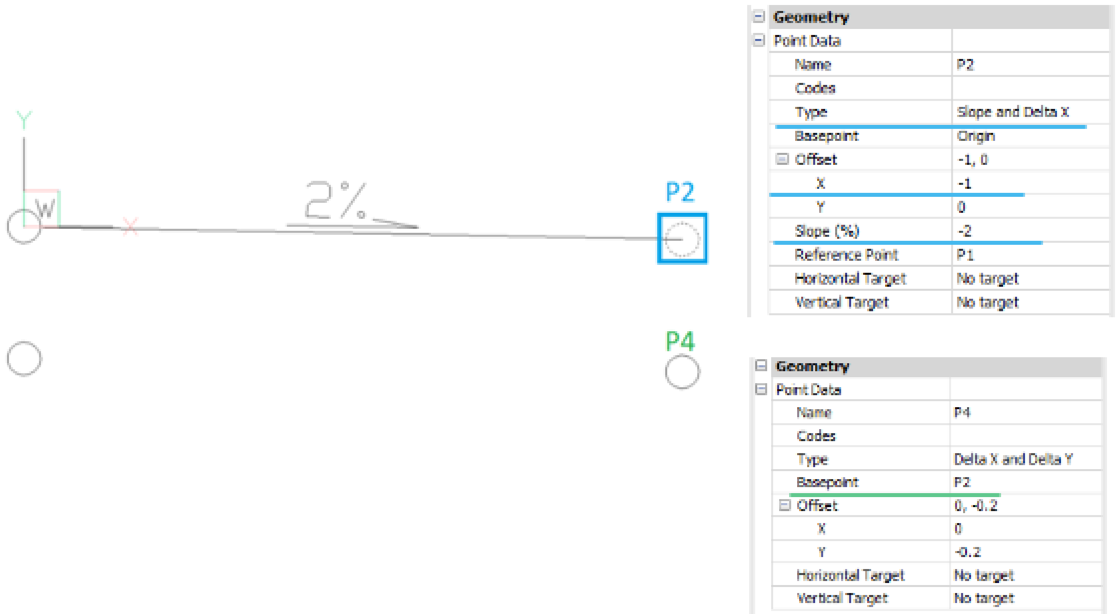
1 Select the Template Element Point P2 by holding the Ctrl button, selecting the desired point, or by selecting the Corridor Template Element in a drawing and navigate to the point P2 in the **Properties** panel.

2 Select the option **Slope and Delta X** from the drop-down list in the **Type** field.

3 Type the offset distance in the **X** field and slope in the **Slope (%)** field.

Note:

- The position of the point P2 is determined by the specified offset in the X direction and the slope in %.
- Point P4 moves accordingly when the position of the point P2 changes, because its **Basepoint** is set to P2.



8.6.2.10 Step 4 - Adding links to a Corridor template element

Continue from Step 3.

1 Launch the CORRIDORTEMPLATEELEMENTEDIT command.

2 Select the template element in a drawing.

3 Select the option **Add Link** in the Command line.

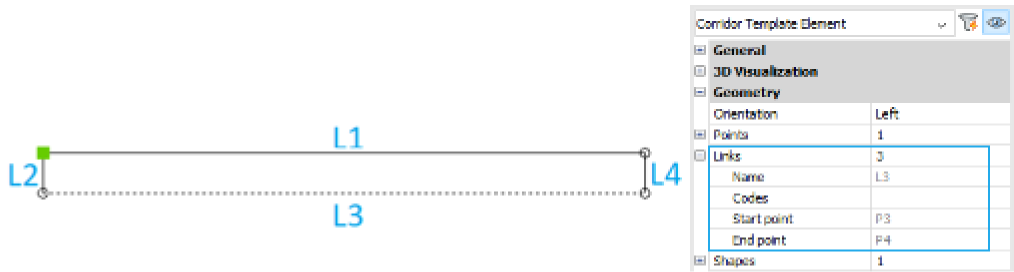
4 Select template element points P1 and P3 to create link between them.

5 Select template element points P3 and P4 to create link between them.

6 Select template element points P4 and P2 to create link between them.

7 Press the Enter button to finish the command.

8



8.6.2.11 Step 5 – Changing Template Element Basepoint

1 Select the template element in a drawing.

2 Click on the Basepoint.

3 Hold the Ctrl button and specify the new Template Element basepoint position in a drawing.

4 Select the template element in a drawing to see a new Template Element Basepoint position.

8.6.2.12 Step 6 - Adding shape to a Corridor template element

Continue from Step 5.

1 Launch the CORRIDORTEMPLATEELEMENTEDIT command.

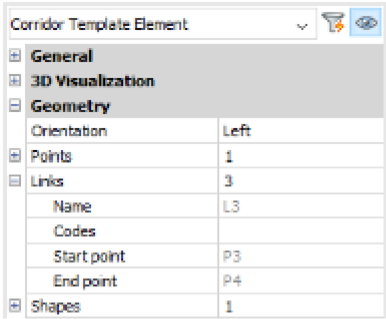
2 Select the template element in a drawing.

3 Select the option **Add Shape** in the command line.

4 Click on the links L1, L2, L3 and L4 in the written order.

5 Press the Enter button to finish the command.

6



8.6.2.13 Step 7 - Adding an embankment line to existing Template Element

Continue from the Step 6.

1 Launch the CORRIDORTEMPLATEELEMENTEDIT command.

2 Select the template element in a drawing.

3 Select the option **Add Point** in the Command line.

4 Specify the new point position in a drawing

5

6 Launch the CORRIDORTEMPLATEELEMENTEDIT command.

7 Select the template element in a drawing.

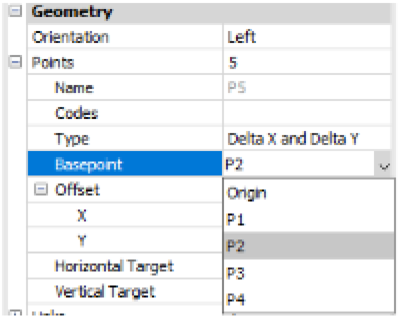
8 Select the option **Add Link** in the Command line.

9 Select points P2 and P5 to create a new link.

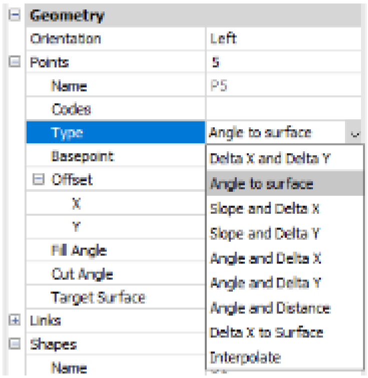
10 

11 Select the Template element point P5 and change its Basepoint to the point P2.

12 



13 Select the Template element point P5 and change the Type parameter to **Angle to Surface**.



8.6.2.14 Step 8 - Adding a shape to the Embankment line

Continue from Step 7.

1 Launch the CORRIDORTEMPLATEELEMENTEDIT command.

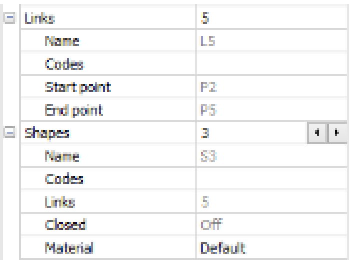
2 Select the template element in a drawing.

3 Select the option **Add Shape** in the command line.

4 Select template element link L5.

5 Press the Enter button to finish the command.

6 



8.6.2.15 Step 9 - Adding the Surface target to embankment

Continue from Step 8.

1 Launch the CORRIDORTEMPLATEELEMENTEDIT command.

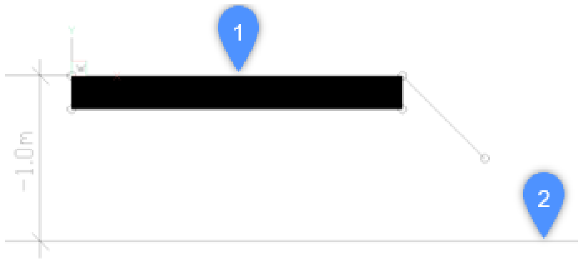
2 Select the template element in a drawing.

3 Select the option **Add Target** in the command line.

4 Enter target name: **Surface-1**.

5 Select target type: Surface.

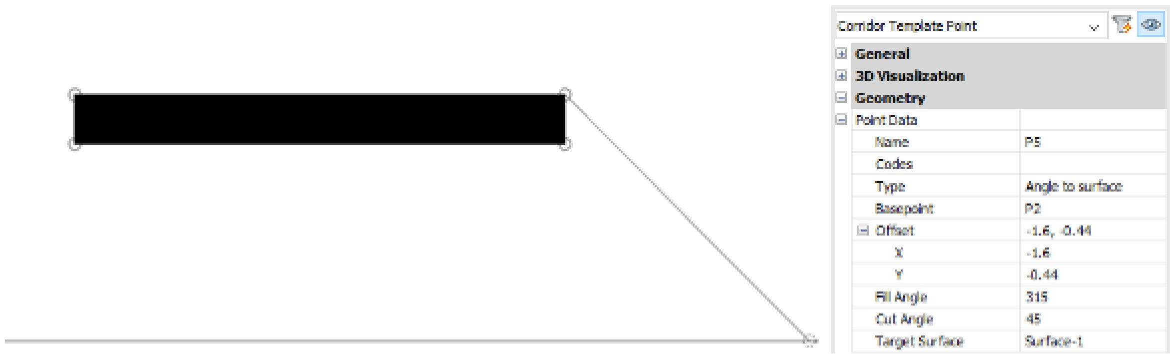
6 Enter the target default value: **-1**.



1 - Template Element

2 - Surface Target

7 Select the Template Element Point P5 and select **Surface-1** in the **Target Surface** field.



8.6.2.16 Step 10 – Appending Corridor template element to a Corridor template

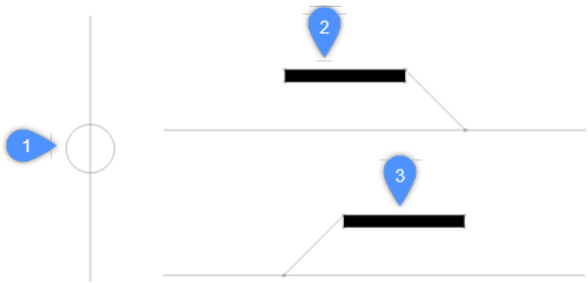
Continue from Step 9.

1 Launch the CORRIDORTEMPLATE command.

2 Pick a point in a drawing to create corridor template.

3 Copy and paste an existing Template Element in the same drawing.

4 Select the copied Corridor Template Element and change the **Orientation** parameter to **Left** in the **Properties** panel.



1 - Corridor Template

2 - Template Element Right orientation

3 - Template Element Left orientation

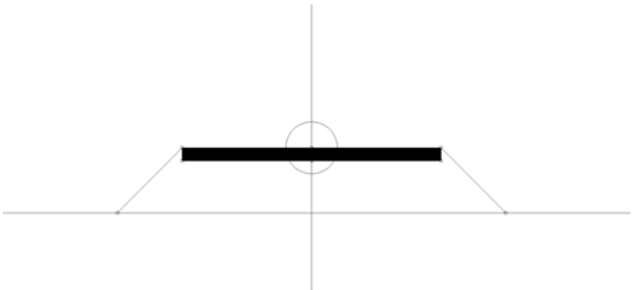
5 Launch the CORRIDORTEMPLATE command.

6 Select the option **Add template element** in the Command line.

7 Select the corridor Template Element with the right orientation.

8 Select the Corridor Template to append selected Template Element to it.

9 Repeat steps 5 – 8 also for the Left oriented Template Element.



8.7 Alignments

8.7.1 Creating alignments

8.7.1.1 Overview

In BricsCAD, alignments are created using the Alignment tool.

Several methods are available for creating new alignment:

- Method **Alignment by Points of intersections (PI)** enables you to create Alignment by Tangent Polygon.
- Several methods enable you to create **Alignment by Elements**:
    - **Alignment Line**.
    - **Alignment Curve**.
    - **To/From Element**.
    - **Between Elements**.
- Import Alignment from LandXML file.

- Import Alignment from AutoCAD® Civil 3D.

Alignment is the route of the linear objects, defined as a combination of lines and curves that are viewed as one object.

8.7.1.2 About ALIGNMENTS

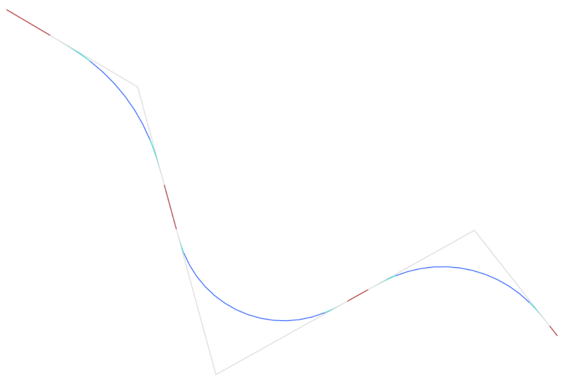
Alignment is a reference system used to position linear objects, like roads, railways, pipe networks, streets, retaining walls and bridges, in space. An alignment is represented with horizontal, vertical 2D curves, resulting in 3D alignment.

Horizontal alignments

Horizontal alignment is a 2D curve that is needed to precisely position linear objects in a horizontal direction. It can be created by Points of Intersections (PI), by Alignment Elements, imported from LandXML and from AutoCAD® Civil 3D. Horizontal alignment is the horizontal trace on your TIN Surface which is used to generate the elevation profile and vertical alignment.

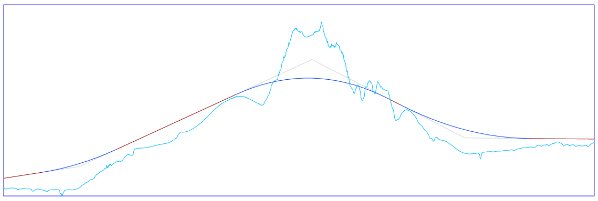
Note: The Horizontal Alignment is not necessarily always projected (draped) on the TIN Surface. Projection is required to create Vertical Alignments and Corridors.

A single horizontal alignment can have one vertical alignment. The horizontal alignment will appear under the surface.



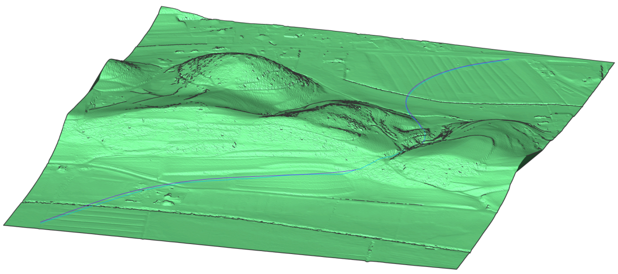
Vertical alignments

A vertical alignment is needed to precisely define a linear object in the vertical sense. The horizontal path defined by the horizontal alignment corresponds with a vertical profile of the TIN Surface. By defining Points of Vertical Intersection (PVI) on these vertical profiles of the target surfaces, the new 2D curve defines the vertical alignment.



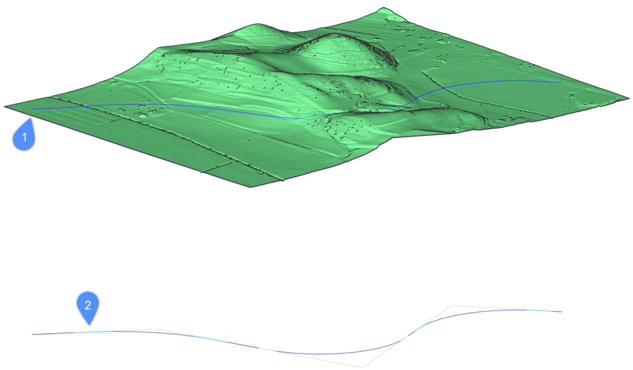
3D alignments

A 3D Alignment is a 3D polygonal curve defined by the horizontal and vertical alignment.



8.7.1.3 Procedure: Creating an Alignment by PI

1. Open the drawing file that contains a TIN Surface.
2. Launch the ALIGNMENT command in the Command line.
3. Select the TIN surface.
4. Optionally, select the **Specify start station** option in the Command line to define start station for the alignment
5. Pick the horizontal alignment PI points on your surface. Press ENTER when done. Your 3D alignments (1) and horizontal alignment (2) are now generated.



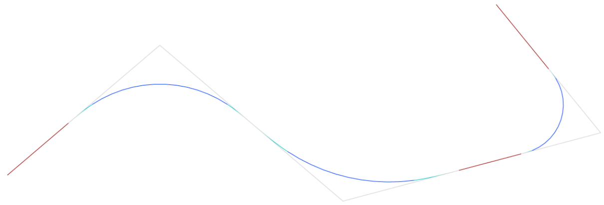
Note: Three alignments are shown. The horizontal alignment on plane 0,0, light blue 3D alignment shows the projection of horizontal alignment on TIN Surface and the purple 3D alignment shows the design alignment that is automatically generated.

8.7.1.4 Procedure: Creating a Horizontal Alignment without 3D alignments

1. Open a drawing and launch the ALIGNMENT command in the Command line.
2. Select the **Pick pi point** option in the Command line.

   Note: The Alignment is not projected on the TIN Surface if this option is selected in the Command line.
3. Optionally, select the **Specify start station** option in the Command line to define start station for the alignment
4. Pick the horizontal alignment PI points. Press ENTER when done.

   Your horizontal alignment is now generated.

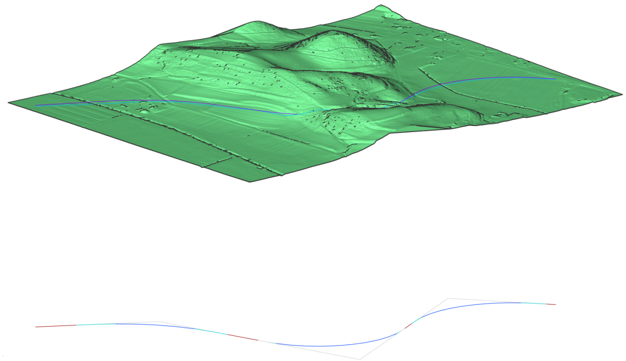


8.7.1.5 About Vertical Alignment View

The horizontal alignment creates a vertical section of the target TIN surface. This 2D curve can be displayed in a separate viewport inside your model space using the command ALIGNMENTVIEW.

8.7.1.6 Creating a Vertical Alignment View

1 Open the drawing file that contains a TIN Surface, Horizontal Alignment and 3D Alignments.



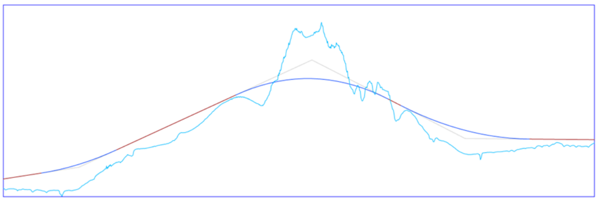
2 Launch the ALIGNMENTVIEW command in the Command line.

3 Select a horizontal or 3d alignment in the drawing.

Note: If you select the Horizontal alignment in a drawing that is not projected yet on TIN Surface, then TIN Surface needs to be selected in the next step.

4 Pick a point as origin for the vertical alignment view.

5 Your vertical alignment view is now generated.

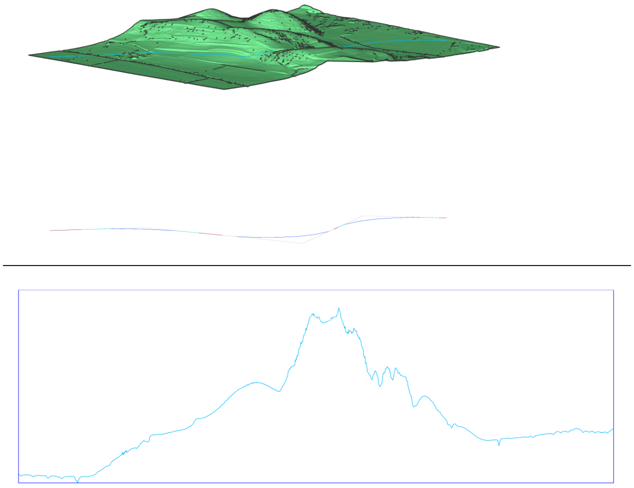


Note: Two vertical alignments are shown. Light blue represents terrain line from the TIN Surface, and the other one is vertical alignment. The vertical alignment has straight lines and vertical curves and is created automatically based on the accuracy factor of the Vertical Tolerance parameter. You can change the accuracy factor in the **Properties** panel.

8.7.1.7 Creating a Vertical Alignment

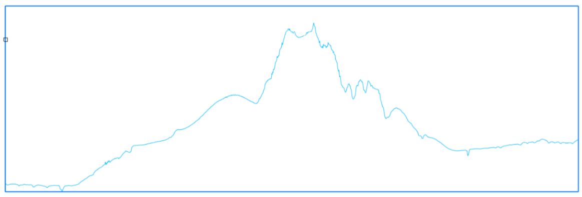
1 Open the drawing file that contains a TIN Surface, Horizontal Alignment and Vertical Alignment View.

1

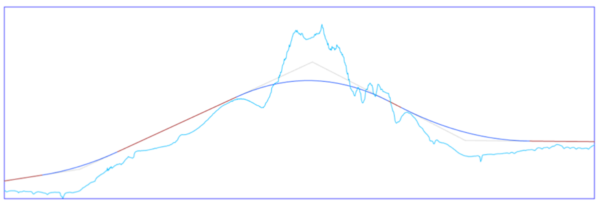


2 Launch the ALIGNMENTVERTICAL command in the Command line.

3 Select vertical alignment view in the drawing.



4 Pick the vertical alignment PVI points. Press ENTER when done. If you want to go one step back, use Undo. Automatic 3d alignment option generates vertical alignment automatically. Your vertical alignment is now generated.



8.7.1.8 Attaching new alignment element before or after an existing alignment element

1 Draw a new Horizontal alignment using the ALIGNMENTLINE or ALIGNMENTCURVE commands.

2 Launch the ALIGNMENTELEMENTFROM command.

3 You are prompted:

Alignment element [Line/Curve/Spiral/Line with Spiral/Curve with Spiral/Curve-Spiral-Spiral]:

4 In the Command line, select which type of element you want to attach. In this example, we will use the **Curve with Spiral** option.

5 You are prompted:

6 Attach to: [Element/element END]

- **Element**: select this option to attach new element to the specified point on the attachment element.
- **element END**: select this option to attach new element to the start/end point of the attachment element.

Note: To attach a curve before the existing element, select it closer to the element start point. Otherwise click closer to attachment element end point to attach curve after (starting from) the element.

7 Select an existing line element (3).

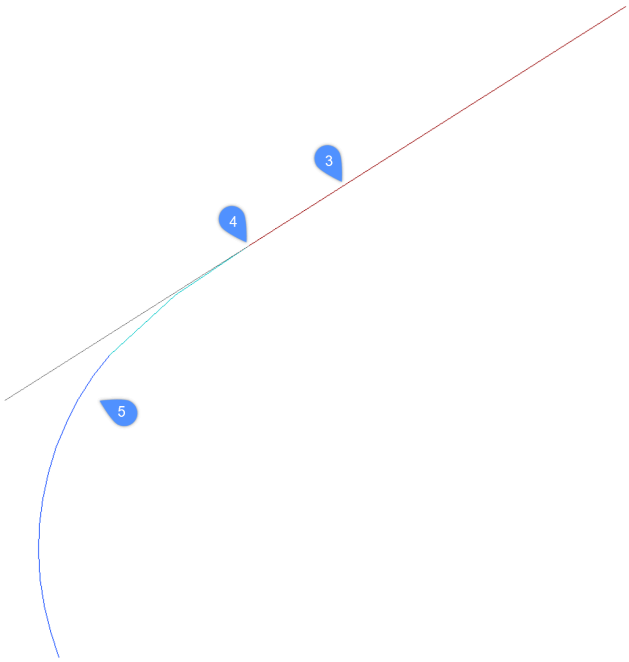
8 Specify the spiral radius.

9 Specify the spiral length.

10 Specify the curve solution angle (greater or less than 180°).

11 Specify the curve pass-through point (4) to draw the Curve-Spiral combination (5).

Note: The new element is always drawn tangential to the specified point on the selected attachment element.



8.7.1.9 Attaching new alignment element between two selected alignment elements

1 Draw a Horizontal alignment with two elements, using the ALIGNMENTLINE or ALIGNMENTCURVE commands.

2 Launch the ALIGNMENTELEMENTBETWEEN command.

3 You are prompted:

Alignment element [Line/Curve/Spiral/Spiral-Curve-Spiral/Spiral-Spiral/Spiral-Line-Spiral]:

4 In the Command line, select which type of element you want to attach. In this example, we will use the **Spiral-Curve-Spiral** option.

5 Select the first attachment element (6).

6 Select the second attachment element (7).

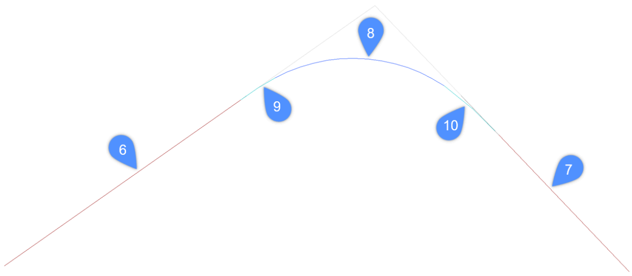
7 Specify the curve radius (8).

8 Specify the spiral in length (9).

9 Specify the spiral out length (10).

10 The new Spiral-Curve-Spiral combination between the selected alignment elements will be added.

Note: The new element is drawn tangential to both attachment elements.



8.7.2 Editing alignments

8.7.2.1 Overview

Several alignment editing options are available in BricsCAD Civil, using the ALIGNMENTEDIT command:

- Continue a horizontal or vertical alignment;
- Add or delete Pi/Pvi points;
- Delete elements;
- Add or remove Station equations;
- Change the TIN Surface

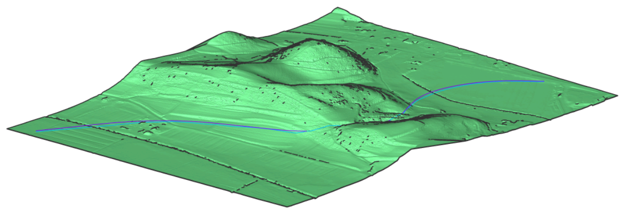
Note: The 3D alignment is updated automatically.

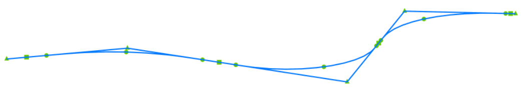
8.7.2.2 Continuing a horizontal alignment with ALIGNMENTEDIT

1 Draw a horizontal alignment following the steps from the **Create alignments** article, **Creating an alignment by PI**.

2 Type ALIGNMENTEDIT in the Command line.

3 Select the horizontal alignment you want to edit.

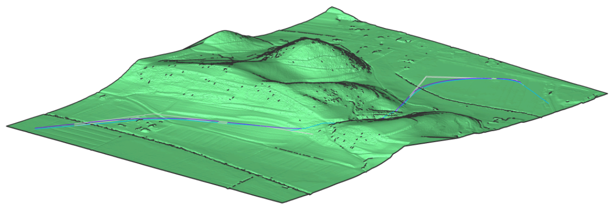


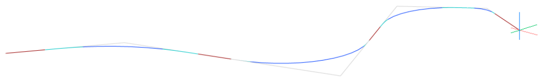


4 Select the option **Continue by Pi** in the Command line.

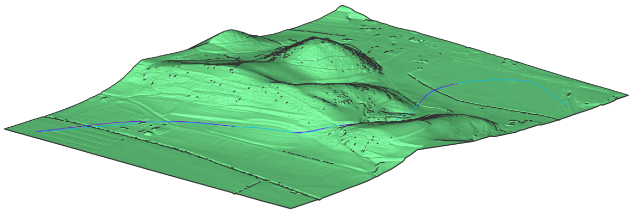
5 Pick new horizontal alignment PI points to continue your existing alignment.

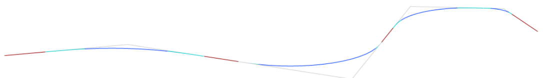
6 Use the **Undo** option in the Command line to return to the previous step.





7 Press enter when finished. In doing so, the horizontal alignment is extended accordingly, and the 3D alignment is updated automatically.





8.7.2.3 Adding PI to an existing horizontal alignment

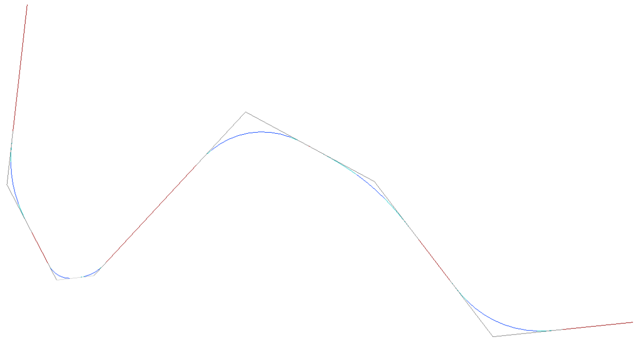
1 Type ALIGNMENTEDIT in the Command line.

2 Select the horizontal alignment you want to edit.

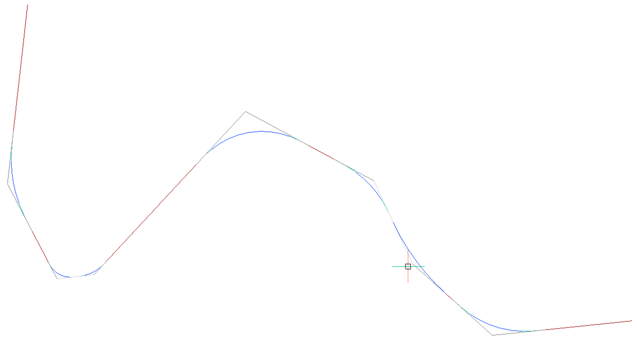
3 Select the option **Add Pi** in the Command line.

4 Pick point to add new PI.

Before adding PI point

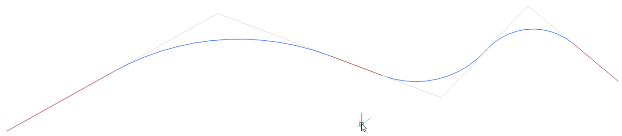


After adding PI point



8.7.2.4 Removing PI from existing horizontal alignment

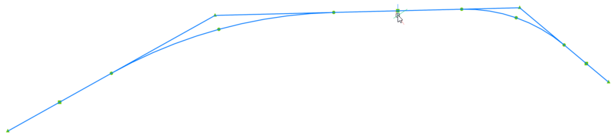
1 Type ALIGNMENTEDIT in the Command line.



2 Select the horizontal alignment you want to edit.

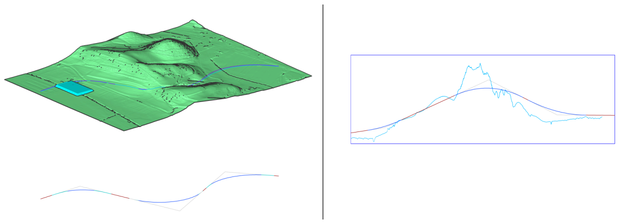
3 Select the **Remove Pi** option in the Command line.

4 Pick a point near PI to remove.



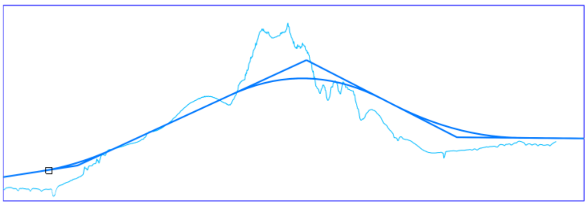
8.7.2.5 Adding additional TIN Surface to Vertical alignment view

1 Open the drawing file that contains a TIN Surface, Alignments, Vertical Alignment View and grading.

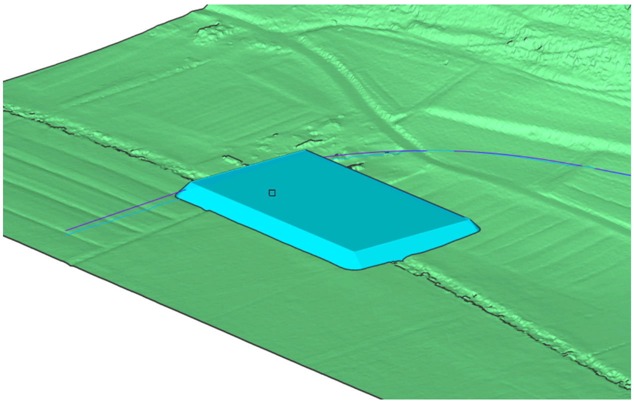


2 Launch the ALIGNMENTEDIT command in the Command line.

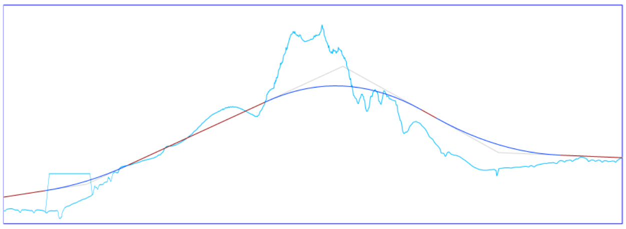
3 Select the **change Tin surface** option in the Command line.



4 Select the TIN surface to add.



In the vertical alignment view, you can see an additional light blue line representing the grading object.



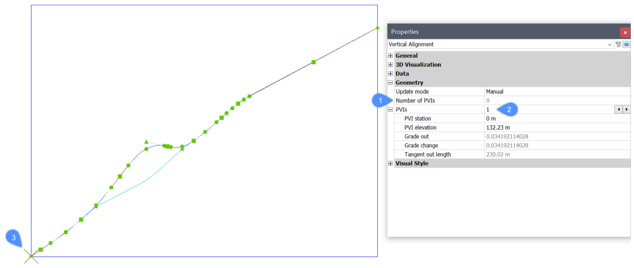
8.7.2.6 Edit Alignments through the Properties panel

You can edit the alignment parameters through the **Properties** panel.

Edit Vertical Alignment

- Select a **Vertical Alignment**.
- In the **Properties** panel, you can see that the **Vertical Alignment** consists of a number of PVIs (1).

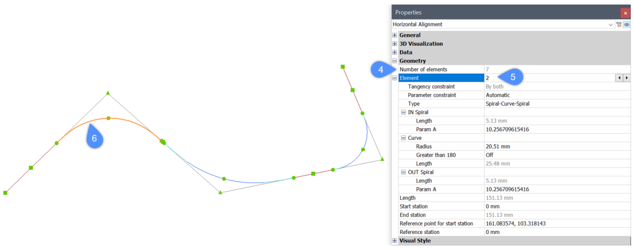
- Once click on the **PVI** field (2), the currently selected PVI is highlighted in the drawing (3).



- You can switch between PVIs and edit the Alignment PVI parameters. The **Vertical Alignment** will be automatically updated in the drawing.

Edit Horizontal Alignment

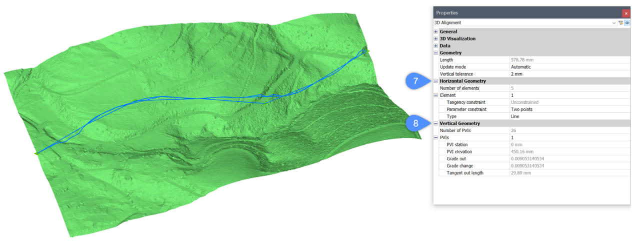
- Select a **Horizontal Alignment**.
- In the **Properties** panel, you can see that the **Horizontal Alignment** consists of a number of elements (4).
- Once click on the **Element** field (5), the currently selected element is highlighted in the drawing (6).



- You can switch between elements and edit the Alignment element parameters. The **Horizontal Alignment** will be automatically updated in the drawing.

Edit 3D Alignment

- Select a **3D Alignment**.
- In the **Properties** panel, you can edit its Horizontal (7) and Vertical (8) alignment parameters.



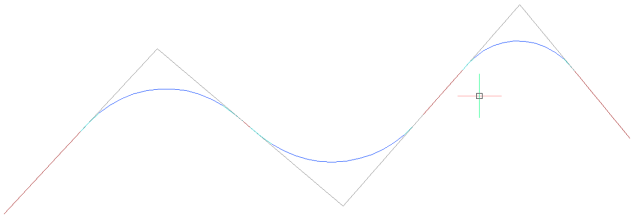
- As described above, you can switch between elements and PVIs and change their parameters. The **3D Alignment** will be automatically updated in the drawing.

8.7.2.7 Working with alignment station equations

The alignment station equation can be set and changed as desired.

Adding station equation to an existing Horizontal alignment

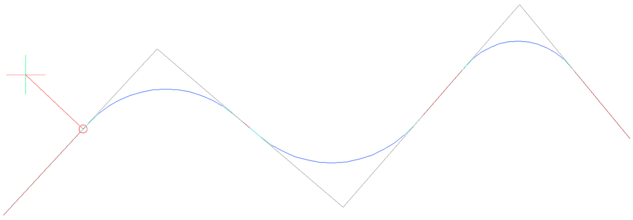
1 Type ALIGNMENTEDIT in the Command line.



2 Select the Horizontal alignment you want to edit.

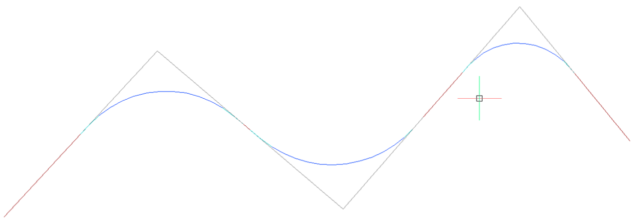
3 Select the **Add Station equation** option in the Command line.

4 Pick point to add new **Station equation**



Removing station equation from an existing horizontal alignment

1 Type ALIGNMENTEDIT in the Command line.



2 Select the Horizontal alignment you want to edit.

3 Specify the index number of station equation you want to remove.

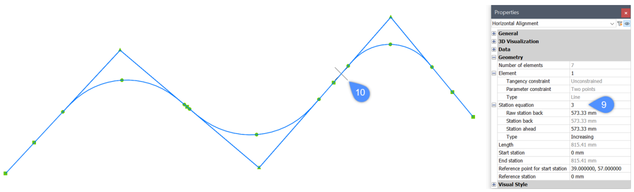
Editing station equation parameters through the Properties panel

You can switch between created **Station equations** and edit the parameters in the **Properties** panel.

1 Select a **Horizontal Alignment**.

2 In the **Properties** panel, you can see if the **Horizontal Alignment** have station equations added.

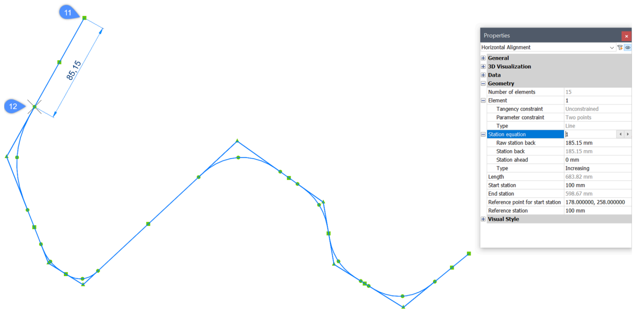
3 Once click on the **Station equation** field (9), the currently selected station equation is highlighted in the drawing (10).



- The **Raw station back** value is relative to the Alignment start point (11) and the specified point for the **Station Equation** (12).

> ***Note:*** **Start station** at the Alignment start point may be different from zero (specified by the user). If only one **Station Equation** is defined on the Alignment, then **Raw station back** value matches to **Station back** value.

Edit the **Raw station back** parameter in the **Horizontal Alignment** Properties to move the point for the **Station Equation** on the Alignment.



- The **Station back** value of all Station equations, except for the first one, is relative to the **Station equation** value and the **Station ahead** value, placed directly before it. If only one Station Equation is defined on the Alignment, then **Station back** matches to **Raw station back**.
- The **Station Ahead** specifies the new station value starting after the new point for the **Station Equation** location on the Alignment.
- The **Type** parameter specifies whether the stations increase or decrease after the (point for the) Station Equation location on the Alignment.

9. Parametric modeling

9.1 Parametric modeling overview

BricsCAD provides a rich toolset to control 2D and 3D geometric models with parameters of different kinds. This includes:

- **Global parameters**: parameters of 2D/3D dimensional constraints, user-defined parameters, configuration parameters of design tables, etc.
- **Local parameters**: parameters of arrays and constrained entities inside arrays, parameters of inserts of parametric components, etc.

All parameters and their current values can be found in the **Parameters Manager** panel or in different sections in the **Mechanical Browser** panel. Parameter values can be linked together via expressions. When you change the value of a parameter, the model is updated automatically. This workflow allows you to add parametric behavior to any geometry and explore design variants easily.

Note: In BricsCAD, there are no parent-child dependencies between geometric elements. For example, if you change a sketch used to create an extruded 3d solid, the solid is not changed accordingly. However, you can create dependencies with parameters and expressions afterward.

9.1.1 Global and local parameters

Commands:

- The -PARAMETERS command manages dimensional constraint parameters through the Command line.
- The PARAMETERSPANELOPEN command opens the **Parameters Manager** panel.
- The MECHANICALBROWSEROPEN command opens the **Mechanical Browser** panel.
- The CLEANUNUSEDVARIABLES command purges parametric variables not used by constraint expressions and not linked to dimensions.

Parameters in BricsCAD can be either global or local.

Local parameters are attached to a particular entity, an associative array or a mechanical component.

Global parameters are not attached to a particular entity, they are subdivided into four groups:

- Parameters of 2D dimensional constraints
- Parameters of 3D dimensional constraints
- User-defined parameters
- Configuration parameters of design tables

9.1.2 Defining an expression for a parameter

You can define an expression for both parameter types, global and local. The simplest expression is a number or the name of a global parameter (local parameters cannot be referenced in expressions). A more complex expression can include standard operators and functions.

To define an expression for a parameter, open the **Mechanical Browser** panel, click on the parameter name and fill out the **Expression** field.

Note: You can also define an expression for a parameter in the Command line using the -PARAMETERS command.

9.1.3 Defining dependent parameters

In expressions, you can use both: names of global parameters and numbers. This is known as a 'non-constant expression'. If a non-constant expression is set to a parameter, its icon changes from an open padlock ( ) to a closed padlock ( ) in the **Mechanical Browser** panel.

This means that the parameter becomes dependent on (an)other (defining) parameter(s) and its value will be automatically recalculated when the value(s) of the defining parameter(s) change(s).

You can create chains of parameters, where each element depends on the previous one. There is no limit to the length of such a chain. However, be careful not to create a loop where the next element in a chain drives one of the previous elements. This might result in an infinite evaluation of expressions. BricsCAD automatically detects such problems and does not accept expressions that lead to a parametric loop.

9.1.4 Parameters of associative arrays

Associative arrays are created by the ARRAY, ARRAYRECT, ARRAYPOLAR and ARRAYPATH commands. The properties of associative arrays that can be controlled with an expression are the following.

- **Rectangular arrays**: Number of columns, Column spacing, Number of rows, Row spacing, Row elevation increment, Number of levels, Level spacing, and Axis angle.
- **Polar arrays**: Radius, Number of Items, Angle between items, Fill angle, Number of rows, Row spacing, Row elevation increment, Number of Levels, and Level spacing.
- **Path arrays**: Item spacing, Start offset, Rows, Row spacing, Row elevation increment, Levels and Level spacing.

To provide the availability of blocks' parameters inside an array, parameters must be linked (using the **Link to parameter** context menu command) before the array creation. After array creation, parameters will be available for editing in the **Mechanical Browser** panel or **Parameters Manager** panel.

The link to the parameter is removed, but it can be restored for array's blocks parameters using the same **Link to parameter** context menu item.

Technically, the parameters are associated with the source element of an array. When this source element is edited parametrically, all other items of the array are changed after it.

For the former implementation of parametric components (MECHANICALBLOCKS system variable is Off and BMINSERT is used) parameters of the components nested inside array are available simply after the array creation.

To define an expression for any of these properties, select the corresponding array entity, either in the drawing area or in the **Mechanical Browser** panel and enter the expression in the corresponding properties field in the **Mechanical Browser** panel or in the **Properties** panel.

If an associative array contains entities constrained with 2D or 3D dimensional constraints, the parameters of these constraints can also be evaluated with expressions. These nested parameters are listed in the **Mechanical Browser** panel. If you select any of them, you can define an expression.

9.1.5 Parameters of mechanical components

The BMUPDATEMODE system variable controls whether external assembly components are always reloaded or only when modified.

If you insert a mechanical component or just a plain .dwg file into a drawing with BMINSERT command and the insert contains global exposed parameters, these parameters are converted to local parameters. They are associated with the corresponding component insert and are visible in the **Mechanical Browser** panel under the name of this insert. Click the parameter(s) to edit.

9.1.6 Parameter properties

You can manage all parameters such as user-defined, 3D constraints, design tables, parameters of parametric components and arrays via the **Mechanical Browser** panel.

- **Name**: identifies the parameter. The parameter name can be used in expressions of other parameters in the same **component**. The name cannot begin with a number, it should be alphanumeric without spaces.
- **Expression**: type a value or an expression.
- **Value**: displays the current value of the parameter.
- **Geometry-driven**: if set to **On**, it disables the **Expression** field. The value of a parameter value is obtained from the model and can be changed with direct modeling operations.
- **Description**: optional description of the parameter.
- **Exposed**: defines whether the parameter is visible and can be modified when the component is inserted in an assembly. Select the field, then click the down-arrow button and choose an option. A new parameter gets the **OFF** option by default.
    - **ON**: the parameter is always exposed.
    - **OFF**: the parameter is never exposed.
    - **Reset**: the option is set to it's default value.
- **Units**: sets the dimension of the parameter: **Linear**, **2-dimensional** or **3-dimensional**. Select the field, then click the down-arrow button and choose an option.

9.1.7 3D constraints properties

BricsCAD provides a set of properties for 3D constraints which allow you to control mutual placement of the constrained geometric entities (or constraint arguments) upon 3D constraints recalculation.

The properties may be assigned both to a constraint itself (**Directions** property) and/or to the constraints arguments, for each argument separately (**Placement** and **Use as** properties).

- **Type**: indicates the constraint type: Path, Distance, Radius, Angle or Cone Angle.
- **Enabled**: controls whether the constraint is On or Off.

- **Directions**: specifies whether the directions of the constraint arguments are **Same** or **Opposite**. Changing from one to another flips constrained entities with respect to each other.

  Direction is used for lines, planes, circles, cylinders, cones and tori. It is not used for points and spheres.

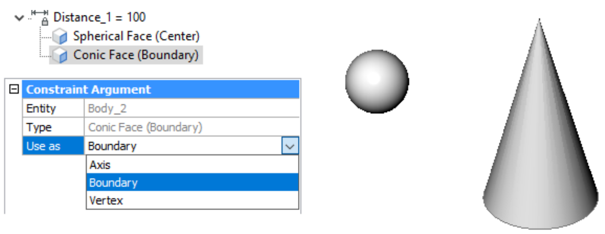
  The direction of a plane is its normal vector. The direction of a circle is the normal vector of the circle's plane. The direction of a cylinder, cone, or torus is the direction of its axis. The direction of a planar face of a 3D solid is additionally adjusted to be external with respect to the 3D solid object.

  Any option means that the directions may change upon direct modeling operations or upon a 3D constraint parameter change.

  The **Keep** option forces BricsCAD to keep the current directions. This option forces BricsCAD to analyze the mutual location of the arguments prior to 3D constraints solving and use either **Same** or **Opposite** (not both).
- **Placement**: specifies how to place an argument with respect to another argument of the same constraint. For example, a point in a point-sphere distance may be placed either **Outside** or **Inside** of a sphere. Changing this property for a constraint argument moves the argument to the other side of another argument. The sides are defined for planes, spheres, cylinders, cones, and tori. And not defined for points, lines, and circles.

  Any means that both **Outside** and **Inside** placement is suitable for user's intentions.

  Keep option forces BricsCAD to keep the current placement of the argument. In other words, this option forces BricsCAD to analyze the mutual location of the arguments prior to 3D constraints solving and use either **Outside** or **Inside** (not both). The **Keep** option is used by default.
- **Use as**: this property is assigned to the arguments of the **Distance** and **Concentric** constraints and applies to circles, spheres, cylinders, cones and tori and defines which geometry is used in a 3D constraint to measure the distance or to coincide with the other entity.



  There are four options for the **Use as** property:

  - **Boundary**: the constraint is applied to the boundary of the argument.
  - **Axis**: applies to an argument that has an axis: circle, cylinder, cone, or torus.
  - **Center**: applies to an argument that has a central point: circle, sphere, or torus.
  - **Vertex**: applies to a cone vertex only.

9.1.8 Context menus

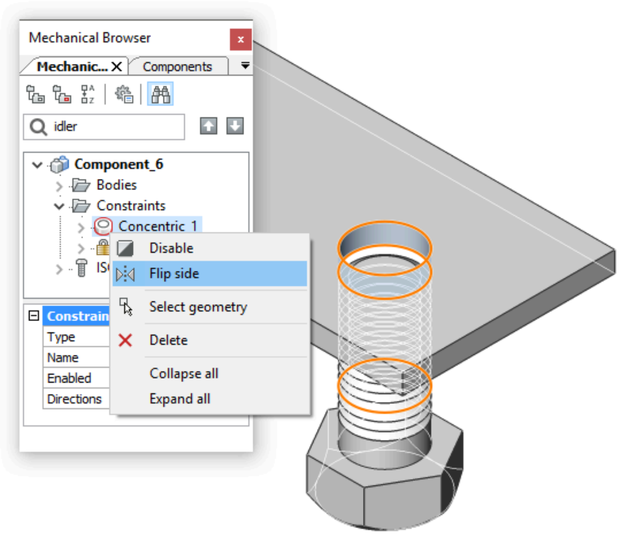
In the **Mechanical Browser** panel, the following context menus are available.

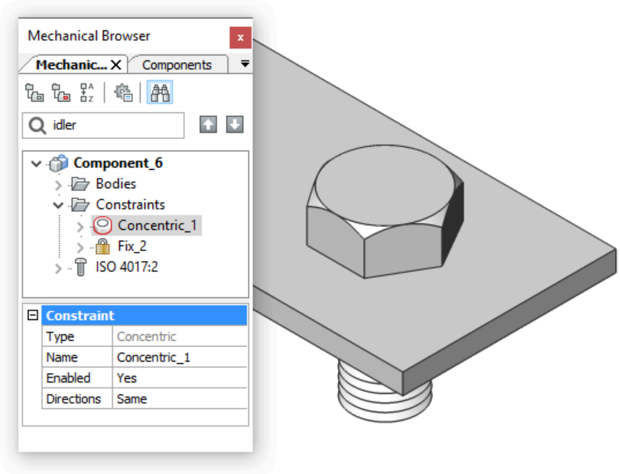
Main Node

- **Add new parameter**: creates a new parameter.
- **Collapse All**: collapse all nodes.
- **Expand All**: expand all nodes.

Sub-nodes

- **Delete**: deletes the selected parameter from file.
- **Create design table**: creates a design table to drive parametric block parameters.
- **Animate**: animates the parameter value within some range in the drawing.
- **Link to parameter**: links a subcomponent parameter to the main level parameter.
- **Enabled**: controls whether the constraint or parameter is evaluated or not.
- **Geometry driven**: if selected, disables the **Expression** property. The value of a parameter value is obtained from the model and can be changed with direct modeling operations.
- **Flip Side**: allows you to change the relative position of an entity (vectors normal of the selected faces point in the same direction or in the opposite direction). This option can be applied to Parallel, Coincident and Concentric constraints only and on condition the constraint is applied to faces.





- **Select geometry**: selects the geometry that is affected by the constraint, in the drawing.

9.1.9 The Parameters Manager panel

The **Parameters Manager** panel allows you to browse and manage global parameters of your model in tabular form. Every row in this table corresponds to one global parameter, while every column represents a particular property of a parameter - name, expression, value.

In the **Parameters Manager** panel you can:

- Create a new parameter.
- Delete an existing parameter.
- Click a parameter name field and change it.
- Click a parameter expression field and change it.
- Search a parameter by name.
- Sort all parameters by name, expression, and value.
- Hide all parameters except those relevant to the selected parameter.
- Hide all parameters except those dependent on the selected parameter.
- Animate a parameter.
- Clean unused variables.

9.2 Operations and functions in expressions

9.2.1 Operators

Operation	Character	Description
Addition	+	Produces the sum of numeric values.
Subtraction	-	Subtracts the two numeric values.

Operation	Character	Description
Multiplication	*	Multiplies the numeric values.
Division	/	Divides two numeric values.
Exponentiation	^	Calculates the exponential value of the given numbers.
Remainder operator	%	Gives the remainder after the division of one numeric value to another. e.g. 5%2 = 1, because 5/2 leaves remainder of 1.

9.2.2 Expressions

Expressions are evaluated according to the standard mathematical rules of precedence:

1 Expressions within brackets; innermost sets first.

2 Standard operations order:
- 1. exponent
- 2. multiplication and division
- 3. addition and subtraction

3 Operators of equal precedence from left to right.

The following functions can be used in expressions:

Function	Syntax
Sine	sin(expression)
Cosine	cos(expression)
Tangent	tan(expression)
Arc sine	asin(expression)
Arc cosine	acos(expression)
Arc tangent	atan(expression)
Hyperbolic sine	sinh(expression)
Hyperbolic cosine	cosh(expression)

Function	Syntax
Hyperbolic tangent	tanh(expression)
Arc hyperbolic sine	asinh(expression)
Arc hyperbolic cosine	acosh(expression)
Arc hyperbolic tangent	atanh(expression)
Square root	sqrt(expression)
Signum function (-1,0,1)	sign(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Round down	floor(expression)
Round up	ceil(expression)
Absolute value	abs(expression)
Largest element in array	max(expression1;expression2)*
Smallest element in array	min(expression1;expression2)*
Degrees to radians	d2r(expression)
Radians to degrees	r2d(expression)
Logarithm, base e	ln(expression)
Logarithm, base 10	log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Power function	pow(expression1;expression2)*

Function	Syntax
Random decimal between 0 and 1	Random

* Use the list separator character as defined on your system: , (comma) or ; (semicolon).

Note:

- The constants **Pi=3.14...** and **e=2.72...** can be used in expressions. The constants names are not allowed to be used as a parameter or constraint name.
- The CLEANUNUSEDVARIABLES command purges parameters that are not used in constraint expressions or linked to dimensions.

9.3 3D constraints

9.3.1 Variational Direct Modeling

Use geometric constraints (aka variational direct modeling) to model parametric solids and surfaces. Define constraints between edges and faces to modify solid and surface models.

As soon as a constraint is added, BricsCAD automatically modifies the model and these constraints are updated with subsequent modifications to the model. The key advantage of variational direct modeling is that all constraints are taken into account simultaneously and the model behavior does not depend on the constraint creation order. It allows you to parametrize any feature of the 3D model without the worry of model creation history.

Dimensional constraints control the dimensions of the model. When such a constraint is introduced or the value of its parameter is changed, BricsCAD automatically updates the solid and surface geometry. For example, the dimensions of a box can be controlled by the parameters of three distance constraints applied to its opposite faces.

BricsCAD provides the following 3D constraints:

- Geometric constraints: **Fix** (), **Coincident** (), **Concentric** (), **Parallel** (), **Perpendicular** (), **Tangent** (), and **Rigid Set** ().
- Dimensional 3D constraints: **Distance** (), **Radius** (), **Angle** (), **Cone Angle** (), **Path Constraint** ().

3D Constraints are available in the **3D Constraints** toolbar, the **Parametric/3D Constraints** menu, the **3D Parametric** ribbon panel and the **Constraints** tab on the Quad.

All 3D constraints are listed in the **Mechanical Browser** panel. Select a constraint to check and edit its properties.

The following (sub)entities and items are supported for 3D constraint creation:

- Vertices, edges and faces of 3D solids, surfaces, meshes and regions and surfaces of planar, cylindrical, spherical, toroidal and conical geometry.
- Lines, rays, xlines, circles and arcs.

- Segments and vertices of 3D polylines.
- Arbitrary curves for coincident and path constraints with a point as another argument.
- Planes, axes and origin points of the WCS or a block coordinate system.

3D constraints have a number of properties that allow you to control how BricsCAD treats them in various situations.

9.3.2 Geometry-driven constraints

Dimensional 3D constraints can also be **geometry-driven**. They update to accurately reflect the current state of the model. You cannot specify their expression. Instead, their value is obtained from the model and can be changed with direct modeling operations and changes to other constraints.

Other parameters, including constraints, can depend on the values of geometry-driven constraints. In some circumstances, a constraint dependent on geometry-driven constraints can make the model unresolvable. This should be avoided.

You can turn regular constraints into geometry-driven constraints and vice versa, in the **Mechanical Browser** panel and the **Parameters Manager** panel or using the DMCONSTRAINT3D command. After a regular constraint is turned into a geometry-driven one, its expression will be overwritten with the first modification of the model that affects this constraint.

9.3.3 3D constraints and direct modeling

3D constraints are taken into account when direct modeling operations are applied. If there is a 3D constraint that fixes the placement of faces or edges, this constraint will be preserved during Push/Pull, Move and Rotate operations. A 3D constraint can be temporarily disabled, or erased permanently in the **Mechanical Browser** panel.

Geometry-driven constraints update their values when a direct modeling operation is completed, which possibly leads to additional changes in the model.

If some faces or edges disappear during direct modeling or Boolean solid editing operations, 3D constraints applied to such entities are removed. However, if there is an exact correspondence between the initial faces and the faces obtained, 3D constraints are automatically applied to the new entities.

Depending on the value of the DMRECOGNIZE system variable, BricsCAD automatically recognizes geometric relations between the surfaces of a solid and will preserve them during direct modeling operations. This feature is called **design intent recognition**.

9.4 Suppression feature

The suppression feature temporarily suppresses a set of entities in your drawing.

Suppressed entities are internally placed on a special frozen layer. This allows you to speed up orbit, zoom and pan operations, improves object selection performance and reduces regeneration time for complex drawings. Suppressed entities are also excluded from mass properties calculations (generated drawing views and bill of materials). Use a parametrically-driven expression for the condition parameter of a suppression feature, to create different design variants of your parametric components.

9.4.1 Creating a suppression feature

Execute the SUPPRESS command.

9.4.2 Managing suppression features

Select the suppression feature in the **Mechanical Browser** panel, then right-click to display a context menu.

The available options are:

- **Suppress**: Suppresses the entity set.
  - The **Suppressed** icon ( ) indicates the suppressed state.
  - The option now reads **Unsuppress**.
- **Unsuppress**: Unsuppresses the entity set.
  - The **Unsuppressed** icon ( ) indicates the unsuppressed state.
  - The option now reads **Suppress**.
- **Dissolve**: Deletes the suppression feature.
- **Edit Selection**: Allows you to add/remove entities from the selection.

  All members of the selection are highlighted.
  - Select entities to add.
  - Shift + Select (hold down the Shift key, then use a selection method on) entities to remove.
  - Right-click to stop editing.
- **Zoom to**: Zooms onto selected entities.

  To compose a selection set:
  - Click a suppression feature to select all its members.
  - Expand a suppression feature, then click to select an entity.
  - **Ctrl + Click** to add/remove entities from the selection.
- **Select**: Selects all members of a suppression feature.

9.4.3 Setting a condition for a suppression feature

You can set an expression condition for a suppression feature. Depending on the value of the current expression, the entities are suppressed or unsuppressed automatically.

In the Mechanical Browser panel:

1. Select the suppression feature (use the "Search" field to search by name).
2. Enter an expression in **Condition** field in the properties grid.

   Note: If the **Condition** field is equal to zero, all entities are displayed. Otherwise, entities are suppressed.

In the Parameters Manager panel:

1. Select the parameter with the same name as the suppression feature (use the "Search" field to search by name).
2. Enter an expression in the **Expression** field.

Note: If the **Expression** field is equal to zero, all entities are displayed. Otherwise, entities are suppressed.

9.5 Working with design tables

A design table allows you to evaluate several global parameters at once with the value of one configuration parameter, named configuration name. You can create design tables from scratch in BricsCAD or import them from a .csv file (a comma-separated values text file). You can create such files with a spreadsheet software package or a text editor.

9.5.1 Creating a design table from a .csv file

The **From file** option of the DESIGNTABLE command imports .csv files that contain sets of parameter values, or else creates them from scratch. Parameters are added to the **Mechanical Browser** panel and to parametric component properties.

The expected format of the .csv file is:

Configuration_Name,Parameter1,Parameter2,...,ParameterN

Configuration_1,Value1Parameter1,Value1Parameter2,...,Value1ParameterN

Configuration_2,Value2Parameter1,Value2Parameter2,...,Value2ParameterN

Configuration_3,Value3Parameter1,Value3Parameter2,...,Value3ParameterN

...

The first row lists the parameter names separated by a comma. The first name must be different from the name of any global parameter in your drawing. It is called the configuration name. No spaces are allowed in the configuration name. Other names in the first row can be identical to the names of a global parameter. The next rows list possible values for these parameters as strings or numbers.

Example:

Size,Length,Width,Height

Small,300,200,100

Medium, 450,300,150

Large,600,400,200

When the .csv file is imported into BricsCAD using the DESIGNTABLE command, a special global parameter is created by the name of the first name in the first row of the .csv file ('Size' in the above example). This is called the 'configuration name'. It will drive the values of other parameters in the design table. These are called 'configuration-dependent parameters'.

Note: BricsCAD does not maintain an association with the .csv file used to create a design table: if this file is modified, then the design table is not updated automatically. Use the -DESIGNTABLEEDIT command to replace a design table with a new or modified .csv file.

9.5.2 Direct lookup in a design table

To evaluate a configuration parameter:

- Select it in the **Mechanical Browser** panel and type the desired value in its **Expression** field.

- Select a value in the drop-down list of the **Value** field.

If you change the value of a configuration name, all configuration-dependent parameter values change automatically in the corresponding design table. The values of these parameters are taken from the row in the design table. The first element coincides with the current value of the configuration name.

9.5.3 Reverse lookup in a design table

You can evaluate parameters that are defined by a configuration name with a design table independently.

Select the dependent parameter and apply an expression to it.

- If the value of this expression matches one of the values in the corresponding design table, then the configuration name and all dependent parameters of the design table will automatically change to the values specified in the corresponding row in the table.
- If several configurations are possible for a particular value of a dependent parameter, BricsCAD selects the one closest to the current values of the other parameters.
- If you enter an expression and the result is not available in the design table, or if you enter expressions for two or more design table parameters and the values cannot be found in a single configuration (the values are not found in the same row in the design table), then the values of the configuration name are changed to 'Custom' and the values of the other dependent parameters are not changed. This case corresponds to a configuration that does not exist in the design table.

9.5.4 Adding a new configuration to a design table

You can add any 'Custom' configuration to a design table and assign it a name. To do so, right-click the configuration parameter in the **Mechanical Browser** and select **Save current configuration** in the context menu. The configuration will be added to the design tables as Configuration_N:

- **Configuration**: the name of the configuration.
- **N**: an index number - one more than the current number of configurations in the design table.

If you want to save a configuration in the design table under a different name, launch the -DESIGNTABLEEDIT command.

9.5.5 Creating an empty design table

If you use the procedure described above, it is possible to start your design with an empty design table and then fill it with the desired values.

Use the **Empty** option of the DESIGNTABLE command.

You will be prompted to specify:

- The name of a new design table
- The name of the corresponding configuration name
- The names of global parameters that will be dependent on this design table

9.5.6 Using a parametric component with design tables

You can use both configuration name and configuration-dependent parameters in assemblies. If you then insert such a component into your drawing, its parameters will include the exposed parameters of design tables. A drop-down list of available values will be available. Both direct and reverse lookup actions are supported for such parameters.

You must associate the design table with the parametric component drawing first.

1 Open the drawing.

2 On the **Mechanical Browser** panel, you will see the list of parameters and their values.

3 Launch the DESIGNTABLE command and press Enter to accept the **From file** option.

   The **Select design table file** dialog box appears.

4 Select the .csv file, then double-click or click the **Open** button.

5 Check the **Value** property for each of the main parameters. You can select the default values to use when you insert the component.

6 Check the **Exposed** property for each of the main parameters. They should all be set to **ON**.

7 Save the component drawing.

9.5.7 Other actions with design tables

The -DESIGNTABLEEDIT command allows you to:

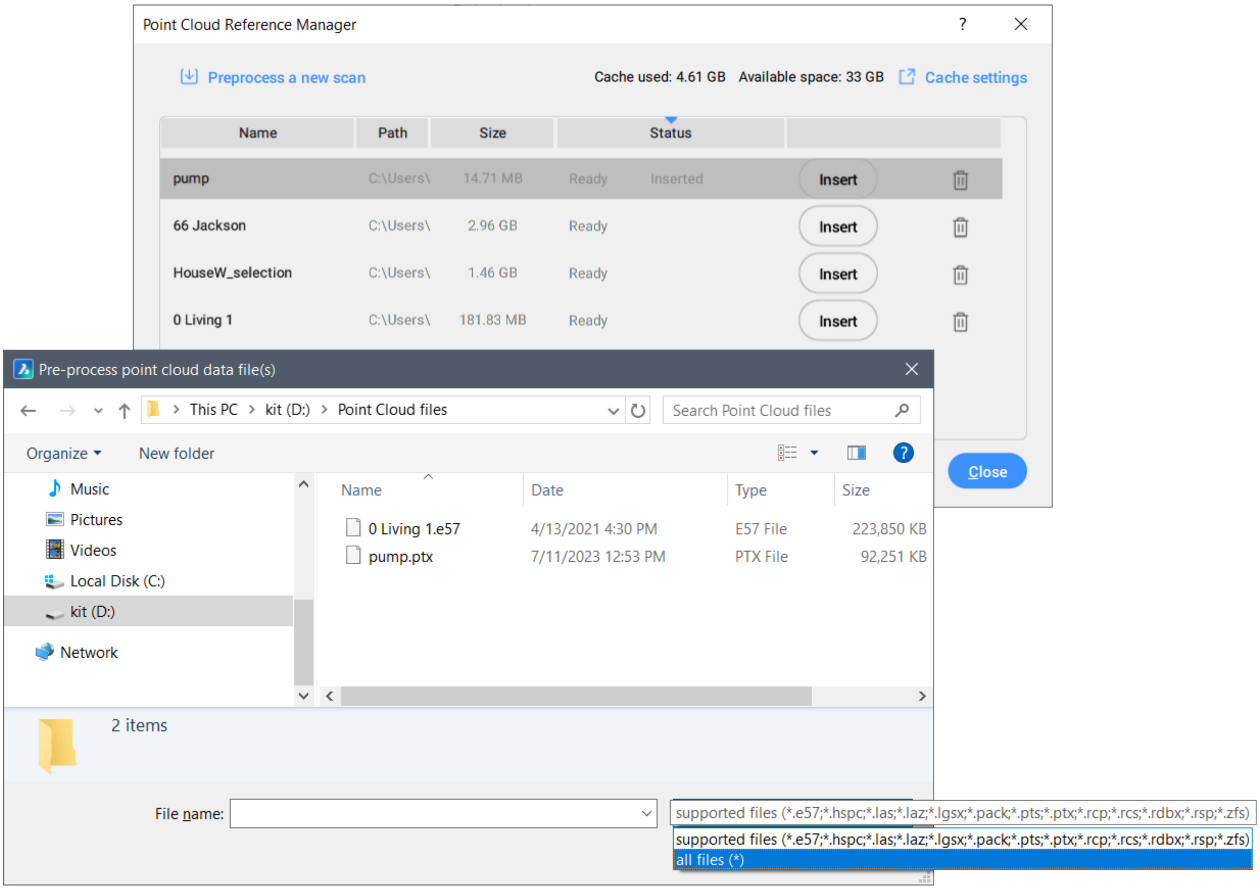
- Delete a design table
- Replace a design table
- Link a new parameter
- Unlink an existing parameter
- Add new configurations, remove an existing one
- Export a design table as a .csv file

10. Point cloud

10.1 Working with Point Clouds

10.1.1 Pre-processing

To pre-process a point cloud, use the POINTCLOUDREFERENCE command to open the **Point Cloud Reference Manager** dialog box, then select **Preprocess a new scan**. You can also use the POINTCLOUDATTACH command.



The progress of the pre-processing is recorded in a log file in the sub-folder related to the point cloud. This sub-folder is located inside the current point cloud cache path. The default path for the current point cloud cache is *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\PointCloudCache\Folder_for_processed_pointcloud*, and can be set in the POINTCLOUDCACHEFOLDER system variable.

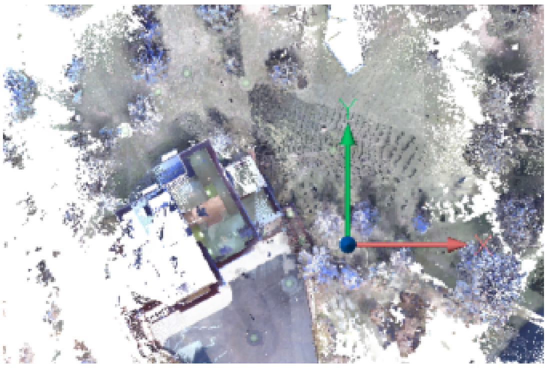
Note: The log file is mainly used for troubleshooting. You can follow the progress of the pre-processing in the **Point Cloud Reference Manager** dialog box.

Note: Some unusual point cloud input files, like LAS files with non-standard column layout or non-standard geo tags, are not automatically preprocessed correctly. However, there is also a shell executable pre-processor (**preprocessor.exe**) in the BricsCAD installation folder that allows you to specify non-standard column layout and/or geo tags (-w skip). The shell pre-processor uses the same library under the hood, but it exposes some manual options that are not automatically used by BricsCAD. To learn about the options, run the pre-processor executable without arguments in an OS shell. A usage summary will be printed. For more detailed information see the **How to use the preprocessor.exe in order to ignore GeoTIFF tags while preprocessing non standard .LAS, .LAZ pointcloud data?** article.

Note: Any subfolder with previously pre-processed point cloud cache data (from another BricsCAD point cloud cache folder or preprocessed with the shell executable preprocessor.exe) can be copied in the folder specified by the POINTCLOUDCACHEFOLDER system variable (default value: *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\PointCloudCache*) and will be available next time you will open the **Point Cloud Reference Manager**. For each subfolder in the point cloud cache folder, there is an item in the reference dialog box.

10.1.2 Alignment

The POINTCLOUDALIGN command automatically rotates a point cloud to optimally align it with the X and Y axis, taking into account the vertical planar surfaces in the point cloud (e.g. walls). To determine the best alignment, you will be asked to specify the most relevant area of the point cloud containing walls.



10.1.3 Bubble Viewer

Depending on the original file format of the point cloud and the type of scanner used during scanning (static or kinematic), bubbles (green spheres) or waypoints (blue spheres) might be shown at all the scan locations. At these locations, you will experience the most realistic visual representations by opening the **Bubble viewer**.

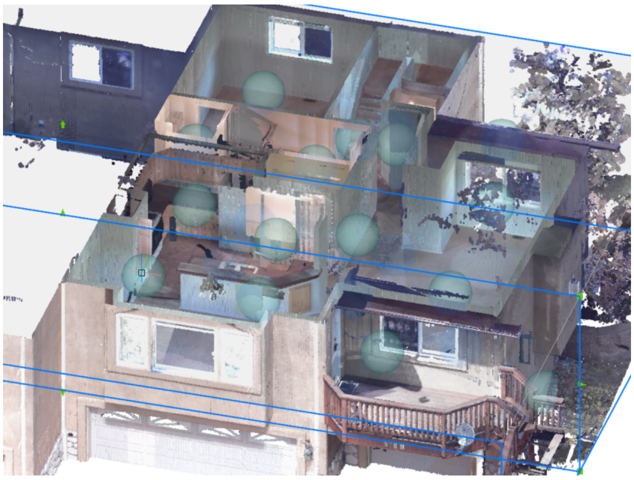
Note: LAS and PTS formats do not contain scanner information, so they cannot produce point cloud bubbles.

Structured data is captured using a static scanner. In this case the location of the scanner is known for every point. Green bubbles are created at the locations of the static scanner from the scanned points.

Note: The bubbles have depth information and can be used for algorithms like fit planar and fit cylinder.

Unstructured data is captured using a kinematic scanner. In this case there is no exact location from where the points were scanned. Some of the kinematic scanners create panoramic images at time intervals. For these images the location is known and waypoints (blue spheres) are created.

Note: The waypoints are purely created from the captured images which don't have depth information. They have a pure visualization functionality and cannot be used for other algorithms.



Note: Change the size of the displayed bubbles from the **Point Cloud Manager**'s **Render Settings** tab. From here you can also turn off the display of point cloud bubbles in Model space and **Bubble viewer**. For cropped point clouds you can choose to display only the non-cropped bubbles (see **Point cloud manager panel** article).

Indicate a bubble index in the POINTCLOUDBUBBLEVIEWER command or double-click one of the bubbles in the model space to open the **Bubble viewer**.

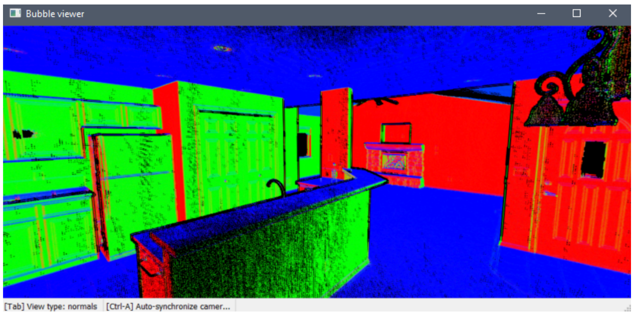
Press and hold the middle mouse button and move the mouse to view the point cloud in any direction from that scan location.

Zoom in and out using the mouse wheel.

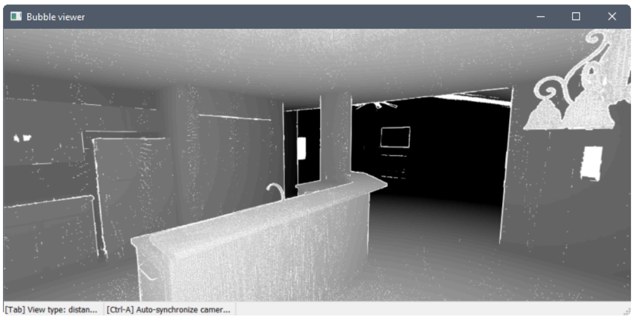
Nearby scans are indicated in the **Bubble viewer** by red bubbles. Hover the cursor over a bubble to display the scan name. Double-click a bubble to open the scan in the viewer.

Press the **Tab** key to cycle through three different visual modes (view types).

- The **colors** and **colors (depth)** modes display the points as their actual colors or in grayscale, depending on how the data was scanned.
- The **normals** mode displays the points as red, green, or blue according to their normal vectors. The colors correspond to the UCS axes.



- The **distances** mode displays the points from light to dark as the distance from the scan location increases.



Sync the drawing view to match the **Bubble viewer** by pressing **Ctrl+A**.

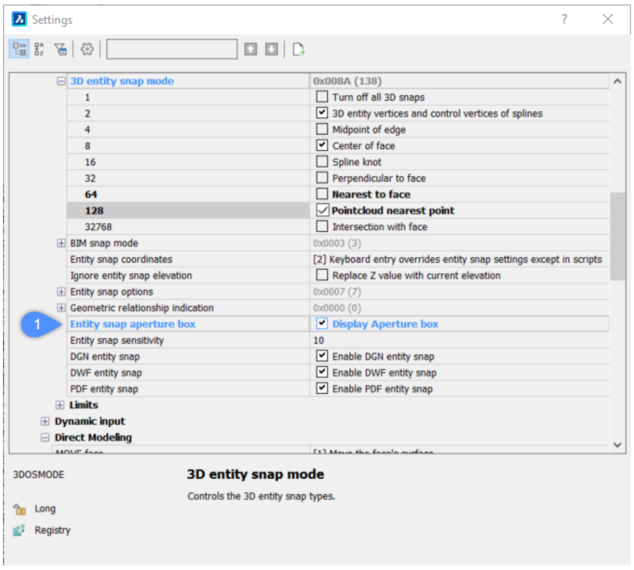
Note: During commands that prompt for selection of a point, e.g. (poly)lines, you can use the **Bubble viewer** to snap to point cloud points.

10.1.4 Entity snaps

The **Pointcloud nearest point** entity snap (PCNEAREST command/3DOSMODE system variable) significantly improves the ability to select relevant point cloud points. It uses an imaginary cylinder from the current viewpoint toward the cursor.

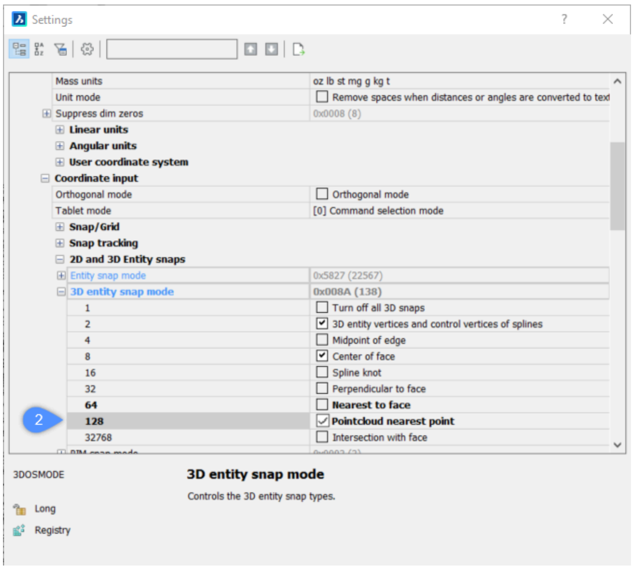
The radius of the imaginary cylinder is defined by the **Entity snap aperture box** setting (APBOX system variable) (1).

Note: The result is the point inside the imaginary cylinder that is the closest to the camera.



Enable the **Pointcloud nearest point** (2) entity snap together with other 3D entity snaps in the Entity Snap menus, toolbar and settings.

Note: Make sure that only the 128 bitcode of the 3DOSMODE system variable is ticked. (Untick **Turn off all 3D snaps**).

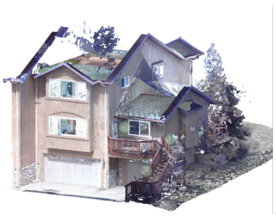


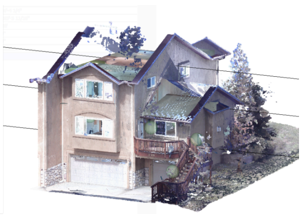
10.1.5 Export

The POINTCLOUDEXPORT command allows you to export the visible parts of a point cloud to a PTS, HSPC, or LAZ file.

10.1.6 Floor Detection

The POINTCLOUDDETECTFLOORS command detects floors in a point cloud of a building. The detection is based on regions of points with similar Z-coordinates. The detected floors can help in navigating point clouds of buildings.

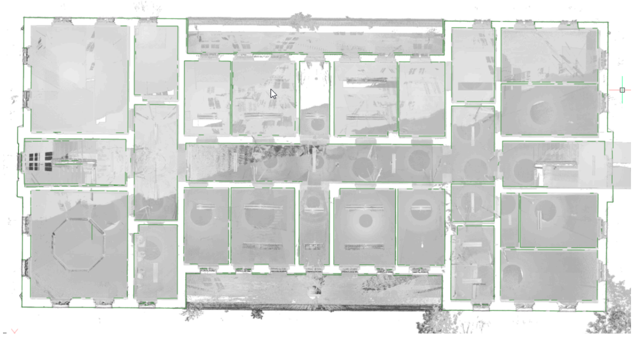




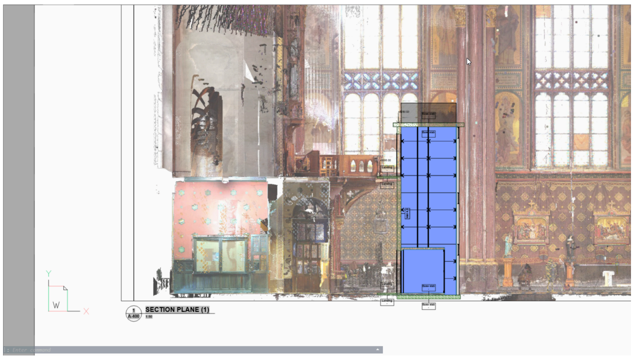
Note: The POINTCLOUDDETECTFLOORS command is used as a step in the point cloud Scan to BIM workflow. See the article **Point cloud Scan to BIM workflow**.

10.1.7 Point cloud projection

The POINTCLOUDPROJECTSECTION command enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections with the BIMSECTION command. You can use these sections to generate 2D lines to create a 2D floor-plan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way, it is possible to run this command in full resolution on all sections.



At the same time, a raster image is generated to give the user some context. In some cases, it is not necessary to recreate the existing building. Background images can give much more context to the design documents. These can be used to verify the created 2D geometry but in high quality scans, these images can also be used as graphical material. For example, as a background image for a BIM model in renovation projects where modern intervention are made in historical buildings.



10.1.8 Planar Fit

The POINTCLOUDFITPLANAR command enables you to create 3D geometry based on the point cloud. It will create a planar surface or solid after selecting a seed point in a point cloud. To detect the points that lie on the same plane as the seed point, a threshold value is used that can be set as a property of the point cloud entity. By holding down the **Shift** button and selecting the adjacent walls of a room, the fitted planes will be extended/trimmed to form a closed surface.

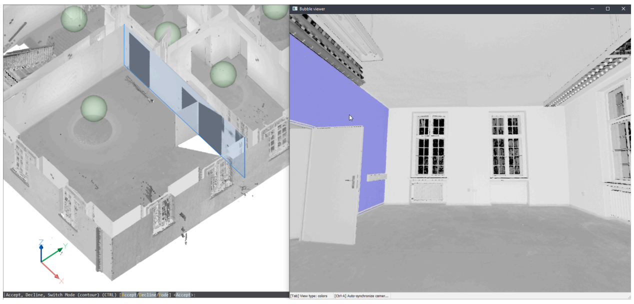
Note:

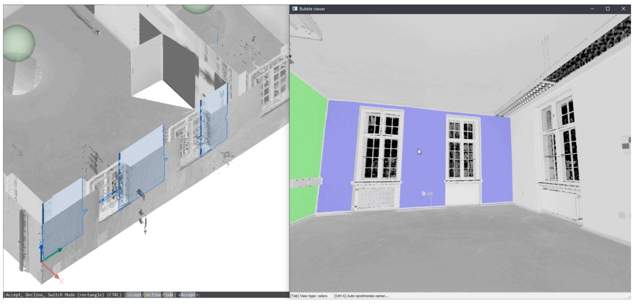
- **Select openings** allows you to take into account the openings of doors and windows (only in **Bubble viewer**).
- **Adjust borders** allows you to adjust the planar border contour.
- **Stitch surfaces** provides stitching the result of multiple fitted planes into, for example, a solid which can be later used with the BIMINVERTSPACES command.

In bubble view

If the **Bubble viewer** is open when the command is launched, BricsCAD expects you to select the plane seed points inside the **Bubble viewer**. The cursor will give you a preview of the normal direction of the plane. Next, you get a preview in both **Bubble viewer** and model view. You can toggle between the two shape representations using the **Ctrl** key.

Note: While still in selecting mode, the corresponding points in the bubble are shown in purple. These points are shown in green once the user accepts the generated result.

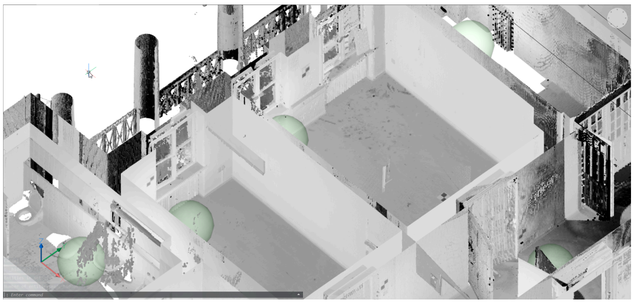


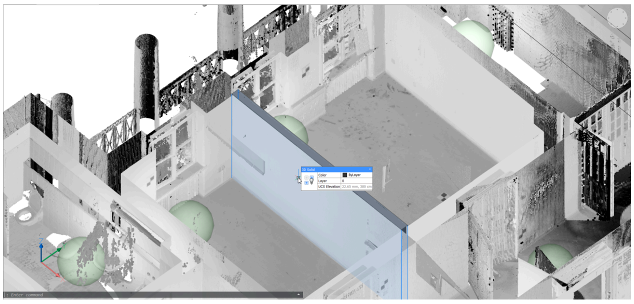


In model space

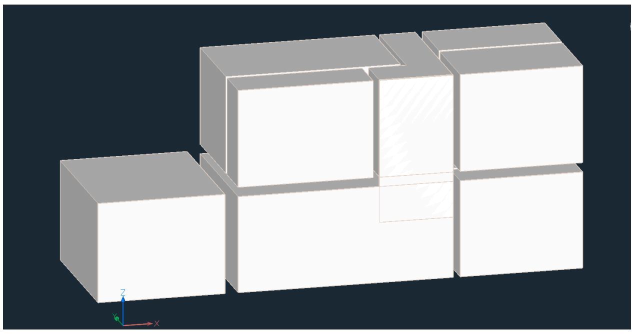
You can also use this command in the model space when the **Bubble viewer** is not open. BricsCAD will ask you to select a point of the point cloud in model space. Depending on the size of the visible parts of the point cloud, this might take more time compared to running the command inside the **Bubble viewer**. However, this has two advantages by searching multiple scan positions:

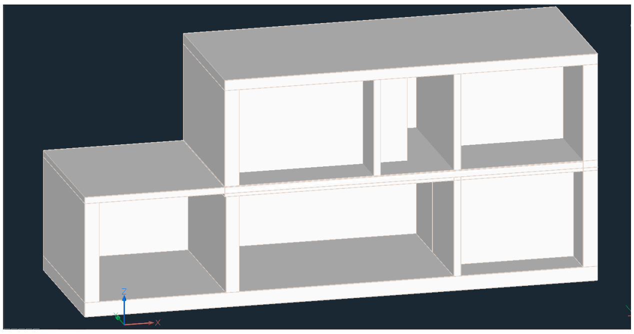
- It can create larger surfaces by combining parts of different scans.
- It can detect wall and slab thickness.





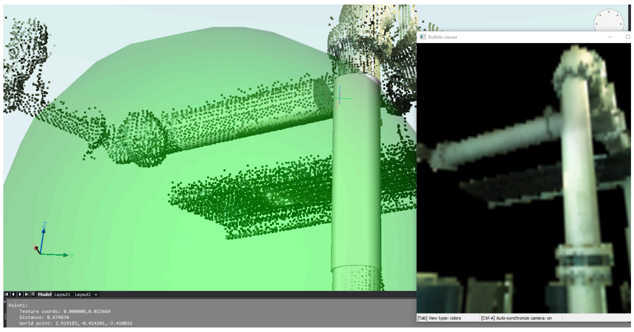
Note: Here, a work-flow of converting a scan to building model is provided. The point cloud (top image) is first converted to a set of solids, one per room, using stitched POINTCLOUDFITPLANAR results (middle image). The command BIMINVERTSPACES converts these temporary room solids into a building model with walls and slabs from the inversion of these solids.





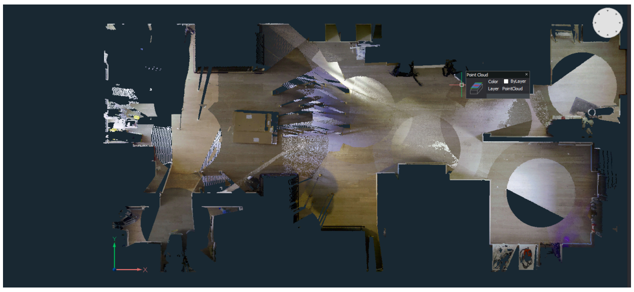
10.1.9 Fit cylinder in bubbles

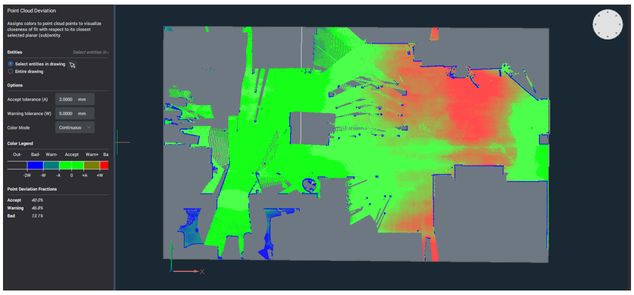
By using the POINTCLOUDFITCYLINDER command inside the **Bubble viewer**, you can fit a cylinder to the cylindrical parts of a point cloud (e.g. pipes) Two clicks along the cylinder axis in the **Bubble viewer** are needed to extract a cylinder (a solids is created in model view).



10.1.10 POINTCLOUDDEVIATION

- The POINTCLOUDDEVIATION allows you to visually assess the fit between planar structures and points of the point cloud.



- Point cloud section of a floor of a house and a fitted plane:
- 

  The distances of points from floor to the fitted plane are visualized using a colormap. In the left panel, an explanation of the colors is provided (green is planar, gradient from green to blue here is further above the plane, gradient from green to red is further below the plane). Also, a summary of the percentages of points in which category is presented (OK, warning level, ...)

10.2 HSPC (Hexagon Smart Point Cloud)

10.2.1 Introduction

The HSPC is the new point cloud format.

HSPC is a powerful, robust format for handling point cloud data. In BricsCAD, point clouds can be processed two ways:

- The classic point cloud cache format with all of its advantages: strong compression, best graphical representation of point cloud data.

- The HSPC format, a format created by the Hexagon research group, allows for the storage of metadata, per point, during or after pre-processing.

10.2.2 About HSPC

The HSPC format can result in slightly less appealing graphical representation (surfaces may appear less dense in some regions). However, the advantage of this format is that we are able to limit the displayed points based on the available metadata. If you choose this cache format to import a structured, classified point cloud, you will be able to:

- Show, create and edit **regions** in point clouds. This allows you to store any cropping you make on a point cloud. The regions are stored next to the cache, so they are available in any drawing you attach the point cloud to.
- Toggle the visibility of scan positions. You are no longer limited to show the entire registered point cloud (i.e., the sum of all fixed scan positions). You can choose to work only on the data of one or more scan positions, reducing the number of points displayed, and therefore hugely accelerating performance and avoiding the need to manually crop the point cloud to provide a performant, workable dataset.
- Use any classification inside the imported point cloud. Again, this allows you to work on a limited data set. Alongside the supported predefined classes in LAS/LAZ format, you can add classifications using the POINTCLOUDCLASSIFY command.
- Access the geographic information of a point cloud (when it is available in the original data) in the **Properties** panel (under the **Misc** category) and use it to overwrite the DWG geographic information by setting the **Geolocate**property to On (possible only if the CRS of the point cloud is the same as the CRS of the DWG file).

You can choose the point cloud processing format in the **Settings** dialog box, by defining the value of the POINTCLOUDHSPC system variable. The default cache format is set to HSPC.

10.3 Point cloud Scan to BIM workflow

The point cloud Scan to BIM semi-automatic workflow consists of four steps:

1. Detecting floors
Use the POINTCLOUDDETECTFLOORS command to detect the floors. A floor number is assigned to each selected point in the point cloud.

2. Detecting rooms
Use the POINTCLOUDDETECTROOMS command to detect rooms. A room number is assigned to each selected point in the point cloud.

3. Creating room solids
Use the POINTCLOUDFITROOMS command to create room solids based on the previously found rooms.

4. Creating BIM components
Use the BIMINVERTSPACES command to create walls in between the selected solids, wall openings, outer walls and slabs.

Note: The workflow is applicable for point clouds for which normal vectors information is available.

For structured point cloud data, normal vectors are calculated during point cloud preprocessing. In case a point cloud has been preprocessed on an older BricsCAD version where normals have not yet been calculated, use the POINTCLOUDNORMALS command first.

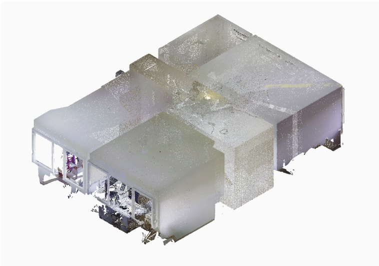
For unstructured point cloud data, normal vectors information is imported, if present, and used in the workflow. Otherwise, it is not calculated, and the workflow is not applicable.

Note: Normal vectors calculation requires a point cloud processed in HSPC format. Both POINTCLOUDHSPC and POINTCLOUDNORMALS system variables need to be set to On (On by default).

10.3.1 Detecting floors

The first step of the Scan to BIM workflow is to detect the floors and ceiling in the point cloud attached to the drawing.

Note: To learn more about attaching point clouds, see **POINTCLOUDATTACH command** and **POINTCLOUDREFERENCE command** articles.



1 Launch the POINTCLOUDDETECTFLOORS command.

1 **You are prompted**: Select point cloud entity:

2 Select a point cloud attached to your drawing.

3 **You are prompted**: Operate on full point cloud or a selection?

4 Do one of the following:
   - Click **Full** to operate on the full point cloud.
   - Click **Selection** to specify the first point of the selection rectangle, then the opposite point.

   You are prompted: Create spatial elements?

5 Select **Yes** to create spatial elements based on the detected floors height.

6 A message appears in the Command line to let you know when the floors detection process is ready.

7 The command will detect floors for the selected points. The floors are listed in the **Point Cloud Manager** panel under **Buildings**.

8 (Optional) To edit the selection of points assigned to a floor, right-click the floor listed in the **Point Cloud Manager** panel and select the **Modify** option from the context-menu.

10.3.2 Detecting rooms

The second step of the Scan to BIM workflow is to detect the rooms in a floor.

1 Launch the POINTCLOUDDETECTROOMS command.

1 **You are prompted**: Specify floor id or select all.

2 Do one of the following:
   - Click **All** to detect rooms for all the floors listed in the **Point Cloud Manager** panel.
   - Click **Id** to detect rooms for the floor with the specified Id.

3 The command starts the background processing. Depending on the point cloud size, this can take up to several minutes.

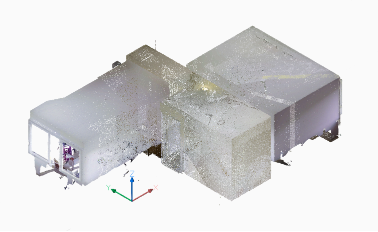
4 ***Note:*** You can continue working during the background processing.

When the background processing finishes, a pop-up message appears. The points found in the specified floors are classified under different rooms. The detected rooms are listed in the **Point cloud manager** panel under **Buildings** with an Id assigned to each.

To check that a room is correctly detected, toggle on/off the room visibility state icon next to the room's name.

Select a room and right-click to access the context menu. Select **Edit** to adjust the room's contour by dragging the polyline points, then select **Accept** or **Decline** from the room's context menu. Use **Rename** or **Delete** options if needed. Merge multiple rooms by selecting the rooms in the list and using **Merge** from the context menu.

To add a new room to a floor, right-click the floor's name in the **Point cloud manager** panel, select **Add Room** and draw a polyline to define the new room's boundary.



Once the points in the selection have been classified into different rooms, you can create solids based on this classification.

10.3.3 Creating room solids

The third step of the Scan to BIM workflow is to create room solids based on the detected rooms.

1 Launch the POINTCLOUDFITROOMS command.

1 **You are prompted**: Specify building Id or select all.

2 Do one of the following:
   - Click **All** to create solids for all the detected rooms.
   - Click **Id** to create a solid for the detected room with the specified Id.

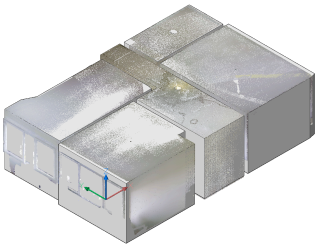
You are prompted: Restrict detection to perpendicular (90°) walls?

3 Select **Yes** if the rooms have two perpendicular wall directions.

The command starts the background processing. Depending on the point cloud size and wall directions, this can take up to several minutes.

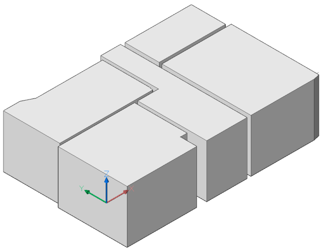
Note: You can continue working during the background processing.

When the background processing finishes, a pop-up message appears. The created solids are assigned to the automatically generated **Rooms** layer.



4 Turn off the attached point cloud's visibility. The rooms solids are now visible.

5



Optionally, you can edit the rooms by using the DMPUSHPULL command to adjust the faces of the created solids.

10.3.4 Creating BIM components

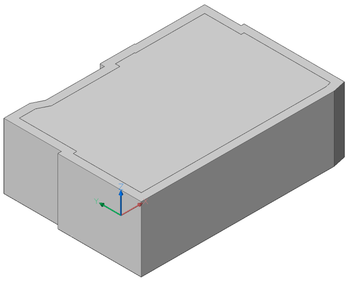
The last step of the Scan to BIM workflow is to generate walls in between the previously created solids, as well as the outer shell.

1 Select a layer on which the BIM components will be created.

2 Launch the BIMINVERTSPACES command. The **Invert Spaces** Command Context panel opens to let you define the settings. For detailed information about each option, see the article **BIMINVERTSPACES command**.

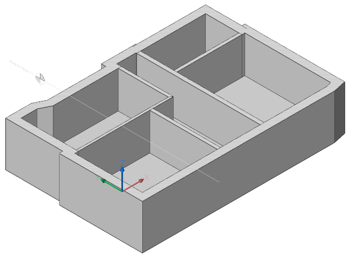
3 Click **Apply**.

The Scan to BIM workflow is now complete, with the 3D model based on the point cloud being fully generated.



Next you can verify and edit the result:

1 Turn off the layer on which the room solids were created.
2 Use the BIMSECTION command to create a horizontal section plane that displays the internal walls.
3 Use wall editing tools like the LCONNECT or BIMEXTEND commands to adjust the result as needed.
4



11. Blocks, attributes, external references and components

11.1 Creating blocks

The BLOCK command groups existing entities into a block using a dialog box.

The -BLOCK command creates blocks from existing entities and prompts you in the command bar.

The BEDIT and BCLOSE commands create or edit blocks in a temporary drawing session.

The WBLOCK command writes blocks and other drawing parts to a separate drawing file (short for "write block").

The PASTEBLOCK command pastes entities from the **Clipboard** as a block in the current drawing.

The SECTIONPLANETOBLOCK command saves the selected section plane result to a 2D cross section / elevation block or a 3D cutaway section block.

11.1.1 To create a block

1 Optionally select entities for the block.

2 Launch the BLOCK command.

3 The **Create Block Definition** dialog box appears:

4 Type a name for the new block in the **Name** field.

5 Do one of the following to specify the **Base Point**:

- Check the **Specify On-screen** option.

  You will be prompted to specify the base point after clicking the **OK** button.

- Click the **Pick point** button ( ).

  The **Create Block Definition** dialog closes temporarily to let you pick a point in the drawing.

- Type the coordinates of the base point in the X-, Y- and Z-field.

6 The number of preselected entities in step 1 is indicated.

7 To select the more entities for the new block, do one of the following:

- Select the entities before starting the BLOCK command.
- Check the **Specify On-screen** option.

  You will be prompted to select the entities after clicking the **OK** button.

- Click the **Select entities** button ( ).

  The **Create Block Definition** dialog closes temporarily to let select the entities in the drawing.

8 Set the block properties:

- **Annotative**: creates an annotative block.
- **Match block orientation to layout**: if checked, the orientation of an annotative block is preserved in a rotated viewport.
- **Scale uniformly**: if checked, the X-, Y- and Z-scaling cannot be set differently.
- **Allow exploding**: if not checked, prevents the block from being exploded.

- **Block unit**: sets the insertion unit for the block (see also: Inserting Blocks).

9 Set the behavior of the selected entities after the block is created:
  - **Convert to block**: The selected entities are replaced by the newly created block.
  - **Don't convert to block**: The selected entities remain in the drawing.
  - **Delete**: The selected entities are deleted.

10 Click the **OK** button to create the block.

Note:

- If the **Delete** option in step 7 was set, the OOPS command brings back the erased block entities.
- If you type a name of an existing block in step 3, you will be prompted to overwrite the existing block definition. Existing block instances will be replaced with the new block definition.

11.1.2 To save a block to a separate file

1 Optionally select entities for the block.

2 Launch the WBLOCK command.

3 The **Write Block to File** dialog box opens.

4 Click the browse button next to the **Name and path** field.

The **Save Block** dialog box is displayed.

5 Select a folder and type a name in the **File name** field, then click the **Save** button.

The **Save Block** dialog closes.

6 **Insert units**: sets the value of the INSUNITS system variable for the block file.

By default this is the INSUNITS value of the source drawing.

7 The number of preselected entities in step 1 is indicated.

8 To select the more entities for the new block, do one of the following:
  - Click the **Block** radio button, then select a block from the drop-down list.
  - Click the **Entire Drawing** radio button, to select all the entities in the model space of the current drawing.
  - Click the **Entities** radio button, then click the **Select Entities** button.

    The **Write Block to File** dialog temporarily closes to let you select entities in the drawing.

    Right click to stop selecting entities.

9 Click the **Pick point** button () to define the **Base point** of the new drawing

or

Type the coordinates of the base point in the X-, Y- and Z- field.

10 Set the behavior of the selected entities after the block is created:
  - **Retain**: The selected entities remain in the drawing.
  - **Convert to block**: The selected entities are replaced by the newly created block.

- **Delete**: The selected entities are deleted.

11 Click the **OK** button to create the drawing file.

Note: If the **Delete** option in step 8 was set, the OOPS command brings back the erased block entities.

11.2 Using Blockify

The **Blockify** tool enables you to automatically convert repetitive sets of entities to block definitions. Select a set of 2D entities or 3D solids and BricsCAD searches the drawing for identical sets of entities and automatically replaces them by block references.

For more information about this command, see the BLOCKIFY Command Reference article.

11.2.1 Blockify from a selected input

1 Launch the BLOCKIFY command.

1 You are prompted: Select source entities or [match Equal solids/find Collections/match existing Blocks/selection options (?)] <match Equal solids>:

2 Select the entities you want to convert into a block and press Enter.

3 Entities selected: xx

You are prompted: Select source entities or [Settings/selection options (?)]:

4 Select more entities or press Enter or right click to stop selecting.

5 You are prompted: Select block insertion point or [use Default point] <use Default point>:

6 Identify the insertion point for the block.

7 You are prompted: Select search space or [Entire drawing/selection options (?)] <Entire drawing>:

8 Press Enter or right click to search the entire drawing.

9 You are prompted: Specify block name or <ENTER to use default name>:

10 Type a name for the block and press Enter.

11 You are prompted: Select matches to replace with block references [Replace/cAncel] <Replace>:

12 Click on the individual green check marks/red cross signs in the model space to further select/ deselect the entities you want to be replaced by block references and press Enter. (or press Esc/A to cancel the command)

13 BricsCAD reports:

Created new block definition 'block_name'

Inserted 'x' block references

Note: Depending on the value of the BLOCKIFYMODE system variable, some prompts are suppressed and defaults are used for the insertion point, search space, block name and/or matches to replace.

11.2.2 Existing block definitions

One of the aims of the BLOCKIFY command is to automatically add and improve the structure of a model. If the model already contains block definitions, the existing block definitions are checked.

- **Use a selected input set of entities**: when one of the existing block definitions matches the BLOCKIFY input set, no new definition will be created and a message will appear in the Command line: Using existing block definition 'block_name'. You are prompted first to select the search space and then to select the found matches to replace. To do that, click on the individual green check marks/red cross signs in the model space or press Enter/R to accept the current selection. The selected elements will be replaced by block references of the existing block definition.
- **Match existing blocks**: the entire drawing or selection set is searched for geometry that matches the existing block definitions. Tick the **Show preview** option in the **Settings** area of the Command Context panel. Next, select from the Command Context panel which of the found group(s) matching existing block definitions to be replaced by block references. For each selected group you can then select/deselect individual entities in the drawing by clicking on the green check marks/red cross signs.

  Note: You can use this option to recreate block instances that were accidentally exploded.

11.2.3 Combining duplicate block definitions

Identical block definitions can be combined into a single one and optionally purge the duplicate using the OVERKILL command.

1 Launch the OVERKILL command.

1 You are prompted: Select entities or [Combine duplicate block definitions/selection options (?)]:

2 Choose the **Combine duplicate block definitions** option.

The **Combine duplicate block definitions** dialog box appears.

3 (option) Select the properties to ignore.

4 (option) Define the comparison tolerance which is set to 1e-06 by default.

5 (option) Tick the **Purge duplicate block definitions** option.

6 Click the **Apply** button.

The **Overkill** Command Context panel opens and displays:

- All groups of duplicate block definitions found.
- Names of duplicate block definitions for each group.

Select the groups of blocks you want to combine and select the block definition names to keep. Then click **Combine groups** to finish the command.

11.3 Editing blocks

You can access the **Block Editor** with the BEDIT command or by double-clicking on the block you wish to edit. When using the **Block Editor**, BricsCAD displays the block entities in a temporary drawing session. It includes all the tools you need to create and edit a block definition.

To exit the block editing session, use the BCLOSE command and choose whether to save or discard the changes.

You can use the BSAVEAS command while in the block editing session to copy the current block definition to a new name. Call the command at the Command line or click the icon in the Ribbon.

If you use the SAVE command while in the block editing session, BricsCAD saves the changes to the block definition and exits the **Block Editor**.

Blocks can also be edited using the REFEDIT command.

11.4 Inserting blocks

The INSERT command lets you insert blocks through a dialog box.

The -INSERT command inserts blocks by prompting in the Command line. BricsCAD searches the block name in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the SRCHPATH system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'.

The INSERTALIGNED command inserts a block entity, with easy alignment on existing entities. The INSERTALIGNED command is similar to the INSERT command, but during the placement the block dynamically aligns with existing lines, polylines, arcs or circles. The block aligns with the entity to which the cursor snaps. If no entity is snapped, the block is aligned with the X-axis of the WCS.

The MINSERT command inserts a block as a rectangular array; combines the -INSERT and ARRAY commands (short for "multiple insertion").

You can choose to select an existing block definition or to insert an entire drawing as a block. When you insert a drawing, a new block definition is created in the current drawing. If you change the original drawing file, those changes have no effect on the current drawing unless you redefine the block by reinserting the changed drawing. When a block is inserted in a drawing, it is treated as a single entity.

Note:

- When inserting a drawing as a block the origin is defined by the INSBASE system variable of the inserted drawing. The variable is set by the BASE command.
- If a block contains attributes, you will be prompted to fill out the text for each attribute in the Command line.
- If the DRAGOPEN system variable is set to zero (OFF), drawings can be inserted as a block by dragging them from the **Windows Explorer** dialog into the current drawing; if DRAGOPEN = 1 (ON), the drawing will be opened instead.
- The INSUNITS and INSUNITSSCALING system variables control the automatic scaling when inserting blocks or attaching Xrefs when the value of INSUNITS is different in the block or Xref and the target drawing.
- If the value of INSUNITS in the source drawing is 0 (Unspecified), the value of the INSUNITSDEFSOURCE system variable is used instead.
- If the value of INSUNITS in the target drawing is 0 (Unspecified), the value of the INSUNITSDEFTARGET system variable is used instead.

- The values of INSUNITSDEFSOURCE and INSUNITSDEFTARGET are saved in the registry and therefore apply to all drawings in which the value of the INSUNITS system variable is not specified (unitless).
- When you type the name of a block at the ‘Block to insert:’ prompt, BricsCAD will look up the block in the block definitions in the current drawing. If the block is not found there, the paths defined by the SRCHPATH system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'.

In the **Insert Block** dialog box:

- The **Unit** field under **Block Unit** is set by the INSUNITS system variable in the source drawing or the **Block Unit** property of a block definition (see Creating Blocks).
- The **Factor** field expresses the relation between the value of the INSUNITS variable in the source drawing and the target drawing. E.g. if INSUNITS is **Millimeters** in the *source* and **Centimeters** in the *target*, the value of the **Factor** field is **0.1** and if INSUNITS is **Millimeters** in the *source* and in the *target*, the value of the **Factor** field is **1**.

11.4.1 Using the INSERT command

1 Launch the INSERT command.

1 The **Insert Block** dialog box appears.

2 Do one of the following:

- Select an existing block definition in the **Name** list.

  The name of the selected block is displayed in the **Name** field.
- Click the **Browse** button and select a drawing file.

  The path field shows the location of the selected drawing.

3 (option) Check the **Use reference curves for insert** option.

4 Set the **Insertion Point**, **Scale** and **Rotation** modes.

The options are:

- Specify the settings in the **Insert Block** dialog.
- **Specify On-screen**: You will be prompted during the insertion process.

5 (option) Check the **Explode** option to explode the block upon insertion.

6 Click the **OK** button.

You will be prompted in accordance with the selected options.

11.4.2 To insert a block using the Drawing Explorer dialog box

Inserting blocks from the current drawing.

1 Launch the EXPBLOCKS command.

1 The **Drawing Explorer – Blocks** dialog box appears.

2 (option) Set the view mode: **Detail** or **Icon**.

3 (option) Select a block, then right click and choose **Options...** in the context menu.

The **Drawing Explorer Options** dialog box appears.

Set the insertion options, then click the **OK** button.

4 Double click the block icon or the number when in **Detail View** mode.

The **Drawing Explorer** dialog closes to let you insert the block in the drawing.

5 Specify the insertion point.

- Depending on the current insertion options, you are prompted to specify the scale and/or rotation.
- When the **Align** option is selected, you must specify the position of the block with respect to the selected linear entity.

  You are prompted: Enter block mirroring control point:

The **Drawing Explorer** dialog box reopens.

Inserting blocks from another drawing.

1 Open the **Drawings Explorer** dialog box.

2 In the **Drawings** pane, click the **Folders** tab.

3 (option) If no folder are available yet in the **Local Folders** tree, or if you want to add a folder, click the **Add local folder...** button.

The **Choose a folder** dialog box appears.

Browse to the folder you want to add, then click the **Select Folder** button.

The folder and its subfolders are added in the **Local Folders** tree.

4 Select a drawing in the **Local Folders** tree.

5 Expand the drawing, then select **Blocks**.

All blocks in the selected drawing are displayed in the **Blocks** pane.

6 Proceed from step 2 in the previous procedure.

11.4.3 To insert a block aligned with an entity

1 Make sure the appropriate Entity Snaps are active.

2 Launch the INSERTALIGNED command.

3 **You are prompted**: Block to insert (~ to open file dialog) or [? to list blocks in drawing] <current block>:

4 Do one of the following:

- Press Enter to accept the currently selected block.
- Type the name of the block to insert.
- Press the ? key to see a list of available blocks.

You are prompted: Insertion point for block or [Scale/X scale/Y scale/Z scale/Multiple blocks]:

5 (option) Choose the **Multiple blocks** option or specify a fixed scale.

6 Specify a point.

7 The block displays the selected entity snap point, aligned with the linear entity.

You are prompted: Enter block mirroring control point:

8 Move the cursor to select the desired position, then click to insert the block.

Note: The block is aligned as follows:

The block is rotated so that its local Z-axis is parallel with the Z-axis of the active UCS, and its local X-axis should be parallel with the tangent to the entity at the position of the snap point. When you click a point on an entity, the block is aligned with the entity. Then, by moving the mouse pointer around the insertion point, you can choose how the block should be mirrored. The origin point of the block lies on the entity.

11.4.4 To insert multiple instances of a block in a rectangular array

1 Launch the MINSERT command.

1 **You are prompted**: Block to insert (~ to open file dialog) or [? to list blocks in drawing] <current block>:

2 Do one of the following:

- Press Enter to accept the currently selected block.
- Type the name of the block to insert.
- Press the ? key to see a list of available blocks.

You are prompted: Insertion point for block or [Scale/X scale/Y scale/Z scale/Multiple blocks]:

3 (option) Choose the **Multiple blocks** option or specify a fixed scale.

4 Specify a point.

5 **You are prompted**: Rotation angle for block <0°0'0">:

6 Press Enter to accept the default rotation angle or specify an angle.

7 **You are prompted**: Number of rows in the array <1>:

8 Specify a number.

9 **You are prompted**: Number of columns <1>:

10 Specify a number.

11 **You are prompted**: Vertical distance between rows, or spacing rectangle:

12 Do one of the following:

- Specify a value.
- **You are prompted**: Horizontal distance between columns:

  Specify a value.
- Click two point to specify a rectangle, which sets the horizontal and vertical spacing distances.

The block is inserted as a **Minsert Block** entity.

Note:

- When you choose **Multiple blocks** in step 3, you are prompted to create multiple instances of the block array.
- It is not possible to explode a **MInsert Block** entity.
- You can edit the properties of a **MInsert Block** entity in the **Properties** panel.

11.5 Exploding blocks

If the **Explodable** property of a block is set, you can explode an inserted block to its original component entities. Exploding a block affects that single instance of the block only. The original block definition remains in the drawing and you can still insert additional copies of the original block. If you explode a block that contains attributes, the attributes are lost, but the original attribute definitions remain.

The EXPLMODE system variable controls whether it is possible to explode non-uniformly scaled blocks.

11.5.1 To explode a block

1 Launch the EXPLODE command.

1 **You are prompted**: Select entities to explode [selection options (?)]:

2 Select one or more blocks.

3 Entities in set: x

You are prompted: Select entities to explode [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 The selected blocks are exploded.

Note: Exploding dissociates component entities to their next simplest level of complexity: nested blocks or polylines in a block become blocks or polylines again.

11.5.2 To set the Explodable property of a block

The **Explodable** property of a block is set when creating a block using the BLOCK command.

To edit the property, do the following:

1 Launch the EXPBLOCKS command.

1 The **Drawing Explorer – Blocks** dialog box appears.

2 Choose the **Detail view** mode.

3 Click the **Explode** tick box to toggle the **Explodable** property.

11.6 2D parametric blocks

11.6.1 Context

Parametric blocks are blocks which contain not only pure geometry, but also some metadata, which affect the geometry (for example their size, visibility). The metadata controls the way the block references behave. Thus, the size and appearance of a parametric block can be modified without editing the block definition.

11.6.2 Workflow

This workflow will describe one way of working with 2D Parametric Blocks, making use of some advanced features that BricsCAD offers for creating these blocks.

One use case of the 2D Parametric Blocks is architectural symbols. The symbol used in this example is a simple parametrized door.

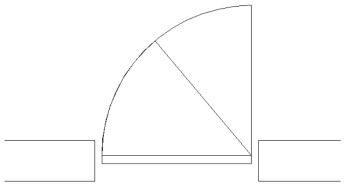
Note: In this case, the block will be built in a separate drawing. Then, it will be inserted in the target drawing using the INSERT command and choosing the file from browser.

11.6.3 Creating the geometry

To ensure all the necessary tools are easily accessible, it is helpful to set the workspace to **Drafting**.

The geometry will contain the entities to represent three distinct configurations of the door. These configurations are called **visibility states**. When creating the geometry, the entities for all the states will be drawn.

For this example, the block geometry will look like this:



11.6.4 Adding the visibility states

To start creating the visibility states, open the **Visibility States** panel. To do this, go to the **2D Parametric** tab of the ribbon. In the **Operations** panel, click the **Visibility** button (or right-click the ribbon and select **Panels** > **Visibility States**). You can also launch the VISIBILITYSTATESPANELOPEN command in the Command line.

For more information about the visibility states, see the article **Visibility states panel**.

Next, three states of the door symbol will be defined. These states are created as values of a visibility parameter. This operation can be achieved using the **Visibility States** panel mentioned above.

The workflow for this task is the following:

- Create a visibility parameter.
- Add three states to it.
- Add entities to the parameter.
- Make attached entities visible or invisible depending on which state is active.

To add the visibility states to the block, follow the next steps:

1 Open the block drawing.

2 To add a visibility parameter, click the **Add parameter** button (⊕) at the top of the **Visibility States**panel. A new parameter will be created, and its name will be available to edit. For this example, the visibility parameter will be called **Angle**.

3 To add a state, click the **+Add state** button under the parameter name label. A new visibility state is created, and its name is available for edit. For this example, the names of the three visibility states are **Open**, **Half Open** and **Closed**.

4 To add entities to a state, hover over the state name label and click the menu button (⋮) that appears at the right end. From the context menu, choose **Make entities visible**. This option allows you to select

the entities that will be made visible to this state only. To attach entities to each of the three states, follow the next steps:

- Make the **Open** state active and add the vertical line and the larger arc.
- Make the **Half Open** state active. The entities added to **Open** will be hidden. Add the slanted line and the smaller arc.
- Make the **Closed** state active. The entities for **Open** and **Half Open** will be hidden. Because the entities for the other states have been automatically hidden when they were attached, there are no changes to be made. This happens for the last state of the parameter.

Note:

a When a certain visibility state is active, to view the entities that are hidden for that state, click the menu button (⋮) at the top right corner of the panel and then click **Show invisible**. The hidden entities will be shown in light blue. To hide them again, deselect the **Show invisible** option.

b There is a second way to add entities to a visibility parameter. Click the arrow button (>) at the right of one of the state name. A new state specific panel opens. Then click the Select add ( ) button at the top of the panel and select the desired entities.

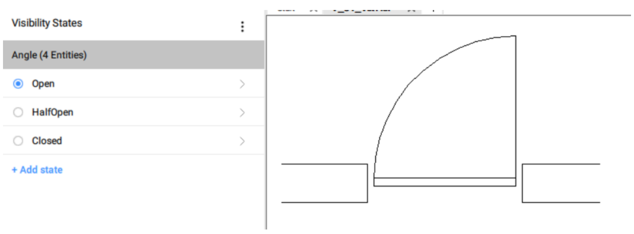
c To remove entities from a visibility parameter, click the arrow button (>), then click the Select remove ( ) button and select the entities to be removed from the parameter.

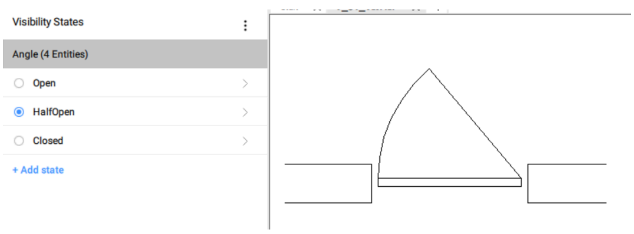
d If, by mistake, some entities are specified to be visible in all the states of the parameter, they will appear in the state specific panel with the label **Visible for all states of this parameter**. These entities are redundant, and they should be removed from the parameter.

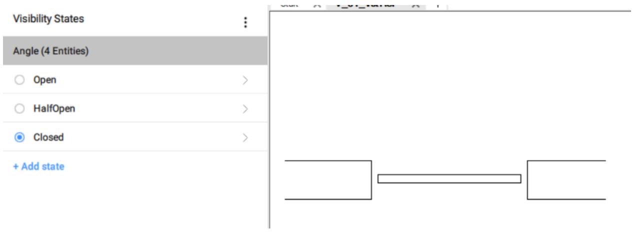
e To remove many entities at once, select them using the **Shift** key. Then, right-click and select **Remove from parameter**.

f To edit the visibility of the entities for a certain state, make the state active and click the arrow button (>). The state specific panel opens and displays a list containing the entities that are attached to the parameter. At the left side of each entity label there is an eye icon. If the eye icon has a slanted line over it ( ), then the entity is hidden for that state. Clicking on the icon will toggle the visibility of that entity.

This is how the block will look for each visibility states that were defined:







5 Save the block drawing.

6 Insert the block into the target drawing.

7 To change the visibility state, follow these steps:

8 - Select the block.
  - Open the **Properties** panel and go down to **Parameters** section.
  - Select a visibility state from the **Angle** parameter drop-down list.

11.6.5 Adding the reference curves

Reference curves are entities which are created on the **REFERENCE_CURVES**layer. These specific entities of a block will be used to align it with entities in the target drawing. The alignment will take place during the insert operation.

The block will be placed in the desired position without the need of further editing operations as moving or rotating.

For more information on reference curves, see the article **REFERENCECURVES command**.

To add the reference lines, follow the next steps:

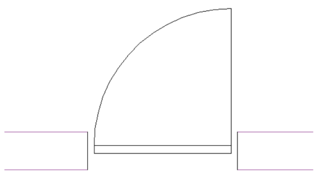
1 Open the block drawing.

2 Launch the REFERENCECURVES command in the Command line. You can also go to the **2D Parametric** tab of the ribbon. In the **Operations** panel, click the **Reference curves** button.

3 Select the four horizontal lines that represent the wall. These lines are now put on the **REFERENCE_CURVES** layer.

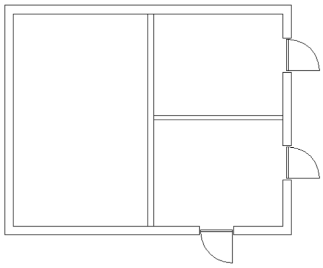
4 Save the block drawing. The block will look like this:

5



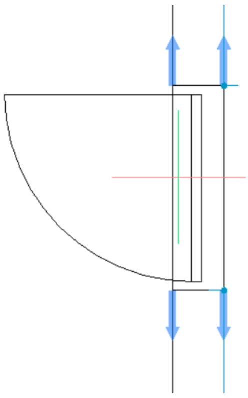
Note: At this stage, the block will snap the reference curves to similar curves in the target drawing. For this case, the block contains two times two parallel lines, with an offset distance of 300 mm. Thus, this block will be dynamically snapped to lines in the target drawing that are parallel with an offset distance of 300 mm.

6 Insert the block into the target drawing. After a few insertions, the result will be similar to this:



The insertion process is like this:

After launching the INSERT command, the **Insert Block** dialog opens. In the **Advanced settings** section, make sure the checkbox **Use reference curves for insert** is ticked, then press **OK**. The block will be loaded and attached to the mouse cursor. When moving in the neighborhood of two lines that match the reference curves in the block, the block will snap to those lines. Here is a capture of this snapping behavior:



The four blue arrows represent the reference curves in the block. The block will be aligned to matching lines. Moving the mouse along the lines allows you to position the block. Left clicking the mouse will insert the block in the drawing, aligned to the matching lines. In this specific use case, the lines in the target drawing will be trimmed between the connection points (i.e., the points at which the reference curves arrows are attached).

If the lines under the mouse cursor do not match the reference curves, the block can be inserted in the normal way, but will not align with any curves in the target drawing. For this case, if the distance between the parallel lines in the drawing is different than 300 mm (as it is in the block), the reference curves will not snap to those lines.

Note: The reference curves might not work well with the geometric and dimensional constraints. The behavior of the block when applying reference curves together with constraints is highly dependent on the complexity of the block geometry and the number and type of the constraints.

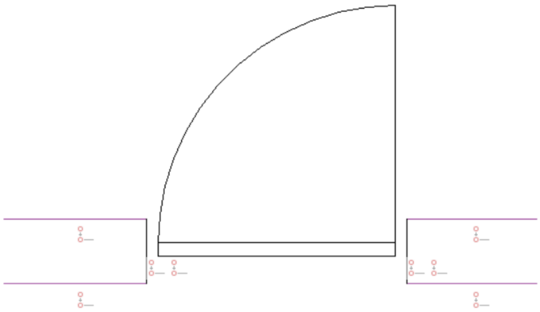
Note: The **Fuzzy Insert Guided** functionality parametrizes reference curves to enable you to insert the door block in walls (pairs of parallel lines) with different wall thicknesses (distances between the lines).

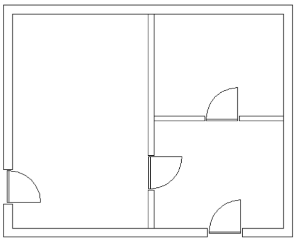
11.6.6 Enabling the Fuzzy InsertGuided functionality

To make the door block more flexible/fuzzy, the reference curves will be parametrized, and the surrounding geometry will be constrained to them. As a result, the parametric block will snap to walls with different wall thicknesses in the target drawing.

For more information on this functionality see the article **Fuzzy Insert Guided Workflow**.

To enable the **Fuzzy InsertGuided** functionality to the block, follow the next steps:

1. Open the block drawing.
2. Launch the REFERENCECURVES command from the ribbon or the Command line. For this case, the reference lines are already defined.
3. Select the **Parametrize** option. This will create the parameters between the reference curves automatically (see the panels **Parameters Manager** or **Mechanical Browser**).
4. Add constraints to attach some entities to the parametrized reference curves. See the picture below:
5. 

   For this case, four coincident constraints were added between the ends of the reference curves and the ends of the two vertical lines that represent the wall break.
6. Save the block drawing.
7. Insert the block into the target drawing. This time, due to the parameters attached to them, the reference curves will also snap to the inner walls which have a different thickness. See the picture below.
8. 

11.6.7 Adding the flip lines

Another useful feature that can be added to the parametric blocks is flipping the block geometry along certain lines. This task can be achieved using a flip parameter. The flip parameter will mirror the block across one axis. For this example, two flips will be added, mirroring horizontally and vertically.

To have access to the flip functionality, go to the **2D Parametric** tab of the ribbon. In the **Operations** panel, click the button **Flipline**. Another way to achieve a flip operation is by typing the FLIPLINE command in the Command line.

For more information on the flip lines, see the article **FLIPLINE command**.

To add one flip line, follow the next steps:

1 Open the block drawing.

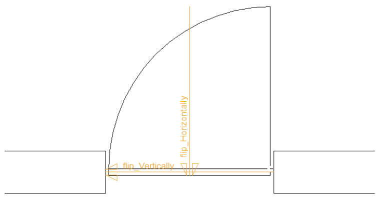
2 Launch the FLIPLINE command in the Command line and draw a vertical line on the center of the block. This command creates an orange line.

3 Individually select entities to be flipped or use the **flip Everything**option.

4 Give a name to the flip parameter. This parameter will be named **flip_Horizontally**.

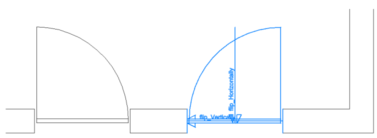
5 Using the same process, define a new flip line on the middle of the wall. This will be called **flip_Vertically**.

After defining the flip lines, the block drawing will look like this:



6 Save the block drawing.

7 Insert the block into the target drawing. After that, the block can be flipped by selecting it and using the flip parameters in the **Parameters** section of the **Properties** panel.



8 (Optional) Change the parametric block operation geometries color by setting the POPERATIONSCOLOR system variable as needed.

Note:

- The parametric block operation geometries are visible by default after these are created, but hidden when opening a drawing that contains parametric blocks. Use the PBLOCKOPERATIONSDISPLAY command to hide/show the parametric block operation geometries.

- The visualization of the parametric operation geometries when hovering the cursor over parametric block references is controlled by the PBLOCKREFERENCEOPERATIONSVISUALIZATION system variable (**Off** by default).

11.6.8 Adding the stretch parameter

The stretch parameter works similar to the STRETCH command. It can be useful when there are many entities to be stretched or moved in the same direction, over the same distance, without editing the block. In this case, the stretch parameter can replace several constraints, thus simplifying the drawing and the process of parametrizing the geometry.

Note: The stretch parameter might not work well in combination with the geometric and dimensional constraints. The behavior of the block when applying the stretch parameter together with constraints is highly dependent on the complexity of the geometry and the number and type of the constraints.

For more information on the stretch parameter, see the article **PARAMETRICSTRETCH command**.

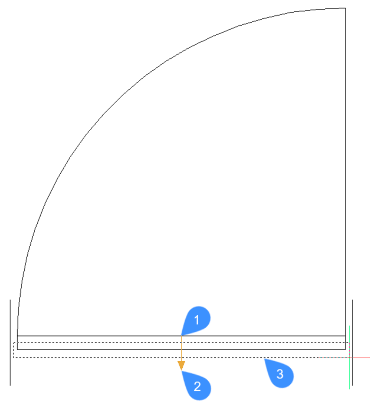
The following steps describe how to add a parametric stretch to a block:

1 Open the block drawing.

2 Launch the PARAMETRICSTRETCH command in the Command line. You can also go to the **2D Parametric** tab of the ribbon. In the **Operations** panel, click the **Stretch** button.

3 Create the stretch vector by selecting two points. For this example, the stretch vector will have its base point of displacement in the point marked with 1, and the second point of displacement, in the point marked with 2. See the picture below.

4



Note:

a The direction of this vector defines the direction of the stretch, and the length of this vector will be the initial value of the stretch parameter.

b If possible, align the vector with an important feature/dimension in the drawing. By doing this, the stretch parameter value will have a clear meaning.

c The stretch vector is drawn with a soft red color so it should be easily observed.

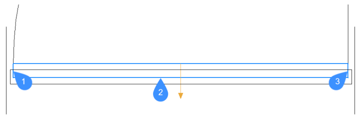
5 Draw the stretch frame.

The stretch frame can be a polygon or a rectangle. The stretch frame is used to specify which stretch points will be moved when the stretch is applied. Only the points of selected entities (see next step) which lie inside the stretch frame will be moved.

For this example, the stretch frame will be the rectangle marked with 3 in the picture above.

6 Edit the selection of the entities which will be affected when a stretch is applied, if needed. By default, the entities that have stretch points inside the stretch frame are already selected.

For this example, the default selection includes the lower line and the two side lines of the rectangle. See the highlighted lines marked with 1, 2 and 3 in the picture below.

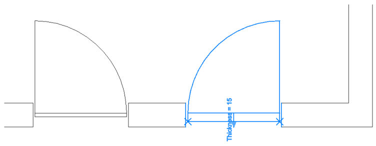


7 (Optional) Change the linked behavior of the stretch operation to have entities follow operation definition points when adjusted by other operations. The **Off** option corresponding to **not linked** behavior is set as default for newly created parametric operations.

8 Give a name to the stretch parameter. For this example, the parameter will be named **Thickness**.

9 Save the block drawing.

10 Insert the block in the target drawing and change the stretch parameter.

11 

The block on the left has its **Thickness** parameter set to the default value of 60 mm. The block on the right has its **Thickness** parameter changed to 150 mm. This might make sense for insulation purposes, in this use case.

12 The parametric block operation geometries are visible by default after these are created. You can use the PBLOCKOPERATIONSDISPLAY command to hide them. Use the same command to show them back.

13 ***Note:*** Points of parametric operations that are affected by a parametric stretch operation are marked by an "X".

11.6.9 Grip editing a parametric block

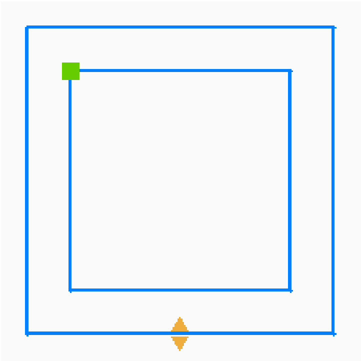
Grip points are displayed on parametric block references, for exposed parameters related to the following parametric operations:

- Flip
- Stretch
- Move
- Scale
- Rotate
- Offset

Note: Currently, no grip points are displayed for visibility parameters, design tables, and any non-exposed parameters.

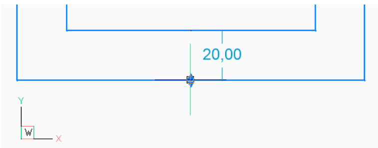
1 Select a parametric block in the drawing.

1 The grip points corresponding to the block reference's parametric operations are displayed.



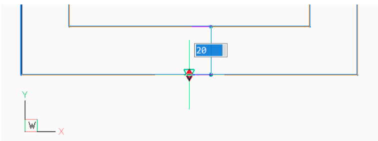
2 Hover the cursor over a parametric operation grip point.

3 The current value of the parametric operation is displayed.



4 Click the operation grip point.

5 The dynamic input field is activated.



Note: For parametric flip operations, click the grip point to mirror the block.

6 Drag the selected operation grip point.

7 The parametric operation's value is updated dynamically.

8 Click again to set the new value of the parameter or type a value in the dynamic input field.

9 The block reference's appearance is updated according to the new parametric operation's value.

11.7 Fuzzy insert guided workflow

11.7.1 Introduction

The reference curves are specific entities of a 2D parametric block. They are used to align the block with entities in the target drawing.

The reference curves can be parametrized. This means that not only the block can be aligned to the entities in the target drawing but also the size and appearance of the block can be adjusted to match the configuration of the entities in the target drawing.

The process of inserting a block with parametrized reference curves into a drawing is called **fuzzy insert guided**.

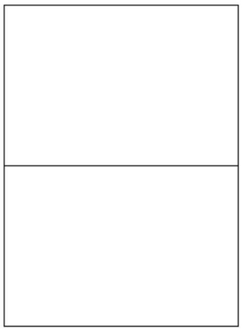
11.7.2 Workflow

The workflow of a **fuzzy insert guided** process is as follows:

1 Create a new drawing for the block definition.

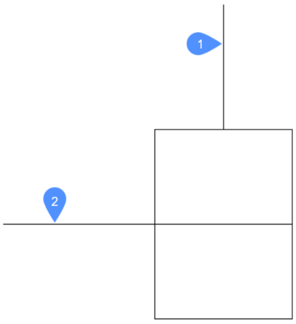
2 Create the entities of the block. In this case, a simple rectangular symbol will be used. See the picture below:

3



4 Create the entities to be used as reference curves, as in the picture below. In this case, the lines marked with 1 and 2 are added and they will be used in the next step.

5



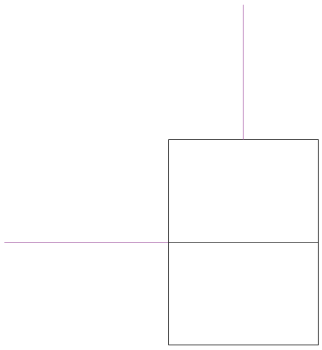
6 Define the reference curves. These curves are not part of the detail curves but will be used as a reference for the insertion (i.e., reference curves).

The lines marked with 1 and 2 (from the previous step) will be converted into reference curves using the REFERENCECURVES command. This command can be launched by typing REFERENCECURVES in the command line or by using the **Reference Curves** button in the **Operations** tab in the ribbon.

To define the reference curves, launch the REFERENCECURVES command and select lines 1 and 2.

After running the command, the drawing will look like this:

1

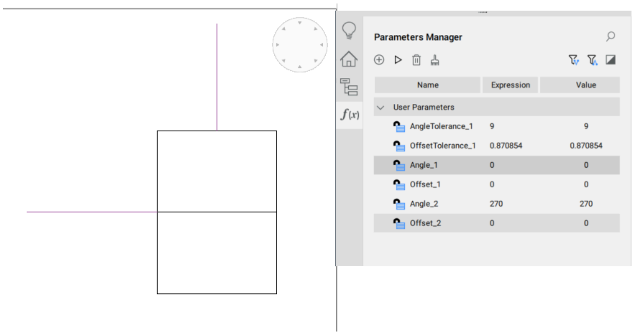


The two lines have been converted into reference curves. Their color has been changed to purple and they have been moved to a special layer, named **REFERENCE_CURVES**. This layer is created automatically by the command.

2 To parametrize the reference curves, run the REFERENCECURVES command and select the PARAMETRIZE option. This option will create parameters, controlling the reference curves configuration. These parameters can now (optionally) be used in the expressions of manually added constraints (for example, to fix the reference curves to the detail entities).

The parameters can be viewed in the **Parameters Manager** panel, as in the picture below.

1



2 Save the block.

3 Open the target drawing and insert the block.

The fuzzy insertion behavior can be achieved by using the INSERT command (with the **Use reference curves for insert** option On in the **Insert Block** dialog box), or by drag and dropping a block with parametrized reference curves from the library into the current model space.

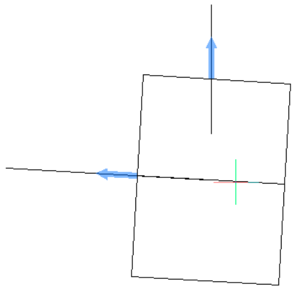
The reference curves of the block should be correctly tweaked and mapped to the target situation. Move the mouse to a position where you want to snap it and click to place the block.

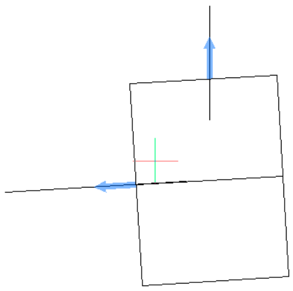
Here is a simple drawing used to see how fuzzy insert guided works:

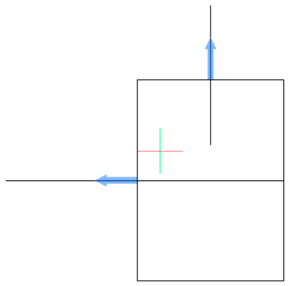
1

After running the INSERT command (with the **Use reference curves for insert** option On in the **Insert Block** dialog box) the block will be loaded to the mouse cursor.

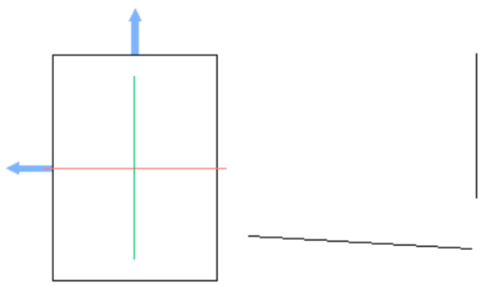
When the mouse cursor is placed near some entities in the drawing, which can match the reference curves, then the block will snap to those entities, like in the pictures below:







When the mouse cursor (with the block attached) is not around any entities or the entities do not match the reference curves, the snapping behavior does not occur, like in the picture below:



In this case, the block can be inserted into the drawing in a normal way.

11.8 Dynamic blocks

If you come from an AutoCAD® design environment, you may be familiar with Dynamic Blocks. You may also be familiar with 2D parametric constraints. They are two different functionalities that somewhat overlap.

11.8.1 Dynamic blocks vs. parametric blocks

AutoCAD® Dynamic Blocks and BricsCAD Parametric Blocks are blocks which automate repetitive tasks by allowing users to create one block instead of many similar, separate block definitions. These blocks contain not only pure geometry, but also some metadata, which affect the geometry (e.g., their size, visibility). The metadata controls the way the components of the block behave. Thus, the size and appearance of these blocks can be modified without editing the block definition. Constraints work in the same way for Parametric Blocks as they do with Dynamic Blocks.

If you insert AutoCAD® Dynamic Blocks into your BricsCAD drawing, in most cases you can edit those block **instances**. However, you cannot create or edit Dynamic Block **definitions** in BricsCAD. Instead of offering dynamic block functionality, BricsCAD focuses entirely on Parametric Blocks. BricsCAD allows you to create both 2D and 3DParametric Block **definitions** using the same set of tools and workflow. It also allows you to convert some Dynamic Block **definitions** to Parametric Block **definitions**.

Note: For more information about parametric blocks and constraints, see the articles **2D parametric blocks** and **2D constraints**.

Note: In BricsCAD, there are no parent-child dependencies between geometric elements. Any kind of dependencies can be created using the parameters and expressions.

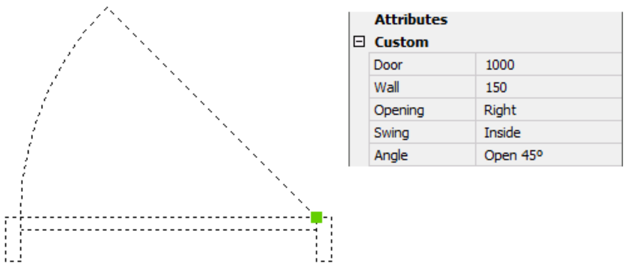
BricsCAD partly supports dynamic blocks. You can:

- Insert dynamic blocks from your existing dynamic blocks libraries.
- Edit the custom properties in the **Properties** panel.
- Use the custom properties of all parameter types and visibility states in fields.
- Grip edit dynamic blocks.

11.8.2 Edit dynamic blocks

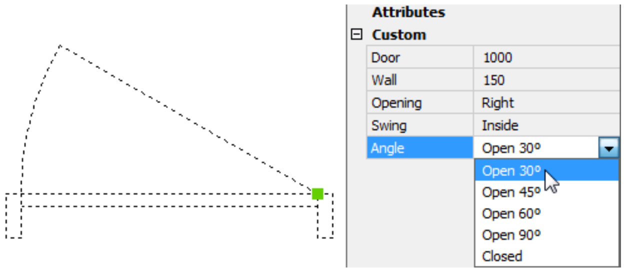
1 Select the dynamic block in the drawing.

1 The properties of the selected block are displayed in the **Properties** panel.



2 Under **Parameters**, edit the properties needed.

The dynamic block is updated automatically.



11.8.3 Grip edit dynamic blocks

1 Select the dynamic block in the drawing.

1 The following grip types are displayed:

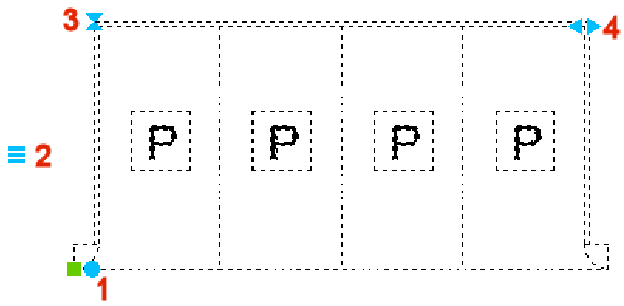
- Point
- XY
- Linear
- Polar
- Rotation
- Flip
- Visibility
- Alignment

2 Drag a grip to edit the block.

3 In the example below:

- Rotate the block (1): drag the grip to rotate the block dynamically or type a value in the dynamic entry field.
- Show/hide entities (2): click to select an option.
- Flip (3): click to mirror the block.
- Edit the length (4): select the grip to dynamically edit the length or type a value in the dynamic

entry field.



11.8.4 Convert dynamic blocks to parametric blocks

Some AutoCAD® Dynamic Blocks can be converted to BricsCAD Parametric Blocks by using the BLOCKCONVERT command. For more information, see the article **BLOCKCONVERT command**.

Note: The history of the dynamic blocks will be lost during the conversion to parametric blocks. A warning message about the possible different look of converted block references will be displayed.

Dynamic Block features supported for conversion

- visibility actions
- one-grip stretch action
- one-grip move action
- chained actions
- rotate action
- flip action
- array action
- alignment action parameter
- scale actions associated with a linear parameter
- scale actions associated with an XY parameter
- lookup action
- history-dependent actions
- block table feature
- linear parameters with two grip points, one of which is not used
- the base point parameter
- move and stretch actions associated with a polar parameter
- polar stretch actions associated with a polar parameter
- interactions between a polar stretch action and a move or a flip action
- actions associated with polar parameters with 2 grip points
- action parameters and dimension constraint parameters of list type
- dimension constraint parameters of increment type
- dynamic blocks with multiple actions associated with a single parameter

- actions with their start point on the X- or Y- axes

Parametric Block features used to reconstruct Dynamic Block features

- visibility states
- parametric move operation
- parametric stretch operation
- parametric rotate operation
- parametric scale operation
- linked behavior of parametric operations
- parametric flip operation
- reference curves
- design tables
- parameter formulas
- associative arrays

11.9 Defining attributes

The ATTDEF command defines attributes through a dialog box.

The -ATTDEF command defines attributes at the command prompt.

11.9.1 General procedure to define attributes

- Specify the characteristics of the attribute, including its name, prompt, and default value.
- Specify the insert coordinates.
- Define the text properties.
- (option) Set the attribute flags.

11.9.2 To define an attribute

1 Do one of the following:

- Click the **Attributes** > **Define Attributes...** tool button () on the **Tools** toolbar.
- Choose **Define Attributes...** in the **Tools** menu.
- Launch the ATTDEF command.

The **Define Attribute** dialog appears.

2 Define the Attribute:

- Type a name in the **Tag** field.

  The Tag identifies each occurrence of an attribute in the drawing. The name you type in the **Tag** field can be any combination of characters except spaces. Lowercase letters are automatically converted to uppercase.
- Type a prompt in the **Prompt** field.

  The attribute prompt appears when you insert a block containing the attribute. If a prompt is omitted, the Tag is used as the prompt instead.

- Do one of the following:
• Type a default value in the **Default** text field.

  For variable attributes, the default value is replaced by the actual value when you insert a block containing the attribute.

• Click the **Insert Field** button ( ) to assign a field value to the attribute (see 'Using sheet set and sheet properties in a title block').
• If the **Multiple lines** flag is checked, the **Default** text field is dimmed.
• Click the button next to the **Default text** field.

  The **Define Attribute** dialog temporarily closes to let you create an Mtext entity.

- Do one of the following:
• Specify the X, Y, and Z coordinates for the attribute insertion point.
• Click the Pick Point button ( ) to specify the insertion point in the drawing.

  You are prompted: Select insertion point.

  The **Define Attribute** dialog closes temporarily.

- (option) Set the **Invisible** attribute flag.

  Attributes with the **Invisible** flag set are neither displayed nor printed.

  If the ATTMODE (Attribute Display Mode) variable is set to **2**, all attributes are displayed, including **Hidden** attributes.

- (option) Set the **Constant** attribute flag.

  Attributes with the **Constant** flag set cannot be edited.

- (option) Set the **Verify** attribute flag.

  Attributes with the **Verify** flag set must be explicitly validated when a block containing such attributes is inserted.

- (option) Set the **Preset** attribute flag.

  When inserting a block, you are not prompted to define attributes of which the **Preset** flag is set. You can still edit the attributes afterwards.

- (option) Set the **Lock Position** attribute flag to lock the position of the attribute within the block reference.

  Unlocked attributes can be moved relative to the rest of the block using grip editing, and multiline attributes can be resized.

- Define the **Text** properties: **Text Style**, **Justification**, **Annotative**, **Text height** and **Rotation**.

3 Click the **OK** button to create the attribute in the drawing.

  The **Define Attribute** dialog closes.

11.9.3 Using sheet set properties in a title block

1 Create the sheet set.

2 Define the custom sheet set and sheet properties.

3 Open a sheet (layout) from the sheet set.

4 Create the title block source entities in the paper space layout:

- Create the borders, line work and fixed texts.
- Create attributes with fields that refer to sheet set, subset and sheet properties.

See the SHEETSET command to learn more about default and custom sheet set, subset and sheet properties.

Custom sheet set properties:

Sheet Set Custom Properties	
Architect	Bricsys & Partners
ArchitectAddress	Bellevue 5/201
ArchitectCity	9050 Gent
ArchitectPhone	09 244 01 91
Contractor	Robert Stone ltd.
ContractorAddress	123 Main Street
ContractorCity	New Town
Owner	Carpenter
OwnerAddress	1 Church Road
OwnerCity	New Town
OwnerTitle	Mr. & Mrs.
SiteAddress	48 Brick Lane
SiteCity	New Town
SiteReference	Section A no. 123

Sheet properties and sheet custom properties:

Sheet	
Number	08
Title	Elevation South
Description	South elevation drawing
Publish	☑ Include for Plot/Publish
Layout	LAYOUT1 (C:\BricsCAD Trainir
Revision Number	1
Revision Date	15/08/2013
Issue Purpose	Final Design
Category	Elevation
Sheet Custom Properties	
Date	12/08/2013
PaperSize	A2
Printer	pdf
Remark	none
Scale	1/50

5 Create a block from the title block source entities.

6 Copy the block to a drawing in one of your block folders.

7 Delete the title block source entities.

The title block can be inserted in the other sheets of the sheet set and in any other sheet set where the same custom sheet set and sheet properties exist. All fields that refer to the default or custom sheet set, subset or sheet properties are filled out automatically. If no value is available, a field displays hyphens (----).

11.10 Working with attributes

An attribute is a particular entity that you can save as part of a block definition. Attributes consist of text-based data. You can use attributes to track such things as part numbers and prices. Attributes have either fixed or variable values. When you insert a block containing attributes, the program adds the fixed values to the drawing along with the block, and you are prompted to supply any variable values.

After you insert blocks containing attributes, you can extract the attribute information to a separate file and then use that information in a spreadsheet or database to produce a parts list or bill of materials. You can also use attribute information to track the number of times a particular block is inserted into a drawing.

Attributes can be visible or hidden. Hidden attributes are neither displayed nor plotted, but the information is still stored in the drawing and written to a file when you extract it.

ATTDEF command - defines attributes through a dialog box.

-ATTDEF command - defines attributes through the Command line.

ATTDISP command - toggles the display of attribute text.

ATTEDIT command - edits attribute values and properties.

-ATTEXT command - extracts attributes from drawings to data files through the command bar.

ATTREDEF command - redefines a block and updates associated attributes.

ATTSYNC command - synchronizes attribute definitions in all block references of a specified block definition.

BATTMAN command - (short for "block attribute manager") edits all aspects of attributes in a block definition, and then optionally applies the changes to all blocks of the same name in the drawing.

EATTEDIT command - edits attribute values and most properties.

11.10.1 General procedure to work with attributes

1. Define the attributes.
2. Create a block.

You can attach attributes to a block. Include the attributes when the program prompts you to select the entities to be included in the block definition. If a block contains attributes, the program prompts you each time you insert the block, so you can specify different values for the attributes each time you insert it into a new drawing.

1. Insert the block in a drawing.
2. (option) Edit the attributes in a block.
3. Extract attribute information.

11.11 Editing attribute definitions

Attribute properties can be edited in the **Properties** panel.

The ATTEDIT command edits attribute values and properties.

The BATTMAN command (short for "block attribute manager") edits all aspects of attributes in a block definition, and then optionally applies the changes to all blocks of the same name in the drawing.

Note: Attributes in a block definition can be edited in the **Properties** panel when using the BEDIT commands to edit the block.

11.12 Editing block attributes

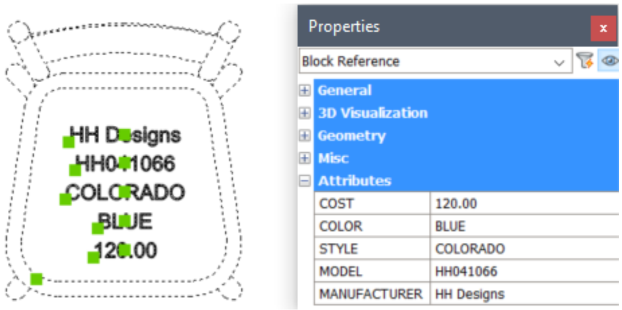
Block attributes can be edited in the **Properties** panel.

1 Select a block containing attributes.

1 The current value of the attributes is displayed in the **Properties** panel.

2 Select the attribute you want to edit.

3 The settings field of the selected attribute is active.



4 Type a new value, then press enter or select another attribute.

11.13 Inserting blocks with attributes

When you place a block which contains attributes, you will be prompted to define the attributes in the Command line or through a dialog box, depending on the value of the ATTDIA system variable:

- ON: attributes are displayed in a dialog box.
- OFF: you are prompted in the Command line to define the attributes.

11.13.1 To place a block with attributes

1 Do one of the following:

   - Click the **Insert Block** tool ( ) on the **Insert** toolbar or the **Insert / Blocks** ribbon panel.
   - Choose **Insert Block** in the **Insert** menu.
   - Launch the INSERT command.

   The **Insert Block** dialog box appears.

2 Select the block, then click the **OK** button.

   You are prompted: Insertion point for block or [Scale/X scale/Y scale/Z scale/Rotate/Multiple blocks]:

3 Specify a point.

4 All block entities are placed in the drawing, except the attributes.

5 Depending on the value of the ATTDIA system variable:

   - **You are prompted**: <Attribute Prompt 1> <Default Text 1>:

- The **Edit Attributes** dialog box appears.

6 Do one of the following:
   - In the Command line, for each attribute:
   - Press Enter or right click to accept the default text.
   - Type a new text in the Command line, then press Enter and define the next attribute.
   - In the **Edit Attributes** dialog box:
   - Select an attribute in the list.
   - The current value is displayed in the **Value** field.
   - Type a new value in the **Value** field.
   - Press **OK** when all attributes are properly defined.

11.14 Extracting attribute information

The ATTEXT command copies data from attributes to a text file via a dialog box.

The -ATTEXT command copies data from attributes to a text file via the Command line.

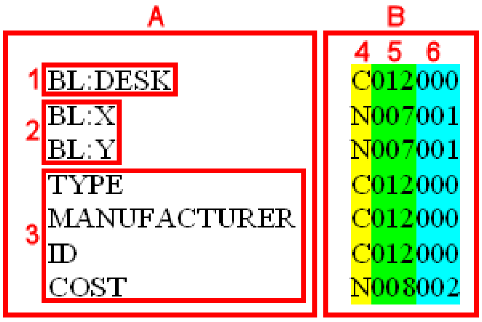
The text file can be imported in a database or a spreadsheet.

You can save the file in any of the following formats:

- **Comma Delimited Format** (CDF): Contains one line for each instance of a block, with individual attribute fields separated by commas. Character string fields are enclosed with single quotation marks. You must specify a template file when extracting to a CDF file.
- **Space Delimited Format** (SDF): Contains one line for each instance of a block. Each attribute field has a fixed length; there are no separators or character string delimiters. You must specify a template file when extracting to a SDF file.
- **Drawing Exchange Format** (DXF): Creates a subset of a standard DXF file (a *.dxx file) containing all the information about each block, including the insertion points, rotation angles, and attribute values. No template file is required.

Before extracting attributes to a CDF or SDF file, you must create a template file. The template file is an ASCII text file that specifies the attribute data fields to be written in the extract file. Each line of the template file specifies one attribute field. BricsCAD recognizes 15 different fields, which contain elements such as the block name, the X, Y, and Z coordinates of its insertion point, the layer on which it is inserted, etc. You can include any of these fields. The template file must include at least one attribute name.

Each line in the template file must start with the field name. Block name and insertion-point values must begin with **BL:**. The next nonblank character must be either a C (indicating a character string field) or an N (indicating a numeric field). This character is then followed by three digits indicating the width of the field (in characters). The final three digits indicate the number of decimal places (for numeric fields). In the case of character fields, the last three digits must be zeros (000). A typical template file is similar to the one shown here:



A. Field name

B. Field format

1. Block name (must be preceded by **BL:**)

2. Coordinate fields (must be preceded by **BL:**)

3. Attribute tags

4. Indicates the field data type: **C** for character fields, **N** for numeric fields

5. Field width (number of characters)

6. Number of decimal places for numeric fields, **000** for character fields

11.14.1 To extract attribute information

1 Do one of the following:

- Click the **Extract Attributes** tool button on the **Tools** toolbar.
- Choose **Extract Attributes** in the **Tools** menu.
- Launch the ATTEXT command.

The **Extract Attributes** dialog box appears.

2 Click the **Select** button.

The **Extract Attributes** dialog box closes temporarily.

3 Select the blocks, then press Enter or right click.

4 The **Extract Attributes** dialog box reopens.

5 Select the text file format: **DXF** (drawing exchange format), **CDF** (comma delimited format) **or SDF** (space delimited format).

6 Select the **Template file**.

7 Specify the **Output file** name.

8 Click the **Extract** button.

You are prompted: <number> records extracted.

11.15 Managing Xrefs

The EXPXREFS command opens the **External References** section of the **Drawing Explorer** dialog box.

The XREF command opens the **Attachments** panel.

The **Drawing Explorer - External References** dialog box allows you to:

- Attach a drawing as an external reference ( ).
- Detach an external reference ( ).
- Reload an external reference ( ).
- Unload an external reference ( ).
- Bind an external reference ( ).
- Insert an external reference ( ).
- Modify the attachment type: **Attach** or **Overlay**.
- Modify the **Saved Path** of an Xref.
- Modify the **Found Path** of an overlay.

11.15.1 To detach external references

1 Select the Xref(s) you want to detach.

1 Press and hold the Ctrl key to select multiple Xrefs or to unselect a selected Xref.

2 Click the **Detach Xref** tool button ( ) on the **Details** toolbar.

The selected Xref(s) are detached. You are not prompted to confirm the detachment.

11.15.2 To unload external references

Do one of the following:

- Click the **Loaded** column of a loaded Xref.
- Select the Xref(s), then either click the **Unload Xref** tool button ( ) on the **Details** toolbar or click the **Loaded** column of one of the selected Xrefs.

11.15.3 To reload external references

Do one of the following:

- Click the **Loaded** column of an unloaded Xref.
- Select the Xref(s), then either click the **Reload Xref** tool button ( ) on the **Details** toolbar or click the **Loaded** column of one of the selected Xrefs.

11.16 Attaching an external reference

The XATTACH command attaches a drawing as an external reference (Xref). The attached Xref appears as an insert in the current drawing. Just like blocks, attached Xrefs can be nested. The attached drawing represents the most recently saved version.

11.16.1 To attach an external reference

1 Do one of the following:

  - Click the **Attach Xref...** tool button ( ) on the **Insert** toolbar or the **Insert / References** ribbon panel.

- Choose **Attach Xref...** in the **Insert** menu.
- Click the **Attach Xref** tool button () on the **Details** toolbar on the **Drawing Explorer / Xrefs** dialog box.
- Choose the **Attach DWG** from the drop-down menu opened by pressing the tool button () on the **Attachments** panel.
- Launch the XATTACH command.

The **Select Reference File** dialog box appears.

2 Select the drawing you want to attach, then press the **Open** button.

The **Attach External Reference** dialog box appears.

3 The full path to the selected drawing is displayed in the **Path** field.

(option) Select the **Path Type**: **Full**, **Relative** or **No path**.

4 Choose the **External Reference Type**:

- **Attachment**: if the current drawing is attached as an Xref to another drawing, an attachment becomes a nested Xref.
- **Overlay**: an overlay is ignored when the drawing to which it is attached is then attached as an Xref to another drawing.

5 Choose whether to specify the **Insertion Point** on screen or in the **X**, **Y**, **Z** fields.

6 Choose whether to specify the **Scale** on screen or in the **X**, **Y**, **Z** fields.

Check the **Uniform Scale** option to specify the scale in the **X** field.

7 Choose whether to specify the **Rotation** angle on screen or in the **Angle** field.

8 Click the **OK** button to attach the Xref.

Depending on the options in steps 5, 6 and 7 you are prompted to specify the insertion point, scale and/or rotation angle.

Note:

- Parametric external references are also kept in the **Attachments** panel, in order to be reloaded, and in this case all its inserts including parametric ones are rebuilt.
- The XDWGFADECTL (Xref database fade control) system variable sets the fading level for external references.
- The origin point of an external reference is controlled by the INSBASE (insertion base point) system variable. Use the BASE command to define the INSBASE system variable in the external reference drawing.
- The **Unit** field under **Block Unit** is set by the INSUNITS (Insertion Units) system variable in the external reference. The **Factor** field expresses the relation between the value of the INSUNITS system variable in the Xref and the parent drawing.

11.17 Working with external references

You can link entire drawings to the current drawing as external references. Unlike inserting a drawing as a block, in which you add all the entities from the separate drawing into the current drawing, external references attach a pointer to the external file. The entities in the external reference appear in the current drawing, but the entities themselves are not added to the drawing. Thus, attaching an external reference does not significantly increase the size of the current drawing file.

External references provide additional capabilities not available when you insert a drawing as a block. When you insert a drawing as a block, the entities are stored in the drawing. Any changes you make to the original drawing are not reflected in the drawing in which you inserted it. When you attach an external reference, however, any changes you make to the original drawing file are reflected in the drawings that reference it. These changes appear automatically each time you open the drawing containing the external reference. If you know that the original drawing was modified, you can reload the external reference anytime you're working on the drawing.

External references are useful for assembling master drawings from component drawings. Use external references to coordinate your work with others in a group. External references help reduce drawing file size and ensure that you are always working with the most recent version of a drawing.

The XREF command (short for "external reference") attaches DWG files to the current drawing through the **Drawing Explorer**.

The -XREF command attaches DWG files to the current drawing through the Command line.

The XATTACH command (short for "external attach") attaches externally-referenced drawings through a dialog box.

The XCLIP command (short for "Xref clip") clips externally-referenced drawings, and adjusts front and rear clipping planes.

The XDWGFADECTL system variable controls the fading of external references. Values between 0 (no fading) and 90 are accepted. The fading factor affects the screen display only, not the printing.

The XNOTIFYTIME system variable defines the frequency to check for modified externally referenced resources: Xrefs, images and pdf attachments if XREFNOTIFY, IMAGENOTIFY and/or PDFNOTIFY is **On**.

The XREFNOTIFY system variable controls whether warnings for missing or modified Xrefs are issued.

11.18 Editing blocks and external references

The REFEDIT command (short for "reference editor") edits block references and externally-referenced drawings (Xrefs) through a dialog box. What REFEDIT does is referred to as **in-place reference editing**. The command either works on a portion of the reference or on the entire drawing or all entities in the block.

The -REFEDIT command edits block references and externally-referenced drawings through the Command line.

The REFSET command adds and removes entities from the reference being edited.

The REFCLOSE command closes the reference editor.

The BEDIT command edits the entities that make up blocks in a temporary drawing session called the **Block Editor**, which uses the local coordinate system of the block.

The -BEDIT command prompts for a block name at the Command line, and then opens the **Block Editor** environment.

The XFADECTL system variable sets the fading level for the entities which are not in the working in the current refedit session. Values between 0 (no fading) and 90 are accepted.

11.18.1 To edit Xrefs

1 Do one of the following:
   - Double click the Xref.
   - Click the **Refedit** tool button () on the **Refedit** toolbar.
   - Launch the REFEDIT command.

   You are prompted: Select reference:

   Click the Xref or block.

   The **Reference Edit** dialog box appears.

   The dialog box helps you to visually identify the reference to edit and controls how the reference is selected.
   - **Path**: Location of the selected reference; if the reference is a block, no path is displayed.
   - **Reference Name**: The name of the currently selected reference and any references, that are nested within the selected reference.
   - **Preview**: Displays the reference as it was last saved in the drawing. Please note that the preview image is not updated when changes are saved back to the reference.

2 (option) If multiple nested references are displayed in the **Reference name** tree, select a reference to modify.

   Click the expand/collapse button to show/hide nested references.

   You can edit only one reference a time.

3 Choose one of the following to **Select nested entities ...** :
   - **Automatically**: All the entities and nested blocks in the selected reference are included in the reference editing session.
   - **Prompt**: After you close the **Reference Edit** dialog, you are prompted to select the entities and nested blocks in the reference that you want to edit. Only these entities will be temporarily extracted and be made available for modification within the context of the current drawing.

   This temporary working set of entities can be edited and then saved back to update the reference being edited. To distinguish 'current drawing' from 'editing set', BricsCAD displays all of the entities belonging to the current drawing as faded. The fading level is controlled through the XFADECTL settings variable, which has a default value of **50**.

4 Check the **Settings**:
   - **Enable unique layer and symbol names**:
   - If selected, named objects in Xrefs are altered, adding a **$#$** prefix.
   - If cleared, the names of layers and other named objects remain the same as in the reference drawing.
   - **Display attribute definitions for editing**: Controls whether attribute definitions in blocks are extracted and displayed during reference editing.
   - **Lock entities not in working set**: Entities in the reference being edited that are not included in the temporary working set cannot be manipulated, nor can entities in the host drawing or other Xrefs. This behavior is similar to objects on locked layers.

5 Click the **OK** button to start editing the working set.

The **Reference Edit** dialog closes.

6 (option) Click the **Add to Refedit** tool button () to add entities to the working set.

The selected entities will be removed from the host drawing and added to the reference being edited when the working set is saved back.

7 (option) Click the **Remove from Refedit** tool button () to remove entities from the working set.

The selected entities will be removed from the reference being edited when the working set is saved back. These entities are added to the host drawing.

8 Do one of the following:
   - Click the **Refclose and Save** button () to save the changes back to the Xref source drawing or the block definition in the current drawing.
   - Click the **Refclose and discard** button () to stop the refedit session without saving. The source drawing or block definition is left unchanged.

11.19 Working with groups

The GROUP command creates and modifies named groups of entities, in a dialog box.

The -GROUP command creates and modifies named groups of entities, at the command line.

Working with groups is a technique to manipulate a set of several objects as though it were a single object. A group can also be considered as a 'Named Selection'. You can pick a single object and the entire set is selected as though it were a block, or you can edit individual objects without affecting the rest of the group and without breaking up the group. You can use nested groups if necessary and a single entity can be a member of different groups.

Note: Make sure bitcode 1 of the PICKSTYLE system variable is set (value = 1 or 3) to enable the selection of groups. The Ctrl + H keyboard shortcut toggles the value of the PICKSTYLE system variable: 0/1 if 0 or 1 and 2/3 if 2 or 3.

11.19.1 To create a group

1 Do one of the following:

- Click the **Group** tool button ( ) on the **Tools** toolbar or the **Home / Groups** ribbon panel.
- Choose **Group** in the **Tools** menu.
- Launch the GROUP command.

The **Entity Grouping** dialog box appears:

2 Do one of the following:

- Type a name in the **Name** field.
- Check the **Unnamed** option to create an unnamed group.

3 (option) Type a description in the **Description** field.

4 (option) Check / uncheck the **Selectable** option.

5 Click the **Select entities and create group** button.

The **Entity Grouping** dialog box temporarily closes to let you select entities.

6 Select the entities you want to include in the new group.

7 Right click to conclude the selection of entities.

8 The **Entity Grouping** dialog box reopens.

The newly created group is added.

9 Click the **OK** button to close the **Entity Grouping** dialog box.

11.19.2 To modify a group

1 Do one of the following:

- Click the **Group** tool button ( ) on the **Tools** toolbar or the **Home / Groups** ribbon panel.
- Choose **Group** in the **Tools** menu.
- Launch the GROUP command.

The **Entity Grouping** dialog box appears.

2 Select the group you want to modify.

3 (option) Double click in the **Name** field to edit the group name.

4 (option) Double click in the **Description** field to edit the group description.

5 (option) Click in the **Selectable** column to toggle the **Selectable** option on/off.

6 (option) Click the **Add entities to group** button to add new entities to the group.

The **Entity Grouping** dialog box temporarily closes to let you select the new group entities.

7 (option) Click the **Remove entities from group** button to remove entities from the group.

The **Entity Grouping** dialog box temporarily closes to let you select the entities.

8 Click the **OK** button to close the **Entity Grouping** dialog box.

11.19.3 To ungroup entities

1 Do one of the following:

- Click the **Group** tool button ( ) on the **Tools** toolbar or the **Home / Groups** ribbon panel.
- Choose **Group** in the **Tools** menu.
- Launch the GROUP command.

The **Entity Grouping** dialog box appears.

2 Select the group or groups you want to delete.

3 Click the **Un-group selected groups** button.

The selected groups are removed.

Note: When you ungroup entities, the group is deleted, but the entities remain in the drawing.

11.19.4 To change the order of entities

1 Do one of the following:

- Click the **Group** tool button ( ) on the **Tools** toolbar or the **Home / Groups** ribbon panel.
- Choose **Group** in the **Tools** menu.
- Launch the GROUP command.

The **Entity Grouping** dialog box appears.

2 Click the **Re-Order entities** button.

The **Order Group** dialog box appears:

3 (option) Click the **Highlight** button to see the current position of each entity.

The **Object Grouping** dialog box opens.

4 (option) Click the **Reverse Order** button to reverse the order of all entities in the group.

5 (option) To change the order of a single entity do the following:

- In the **Remove from position** field, type the current position of the entity you want to move.
- In the **Enter new position number for the entity** field, type the new position.

6 (option) Type the number of objects you want to reorder in the **Number of objects** field.

7 Click the **Re-Order** button.

A message box confirms that the group has been re-ordered.

11.20 Working with underlays

PDF files can be attached as an underlay to a drawing file. Much like raster image files and external references (Xrefs), a PDF underlay is not part of the drawing, but is linked to it. The path to the underlay is saved in the parent drawing. You can edit the path to make sure the correct PDF is found. By default the folder of the parent drawing is searched first. If the PDF underlay is not found there, the saved path folder is searched. If the PDF underlay is not found there either, BricsCAD reports a 'Missing or invalid reference' in the drawing.

The PDFADJUST command allows to adjust the fade, contrast and monochrome settings of a PDF underlay in the Command line.

The PDFATTACH and -PDFATTACH commands insert a PDF file as an underlay into the current drawing.

The PDFLAYERS command allows you to control the display of layers in a PDF underlay.

The PDFCLIP command crops the display of a PDF. You can choose between a polygonal or a rectangular boundary. Each instance of the same PDF underlay file can have a different boundary.

The PDFOSNAP system variable allows to snap to the geometry in the PDF underlay if the system variable is **On**.

The PDFNOTIFY system variable controls whether warnings for missing or modified PDF underlays are issued.

The XNOTIFYTIME system variable defines the frequency to check for modified externally referenced resources: Xrefs, images and PDF attachments if XREFNOTIFY, IMAGENOTIFY and/or PDFNOTIFY is **On**.

If the value of the FRAME system variable = 3, the PDFFRAME system variable controls the visibility of PDF frames. The options are:

- 0: Hide PDF frames.
- 1: Display and plot PDF frames.
- 2: Display but do not plot PDF frames.

Note:

- Underlays can be viewed only when **Visual style** is set to 2dWireframe.
- Hypertext links from PDF files are converted to text.

11.20.1 Controlling the PDF underlay caching

A multi-resolution persistent image cache is used to display attached PDF underlays, enabling (very) fast zoom and pan operations. The highest cached resolution is 5000 x 5000 pixels. Still, when zooming in very close, the display of the PDF underlay will become pixelated. So a hybrid modus can be used which switches to the real-time generation of a crisp PDF underlay display when zooming in very closely. The initial generation of the image cache may take a few seconds, but from then on processing gets (very) fast, and remains just as swift in subsequent sessions.

The PDFCACHE system variable controls how PDF caching is applied.

- 0 = don't use PDF cache, always use real-time generation of the PDF underlay image. (this mode was the default in previous BricsCAD versions).
- 1 = use PDF cache, switch to real-time generation when zooming in very close.
- 2 = always use PDF cache (default).

11.20.2 Attaching a PDF underlay

1 Do one of the following:

- Click the **Attach PDF** tool button ( ) on the **Insert** toolbar or the **Insert / PDFs** ribbon panel.
- Click the **New** tool button ( ) on the **Details** toolbar of the **Drawing Explorer - PDF Underlays**

dialog box.

- Type **PDFattach** in the Command line, then press Enter.

The **Select PDF Underlay File** dialog opens.

2 Select the file, then double click the file or click the **Open** button on the **Select PDF Underlay File** dialog.

The **Attach PDF Underlay** dialog box appears.

3 In case of a multi-page PDF, click the down arrow in the **Name** field, then select the page of the PDF you want to load.

4 Click the **Path** option button, then select either:
- **Full path**: The full path will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" is displayed at the insertion point of the PDF overlay.
- **Relative path**: The relative path, with respect to the folder of the parent drawing, will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" is displayed at the insertion point of the PDF overlay.
- **No path**: Only the folder of the parent drawing is searched when the drawing is loaded. If the overlay file is not found there, "Missing or invalid reference" is displayed at the insertion point of the PDF overlay.

5 Choose whether you want to specify the **Insertion Point**, **Scale** and the **Rotation** angle on-screen or not.

6 Click the **OK** button.

The **Attach PDF Underlay** dialog closes.

7 Depending on the insertion options chosen in the previous step, **you are prompted** to specify the **Insertion Point**, **Scale** and/or **Rotation** angle.

Note: You can also drag and drop underlays into the current drawing.

11.20.3 Inserting a PDF underlay

If you want to insert a second instance of a PDF that is already attached or another page of such underlay do the following:

1 Launch the EXPPDFS command.

1 The **Drawing Explorer - PDF Underlays** dialog box appears.

2 Select the PDF underlay.

3 Do one of the following:
- Right click and choose **Insert** in the context menu.
- Choose **insert** in the **Edit** menu on the **Drawing Explorer - PDF Underlays** dialog.

The **Drawing Explorer - PDF Underlays** dialog closes.

4 Continue with steps 3 through 7 of the previous procedure.

11.20.4 Controlling the display of a PDF underlay

1 Click the frame of the PDF underlay.

1 The frame of the selected PDF highlights.

The properties of the underlay are displayed in the **Properties** panel.

2 Under **Misc**, click **Show underlay** in the **Properties** bar, then select **Yes** or **No**.

11.20.5 Setting the layer display in a PDF underlay

1 Launch the PDFLAYERS command.

1 **You are prompted**: Select PDF underlay.

2 Click the frame of the PDF underlay.

3 The frame of the selected PDF highlights.

The **Underlay Layers** dialog box appears.

4 The icon in the **On** column indicates the current state of the layers; **On** ( ) or **Off** ( ).

5 Click the icon in the **On** column to toggle the display of a layer.

6 (option) Click the **Search for layer** field, then type a layer name to search for a layer.

Use wildcard characters (? or *) to limit the number of layers in the list.

7 (option) Click the **Reference name** button to select another PDF underlay.

8 Click the **OK** button to stop.

11.20.6 Clipping a PDF underlay

1 Launch the PDFCLIP command.

1 **You are prompted**: Select PDF underlay:

2 Click the frame of the PDF underlay.

3 The frame of the selected PDF highlights.

You are prompted: Enter PDF clipping option [ON/OFF/Delete/New] <New>:

4 Press Enter to accept the **New** default command option.

You are prompted: Enter PDF clipping type [Polygonal/Rectangular] <Rectangular>:

5 Do one of the following:

- Press Enter to accept the **Rectangular** default command option to define a rectangular clipping boundary.
- Type P, then press Enter to define a polygonal clipping boundary.

6 Define the clipping boundary.

7 The defining points must lie inside the frame of the PDF underlay. If you click outside the PDF underlay the point is placed on the PDF underlay frame.

11.20.7 Toggling the display of the clipped part of a PDF underlay

1 Click the frame of the PDF underlay.

1 The frame of the selected PDF highlights.

The properties of the underlay are displayed in the **Properties** panel.

2 Under **Misc**, click **Show clipped** in the **Properties** bar, then select **Yes** or **No**.

11.20.8 Deleting a clipping boundary

1 Launch the PDFCLIP command.

1 **You are prompted**: Select PDF underlay.

2 Click the frame of the PDF underlay.

3 The frame of the selected PDF highlights.

You are prompted: Enter PDF clipping option [ON/OFF/Delete/New] <New>:

4 Type D in the Command line, then press Enter.

5 The clipping boundary is deleted and the complete PDF underlay appears.

11.20.9 Adjusting the display settings of a PDF underlay

The PDFADJUST command allows to adjust the **Fade**, **Contrast** and **Monochrome** settings of a PDF underlay in the Command line.

Select the PDF underlay, then adjust the display settings under **Underlay Adjust** in the **Properties** panel.

11.21 Components and component inserts

A component is a named group of entities, that can be inserted in a drawing with the BmInsert command. Components can be parametric: e.g. windows and doors, in BricsCAD BIM and standard mechanical components in BricsCAD Mechanical.

Any .dwg file can be used as a component. The same component can be inserted once or several times into one or more other components. When the component is modified, all inserts of the component will reflect these changes automatically. Components cannot self-reference.

A component insert is a named entity. The default name of a component insert is composed of the name of the component and the serial number of the insert. Internally a component insert is represented either as a reference to an anonymous block or as an external reference.

Properties of inserted components, including their parameters, can be edited in the **Mechanical Browser** panel and **Properties** panel.

11.21.1 External and local components

Components can be either local or external. A local component is stored in the .dwg file with the assembly as an anonymous block. An external component is defined in a separate .dwg file and represented by an Xref or as a local copy stored as an anonymous block. When the corresponding .dwg file is modified all inserts of external components can be updated using the BMUPDATE command or the **Mechanical**

Browser context menu. If you use external components in your assembly, always remember to transfer all related files when you want to share your model.

Use the ETRANSMIT command to create a package of a drawing file and all its dependencies, such as external references, images, font files, plot configuration files, plot style tables and font map files.

You can always convert a local component to an external one and vice versa.

To edit a local component, it first needs to be converted to an external component using the BMEXTERNALIZE command. Alternatively, you can use the BMOPENCOPY command to open a copy of a component, edit and save it. Then use the BMREPLACE command to replace the original with the copy.

You can choose the default insertion type for your component: **Local** or **External**. To change the type, select the root node in the **Mechanical Browser** and select **Local** or **External** from **Insert as properties**.

Depending on the value of BMAUTOUPDATE, inserts of external components are updated automatically on opening the assembly document or manually with the BMUPDATE command. You can change this value in the **Settings** dialog box. The options are:

- Update only when the BMUPDATE command is used.
- Update automatically upon opening the file.

11.21.2 Converting components

The BMEXTERNALIZE command converts local components to external components.

The BMLOCALIZE command converts external components to local components.

Do one of the following:

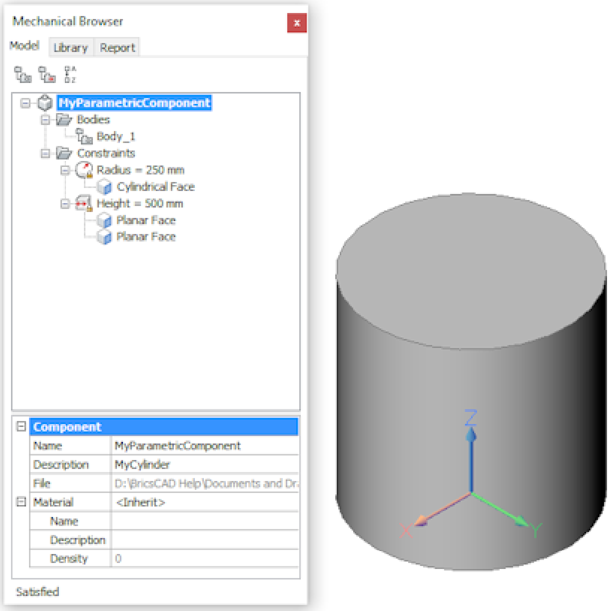
- Click **Switch Component to External** ( ) or **Switch Component to Local** on the **Assembly** ribbon tab. The tools are also available from the **Assembly** toolbar and the **Assembly** menu.
- Right-click on the component insert in the **Mechanical Browser** and select **Switch to External** or **Switch to Local** in the context menu.
- Hover your mouse cursor over a component insert in a drawing window and select **Switch to External** ( ) or **Switch to Local** in the **Assembly Modeling** command group in the Quad.

When you convert an external component to a local one, the .dwg file with the component definition is not deleted, but future changes will not affect your assembly.

You can always distinguish between inserts of local components ( ) and inserts of external components ( ) by their icons in the **Mechanical Browser**.

11.21.3 Parametric Components

You can make parametric changes of inserted components at the assembly level. A component is considered parametric if it contains at least one 2D or 3D dimensional constraint or an associative array with an expression.



Note:

- BricsCAD automatically maintains associativity between the definition of a parametric component and its inserts. So, you can open the parametric component for editing, modify its geometry and save it. To see the result, execute the BmUpdate command to update the mechanical structure of your assembly.
- If you need to go back to the default value of a component's parameter, just edit the **Expression** field and delete its content.

You can control if a particular component parameter will be available for editing in a drawing that contains the inserts of these components (at the assembly level). To do that, select a parameter in the **Mechanical Browser** and change the value of the **Exposed** field in the properties grid of the parameter.

The options are:

- **Off**: Hides the parameters at the assembly.
- **On**: Makes the parameter visible at the assembly level.
- **Reset**: Hides all parameters with values controlled by expressions and exposes all other parameters.

You can associate units with a parameter. If you assign a value, directly or via an expression, to a component parameter at the assembly level, it automatically converts from drawing units of the assembly drawing to the units of the parameter. Values assigned to unitless parameters are not converted.

By default, all parameters are unitless except parameters of dimensional constraints, and parameters which are used as expressions for dimensional constraints. BricsCAD automatically assigns drawing units to such parameters.

11.21.4 To edit the parameters of a component insert

When a parametric component is inserted in a model, you can edit the parameters either in the **Mechanical Browser** panel or in the **Properties** panel.

The component insert immediately updates according to the new value of a parameter. Please note that this does not modify the definition of the parametric component, which is a separate drawing file. As a result, multiple inserts of the same parametric component can exist in the same model, with different values for each of their parameters:

To edit a parameter in the Mechanical Browser

1 Expand the **Parameters** node of the component.

2 Select the parameter.

3 In the **Parameter** section, type a new value in the **Expression** field.

To edit a parameter in the Properties bar

1 Select the component in the drawing.

2 Expand the **Parameters** settings group.

3 Type a new value for one of the available parameters.

Note:

- In the **Properties** panel, the current value is displayed. If this value is controlled through an expression, the expression will be overwritten by the newly entered value.
- If multiple parametric components are selected, the shared parameters can be edited simultaneously. The settings fields of parameters that currently have different values reads: *varies*.

11.21.5 To replace components

The BMREPLACE command replaces component inserts. Only top-level inserts can be replaced.

Note: Parameter values that are shared between the replacement component and the replaced component are copied automatically.

11.21.6 Components, blocks or external references?

Components are based on blocks and external references (Xrefs) but the behavior of components is different. An external component is linked to a file, much like an Xref, but an external component can be represented by either an Xref (BMINSERT acts like XATTACH) or by a local anonymous block (BMINSERT acts like INSERT).

Components can be either parts or assemblies:

- Part: composed of entities only; no inserts of subcomponents.
- Assembly: composed of entities and subcomponent inserts. Even when the subcomponents are removed, the assembly status is kept.

When an external component is a part and its parameters are not modified, the component is inserted as an Xref. This is listed in the **External References** section of the **Drawing Explorer** dialog. Xref layers have a 'file_name' prefix.

When an external component is an assembly, it is inserted as a local anonymous block. All of its layers are merged with the layers of the main document and no prefix is added to the layer name.

11.21.7 Converting blocks and external references

The BMMECH command converts the current drawing into a mechanical component. If the drawing already is a mechanical component BMMECH does nothing.

Block references are converted to inserts of local components and external references are converted to inserts of external components.

The BMUNMECH command converts the current mechanical component into a plain drawing. Mechanical components are converted back to blocks and external references. The command applies to drawings that are a mechanical component only.

11.21.8 Component-based features

A feature is a physical constituent of a part with engineering significance. Examples of features are holes, ribs, slots, pockets, forms. If applied to a 3D solid, a feature removes and/or adds geometry. To create your own features, use the BricsCAD component extension technology.

An extended component has BC_SUBTRACT and/or BC_UNITE layers. When you use the BMINSERT command to insert such a component on the face of a 3D solid, the target solid is modified as follows:

- All 3D solids on the BC_SUBTRACT layer of the component are subtracted from the target 3D solid.
- All 3D solids on the BC_UNITE layer of the component are united with the target 3D solid.

All faces created after Boolean operations form a component-based feature. They are updated when you move the component insert or when you modify any parameters. When the insert is deleted, the target solid is healed. Adjust the target 3D solids with the BMLINK command.

Some examples of parametric component-based features are included with BricsCAD. You can use them as samples to create your own features.

11.21.9 Assign physical materials to components

You can assign a physical material to a mechanical component. If a mechanical component does not have an assigned material, it will inherit the material from its parental component. If the parental component does not have an assigned material, the material of the nearest component, will be used. As a result, the same material can be assigned to all components that have a common parent.

1 Select the root component in the **Mechanical Browser**.

2 Select the **Material** property and press the **Browse** button ().

3 In the **Physical Materials** dialog box, select a material from either the **In Project** or the **In Library** list.

  Materials that are chosen from the **In Library** are copied to the **In Project** database automatically.

4 Click the **OK** button to assign the selected material to the component.

To remove the material from the component:

1 Select the root component in the **Mechanical Browser**.

2 Select the **Material** property and press the **Clear** button (X).

Inserting a component with a physical material:

When you insert a mechanical component in a model, all its materials are copied to the material library of the model. If a material with the same name already exists in the target model, this material will be used instead. As a result, the properties of the material in the target model are used. This replaces the properties of the material from the inserted component.

Note: Materials that are copied to the document material library will not be erased in **Undo** operations.

11.21.10 Assign custom properties to components

You can assign custom properties to components and component inserts. Similarly to BIM custom properties, component custom properties are grouped into property sets, that correspond to groups in mechanical browser or properties panel. A property set contains property definitions, which define the name and the type of this custom property. You may also add predefined values for a specific custom property.

By default, property sets are applicable for all mechanical components in the document. For example, you may define a property set with "Part number" custom property, and this property will appear for every part in the model. If a component with this property is inserted into another document, the property will also appear for all components in that document. To avoid that, you may use per-instance property sets. Those property sets will be applied only to specified components. To enable or disable a per-instance property set for selected components, use the context menu of the mechanical browser.

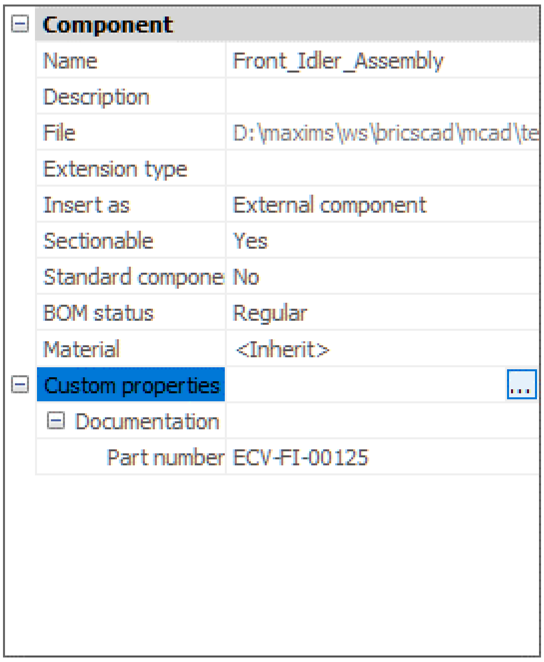
Component inserts inherit custom properties of their corresponding component. However, you are free to change any instance custom property as you wish; those changes will be applied only to selected component instances and will not affect other instances or the component itself. To change component properties, you will need to open the corresponding component and change its properties by selecting the root node in the mechanical browser.

To change component properties:

1 Select the root component in the Mechanical Browser.

2 (optional) If necessary, configure the available custom properties:

  - Call BMPROPERTIES command from the ribbon or command line
  - or select the **Custom properties** property and press the Browse button ().



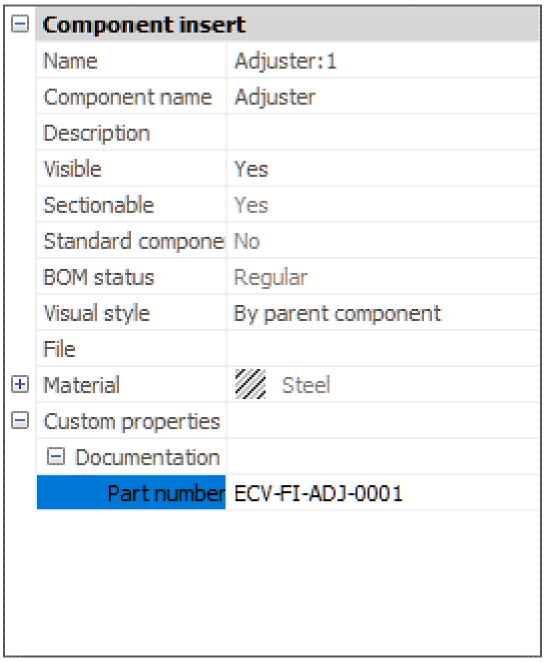
3 (optional) If necessary, enable per-instance property sets for this component:

- Right-click on the root node in the browser, then select necessary property sets in Custom properties submenu.

4 Select a required property and type or select a new value.

To change top-level instance properties:

1 Select the component instance in the Mechanical Browser.



2 (optional) If necessary, configure the available custom properties:
   - Call BMPROPERTIES command from the ribbon or command line
   - or select the **Custom properties** property and press the Browse button ().
3 (optional) If necessary, enable per-instance property sets for this component:
   - Right-click on the corresponding node in the browser, then select necessary property sets in Custom properties submenu.
4 Select a required property and type or select a new value.

12. Managing images and PDF files

12.1 Managing images

The EXPIMAGES command displays the **Drawing Explorer – Images** dialog box.

The IMAGE command opens the **Attachments** panel.

The IMAGEADJUST command adjusts the properties of images through the **Properties** panel.

The IMAGEATTACH command attaches raster images to the drawing.

The IMAGECLIP command clips images.

The IMAGEFRAME command toggles the display of the frame around images; sets the value of the IMAGEFRAME system variable.

The IMAGEQUALITY command toggles the display quality of images between draft and high.

The IMAGENOTIFY system variable controls whether warnings for missing or modified image files are issued.

The XREFNOTIFYTIME system variable defines the frequency to check for modified externally referenced resources: Xrefs, images and pdf attachments if XREFNOTIFY, IMAGENOTIFY and/or PDFNOTIFY is ON.

You can manage images in the **Drawing Explorer - Images** dialog box and in the **Attachments** panel.

Here you can:

- Attach an image ( ) or ( ).
- Load / Unload inserted images.
- Detach images ( ).
- Edit the path to the image.
- See a preview of the attached images (**Drawing Explorer** dialog box only).

To open the **Drawing Explorer - Images** dialog box, do one of the following:

- Choose **Drawing Explorer** > **Images** in the **Tools** menu.
- Select **Images** in the **Open Drawings** sub-window of the **Drawing Explorer** dialog box.
- Launch the EXPIMAGES command.

To open the **Attachments** panel, do one of the following:

- Click the **Image Management** tool button ( ) in the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGE command.

12.1.1 Attaching an image

1 Do one of the following:

  - Click the **New** tool button ( ) in the **Drawing Explorer - Images** dialog box.
  - Click the **Attach Raster Image** tool button ( ) in the **Insert** toolbar or the **Insert / Images** ribbon panel.

- Launch the IMAGEATTACH command.

The **Select Image File** dialog box appears.

2 Select the image file, then double-click or click the **Open** button.

The **Attach Raster Image** dialog box appears.

3 (option) Check the **Use Geocoding Information** option, then click the **Browse** button to open an appropriate world file to be used with the selected raster image.

If the **Use Geocoding Information** option is checked, the **Insertion Point**, **Scale** and **Rotation** settings fields are not available.

4 Specify an **insertion point** or select the **Specify on-screen** check box.

5 Specify a **scale** or select the **Specify on-screen** check box.

6 Specify a **rotation** angle or select the **Specify on-screen** check box.

7 Click the **OK** button.

8 If none of the **Specify on-screen** options are selected, the image is inserted at the desired insertion point, scale and rotation angle;

else you are prompted to specify the insertion point, scale and/or rotation angle on-screen.

12.1.2 Loading and unloading images

When an image is attached and then inserted in the drawing, you can unload the image to temporarily hide it. Unloaded images are still inserted in the drawing, but they are no longer displayed. If the **Imageframe** property is ON, the frame around the image is still displayed.

To load / unload a single image

Click the **Load** column of the image in the **Attachments** panel or the **Drawing Explorer – Images** dialog box.

To load / unload multiple images

1 Hold down the Ctrl key, then select the images in the **Attachments** panel or the **Drawing Explorer – Images** dialog box. Click a selected image to de-select.

2 Click the **Load** column of a selected image.

12.1.3 Detaching images

Since an image is first attached, then inserted in the drawing, you can delete an inserted image, in the drawing without detaching it.

To detach an image:

1 Select the image(s) in the **Attachments** panel or the **Drawing Explorer – Images** dialog box.

Hold down the Ctrl key to select multiple images; click a selected image to de-select.

2 Do one of the following:

- Click the Delete or Detach (✕) tool button.
- Right-click, then choose **Delete** or **Detach** in the context menu.

12.2 Image utilities

The IMAGEADJUST command adjusts the properties of images through the Properties panel.

The IMAGECLIP command clips images.

The IMAGEFRAME command toggles the display of the frame around images; sets the value of the IMAGEFRAME system variable.

The TRANSPARENCY command toggles the transparency of images that have a transparent alpha channel. The alpha channel is a special channel that handles transparency. When an image has an alpha channel on it, you can adjust the image's opacity levels and make bits translucent or totally see-through.

Note: When the image frames do not appear (IMAGEFRAME = 0), you must use a window selection method to select an image.

12.2.1 Clipping an image

1 Do one of the following:

- Click the **Clip Image** tool button ( ) on the **Image** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGECLIP command.

You are prompted: Select image:

2 Select the image.

3 **You are prompted**: Enter image clipping option [ON/OFF/New boundary] <New boundary>:

4 Press Enter to accept the **New Boundary** default option.

5 **You are prompted**: [Select polyline/Polygonal/Rectangular] <Rectangular>:

Press Enter to accept the **Rectangular** default option.

6 **You are prompted**: Specify first corner.

7 Click a point to define the first corner of the clipping rectangle.

8 The clipping rectangle is dynamically displayed.

You are prompted: Specify opposite corner:

9 Click a point to define the clipping rectangle.

10 The image is clipped.

12.2.2 Modifying the Image Frame setting

Using the Image Frame tool (IMAGEFRAME command):

1 Do one of the following:

- Click the **Image Frame** tool ( ) on the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGEFRAME command.

You are prompted: Enter image frame setting <current value>:

2 Do one of the following:

- Type 0 to hide image frames.
- Type 1 to display and plot image frames.

- Type 2 to display but not print image frames.

Editing the IMAGEFRAME system variable:

1 Open the **Settings** dialog box.

2 Go to **Drawing / Underlays**.

3 Set the FRAME system variable = 3: Use individual settings.

4 Select the IMAGEFRAME system variable.

5 The options are:

- 0: Hide image frames.
- 1: Display and plot image frames.
- 2: Display but do not plot image frames.

12.2.3 Setting the image transparency

1 Do one of the following:

- Click the **Image Transparency** tool button () on the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the TRANSPARENCY command.

You are prompted: Select images.

2 Select the images you want to set the **Transparency** property of.

You are prompted: Enter transparency mode [ON/OFF]<current mode>:

3 Do one of the following:

- Press Enter to accept the current mode.
- Choose **On** to switch the **Transparency** property of the selected images on.
- Choose **Off** to switch the **Transparency** property of the selected images off.

Transparency of the camera image: OFF (left) and ON (right)

12.2.4 Setting the display properties of an image

1 Select an image.

2 The properties of the image appear in the **Properties** panel.

3 Go to the **Misc** properties category.

4 The following properties can be set **Yes** or **No**:

- **Show Image**: shows or hides the image.
- **Show clipped**: sets the clipping boundary on/off.
- **Clip inverted**: inverts the clipping boundary.
- **Background transparency**: sets the transparency on/off.

12.3 Managing PDF files

12.3.1 About

You can manage PDFs in the **Drawing Explorer - PDF Underlays** dialog box and in the **Attachments** panel. Here you can:

- Attach PDF files ().
- Detach PDFs ().
- Load/Unload inserted PDFs.
- Edit the path to the PDF.
- See a preview of the attached PDFs. A preview is only available in **Drawing Explorer** dialog box.

Note:

- To open the **Drawing Explorer - PDF Underlays** dialog box, do one of the following:
  - Run the EXPPDFS command.
  - Click the **PDF Underlays** tool button () in the **Insert / PDFs** tab of the **Ribbon** panel.
- To open the **Attachments** panel, run the PDF command.

12.3.2 Attaching a PDF file

1 Do one of the following:

- Click the **New** tool button () in the **Drawing Explorer - PDF Underlays** dialog box.
- Click the **Attach PDF** tool button () in the **Insert** toolbar, in the **Attachments** panel or in the **Insert / PDFs** tab of the **Ribbon** panel.
- Launch the PDFATTACH command.

The **Select PDF Underlay File** dialog box displays.

2 Select the PDF file, then double-click or click the **Open** button.

The **Attach PDF Underlay** dialog box appears.

3 Specify an **Insertion Point** or select the **Specify On-screen** check box.

4 Specify a **Scale** or select the **Specify On-screen** check box.

5 Specify a **Rotation** angle or select the **Specify On-screen** check box.

6 Click the **OK** button.

7 If none of the **Specify On-screen** options are selected, the PDF is inserted at the desired insertion

point, scale and rotation angle, else you are prompted to specify the insertion point, scale and/or rotation angle on-screen.

Note:

- You can adjust the face, contrast and monochrome level of one or more PDF underlays attached to the current drawing, using the PDFADJUST command.
- You can clip PDF underlays with a boundary, using the PDFCLIP command.
- You can specify the visibility of PDF underlay frames, using the PDFFRAME system variable.

12.3.3 Loading and unloading PDFs

When a PDF is attached and then inserted in the drawing, you can unload the PDF to temporarily hide it. Unloaded PDFs are still inserted in the drawing, but they are no longer displayed.

To load / unload a single PDF

In the **Attachments** panel or the **Drawing Explorer - PDF Underlays** dialog box, click the checkbox in the **Load** column of the desired PDF.

To load / unload multiple PDFs

1 Hold down the Ctrl key, then select the desired PDFs in the **Attachments** panel or the **Drawing Explorer - PDF Underlays** dialog box. Click a selected PDF to deselect, while still holding down the Ctrl key.

2 Click a checkbox in the **Load** column of the selected PDFs to load/unload all selected PDFs.

Note: When the PDF is loaded, a green check mark appears.

12.3.4 Detaching PDFs

To detach a PDF:

1 Select the PDF(s) in the **Attachments** panel or the **Drawing Explorer - PDF Underlays** dialog box.

Hold down the **Ctrl** key to select multiple PDFs. Click a selected PDF to deselect.

2 Do one of the following:

- Click the Delete or Detach (×) tool button.
- Right-click, then choose **Delete** or **Detach** in the context menu.

12.3.5 Importing a PDF

You can import PDF files and convert their contents to simple CAD entities, using the PDFIMPORT command. This command also converts a PDF underlay already attached to the drawing to CAD entities.

Note: The accuracy of the resulting BricsCAD content depends on the quality of the original PDF file.

12.3.6 Exporting a PDF

You can generate PDF files, using the EXPORTPDF command.

To export a layout tab to a PDF file, first select the desired layout and then run the EXPORTPDF command.

Note: You can export drawings to 3D PDF, using the EXPORT command. You need:

- Communicator for BricsCAD®.

- A minimum BricsCAD Pro license level.

13. Annotate

13.1 Dimensions

13.1.1 Dimensioning tools overview

Dimensioning tools can be found in:

- The **Dimensions** toolbar.
- The **Dimensions** menu.
- The **Annotate / Dimensions** ribbon panel.
- The **Draw** tab of the **No Selection** Quad.

Icon	Tool name	Command	Description
	Linear Dimension	DIMLINEAR	Creates horizontal or vertical linear dimensions.
	Aligned Dimension	DIMALIGNED	Creates linear dimensions which are aligned with the dimension points.
	Jog Line	DIMJOGLINE	Adds or removes jog line to linear or aligned dimensions.
	Arc	DIMARC	Creates arc length dimensions.
	Radius	DIMRADIUS	Creates radial dimensions in circles and circular arcs.
	Diameter	DIMDIAMETER	Creates diametrical dimensions in circles and circular arcs.
	Jogged Dimension	DIMJOGGED	Creates jogged dimension for arcs or circles.
	Disassociate Dimensions	DIMDISASSOCIATE	Removes the associativity from selected dimension entities.

Icon	Tool name	Command	Description
	Reassociate Dimensions	DIMREASSOCIATE	Reassociates or associates dimensions to entities or points on entities.
	Angular	DIMANGULAR	Creates angular dimensions.
	Baseline	DIMBASELINE	Creates stacked linear dimensions.
	Continue	DIMCONTINUE	Creates a chain of linear dimensions, starting from an existing linear dimension entity.
	Ordinate	DIMORDINATE	Creates ordinate dimensions.
	Rotated	DIMLINEAR	Creates rotated linear dimensions.
	Make Oblique	DIMEDIT + O	Makes the extension lines of the selected dimension(s) oblique.
	Multileader	MLEADER	Creates Mleaders (multileaders) using a multileader style.
	Edit Multileader	MLEADEREDIT	Adds leader lines to and removes leader lines from a multileader entity.
	Multileader add leaders	AIMLEADEREDITADD	Adds leader lines to a multileader.
	Multileader remove leaders	AIMLEADEREDITREMOVE	Removes leader lines from a multileader.
	Tolerance	TOLERANCE	Defines the layout of geometric tolerances.

Icon	Tool name	Command	Description
	Dimcenter	DIMCENTER	Draws center lines in circles and circular arcs.
	Edit Dimension Text	DIMEDIT + E	Defines the dimension text of the selected dimension(s).
	Rotate Dimension Text	DIMEDIT + R	Rotates the dimension text of the selected dimension(s).
	Reposition Dimension Text	DIMTEDIT	Lets you reposition the dimension text.
	Restore Text Position	DIMEDIT	Undoes the rotation or repositioning of the dimension text of the selected dimension(s).
	Apply Dimension Style	DIMSTYLE + A	Applies the current dimension style to the selected dimension(s).
	Save Dimension Style	DIMSTYLE + S	Saves the current dimension settings in a new dimension style.
	Restore Dimension Style	DIMSTYLE	Sets the current dimension style.
	Dimension Variable Status	DIMSTYLE + ST	Displays the status of the dimension variables in the Command line and the Prompt History window.
	Leader	DIMLEADER	Creates a leader with or without text.

Icon	Tool name	Command	Description
	Quick Leader	QLEADER	Draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.

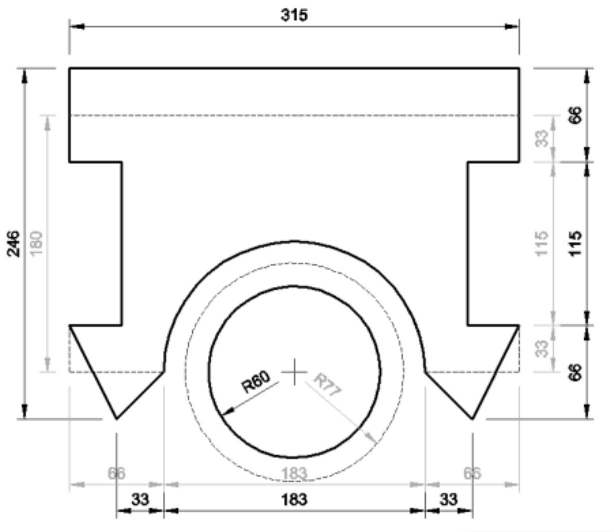
13.1.2 Associative dimensions

Associative dimensions update automatically when the dimensioned entity is moved or modified.

BricsCAD creates associative dimensions if the value of the DIMASSOC system variable is 2 (default).

The associative property of dimensions also applies to dimensions that are placed in paper space and are associated with model space entities.

If necessary, you can use the DIMREGEN command to update all associative dimensions.



The following Entity Snaps can be used to create associative dimensions: **endpoint**, **midpoint**, **center**, **perpendicular**, **quadrant**, **insertion**, **point**, **intersection**, and **apparent intersection**.

The DIMDISASSOCIATE command removes the associativity from selected dimension entities.

DIMREASSOCIATE command re-associates or associates dimensions to entities or points on entities.

The GENERATEASSOCVIEWS system variable enables the creation of associative dimensions for generated views by the VIEWBASE and VIEWSECTION commands and generated drawings by the BIMSECTIONUPDATE command.

The ANNOMONITOR system variable controls if a disassociated dimension is marked with a warning sign.

13.1.3 Dimension settings

Dimension settings are saved in a dimension style. Dimension entities are created using the settings in the current dimension style.

The DIMSTYLE command creates and modifies dimension styles, through the **Drawing Explorer**.

The -DIMSTYLE command creates and modifies dimension styles at the Command line.

A dimension style consists of six settings groups

- **Lines and Arrows**
- **Text**
- **Fit**
- **Primary Units**
- **Alternate Units**
- **Tolerances**

13.1.3.1 To list the current status of the dimension variables

1 Do one of the following:

  - Click the **Dimension Variable Status** tool button () on the **Dimensions** toolbar.
  - Choose **Dimension Variable Status** in the **Dimensions** menu.

  A list of all dimension variables displays in the Command line.

2 Press the **F2** function key to open the **BricsCAD Prompt History** window.

3 Press Enter to continue the listing.

4 Press the **F2** function key to close the **BricsCAD Prompt History** window.

13.1.4 Working with dimension styles

The DIMSTYLE command opens the **Drawing Explorer – Dimension Styles** dialog box.

Here you can:

- Create a new dimension style
- Create dimension child styles
- Edit a dimension style
- Define dimension style overrides
- Rename a dimension style
- Set a dimension style current
- Copy dimension styles between drawings
- Compare dimension styles
- Delete a dimension style

13.1.4.1 To set a dimension style current

First method:

1 Right click the dimension style field in the **Status bar**.

A context menu appears.

2 Select the dimension style in the context menu.

Second method:

1 Launch the DIMSTYLE command.

1 The current dimension style is marked in the **Current** column.

2 Click the current column of the dimension style.

13.1.4.2 To create a new dimension style

1 Launch the DIMSTYLE command.

2 Select an existing dimension style, which will be used as a template for the new style.

3 Do one of the following:
   - Click the **New** button.
   - Place the cursor on an existing dimension style, then right click and choose **New** in the context menu.

4 Select the **Name** field and type a name for the new dimension style.

5 Define the settings for the new dimension style.

Note: Use the **Save to new style** option of the context menu to create a new dimension style as a copy of the selected dimension style.

13.1.4.3 Creating a dimension child style

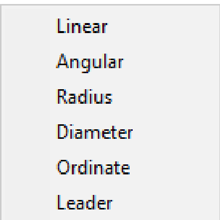
For each dimension style, you can create a child style for each dimension type: **Linear**, **Angular**, **Radius**, **Diameter**, **Ordinate** and **Leader**.

If the current dimension style has a child style for a dimension type, the child style will be used automatically when such dimension type is created.

To create a dimension child style, do the following:

1 Select the parent dimension style, then right click and choose a **New child style** in the context menu.

A context menu appears.



2 Select a dimension entity type in the context menu.

3 The name of the child style is: <parent style>:<dimension entity type>

4 Select the child style and define the properties that need to be different from the parent style for the selected dimension entity type.

13.1.4.4 To edit a dimension style

1 Launch the DIMSTYLE command.

2 Select the dimension style.

3 Adjust the dimension settings.

4 Close the **Drawing Explorer** dialog box.

13.1.4.5 To define dimension style overrides

1 Launch the DIMSTYLE command.

2 Select the dimension style.

3 The first line of the dimension style list reads: <overrides on dimstylename>

4 Define the overrides.

5 Close the **Drawing Explorer** dialog box.

Note:

- It is not recommended to create dimensions with overrides.
- Use the dimension override settings to test changes, without modifying the current dimension style definition, then use the **Save overrides to current style** or **Save to new style** option to save the overrides.
- Setting another style current will discard the existing dimension style overrides.

13.1.4.6 To rename a dimension style

1 Launch the DIMSTYLE command.

2 Select the dimension style.

3 Right click, then choose **Rename** in the context menu.

4 Type a new name in the **Name** field, then press enter.

13.1.4.7 To copy dimension styles between drawings

1 Open both the source and the target drawing.

2 Launch the DIMSTYLE command.

3 Open drawings are listed in the **Open drawings** list on the **Drawing Explorer** dialog box.

4 Select **Dimension styles** of the source drawing in the **Open drawings** list.

5 Select the dimension style(s) to be copied.

6 Hold down the Ctrl key to select multiple dimension styles.

7 Place the cursor over one of the selected dimension styles, then press and hold the left mouse button and drag the selection onto the target drawing, then release the left mouse button.

13.1.4.8 To compare dimension styles

1 Launch the DIMSTYLE command.

2 Select the dimension styles.

3 Click the first dimension style, then hold down the Ctrl key to select the next dimension style(s).

The properties of the selected styles display. The settings that are different, are highlighted.

4 (option) Hold down the Ctrl key to add or remove a dimension style from the comparison table.

13.1.5 Using the dimension style tools

The dimension style tools on the **Dimensions** toolbar and the **Dimensions** ribbon panel allow to:

- Apply the current dimension style to a selection of dimension entities.
- Create a new dimension style.
- Restore a dimension style.
- List the current status of the dimension settings.

13.1.5.1 Applying the current dimension style

1 Do one of the following:

- Click the **Apply Dimension Style** tool ( ).
- Choose **Apply Dimension Style** in the **Dimensions** menu.
- Launch the -DIMSTYLE command, then type **a** or **apply** and press Enter.

2 **You are prompted**: Select dimensions to apply the current style.

3 Select the dimensions, then press Enter.

Note: Since the dimension style settings only apply to dimension entities, you can select all entities in step 2 to apply the current dimension style to all dimensions in the drawing.

13.1.5.2 Saving a dimension style

1 Do one of the following:

- Click the **Save Dimension Style** tool button ( ) on the **Dimensions** toolbar.
- Choose **Save Dimension Style** in the **Dimensions** menu.
- Launch the -DIMSTYLE command, then type **S** or **save** and press Enter.

You are prompted: Enter name for style to save or [? to list styles]:

2 Type a name for the new dimension style, then press Enter.

3 If the name already exists, the Command line prompts you to redefine it or not.

13.1.5.3 Restoring a dimension style

1 Do one of the following:

- Click the **Restore Dimension Style** tool button ( ) on the **Dimensions** toolbar.
- Choose **Restore Dimension Style** in the **Dimensions** menu.
- Launch the -DIMSTYLE command, then type **r** or **restore** and press Enter.

You are prompted: Dimension style to restore or [? to list styles] <Select dimension>:

2 Do one of the following:
   - Type a dimension style name and press Enter.
   - Press Enter, then click a dimension entity in the drawing.
   - The dimension style of the selected entity is the current dimension style now.
   - Type ? to see a list of the dimension styles in the current drawing.

13.1.5.4 Listing the settings in the current dimension style

Do one of the following:

- Click the **Dimension Variable Status** tool button ( ) on the Dimensions toolbar.
- Choose **Dimension Variable Status** in the **Dimensions** menu.
- Launch the -DIMSTYLE command, then type **st** or **status** and press Enter.

The status of the settings in the current dimension style are listed in the **Prompt History** window. Press the F2 function key to close the **Prompt History** window.

13.1.6 Creating linear dimensions

Linear dimensions annotate linear distances or lengths and can be oriented horizontally, vertically or aligned parallel to an existing entity or to the selected origin points.

The following commands create linear dimension entities:

- DIMLINEAR: creates horizontal and vertical linear dimensions with respect to the current coordinate system (World or UCS).
- DIMALIGNED: creates dimensions parallel to (aligned with) lines and polyline segments. The command also works with polyline arc segments, arcs, and circles, but dimensions their diameters (see DIMDIAMETER command).
- DIMBASELINE: places multiple linear and angular dimensions, all starting at the same base point of an existing dimension. This command works only when at least one other dimension already exists in the drawing.
- DIMCONTINUE: places a chain of dimensions by continuing linear or angular dimensions from the endpoint of the previous dimension or another existing dimension.
- DIMARC: places dimensions that measure the lengths of arcs and polyline arc segments.

13.1.6.1 Creating a horizontal or vertical linear dimension

1 Launch the DIMLINEAR command.

1 **You are prompted**: Origin of first extension line <Select entity>:

2 Do one of the following:
   - Specify the origin of the first extension line, then specify the origin of the second extension line.
   - Press Enter, then select the line or linear segment of a polyline you want to dimension.

The dimension line is dynamically displayed.

Move the cursor in to create a horizontal dimension, or horizontally to create a vertical dimension.

3 Specify a point to position the dimension line.

13.1.6.2 Creating an aligned linear dimension

1 Launch the DIMALIGNED command.

1 **You are prompted**: Origin of first extension line <Select entity>:

2 Do one of the following:

- Specify the origin of the first extension line, then specify the origin of the second extension line.
- Press Enter, then select the line or linear segment of a polyline you want to dimension.

The dimension line is dynamically displayed.

3 Specify a point to position the dimension line.

13.1.6.3 Creating a rotated linear dimension

1 Do one of the following:

- Click the **Rotated** tool button on the **Dimensions** toolbar or ribbon panel.
- Choose **Rotated** in the **Dimensions** menu.

You are prompted: Origin of first extension line <Select entity>:Do one of the following:

2 Do one of the following:

- Specify the origin of the first extension line, then specify the origin of the second extension line.
- Press Enter, then select the line or linear segment of a polyline you want to dimension.

You are prompted: Angle of dimension line <0>:

3 Do one of the following:

- Type the rotation angle in the Command line, then press Enter.
- Enter the rotation angle by specifying two points.

The dimension line is dynamically displayed.

4 Specify a point to position the dimension line.

13.1.6.4 Creating baseline and continued dimensions

Note:

- When selecting the starting dimension, the baseline or continued dimension starts from the selected extension line or the nearest extension line, when you click the dimension line itself.
- If you select an angular dimension, stacked or continued angular dimensions are created.
- The distance between the stacked dimension lines is controlled by the DIMDLI system variable (Dim baseline spacing property in the dimension style).

Creating baseline dimensions

1 Launch the DIMBASELINE command.

2 **If no linear or angular dimensions have been created yet in the current session, you are prompted**: Select dimension for baseline:

Select a linear or angular dimension.

A baseline dimension displays dynamically.

Otherwise you are prompted:

Baseline: Origin of next extension line or [Select starting dimension/Undo] <Select starting dimension>:

A Baseline dimension displays dynamically, from the most recently created linear or angular dimension.

Optionally, press Enter and select another linear or angular dimension.

3 Specify a point to create the first baseline dimension.

A new baseline dimension displays from the same starting dimension dynamically.

4 Do one of the following:

- Specify a point to create a new baseline dimension.
- Press Enter to select another starting dimension for the next baseline dimension(s).
- Press the Esc key to stop.

5 Repeat step 4.

Creating continued dimensions

1 Launch the DIMCONTINUE command.

2 **If no linear or angular dimensions have been created yet in the current session, you are prompted**: Select dimension to continue:

Select a linear or angular dimension.

A baseline dimension displays dynamically.

Otherwise you are prompted:

Continue: Origin of next extension line or [Select starting dimension/Undo] <Select starting dimension>:

A new dimension displays dynamically, from the most recently created linear or angular dimension.

Optionally, press Enter and select another linear or angular dimension.

3 Specify a point to create the first continued dimension.

A new continued dimension displays dynamically.

4 Do one of the following:

- Specify a point to create a new continued dimension.
- Press Enter to select another starting dimension for the next continued dimension(s).
- Press the Esc key to stop.

5 Repeat step 4.

13.1.6.5 Creating an arc length dimension

1 Launch the DIMARC command:

1 **You are prompted**: Select arc or polyline arc segment:

2 Select the arc or polyline arc segment.

3 The arc length dimension is dynamically displayed.

4 Specify a point to position the dimension line.

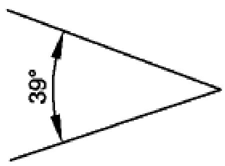
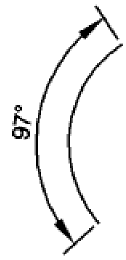
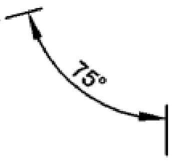
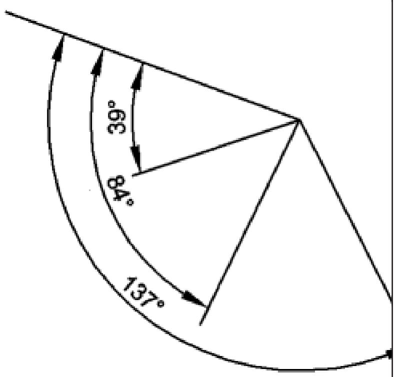
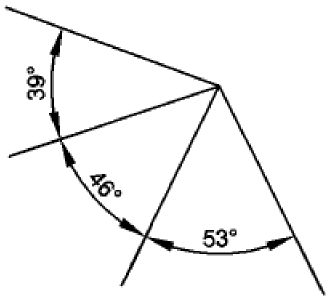
13.1.7 Creating angular dimensions

The DIMANGULAR command creates angular dimension.

Angular dimensions annotate the angle measured between two lines, two linear segments of a polyline or the angle of a circular arc. You can also dimension an angle by selecting an angle vertex and two endpoints.

Starting from an existing angular dimension, you can add a baseline dimension or a continued dimension.

- An angular baseline dimension inserts an additional dimension from a common first extension line origin.
- An angular continued dimension continues an angular dimension from the second extension line of a previous angular dimension.

		
angular dimension measured between two lines	angular dimension of an arc	angular dimension defined by a vertex and two endpoints
		
angular baseline dimensions	angular continued dimensions	

13.1.7.1 To dimension an angle between two lines

1 Launch the DIMANGULAR command.

1 **You are prompted**: Select line, arc, or circle [selection options (?)] <Press ENTER to specify angle>:

2 Select a line or linear polyline segment.

3 **You are prompted**: Other line for angular dimension [selection options (?)]:

4 Select a second line or linear polyline segment.

5 The angular dimension is dynamically displayed.

You are prompted: Location of dimension arc [Angle/Text]:

6 Specify a point to position the dimension.

7 Move the cursor to dimension either the interior angle between the lines or its supplement.

13.1.7.2 To dimension an angle encompassed by an arc

1 Launch the DIMANGULAR command.

1 **You are prompted**: Select line, arc, or circle [selection options (?)] <Press ENTER to specify angle>:

2 Select an arc.

3 The angular dimension is dynamically displayed.

You are prompted: Location of dimension arc [Angle/Text]:

4 Specify a point to position the dimension.

13.1.7.3 To create an angular dimension defined by a vertex and two endpoints

1 Launch the DIMANGULAR command.

1 **You are prompted**: Select line, arc, or circle [selection options (?)] <Press ENTER to specify angle>:

2 Press Enter.

3 **You are prompted**: Vertex of angle:

4 Specify a point.

5 **You are prompted**: First side of angle:

6 Specify a point.

7 **You are prompted**: Other side of angle:

8 Specify a point.

9 The angular dimension is dynamically displayed.

You are prompted: Location of dimension arc [Angle/Text]:

10 Specify a point to position the dimension.

11 Move the cursor to dimension either the interior or the exterior angle.

13.1.8 Creating diametrical and radial dimensions

The DIMCENTER and DIMDIAMETER commands create diametrical and radial dimensions to annotate the radii and diameters of circles and circular arcs.

13.1.8.1 To create a diametrical dimension

1 Launch the DIMDIAMETER command.

1 **You are prompted**: Select arc or circle to dimension [selection options (?)]:

2 Select an arc or a circle.

3 The diametrical dimension is dynamically displayed.

4 Specify a point to position the dimension entity.

13.1.8.2 To create a radial dimension

1 Launch the DIMRADIUS command.

1 **You are prompted**: Select arc or circle to dimension [selection options (?)]:

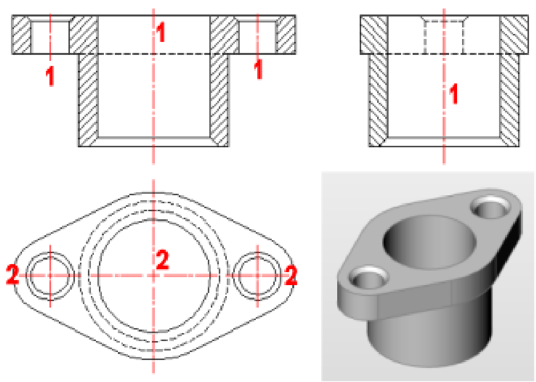
2 Select an arc or a circle.

3 The radial dimension is dynamically displayed.

4 Specify a point to position the dimension entity.

13.1.9 Centerlines and center marks

Centerlines and center marks are drawing reference entities to indicate axes of symmetry and centers of circular openings. Centerlines and center marks are associative entities: when the referenced lines, circles or circular arcs are modified they are adjusted automatically.



1 Centerline

2 Center mark

The CENTERLINE command creates a centerline entity associated with two selected lines.

The CENTERMARK command creates a center mark entity associated with a selected circle or arc.

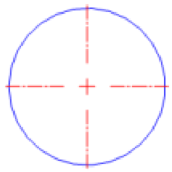
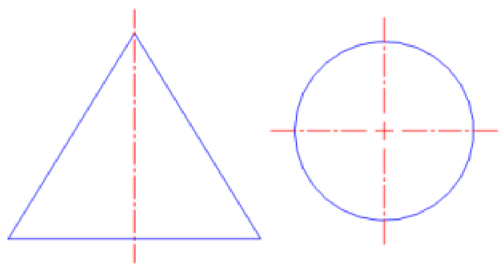
The CENTERREASSOCIATE command recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc.

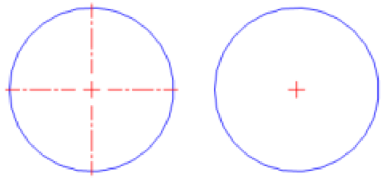
The CENTERDISASSOCIATE command breaks the association of centerline and a center mark entities.

The CENTERRESET command resets centerline and center mark entities.

13.1.9.1 System variables for center entities

The properties and geometry of centerline and center mark entities are controlled through system variables.

Name	Title	Description
CENTERCROSSGAP	Center mark cross gap	Defines the gap between the center mark and its centerlines.  Type a value to specify the absolute size of the gap in drawing units. Type a value followed by x to set the size of the gap relative to the diameter of the circle or circular arc. The default value 0.05x creates a gap of 5/100 of the diameter. . (dot) creates no gap.
CENTERCROSSSIZE	Center mark cross size	Defines the size of the associative center mark. Type a value to specify the absolute size of the center mark lines in drawing units. Type a value followed by x to set the size of the center mark relative to the diameter of the circle or circular arc. The default value 0.1x creates center marks 1/10th of the diameter. Type a . (dot) to create center marks with zero length lines.
CENTEREXE	Centerline extensions length	Defines the length of the centerline extensions. The value is expressed in drawing units. 

Name	Title	Description
CENTERLAYER	Default layer for center mark or centerline	Specifies the default layer for new centerlines and center marks. Type a name. If such layer not yet exists, the layer will be created when a centerline or centemark is created. Type a . (dot) to create centerlines and center marks on the current layer.
CENTERLTYPE FILE	Linetype file for center mark or centerline	Specifies the linetype file used to create centerlines and center marks. Default in imperial unit drawings: **default.lin** Default in metric unit drawings: **iso.lin**
CENTERLTYPE	Linetype for center mark or centerline	Specifies the linetype for newly created centerlines and center marks. Type a new linetype name or press Enter to accept the current name. Type a . (dot) for continuous lines.
CENTERLTSCA LE	Linetype scale for center mark or centerline	Specifies the linetype scale for newly created centerlines and center marks.
CENTERMARK EXE	Automatic extension for center mark or centerline	Determines whether centerlines extend (left) or not (right) for new center marks. 

13.1.9.2 Editing center entities

The properties of center entities can be edited in the **Properties** panel.

13.1.10 Creating ordinate dimensions

An ordinate dimension annotates the perpendicular distance from an origin or base point (the origin of the current coordinate system (WCS or UCS). Ordinate dimensions consist of an x- or y-coordinate and a leader. An x-ordinate dimension measures distances along the x-axis, while a y-ordinate dimension measures distances along the y-axis.

As you select ordinate points, the program automatically detects whether the point is x- or y-ordinate based depending on the direction you drag the second point. You can also specify whether the ordinate represents an x- or y-ordinate. Ordinate dimension text is always aligned with the ordinate leader lines, regardless of the text orientation specified by the current dimension style.

13.1.10.1 To create an ordinate dimension

1 Set the origin of the coordinate system at the reference point for the ordinate dimensions.

2 Launch the DIMORDINATE command.

3 **You are prompted**: Select point for ordinate dimension:

4 Specify a point.

5 The ordinate dimension is dynamically displayed.

You are prompted: Leader endpoint [Xdatum/Ydatum/Text/Angle]:

6 Click to position the ordinate dimension.

13.1.11 Editing dimensions

You can edit dimension entities:

- Using grips.
- Edit the dimension text.
- Rotate the dimension text.
- Reposition the dimension text.
- Restore the text position.
- Make the extension lines oblique.

13.1.11.1 Using grips to edit a linear dimension

1 Select the dimension entity.

1 5 grips display:

2 origin point grips (1)

2 dimension line grips (2)

1 dimension text grip (3)

2 (option) Drag the origin point grips to adjust the dimension origin points.

3 The dimension text is adjusted automatically.

4 (option) Drag dimension line grips to move the dimension line.

5 (option) Drag the text grip to move the dimension text.

Note:

- Depending on the **Text movement** setting (DIMTMOVE system variable), the dimension line will move along with the text or the text will move independently. A leader will be drawn if the **Text movement** setting is set to **Move text, add leader**.
- If two adjacent dimensions (continued dimensions) are selected both, you can move their coinciding grips simultaneously.

13.1.11.2 Editing the dimension text

Dimension text can be edited:

- in the MText editor:
  - using the DDEDIT command
  - double clicking the dimension entity
- in the **Text override** field in the **Properties** panel.
- using the **Edit Text** option of the DIMEDIT command.

Note:

The following tags can be used to format the dimension text:

- **\L** (backslash upper case L) starts underlined text, **\l** (backslash lower case L) to stop underlined text
- **<>** to add the default dimension text (= measured distance)
- **[]** to add the alternate units, when alternate units are currently switched off in the dimension style.
- **%%d** to add the degree symbol (°)
- **%%c** to add the diameter symbol (Ø)
- **%%p** to add the plus/minus symbol (±)

e.g. if you type **%%c\L<> cm** in the **Text override** field, dimension text displays as:

Ø24,43cm

13.1.11.3 To rotate the dimension text

1 Do one of the following:

- Click the **Rotate Dimension Text** tool ( ) in the **Dimensions** toolbar or ribbon panel.
- Choose **Rotate Dimension Text** in the **Dimensions** menu.
- Launch the DIMEDIT command then choose the **Rotate text** option.

You are prompted: Angle of dimension text:

2 Type a value, then press Enter.

3 **You are prompted**: Select entities to rotate text [selection options (?)]:

4 Select the dimensions and press enter.

13.1.11.4 To reposition the dimension text

1 Do one of the following:

- Click the **Reposition Dimension Text** tool ( ) in the **Dimensions** toolbar or the ribbon panel.
- Choose **Reposition Dimension Text** in the **Dimensions** menu.
- Launch the DIMTEDIT command.

You are prompted: Select dimensions to reposition text:

2 Select the dimension.

3 The dimension text moves dynamically.

4 Click to reposition the dimension text.

13.1.11.5 To restore the text position

1 Do one of the following:

- Click the **Restore Text Position** tool ( )
- Choose **Restore Text Position** in the **Dimensions** menu.
- Launch the DIMEDIT command and choose the **Restore text** option.

You are prompted: Select dimensions to restore text to default position [selection options (?)]:

2 Select the dimensions, then press Enter.

13.1.11.6 To make the extension lines oblique

1 Do one of the following:

- Click the **Make Oblique** tool ( ) in the **Dimensions** toolbar or ribbon panel.
- Choose **Make Oblique** in the **Dimensions** menu.
- Launch the DIMEDIT command and choose the **Oblique lines** option.

You are prompted: Select linear dimensions to make oblique [selection options (?)]:

2 Select the dimension entities, then press Enter.

3 **You are prompted**: Enter obliquing angle:

4 Type a value, then press Enter.

Note: To restore non-rotated extension lines, execute the above procedure and type 0 (zero) at the 'Enter obliquing angle' prompt.

13.1.12 Creating leaders and annotations

A leader consists of an arrowhead, a line, polyline or a spline and an annotation.

The general procedure to place an annotation is:

- Specify the position of the arrowhead.
- Create the leader line segments.
- Type the annotation text.

13.1.12.1 Creating a leader and an annotation

1 Launch the LEADER or DIMLEADER command.

1 **You are prompted**: Start of leader

2 Specify the position of the arrowhead.

3 **You are prompted**: Next point.

4 Specify the endpoint of the first leader segment.

5 **You are prompted**: To point [Format/Undo/Annotation] <Annotation>:

6 Add more segments to the leader, then press Enter or right click to stop adding segments.

7 **You are prompted**: First line of annotation text/<options>:

8 Type the first line of the annotation text, then press Enter.

9 **You are prompted**: Next line of annotation text:

10 Do one of the following:

- Press Enter or right click to create the leader and annotation.
- Keep adding lines to the annotation text, then press Enter or right click to create the leader and annotation.

13.1.12.2 Creating a leader only

1 Launch the LEADER or DIMLEADER command.

1 **You are prompted**: Start of leader

2 Specify the position of the arrowhead.

3 **You are prompted**: Next point.

4 Specify the endpoint of the first leader segment.

5 **You are prompted**: To point [Format/Undo/Annotation] <Annotation>:

6 Add more segments to the leader, then press Enter or right click to stop adding segments.

7 **You are prompted**: First line of annotation text/<options>:

8 Press Enter.

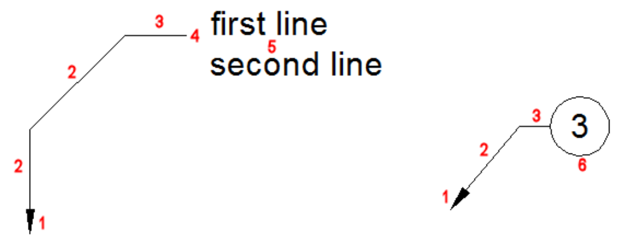
9 **You are prompted**: Dimension text options [Block/Copy/None/Tolerance/Mtext] <Mtext>:

10 Choose the **None** option.

13.1.13 Creating multileaders

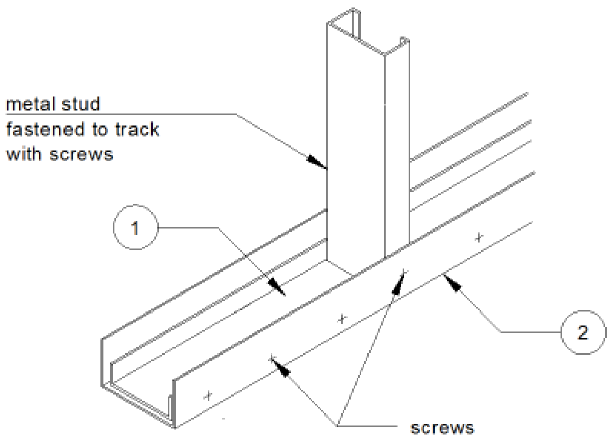
A multileader consists of:

- an arrowhead (1)
- one or more leader lines (2), which can be straight or curved
- optionally a horizontal landing (3) and landing gap (4)
- content: either a multiline text (5) or a block (6)



Multileaders are created using the current multileader style. Each drawing contains a **Default** leader style, which cannot be renamed or deleted; the **Default** leader style can be customized though. The MLEADER command allows to define overrides on the current multileader style.

Multileaders can be created arrowhead first, landing first or content first.



13.1.13.1 To create a multileader

1 (option) Launch the MLEADERSTYLE command, then set the current multileader style.

2 Do one of the following:

- Click the **Multileader** tool button ( ) on the **Dimensions** toolbar or ribbon panel.
- Choose **Multileader** in the **Dimensions** menu.
- Launch the MLEADER command.

You are prompted: Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>:

3 (option) Select a placement option:

- arrowhead first
- landing first
- content first.

The most recently used placement option is repeated.

4 The default sequence is: arrowhead - landing - content.

5 (option) Choose **Options** if you want to define overrides on the current multileader style.

You are prompted: Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/

Second angle/eXit options] <eXit options>:

Specify the option(s), then choose **Exit** to proceed.

6 Specify a point to define the leader arrowhead location.

7 **You are prompted**: Specify leader landing location:

8 Specify a point to define the leader landing location.

9 Depending on the content type:
   - **MText**: The MText command is launched.
   - **Block**: A block is inserted.
   - **None**: The command is concluded.

13.1.13.2 Adding vertices to a leader line

1 If necessary press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the leader line you want to add a vertex to, then hold down the Ctrl key.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu and expand the **Modify** tab.

5 Click the **Add Vertex** tool ( ).

A new vertex is added dynamically.

6 Specify the position of the vertex.

13.1.13.3 Removing vertices from a leader line

1 If necessary press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the leader line you want to add a vertex to, then hold down the Ctrl key.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu expand the **Modify** tab.

5 Click the **Remove Vertex** tool ( ).

The vertex is removed.

13.1.13.4 Adding leader lines to a multileader

1 If necessary press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the multileader entity.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu and expand the **Modify** tab.

5 Click the **Add Leader** tool ( ).

A new leader line is dynamically displayed.

6 Specify the origin of the new leader line*.

7 Another new leader line is dynamically displayed.

8 Do one of the following:
- Specify the origin of the new leader line.
- Right click or press Enter to stop adding leader lines.

(*) When the Maximum leader points property of the multileader style is higher than 2, right click to stop adding vertices for the new leader.

13.1.13.5 Removing leader lines from a multileader

1 If necessary, press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the leader line you want to remove.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu and expand the **Modify** tab.

5 Click the **Remove Leader** tool ( ).

The leader line is removed.

13.1.14 Adding geometric tolerances

Geometric tolerances specify the maximum variation that is allowed in form or position from true geometry. The geometric tolerance is, in essence, the tolerance zone within which a surface or axis of hole or cylinder can lie which results in resulting feature being acceptable for proper function and interchangeability.

BricsCAD draws geometric tolerances using a feature control frame, which is a rectangle divided into compartments.

Each feature control frame consists of at least two compartments. The first compartment contains a geometric tolerance symbol that indicates the geometric characteristic to which the tolerance is applied, such as location, orientation, or form. For example, a form tolerance may indicate the flatness or roundness of a surface.

The second compartment contains the tolerance value. When appropriate, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol. The material conditions apply to features that can vary in size.

The tolerance value can then be followed by primary, secondary, and tertiary datum reference letters, along with the material conditions of each datum. Datum reference letters are generally used as reference tolerances to one of up to three perpendicular planes from which a measurement is made, although datum reference letters can also indicate an exact point or axis.

1 Geometric characteristic symbol

2 Diameter symbol

3 Tolerance value

4 Material condition symbol

5 Datum reference

When two tolerances apply to the same geometry, you can also add a composite tolerance consisting of a primary tolerance value followed by a secondary tolerance value. To make a tolerance even more specific, it can also contain a projected tolerance consisting of a height value followed by a projected tolerance symbol. For example, you can use a projected tolerance to indicate the perpendicularity of an embedded part.

13.1.14.1 Creating a geometrical tolerance frame

1 Do one of the following:

- Click the **Tolerance** tool button (⊥) in the **Dimensions** toolbar or ribbon panel.
- Choose **Tolerance** in the **Dimensions** menu.
- Type **tolerance** in the Command line, then press Enter.

The Geometric Tolerance dialog opens.

2 Compose the geometrical tolerance frame in the **Geometric Tolerance** dialog box.

3 Click the **OK** button.

The **Geometric Tolerance** dialog box closes.

4 Specify a point to insert the geometric tolerance frame in the drawing.

13.1.14.2 Editing a geometrical tolerance frame

1 Double click the geometrical tolerance frame.

1 The **Geometric Tolerance** dialog box opens.

2 Edit the settings.

3 Click the **OK** button.

The **Geometric Tolerance** dialog box closes.

The content of the geometrical tolerance frame is updated.

13.2 Texts

13.2.1 Text system variables

System variables can be set using the **Settings** dialog box.

Variable Name	Variable Title	Description
FONTALT	Alternate font	Defines the font which will be used if a text font cannot be found.

Variable Name	Variable Title	Description
HIDETEXT	Hide text on hide	Specifies whether the HIDE command processes text entities.
MIRRTEXT	Mirror text	Determines whether text is mirrored by the MIRROR command or not.
MTEXTED	Multiline text editor	Sets the primary and secondary text editors to use for multiline text entities.
MTEXTFIXED	Multiline text fixed	Controls whether BricsCAD zooms, rotates and/or pans the view to fit the Mtext being edited.
QTEXTMODE	Quick text mode	Controls how text entities are displayed. If ON, displays text as a box.
TEXTANGLE	Text angle	Stores the angle of the last added text entity.
TEXTFILL	Text fill	Determines whether True Type fonts show as either filled or outlined. **Filled Text** Outlined Text
TEXTQLTY	Text quality	Defines the smoothness of True Type fonts for plotting and rendering.
TEXTSIZE	Text size	Sets the default height for new text entities. TEXTSIZE has no effect if the current text style has a fixed height.
TEXTSTYLE	Text style	Stores the name of the current text style.
TSPACEFAC	Text space factor	Specifies the line spacing distance of multiline text measured as factor of the text height. Values between 0.25 and 4 are accepted.
TSPACETYPE	Text space type	Specifies the type of line spacing used for multiline text. At least: adjusts line spacing based on the tallest character(s) in a line. Exactly: used the specified line spacing, regardless of in individual character sizes.
TSTACKALIGN	Text stack align	Determines the vertical alignment of stacked text: bottom, center or top.

Variable Name	Variable Title	Description
TSTACKSIZE	Text stack size	Specifies the percentage a which the stacked text fraction height is relative to the current height of the selected text. Values between 25 and 125 are accepted.

13.2.2 Working with text styles

When you add text to a drawing, it uses the current text style, which determines the font, size, angle, orientation and other properties of the new text entity.

Every drawing has a default text style, named **Standard**, which initially uses the **Arial** font. You cannot delete or rename the **Standard** style, but you can modify it.

If you change the **font** or **orientation** of an existing style, all existing text entities created using that style are automatically updated to reflect the change. Changing any other property has no effect on existing text.

You can create and use an unlimited number of additional text styles.

Annotative text styles create annotative text entities. In an annotative text style, the **Height** defines the paper space height of the text; the model space height depends on the current annotation scale as defined by the CANNOSCALE and CANNOSCALEVALUE system variables.

13.2.2.1 To open the Text Style Explorer dialog box

Do one of the following:

- In the **Tools** > **Drawing Explorer** menu, choose **Text Styles ...**
- Click the **Style** field in the **Status** bar.
- Launch the STYLE command.

13.2.2.2 To create a text style

1 Open the **Text Style Explorer** dialog box.

2 Do one of the following:

  - Click the **New** tool button ( ) in the **Details** section of the **Drawing Explorer** dialog box.
  - Select an existing style, then right click and choose **New** in the context menu.

  A **NewStyle** text style is created, inheriting the properties of the currently selected style.

3 Type a name in the **Text Style Name** field, replacing the **NewStyle** default name.

4 Define the text style properties.

5 Close the **Text Style Explorer** dialog.

13.2.2.3 To edit a text style

1 Open the **Text Style Explorer** dialog box.

2 Select the text style.

3 Define the text style properties.

4 Close the **Text Style Explorer** dialog.

13.2.2.4 To set a text style current

Do one of the following:

- Right click the **Style** field in the **Status** bar, then click the text style.
- Open the **Text Style Explorer** dialog box.

  The current text style is indicated in the **Current** column.

  Do one of the following:

  - Click the **Current** field of the text style.
  - Right click the text style, then choose **Set Current** in the context menu.

13.2.2.5 To delete a text style

1 Open the **Text Style Explorer** dialog box.

2 Select the text style.

3 Do one of the following:

  - Click the **Delete** tool button (✕).
  - Select the text style, then choose **Delete** in the context menu.

  If the text style is not used in the drawing or in a dimension style, the text style is deleted, else an alert box appears.

4 Do one of the following:

  - Click the **Delete** button to delete the text style and all entities using it.
  - Click the **Change...** button to delete the text style and assign a different text style to the entities, which currently use the selected style.
  - Click the **Cancel** button to cancel the delete procedure.

Note: It is not possible to delete the current text style.

13.2.3 Creating text

The TEXT command places single lines of text in the drawing and optionally evaluates LISP expressions.

The MTEXT command places formatted paragraph text in a bounding box that limits the extent of the text.

13.2.3.1 Creating single line texts

1 Do one of the following:

  - Click the **Text** tool button (⊤) on the **Draw** toolbar or the **Annotate / Text** ribbon panel.
  - Choose **Text** in the **Draw** menu.
  - Launch the TEXT command.

2 **You are prompted**:

  Text start point or [use defined Style/Align on line/Fit between points/Center horizontally/Middle (hor/

vert)/Right justify/Justification options...]:

3 Specify a point.

4 **You are prompted**:

Height of text <current height>:

5 Do one of the following:

- Press Enter or right click to accept the current height.
- Enter a value.
- Click a point to define the height graphically.

You are prompted:

Rotation angle of text <current angle>:

6 Do one of the following:

- Press Enter or right click to accept the current rotation angle.
- Type a value.
- Click a point to define the rotation angle graphically.

7 Type the text, then press Enter.

8 A new empty text is created.

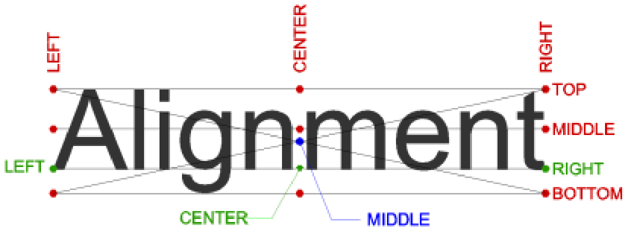
9 Do one of the following:

- Type the text.
- Press Enter to stop.

10 (Optional) Keep adding texts then press Enter twice to stop.

Note: When you have already created text in the current BricsCAD session, the most recent text entity highlights when you repeat the TEXT command. Press Enter or right click to create a new text below the most recently created text, using the same text height and rotation angle.

Single line text justification options:



Left
Center
Right
Middle
Bottom left
Bottom center
Bottom right
Middle left
Middle center
Middle right
Top left
Top center
Top right

13.2.3.2 Creating multiline text

1 Do one of the following:

- Click the **Multiline Text** tool button ( ) on the **Draw** toolbar or the **Annotate / Text** ribbon panel.
- Choose **Multiline Text** in the **Draw** menu.
- Launch the MTEXT command.

You are prompted:

First corner for block of text:

2 Specify the insertion point of the multiline text entity.

3 **You are prompted**:

Select Opposite corner for block of text or [Justification/Rotation angle/text Style/text Height/Direction/Width/Line spacing/Columns]:

4 Do one of the following:

- Specify the opposite corner of the text block to define the maximum width of the multiline text.
- If **Dynamic Dimensions** are active, the height and width of the text field are displayed dynamically. Type the height and width in the dynamic dimension fields. Press the Tab key to switch; press Enter to confirm.
- Type @ in the Command line, then press Enter for non-wrapped (zero width) text (= unlimited line width).

The **Text formatting** toolbar opens right above the text block.

5 Type the text.

6 The maximum line width is defined by the width of the text block as defined in step 3.

Press Enter to start a new paragraph.

7 To conclude the command and close the **Text formatting** toolbar do one of the following:

- Click the **OK** button
- Press Ctrl+Enter.

- Click outside the MText entity.

Text formatting toolbar tools and settings

Button or setting	Name	Description
Standard	Text Style	Sets the text style.
[Arial]	Text Font	Sets the text font.
2.5 (Default)	Text Height	Sets the text height.
B	Bold	Creates bold text.
I	Italic	Creates italic text.
T	Underscore	Creates underscored text.
T	Overscore	Creates overscored text.
T	Strike	Creates a strike-through text.
	Uppercase	Converts selected text to uppercase.
	Lowercase	Converts selected text to lowercase.
	Annotative	Toggles the Annotative property.
	Line Spacing	Sets the line spacing.
	Columns	Arranges the text in columns.
ABC	Toggle spellchecker	Toggles the Spell Checking for Mtext editor.
	Match Properties	Matches Mtext fragment formatting.

Button or setting	Name	Description
ByLayer	Color	Sets the color.
0	Oblique Angle	Sets the oblique angle.
1	Width Factor	Sets the width factor.
1	Character Spacing	Sets the character spacing.
	Undo	Undoes recent actions.
	Redo	Redoes previously undone actions.
	Stacked text	Creates stacked text (fractions, subscripts and superscripts). ***Note:*** See **Formatting stacked text** below for more information. To create a subscript or superscript, insert the ^ character before the text that should become a subscript or after the text that should become a superscript. Then select the text and the ^ character and use the **Stacked Text** option.
@	Special Character	Inserts special characters.
	Field	Inserts a variable field.
	Justify	Sets the text justification.
	Paragraph alignment	Sets the paragraph alignment.
	Background Mask	Adds a colored background.
T^2	Toggle superscript	Creates superscript text.
T_2	Toggle subscript	Creates subscript text.
	Ruler	Toggles the ruler display.

Button or setting	Name	Description
	Numbered and bullet lists	Creates numbered/bullet lists.
OK	OK	Closes the **Text formatting** toolbar and applies the changes.
Cancel	Cancel	Closes the **Text formatting** toolbar and leaves the MText entity unchanged.

Multiline text justification

The red dot indicates the insertion point of the multiline text entity.

This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.	This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.	This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.
Top left	Top center	Top right
This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.	This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.	This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.
Middle left	Middle center	Middle right
This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.	This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.	This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.
Bottom left	Bottom center	Bottom right

13.2.3.3 Including special text characters and symbols

You can use control codes and Unicode strings to include special characters in text entities.

To include control codes, type 2 percent symbols (%%) followed by the special control code or character. A single percent sign is treated as a normal text character.

Special text characters

Control Code	Unicode String	Result
%%d	\U+00B0	Draws a degree symbol (°)

Control Code	Unicode String	Result
%%p	\U+00B1	Draws a plus-or-minus symbol (±)
%%c	\U+2205	Draws a diameter symbol (Ø)
%%k	\U+0336	Starts to strike-through text. Repeat the code to stop.
%%u	\U+005F	Starts to underscore text. Repeat the code to stop.
%%o	\U+203E	Starts to overscore text. Repeat the code to stop.

Symbols and Unicode strings

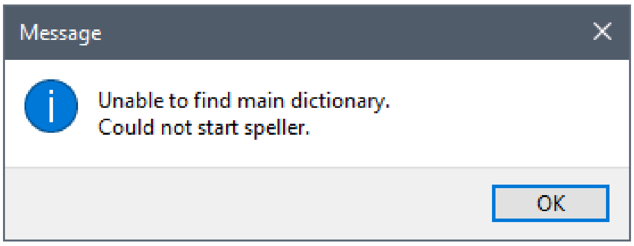
Name	Symbol (*)	Unicode String
Almost equal	≈	\U+2248
Angle	∠	\U+2220
Center line	℄	\U+2104
Delta	Δ	\U+0394
Electrical phase	ɸ	\U+0278
Identity	≡	\U+2261
Not equal	≠	\U+2260
Omega	Ω	\U+03A9
Squared	²	\U+00B2
Cubed	³	\U+00B3

(*) If supported by the text font.

13.2.3.4 Spellchecker

When the spellchecker button ( ) is activated, the spell checking is available in the Mtext editor. The misspelled words are underlined in red in real time. Right click the string to display spelling suggestions.

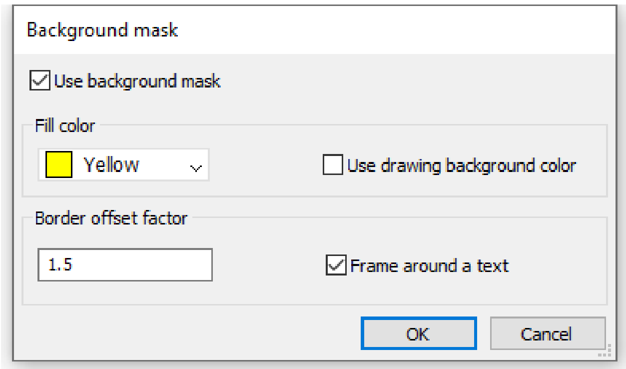
A warning message appears when a valid dictionary is missing. See the SPELL command.



13.2.3.5 Adding a background mask

1 When creating or editing a multiline text entity, click the **Background Mask** tool button ( ) on the **Text Formatting** toolbar.

The **Background Mask** dialog box appears:



2 Check the **Use background mask** option.

3 Do one of the following:
- Choose a color in the **Fill color** drop-down list.
- Choose **Select color...** in the **Fill color** drop-down list and choose a color in the **Color** dialog box.
- Check the **Use drawing background color** option.

4 Define the **Border offset factor**, which sets the margin around the text for the background. The value is based on the text height and must be in the range 1-5. A factor of 1.0 fits the multiline text entity exactly.

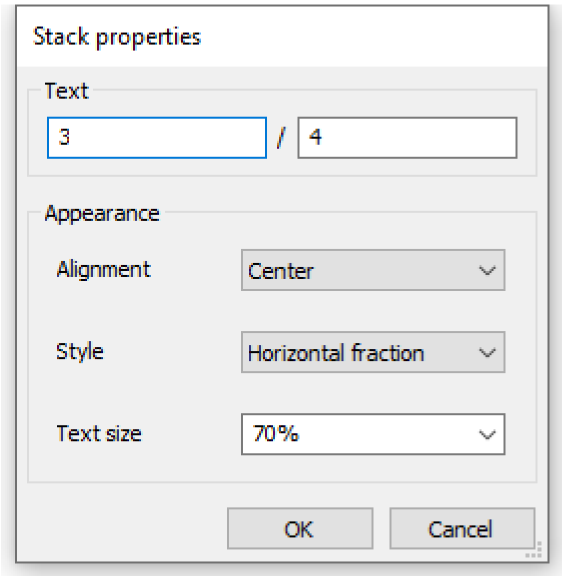
5 (Optional) Tick the **Frame around a text** option to create a text frame.

6 Click **OK** to create the background mask.

13.2.3.6 Formatting stacked text

Stacked fraction text can be formatted using the **Stack properties** dialog box.

Right-click the stacked fraction text, then choose **Stack properties** in the context menu. The **Stack properties** dialog box displays:



The **Stack properties** dialog box allows you to edit the values of the fraction as well as the **Appearance**:

- **Alignment**: allows you to switch between **Bottom**, **Center** and **Top**.
- **Style**: allows you to switch **Diagonal fraction**, **Horizontal fraction** and **Tolerance**.
- **Text size**: adjusts the text size based on a percentage of the set text height. Enter a value or choose a value from the drop-down list.

13.2.4 Editing text

You can edit and modify a text entity as you would any other drawing entity. That is, you can delete, move, rotate, and scale text.

You can edit text properties in the **Properties** panel.

The TEXTED system variable controls the text edit behavior Text entities:

- 0 = in-place editor
- 1 = pop up dialog
- 2 = in-place editor with repeated input

The MTEXTED system variable allows to use an external text editor for editing Mtext entities.

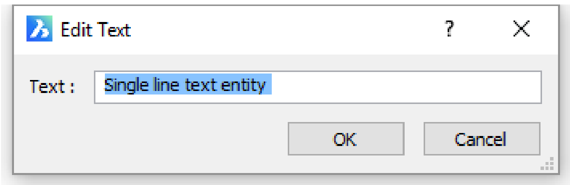
The MTEXTFIXED system variable controls whether the view is zoomed, rotated and/or panned to fit the edited Mtext.

13.2.4.1 To edit a Text entity

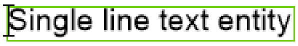
1 Double click the text entity.

2 Depending on the value of the TEXTED system variable:

3 The **Edit Text** dialog box appears



or the text is edited 'in place'.



4 Edit the text, then press Enter.

5 Do one of the following:
  - Click another text entity to edit.
  - Press Enter to stop editing text entities.

13.2.4.2 To edit an Mtext entity

1 Double click the Mtext entity.

2 The **Text formatting** toolbar displays.

3 Edit the text, then click the **OK** button or click outside the text block.

4 Do one of the following:
  - Click another Mtext entity to edit
  - Press Enter to stop editing Mtext entities.

13.2.4.3 To edit a selection of Mtext entities

1 Select the Mtext entities.

2 In the **Properties** panel select **Contents**.

3 Press the **Browse** button ( ) on the right hand side of the **Contents** field.

4 The **Text formatting** toolbar opens for the first Mtext entity in the selection.

5 Edit the Mtext.

6 Press the **OK** button or click outside the text block.

The next Mtext entity in the selection can be edited now.

7 Repeat steps 5 and 6 until the last Mtext is edited.

13.2.4.4 To select an external text editor for Mtext entities

1 Launch the SETTINGS command to open the **Settings** dialog box.

2 Under **Drawing** > **Drafting** > **Entity Creation** > **Texts** > **Multiline Text**, select the MTEXTED system variable.

3 Type the file name and full path of the external text editor in the Command line, then press Enter.

4 e.g. C:\Program Files\Windows NT\Accessories\Wordpad.EXE

Note: To restore the built-in text editor, delete the content of the MTEXTED system variable.

13.2.5 Exploding text

The TXTEXP command explodes Text and Mtext entities into 2D polylines.

13.2.5.1 To explode text

1 Select the text(s)

2 Do one of the following:

- Choose **Explode** in the **Edit** tab of the **Quad**.
- Launch the TXTEXP command.

Note:

- The solid fill of certain TTF fonts is removed when exploding such texts, leaving the text outlines only.
- Use the JOIN command to create closed polylines from the exploded text outlines.
- The **Explode** tool in the **Edit** tab of the **Quad** launches the TXTEXP command. The EXPLODE command cannot explode text entities.

13.2.6 Finding and replacing text

The FIND command lets you specify the text you want to find. You can than find or replace the text. Optionally you can zoom in on the various instances of the text entities that contain the search text string.

13.2.6.1 To set the Find and Replace options

1 Launch the FIND command.

1 The **Find and Replace** dialog box appears.

2 Click the **Options** button.

3 Tick the options on the **Find and Replace Options** dialog box.

4 Click the **OK** button.

13.2.6.2 To find text

1 Launch the FIND command.

1 The **Find and Replace** dialog box appears.

2 (option) Set the **Find and Replace** options.

3 (option) Click the **Select Entities** button ( ) to compose a selection set.

If no entities are selected, the entire drawing is searched.

- The **Find and Replace** dialog temporarily closes to let you select entities.
- Right click to conclude the selection of entities.

The **Search in** field reads **Current Selection** instead of **Entire Drawing**.

4 Type the search text in the **Find Text String** field.

5 Click the **Find** button.

6 The content of the first text entity that contains the search string is displayed in the **Search results** panel.

7 (option) Click the **Zoom To** button.

The drawing is zoomed in on the text entity.

8 Do one of the following:

9 Repeat steps 5 and 6 to find more instances of the same search string.

10 Repeat steps 2 through 6 to find another text string.

11 Press the Esc key or click the **Close** button to stop.

13.2.6.3 To replace text

1 Launch the FIND command.

1 The **Find and Replace** dialog box appears.

2 (option) Set the **Find and Replace** options.

3 (option) Click the **Select Entities** button ( ) to compose a selection set.

If no entities are selected, the entire drawing is searched.

- The **Find and Replace** dialog temporarily closes to let you select entities.
- Right click to conclude the selection of entities.

The **Search in** field reads **Current Selection** instead of **Entire Drawing**.

4 Type the search text in the **Find Text String** field.

5 Type the replacement text in the **Replace with** field, then press Enter.

All text entities that contain the search string are listed.

6 Do one of the following:

- Click an item in the list to highlight the text entity in the drawing, then click the **Replace** button.
- Click the **Replace All** button.

13.2.7 Checking spelling

The spelling checker checks the spelling of the text in the current drawing. You can check the spelling of one or more selected text entities or check the entire drawing.

During the spelling check, the SPELL command matches the words in the drawing or the current selection set to the words in the current main dictionary and the current custom dictionary. If a word is not found in either dictionaries, you can select the correct spelling in the suggestions field or you can add the word to the current custom dictionary. Custom dictionaries can be used for discipline-specific words.

13.2.7.1 Dictionary settings

It is defined by the DCTMAIN (main dictionary) and DCTCUST (custom dictionary) system variables which dictionaries are used by the SPELL command

Note: Only the English dictionary are delivered because of the license type on the dictionary files: the English version can be distributed freely, while all other languages are not free. However, you can download other languages and use them for free (see the part **Adding a main dictionary** below).

13.2.7.2 Opening the Spell Checking dialog box

Do one of the following:

- Choose **Check Spelling** in the **Tools** menu.
- Click the **Check Spelling** tool in the **Annotate / Text** ribbon panel.
- Launch the SPELL command.

13.2.7.3 Checking the spelling

1 Open the **Spell Checking** dialog box.

2 (option) Click the **Select** button ( ) to compose a selection set.

If no entities are selected, the entire drawing is searched.

- The **Spell Checking** dialog box temporarily closes to let you select entities.
- Right click to conclude the selection of entities.

The **Where to check** field reads **Current Selection** instead of **Entire Drawing**.

3 (option) Change the current dictionaries (see below).

4 Click the **Start** button.

The first possibly misspelled word is displayed in the **Current word** section.

5 The content of the text entity in which the **Current word** is found, is shown in the **Context** field.

6 Do one of the following:

- Click the **Ignore** button to leave this instance of the **Current word** unchanged.
- Click the **Ignore All** button to leave all instances of the **Current word** unchanged.
- Accept the word in the **Suggestions** field or select a word in the **Suggestions** list, then click the **Change** button to replace this instance of the **Current word** by the word in the **Suggestions** field.
- Accept the word in the **Suggestions** field or select a word in the **Suggestions** list, then click the **Change All** button to replace all instances of the **Current word** by the word in the **Suggestions** field.
- Click the **Add** button to add the **Current word** to the current custom dictionary.
- Click the **Lookup** button to list words similar to the word in the **Suggestions** field.

7 Repeat the previous step until the **Spelling check is complete** message appears.

13.2.7.4 Changing dictionaries

1 Open the **Spell Checking** dialog box.

2 Click the **Change Dictionaries...** button.

The **Change Dictionaries** dialog box appears.

3 Click the **Main dictionary** list button to change the main dictionary.

4 Select a new main dictionary.

5 (option) Click the **Browse...** button to select a new **Custom dictionary**.

6 (option) Type a word in the **Custom dictionary words** field, then click the **Add** button to add a word to the current custom dictionary.

7 Click the **Apply & Close** button.

Note:

- Custom dictionaries are best saved in the **Support** folder of the **Roamable** root folder.
- If you want to share the custom dictionary with MS Word, point to the MS Word custom dictionary in step 5 in the above procedure. By default, CUSTOM.DIC is the custom dictionary in MS Word, which can be found under *C:\Users\%username%\AppData\Roaming\Microsoft\Proof\CUSTOM.DIC*.

13.2.7.5 Adding a new main dictionary

1 Open the **Spell Checking** dialog box.

2 Click the **Change Dictionaries...** button.

3 Click the **Download...** button.

Your browser opens: https://extensions.openoffice.org/dictionaries

4 Scroll down to the language of your choice, then click the link to open the page of the selected language.

5 Click the **Download extension** button to start downloading the ***.oxt** file.

6 Change the **oxt** extension to **zip**.

7 Extract the ***.aff** and ***.dic** files from the ZIP file to the folder *C:\Program Files\Bricsys\BricsCAD Vx \Support*.

The dictionary is now available in the **Main Dictionary** list in the **Change Dictionaries** dialog box.

13.2.7.6 Adding a new custom dictionary

1 Open a text editor, such as **Notepad**.

2 (option) Type the words you want to use in your custom dictionary.

3 Each word must be on a separate line.

4 Save the file in the **Support** folder of the **Roamable** root folder using the **.cus** extension.

13.2.8 Working with fields

A field is a text entity which contains instructions to display the status or property of another entity or setting in the drawing. When a field is updated it displays the latest value of the data source it refers to. Fields can be used to insert drawing properties, custom properties or object properties in multiline texts or tables or as an attribute in a block. Fields in attributes are especially useful in title blocks: using drawing, sheet set, subset and sheet properties title blocks can be filled out automatically.

See the DWGPROPS command to learn more about default and custom drawing properties.

See the SHEETSET command to learn more about default and custom sheet set, subset and sheet properties.

If no value is available, a field displays hyphens (----).

If the FIELDDISPLAY system variable is ON, fields are displayed on a light gray background which does not print.

The FIELDEVAL system variable controls when fields are updated.

Note: Date fields are only updated by the UPDATEFIELD command; they are not automatically updated based on the FIELDEVAL system variable value.

13.2.8.1 General procedure to create fields

1 Do one of the following:

1
- Click the **Insert Field** tool button ( ) on the **Insert** toolbar and the **Annotate / Text** ribbon panel.
- Choose **Insert Field...** in the **Insert** menu.
- Launch the FIELD command.
- Click the **Field** tool button ( ) on the **Text Formatting** toolbar when creating or editing Mtext or a table cell content.

The **Field** dialog box appears.

2 Under **Field Names**, expand a group, then choose a field name.

The settings for the selected field name appear.

3 Adjust the settings, then click the **OK** button.

The **Field** dialog box closes.

The field is created in the Mtext or table cell being edited.

or

You are prompted: Specify start point or [Height/Justify]:

Do the following:

- (option) Type **H** or choose **Height** in the context menu to define the text height.
- (option) Type **J** or choose **Justify** in the context menu to set the text justification.
- Specify the insertion point of the field in the drawing.

An Mtext entity containing the field is created.

13.2.8.2 Working with Field Formula

Field Formula refers to functions and operations that can be used in the **Field** dialog box.

1 Launch the FIELD command to open the **Field** dialog box.

2 Expand **Objects** group in the **Field names** list and select **Formula**.

3 In the **Formula** field you can insert different types of operations and formulas. The result will be shown in the **Field value** field.

The following table shows the operations that can be used in the Field Formula.

Operation	Character	Description
Addition	+	Produces the sum of numeric values.
Subtraction	-	Subtracts the two numeric values.
Multiplication	*	Multiplies the numeric values.
Division	/	Divides two numeric values.
Exponentiation	^	Calculates the exponential value of the given numbers.

The following table shows the functions that can be used in the Field Formula.

Function	Syntax
Cosine	cos(expression)
Sine	sin(expression)
Tangent	tang(expression)
Arc cosine	acos(expression)
Arc sine	asin(expression)
Arc tangent	atan(expression)
Square	sqr(expression)
Square root	sqrt(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Absolute value	abs(expression)
Degrees to radians	d2r(expression)

Function	Syntax
Radians to degrees	r2d(expression)
Logarithm, base e	ln(expression)
Logarithm, base 10	log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Pi = 3.14...	pi

Note: The syntax for **Tangent** function in Field Formula is slightly different from the one in Parameters:

- Field Formula: tang(expression)
- Parameters: tan(expression)

Note: Square of a number, sqr(expression), is available only in Field Formula.

13.2.8.3 To display an entity property in a field

1 Launch the FIELD command.

1 The **Field** dialog box appears.

2 Expand **Objects** in the **Field names** list and select **Object**.

3 Click the **Select Entity** button () on the **Field** dialog box.

The **Field** dialog box closes temporarily.

You are prompted: Select Entity:

4 Click the entity you want to display a property of.

5 The **Field** dialog box appears again.

6 Select the entity property in the **Property** list.

7 Set the display layout:

- **Format**: choose a unit format.
- **Precision**: select a precision in the list button
- **Convert**: this option allows to recalculate the field value.
- Choose a formula in the list; e.g. f*#, where f is the factor you type in the **Convert** field and # is the field value.
- Type a factor in the **Convert** field; e.g. 0.0001 to express the area of a polyline in square meters in a cm drawing.

8 Click the **OK** button.

9 Specify a point to place the field in the drawing.

13.2.8.4 To edit a field

1 Edit the table cell or Mtext entity that contains the field.

2 Double click the field.

3 The **Field** dialog box appears.

Note: When you edit a field associated with a parameter that no longer exists, an error message is displayed in the dialog box.

4 Adjust the properties of the field.

5 Click the **OK** button to apply the changes.

13.2.8.5 To update a field

1 Do one of the following:

- Click the **Update Field** tool button () on the **Redraw / Regen** toolbar.
- Choose **Update Fields...** in the **Tools** menu.
- Launch the UPDATEFIELD command.

You are prompted: Select field(s) to update:

2 Select the field(s) you want to update, then right click or press Enter.

3 The selected fields are updated.

Note: If bitcode 16 of the FIELDEVAL system variable is set, executing the REGEN command updates all fields in the drawing.

Note: Date fields are only updated by the UPDATEFIELD command; they are not automatically updated based on the FIELDEVAL system variable value.

13.2.8.6 To convert a field to text

1 Edit the table cell or Mtext entity that contains the field.

2 Select the field.

3 Right click, then choose **Convert Field to Text** in the context menu.

13.2.9 Working with tables

A table is an entity that is composed of rows and columns, much like a Microsoft Excel spreadsheet. A table entity can be created as an empty table or you can import data from a CSV (Comma Separated Values) file. The appearance of the table text and gridlines is controlled by a table style. Table styles are saved in the DWG file.

A default table consists of:

- a **Title** cell (A)
- a row of **Header** cells (B)
- one or more rows of **Data** cells (C).

A table can have one of two directions:

- **Down** (title row at the top)
- **Up** (title row at the bottom).

A				
B	B	B	B	B
c	c	c	c	c
c	c	c	c	c
c	c	c	c	c

c	c	c	c	c
c	c	c	c	c
c	c	c	c	c
B	B	B	B	B
A				

13.2.9.1 To create an empty table

1. Do one of the following:
   - Click the **Table** tool button ( ) on the **Draw** toolbar or the **Home / Annotations** ribbon panel.
   - Choose **Table...** in the **Draw** menu.
   - Launch the TABLE command.

   The **Insert Table** dialog box appears.
2. Choose a Table Style from the **Table Style** list.
3. (Optional) Set the cell style for the first, second and other rows.
4. By default the cell style for the first row is **Title**, the cell style for the second row is **Header** and the cell style for the other rows is **Data**.
5. Under **Table Options**, choose **Start from Empty Table**.
6. Under **Insertion Behavior**, do one of the following:
   - Choose **Specify Insertion Point**, then specify the number of columns and the column width and number of data rows and the row height.
   - Choose **Specify Window**, then do one of the following:
   - Specify the number of columns and data rows.
   - Specify the column width and the row height.
   - Specify the column width and the number of data rows.
   - Specify the number of columns and the row height.
7. Click the **OK** button.

   Depending on your choice in step 5, **you are prompted** to specify an insertion point or a window.

   The table is created and the **Text Formatting** toolbar appears (see Creating Texts).
8. Do one of the following:
   - Start adding text in the table.
   - Click outside the table to close the **Text Formatting** toolbar.

13.2.9.2 To create a table from data

When importing data from a CSV (Comma Separated Values) file, please make sure the list separator character on your system is the same as the one used in the CSV file (see Defining the list separator character).

1 Do one of the following:

- Click the **Table** tool button ( ) on the **Draw** toolbar or the **Home / Annotations** ribbon panel.
- Choose **Insert Table...** in the **Draw** menu.
- Launch the TABLE command.

The **Insert Table** dialog box appears.

2 Under **Table Options**, choose **From Data**.

3 Click the **Browse** button ( ).

4 The **Open** dialog box appears.

Choose a CSV file, then click the **Open** button.

5 Click the **OK** button on the **Insert Table** dialog.

6 Click a point in the drawing to place the table.

7 The table is created and the **Text Formatting** toolbar appears (see Creating Texts).

8 Do one of the following:

- Start editing the text in the table.
- Click outside the table to close the **Text Formatting** toolbar.

9 (Optional) Select the table and drag the grips to adjust the dimensions of the table.

13.2.9.3 To create a table from an Excel spreadsheet

1 Open the spreadsheet in **Microsoft Office Excel**.

2 Select the cells you want to create a table from.

3 Do one of the following to copy the selection to the clipboard:

- Press **Ctrl + C**.
- Right click and choose **Copy** in the context menu.

4 In BricsCAD, choose **Paste special...** in the **Edit** menu or launch the PASTESPEC command.

5 The **Paste Special** dialog box appears.

6 Choose **XML Spreadsheet** in the **As** option list.

7 Click in the drawing to insert the table.

13.2.9.4 Editing a table using grips

When a table is selected, multiple grips are shown:

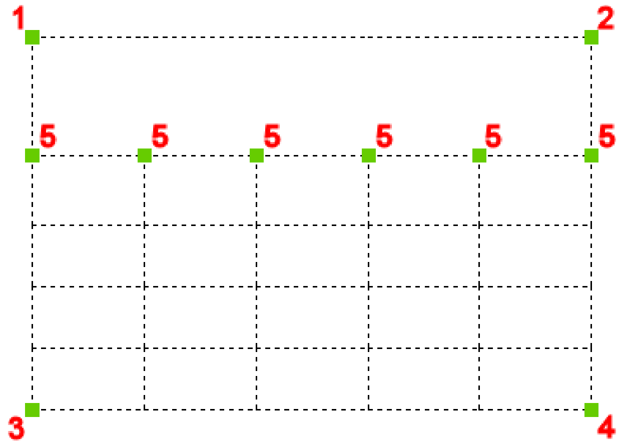
1 Moves the table.

2 Modifies the width uniformly.

3 Modifies the height uniformly.

4 Modifies the width and height uniformly.

5 Modifies the width of a column.



13.2.9.5 Editing a table using the Table toolbar

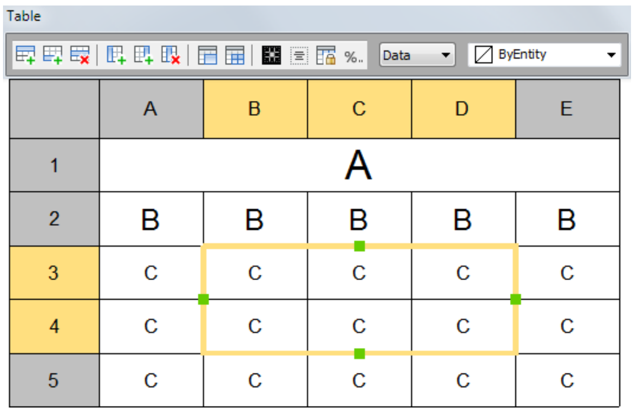
1 To open the **Table** toolbar, do one of the following:

- Click the cell you want to edit.
- Press and hold the left mouse button then define a rectangle to select multiple cells.

2 Column letters and row numbers are displayed.

3 The column(s) and row(s) of the selection are highlighted.

A rectangle with four handles is drawn around the selection.



4 (Optional) Drag the grips of the selection to modify the dimensions.

5 (Optional) Click a tool in the **Table** toolbar.

6 (Optional) Right click to open a context menu:

- **Delete Cell Content**: deletes the text in the selected cells.
- **Clear Cell Overrides**: resets all cell properties of the selected cells to the **Cell Style** settings as defined in the **Table Style**.
- **Clear Table Overrides**: resets all cell properties of all cells to the **Cell Style** settings as defined

in the **Table Style**.

7 (Optional) Repeat steps 3 and 4 to continue editing the table.

8 (Optional) Double click a cell to edit the cell text.

9 (Optional) Do one of the following:

- Click another cell.
- Press and hold the left mouse button, then drag the cursor to select multiple cells.

10 Click outside the table to close the **Table** toolbar.

Table tools

Icon	Tool name	Description
	Insert row above	Inserts an empty row above the selection.
	Insert row below	Inserts an empty row below the selection.
	Delete row	Deletes the row(s) of the selection.
	Insert column left	Inserts an empty column at the left of the selection.
	Insert column right	Inserts an empty column at the right of the selection.
	Delete column	Deletes the column(s) of the selection.
	Merge	Merges the selected cells.
	Unmerge	Splits previously merged cells.
	Border style	Sets the border style of the selection (not implemented yet).
	Text alignment	Sets the text alignment of the selection.
	Lock / Unlock	Locks / Unlocks the selection.
%..	Set cell format	Sets the cell format of the selection (not implemented yet).
Title	Set cell style	Sets the style of the selected cells: **Title**, **Header** or **Data**.
ByEntity	Set cell background	Sets the background color of the selection.

13.2.9.6 Editing a table in the Properties panel

1 Click one of the grid lines.

1 The properties of the table are displayed in the **Properties** panel.

2 (Optional) Apply a different **Table Style**.

3 (Optional) Change the table direction.

4 (Optional) Modify the width and/or height of the table.

5 (Optional) Modify the vertical and/or horizontal cell margins.

6 Press the Esc key to stop editing.

13.2.9.7 Editing a cell in the Properties panel

1 Click the cell.

1 The properties of the cell are displayed in the **Properties** panel.

2 (Optional) Modify the width and/or height of the cell.

3 The width and height of the column and/or row of the cell are changed.

4 (Optional) Modify the text rotation.

5 The options are: 0°, 90°, 180° and 270°.

6 (Optional) Edit the cell content.

7 (Optional) Choose a different text alignment option.

8 Press the Esc key to stop editing.

13.2.9.8 Evaluating formulas

Currently formulas are supported partially. There are more complex formulas which are not supported yet, but in most cases formulas in Excel sheets will remain functional when the Excel sheet is imported as a table. Editing and creating formulas through the field dialog is not supported yet, but in-place editing/creating is possible. E.g.: if you type =SUM(A2:A4) in a cell, it will be interpreted as a formula, and the sum of the values in the cells A2 through A4 is calculated.

13.2.9.9 To edit the table content

1 Do one of the following:

- Double click the cell you want to edit.
- Launch the TABLEDIT command.
- **You are prompted**: Pick table cell to edit.

  Click the cell you want to edit.

The **Text Formatting** toolbar is displayed above the selected cell (see Creating Texts).

2 Edit the cell.

3 (Optional) Press Alt + Enter to create a second line in the same cell.

4 To edit another cell:

- Press the Tab key to go to the next cell in the same row.

- Press Shift + Tab to go to the previous cell in the same row.
- Press Enter to go the next cell in the same column.
- Press Shift + Enter to go to the previous cell in the same column.
- Press one of the arrow keys.

5 Click outside the table to stop editing.

13.2.9.10 To export a table

When exporting data to a CSV (Comma Separated Values) file, the list separator character of your system is used in the CSV file (see Setting the list separator character).

1 Launch the TABLEEXPORT command.

1 **You are prompted**: Select table to export:

2 Click the table you want to export the data of.

3 The **Export Data** dialog opens.

4 Select a folder to save the data in.

5 Type a name in the **File name** field.

6 Click the **Save** button.

The data in the selected table are saved as a CSV file.

13.2.9.11 Setting the list separator character

1 Open the Windows **Settings** dialog box.

2 Open the **Regions** settings category.

3 Click **Additional date, time and regional settings**.

4 Click **Change date, time or number formats**.

5 Click **Additional settings**.

6 Set the **List separator** character.

7 Click **OK** to confirm.

13.3 Annotative Entities

13.3.1 Annotative entities

Annotations are all entities in your drawing which are not a model or geometry, such as dimensions, texts, symbols, etc. A common problem with annotations is that they are often scale dependent. Some annotations must be scaled to the plot scale, or are meant to be shown at certain scales only.

Annotative styles and entities change the way in which they are displayed (size, location, and orientation) depending on the current scale factor and the viewport.

The following entities can have the annotative property:

- text and multiline text

- hatches
- blocks and attributes
- dimensions and tolerances
- leaders and multileaders

The following styles can have the annotative property:

- text styles (see the STYLE command)
- dimension styles (see the DIMSTYLE command)
- multileader styles (see the MLEADERSTYLE command)

Note:

- Entities that are created using an annotative style, get the current annotation scale assigned automatically.
- When an annotative style is applied to a non-annotative entity, the annotative property is not assigned automatically.

13.3.1.1 Commands

ANNORESET: Resets all scale representations of the selected annotative entities to the location of their current scale representation, thus synchronizing all scale representations.

ANNOUPDATE: Updates the selected annotative entities to match their current style definition.

OBJECTSCALE: Adds or removes supported scales for annotative entities through a dialog box.

-OBJECTSCALE: Adds or removes supported scales for annotative entities at the command line.

SCALELISTEDIT: Edits the scale list of a drawing, used in **Plot Scale** section of the **Print** and **Page Setup** dialogs and the **Standard scale** property of a paper space viewport. Defines the available annotation scales for annotative entities.

-SCALELISTEDIT: Edits the scale list of a drawing at the Command line.

13.3.1.2 System variables

ANNOALLVISIBLE: Toggles the display of annotative entities that do not support the current annotation scale.

ANNOTATIVEDWG: Specifies whether or not the drawing behaves as an annotative block when inserted into another drawing.

CANNOSCALE: Sets the name of the current annotation scale for the current space. Only scales that exist in the current scale list are accepted. The CANNOSCALE setting is saved with model space and with each layout viewport in the various layouts. When you create an annotative entity the current annotative scale is applied automatically.

CANNOSCALEVALUE: Displays the value of the current annotation scale. E.g. the value of the 1:2 scale is 0.5.

HPANNOTATIVE: Controls whether new hatch entities are annotative.

MSLTSCALE: If ON, linetypes in model space are scaled to the current annotation scale. A regen is needed when you toggle the value of MSLTSCALE. This variable is the equivalent of the PSLTSCALE (Paperspace linetype scale) variable in paper space.

MSOLESCALE: Controls the size of an OLE object containing text when pasted in model space. The MSOLESCALE system variable only affects the initial scaling. Existing OLE objects are not affected if the variable is modified.

SAVEFIDELITY: Controls whether the current display of the drawing is preserved when opened in a program that does not support annotative entities.

SELECTIONANNODISPLAY: Controls whether all scale representations are displayed when an annotative entity is selected.

13.3.1.3 Setting the current annotation scale

To set the current annotation scale, do one of the following:

- Click the **Annotation Scale** field under **Misc** in the **Properties** panel, then press the down-arrow button and select a scale in the list.
- Right click the **Annotation Scale** field in the **Status** bar, then select a scale in the list. If the **Annotation Scale** field is not available, click the down arrow at the right hand side of the **Status** bar and choose **Annotation Scale** in the list.

Note: Only scales that exist in the **Scale** list of the drawing are available. Use the SCALELISTEDIT command to edit the **Scale** list.

13.3.1.4 To make an annotation entity annotative

Annotation entities which are created using a non-annotative style can be made annotative using the following procedure:

1 Select the annotation entity.

2 In the **Properties** panel, click the **Annotative** property.

3 Select **Yes**.

The current annotation scale is applied to the selected entity.

13.3.2 Annotation scales

Annotation scales are used to calculate the scaling factor for annotative entities. It defines the height of annotative text entities or the overall scaling of annotative blocks. In general, the annotation scale allows you to create annotative entities in model space at the correct paper space size automatically.

- Texts, MTexts, Dimensions, MLeaders: The current annotation scale is applied upon creation if the current style is annotative.
- Hatches and Gradient Fills: The current annotation scale is applied if the HPANNOTATIVE system variable is ON.
- Blocks: The current annotation scale defines the scaling of annotative blocks upon insertion.

Annotative entities can support multiple annotation scales. When the SELECTIONPREVIEW system variable is active or when using the **Quad** cursor menu, an icon is displayed next to the highlighted entity, indicating whether a single annotation scale ( ) applies to an annotative entity or multiple ( ).

13.3.2.1 To add or remove annotation scales

The following procedure allows to:

- Check which annotation scales currently apply.
- Add annotation scales.
- Remove annotations scales.

1 Select the annotative entity or entities.

2 In the **Properties** panel, select the **Annotative Scale** field.

3 Click the **Browse** button ( ) of the **Annotative Scale** field.

The **Annotation Object Scale** dialog box appears.

4 (option) If multiple entities were selected, choose whether to list all scales for the selection, or only the common scales.

5 (option) Do one of the following:

- Select one or more scales in the list, then click the **Delete** button. The selected scale(s) are removed from the list.
- Click the **Add...** button, then select one or more scales in the list and click the **OK** button.

  The selected scale(s) are added to the list.

6 Click the **OK** button to close the **Annotation Object Scale** dialog box.

The changes are applied to the selection set.

Note: Only scales that exist in the **Scale** list of the drawing are available. Use the SCALELISTEDIT command to edit the **Scale** list.

13.3.3 Annotative texts

Text entities are created using the current text style:

- If the text style is annotative, the text entity gets the annotative property automatically and the current annotation scale is assigned.
- If the text style is not annotative, you need to make the text entity annotative manually.

Annotative texts have two height properties:

- **Model text height**: the display height on the screen in model space, with respect to the current annotation scale.
- **Paper text height**: the height on paper if printed on the supported annotation scale.

13.3.3.1 To make a text entity annotative

1 Select the text entity.

1 The properties of the selected text entity are displayed in the **Properties** panel.

2 In the **Properties** panel, select the **Annotative** property.

3 Choose **Yes** from the options list.

The current annotation scale is assigned.

The text is not scaled: the current text height becomes the **Model text height**.

13.3.3.2 To remove the annotative property

1 Select the text entity.

1 The properties of the selected text entity are displayed in the **Properties** panel.

2 In the **Properties** panel, select the **Annotative** property.

3 Choose **No** from the options list.

If the text entity supports multiple annotation scales, the text is scaled back to its initial scale.

13.3.4 Annotative hatches and gradient fills

The HPANNOTATIVE system variable controls whether new hatch or gradient entities are annotative.

HPANNOTATIVE is OFF by default. The system variable is not saved and is reset to its default value when BricsCAD closes.

13.3.4.1 To set the HPANNOTATIVE system variable

Do one of the following:

- In the **Settings** dialog box, under **Drafting** > **Entity Creation** > **Hatches**.
- Check the **Annotative** option on the **Hatch and Gradient** dialog box.
- Choose the **Annotative** option in the command prompt of the -HATCH command.

13.3.4.2 To make a hatch or gradient entity annotative

1 Select the hatch or gradient fill entity.

1 The properties of the selected entity are displayed in the **Properties** panel.

2 Select the **Annotative** property.

3 Choose **Yes** from the options list.

The current annotation scale is applied to the selected hatch or gradient entity, and the hatch is scaled accordingly.

13.3.4.3 To remove the annotative property

1 Select the hatch or gradient fill.

1 The properties of the selected entity are displayed in the **Properties** panel.

2 Select the **Annotative** property.

3 Choose **No** from the options list.

13.3.5 Annotative blocks

The annotative property of a block is either set when creating the block definition using the BLOCK command, or next, in the **Drawing Explorer / Blocks** dialog.

Annotative blocks can have an optional property to always match the orientation of a layout, independent from the rotation of the paper space viewport content.

13.3.5.1 To make a block definition annotative

1 Launch the EXPBLOCKS command.

1 The **Drawing Explorer / Blocks** dialog box appears.

2 If necessary, choose **Detail View**.

Annotative block definitions have the **Annotative** icon ( ) in the **Annotative** column.

3 Click in the **Annotative** field of a block definition to toggle the annotative property on/off.

13.3.5.2 To set the Match Orientation to Layout property

1 Launch the EXPBLOCKS command.

1 The **Drawing Explorer / Blocks** dialog box is displayed.

2 If necessary, choose **Detail View**.

3 Tick the **Match Orientation to Layout** checkbox of a block definition to set the property.

13.3.6 Annotative dimensions

Dimension entities are created using the current dimension style:

- If the dimension style is annotative, the dimensions get the annotative property automatically and the current annotation scale is assigned.
- If the dimension style is not annotative, you need to make the dimensions annotative manually.

Note:

- The **Dim Scale Overall** setting (DIMSCALE system variable) is not editable for annotative dimension styles, and is neglected for annotative dimensions.
- It is recommended to set **Dim Scale Overall** = 1 for annotative dimension styles.

13.3.6.1 To make a dimension entity annotative

1 Select the dimension entity.

1 The properties of the selected dimension entity are displayed in the **Properties** panel.

2 Select the **Annotative** property.

3 Choose **No** from in the options list.

The dimension entity is scaled with respect to the current value of the DIMSCALE system variable.

13.3.7 Annotative leaders

Multileaders are created using the current Multileader style:

- If the Multileader style is annotative, the Multileader entity gets the annotative property automatically and the current annotation scale is assigned.
- If the Multileader style is not annotative, you need to make the Multileader entity annotative manually.

13.3.7.1 To make a Multileader entity annotative

1 Select the **Multileader** entity.

The properties of the selected **Multileader** entity are displayed in the **Properties** panel.

2 Select the **Annotative** property.

3 Choose **Yes** in the options list.

The current annotation scale is assigned.

The **Multileader** entity is scaled with respect to the current annotation scale.

13.3.7.2 To remove the annotative property

1 Select the **Multileader** entity.

The properties of the selected **Multileader** entity are displayed in the **Properties** panel.

2 Select the **Annotative** property.

3 Choose **No** from the options list.

14. Customization

14.1 Startup options

When starting up BricsCAD you can add arguments or switches. For example:

- The Command line "BricsCAD.exe drawing1.dwg drawing2.dwg drawing3.dwg" starts BricsCAD and loads the specified drawings.
- The Command line "bricscad.exe /P MyProfile" starts BricsCAD using the MyProfile user profile.
- The Command line "bricscad.exe /pr LicenseLevelName" starts BricsCAD using a specified license level.

"/P MyProfile" is called a Command line switch. Both a slash and a hyphen are accepted to precede the switch, e.g. "-P MyProfile" will work. The space is optional, e.g. "/PMyProfile" will also work, and the switch character is case insensitive, e.g. "-p MyProfile" also works.

Switch	Description
/P [userProfileName]	Starting without the /P option, BricsCAD uses the current profile (= the most recently used profile) which is not necessarily the Default profile. If the user profile specified with the /P option doesn't exist at startup, a new empty user profile will be created with that name. If the /P argument is an .arg file, the .arg file is used to initialize the profile settings on the first run (See the section **Starting BricsCAD with a specific user profile** in the **User profiles** article).
/pr [LicenseLevelName]	Starting without the /pr option, BricsCAD uses the current license level. Valid strings are: "lite", "pro", "bim", "mechanical" and "ultimate". They are case insensitive. A valid /pr argument will set the RUNASLEVEL system variable for the current session. For example, "C:\Program Files\Bricsys\BricsCAD *V25* en_US\bricscad.exe" /pr lite launches BricsCAD in the Lite license level. ***Note:*** You can start BricsCAD Shape using /pr shape.
/B [scriptFileName.scr] (*)	Runs a script (*.scr) or lisp (*.lsp) at startup. Opens all the file types that can be dragged and dropped in the application main frame window. If you pass filenames without a parameter, they are opened as if they had been passed with /B. Therefore, bricscad.exe mylisp.lsp is identical to bricscad.exe /B mylisp.lsp, and bricscad.exe my.dwg is identical to bricscad.exe /B my.dwg.
/L	Suppresses the startup splash screen.

Switch	Description
/LD [application.arx/brx/drx/lsp/dll] (*)	Loads an application on startup and on every new or opened drawing depending on the file extension. If the file extension is .lsp/.des/.mnl, it is treated as a lisp file and loaded in every drawing. Other file types are loaded only once at startup.
/Safemode	Prevents all third party plugin code, enablers and user lisp files from loading in the current session. Starting in a clean environment can prevent potential crash causes. See the troubleshooting article **How to start BricsCAD in Safe Mode, without any applications?**.
/S [path]	Redefines the SRCHPATH system variable. e.g. BricsCAD.exe /S C:\BricsCAD\
/T [TemplateFileName]	Creates a new drawing based on the template drawing specified.
/regserver /unregserver (Windows-only)	Registers/unregisters BricsCAD's COM interfaces.
/Automation (Windows-only)	Starts BricsCAD Shapewithout displaying the main frame window (with hidden UI). This switch is intended for use by COM automation, but it can also be useful for batch processing tasks that don't use COM. Automation switch could be used together with other switches. Example of loading a script file: <BricsCAD .exe path> /automation /B <path>/scriptFileName.scr The loaded script must finally close the application when it has completed.

Note:

- Switch arguments containing spaces should be enclosed by double quotes, e.g. /P "My user profile".
- Switches can be combined, for example /P MyProfile /L /S C:\BricsCAD\ starts BricsCAD using the **MyProfile** user profile without the splash screen and sets the SRCHPATH system variable to *C:\BricsCAD*
- Switch parameters that require arguments must be followed by a valid argument to have any effect.
- (*) Switches /B and /LD enable all sorts of files to be loaded, some may be not expected. For example, .cui, .dsd, .txt, .pdf types are also processed.
- If GETSTARTED = ON when you launch BricsCAD, the **BricsCAD Launcher** dialog box appears, which lets you choose the workspace.

Besides these switches, BricsCAD supports file names as Command line arguments.

Supported file types are:

- DWG: to load a drawing
- DXF: to load a drawing
- SCR: to load a script
- LSP: to load a LISP program
- DLL: to load an ADS/SDS or DRX/ARX program
- SLD: to load a slide file
- MNU: to load a menu file
- CUI: to load a menu file

14.2 BricsCAD user files

In BricsCAD, user files are stored in specific folders:

- **PlotConfig**: plotter configuration files (*.pc3)
- **PlotStyles**: plot style table files (*.stb) and plot color table files (*.ctb)
- **Support**:
    - hatch pattern files (*.pat)
    - line style files (*.lin)
    - alias files (*.pgp)
    - unit files (*.unt)
    - custom user interface files (*.cui)
    - BIM library files (*.bsyslib) see BIMPROJECTINFO command.
    - Structure Tree Configuration files (*.cst) see STRUCTUREPANEL command.
    - **Library** panel configuration files (*.ccf) see LIBRARYPANELOPEN command.
- **Templates**: template drawings (*.dwt)

Platform	Folders	Subfolders
Windows	*C:\Users\%username%\AppData\Local \Bricsys \BricsCAD\V25x64\en_US*	\Templates
Windows	*C:\Users\%username%\AppData\Roaming \Bricsys\BricsCAD\V25x64\en_US*	\PlotConfig \PlotStyles \Support
Mac	*/Users/<UserName>/Library/Application Support/ Bricsys/BricsCAD/<Version>/en_US*	/PlotConfig /PlotStyles /Support /Templates

Platform	Folders	Subfolders
Linux	*/home/<UserName>/Bricsys/BricsCAD/<Version>/en_US*	/PlotConfig /PlotStyles /Support /Templates

14.2.1 The BricsCAD user file manager

When you update, the first time you launch BricsCAD after the update, the content of the **UserDataCache** subfolder of the BricsCAD installation folder is compared with the content of the **user file folders**. If you have customized files in these folders and if one or more of the corresponding files in the **UserDataCache** folder are more recent, the **BricsCAD user file manager** dialog box displays, asking you whether to keep or to overwrite your files with the new version.

Do the following:

1 For each file in the list, choose what to do:

- **Keep current**: keep your customizations.
- **Overwrite**: replace the file.

2 Click the **Accept** button.

14.3 Loading and unloading CUI files

Shape Lite Pro Mechanical BIM

When you install BricsCAD, the default CUI files are copied to the **Support** folder (see the **BricsCAD User Files** article).

Note: There are three standard CUI files installed by default:

- Modern
- Classic-ribbon
- Classic-toolbars

When you launch BricsCAD the first time, the **classic-ribbon.cui** file is set by default.

There are two procedures to load a CUI file:

- Load main CUI file: replaces the currently loaded main CUI file.
- Load partial CUI file: adds custom customizations to the interface.

When you launch BricsCAD after installing a major update, you have the option to copy the files and settings from the previous BricsCAD installation, preserving user changes in the CUI while adding the new CUI components.

14.3.1 Loading a main CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 In the **File** menu, choose **Load main CUI file...**.

The **Select main CUI file** dialog box displays the content of the **Support** folder.

3 Select a CUI file, then double-click or click the **Open** button.

The selected CUI file is loaded.

4 Close the **Customize** dialog box.

14.3.2 Loading a partial CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 In the **File** menu, choose **Load partial CUI file...**.

The **Choose a customization file** dialog box displays the content of the **Support** folder.

3 Select a CUI file, then double-click or click the **Open** button.

The content of the selected CUI file is appended: definitions which are different from the ones in the main CUI are used.

4 Close the **Customize** dialog box.

Note: If a CUIX file is selected in step 2 it will be converted to a CUI file on-the-fly. When a CUIX file contains bitmaps, they are copied into a .RESZ resource file, which are supported as an alternative to resource DLL (zip file containing tool bitmaps; works on all platforms). The following image formats are supported: BMP, JPG, PNG, ICO, GIF, TIFF, XBM, XPM.

14.3.3 In case of a CUI load error

In the case of a load error due to a corrupted CUI file, an error message is displayed allowing you to try restoring from the latest backup, or using a CUI file with the same name from the UserDataCache, if any.

14.3.4 Unloading a partial CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 Click the **Menus** tab.

3 Right-click the partial CUI name and choose **Unload <PartialCuiName>** in the context menu.

4 Close the **Customize** dialog box.

14.3.5 Managing customization groups

1 Launch the MENULOAD command.

1 The **Customization groups** dialog box opens.

2 (Optional) To load a partial CUI file, click the browse button (...) to select a partial CUI file, then click the **Load** button.

3 (Optional) To unload a partial CUI file, select the partial CUI file in the **Loaded Customization Groups** list, then click the **Unload** button.

4 Click the **Close** button to stop.

14.4 Creating a partial CUI file

1 Launch the CUSTOMIZE command.

2 Do one of the following:

- In the **File** menu, choose **Create new partial CUI file ...**
- Place the cursor on a Menu group, then right-click and choose **Create new partial CUI file ...** in the context menu.

The **Create a Customization File** dialog opens.

By default the new partial CUI file will be created in the **Support** folder.

3 (Option) Select a different folder to save the CUI file in.

4 Type a name for the new CUI file in the **File name** field.

5 Click the **Save** button.

The newly created **Partial CUI file** is loaded and a **Menu Group** of the same name is created in the **Menus**, **Toolbars**, **Ribbon**, **Keyboards**, **Mouse**, **Tablet** and **Quad** tab and in the **Available Tools** pane of the different tabs.

14.5 Managing toolboxes

A toolbox contains a set of tools, which can be added to menus, toolbars, ribbon panels, Quad tabs and keyboard or mouse shortcuts. Toolboxes can be created in the main CUI or in a partial CUI.

14.5.1 Creating a toolbox in a menu group

1 In the **Customize** dialog box, select a menu group in the **Available Tools** panel:

- BricsCAD (= default main CUI)
- <PartialCuiName> (= partial CUI)

2 Right-click and choose **Append toolbox** in the context menu.

The **Add Toolbox** dialog box appears.

3 Type a name for the new toolbox in the **Name** field, then click the **OK** button.

The toolbox is added in the **Available Tools** tree.

14.5.2 Creating a tool in a toolbox

1 In the **Customize** dialog box, select a toolbox in the **Available Tools** panel.

2 Right-click and choose **Append tool** in the context menu.

The **Add tool** dialog box appears.

3 **Title**: type a name for the new tool.

4 **Help**: type a help string.

5 **Command**: type a command or a command macro*.

6 **Image**: Click the **Browse** button (...).

The **Tool Image** dialog box appears.

7 Do one of the following:
   - Select **Builtin**, then choose one of the built-in icons.
   - Select **Bitmap file**, first choose to select one image file or a small and a large image file, then click the **Browse** button (...) to select the image file(s).

8 Click the **OK** button to create the tool.

* Syntax to create command macros:

- Start the macro with **^c^c**, the equivalent of pressing the Esc key twice, which first ends a running command if necessary.
- Type an underscore (_) in front of a command name if you want to use the macro in a non-English version of BRICSCAD.
- A period (.) in front of a command name ensures that the macro still works if this command is temporarily undefined.
- A minus sign (-) in front of a command launches the Command line version of a command, thus preventing the command dialog window to open.
- A semicolon (;) is the equivalent of pressing the Enter key, e.g. to accept a default prompt option.
- A backslash (\) temporarily pauses the macro to allow user input, e.g. specifying a point.

In the following macro: ^c^c_-layer;m;InteriorWalls;;_mline

- **^c^c** concludes a running command, if any.
- **layer;** launches the -LAYER command.
- **m;** selects the Make new current layer option.
- **InteriorWalls** makes the InteriorWalls layer current and creates it first if necessary.
- **;;_mline** concludes the -LAYER command and launches the MLINE command.

14.6 Creating toolbars

1 In the **Customize** dialog box, click the **Toolbars** tab.

2 Do one of the following:
   - Select a menu group, then right-click and choose **Append toolbar** in the context menu.
     The new toolbar will be added below the last existing toolbar.
   - Select an existing toolbar in a menu group, then right-click and choose **Insert toolbar** in the context menu.
     The new toolbar will be added above the selected toolbar.

   The **Add toolbar** dialog box appears.

3 Type a name in the **Title** field.

4 Click the **OK** button.

   The toolbar is created.

5 Define the properties of the toolbar:
   - **Title**: The toolbar name must include alphanumeric characters with no punctuation other than a dash (–) or an underscore (_). The **Title** along with the **Alias** enables the toolbar to be referenced programmatically.
   - **Position**: Defines the display position of the toolbar when the CUI file is loaded.

     The options are: **Floating**, **Top**, **Left**, **Bottom** or **Right**. The latter four refer to the window edge where the toolbar will be docked.
   - **Display**: Defines whether the toolbar appears or not when the CUI file is loaded.
   - **Rows**: Specifies the number of rows when its position is floating. Toolbars with controls cannot have multiple rows.
   - **Xval**: When the position is Floating, specifies the X coordinate in pixels, measured from the left edge of the screen to the left side of the toolbar.
   - **Yval**: When the position is Floating, specifies the Y coordinate in pixels, measured from the top edge of the screen to the top of the toolbar.

6 Add items to the toolbar.

14.6.1 To add a flyout to a toolbar

1 In the **Customize** dialog box, click the **Toolbars** tab.

2 Do one of the following:
   - Select a toolbar, then right-click and choose **Append Flyout** in the context menu, to add the flyout at the end of the toolbar.
   - Expand the toolbar, right-click a tool then choose **Insert flyout** in the context menu, to add the flyout above the selected tool.

3 The **Add flyout** dialog box appears.

4 Type a name in the **Title** field of the **Add flyout** dialog.

5 Click the **OK** button.

   A new empty toolbar is created and inserted as a flyout in the selected toolbar.

6 Define the flyout properties of the toolbar.
   - **Title**: The toolbar name must include alphanumeric characters with no punctuation other than a dash (–) or an underscore (_). The **Title** along with the **Alias** enables the toolbar to be referenced programmatically.
   - **Target alias**: Identifies the referenced toolbar.
   - **Own Icon**:
   - **No**: The icon of the first tool or the icon of the most recently used tool is displayed in the parent toolbar.
   - **Yes**: The selected icon is displayed in the parent toolbar.

7 Define the properties of the toolbar when the flyout is teared off and inserted as a floating toolbar.

8 Add items to the toolbar.

9 Items added to the flyout are automatically added to the toolbar and vice versa.

14.7 Creating menus

Main menus sit in the menu bar at the top of the BricsCAD application window. Submenus show as a flyout of a main menu.

The MENUBAR system variable controls the display of menus.

14.7.1 Creating a main menu

1 In the **Customize** dialog, click the **Menus** tab.

2 Do one of the following:

- Click the **Main Menus** subgroup of a menu group, then right-click and choose **Append main menu** in the context menu.

  The new menu will be added below the last existing menu.

- Select an existing menu in the **Main Menus** subgroup, then right-click and choose **Insert main menu** in the context menu.

  The new menu will be added above the selected menu.

The **Add Main Menu** dialog box appears.

3 Type a name in the **Title**.

4 Click the **OK** button.

The main menu is created.

5 Add items to the menu.

Note: The new main menu is not visible in the menu bar until at least one menu item is added to it.

14.7.2 Adding a submenu to a main menu

1 In the **Customize** dialog, click the **Menus** tab.

If collapsed, expand the menu group.

If collapsed, expand the **Main menus** group of the menu group.

2 Do one of the following:

- Select a main menu, then right-click and choose **Append submenu** in the context menu.

  The new submenu will be added below the last existing submenu.

- Select an existing submenu, then right-click and choose **Insert submenu** in the context menu.

  The new submenu will be added above the selected submenu.

The **Add submenu** dialog box appears.

3 Type a name in the **Title** field.

4 Click the **OK** button.

The submenu is created.

5 Add items to the submenu.

14.7.3 Adding a submenu to a submenu

1 In the **Customize** dialog, click the **Menus** tab.

2 Expand the submenu.

3 Select an item in the submenu, then right-click and choose **Insert submenu** in the context menu.

The **Add submenu** dialog box appears.

4 Type a name in the **Title** field.

5 Click the **OK** button.

The submenu is created.

6 Add items to the submenu.

14.8 Creating context menus

The availability of context aka shortcut menus is controlled through the SHORTCUTMENU system variable. There are three context menu modes:

- **Default mode**: Displays the **Default** context menu when no command is active and nothing is selected.
- **Edit mode**: Displays the **Edit** context menu when a selection set exists. This mode is context sensitive. If a specific shortcut menu for the selected entity or entities exists, its content is added to the **Edit** context menu. For example, the content of the **Multiline Text Object** context menu is available in the **Edit** context menu when an MText entity is selected.
- **Command mode**: Displays the **Command** context menu when a command is running.

Shortcut menus appear when you right-click. However:

- When bitcode 16 of the SHORTCUTMENU system variable is set, a right-click repeats the previous command and you need to hold down the right mouse button longer (= long right-click) to display shortcut menus.
- When bitcode 4 of the QUADDISPLAY system variable is set (= Show Quad on right-click), shortcut menus only show when bitcode 16 of the SHORTCUTMENU system variable is set.

Special shortcut menus are:

- **Grips**: Appears on a right-click when using grips to edit an entity.
- **Entity Snap**:
  - MBUTTONPAN = OFF: appears when you click the mouse wheel (middle mouse button).
  - MBUTTONPAN = ON: displays on Shift + Right-Click

14.8.1 Creating shortcut menus

1 In the **Customize** dialog, click the **Menus** tab.

2 Do one of the following:

- Click the **Context Menus** subgroup of a menu group, then right-click and choose **Append context menu** in the context menu.

  The new context menu will be added below the last existing context menu.
- Select an existing context menu in the **Context Menus** subgroup, then right-click and choose

Insert context menu in the context menu.

The new context menu will be added above the selected context menu.

The **Add Context Menu** dialog box appears.

3 Type a name in the **Menu title** field.

4 (option) To create a regular context menu:

- Choose **As regular context menu**.
- Click the **Add Context...** button.

  The **Add context alias** dialog box appears.
- Select a context alias. The options are: CMCOMMAND, CMDEFAULT, CMEDIT, GRIPS and SNAP.
- This controls when the context menu is available. See above.
- (option) add another context alias.
- Click the **OK** button to create the context menu.

5 (option) To create a context sensitive context menu:

- Choose **As context menu on specific entities**.
- Click the **Add Entity Types...** button.

  The **Add Entity Alias** dialog box appears.
- Tick the **One entity** and/or **Multiple entities** boxes.
- Select one or more entities in the list, then click the **OK** button.
- (option) Click the **Add Custom...** button add a custom alias.
- Click the **OK** button to create the context menu.

6 Add tools to the context menu.

14.9 Creating keyboard shortcuts

1 In the **Customize** dialog box, click the **Keyboard** tab.

2 Do one of the following:

- Select a menu group, then right-click and choose **Append shortcut** in the context menu.

  The new shortcut will be added below the last shortcut.
- Select an existing shortcut in a menu group, then right-click and choose **Insert shortcut** in the context menu.

  The new shortcut will be added above the selected shortcut.

The **Add keyboard shortcut** dialog box appears.

3 Do one of the following:

- Select **Add an existing tool**, then select an existing tool.
- Select **Add a new tool**, then create a new tool.

  See the **Create a tool in a toolbox** procedure in **Managing Toolboxes**.

4 In the **Properties** grid of the new shortcut, select the **Key** field, then press the key combination you want to assign to the shortcut.

14.10 Customizing the Quad

To open the **Quad** tab of the **Customize** dialog box, do one of the following:

- Right-click the **QUAD** field in the **Status** bar and choose **Customize Quad items...** in the context menu.
- Run the CUSTOMIZE command, then select the **quad** tab.

14.10.1 Creating a quad tab

1 Do one of the following:

  - Right-click a menu group and choose **Append quad tab** in the context menu.

    The quad tab will be added below the last existing quad tab in the menu group.

  - Right-click an existing quad tab and choose **Insert quad tab** in the context menu.

    The quad tab will be added above the selected quad tab.

  The **Add Quad Tab** dialog box appears.

2 Type a name in the **Label** field.

  The label appears in the Quad cursor menu.

3 Type a name in the **Title** field.

  The tab title must be unique in the menu group.

4 Click the **OK** button.

5 Add tools to the tab.

14.10.2 Copying a quad tab

You can insert a copy of a quad tab in the same menu group or to another menu group.

- To insert a copy of a quad tab, select the tab, then right-click and choose **Insert Copy** in the context menu.

  The **Add Quad Tab** dialog box appears.

  Specify a label and title (see above), then click the **OK** button.

- To insert a copy of quad tab in another menu group: drag-and-drop the quad tab onto the target menu group.

14.10.3 Setting the Entity Filter property of a tool

The Quad cursor menu is context sensitive. The available tools depend on the entity under the cursor or the current selection set. When no selection exists, the **No Selection Quad** appears on a right-click if bitcode 4 of the QUADDISPLAY system variable is selected.

1 Select the tool.

1 The **Quad Button** properties are displayed in the bottom section of the **Customize** dialog box.

2 Select the **Entity Filter** property.

  The default setting is NO_SELECTION.

3 Click the **Browse** button (...) at the right hand side of the field.

The **Edit Quad Button Filter** dialog box appears.

4 (Optional) Click the **Add Filter...** button.

The **Add quad button filter** dialog box appears.

Click the down arrow at the right hand side of the field and choose a filter:

- NO_SELECTION: available when nothing is selected.
- OBJECT_ANY: available when a single entity is selected.
- OBJECTS_ANY: available when multiple entities are selected.

Click the **OK** button.

5 (Optional) Repeat the previous step to add a filter or remove a filter.

6 (Optional) Click the **Add Entity Type(s)...** button.

The **Add Entity Alias** dialog box appears.

- Tick the **One entity** and/or **Multiple entities** boxes.
- Tick the box the entity type(s) the tool applies to.

Click the **OK** button.

Note: It is not possible to modify the Entity Filter of a built-in Quad Button.

14.11 Creating Custom Quad Filters

The Quad cursor filter allows to customize the set of buttons that the Quad displays for a certain type of entity.

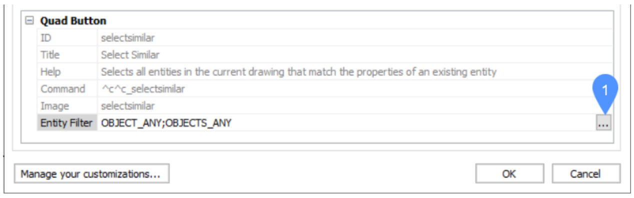
14.11.1 Creating custom filters for custom tools

The Quad displays certain buttons depending on the type of entity that is hovered with the mouse cursor.

The condition which specifies whether a certain Quad Button will appear in the quad is specified for each Quad Button individually in the **Entity Filter** property.

To access the **Entity Filter** property, follow the next steps:

- Open the **Customize** dialog box, **Quad** tab.
- Click the tool node in the left side of the dialog box.
- In the **Quad Button** frame, click the **Entity Filter** label.
- At the far-right side of the same row, click the button with three dots, marked with (1) in the image below. This will open the **Edit Quad Button Filter** dialog box. See details below.



Note: It is not possible to modify the **Entity Filter** of a built-in Quad Button. For built-in buttons the **Entity Filter** field is disabled.

14.11.2 Adding a new filter for a Quad button

To add a new filter to a Quad button, follow these steps:

- Open the **Customize** dialog box, **Quad** tab.
- Click the tool node in the left side of the dialog box.
- Open the **Edit Quad Button Filter** dialog box.
- Click **Add Entity Type(s)** button.
- Select whether the filter will affect one entity or many entities.
- Select the entity type or subentity type that will be associated to the filter. You can select multiple items.
- Click **OK**.
- Or click the **Add Filter** button to enter a custom filter or to select a filter from the available general filters (NO_SELECTION, OBJECT_ANY, OBJECTS_ANY).

  Note: Custom filter allows the user to manually type the DXF name of an entity that they want to be filtered, as well as a subentity specifier (face/edge/vertex). The full filter name consists of a general filter name, either "OBJECT_" or "OBJECTS_", followed by the DXF name (LINE, CIRCLE, etc) of the entity, e.g. OBJECT_CIRCLE, OBJECTS_SURFACE_EDGE.
- Click **OK**.

The selected items will appear in the list of the **Edit Quad Button Filter** dialog box.

Other operations that can be performed here:

1 To delete an item, click it and press **Remove**.

2 To add one or more items, press **Add Entity Type(s)** and select some more items from the list of entity types.

3 After the entity list in the **Edit Quad Button Filter** dialog box is ready, press **OK**. The filter will appear in the **Entity filter** field in the **Customize** dialog box, **Quad** tab.

4 To edit the filter, press the button with three dots at the right end of the **Entity filter** row and repeat the steps described above.

14.12 Adding tools to user interface controls

Tools can be drag-and-dropped from the toolboxes in the **Available Tools** section of the **Customize** dialog box onto menus, context menus, toolbars, rows in a ribbon panel, keyboard shortcuts and Quad tabs.

1 Select the target in the **Customize** dialog box.

2 In the **Available Tools** section, expand the source toolbox.

3 Select the tool and drag-and-drop it on the target item.

4 (option) Expand **Controls** in the **Available Tools** section, then drag-and-drop a control onto the target item.

5 (option) Drag-and-drop tools to reposition them or to move them to a different target item.

14.13 Customizing the ribbon

The ribbon is composed of a series of tabs, which in turn are composed of panels. Ribbon panels are composed of rows with tools, controls and sub-panels.

Ribbon tabs control which ribbon panels are displayed on the ribbon.

To display a ribbon tab on the ribbon, you must add the tab to a workspace.

14.13.1 Creating a ribbon tab

1 In the **Customize** dialog box, click the **Ribbon** tab.

2 Do one of the following:
   - Expand a menu group, then right-click **Ribbon tabs** and choose **Append ribbon tab** in the context menu.

     The new ribbon tab will be added below the last existing ribbon tab.
   - Select an existing ribbon tab in a menu group, then right-click and choose **Insert ribbon tab** in the context menu.

     The new ribbon tab will be added above the selected ribbon tab.

   The **Add ribbon tab** dialog box appears.

3 Define the ribbon tab:
   - **Label**: Name of the tab as displayed in the ribbon.
   - **Title**: Long name of the tab.
   - **ID**: Unique tag to identify the tab in the menu group. The ID is used to reference the ribbon tab programmatically.

4 Click the **OK** button to create the ribbon tab.

14.13.2 Creating a ribbon panel

1 In the **Customize** dialog box, click the **Ribbon** tab.

2 Do one of the following:
   - Expand a menu group, then right-click **Ribbon panels** and choose **Append ribbon panel** in the context menu.

     The new ribbon panel will be added below the last existing ribbon panel.
   - Select an existing ribbon panel in a menu group, then right-click and choose **Insert ribbon panel** in the context menu.

     The new ribbon panel will be added above the selected ribbon panel.

   The **Add ribbon panel** dialog box appears.

3 Define the ribbon panel:
   - **Label**: Name of the panel as displayed in the ribbon.
   - **Title**: Long name of the panel.
   - **ID**: Unique tag to identify the panel in the menu group. The ID is used to reference the ribbon panel programmatically.

4 Click the **OK** button to create the ribbon panel.

14.13.3 Adding a ribbon panel to a tab

1 In the **Customize** dialog box, click the **Ribbon** tab.

2 Expand **Ribbon tabs** in a menu group.

3 Do one of the following:

- Right-click a ribbon tab and choose **Append panel** in the context menu.

  The ribbon panel will be added below the last existing ribbon panel.

- Expand a ribbon tab, then right-click and choose **Insert panel** in the context menu.

  The ribbon panel will be added above the selected ribbon panel.

The **Select ribbon panel** dialog box appears.

All existing ribbon panels in the menu group are listed.

4 Select a panel in the list, then double-click or click the **OK** button.

The panel is added to the tab.

14.13.4 Adding a row to a panel

1 In the **Customize** dialog box, click the **Ribbon** tab.

2 Expand **Ribbon panels** in a menu group.

3 Select a panel, then do one of the following:

- Right-click and choose **Append row** in the context menu.

  The row will be added below the last existing row in the selected panel.

- Select an existing row then right-click and choose **Insert ribbon row** in the context menu.

  The row will be added above the selected row.

14.13.5 Adding items to a row

When you right-click a row, the following options are available in a context menu:

- **Add row panel**: Adds a subpanel containing three subpanel rows by default. You can then add one or more items to these rows and add/remove rows.
- **Add command button**: See **Adding tools to user interface controls**.
- **Add split button**: Adds a button to which you can add multiple items. Only the first item or the most recently used item is displayed.
- **Add toggle button**: Adds a button to toggle a setting or an option. E.g. entity snaps or selection priorities. A toggle button indicates the state of the setting or option.
- **Add Separator**: Adds a space between rows.

14.13.6 Defining the properties of an item

Select the item, then define the properties in the property grid in the bottom section of the **Customize** dialog box.

14.13.7 Repositioning items

To reposition any item in a panel or a row: select the item, then press and hold the left mouse button and move the item to the desired position. Release the mouse button to place the item.

14.13.8 Deleting items

Select the item, then right-click and choose **Delete** in the context menu.

14.14 Working with Tool palettes

On a tool palette you can add blocks, hatches and commands in a tabbed window.

Drag a tool from the palette to the graphic screen or click on it to launch the related tool.

14.14.1 Opening the Tool Palettes panel

Do one of the following:

- Hover the cursor over a toolbar or ribbon panel, then right-click and choose **Panels** > **Tool Palettes** in the context menu.
- Launch the TOOLPALETTES command to open the **Tool Palettes** panel.

14.14.2 Setting the view options

1. Open the context menu by clicking the three dots button from the upper right side of the **Tool Palettes** panel or by right clicking an empty space of the icon area on the **Tool Palettes** panel.
2. Choose **View Options...** in the context menu.

   The **View Options** dialog box appears.
3. Drag the **Image Size** slider to adjust the image size.
4. Set the viewstyle: **Icon with text**, **Icon only** or **List view**.
5. Choose either **Current Palette** or **All Palettes** from the **Apply to** list button.
6. Click the **OK** button to confirm.

Note: Also, it is possible to switch the view mode in the current palette by clicking the three buttons on the upper right side of the panel:

- **Icon with text**
- **Icon only**
- **List view**

14.14.3 Customizing tool palettes

Note: The default folder for tool palettes is defined by the TOOLPALETTEPATH system variable.

Opening the Customize dialog box

1. Open the context menu by clicking the three dots button from the upper right side of the **Tool Palettes** panel or by right clicking an empty space of the icon area on the **Tool Palettes** panel.
2. Choose **Customize Palettes...** in the context menu.

The **Customize** dialog box appears.

Importing a tool palette

1. In the **Palettes** panel, right-click and choose **Import** in the context menu.

   The **Import Palette** dialog box appears.

2. Select a folder, then select a palette file (*.atc, *.btc or *.xtp).
3. Click the **Open** button or double-click to import the palette.

   The palette is added to the **Palettes** list.

4. Click the **Close** button to close the **Customize** dialog box.

Creating a palette group

1. Place the cursor on the **Palette Groups** root folder name.
2. Right-click and choose **New Group** in the context menu.

   A new group folder is created.

3. Type a name for the new group.

Creating a palette subgroup

1. Place the cursor on an existing group name.
2. Right-click and choose **New Group** in the context menu.

   A new group folder is added as a subgroup of the selected group.

3. Type a name for the new group.

Adding a palette to a group

1. Place the cursor on a palette.
2. Select a palette, then drag the palette onto a palette group name.

Removing a palette from a group

1. Select the palette in the group.
2. Right-click and choose **Remove** in the context menu.

Setting a palette group current

1. Select the palette group in the palette groups tree.
2. Right-click and choose **Set Current** in the context menu.

Deleting a palette group

1. Select the palette group in the palette groups tree.
2. Right-click and choose **Delete** in the context menu.

14.14.4 Renaming a tool palette

Method 1:

1. Right-click tool palette tab and choose **Rename Palette** in the context menu.
2. Type a new name for the palette in the **Name** field.
3. Press Save to confirm.

Method 2:
Double click a tool palette tab and type a new name for the palette.

14.14.5 Deleting a tool palette

1 Open the context menu and choose **Delete Palette** in the context menu to delete the current palette.

2 Click the **Yes** button on the **Confirm Palette Deletion** box.

14.14.6 Managing tools

1 Right-click the tool.

1 A context menu appears.

Note: Rectangle selection is available to select multiple tools.

2 Choose an option:

- **Cut**: removes the item from the palette. Open another panel, then right-click and choose **Paste** from the context menu.
- **Copy**: copies the tool. Open another panel, then right-click and choose **Paste** in the context menu.
- **Delete**: deletes the tool. A dialog box appears, asking you to confirm the deletion.
- **Rename**: type a new name for the tool.
- **Specify Image...**: allows you to replace the default image.

  Choose a new image in the **Select Image File** dialog box, then double-click or click the **Open** button.
- **Remove Image...**: restores the default image.
- **Manage Flyout options...**: Opens the **Flyout Options** dialog box that allows you to manage tool flyouts. Tool flyouts via .xtp import are also supported.
- **Properties**: when you click an item on a tool palette, a command is launched. The properties of the tool palette item control the execution of the command.

  The available properties on the **Tool Properties** dialog box depend on the tool type: Command, Hatch, Block or Component.

  Edit the tool properties, then click the **OK** button.

14.14.7 Creating a new tool palette

1 Open the context menu.

2 Choose **New Palette** from the context menu.

A new tab is added.

3 Type a name for the new palette in the **Name** field.

14.14.8 Adding items to a tool palette

The following items can be added to a tool palette:

- Hatch patterns

- Blocks
- Components
- Commands

14.14.9 Adding hatch patterns

1 Select the palette you want to add a hatch pattern to in the **Tool Palettes** panel.

2 Open the **Hatch and Gradient** dialog box.

3 Select either **Predefined** or **Custom** from the **Type** list button.

4 Double-click the **Swatch** field on the **Hatch and Gradient** dialog box.

The **Hatch Pattern Palette** appears.

5 Select the hatch pattern you want to add, then right-click and choose **Add to Current Tool Palette**.

The hatch pattern is added on the selected tool palette.

6 (option) Repeat steps 4 and 5 to add more hatch patterns.

7 (option) Drag-and-drop patterns to change the order on the palette.

14.14.10 Adding blocks

Blocks can be added from:

- The current drawing.
- Any drawing.
- All drawings in a selected folder.

14.14.11 Adding blocks from the current drawing

1 Open the **Drawing Explorer** dialog box.

2 Select **Blocks**.

3 Select the block(s) you want to add. Hold down the Ctrl key to select multiple blocks.

4 Right-click and choose **Add to Current Tool Palette** from the context menu.

14.14.12 Adding blocks from any drawing

1 Open the **Drawing Explorer** dialog box.

2 Select **Folders** on the left hand pane.

3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.

4 Select the drawing in the **Local Folders** list.

5 Click the **Expand** icon (+) of the containing drawing and click the **Blocks** icon.

All blocks in the selected drawing are displayed in the **Details** pane.

6 Select the block(s) you want to add. Hold down the Ctrl key to select multiple blocks.

7 Right-click and choose **Add to Current Tool Palette** in the context menu.

14.14.13 Adding a drawing as a block

1. Open the **Drawing Explorer** dialog box.
2. Select **Folders** on the left hand pane.
3. If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.
4. Select the drawing in **Local Folders** list.
5. Right-click and choose one of the following in the context menu.
   - **Add to Current Tool Palette as block**.

     Blocks are inserted using the INSERT command.
   - **Add to Current Tool Palette as component**.

     Components are inserted using the BMINSERT command.

14.14.14 Adding all blocks from multiple drawings

This procedure collects all the blocks in all the drawings of the selected folder and adds them to a palette of the drawing name. New palettes are created if necessary.

1. Open the **Drawing Explorer** dialog box.
2. Select **Folders** on the left hand pane.
3. If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.
4. Select the folder, then right-click and choose **Add blocks to Tool Palettes** in the context menu.

14.14.15 Adding commands

1. In the **Tool Palettes** panel click the button and choose **Add Tool**.
2. In the **Available Tools** panel, on the **Customize** dialog box, expand a menu group (e.g. BricsCAD).

   All toolboxes in the selected menu group are listed.
3. Expand a toolbox.

   All commands in the selected toolbox are listed.
4. Right-click a tool and choose **Add to Current Tool Palette** in the context menu.

14.14.16 Adding a text

1. In the **Tool Palettes** panel click the button and choose **Add Text**.
2. Type a text in the opened box.
3. ***Note:*** The text can be moved using drag and drop, it can be cut, copied and pasted from the context menu, or it can be changed by clicking the text box and typing a different string.

14.14.17 Adding a separator

1. In the **Tool Palettes** panel click the button and choose **Add Separator**.

Note: The separator can be moved using drag and drop, or it can be cut, copied and pasted from the context menu.

14.14.18 Searching a tool

In the **Tool Palettes** panel it is possible to search for specific tools in the current palette.

1 Click the search button .
2 Start typing a name.
3 The list is shortened to display only the tools that contain the entered sequence of characters, regardless of where they are located in the tool's name.

14.15 Command aliases

Command aliases are alternative names for BricsCAD commands. You can launch the LINE command by entering L at the Command line. There is no need to type the complete name. After typing the command alias the command is executed. Some commands have more than one alias: e.g. REC, RECT and RECTANGLE are all aliases for the RECTANG command, whereas the last alias is even longer than the command name. In general, command aliases can increase your speed in entering the commands using the keyboard.

Command aliases are saved in a *.pgp file (P roG ram P arameters file). By default the **default.pgp** file in the **Support** folder of the Roamable Root folder is used.

If a command can be executed transparently (= while another command is running), the alias can also be used transparently. To execute a command or an alias transparently, precede the command name or alias with an apostrophe (').

Note:

- Unlike commands, aliases are only local and cannot be used as global commands preceded by a underscore (_).
- Command aliases cannot be used in scripts (*.scr).

14.15.1 Editing the alias file in a text editor

In order to modify existing aliases or to create new ones, the **default.pgp** file can be edited using an ASCII text editor (e.g. Notepad).

The first section of the **default.pgp** file defines the **Shell** commands. The second section of the **default.pgp** file defines command aliases.

The following syntax is used to define a command alias:

alias, *command

alias = the character string to be entered at the command prompt. The command alias must be followed by a comma (,).

command = the BricsCAD command. The command name must be preceded by an asterisk (*).

Comment lines must be preceded by a semicolon (;). Comment lines can be used to add additional information, e.g. when the file was last edited and by whom.

Some commands, such as HATCH or BOUNDARY, can be preceded by a hyphen (-) to access the Command line version of the command instead of the dialog box version. Command aliases can be created for these commands also, e.g.:

vi, *-view

Execute the REINIT command to reload the alias file when it was externally edited.

14.15.2 Using the Customize dialog to edit the alias file

1 Click the **Command Aliases** tab on the **Customize** dialog.

2 (option) To create a new alias,

- Do one of the following:
- • Click the **Add** button.
- • Right-click an alias and choose **Add Alias** in the context menu.
- • Right-click a command and choose **Add Alias** in the context menu to create an alias for the selected command.

  The **Add Alias** dialog box appears.
- Type the command alias in the **Alias** field.
- Click the down arrow at the right hand side of the **Command** field, then select a command. You can start typing the command name in the **Command** field to easily find the command in the list.
- Click the **OK** button.

3 (option) To edit an existing alias,

- Select the alias in the list.
- Click the **Edit** button, or right-click the alias and choose **Edit Alias** in the context menu.

  The **Edit Alias** dialog box appears.
- Type a new alias in the **Alias** field.

  If the alias already exists, you are prompted to overwrite the existing alias.
- Click the **OK** button.

4 (option) To delete an alias,

- Select the alias in the list.
- Click the **Delete** button, or right-click and choose **Delete** in the context menu.

  A **Confirm** dialog box appears.
- Click the **OK** button on the **Confirm** dialog box to delete the alias.

Note: If you use these procedures to edit the .pgp file, it is not necessary to execute the REINIT command.

14.16 User profiles

In BricsCAD, you can customize the settings that control your drawing environment, which can then be saved in a **user profile**. You can create multiple user profiles, e.g. a profile for each of your drawing types or workflows. User profiles can also be used when multiple users share the same computer, and each user wants to create their own drawing environment.

User profiles can be exported to a file, which can then be imported on a different computer. This allows you to copy your settings to a new computer or to install the same settings on multiple computers.

If you use third party applications on top of BricsCAD, you might want to create a user profile to use with each of these applications.

The **User Profile Manager** is a standalone tool that lists the existing profiles and provides all tools needed to **Load**, **Create**, **Set Current**, **Copy**, **Rename**, **Delete**, **Export** and **Import** user profiles.

14.16.1 Content of a user profile

User profiles save many settings that control the drawing environment. Once you start BricsCAD using a user profile, it automatically tracks and stores changes that you make to your drawing environment, such as the following.

- Main Cui file.
- Current Workspace. BricsCAD has six standard workspaces: Drafting, Drafting (toolbars), Modeling, Modeling (toolbars), Mechanical and BIM.
- Configuration settings: all system variables which are saved in the Registry and user preferences.
- Default plot settings.
- Settings in dialog windows, for example, search options in the **Settings** dialog box.
- Project settings.
- Recent paths.
- Status bar settings.
- Properties panel settings.
- Toolbar and menu configuration.
- Worksets.

14.16.2 The User Profile Manager

To open the **User Profile Manager** dialog box:

- When BricsCAD is running: launch the PROFILEMANAGER command.
- Execute **profilemanager_app.exe** in the BricsCAD program folder (e.g. *C:\Program Files\Bricsys\BricsCAD Vx*).

In the **User Profile Manager** dialog box:

- The **Current Profile** is marked.
- **Create**: Creates a new profile, using default settings.
- **Set Current**: Loads the selected profile.

- **Copy**: Creates a copy of the selected profile in the **Copy profile** dialog box.
- **Delete**: Deletes the selected profile.
- **Export...**: Exports the selected profile to a .arg file.
- **Import...**: Imports a **.arg** file as a new profile.
- **Start**: Starts BricsCAD with the selected user profile. This option is available only when the dialog box is opened using **profilemanager_app.exe** in the BricsCAD installation folder.
- **OK**: Closes the dialog box.

14.16.3 Starting BricsCAD with a specific user profile

When you restart BricsCAD, the user profile that was current when you closed the previous session is used.

To create a desktop shortcut to launch BricsCAD using a specific user profile, do one of the following:

- Add the **/P option** to the **Target** settings of the BricsCAD shortcut. To open the shortcut settings dialog: right-click the shortcut icon on the desktop and choose **Properties** in the context menu.

  A typical BricsCAD shortcut could be: **"C:\Program Files\Bricsys\BricsCAD Vx\bricscad.exe" /P UserProfileName**
- Execute **profilemanager_app.exe**, then either double-click a user profile or select a user profile and click the **Start** button.

Note:

- The **/P** option is case insensitive.
- A space is needed between the file location and /P option.
- When BricsCAD is already running, a second instance of BricsCAD is launched, at least on condition that the SINGLETON variable is OFF. This variable controls whether multiple BricsCAD sessions can run simultaneously. See the Settings dialog page for more information about variables and settings.
- If the user profile in the argument of the **/P** option does not exist, the user profile is created automatically using default hard-coded values for all settings.
- If the user profile name contains spaces, use double quotes: e.g. /P "background white".

14.17 Workspaces

The WORKSPACE command sets the current workspace and lets you create, modify and save workspaces.

The WSSAVE command saves the current workspace under a different name, thus creating a new workspace.

The WSSETTINGS command opens the **Customize** dialog box at the Workspaces tab.

Workspaces define settings and preferences regarding a specific working environment.

The following workspaces are available:

- Drafting
- Modeling
- Mechanical
- BIM

- Civil
- Complete

The name of the workspace is stored in the WSCURRENT system variable of the current User Profile.

If the GETSTARTED system variable is ON, you can select the workspace on the **BricsCAD Launcher** dialog box on startup.

The availability of a startup option depends your license level.

Startup Option	Workspace	License Level
2D Drafting	Drafting	Lite or higher
3D Modeling	Modeling	Pro or higher
Civil Survey	Civil	Pro or higher
Mechanical Design	Mechanical	Mechanical or Ultimate
BIM	BIM	BIM or Ultimate

14.17.1 Managing workspaces

Workspaces are defined in the **Workspace** tab page on the **Customize** dialog box.

1. In the **Customize** dialog box, click the **Workspaces** tab.
2. Expand a workspace.
3. A workspace definition is composed of:
   - **Menus**: defines which menus are available.
   - **Toolbars**: defines which toolbars are displayed by default.
   - **Panels**: controls the visibility of dockable panels.

     Right-click a panel, then choose an option in the context menu:
   - **Show**: the panel is visible.
   - **Hide**: the panel is not visible.
   - **Don't change**: keep the current state.
   - **Ribbon**: defines the available ribbon tabs.
   - **Quad Tabs**: defines the order and availability of the Quad tabs.
   - **On Switch**: defines the system variables and user preferences to be set, and the commands to be executed, when switching to the workspace.

14.17.2 Setting the current workspace

Do one of the following:

- Right-click the **Current Workspace** field in the **Status** bar, then select a workspace in the context menu.

- Choose a workspace in the **Workspaces** toolbar.
- Type **WSCURRENT** at the command prompt.

  You are prompted: New value for WSCURRENT <"2d DRAFTING">:

  Type the name of the new current workspace, then press Enter.

Note:

- If the WSAUTOSAVE system variable is ON, upon switching to another workspace, the toolbar positions of the current workspace are saved in the CUI file. The default value for WSAUTOSAVE is ON.
- It is not recommended to switch workspaces if the name of the user profile refers to a workspace.

14.17.3 Creating a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Do one of the following:

  - Select Workspaces, then right-click and choose **Append workspace** in the context menu.

    The new workspace will be added below the last existing workspace.
  - Select an existing workspace, then right-click and choose **Insert workspace** in the context menu.

    The new workspace will be added above the selected workspace.

  The **Add workspace** dialog box appears.

3 Type a name in the **Name** field then press Enter or click the **OK** button.

  A newly created workspace contains the following items:

  - A standard menu set
  - A standard toolbar set
  - All panels set to **Don't change**
  - An empty ribbon
  - A default set of Quad tabs
  - An empty **On Switch** sequence

  The newly created workspace has default properties. These properties are also available in the **Settings** dialog box, under **Program Options / Workspace**.

14.17.4 Editing a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Expand the workspace.

3 (option) Edit the menu set:

  - Change the order: Drag a menu.
  - Delete a menu: select the menu, then right-click and choose **Remove** in the context menu.
  - Insert a menu:
  - Right-click a menu in the list and choose **Insert menu** in the context menu.

The **File** menu is inserted by default.

- In the **Workspace Menu** grid, select the **ID** field and select the desired menu in the list.

4 (option) Edit the toolbar set:
  - Change the order: Drag a toolbar.
  - Delete a toolbar: select the toolbar, then right-click and choose **Remove** in the context menu.
  - Insert a toolbar:
  - Right-click a toolbar in the list, and choose **Insert toolbar** in the context menu.

    The **Standard** toolbar is inserted by default.
  - In the **Workspace Toolbar** grid, select the **ID** field and select the desired toolbar in the list.

5 Edit the properties of the panels:
  - Set the desired 'on switch' behavior: **Show**, **Hide** or **Don't change**.
  - Select a panel, then edit its properties in the property grid.

6 Edit the composition of the ribbon:
  - To add a tab: right-click an existing tab, then choose **Insert tab** in the context menu and select the tab in the **Select Ribbon Tab** dialog box.
  - To remove a tab: right-click the tab and choose **Remove** in the context menu.
  - To change the position of a tab: drag the tab.

7 Edit the Quad settings:
  - To change the order of the Quad command groups: press and hold the left mouse button, then drag a command group to its new position.
  - Right-click a Quad command group, then choose **Delete** in the context menu to remove the command group from the Quad cursor menu.
  - Right-click a Quad command group, then choose **Insert** and select from the list one of the tabs which you want to add to the Quad cursor menu.

8 Edit the **On Switch** procedure:

  To add a command to the **On Switch** sequence: select the command in the available commands pane and drag it to the **On Switch** node.

14.17.5 Deleting a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Right-click the workspace, then choose **Delete** in the context menu.

  A **Confirm** dialog box appears.

3 Press the **Yes** button to delete the selected workspace.

Note: Do not delete default workspaces, as they are used in the **BricsCAD Launcher** dialog box. When one of the default workspaces is deleted, use the **Importing a Workspace** procedure below to restore the missing workspace from Default.cui in the *.../UserDataCache\Support\en_US* subfolder of the BricsCAD installation folder.

14.17.6 Importing a workspace

The option in the **File** menu on the **Customize** dialog box can import workspaces from another CUI file into the current main CUI.

1. In the **Customize** dialog box, choose **Import workspaces…** in the **File** menu.

   The **Choose a Customization File** dialog box appears.
2. Browse to the folder where the CUI file, you want to import the workspace from, is saved.
3. By default, BricsCAD saves CUI files in the **Support** folder of the Roamable Root folder. See BricsCAD User Files.
4. Select the CUI file.
5. A dialog box appears, listing all workspaces in the selected CUI file.
6. Select the workspace(s) to import in the list, then click the **OK** button.

   Press and hold the **Ctrl** key to select multiple workspaces.

14.18 Digitizing tablet

A digitizing tablet, also called a graphics tablet, a graphics pad or drawing tablet, is a computer input device that allows you to hand-draw images and graphics, similar to the way one draws images with a pencil and paper. In a CAD program a digitizing tablet can be used to trace paper drawings into a drawing file or to launch drawing commands from a digitizing tablet overlay.

The TABLET command configures and calibrates a digitizing tablet and toggles the tablet mode.

This command requires that Wintab32.dll from Wacom Technology, Corp. be installed on the computer. Download Wintab32.dll from the internet, then place the file in your system directory: C:\Windows \System32.

14.18.1 Using a digitizing tablet in BricsCAD

In order to prepare BricsCAD for the use of a digitizing tablet, do the following:

- Install the appropriate driver for your tablet.
- Make sure Wintab32.dll is present in C:\Windows\System32.
- Download the tablet overlay and CUI files.
- Print the BricsCAD tablet overlay.
- Load the overlay CUI file.
- Start BricsCAD, then initialize the tablet.
- Configure the tablet: part of the tablet surface is used as the menu area, another part of it is used as the screen pointing area.
- Calibrate the tablet: a calibrated tablet can be used to trace the geometry of a paper drawing or image into a Bricscad drawing.

14.18.2 Downloading the tablet overlay and the CUI files

Click here to download the file Tablet.zip.

https://www.bricsys.com/bricscad/tools/Tablet.zip

The ZIP file contains the following files:

- **overlay.png**: an image of the tablet overlay
- **overlay(A3).pdf**: to print the tablet overlay on a A3 paper size.
- **overlay(cm).dwg**: a DWG in which the **overlay.png** is attached as an image.

  The drawing contains two layouts to print the overlay using decimal units.
- **overlay(inch).dwg**: a DWG in which the **overlay.png** is attached as an image.

  The drawing contains two layouts to print the overlay using imperial units.
- **tablet.cui**: a partial cui file to be used with a tablet.
- **tablet(acadLike).cui**: an AutoCAD®-like partial CUI file to be used with a tablet.

14.18.3 Loading the overlay CUI file

1. Extract the overlay CUI files to the **Support** folder of the **roamable root** folder (see BricsCAD User Files).
2. Load **Tablet.cui** as a partial CUI. The next time you start BricsCAD, **Tablet.cui** will be loaded automatically.

Note:

- Please make sure the **overlay.png** image file is extracted to the same folder as the DWG's.
- It may be necessary to repeat the above procedure after upgrading BricsCAD.

14.18.4 Switching the tablet ON / OFF

Do one of the following:

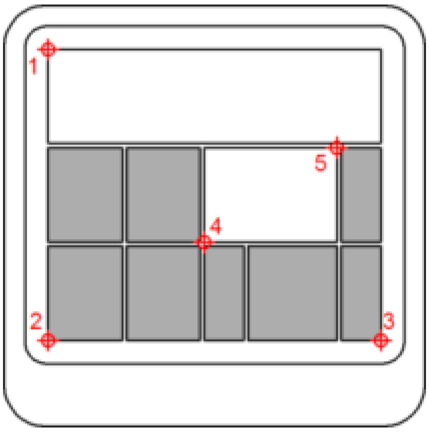
- Click the **TABLET** field in the Status bar.
- Press the **F4** function key.
- Launch the TABLET command.

14.18.5 Configuring the tablet

1. Launch the TABLET command.
2. Choose the **Configure** option.
3. **You are prompted**: Do you want to align the tablet overlay? Yes/No/<No>.

   Type **Y** in the Command line and press Enter or choose **Yes** in the prompt menu.

   You are prompted: Digitize upper left corner of the overlay.
4. Click on the mark in the upper left corner (1) of the digitizing tablet overlay.



You are prompted: Digitize the lower left corner of the overlay.

5 Click on the mark in the lower left corner (2) of the digitizing tablet overlay.

6 **You are prompted**: Digitize the lower right corner of the overlay.

7 Click on the mark in the lower right corner (3) of the digitizing tablet overlay.

8 **You are prompted**: Digitize the lower left corner of the screen pointing area.

9 Click on the mark at position (4) on the of the digitizing tablet overlay.

10 **You are prompted**: Digitize the upper right corner of the screen pointing area.

11 Click on the mark at position (5) on the of the digitizing tablet overlay.

12 **You are prompted**: Tablet configured.

14.18.6 Calibrating the tablet

To calibrate the digitizing tablet you must specify at least two points. The more additional points you specify, the more accurate the digitizing process. Additional points are particularly recommended when digitizing a non-orthogonal image, such as an aerial photograph.

1 Launch the TABLET command.

2 Choose the **Calibrate** option.

3 **You are prompted**: Digitize point #1:

Click a point on the digitizing tablet.

You are prompted: Enter coordinates for point #1:

4 Do one of the following:

- Click a point in the BricsCAD drawing window.
- Enter the coordinates (X,Y) in the Command line.

5 Repeat step 3 to specify the following calibration point.

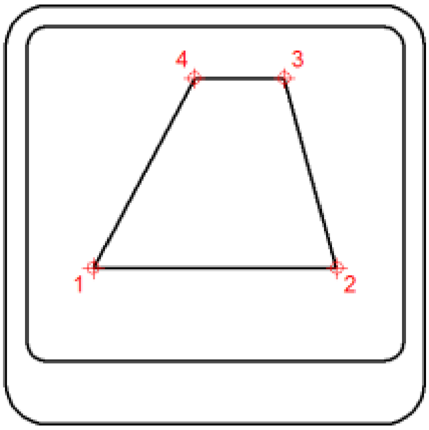
6 **You are prompted**: Digitize point #3 (or ENTER to end):

7 Do one of the following:

- Repeat step 4 to specify an additional calibration point. (You can specify up to 10 calibration points.)

- Press Enter to conclude the calibration process.

8 BricsCAD responds:
- When 2 calibration points are specified: Tablet calibrated.
- When 3 or more than 4 calibration points are specified: Select transformation type Orthogonal/ Affine:
- When 4 calibration points are specified: Select transformation type Orthogonal/Affine/ Projective:



9 Select the transformation type.

10 Transformation refers to the calculation of the points on the screen that correspond to points you digitize on the tablet.

- **Orthogonal** (left): To be used with accurate paper drawings. Orthogonal transformation maintains all angles and preserves relative distances. If only two calibration points are specified, orthogonal transformation is generated automatically.
- **Affine** (middle): Maintains parallel lines, but not necessarily the angles between intersecting lines.
- **Projective** (right): Does not maintain parallel lines, nor angles.

14.19 Lisp

How to create LISP routines for drawing setup.

14.19.1 BricsCAD Lisp

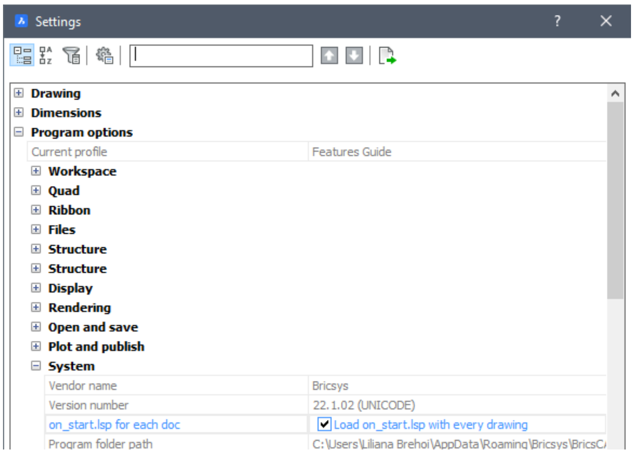
Migrating from AutoCAD®

When you start migrating to BricsCAD, you'll find that there are virtually no differences in BricsCAD LISP versus OtherLISP. Your code loads and runs, and the functionality is identical. The primary differences when "porting" your apps will be a few minor setup steps, Command line structure, and possibly file locations. To start, let's review the setup and launch of your existing programs in BricsCAD.

If you are auto-launching your programs using **Acad.lsp** or **Acaddoc.lsp**, you will need to rename them as **on_start.lsp** or **on_doc_load.lsp** respectively. The best solution is to use on_doc_load.lsp, rather than on_start.lsp.

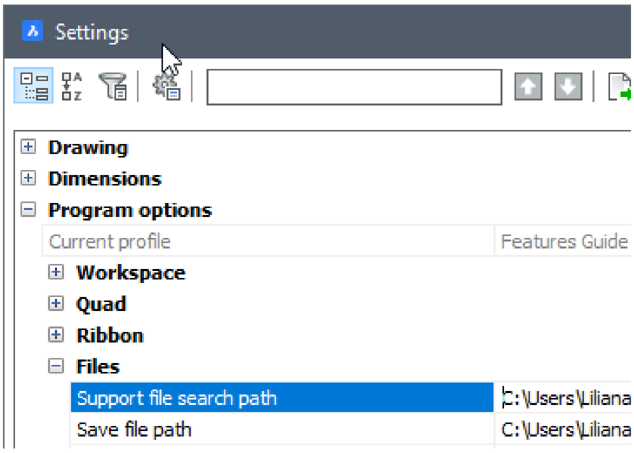
BricsCAD Settings Panel

To enable auto-launching your program set the ACADLSPASDOC system variable to 1 or simply toggle the option in *Settings / Program options / System*:



It is also recommended that you spend a little time familiarizing yourself with the Options/Setting menu in BricsCAD, as there are settings and options specific to BricsCAD environment.

The next step in the settings dialog would be to add your support folders to the Files search path, which is found within Settings / Program Options / Files:



It is recommended to use the support paths added as in the manner above and/or located one level below the current project folder. It is a good practice to use a **findfile** function to validate the file exists before proceeding with your commands, as in the method below:

```
(if (findfile "custom-code.lsp") (load "custom-code"))
```

This is one of the methods you can use to demand-load your functions within the **on_doc_load.lsp** file:

This is one of the methods you may use to demand-load your functions within the **on_start.lsp** file:

The DEFUN command (DEfine FUNction) is the same name as the function inside your **on_doc_load.lsp** file and that function redefines the calling function when it's launched.

14.19.2 Drawing Setup Lisp Routines for BricsCAD

Code sample

The below simple drawing setup routine could be adapted to your needs with minimal changes.

```
(defun C:MYSETUP ( / )                          ;define the command line function
 "mysetup"
  (setvar "cmdecho" 0)
  (command "_audit" "_y")                       ;audit current drawing
  (command "_-purge" "_all" "*" "_n")           ;purge all unused styles, etc
  (command "_-layer" "_m" "1" "_c" "1" "" "")  ; create layer "1" color "1"
  (command "_-layer" "_m" "2" "_c" "2" "" "")
  (command "_-layer" "_m" "3" "_c" "3" "" "")
  (command "_-layer" "_m" "4" "_c" "4" "" "")
  (command "_viewres" "_y" "20000")              ;set view resolution to maximum
  (setvar "filedia" 1)                           ;turn file dialogs on
  (setvar "attdia" 1)                            ;turn attribute dialogs on
  (setvar "attreq" 1)                            ;enable attribute editing on placement
  (setvar "cecolor" "bylayer")                   ;set current color bylayer
  (setvar "clayer" "0")                          ;set current layer to "0"
  (setvar "maxsort" 5000)                        ;set maximum layer sort to 5,000
  (command "_zoom" "_e" "_zoom" "0.95x")         ;zoom extents, then out a little
  (setvar "cmdecho" 1)
  (princ)
)
```

Each individual line in this drawing setup routine could have been typed in at the command line in BricsCAD. By wrapping them into a Lisp routine, the **defun** (*define function*) makes the drawing setup routine "callable" from a menu or by typing its name on the BricsCAD Command line.

If you don't include **defun**, the program will run once, when you load it by its file name. Using **defun** and loading this routine within your **lsp**, will make the function available at any time by typing its function name: **mysetup**.

The commands with preceding dashes (e.g. **-layer** and **-purge**) are used to prevent the command's dialog boxes from launching.

Note: Always use English command and options names to avoid fails on localized (non-English) BricsCAD versions.

A good practice when opening files from others is to execute an **audit** followed by a **purge** to remove any unused items.

Purge command could be run multiple times to remove nested items. See the below example:

```
(repeat 3 (command "-purge" "all" "*" "n"))
```

Note: **Purge** is done before adding the new, empty layers, as they would be removed if we did it after.

The **command** calls for setting layers could be combined into a single command statement, but it's easier to read when each line stands alone.

The **setvar** calls for setting a system variable to a specified value, e.g. style settings for dimensions and text, unit settings, etc.

The **maxsort** setting was introduced for two reasons.

1 **Maxsort** controls the maximum number of dialog box entries that will be sorted.

2 The default value for **maxsort** is 1,000. With today's computers and often found larger project files, that number can be too low.

If you open drawings where the layer names won't sort into alphabetical order, the default **maxsort** value could be the reason.

A more versatile Layer Maker for your drawing setup routine

The following example could be added to your version of mysetup.lsp. By creating a simple text file "Layers.txt" with an editor like Notepad, you can add a list of layer names to be created when the routine is run. This is an alternative to the hard-coded process shown above. By using the findfile function, BricsCAD will first search the current directory, then sequentially through your defined support folders to find the Layers.txt file. You can place your default layer setup text file in the support path and place alternative versions of it in individual project folders.

```
(if (not (findfile "Layers.txt"))                    ;"not" = looking for no result
 from findfile
(alert "Can't find Layers.txt!")                     ;No file? Yell at the user and
 exit!
(progn                                               ;Otherwise, do the stuff in
 the progn group
(setq layerfile (open (findfile "Layers.txt") "r")) ;open layers.txt to "read"
(while (setq layername (read-line layerfile))        ;while reading each line,
 store the text to layername
(command "-layer" "m" layername "")                  ;make layer with the layername
)                                                    ;end the while loop
(close layerfile)                                    ;close the Layers.txt file
)                                                    ;end the progn
)                                                    ;end the if
```

15. Modifying entities

15.1 Modifying commands overview

Icon	Command	Description
	3DARRAY	Creates multiple copies of entities in three dimensions. Entities are arrayed in a three-dimensional rectangular (rows, columns and levels) pattern or a two-dimensional polar (circular) pattern in three-dimensional space.
	ALIGN	Aligns selected entities with other entities in three-dimensional space.
	ARRAY	Creates 2D and 3D arrays of entities in rectangular or polar patterns, or along a path.
	ARRAYCL ASSIC	Creates multiple copies of entities in one of two symmetrical patterns: rectangular (rows and columns) or polar (circular).
	ARRAYPA TH	Associatively distributes entity copies evenly along a path into multiple rows, and levels.
	ARRAYPO LAR	Associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
	ARRAYRE CT	Associatively distributes copies of entities into any number of rows, columns, and levels.
	BREAK	Splits an entity into two entities.
	CHAMFER	Creates chamfers at intersections of linear entities, defined by two lengths, or a length and an angle.
	CHANGE	Changes the properties of entities, through the Command line. This command has largely been superseded by the **Properties** panel.
	COPY	Draws duplicates of the selected entities.
	COPYGUID ED	After selecting the entities, their relation to their reference curves will automatically guide you to inserting the copy in similar locations.
	DIVIDE	Places markers (points or blocks) along a selected entity. The markers evenly divide the entity into the specified number of equal parts.

Icon	Command	Description
	EXPLODE	Reduces complex entities to simpler ones, such as blocks into constituent parts, or polylines into lines and arcs.
	EXTEND	Lengthens lines, arcs, polylines or rays to meet another entity.
	FILLET	Fillets intersections; joins intersecting lines with an arc of any suitable radius.
	FLATTEN	Projects 2D and 3D entities onto the XY-plane of the current view.
	HATCHEDIT	Edits hatches and gradient fills in a dialog box.
	JOIN	Joins lines, lightweight and old-style 2D polylines, 3D polylines, arcs, elliptical arcs, polyarcs, splines, and helices at their common endpoints.
	LENGTHEN	Changes the length of a line, polyline, freehand entity or arc.
	MANIPULATE	Launches the Manipulator widget; rotates, moves, mirrors and scales entities. Keep the Ctrl key pressed while starting an action to manipulate a copy of the selected entities.
	MEASURE	Divides a selected entity into segments by placing markers (points or blocks) at specified intervals along its length or circumference.
	MIRROR	Moves or copies the reflected image of selected entities about a line.
	MIRROR3D	Moves or copies the reflected image of entities about a plane.
	MOVE	Moves entities to another location in the same drawing.
	MOVEGUIDED	After selecting the entities, their relation to their reference curves will automatically guide you to moving the entities in a similar location.
	OFFSET	Creates a parallel or offset copy of linear entities.
	OVERKILL	Deletes duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones; optionally moves duplicates to a dedicated layer. Combines duplicate blocks; optionally purges duplicate block definitions.

Icon	Command	Description
	PEDIT	Edits a two-dimensional or three-dimensional polyline, or a polygon mesh.
	PROPERTIES	Opens the BricsCAD **Properties** panel.
	ROTATE	Rotates entities around a specified point.
	ROTATE3D	Rotates entities around an axis.
	SCALE	Changes the scale of existing entities, either enlarging them or reducing them proportionately in X, Y, and Z directions.
	SIMPLIFY	Reduces the number of vertices of polylines and hatch boundaries without changing their general shape. Simplified entities are easier to manipulate and can significantly reduce file size.
	SPLINEDIT	Edits a two-dimensional or three-dimensional spline.
	STRETCH	Moves a portion of a drawing while retaining connections to other parts of the drawing.
	TRIM	Erases the portions of selected entities that cross a specified boundary.

15.2 Entity modification settings

Entity modification settings are found in the **Settings** dialog box under **Drawing** > **Drafting** > **Entity modification**.

15.3 Match properties

The MATCHPROP command (short for "match properties") assigns the properties of one entity to other entities.

15.3.1 Copying properties between entities

1 Launch the MATCHPROP command.

2 **You are prompted**: Select entity to copy properties from:

3 Select the source entity.

4 **You are prompted**: Active Settings: 'Color' 'Layer' 'Linetype' 'Linetype Scale' 'Lineweight' 'Thickness' 'Plot Style' 'Dimension' 'Polyline' 'Material' 'Text' 'Shadow Display' 'Hatch' 'Multileader' Select entities to copy

properties to or [Settings/selection options (?)]:

5 (option) Choose the **Settings** option, then press Enter.

Adjust the settings, then click the **OK** button.

6 Select the target entities.

7 **You are prompted**: Select entities to copy properties to or [Settings/selection options (?)]:

8 Keep selecting entities or right-click to stop.

15.3.2 Copying properties between entities with BIM data

Whether the MATCHPROP command copies BIM data depends on the value of the BIMMATCHPROP system variable.

- **BimMatchProp = 0**: Match standard properties

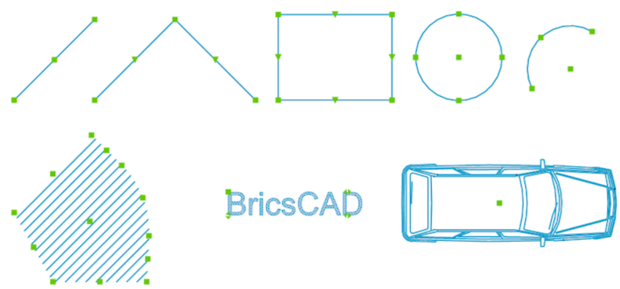
  Only standard properties will be copied, according to the currently selected options (see above).
- **BimMatchProp = 1**: Match standard and BIM properties

  BIM data, including IFC properties and user defined properties are copied, except:
  - BIM Type*
  - Composition/Profile
  - GUID
  - Quantities

* When using MATCHPROP between a BIM classified entity and an entity without any BIM data, the BIM type and the corresponding BIM properties will be copied along to the unclassified entity, except Composition/Profile, GUID and Quantities.

15.4 Grips editing

Grips are displayed as small colored squares at key-points on selected entities.



Whether grips are displayed or not and the appearance of grips are controlled through a series of grips settings in the **Settings** dialog box under **Drawing** > **Drafting** > **Display/Viewing** > **Grips**.

In complex drawings, using too many grips may clutter the display. The GRIPOBJECTLIMIT system variable allows you to suppress the display of grips if the number of selected entities exceeds a certain limit. If set to zero, all grips are displayed.

Grips can be dragged using the mouse in order to stretch, move, mirror, rotate or scale entities.

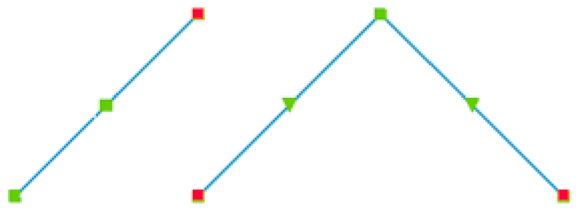
When the GEOMRELATIONS system variable is set, the geometric relation (perpendicular or tangent) between the entity being edited and other lines, polylines, arcs or circles is indicated (see Geometric Relations).

To select a grip, move the mouse over it, then click. Selected grips are displayed in a different color, which is defined by the HOTGRIP system variable. A selected grip is called a hot grip. By default, hot grips are displayed in red.

The **midpoint** grip on lines and infinite lines, the **startpoint** grip on rays, the **center** grip on circles, arcs, ellipses and elliptical arcs, the grip on points and the **insertion point** grip of texts, mtexts, blocks and Xrefs can be used to quickly move such entities.

Note: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

15.4.1 Selecting multiple grips

1. Select all entities you want to edit.
2. Press and hold the Shift key, then click the first grip. The selected grip is displayed in red.
3. Keep the Shift key pressed, then click more grips. All selected grips are displayed in red.
4. 
5. Release the Shift key, then click the hot grip you want to use as the base point. The hot grips move simultaneously.

15.4.2 Grip Edit Command Options

When you start dragging a selected grip, BricsCAD displays a prompt in the Command line:

Enter new point [Base point/Copy/Undo/eXit]:

You can choose an option using the keyboard, or in the **Prompt** menu.

Keyboard	Prompt Menu	Description
B	Base Point	You are prompted to specify a new base point, other than the selected grip.
C	Copy	Copies the selected entities and keeps the originals. By default, multiple copies can be created. Press Enter to stop making copies.
U	Undo	If the **Copy** option is chosen, deletes the latest copy.

Keyboard	Prompt Menu	Description
R	Reference	(Applies to Grip Rotate only) Allows you to specify a reference angle. Type the angle in the Command line or specify two points.
X	Exit	Aborts the grip edit command.

15.4.3 The grips editing context menu

The grips editing commands can be chosen by repeatedly hitting the space bar or the Enter key, or in a right-click context menu.

The command options are: stretch, move, mirror, rotate, scale.

The stretch command option is selected by default.

15.4.4 Stretching entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point.

4 All selected grips move simultaneously.

5 **You are prompted**: Enter new point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

8 Do one of the following to define the displacement point:

- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

15.4.5 Moving entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the move.

4 The **Grip Stretch** command is launched.

5 Do one of the following:

- Right-click, then select **Move** in the context menu.
- Hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Move** is active.

You are prompted: Enter move point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

7 Do one of the following to define the displacement point:
- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

15.4.6 Copying entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the move.

4 The **Grip Stretch** command is launched.

5 Do one of the following:
- Right-click, then select **Move** in the context menu.
- Hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Move** is active.

You are prompted: Enter move point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

7 Do one of the following:
- Choose **Copy** in the context menu.
- Type **C**, then press **Enter**.

8 Do one of the following to define the displacement point:
- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

9 Repeat the previous step to create more copies or press the Space bar or Enter to stop.

15.4.7 Mirroring entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the mirroring.

4 The **Grip Stretch** command is launched.

5 Do one of the following:
- Right-click, then select **Mirror** in the context menu.
- Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Mirror** is active.

You are prompted: Enter second point: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

8 Specify a second point to define the mirror axis.

9 The selection is mirrored.

15.4.8 Rotating entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the rotation.

4 The **Grip Stretch** command is launched.

5 Do one of the following:

- Right-click, then select **Rotate** in the context menu.
- Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Rotate** is active.

You are prompted: Enter rotation angle: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

8 Do one of the following to rotate the selection:

- Specify a point to define the rotation angle.
- Type the rotation angle in the Command line, then press Enter.

15.4.9 Scaling entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the scaling.

4 The **Grip Stretch** command is launched.

5 Do one of the following:

- Right-click, then select **Scale** in the context menu.
- Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Scale** is active.

You are prompted: Enter scale factor: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

8 Specify a point.

9 Type the scale factor in the Command line, then press Enter to scale the selection.

15.5 Selection methods

Before you start using modification commands, you need to know how to compose a selection set.

If the PICKFIRST variable is ON, you can build the selection set before starting the command. If the PICKFIRST variable is OFF, you are prompted to select the entities after you launched the modification command.

Using **Quick Select** (QSELECT command), you can specify filters such as entity type, color, linetype or any other entity property, to compose a selection set.

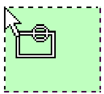
The SELECTIONCYCLING, SELECTIONPREVIEW, and PREVIEWEFFECT system variables facilitate the selection procedure.

The SELECTIONPREVIEWFILTER system variable allows you to exclude entities from being selected: entities on locked layers, entities in Xrefs, tables, MTexts, hatches, entities in group.

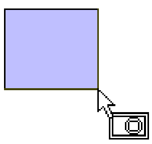
15.5.1 Composing a selection set before (pre-pick)

To compose a selection set before launching a modification command, do one or more of the following:

- Click entities.
- Selected entity highlight and grips are shown.
- Area selection:
    - Click to define the first corner of a selection area.
    - Move the mouse to the left to define a **Crossing area**:

    - All entities that overlap or that are completely inside the rectangle are added to the selection set.
    - By default, the selection area is displayed in green with a dashed line.
    - Click to define the first corner of a selection area.
    - Move the mouse to the right to define a **Window area**:

    - All entities that are completely inside are added to the selection set.
    - By default, the selection area is displayed in blue with a continuous line.

- When working with 3D solids and surfaces, pressing the **Ctrl** key during window selection allows you to select faces or edges instead of the entire solid or surface. The icon next to the cursor indicates the selection mode:
  - Solids (default)
  - Faces
  - Edges
  - Edges Sheet Metal side

    Note: The **Edges Sheet Metal side** option is available in the **Hotkey Assistant** widget only if a document has the sheet metal context. It allows you to select several flanges' edges belonging to a certain side of the model.

  If HOTKEYASSISTANT in ON, the **Hotkey Assistant** widget indicates the current selection mode:

- Press an hold the **Shift** key, then use one of the above methods to remove entities from the selection set.

Note:

- The **Selection Area** system variables control the display of the area selection effects.
- When the PICKADD system variable is OFF, you cannot add entities to a selection set.
- Press and hold the left mouse button to start window selection when the entity under the cursor highlights. This method is especially helpful for window selection in rendered 3D models.
- You can only select vertices of 3D solids if bitcode 8 of the SELECTIONMODES system variable is selected.
- Lasso selections can be made if the bitcode >= 4 of PICKAUTO system variable is selected.

15.5.2 Composing a selection set afterward (post-pick)

If you launch a modification command when no selection set is active, you are prompted to select entities first. BricsCAD provides a range of selection methods to compose your selection:

- **Add to set**: adds selected entities to the selection set.
- **Subtract from set**: removes selected entities from the selection set.
- **Picking**: place the pick-box over a part of the object and click.
- **Select all entities**: selects all entities in the entire drawing, including entities which are on hidden layers. Entities on frozen layers are not selected.
- **Previous selection**: re-uses the previous selection set.
- **Last entity in drawing**: selects the most recent entity.
- **Window**: all entities that are inside, overlap or are completely outside a window are selected. You can choose between a **rectangle**, a **polygon** or a **circle**.

- **Fence**: selects all entities that cross a multi-segment line.
- **Point**: click a point to select all closed entities that enclose this point.
- **Select by Properties**: displays the **Select by Properties** context menu to select entities by **Color**, **Layer**, **Linetype**, **Name**, **Thickness**, **Type**, **Value**, **Width** or **Handle**.
- **Undo**: undoes the last selection action.
- **Location**: displays the **Select Location** context menu.

15.5.3 Selecting overlapping and obscured entities

When entities are on top of each or are obscured by other entities, it is difficult or sometimes impossible to select such entities.

Cycling through the overlapping entities and highlighting each entity one by one helps you select such entities.

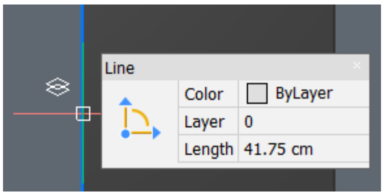
Note: Vertices of 3D solids are only included in the cycling process if bitcode 8 of the SELECTIONMODES system variable is selected.

Selection dialog box method (SELECTIONCYCLING==2)

Make sure that SELECTIONCYCLING == 2 (the Selection dialog box displays) and the Quad cursor menu is active (controlled by the QUADDISPLAY system variable).

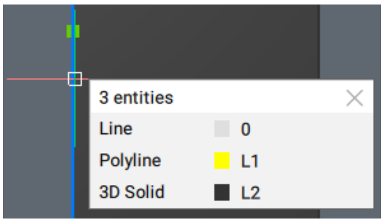
1 Place the cursor over the overlapping entities.

1 A badge displays to signal overlapping entities. The entity on top is highlighted.



2 Click the overlapping entities.

3 A selection dialog box displays listing the entities from front to back.



Note: If you click away without making a selection, the top entity is automatically selected.

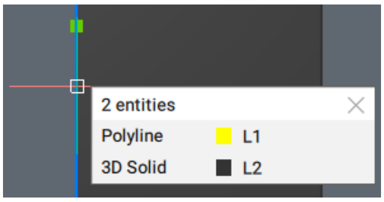
4 (Optional) Press and hold the **Ctrl** key.

The subentities are listed in the selection dialog box.

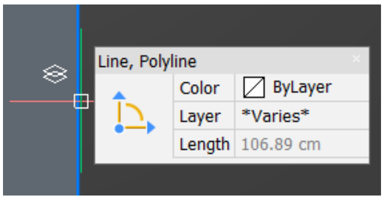
5 Click the desired entity or subentity in the list, then click the **Close** button or move the cursor away from the dialog box.

Note: Multiple selection is accepted by using **Ctrl-Click** and **Shift-Click**.

The entity or subentity is now selected and removed from the selection dialog box.



6 Repeat the previous steps to select more entities or subentities.

7 

SELECTIONPREVIEW method

Make sure that SELECTIONPREVIEW ==1 or 3, and ROLLOVERTIPS == 1.

1 Place the cursor over the overlapping entities.

1 One of the entities highlights.

2 Repeatedly press the **Tab** key.

The overlapping entities highlight one by one.

3 Click when the desired entity highlights.

4 Repeat the previous steps to select more entities.

Note:

- Do not move the mouse while pressing the **Tab** key, because this will restart the highlight order.
- Hold down the **Shift** key to cycle the overlapping entities in reverse order.

15.5.4 Selecting visible subentities

By default, the entire entity is highlighted when hovering the cursor over an entity.

This method allows you to select subentities of compound entities, such as polylines, 3D polylines, regions, 3D solids, etc.

Note: This method applies when the SELECTIONCYCLING system variable is not set to 2. Use the keyboard shortcut **Ctrl + W** to toggle on/off the current setting of the SELECTIONCYCLING system variable.

1 Place the cursor over the visible subentity.

1 The entire entity highlights.

2 Hold down the **Ctrl** key.

The visible subentity under the cursor highlights.

3 Click the highlighted subentity.

4 The properties of the selected subentity are displayed in the **Properties** panel.

5 (Optional) Repeat the previous steps to select more subentities.

The following subentities can be selected: polyline segments, vertices and segments of 3D polylines, vertices, edges and faces of regions, surfaces, meshes and 3D solids.

15.6 Quick select

Quick Select allows you to specify filters such as entity type, color, line type or any other entity property, to compose a selection set. **Quick Select** applies to the entire drawing or to a previously created selection.

15.6.1 General procedure

- Launch the QSELECT command.
- (Optional) Tick the **Apply to current selection set** option.
- (Optional) Set an entity type filter.
- (Optional) Define the property filter(s).
- Apply the filter set to either:
    - Create a new selection set ( ).
    - Add the selection to an existing selection set ( ).
    - Remove the selection from an existing selection set ( ).

15.6.2 Launching the QSELECT command

Do one of the following:

- Display the **Properties** panel, then click the Quick Select tool button ( ) on the panel.
- Type **qselect** in the Command line.

The **Properties** panel displays in quick select mode.

15.6.3 Setting an entity type filter

This step is optional. If not set, the quick select session applies to all entity types.

1 Click the **Selection list** button.

2 A list of all entities in the drawing (or in the current selection set, when the **Apply to current selection set** option is set) and the number of entities of each type displays.

3 Select an entity type in the list.

The properties of the selected entity type display.

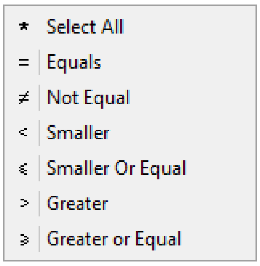
15.6.4 Defining a property filter

By default, an asterisk (*) displays in all property fields.

The available properties depend on the selected entity type. If the entity type is not specified, only general properties are listed.

1 Click the asterisk of the property you want to set a filter for.

2 A context menu displays.



3 Select an operator.

4 Click the property field.

5 Depending on the property, do one of the following:

- Type a value.
- Select an option in a drop-down list.

6 (Optional) Repeat the previous steps to define more property filter(s).

15.7 Draw order

When new entities overlap existing entities, they display and print on top of the previously drawn entities. The DRAWORDER command can change the order in which overlapping entities are displayed and printed. You can move entities to the front or the back, or above or under another entity.

1 Launch the DRAWORDER command.

2 **You are prompted**: Select entities to change the draw order:

3 Select the entities, then right-click or press enter.

4 **You are prompted**: Change draw order: Above/Under/Clear/Front/<Back>:

5 Choose an option.

6 Depending on your choice:

- If you have chosen **Above** or **Under**, the Command line reads: Select reference entity:

  Identify the reference entity. The display of the selection set is changed with respect to the reference entity.
- If you have chosen **Front** or **Back**, the display of the selection set is changed accordingly.

15.7.1 Using Draw Order tools

Draw Order tools are available in the **Tools** menu and the **Draw Order** toolbar.

Icon	Tool Name	Description
	Bring To Front	Places the selection set on top.
	Bring Above	Places the selection set above the reference entity. (*)

Icon	Tool Name	Description
	Send Under	Place the selection set under the reference entity. (*)
	Send to Back	Moves the selection set to the back.
	Hatch to Back	Sends all hatch entities to the back (HATCHTOBACK command).
	Text and Dimensions to Front	Brings all texts and/or dimension entities to front. Options: **Text** / **Dimensions** / **Leaders** / **All** can be chosen either in the Command line or in the prompt menu. The default option is **All** (TEXTTOFRONT command).

15.8 The manipulator

The manipulator is a versatile grip tool to swiftly move, rotate, scale and mirror selected entities. Holding down the Ctrl key when starting the action keeps the original entities, creating copies. When selecting an entity, the manipulator is displayed at a user-controlled location and orientation. The location and orientation are controlled in a similar way to a dynamic UCS. When multiple entities are selected simultaneously, the manipulator is displayed at their geometric center. When a selection is built up step by step, the initial location of the manipulator is preserved. The manipulator can be moved to any desired location or orientation using its handles and the context menu options.

Depending on the operation type and if dynamic dimensions are active, a dynamic entry field is available to set the distance, angle or scale factor.

15.8.1 Adaptive snap step

The snap step size automatically adapts to the current screen zoom factor if SNAPTYPE= 2. The ADAPTIVEGRIDSTEPSIZE system variable controls the smallest available step size. The adaptive step size is used to dynamically adjust the step size in the Manipulator Ruler, which enables you to quickly modify entities using precise values, without having to key in numbers.

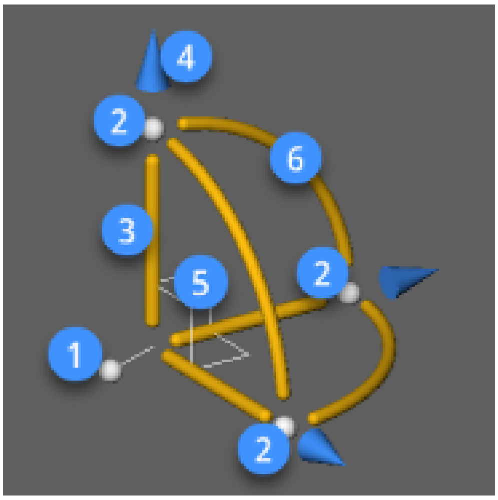
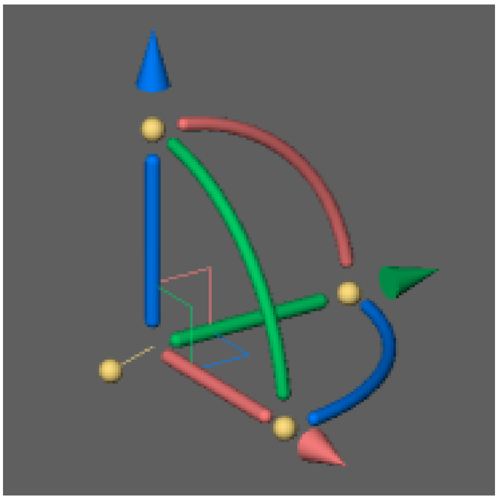
15.8.2 The manipulator settings

The manipulator settings can be found in the **Settings** dialog box under **Program Options** > **Manipulator**.

When the Manipulator is displayed, select **Manipulator Settings** in the right-click context menu.

15.8.3 The manipulator layout

Check the **Classic Color** option in the manipulator context menu for the classic layout. Untick the option for the default layout.

	
Default	Classic

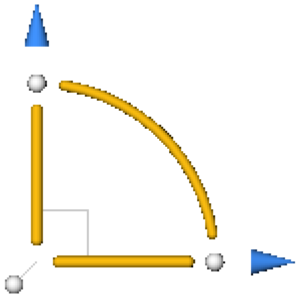
4 handles

- **1**: manipulator handle
- **2**: axis handle

12 command trigger zones

- **3**: axis
- **4**: arrowhead
- **5**: plane swatch
- **6**: rotation arc

When the current view is parallel to one of the main axes of the coordinate system, as in a plan view, the manipulator is simplified to a 2D version.



15.8.4 The location and orientation of the manipulator

The initial location and orientation of the 3D manipulator tripod depends on:

- The entity type.
- The point you click on the entity.

Entity Type	Selection Point	Manipulator - Anchor Point	Manipulator - Orientation
Line	Near endpoint	At endpoint	X-axis through the other endpoint
Line	Near midpoint	At midpoint	X-axis through the nearest endpoint
Polyline Line Segment	Near vertex	At vertex	X-axis through the other vertex of the selected line segment
Polyline Line Segment	Near segment midpoint	At the segment midpoint	X-axis through the nearest vertex
Polyline Arc Segment	On the arc segment	At the arc center	X-axis through the nearest vertex
Arc	Random	At the arc center	X-axis through the nearest endpoint
Circle	Random	At the circle center	X-axis parallel with the X axis of the current coordinate system
Spline	Random	At the nearest endpoint	Through the other endpoint
Region	On a line segment	At the nearest endpoint of the line segment	X-axis through the other endpoint
Region	On an arc segment	At the center of the arc segment	X-axis through the nearest endpoint of the arc segment
Text, MText, Block, Xref	Random	At the origin of the entity	X-axis at the rotation angle of the entity
3D Solid 3D Solid Face	On a face of the solid	At the nearest vertex	X-axis along the nearest edge XY-plane in the selected face
Multiple Entities	Random	At the geometric center of the selection set**	Parallel with the axes of the current coordinate system

* See Selection Modes to learn how to select a 3D solid face or the entire solid.

** The position of the manipulator does not change when more entities are added to the selection set.

15.8.5 Displaying the manipulator

The value of the MANIPULATOR system variable controls when the MANIPULATOR appears automatically upon selection:

- 0: Do not display Manipulator
- 1: Display Manipulator when entities are selected
- 2: Display Manipulator if the left mouse button is pressed longer than the duration defined in the MANIPULATORDURATION system variable.

In order to display the Manipulator when multiple entities are selected:

- Select the entities one by one or use window selection, then long-click to select the last entity.
- Set the value of the MANIPULATOR system variable to 1.
- Compose the selection set, then launch the MANIPULATE command.

15.8.6 Moving the manipulator

Do one of the following:

- Long-click anywhere on the manipulator and move it to a new location.
- MANIPULATORHANDLE = 0: Click the manipulator handle (1) and move it to a new location*.
- The anchor handle stands out to avoid overlap with entity grips.
- MANIPULATORHANDLE = 1: Long-click the manipulator handle (1) and move it to a new location*.
- Place the cursor over the manipulator, then right-click and choose **Move** in the context menu and move it to a new location*.

* Specify the new location using entity snaps.

15.8.7 Rotating the manipulator

Select one of its axis handles (2) to rotate the manipulator. Pick a point or type a value in the dynamic entry field to specify the rotation angle.

Press the Shift key to rotate the Manipulator 90 degrees around its normal axis (Z-axis).

15.8.8 Reorienting the manipulator

1. Place the cursor over the manipulator, then right-click and choose **Reorient Manipulator** in the context menu.

   You are prompted: Specify origin of manipulator:
2. Pick a point.
3. The anchor point of the manipulator is placed at the specified point.

   You are prompted: Point on X-axis or <Accept>:
4. Pick a point.
5. The manipulator rotates about its X-axis.

   You are prompted: Point on the XY-plane with positive Y value or <Accept>:
6. Pick a point to define new position of the manipulator.

15.8.9 Aligning the manipulator with a coordinate system

Move the cursor over the manipulator, then right-click and select an option in the context menu:

- **Align with WCS**: the manipulator axes are forced to become parallel to the corresponding axes of the World Coordinate System.
- **Align with UCS**: if a UCS is currently the active coordinate system, the manipulator axes are forced to become parallel to the corresponding axes of the UCS.

15.8.10 Aligning the manipulator with the face of a solid

Dynamic UCS (DUCS) must be ON for this operation to succeed.

1 Place the cursor over the manipulator then right-click and choose **Move** in the context menu.

2 Hover over the face of a solid.

3 The XY-plane of the manipulator aligns with the solid face under the cursor.

4 Do one of the following:
   - Snap to a point on the selected face or on one of its edges.
   - Press the Shift key to lock the XY-plane to the selected face, then snap to a point outside the face.

15.9 Moving entities

The MOVE command moves entities in the drawing.

The CUTCLIP command copies entities to the Clipboard, and then erases them from the drawing.

The PASTECLIP command (short for "paste clipboard") pastes entities from the Clipboard into the current drawing.

The PASTEORIG command pastes entities from the Clipboard into another drawing, using the coordinates of the source drawing.

The PASTEBLOCK command pastes entities from the Clipboard as a block into the current drawing.

Use CUTCLIP, then either PASTECLIP, PASTEORIG or PASTEBLOCK to move entities to another drawing.

The **Copy** option of the Grips Editing commands allows you to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

Note: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

15.9.1 Moving entities in a drawing

The default method is to create a selection set and then specify a starting point (base point) and an endpoint (displacement point) to define the relocation of the entities. You can also move the entities using a direction vector.

1 Launch the MOVE command.

1 **You are prompted**: Select entities to move [selection options (?)]:

2 Select the entities, then right-click or press Enter.

3 The selection set is now attached to the cursor.

You are prompted: Enter base point [Displacement] <Displacement>:

4 Specify a point.

5 **You are prompted**: Enter second point or <Use base point as displacement>:

6 Specify a point.

7 The selection set is moved.

15.9.2 Moving entities between drawings

1 In the source drawing, select the entities you want to move.

2 Do one of the following:

- Right-click, then choose **Cut** in the context menu.

  Hold down the right-mouse button longer when the Quad is active.
- Choose **Cut** in the **Edit** menu.
- Press **Ctrl + X**.

The entities are deleted in the source drawing and copied to the Clipboard.

3 Switch to the target drawing.

4 Do one of the following:

- Choose **Paste** in the **Edit** menu.
- Press **Ctrl + V**.

The bottom left corner of the bounding rectangle of the selection is attached to the cursor.

5 Do one of the following:

- Specify a point.
- Press to paste the selection set using the coordinates of the source drawing.

Note: Launch the UNDO command in the source drawing bring back the moved entities.

15.9.3 Moving a selection as a block

1 Select the entities you want to move.

2 In the source drawing, select the entities you want to move.

3 Do one of the following:

- Right-click, then choose **Cut** in the context menu.

  Hold down the right mouse button longer when the Quad is active.
- Choose **Cut** in the **Edit** menu.
- Press **Ctrl + X**.

The entities are deleted in the source drawing and copied to the Clipboard.

4 The entities are deleted in the drawing and copied to the Clipboard.

5 (option) Switch to the target drawing.

6 Choose **Paste as a Block** in the **Edit** menu or press **Ctrl + Shift + V**.

The bottom left corner of the bounding rectangle of the selection is attached to the crosshairs in the target drawing.

7 Specify the insertion point.

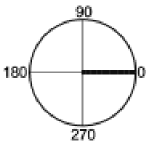
8 The selection is pasted as a block, using a default name.

9 (option) Launch the EXPBLOCKS command and rename the block in the **Drawing Explorer / Blocks** dialog box.

15.10 Rotating entities

The ROTATE command rotates entities at a specified point along a specified rotation angle or by an angle referenced to a base angle. The default method rotates the entities using a relative rotation angle from their current orientation.

By default, angles start at 3 o'clock (East) and increase in a counter-clockwise direction. If you want to rotate in a clockwise direction, you can enter a negative value.



Note:

- The **Angular Base** is controlled by the ANGBASE system variable, which in turn refers to the current UCS.
- The **Angle Direction** is controlled by the ANGDIR system variable, which sets the positive angle direction from angle 0 with respect to the current UCS.

15.10.1 General procedure

1 Launch the ROTATE command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to rotate [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Rotation point:

6 Specify a point.

7 **You are prompted**: Rotation angle or [Base angle/Copy] <0>:

8 Do one of the following:

- Specify a point.
- Type a value and press Enter.

The selection set is rotated.

15.10.2 Rotating a selection set relative to a base angle

1 Launch the ROTATE command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to rotate [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Rotation point:

6 Specify a point.

7 **You are prompted**: Rotation angle or [Base angle/Copy] <0>:

8 Choose the **Base angle** option.

You are prompted: Base angle <0>:

9 Do one of the following:

- Type a value and press Enter.
- Click two points to specify the base angle.

10 **You are prompted**: New angle <30.84>:

11 Do one of the following:

- Type a value and press Enter.
- Click a point.

The selection is rotated.

15.10.3 Rotating in 3D

1 Launch the ROTATE3D command or 3DROTATE command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to rotate [selection options (?)]:

4 Select more entities, or press Enter to stop selecting.

5 **You are prompted**: Specify first point on axis or define axis by [Xaxis/Yaxis/Zaxis/2Points/Object/Last/View] <2Points>:

6 Press Enter to accept the 2points method to select the rotation axis.

7 **You are prompted**: Specify start point of axis:

8 Specify a point.

9 **You are prompted**: Specify end point of axis or <use axis parallel to view direction>:

10 Specify a point.

11 **You are prompted**: Specify rotation angle or [Base angle/Copy]: 45

12 Type a value and press Enter.

13 The selection is rotated.

15.11 Mirroring entities

The MIRROR command creates a mirrored copy of a selection set. The selection is mirrored along a mirror line, which you define by specifying two points. You can choose to either delete or retain the original entities.

The MIRROR3D command creates a mirrored copy of selected entities in three-dimensional space. In this case the selection is mirrored along a mirror plane, which can be defined by either specifying three points, selecting an existing two-dimensional planar entity, aligning the plane parallel to the XY, YZ, or XZ plane of the current UCS, or aligning the plane with the current view. You can choose to either delete or retain the original entities.

Note: Whether text is mirrored or not by the MIRROR command is controlled by the MIRRTEXT system variable.

15.11.1 Mirroring entities along a line

1 Launch the MIRROR command.

1 **You are prompted**: Select entities to mirror [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to mirror [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Start of mirror line:

6 Specify a point.

7 The selection set rotates dynamically.

You are prompted: End of mirror line:

8 Specify a point.

9 The selection is mirrored.

You are prompted: Delete the original entities? [Yes-delete entities/No-keep entities] <No-keep entities>:

10 Decide whether or not to keep the original entities.

15.11.2 Mirroring entities along a plane

1 Launch the MIRROR3D command.

1 **You are prompted**: Select entities to mirror [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to mirror [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Define mirror plane by [Entity/Last/View/Z axis/X-Y plane/Y-Z plane/Z-X plane/3 points] <3 points>:

6 Press Enter to accept the 3 points method to define the mirror plane.

7 **You are prompted**: First point on plane:

8 Specify a point.

9 **You are prompted**: Second point on plane:

10 Specify a point.

11 **You are prompted**: Third point on plane:

12 Specify a point.

13 **You are prompted**: Delete the original entities? [Yes-delete entities/No-keep entities] <No-keep entities>:

14 Decide whether or not to keep the original entities.

15.12 Aligning entities

The ALIGN command lets you reposition a selection set with respect to an existing entity in the drawing. The selection set is moved and rotated in a single action. If necessary, you can also scale the selection to fit in its new location.

The command can be used both in a 2D and 3D environment.

15.12.1 Aligning in 2D

1 Launch the ALIGN command.

2 **You are prompted**: Select entities:

3 Select the entities, then right-click or press Enter.

4 **You are prompted**: Specify first source point:

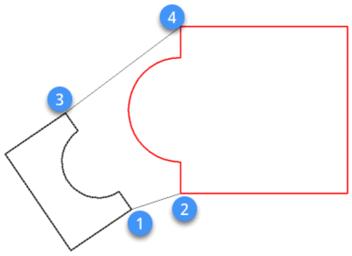
5 Snap to the first source point in the selection set (1).

6 **You are prompted**: Specify first destination point:

7 Snap to the first destination point on the reference entity (2).

8 A witness line is drawn between the source point and the target point.

You are prompted: Specify the second source point:



9 Snap to the second source point in the selection set (3).

10 **You are prompted**: Specify second destination point:

11 Snap to the second destination point on the reference entity (4).

12 A witness line is drawn between the source point and the target point.

You are prompted: Specify the third source point:

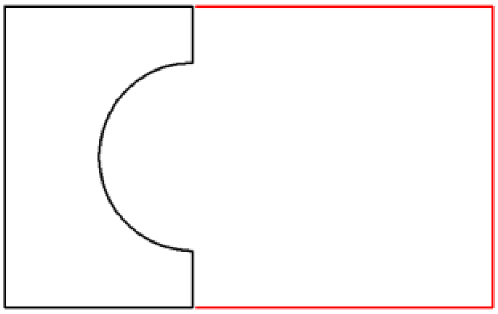
13 Right-click to use the ALIGN command in 2D.

14 **You are prompted**: Scale objects based on alignment points [Yes/No] <No>:

15 Do one of the following:

- Press Enter if you don't want to scale the selection set.
- Type Y and press Enter to scale the selection set.
- Scaling adjusts the distance between the source points to the distance between the target points.

16



Result of the Align procedure with scaled selection set.

15.12.2 Aligning in 3D

1 Launch the ALIGN command.

1 **You are prompted**: Select entities:

2 Select the entities, then right-click or press Enter.

3 **You are prompted**: Specify first source point:

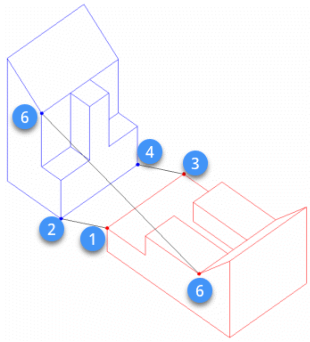
4 Snap to the first source point in the selection set (1).

5 **You are prompted**: Specify first destination point:

6 Snap to the first destination point on the reference entity (2).

7 A witness line is drawn between the source point and the target point.

You are prompted: Specify the second source point:



8 Snap to the second source point in the selection set (3).

9 **You are prompted**: Specify second destination point:

10 Snap to the second destination point on the reference entity (4).

11 A witness line is drawn between the source point and the target point.

You are prompted: Specify the third source point:

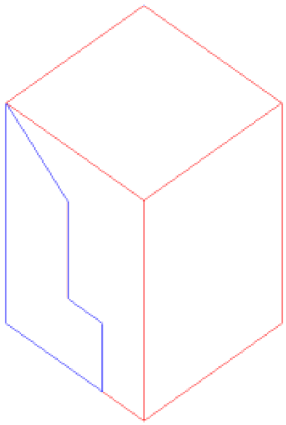
12 Snap to the third source point in the selection set (5).

13 **You are prompted**: Specify third destination point:

14 Snap to the third destination point on the reference entity (6).

15 A witness line is drawn between the source point and the target point.

The entity is aligned.



15.13 Changing entities

The CHANGE command lets you change the insertion point and orientation of texts and block inserts.

The **Properties** option of the command can also change the layer, line type, line type scale, line weight, etc. but the BricsCAD **Properties** panel is much more versatile.

15.13.1 Changing text entities

1 Launch the CHANGE command.

1 **You are prompted**: Select entities to change [selection options (?)]:

2 Select the text entities, then right-click or press Enter.

3 **You are prompted**: Change point [special Entities/Properties...]:

4 Specify a new insertion point or press Enter to keep the original.

5 **You are prompted**: New text style Standard:

6 Type the name of a new text style or press Enter to keep the current style.

7 The text entity scales dynamically.

You are prompted: New height <current height>:

8 Do one of the following:

- Click to define the new text height.
- Type a value to specify the new text height.
- Press Enter to keep the current text height.

The text entity rotates dynamically.

You are prompted: New rotation angle <0>:

9 Do one of the following:

- Specify a point
- Type a value and press Enter.
- Press Enter to keep the current rotation angle.

You are prompted: New text <current text>:

10 Type a new text or press Enter to keep the current text.

11 When multiple text entities are selected, the procedure restarts for each entity.

15.13.2 Relocating blocks

1 Launch the CHANGE command.

1 **You are prompted**: Select entities to change [selection options (?)]:

2 Select the blocks, then right-click or press Enter.

3 **You are prompted**: Change point [special Entities/Properties...]:

4 Specify a point.

5 The block rotates dynamically.

You are prompted: New rotation angle <current angle>:

6 Do one of the following:

- Specify a point.
- Type a value and press Enter.
- Press Enter to keep the current rotation angle.

7 When multiple blocks are selected, the procedure restarts for each entity.

15.14 Copying entities

The COPY command duplicates entities within the current drawing. The default method is to create a selection set and then specify a starting point (base point) and an endpoint (second point) for the copy. You can also specify the displacement as a direction vector (X,Y,Z).

Depending on the COPYMODE system variable, you can create multiple copies of the selection set or just one.

The COPYCLIP command copies entities to the clipboard, which you can then paste in the current drawing or in another drawing using the PASTECLIP or the PASTEBLOCK commands.

Note: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

15.14.1 General procedure

1 Launch the COPY command.

1 **You are prompted**: Select entities to copy [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

4 **You are prompted**: Select entities to copy [selection options (?)]:

Current copy mode: Multiple

You are prompted: Enter base point [Displacement/mOde] <Displacement>:

5 Specify a point.

6 The copied selection moves dynamically.

You are prompted: Enter second point or [Array] <Use base point as displacement>:

7 Specify a point.

8 A first copy of the selection is created.

A second copy moves dynamically.

You are prompted: Enter second point [Undo/Array/Repeat/Exit]:

9 Do one of the following:

- Keep specifying points to create multiple copies.
- Press Enter to stop.

15.14.2 Using the Repeat option

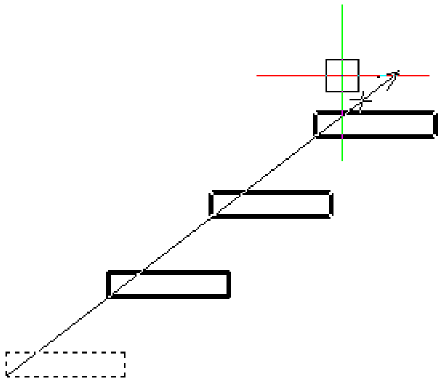
1 Execute steps 1-5 of the general procedure (see above).

1 **You are prompted**: Enter second point [Undo/Array/Repeat/Exit]:

2 Choose the **Repeat** option.

Multiple copies at regular intervals are created when you move the cursor.

You are prompted: Select end point (<ENTER> to key in number of repeats):



3 Do one of the following:
   - Click when the desired number of copies is displayed.
   - Press Enter.
   - **You are prompted**: Enter number of repeats <1>:

     Type the desired number of copies, then press Enter.

15.14.3 Using the Array option

1 Execute steps 1-4 of the general procedure (see above).

1 **You are prompted**: Enter second point or [Array] <Use base point as displacement>:

2 Choose the **Array** option.

You are prompted: Enter number of items to array:

3 Specify the number of items.

4 The array is dynamically displayed.

You are prompted: Enter second point or [Fit]:

5 Do one of the following:
   - Specify a point to define the distance between each item in the array.
   - Choose the **Fit** option.

     You are prompted: Enter second point or [Array]:

     Specify a point to define the distance between the first and the last item in the array.

6 Do one of the following:
   - Keep creating copies.
   - Press Enter to stop.

15.14.4 Copying a selection as a block

1 Select the entities you want to copy as a block.

2 Launch the COPYBASE command.

3 **You are prompted**: Select base point:

4 Specify a point.
5 xx selected
6 (option) Switch to another drawing.
7 Launch the PASTEBLOCK command.
8 **You are prompted**: Select insertion point:
9 Specify a point.
10 The selection is pasted as a block.
11 (option) Launch the EXPBLOCKS command and rename the block in the **Drawing Explorer / Blocks** dialog box.

15.14.5 Copying entities between drawings

Entities can be copied between drawings using the following methods:

- COPYCLIP and PASTECLIP*: to copy the selection using a default base point.
- COPYCLIP and PASTEORIG: to copy the selection using the coordinates of the source drawing.
- COPYBASE and PASTECLIP*: to copy the selection using a user-defined base point.
- COPYBASE and PASTEBLOCK: to copy the selection as a block in the target drawing.

* Press Enter at the 'Specify insertion point' prompt to paste the selection using the coordinates of the source drawing (= equivalent of the PASTEORIG command).

15.15 Making parallel copies

The OFFSET command creates a copy of linear entities and aligns them parallel to the original entities at a specified distance. You can make parallel copies of arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays and infinite lines.

Depending on the selected options set by the SELECTIONMODES system variable, the OFFSET command accepts:

- an edge of a 3D solid if the **Select Edges** option is set.
- the outline of a 3D solid face if the **Select Faces** option is set.
- enclosed boundaries if the **Select Detected Boundaries** option is set.

Note: The OFFSETGAPTYPE settings variable controls how potential gaps between segments are treated when closed polylines are offset.

15.15.1 Making a parallel copy at a specified distance

1 Launch the OFFSET command.
1 **You are prompted**: Enter offset distance or [Through point/Erase/Layer] <Through point>:
2 Do one of the following:
   - Press Enter to accept the current distance.
   - Type a new distance in the Command line.

- Define a new distance by specifying two points.

You are prompted: Select entity/subentity or [Exit/selection options (?)] <Exit>:

3 Select a linear entity.

4 A preview of the parallel copy is displayed depending on the current cursor position with respect to the selected entity.

You are prompted: Select side for parallel copy or [Both sides/Multiple]:

5 Click a point.

6 The parallel copy is created.

If OFFSETERASE = ON, the original entity is deleted.

7 **You are prompted**: Select entity/subentity or [Undo/Exit/selection options (?)] <Exit>:

8 Do one of the following:
- Select another entity.
- Press Enter to stop.

15.15.2 Making a parallel copy through a point

1 Launch the OFFSET command.

1 **You are prompted**: Enter offset distance or [Through point/Erase/Layer] <Through point>:

2 Do one of the following:
- Press Enter to accept the default **Through point** option (= when last used).
- Choose the **Through point** option.

You are prompted: Select entity/subentity or [Exit/selection options (?)] <Exit>:

3 Select a linear entity.

4 The parallel copy is dynamically displayed.

5 Click a point to create the parallel copy.

6 If OFFSETERASE = ON, the original entity is deleted.

You are prompted: Select entity/subentity or [Undo/Exit/selection options (?)] <Exit>:

7 Do one of the following:
- Select another entity.
- Press Enter to stop.

15.16 Arraying entities

The **ARRAY** commands copy a selection set in a rectangular or polar (circular) pattern or along a linear entity. For a rectangular array, you must specify the number of rows and columns and also the spacing between subsequent rows and columns. To create a polar array, you will be prompted to specify the center point of the array, the rotation step and the number of items in the array or the angle to fill. You can choose to rotate the selection set around the center point or to keep its original orientation.

The ARRAYCLASSIC command creates a static polar or rectangular array of entities through a dialog box.

The -ARRAY command creates a static polar or rectangular array of entities.

The 3DARRAY command constructs static 3D rectangular arrays and rotated polar arrays.

The ARRAY command distributes entity copies in an evenly spaced associative rectangular array (ARRAYRECT command), a polar array (ARRAYPOLAR command) or a path array (ARRAYPATH command) array.

The ARRAYEDIT command allows you to edit associative arrays and their source entities.

15.16.1 Creating a static rectangular array

1 Launch the ARRAYCLASSIC command.

1 The **Array** dialog box appears.

2 Under **Array Type**, choose **Rectangular**.

3 Click the **Select Entities** button.

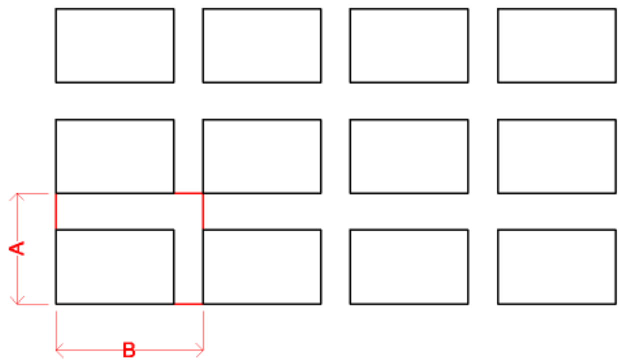
The **Array** dialog box closes temporarily to let you select entities.

4 Select the entities, then right-click or press Enter.

5 The **Array** dialog box reopens.

6 Fill out the **Settings** fields or press the **Pick Points** buttons to define the settings graphically.

The **Array** dialog box closes temporarily to let you pick a point in the drawing.



Row offset (A) and column offset (B) can be defined by the spacing rectangle (red).

7 Click the **OK** button to create the array.

Note: Positive values in step 5 are measured along the positive X- and Y-axis of the current UCS. Negative values are measured in the opposite direction.

15.16.2 Creating a static polar array

1 Launch the ARRAYCLASSIC command.

1 The **Array** dialog box appears.

2 Under **Array Type**, choose **Polar**.

3 Click the **Select Entities** button.

The **Array** dialog box closes temporarily to let you select entities.

4 Select the entities, then right-click or press Enter.

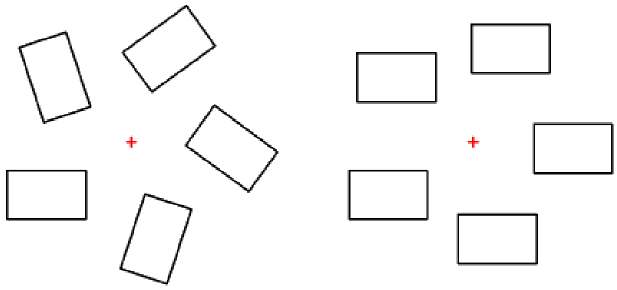
5 The **Array** dialog box reopens.

6 Choose a **Calculate** option: **Angle between**, **Angle to fill** or **Number of items**.

7 Fill out the **Settings** fields or press the **Pick Points** buttons to define the settings graphically.

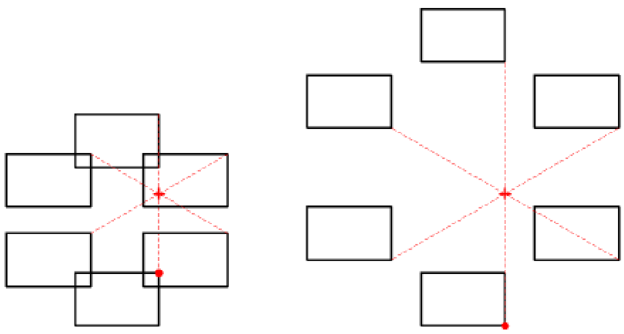
The **Array** dialog box closes temporarily to let you pick a point in the drawing.

8 (option) Check the **Rotate items as copied** option.



Entities rotated (left) or not (right) about the center point of the array.

9 (option) Define an **Entity base point**.



Polar array with base point (dot) and center point (cross).

10 Click the **OK** button to create the array.

15.16.3 Creating a static 3D rectangular array

1 Launch the 3DARRAY command.

1 **You are prompted**: Select entities to array [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to array [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Type of array [Polar/Rectangular] <Rectangular>:

6 Choose the **Rectangular** option.

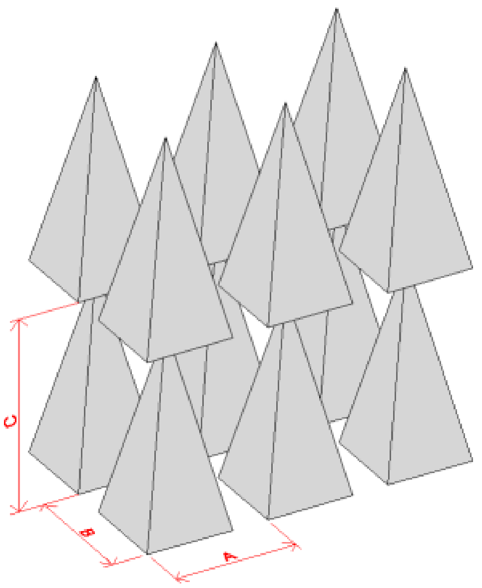
You are prompted: Number of rows in the array <1>:

7 Specify the number or rows.

8 **You are prompted**: Number of columns <1>:

9 Specify the number of columns.

10 **You are prompted**: Number of levels <1>:



Rectangular 3D Array:

row spacing A, column spacing B, level spacing C

11 Specify the number of levels.

12 **You are prompted**: Vertical distance between rows:

13 Type a value or click two points.

14 **You are prompted**: Horizontal distance between columns:

15 Type a value or click two points.

16 **You are prompted**: Depth between levels:

Type a value or click two points.

The array is created.

Note: Positive values in steps 7, 8 and 9 are measured along the positive X-, Y- and Z-axis of the current UCS. Negative values are measured in the opposite direction.

15.16.4 Creating a static 3D polar array

1 Launch the 3DARRAY command.

1 **You are prompted**: Select entities to array [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to array [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Type of array [Polar/Rectangular] <Rectangular>:

6 Choose the **Polar** option.

You are prompted: Number of items to array or [enter to specify Angle between items] <enter to specify Angle between items>:

7 Specify the number of items.

8 **You are prompted**: Angle to fill (+ for ccw, - for cw <360>):

9 Type a value or press Enter to accept the default.

10 **You are prompted**: Rotate entities around the array? [Yes/No] <Yes>:

11 Choose **Yes** or **No**, or press Enter to accept the default.

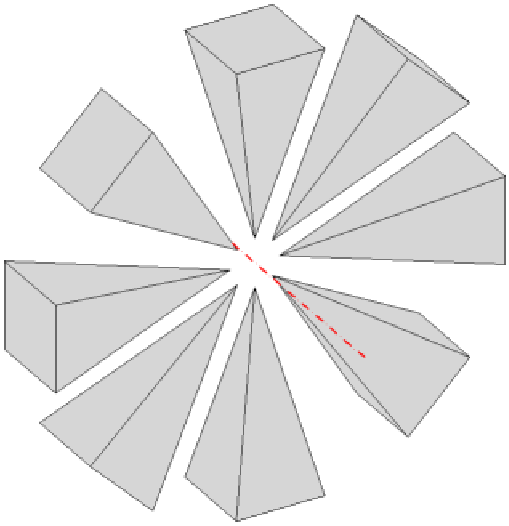
You are prompted: Center point of array:

12 Specify a point.

13 **You are prompted**: Specify second point along central axis of array:

14 Specify a point.

15 The array is created.



Polar 3D array around a horizontal axis (red).

15.17 Extending entities

The EXTEND command extends entities to a boundary, which is defined by one or more other entities.

If the EDGEMODE system variable is **On**, entities extend to an implied edge of the boundary entities.

If the boundary entity is not in the same plane as the entity you want to extend, the PROJMODE system variable lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS

- Project to the current view plane
- True 3D mode (No projection).

When extending entities, first select the boundary edges, and then specify the entities to extend, selecting them either one by one or using the fence selection method.

The following entities can be extended: arcs, lines, two-dimensional polylines, rays.

Boundary entities can be: arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, layout viewports.

15.17.1 Extending entities

1 Launch the EXTEND command.

1 **You are prompted**: Select boundary entities for extend [selection options (?)] <ENTER to select all>:

2 Select a boundary entity.

3 Entities in set: 1

You are prompted: Select boundary entities for extend [selection options (?)] <ENTER to select all>:

4 Select more boundary entities or press Enter to stop selecting.

5 **You are prompted**: Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/Projection/eRase]:

6 Select an entity:

7 The entity extends to the boundary entity.

You are prompted: Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/Projection/eRase/Undo]:

8 Select more entities or press Enter to stop.

Note:

- Entities which are selected when you launch the EXTEND command will be used as boundary entities.
- Hold down the Shift key when you select an entity to trim the line by the boundary entity.
- Use the Fence or Crossing selection method to select a series of entities simultaneously.
- If you select an entity near the end that cannot make an extension to one of the boundary entities, the EXTEND command is aborted.
- Use the **QUICK** mode where all objects automatically act as bounding edges.

15.18 Trimming entities

The TRIM command clips or trims entities by cutting entities.

If the EDGEMODE system variable is **On**, entities extend to an implied edge of the boundary entities.

If the boundary entity is not in the same plane as the entity you want to extend, the PROJMODE system variable lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS.
- Project to the current view plane.

- True 3D mode (No projection).

When trimming entities, you first select the cutting edges, and then specify the entities to trim, selecting them either one by one or using the fence selection method.

The following entities can be trimmed: lines, two- and three dimensional polylines, arcs, circles, ellipses, elliptical arcs, splines, rays and infinite lines.

Cutting entities can be: lines, splines, polylines, arcs, circles, elliptical arcs, ellipses, rays, infinite lines, layout viewports.

Entities in a block can be selected as cutting entities.

15.18.1 Trimming entities

1 Launch the TRIM command.

1 **You are prompted**: Select cutting entities for trim [selection options (?)] <ENTER to select all>:

2 Select a cutting entity.

3 Entities in set: 1

You are prompted: Select cutting entities for trim [selection options (?)] <ENTER to select all>:

4 Select more cutting entities or press Enter to stop selecting.

5 **You are prompted**: Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/Projection/eRase]:

6 Select an entity.

7 The entity is trimmed by the cutting entity.

You are prompted: Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/Projection/eRase/Undo]:

8 Select more entities or press Enter to stop.

Note:

- Entities which are selected when you launch the TRIM command will be used as cutting entities.
- Hold down the Shift key when you select an entity to extend the line to the cutting entity.
- Use the Fence or Crossing selection method to select a series of entities simultaneously.
- Use **QUICK** mode where all entities automatically act as trim edges.

15.19 Lengthening entities

The LENGTHEN command changes the length of lines, open polylines and arcs dynamically, incrementally or relatively. You can also modify the included angle of arcs.

15.19.1 Changing the length dynamically

1 Launch the LENGTHEN command.

1 **You are prompted**: Edit length: Select entity to list length or [DYnamic/Increment/Percent/Total]:

2 Choose the **Dynamic** option.

You are prompted: Select entity to change or [edit Mode]:

3 Click the entity near the endpoint you want to lengthen.

4 The entity is lengthened (shortened) dynamically.

You are prompted: New end point for line:

5 Specify a point to define the new length.

6 **You are prompted**: Select entity to change or [edit Mode/Undo]:

7 Do one of the following:

- Edit another entity.
- Press Enter to stop.

Note: The **Geometry** properties in the **Properties** panel allow you to edit the total length of lines and arcs and the start and end angle of arcs.

15.20 Selection and grip settings

When you select entities in a drawing, the selection method and the display of selected entities (for example whether grips are displayed or not) is controlled by a number of settings.

15.20.1 Selection settings

Most of the selection settings are located in the **Selection settings** group of the **Program Options** settings category in the **Settings** dialog box.

Name	System Variable	Description
Highlight	HIGHLIGHT	Determines whether or not entities highlight on screen when selected.
Image highlight	IMAGEHLT	Determines whether the entire raster image highlights or only the raster image frame when selected.
Pick add	PICKADD	Controls whether subsequent selections replace the current selection set or add to it. **ON**: the selection set is extended if you select additional entities. Press and hold the Shift key to remove entities from the selection set. **OFF**: you cannot add entities to a selection set. The newly selected entity or entities replace the existing selection set. However, if you press and hold the Shift key, you can add entities. If you select an entity that was already selected while pressing the Shift key, this entity is removed from the selection set.

Name	System Variable	Description
Pick automatic	PICKAUTO	Controls automatic window selection (inside or crossing) when selecting entities. The value is stored as a bitcode using the sum of the selected options. negative: disables window selection, saving current value. 1 = Draws a selection window if the mouse is over a blank area of the drawing when clicked. 2 = Draws a selection window if the mouse is over an entity when dragged. 4 = Draws a lasso selection if the mouse is not over any entity when clicked.
Pick box	PICKBOX	Defines the size of the small square at the end of the selection cursor (). If you select an entity by clicking, the Pick Box must touch or overlap the entity. The default size of the Pick Box is 3.
Pick drag	PICKDRAG	Controls the method of drawing a selection window. **ON**: allows you to define a selection window by dragging: press and hold the left mouse button to define the first corner of the rectangle, then move the mouse to define the size of the selection window and release the mouse button to define the opposite corner. **OFF**: define the selection window by clicking two opposite corners.
Pick first	PICKFIRST	Controls whether you select objects before or after you issue a modification command. **ON**: allows you to compose a selection set first, then launch a modification command. **OFF**: first start the command, then compose the select entities.
Linetype gap selection	LTGAPSELECTION	Makes it possible to snap to gaps on non-continuous linetypes. **ON**: does not allow selection or snapping within gaps (legacy behavior). **OFF**: allows the selection or snapping within gaps.
Pick style	PICKSTYLE	Controls the selection of groups and associative hatches. 0 = Either group selection or associative hatch selection. 1 = Group selection: if a member of a selectable group is selected, all members of the group are selected. 2 = Associative hatch selection: the hatch and its boundary are selected, no matter what is picked: the hatch or the boundary (not supported yet).

Name	System Variable	Description
Match options for SELECTSIMILAR	SELECTSIMILARMODE	Controls which properties must match for an entity of the same type to be selected with the SELECTSIMILAR command.
Frame selection	FRAMESELECTION	Controls if the hidden frame of an image, underlay, clipped XRefs, or wipeout can be selected.
Nearest Distance	NEARESTDISTANCE	Controls nearest distance dimension between a pair of selected entities. 0 = Do not show distance dimension 1 = Show nearest distance dimension 2 = Show OX aligned nearest distance dimension 4 = Show OY aligned nearest distance dimension 8 = Show OZ aligned nearest distance dimension
Selection area	SELECTIONAREA	Controls the display of selection area effects.
Selection area opacity	SELECTIONAREAOPACITY	Controls the transparency of the selection area: the higher the value, the more opaque the selection area.
Crossing area color	CROSSINGAREACOLOR	Specifies the color for crossing selection areas.
Window area color	WINDOWAREACOLOR	Specifies the color for window selection areas.

Name	System Variable	Description
Selection preview display	SELECTION PREVIEW	Controls when entities highlight when the cursor hovers over them: 0 = Never. 1 = When no commands are active. 2 = When a command prompts for entity selection. 3 = Both, when no commands are active and when a command prompts for entity selection. ***Note:*** When the SELECTIONPREVIEW system variable is Off: • the **Display the Quad when the cursor hovers over an entity** option of the QUADDISPLAY system variable is ignored and the Quad is not displayed, • the ROLLOVERTIPS system variable is ignored and entity properties are not displayed (the Quad is not displayed), • the SELECTIONCYCLING system variable is ignored and no badge or selection dialog box is displayed (the Quad is not displayed).
Selection preview effect	PREVIEWEF FECT	Controls the appearance of highlighted entities: 0=dashed lines; 1=thickened lines; 2=dashed and thickened lines (not yet supported).
Selection preview filter	PREVIEWFI LTER	Controls which entity types are excluded from selection previewing. The options are: • Entities on locked layers • Entities in Xrefs • Tables • MText entities • Hatches • Groups
GL Selection Highlight Style	HIGHLIGHT EFFECT	Specifies the highlight method: 0 = Dashed lines. 1 = Use different color. 2 = Use thickened line. 3 = Use different color and thickened line.
Selection Highlight color	HIGHLIGHT COLOR	Specifies the highlight color to be used when HIGHLIGHTEFFECT is 1 or 3.

Name	System Variable	Description
Selection cycling	SELECTION CYCLING	Controls the display options associated with overlapping objects and selection cycling. Negative = Disable selection cycling, saves earlier value 0 = Disable selection cycling 1 = A badge displays when you hover over objects that overlap 2 = Both a badge and a Selection dialog box displays
Selection Modes	SELECTION MODES	Defines which subentities should highlight in selection preview. The variable is stored as a bitcode using the sum of the values of all selected options. Click the buttons on the **Selection Modes** toolbar to set the value of SELECTIONMODES. A pressed button indicates the option is selected. 1 = Enable detection of 3D solid edges () 2 = Enable detection of 3D solid faces ()* 4 = Enable boundary detection (). Controls whether closed boundaries in XY-plane of the current coordinate system or on the face of 3D solids highlight when no commands are active. The BOUNDARYCOLOR system variable specifies the color for highlighting detected boundaries. * Hold the Ctrl key to invert the status of the **Select Faces** option: if ON it is possible to select 3D solids, if OFF it is possible to select solid faces. Repeatedly pressing the Tab key while keeping the cursor at the same position highlights all included subentity types one by one. See also Window Selection.
Preview selection delay	PREVIEWD ELAY	Specifies the number of milliseconds to wait before highlighting the (sub)entities under the cursor.
Constrai nt Bar Display	CONSTRAI NTBARDISP LAY	Controls the visibility of a geometric constraint at its creation or selection.
Dynamic Constrai nt Mode	DYNCONST RAINTMOD E	Controls the visibility of hidden dimensional constraints when the constrained entity is selected. When the cursor is over an entity with a dimensional constraint, the blue constraint glyph is displayed.

15.20.2 Grips settings

If the GRIPS system variable is different from zero, grips are displayed on selected entities.

Grips settings are located in **Display/Viewing** in the **Drafting** settings group of the **Drawing** settings category in the **Settings** dialog box.

Grips can be used to manipulate entities (see the **Grips editing** article).

Name	System Variable	Description
Grips	GRIPS	The value of GRIPS can be 0, 1 or 2. 0= Turn off grips. 1= Turn on endpoint grips. 2= Turn on endpoint and midpoint grips
Grips in blocks	GRIPBLOCK	Toggles the display of grips on entities in blocks on/off.
Grip color	GRIPCOLOR	Sets the color of unselected grips. Default value is green (index color no. 72).
Selected grip color	GRIPHOT	Sets the color of selected grips, so called *hot grips*. Default value is red (index color no. 240).
Hover grip color	GRIPHOVER	Sets the color of an unselected grip when the cursor pauses over it. Default value is blue (index color no. 150).
Grip size	GRIPSIZE	Sets the size of the grip box, expressed in pixels. Default value = 4.
Grip object limit	GRIPOBJLIMIT	Suppresses the display of grips if the number of selected objects exceeds the value of GRIPOBJLIMIT. If set to 0 (zero), all grips are displayed.
Grip tips	GRIPTIPS	(Not supported yet) Determines whether grip tips are displayed when the cursor hovers over grips on custom entities or dynamic blocks that support grip tips.
Grips attraction	ENABLEATTRACTION	Toggles the magnetism of grips on/off.
Grips attraction distance	ATTRACTIONDISTANCE	Sets the grips attraction distance: if the cursor is within this distance, the cursor is attracted by the grip and jumps to it.

15.21 Stretching entities

The size and shape of entities can be changed by stretching them. You select an area in your drawing using either a rectangular window or a polygon, then you specify a base point and a displacement point. All points and nodes inside the selected area will be moved over the specified distance. As a result, entities that cross the window or polygon boundary are stretched; those completely within the window or polygon are simply moved.

15.21.1 Stretching entities

1 Launch the STRETCH command.

1 **You are prompted**: Select entities to stretch by [Crossing window/Crossing Polygon/Remove/Add/selection options (?)]:

2 Use a crossing window (or crossing polygon) to specify the points that need to be moved.

3 Specify the bottom right corner of the crossing window.

A green rectangle is dynamically displayed.

You are prompted: Opposite Corner:

4 Specify the top left corner of the crossing window.

5 Entities in set: xx.

You are prompted: Select entities to stretch by [Crossing window/Crossing Polygon/Remove/Add/selection options (?)]:

6 Do one of the following:

- Use a crossing window (or crossing polygon) to specify the points that need to be added or removed from the selection.
- Right-click or press Enter to confirm the current selection.

You are prompted: Base point of displacement:

7 Specify a point.

8 The selected points move with the cursor.

You are prompted: Second point of displacement:

9 Specify a point to confirm the stretch distance.

15.22 Scaling entities

The SCALE command resizes a selection set in relation to a base point. You can specify the scale factor by selecting a base point and a length or by typing an explicit scale factor in the Command line. The scale factor can also be defined with respect to a base scale factor. E.g. when the base scale factor is 2 and the new scale is 3, the new size is 3/2 of the original. The base scale and the new scale can also be defined graphically in the drawing.

15.22.1 Scaling a selection set

1 Launch the SCALE command.

1 **You are prompted**: Select entities to scale [selection options (?)]:

2 Select entities.

3 Entities in set: xx

You are prompted: Select entities to scale [selection options (?)]:

4 Select more entities or press Enter to continue.

5 **You are prompted**: Base point:

6 Specify a point.

7 The base point is not moved in the scaling.

You are prompted: Scale factor or [Reference/Copy] <current scale*>:

8 Type a scale factor or press Enter to accept the previously used value.

* The previously used scale factor in a drawing is remembered.

15.22.2 Scaling a selection set using a reference scale

1 Repeat steps 1 through 4 of the procedure above.

1 **You are prompted**: Scale factor or [Reference/Copy] <current scale*>:

2 Choose the **Reference** option.

You are prompted: Reference scale <1>:

3 Click the base point again (see step 3 in the previous procedure) (1).

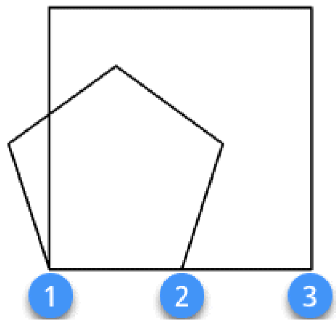
4 **You are prompted**: Second point:

5 Specify a point. (2)

6 The distance between the base point and the second point will be used as the reference scale.

The selection scales dynamically.

You are prompted: New scale factor or [Points] <2>:

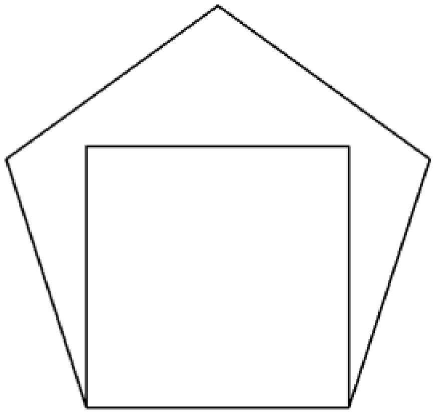


7 Do one of the following:

- Type a value.
- The distance between the reference points is set to the specified value.

The size of the selection set is scaled accordingly.

- Specify a point to position the second reference point (see step 4) (3).



15.23 Breaking entities

The BREAK command removes a portion of an entity, thus breaking it into two parts.

You can break arcs, circles, ellipses, lines, polylines, rays and infinite lines. Breaking a circle converts it to an arc. A ray is broken into a ray and a line, an infinite line is broken into two rays.

When breaking entities, you must specify two points for the break. By default, the point you use to select the entity becomes the first break point. However, you can use the **First** option to select a break point different from the one that selects the entity.

15.23.1 To break an entity

1 Launch the BREAK command.

1 **You are prompted**: Select entity to break [selection options (?)]:

2 Select the entity.

3 **You are prompted**: Select second breakpoint or [First break point/same as first point (@)]:

4 Select the **First** break point option.

You are prompted: First break point:

5 Specify a point.

6 **You are prompted**: Second break point:

7 Specify a point.

8 The part of the entity between the two break points is removed.

15.24 Joining entities

The JOIN command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.

When you join collinear lines, the farthest endpoints remain at their existing locations. BricsCAD draws a new line between these points.

The resulting entity type depends on the types of the input entities and their coplanarity.

15.24.1 General procedure

1 Launch the JOIN command.

1 **You are prompted**: Select source entity or multiple entities to join at once [selection options (?)]:

2 Select source entities, then right-click to stop selecting.

3 BricsCAD reports the result:

4 37 entities joined into 10 polylines, 4 entities discarded from the operation

15.24.2 Joining two arcs

Coplanar arcs, sharing midpoint and radius (= lie on the same circle), are joined counterclockwise, therefore the result depends on the selecting order.

1 Launch the JOIN command.

1 **You are prompted**: Select source entity or multiple entities to join at once [selection options (?)]:

2 Select the first arc.

3 **You are prompted**: Select entities to join [selection options (?)]:

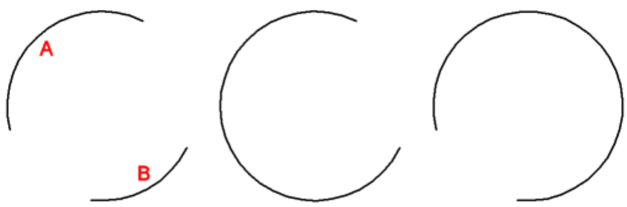
4 Select the second arc.

5 Entities in set: 1

6 **You are prompted**: Select entities to join [selection options (?)]:

7 Right-click.

8 BricsCAD reports: 2 arcs joined into 1 arc.



start situation (left) and result when selecting A first (middle) or B first (right).

15.25 Chamfering entities

The CHAMFER command connects two non-parallel entities by extending or trimming them and then joining them with a line to create a beveled edge.

In BricsCAD you can choose between two chamfer methods:

- **distance-distance**: specify how far to trim the entities back from their intersection
- **distance-angle**: specify the length of the chamfer and the angle it forms along the first entity.

The current chamfer method, distances and angle are saved through a series of system variables:

- CHAMMODE: sets the chamfer method
- CHAMFERA: defines the first distance
- CHAMFERB: defines the second distance
- CHAMFERC: defines the chamfer length
- CHAMFERD: defines the chamfer angle
- TRIMMODE: defines whether or not to keep the original length of the chamfered entities or to trim or lengthen them to the endpoints of the chamfer line.

You can set the values in the **Settings** dialog box under **Drawing** > **Drafting** > **Entity Modification** > **Chamfer/Fillet**. The **Settings...** option of the CHAMFER command allows you to edit the variables in the CHAMFER command procedure.

The following entities can be chamfered: lines, polylines, rays and infinite lines. When chamfering a polyline, you can create a chamfer between two polyline segments or you can chamfer all vertices of the polyline. When chamfering a polyline, the chamfer line is joined with the polyline segments if TRIMMODE=ON.

15.25.1 Chamfering using the distance-distance method

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Select the first entity.

4 **You are prompted**: Directional/Select second entity (select with pressed SHIFT to make corner) [Directional]:

5 Select the second entity.

6 The chamfer is created.

15.25.2 Chamfering using the length-angle method

1 Launch the CHAMFER command.

1 Chamfer (length=20, angle=45) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Select the first entity.

4 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

5 Select the second entity.

6 The chamfer is created.

15.25.3 Chamfering all vertices of a polyline

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Choose the **Polyline** option.

You are prompted: Select 2D polyline to chamfer:

4 Select the polyline.

5 The polyline is chamfered.

Note: When the chamfer method is **distance-angle**, the direction of the polyline defines which is the first entity of a vertex. Use the **Reverse direction** option of the PEDIT command if necessary to get the desired result.

15.25.4 Removing a chamfer

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 Select the first entity.

3 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

4 Hold down the Shift key, then select the second entity.

5 The chamfer is removed.

Note: You can use this method to create a corner between two lines or an end segment of a polyline. When the two entities intersect, the part you click is kept.

15.26 Filetting entities

The FILLET command connects two entities with an arc of a specified radius to create a rounded edge.

You can fillet pairs of line segments, straight polyline segments, arcs, circles, rays, and infinite lines. You can also fillet parallel lines, rays, and infinite lines.

When filleting a polyline, you can fillet multiple segments between two selected segments or you can fillet the entire polyline.

The TRIMMODE system variable defines whether or not to keep the original length of the filleted entities or to trim or lengthen them to the endpoints of the fillet arc.

15.26.1 Filleting two entities or polyline segments

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Choose the **Radius** option.

You are prompted: Fillet radius <current radius>:

3 Type a value.

4 **You are prompted**: Fillet (radius=new radius): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

5 Select the first entity or polyline segment.

6 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

7 Select the second entity or polyline segment.

8 The fillet is created.

15.26.2 Filleting all vertices of a polyline

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Choose the **Radius** option.

You are prompted: Fillet radius <current radius>:

3 Type a value.

4 **You are prompted**: Fillet (radius=new radius): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

5 Choose the **Polyline** option.

You are prompted: Select 2D polyline to fillet [selection options (?)]:

6 Select the polyline.

7 The polyline is filleted.

15.26.3 Filleting two parallel lines

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Select the first line.

3 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

4 Select the second line.

5 The fillet is created.

15.27 Editing polylines

Use the edit polyline segments tools in the **Quad** or use the PEDITEXT command to:

- Add a vertex.
- Add multiple vertices

- Add vertices at the start or end.
- Modify the bulge factor of a segment.
- Add or remove a bulge.
- Stretch a vertex or a segment.
- Delete multiple vertices or segments.
- Delete a segment or a vertex.

Use the PEDIT command to:

- Convert an entity into a polyline.
- Fit a curve to a polyline.
- Taper a polyline uniformly along its length.
- Reverse the direction of a polyline.

Edit a polyline in the **Properties** panel to:

- Open or close a polyline.
- Apply a uniform width to a polyline.
- Change the width of a polyline segment.
- Set the linetype mode (PLINEGEN system variable).

Use grips editing to:

- Move a vertex.
- Move a segment.
- Remove a vertex.

15.27.1 Using the edit polyline segments tools in the Quad

1 Place the cursor over the segment you want to edit without selecting the polyline.

1 The entire polyline is highlighted.

2 Choose one option from the **Modify** section of the **Quad**.

__Note:__ If you press Ctrl key, only the segment under the cursor is highlighted and one of its vertices is marked with a dot.

The editing tools, similar to the options of the PEDITEXT command, are available in the Quad:

- **Add vertex**: adds a vertex to the segment under the cursor.
- **Add vertex at end**: adds as a new vertex at the end of the polyline.

  This option is only available if the cursor is on one of the endpoints of a polyline.
- **Add Multiple vertices**: adds multiple vertices
- **Add bulge**: adds the bulge of the segment under the cursor.
- **Remove bulge**: removes the bulge of the segment under the cursor.
- **Remove segment**: deletes the segment under the cursor or the highlighted vertex.
- **Remove Multiple segments**: deletes the segments under the cursor or the highlighted vertex.

- **Stretch segment**: moves the segment under the cursor.
- **Stretch vertex**: moves the vertex closest to the cursor.

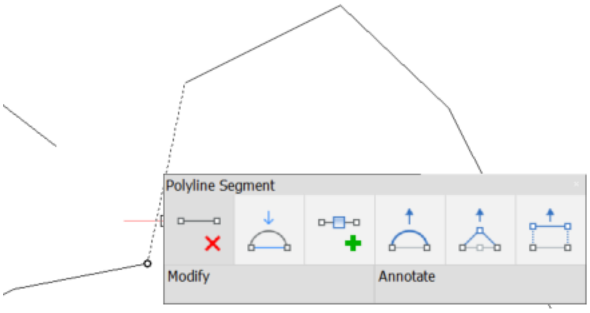
3 Select a tool, then follow the instructions in the Command line.

15.27.2 How to remove a vertex using Quad

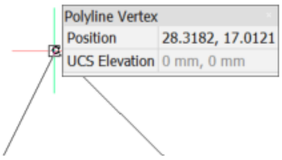
1 Place the cursor over a segment and hold down the **Ctrl** key to highlight it.

2 Activate the vertex by moving the cursor near the desired end of the segment. A dot will mark the active vertex.

3 Click the **Remove segment** button in the **Quad**.



Note: When you place the cursor over a vertex and hold down the **Ctrl** key, information about the vertex of the selected polyline is displayed.



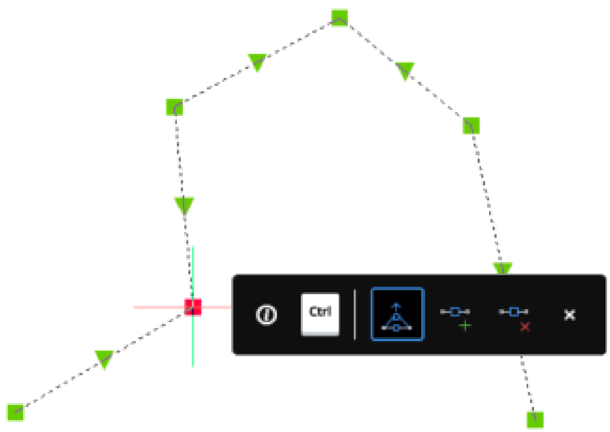
15.27.3 How to edit a vertex using Grips

Start or End vertex

1 Select a start or end vertex grip. It will be marked with red.

2 When the Hotkey Assistant is enabled, you can cycle through stretching, adding, and removing vertex by pressing the Ctrl key.

3

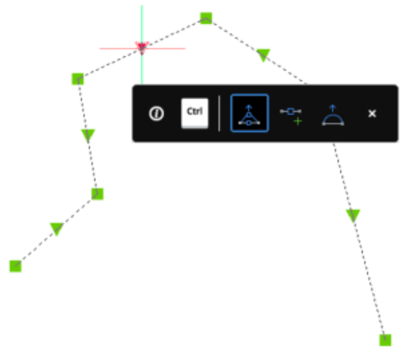


Middle vertex

1 Select the middle vertex grip. It will be marked with red.

2 When the Hotkey Assistant is enabled, you can cycle through stretching, adding vertex, and adding or removing bulge by pressing the Ctrl key.

3



15.27.4 Launching the PEDIT command

Do one of the following:

- Double-click a polyline.
- Click the **Edit Polyline** tool in the **Modify** menu.
- Click the **Edit Polyline** tool on the **Modify** toolbar.

15.27.5 Converting an entity into a polyline

1 Launch the PEDIT command.

2 Select the entity.

3 **You are prompted**: The selected entity is not a polyline. Turn it into one? [Yes-turn into polyline/No-leave as is] <Yes-turn into polyline>:

4 Press Enter or right-click.

5 **You are prompted**: Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse

direction/Spline/Taper/Width/Undo] <eXit>:

6 Choose an option or right-click to stop.

15.27.6 Fitting a curve to a polyline

1 Launch the PEDIT command.

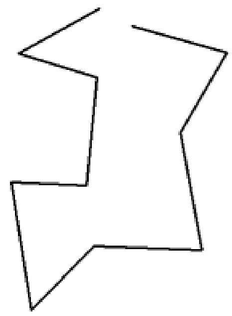
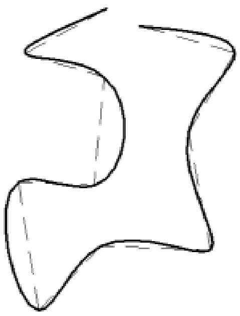
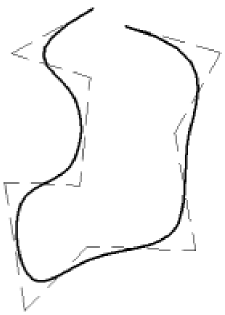
2 Select the polyline.

3 Do one of the following:

- Choose the **Fit** option.
- Choose the **Spline** option.

4 Press Enter or right-click to conclude the PEDIT command.

5

		
Original polyline	After applying Fit	After applying Spline

Note:

- Use the **Decurve** option to restore the original polyline.
- The SPLINESEGS (Spline Segments) system variable controls the number of segments (lines or arcs) to be generated for spline-fit polylines.

15.27.7 Tapering a polyline uniformly along its length

1 Launch the PEDIT command.

2 Select the polyline.

3 Choose the **Taper** option.

You are prompted: Enter new starting polyline width <current width>:

4 Type a value.

5 **You are prompted**: Enter new ending polyline width <current width>:

6 Type a value.

7 The polyline is tapered.

8 Press Enter or right-click to conclude the PEDIT command.

15.27.8 Reversing the direction of a polyline

1 Launch the PEDIT command.

2 Select the polyline.

3 Choose the **Reverse direction** option.

The direction is reversed.

4 Press Enter or right-click to conclude the PEDIT command.

15.27.9 Opening or closing a polyline

1 Select the polyline.

2 Select the **Closed** property under **Misc** in the **Properties** panel.

3 Choose **No** or **Yes**.

Or

1 Double-click the polyline.

2 Type **C** to close, or **O** to open.

15.27.10 Applying a uniform width to a polyline

1 Select the polyline.

2 Select the **Global width** property under **Geometry** in the **Properties** panel.

3 Type the desired width.

15.27.11 Changing the width of a polyline segment

1 Select the polyline.

2 Select the **Vertex** property under **Geometry** in the **Properties** panel.

An X mark is displayed at the start vertex of the polyline.

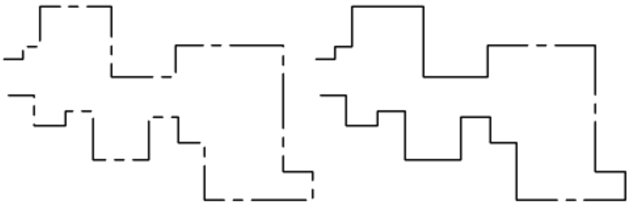
3 Click the arrow keys at the right hand side of the vertex property field to set the X marker at the start vertex of the segment you want to change the width of.

4 Select the **Start width** property and type the desired width.

5 Select the **End width** property and type the desired width.

15.27.12 Setting the linetype mode

The Linetype generation mode (PLINEGEN system variable) controls how a dashed linetype is applied to a multi-segment polyline. When disabled, dashed linetypes are applied per segment: start/end with a dash at each vertex.



PLINEGEN On (left) and Off (right)

1 Select the polyline.

2 Select the **Linetype generation** property under **Misc** in the **Properties** panel.

3 Select **disabled** or **enabled**.

15.28 Exploding entities

The EXPLODE command converts complex entities, such as blocks, polylines, solids or dimensions into their component parts.

Exploding a polyline or dimension reduces it to a collection of individual line and arc entities that you can then modify individually.

Blocks are converted to the individual entities, possibly including other, nested blocks that composed the original entity.

MText entities explode into single line texts.

In general, exploding entities will have no visible effect in the drawing, except for:

- If the original polyline had a width, the width information is lost when you explode it. The resulting lines and arcs follow the centerline of the original polyline.
- If you explode a block containing attributes, the attributes are lost, but the original attribute definitions remain.
- Colors and linetypes assigned BYBLOCK may appear different after exploding an entity, because they will adopt the default color and linetype.

The TXTEXP command explodes text entities into polylines. You can then use the JOIN command to create a single polyline for each letter. For some letters, such as a, e or p, a second polyline is needed for the ‘hole’).

15.28.1 Exploding entities

Do one of the following:

- Launch the EXPLODE command.
- Select entities, then press Enter.
- Select entities, then choose **Explode** in the **Modify** command group in the Quad.

15.29 Measuring entities

The MEASURE command places markers - points or blocks - at a specified interval along the length or circumference of an entity. The markers are placed starting at the closest endpoint to where you select the entity.

You can measure lines, polylines, arcs, circles, ellipses, elliptical arcs and splines. You cannot measure rays and infinite lines.

15.29.1 Measuring an entity using points

1 Launch the MEASURE command.

1 **You are prompted**: Select entity to measure:

2 Select an entity.

3 **You are prompted**: Segment length [insert Blocks]:

4 Do one of the following:

- Type a value.
- Specify two points.

The points are placed along the entity at the specified interval.

Note: The visibility of the points depends on the current value of the PDMODE system variable.

15.29.2 Placing blocks a specified interval along an entity

1 Launch the MEASURE command.

1 **You are prompted**: Select entity to measure:

2 Select an entity.

3 **You are prompted**: Segment length [insert Blocks]:

4 Choose the **Blocks** option.

You are prompted: Name of block to insert [? to list]:

5 (option) Press the ? key to see a list of available blocks in the drawing.

6 Type the name of a block.

7 **You are prompted**: Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-align blocks>:

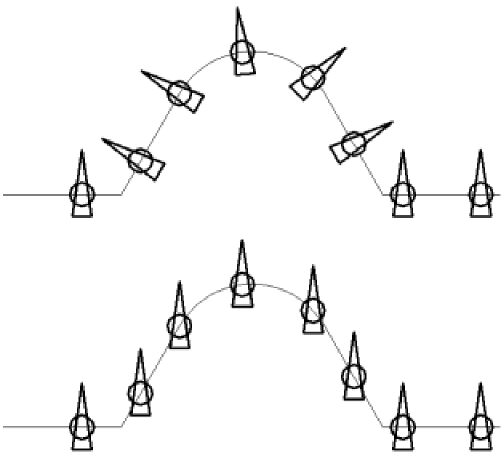
8 Choose whether or not to align the blocks.

9 **You are prompted**: Segment length:

10 Do one of the following:

- Type a value.
- Specify two points.

The blocks are placed along the entity at the specified interval.



Blocks aligned (top) or not aligned (bottom)

15.30 Dividing entities

The DIVIDE command places markers - points or blocks - along a selected entity. The markers divide the entity into a specified number of equal parts.

You can divide lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

15.30.1 Dividing an entity using points

1 Launch the DIVIDE command.

1 **You are prompted**: Select entity to divide:

2 Select an entity.

3 **You are prompted**: Number of segments [insert Blocks]:

4 Type a value, then press Enter.

5 The entity is divided by points.

Note: The visibility of the points depends on the current value of the PDMODE system variable.

15.30.2 Dividing an entity using blocks

1 Launch the DIVIDE command.

1 **You are prompted**: Select entity to divide:

2 Select an entity.

3 **You are prompted**: Number of segments [insert Blocks]:

4 Choose the **Blocks** option.

You are prompted: Name of block to insert [? to list]:

5 (option) Press the ? key to see a list of available blocks in the drawing.

6 Type the name of a block.

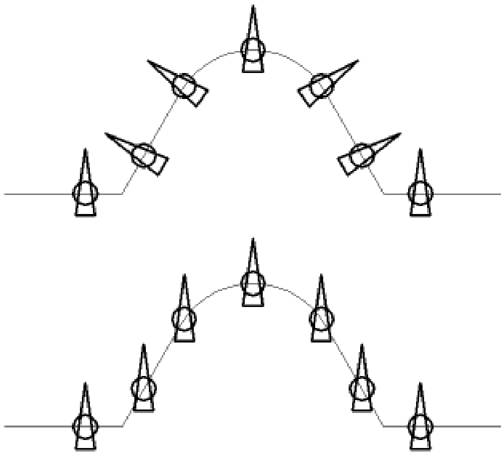
7 **You are prompted**: Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-align blocks>:

8 Choose whether or not to align the blocks.

9 **You are prompted**: Number of segments:

10 Type a value, then press Enter.

11 The entity is divided by blocks.



Blocks aligned (top) or not aligned (bottom)

16. Printing and plotting

16.1 General procedure for printing

The PLOT and PRINT commands plot drawings to printers and to files, through the **Print** dialog box.

The -PLOT command plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines.

The QPRINT command prints the drawing with the default plot configuration, without displaying the **Print** dialog box.

The PAGESETUP command creates and edits page setups for plotting drawings in the **Drawing Explorer** dialog box.

A standard print procedure consists of the following steps:

- Choose either a **Layout** tab or the **Model** tab.
- Open the **Print** dialog box.
- Select a **Page Setup name**. (1)

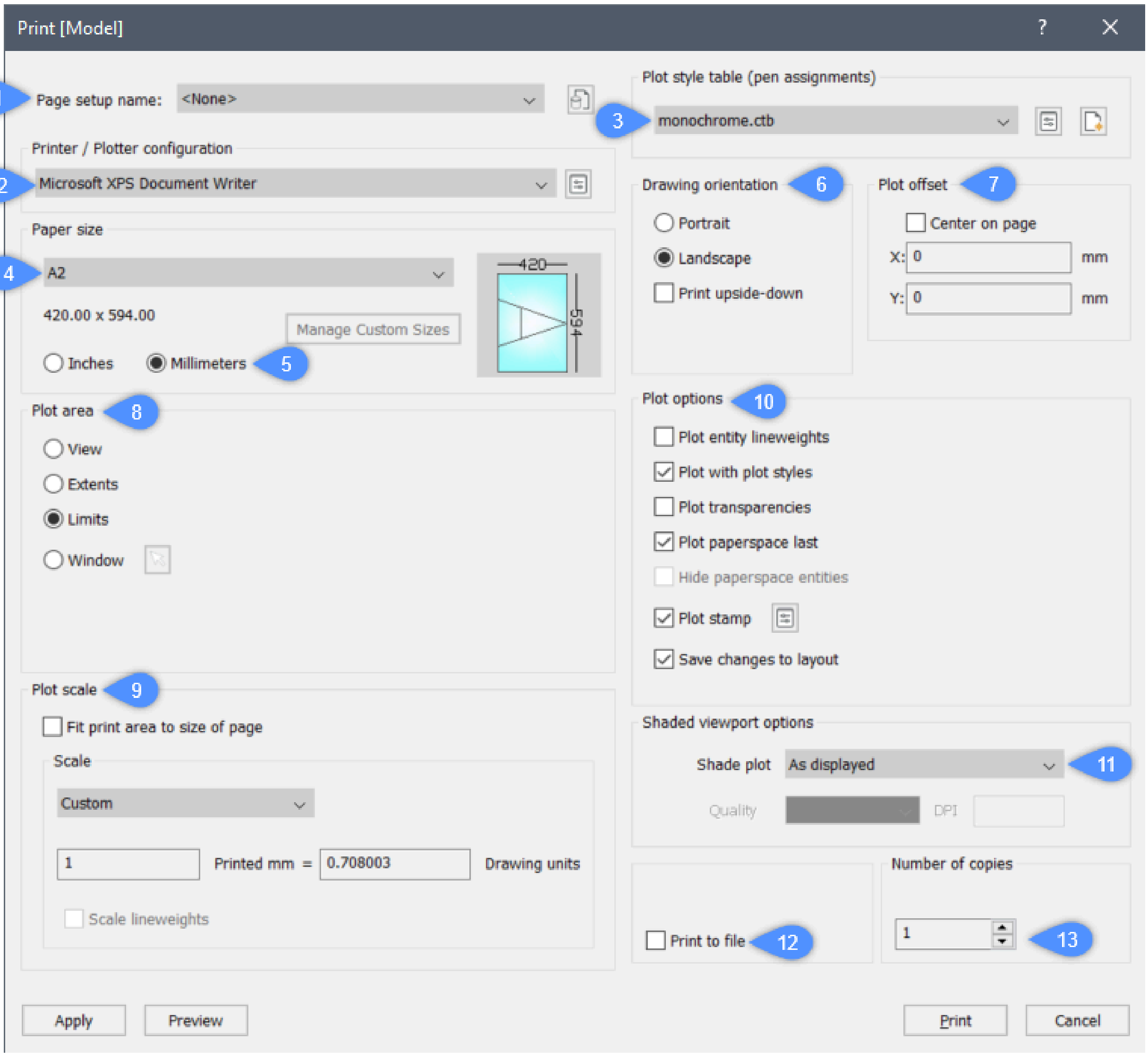
The following steps are optional and only necessary if you want to use overrides on the selected page setup definition:

- Choose a **Printer or Plotter configuration**. (2)
- Choose a **Plot style table**. (3)
- Choose a **Paper size**. (4)

  The image reflects the position and orientation of the drawing on the selected paper size.
- Choose the paper size unit. (5)
- Choose the **Drawing orientation**.(6)
- Define the **Plot offset**. (7)
- Specify the **Plot area** (the area to plot). (8)
- Define the **Plot scale**. (9)
- Check the **Plot options**. (10)
- When printing model space, choose a **Shade plot** mode. (11)

  This option is disabled when printing a paper space layout.
- Check/uncheck the **Print to file** option. (12)
- Specify the **Number of copies**. (13)
- Click the **Preview** button to see a preview of the plot output.
- When in a layout, click the **Apply** button to update the layout.
- Click the **Print** button to start printing.

Note: All print settings and options are saved in the **Model** and each of the **Layouts**.



Remember:

- On macOS and Linux it is not possible to print to system printers. It is only possible to print to the **Print As PDF.pc3** printer. So, a PDF document will be generated, which needs to be sent to the printer for physical printing.
- On macOS and Linux there is a relevant PAGESETUP command.

16.1.1 Opening the Print dialog box

Do one of the following:

- Launch the PRINT or PLOT command.
- Press **Ctrl + P**.
- Choose **Print...** in the **File** menu.
- Click the **Print...** tool in the **Output** > **PLOT** ribbon panel (Drafting workspace).

- Select the **Model** tab or a **Layout** tab, then right-click and choose **Print** in the context menu.

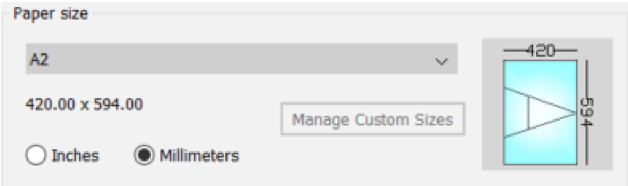
16.1.2 Choosing the paper size

Press the down arrow, then select the paper size in the **Paper Size** list.

The available sizes depend on the selected printer.

16.1.3 Choosing the paper unit

Select either **Inches** or **Millimeters**.



The paper size image indicates the size, position and orientation of the plot area on the selected paper size.

Note: The **Scale** and **Plot offset** setting units are changed according to the chosen **Paper unit**.

16.1.4 Choosing the paper orientation

Select either **Portrait** or **Landscape**.

- **Portrait**: The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- **Landscape**: The drawing or layout x-axis is aligned with the longest edge of the selected paper size.

The paper size image changes accordingly.

When in a layout, click the **Apply** button to update the display.

16.1.5 Defining the plot offset

Select either **Center on page** or type the plot offset in the **X:** and **Y:** fields.

The offset is calculated from the bottom left corner of the paper.

When in a layout, click the **Apply** button to update the display.

To specify the plot area:

Option	Result
View	Prints the portion of the drawing that is displayed in the current viewport or a saved view.
Extents	Prints all entities in the drawing.
Limits	Prints to the limits defined in the drawing. This option is only available when printing in Model space.

Optio n	Result
Layo ut	Prints the current layout. This option is only available when printing in Paper space.
Wind ow	Prints the portion of the drawing contained in a user defined window, maintaining the aspect ratio of the windowed area to the drawing. Click the **Select print area** button ( ), then specify the two corners of the print area in the drawing. or Type the X- and Y-coordinates of the corners of the print area in the **X:** and **Y:** fields.

16.1.6 Defining the plot scale

Note: The SCALELISTEDIT command allows you to edit the list of available scales in the **Print** dialog and the **Standard Scale** property of a paper space viewport.

... when printing a paper space layout:

If you print in paper space, the final scale of the plot output is the product of the viewport scale and the plot scale.

In order to keep control over the scale of the plot output:

- Specify the width and height of the viewports in drawing unit.
- Set the scale of each viewport to the scale you want in the plot output.
- The **plot scale** reflects the relationship between paper unit and drawing unit. (see table below)

Paper unit	Drawing unit	Plot scale	Printed mm or inches =	Drawing unit
mm	mm	1:1	1	1
mm	cm	10:1	10	1
mm	m	1000:1	1000	1
inches	inches	1:1	1	1
inches	feet	12:1	12	1

... when printing model space:

If you print in model space, the scale of the plot output equals the plot scale, on the condition that your drawing unit matches the paper unit. Otherwise you need to multiply the plot scale with respect to the relationship between the paper unit and the drawing unit. For example, if you want to plot at a 1/50 scale in a drawing of which 1 DU = 1 cm, the **plot scale** = 1/5, if 1 DU = 1 m the **plot scale** = 20/1 or 1/0.05

16.1.7 Setting the plot options

Click to toggle the **plot options** On / Off.

Option	Result
Plot object lineweights	If **Plot with plot styles** is off, object lineweights are plotted.
Plot with plot styles	Settings of the assigned STB or CTB file are used. ***Note:*** This applies to viewports which have the 2D Wireframe or Wireframe visual style only.
Plot transparencies	Plots entities using the value of the **Transparency** property.
Plot paperspace last	Entities in Model space are plotted first, paperspace entities are plotted last.
Hide paperspace entities	Removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.
Plot stamp	Adds a header and / or footer. Click the **Edit plot stamp button** () to define the plot stamp.
Save changes to layout	All changes made in the **Print** dialog are saved in the layout. This option toggles the **Save changes to layout** variable.

16.1.8 Setting the shade plot mode

By default the Shade Plot mode is set to **As displayed**.

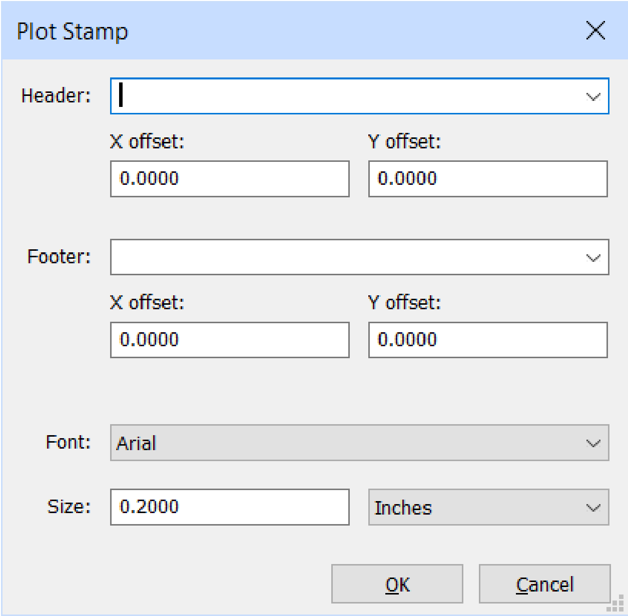
Click the list button, then select an option to plot using a different visual style.

Note: This option applies to model space printing only. The Shade Plot mode of a paper space viewport is defined in the Shade Plot property of the viewport.

16.1.9 Defining the plot stamp

1 Click the **Edit plot stamp** button.

The **Plot stamp** dialog box appears.



2 In the **Header** field, do one of the following:
   - Type the header text.
   - Click the down arrow button, then choose a variable in the drop down list.
   - Type one or more variable names (see table below).
   - Leave the field empty.
3 Define the X and Y offset for the header.
4 In the **Footer** field, do one of the following:
   - Type the footer text.
   - Click the down arrow button, then choose a variable in the drop down list.
   - Type one or more variable names (see table below).
   - Leave the field empty.
5 Define the X and Y offset for the footer.
6 Select a font in the **Font** list button.
7 Enter a value in the **Font Size** field.
8 Set the unit type to **Millimeters** or **Inches**.
9 Click the **OK** button to save the plot stamp.

Variable Name	Description
$Name	Adds the value assigned to the RegisteredOwner from the Windows registry. ***Note:*** User must add the new key as String Values into Windows registry: *Computer \HKEY_LOCAL_MACHINE\SOFTWARE\Bricsys\RegisteredOwner*

Variable Name	Description
$Company	Adds the value of the RegisteredOrganization from the Windows registry. ***Note:*** User must add the new key as String Values into Windows registry: Computer \HKEY_LOCAL_MACHINE\SOFTWARE\Bricsys\RegisteredOwner
$Date	Adds the current date.
$DateTime	Adds the current date and time.
$DwgName	Adds the file name and path of the drawing.

Note:

- The header and footer text consists of three parts (left, center and right), separated by commas.
- By default, the header and footer text is centered.
- The header and footer text can be outlined left center and/or right by separating the text with commas.
- For example: $Name,$Company,$Date places your name left, company name centered and the current date at the right.
- Type a space before one of the commas to leave one of the parts empty, thus outlining the header or footer left or right.
- For example: $Company, , ($Company,<space>,<space>) outlines the company name left. , ,$Company (<space>,<space>,$Company) outlines the company name right.
- The plot stamp settings are saved to a series of user preferences in the current user profile, which can be edited in the **Settings** dialog box under **Program Options** > **Plot and Publish** > **Plot Stamp**.

16.1.10 Specifying the number of copies

Do one of the following:

- Type the desired number in the **Number of copies** field.
- Click the up-arrow and the down-arrow buttons to specify the number of copies.

16.1.11 Previewing the plot output

Click the **Preview** button on the **Print** or **Page Setup** dialog box.

Choose **Print Preview...** in the **File** menu.

Type **preview** in the Command line, then press Enter.

16.2 Working with Shade plot

16.2.1 About

Shade plots allow you to choose a printing style for your plot.

A drawing containing shaded 3D entities can be plotted as it is displayed in a wireframe, with the hidden lines removed, or as rendered.

16.2.2 The difference between Visual Styles and Shade Plot

Visual Styles help control the display of entities, while Shade plots control the plot output.

16.2.3 General Procedure

Set the Shade plot settings of a Viewport:

1 Select the layout for which you want to make the changes.

2 Select the viewport and open the **Properties** panel.

3 In the **Misc** area set the required **Shade plot**.

Set the Shade plot setting in Model Space:

1 Go to the Model Space and make sure that no entities are selected.

2 Open the **Print** dialog box and select the page setup you want to modify.

3 Under **Shaded viewport options** section select the desired **Shade plot**.

4 Click **Apply**.

PDF files are displayed as vector graphics when:

- Shade plot is set to **Legacy wireframe** or **Legacy hidden**.
- Visual Style is set to **2DWireframe**, and Shade plot to **As displayed**.

The vector file format allows unlimited scalability without sacrificing the viewing quality.

PDF files are displayed as bitmap when:

Shade plot is set to **Conceptual**, **Hidden**, **Modeling**, **Realistic**, **Shaded**, **Shaded with edges**, **Shades of gray**, **Sketchy**, **Wireframe**, **X-Ray**, **BIM**, **Maquette**, **Render**, **See-through** or **Mechanical**.

Note: The available Shade plots depend on your drawing template.

The resolution for the bitmap format can be changed by modifying the **Image DPI** setting: open the **Plotter Configuration Editor** dialog box, go to the **Settings** tab, click **Custom Properties...** button, and under the **Output quality** section you can modify the **Image DPI** setting.

Note: For every new Visual Style created, a new Shade plot is automatically generated.

16.3 Plotter configuration

Configurations for Windows system printers are stored in plotter configuration files (.PC3 files). Unlike in AutoCAD®, PC3 plotter configuration files in BricsCAD apply to Windows system printers only.

In a plotter configuration file, you override one or more settings of the system printers installed on your computer. You can configure BricsCAD many plotting/printing devices and keep multiple configurations for each single device.

Each plotter configuration file contains information such as:

- The device driver and model
- The output port to which the device is connected
- Various device-specific settings (not supported on the Linux platform).

PC3 files are saved in the **Plotconfig** subfolder of your Roamable root folder.

After creating a PC3 file, it is available in the list of plotter configuration names in the **Printer / Plotter configuration** list on the **Print** or **Page Setup** dialog boxes.

16.3.1 Configuring the built-in PDF printer

1 Open the **PRINT** dialog box.

2 Click the **Printer/Plotter configuration** list button and choose **Print As PDF.pc3** in the list.

3 Click the **Edit Plotter Configuration** button.

The **Plotter Configuration Editor** dialog is displayed with the **Print As PDF.pc3** configuration file loaded.

4 Click the **Settings** tab on the **Plotter Configuration Editor** dialog.

5 (Option) Edit the **Graphics** settings.

6 (Option) Execute the **Scale Calibration**.

7 Click the **Custom Properties** button.

The **Print As PDF - Custom Properties** dialog is displayed.

8 (Option) Adjust the **Output quality** and **Layers support** settings.

9 Click the **Manage Custom sizes ...** button.

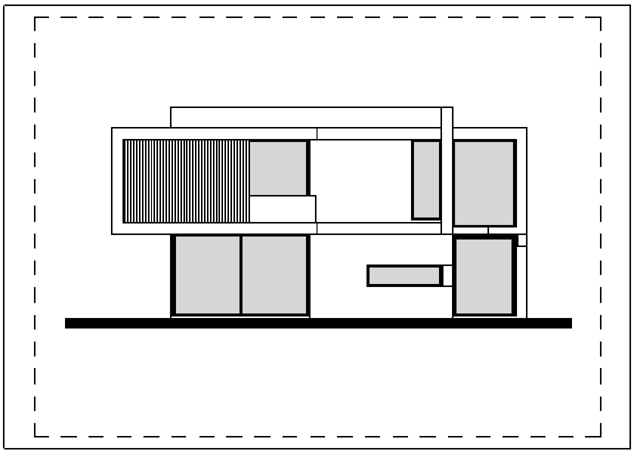
10 To create a custom paper size, do the following:

- Click the **Add** button.

  A new paper size is added.
- Double-click the newly created paper size.
- Type a name for the new paper size, then click the **OK** button.
- Define **Width** and **Height** for the new paper size.
- Define the **Paper Margins**.

  Paper margins define the printable area of paper sheet.

  If the DISPPAPERMARGINS system variable is ON, dashed lines indicate the paper margins on the paper sheet in a layout.



The paper sheet background is displayed if the DISPPAPERBKG system variable is ON.

- Click the **OK** button to create the custom paper size.

11 Click the **OK** button on each of the previously opened dialogs.

The custom paper size(s) can now be selected in the **Paper Size** list button.

16.3.2 Creating a plotter configuration

1 Launch the PLOTTERMANAGER command.

1 A **File** dialog box is displayed with the folder, defined by the PLOTCFGPATH system variable.

2 Double-click **Create a Plotter Configuration** shortcut.

The **Plotter Configuration Editor** dialog box is displayed.

3 In the **Printer name** list, select the printer you want to create a plotter configuration (PC3) for.

4 Click the **Settings** tab on the **Plotter Configuration Editor** dialog.

5 Click the **Custom Properties...** button.

The **<Selected Printer> Custom Properties** dialog box opens.

The layout of this dialog box is different for each printer.

Note: For **Print as PDF - Custom properties**, the **Create bookmarks** option is available to automatically generates bookmarks for named view in the PDF file.

Custom Properties are not supported on the Linux platform.

6 On the **Properties** dialog of the selected printer, adjust the printer settings, then click the **OK** button to confirm.

The **<Selected Printer> Custom Properties** dialog box closes.

7 Click the **OK** button on the **Plotter Configuration Editor** dialog.

The **Save Plotter Configuration As** file dialog appears.

8 (option) Edit the PC3 file name.

9 The default name is **<Selected Printer>.pc3**.

10 Click the **OK** button on the **Changes to a Printer Configuration File** dialog.

The PC3 file is created.

Note: You can create multiple plotter configurations for the same printer if you save them under different names.

16.3.3 Editing a plotter configuration

1 Launch the PRINT command.

1 The **Print** dialog box appears.

2 Select a plotter configuration (PC3) in the **Printer / Plotter configuration** list.

3 Click the **Edit Plotter Configuration** button.

The **Plotter configuration editor** dialog opens.

4 Proceed in the **Creating a plotter configuration** procedure above starting from step 4.

16.3.4 Assigning a plotter configuration

1 Choose **Model** or the **Layout** to which you want to assign a specific plotter configuration file.

2 Do one of the following:
- Choose **Page Setup** in the **File** menu.
- Type **pagesetup** in the Command line, then press Enter.

3 Choose one of the following from the **Printer / Plotter configuration** list:
- A system printer
- A configuration file (PC3 file)
- None (uses Default)

4 Click the **OK** button.

Note:

- When BricsCAD cannot find the plotter configuration file assigned to the drawing, it changes the configuration file assigned to your drawing to **None**.
- The **None** printer device has its own set of paper sizes that can be stored in the drawing, very much the same as for any 'real' printer. If a layout is opened and the stored printer is absent, the printer switches to **None** while the paper size is maintained. The missing printer and the previous paper size are indicated upon opening the **Print**dialog. The previous paper size can be saved in the drawing.
- PREVIOUS PAPER SIZE: drawings created by other parties are often set up for a printer that is not available in your office. In such case, BricsCAD resets the printer device to **None**, which results in using the default printer. Instead of resetting the paper size to a default, BricsCAD shows the Previous Paper size, in order to inform you about the intended paper size and allow you to choose a similar size on any of the available printers.

16.4 Plot style tables

The **Plot Style Table Editor** displays the plot styles contained in the specified plot style table.

16.4.1 Opening the Plot Style Table editor

Do one of the following:

- Click the **Edit Plot Style** button on the **Print** or **Page Setup** dialog box.
- Choose **Plotstyle Manager...** in the **File** > **Print** menu, then double-click a **CTB** or **STB** file.
- In a **Windows Explorer** window, double-click a **CTB** or **STB** file.

The **Plot Style Table Editor** dialog box includes the following tabs:

- The **General** tab
- The **Form View** tab
- The **Lineweights** tab

The **General** tab lists general information about the plot style table.

Name	Description
Plot style table file name	Displays the name of the plot style table file you are editing.
Description	Displays a description you want to include about the plot style table.
File info	Displays information about path, number of plot styles and the version number of the Plot Style Table Editor. ***Note:*** A color-dependent plot style table always contains 255 plot styles.
Apply global scale factor to non-ISO linetypes	Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.
Scale factor	Specifies the factor by which to scale non-ISO linetypes and fill patterns.

The **Form View** tab lists the plot styles contained in the plot style table and their settings. Plot styles are style overrides for your drawings that occur during plotting.

The first plot style in a **Named Plot Style** table is **NORMAL** and represents an entity's default properties (no plot style applied). You cannot modify or delete the **NORMAL** style.

Property	Description
Plot styles	Displays the names of plot styles contained in the selected plot style table. • Always contains 255 styles (1 for each color). • They are tied to entity color. • You cannot add nor delete a plot style. • You cannot rename a plot style. Named plot style table (STB): • Contains one or more plot styles. • You can add or remove plot styles, except for the **Normal** style, which cannot be deleted. • Plot styles can be renamed, except for the **Normal** style, which cannot be renamed.
Description	Description for the selected plot style.
Color	Specifies the plotted color for an entity. The default setting for plot style color is **Use Current**. If you assign a plot style color, the color overrides the entity's color in the plot output.
Dither	Depending on the capabilities of your plotter, dithering approximates the colors with dot patterns. When this option is not active, the colors are mapped to the nearest color, resulting in a smaller range of colors when plotting. Dithering is available whether you select the object's color or assign a plot style color. By default, this option is active. Setting it to Off has no effect in BricsCAD.
Grayscale	Depending on the capabilities of your plotter, converts the colors to grayscale in the plot output.
Physical Pen #	Specifies a pen to use when plotting entities that use this plot style. You can select a pen number from 1 to 32.

Property	Description
Virtual pen #	Specifies a virtual pen number between 1 and 255. Choose **Automatic** to specify that BricsCAD should make the virtual pen assignment from the BricsCAD Color Index. The virtual pen setting in a plot style is used only by non-pen plotters and only if they are configured for virtual pens (select **255 Virtual Pens** under **Color Depth** under **Vector Graphics** on the **Settings** tab in the **Plotter Configuration Editor**).
Screening	Specifies the color intensity of the plot on the paper. The valid range is 0 through 100. If you select 100, the drawing will be plotted with its full color intensity. In order for screening to work, the **Dither** option must be active.
Linetype	Displays a list with an example and a description of each linetype. The default setting for plot style linetype is **Use Object Linetype**. The selected linetype overrides the entity's linetype in the plot output.
Adaptive	Adjusts the scale of the linetype to complete the linetype pattern. You can activate this property if a complete linetype pattern is more important than a correct linetype scaling.
Lineweight	Displays a sample of the lineweight as well as its numeric value. You can modify an existing lineweight if the one you need is not available. The default setting for plot style lineweight is **Use object Lineweight**. The selected lineweight overrides the entity's lineweight in the plot output.

Property	Description
Line end style	Displays a list of several end styles for the lines. By default, the option **Use entity end style** is selected. If you select a line end style, the object's line end style will be overridden in the plot output.
Line join style	Displays a list of several join styles for the lines. By default, the option **Use entity join style** is selected. If you select a line join style, the object's line join style will be overridden in the plot output.
Fill style	Displays a list of several fill styles. By default, the option **Use entity fill style** is selected. If you select a fill style, the object's fill style will be overridden in the plot output.

Note: Entities which are created in **true color** are always plotted using their own object properties (color, lineweight, etc.).

The **Lineweights** tab lists the available lineweights.

Note: Lineweights with a tick are used in the **Lineweight** field on the **Form View** tab.

To edit a lineweight, select the lineweight in the list, then click the **Edit** button.

16.4.2 Assigning a plot style table

In BricsCAD, you can assign plot style tables to model space and to each of the layouts in a drawing. By assigning different plot style tables to each layout in your drawing, you can control how entities in the layout are plotted. The plot style table affects both model space and paper space entities. To plot the drawing without applying plot style properties, choose **None (use Default)** from the list of plot style tables.

In STB-type drawings, each entity in the drawing either is assigned a plot style explicitly or **By Layer**.

Plot style tables are used only if the **Plot with plot styles** plot option on the **Print** dialog box is ticked. When you turn off plot style tables, entities print according to their own properties.

Assigning a plot style table

1 Open the **Print** or **Page Setup** dialog box.

2 Select a **Plot style table** from the list box in the **Plot Style Table (pen assignments)** section.

3 Click the **OK** button.

16.5 The plot styles manager

The **Plot Styles Manager** is a window that lists plot style table files (CTB and STB files).

The Plot Styles Manager allows you to:

- Create Color-dependent Plot Style (CTB) and Named Plot Style files (STB) using the **Create a Plot Style Table** wizard.
- Edit existing Plot Style Table files.

16.5.1 Launching the plot styles manager

Do one of the following:

- Choose **Plotstyle manager...** in the **File** > **Print** menu.
- Launch the STYLESMANAGER command.

A **File** dialog box is displayed, showing the content of the PlotStyles folder, which is defined by the PLOTSTYLEFOLDER system variable.

16.5.2 Creating a new plot style table

1. Launch the Plot Styles Manager.
2. In the PlotStyles folder, double-click the **Create a Plot Style Table** shortcut. The **Add Plot Style Table Editor** dialog box opens.
3. Choose **Create new from scratch**.
4. Do one of the following:
    - Select either **CTB** or **STB** from the **Table type** selection button.
        - **CTB** (Color-dependent Plot Style Table) - bases plots on entity colors. Plotting parameters, such as pen thickness and linetype, are matched to the color of entities. This system is older and less flexible than named plot styles, but is the default in all new drawings. Creates a CTB file.
        - **STB** (Named Plot Style Table) - bases plots on plot styles. Plotting parameters can be given to every entity and every layer. This system is new and more flexible, but may be incompatible with other software that reads DWG files. Creates an STB file.
    - Choose **Create from existing table**, then choose a file in the drop down list box or click the browse button to select an existing CTB or STB file.
5. Type a name in the **Table name** field, then click the **OK** button.

   The **Plot Style Table Editor** dialog box is displayed.

16.5.3 Creating a new plot style table in the Print dialog box

1 Launch the PRINT command.

1 The **Print** dialog box appears.

2 Under **Plot style table**, click the **Create new plot style**.

The **Add Plot Style Table** dialog box appears.

3 Type a name in the **Table name** field, then click the **OK** button.

If the **Launch Editor** option is chosen, the **Plot Style Table Editor** dialog box appears.

If the **Use this plotstyle for the current drawing** option is chosen, the new plot style table is used in the current layout.

Note: The above procedure creates a CTB file in CTB-type drawing and a STB file in a STB-type drawing.

16.5.4 Editing a plot style table

1. Launch the Plot Styles Manager.
2. In the PlotStyles folder, double-click the plot style table you want to edit.
3. The **Plot Style Table Editor** dialog appears.

16.5.5 Editing a plot style table in the Print dialog box

1. Open the **Print** dialog box.
2. Select a plot style table from the **Plot Style Table** list button.
3. Click the **Edit plot style** button.

16.6 Batch plot

The PRINT command prints a single layout at a time. The PUBLISH command allows to print a sheet list (= a list of model space or paper space layouts) to a printer. A sheet list can be saved to a DSD (Drawing Set Description) file.

The SHEETSET command manages sheet sets and allows you to print, publish or etransmit a sheet set as a whole or a selection of sheets. A sheet set is saved to a DST file (Sheet Set Data) file.

17. Rendering

17.1 Using TwinMotion

Note: This feature is only available for BricsCAD on Windows.

Twinmotion offers real-time, immersive 3D architectural visualization using the power of Unreal Engine from Epic Games. All your BricsCAD model data are exported using the Unreal Engine file type (*.udatasmith) in the EXPORT command or synchronized with the DATASMITHSYNC command.

To use Twinmotion with BricsCAD you need to:

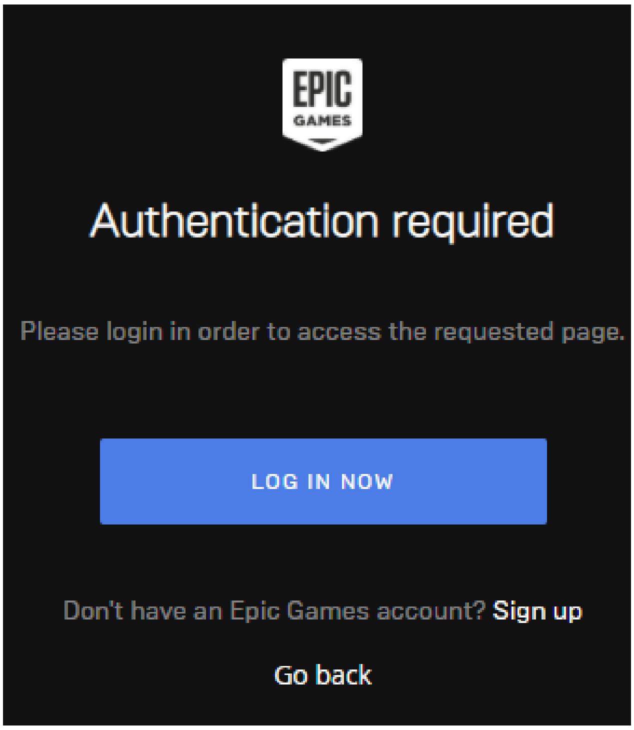
- Install the **Epic Games Launcher**.
- Download and install **Twinmotion**.
- Export your model to a UDATASMITH file and import this file to Twinmotion, or create a direct link between BricsCAD and Twinmotion.

17.1.1 Installing Twinmotion

From the Twinmotion Help pages:

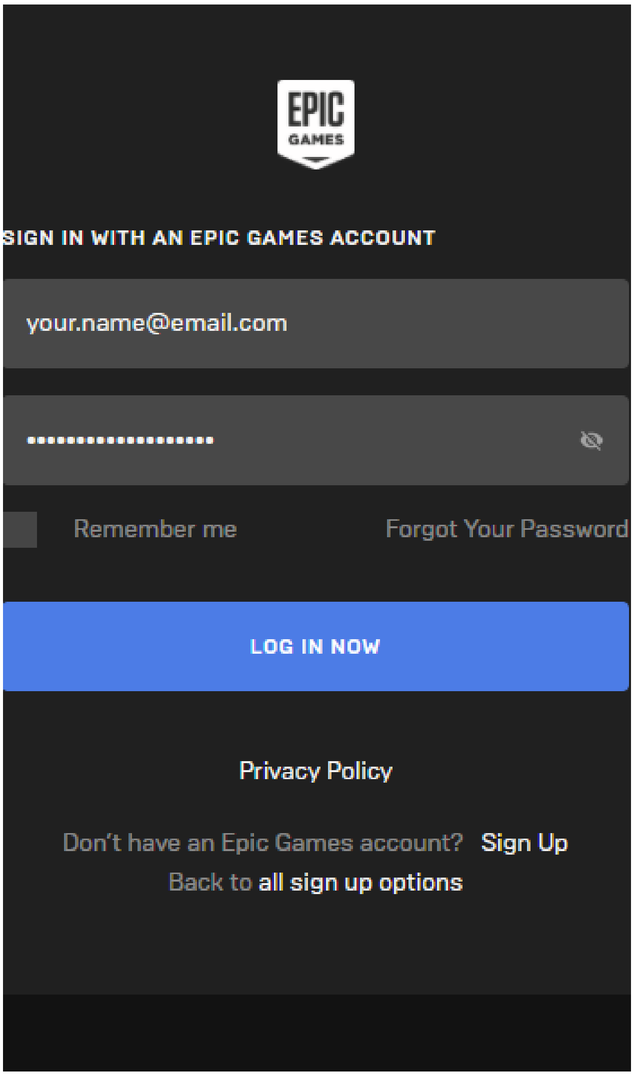
1 Go to https://www.unrealengine.com/en-US/twinmotion.

2 Click **Get Started Now**.



If you do not have an Epic Games account yet, click **Sign up**, then fill in all required information and click **Create Account**.

3 Click **LOG IN NOW**.



4 Once you are logged in, click **Download the Epic Games Launcher**.

5 Once the Epic Games Launcher is downloaded, make sure to click the **Unreal Engine** tab on the left side.

You will now see the **Twinmotion** tab in the top right.

6 Click **Install** to install Twinmotion.

7 Carefully choose your file location, it is strongly recommended to use the default location.

17.1.2 Exporting your model

1 Open your model in BricsCAD BIM.

2 Launch the DATASMITHEXPORT command.

3 The **Export drawing to** dialog box appears.

The **Save as type** is Unreal Engine (*.udatasmith).

4 Type a name or accept the default name, then click the **Save** button.

The exported model can now be imported in Twinmotion.

Note: The Site Location, Project Location, and Survey Location, as well as geographical coordinates, can be exported to Datasmith (*.udatasmith via the EXPORT or DATASMITHEXPORT commands). See the related article **Using TwinMotion**.

17.1.3 Importing your model in Twinmotion

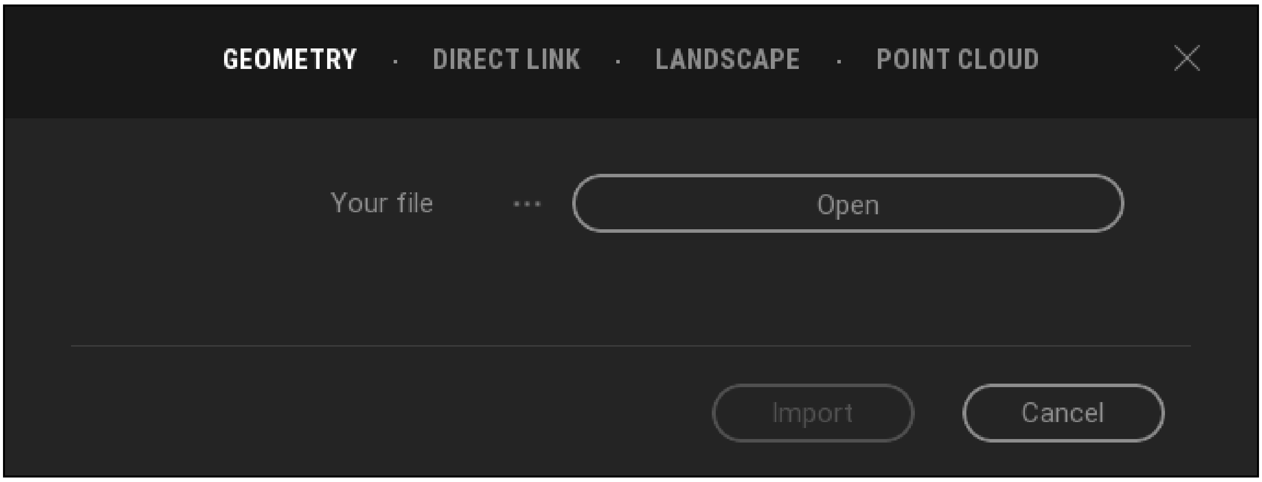
Note: This procedure requires:

- BricsCAD V21 or above, with Pro, BIM, Mechanical, or Ultimate license.
- Twinmotion 2020 or above.
- Optionally Unreal Engine 4.24 or above.

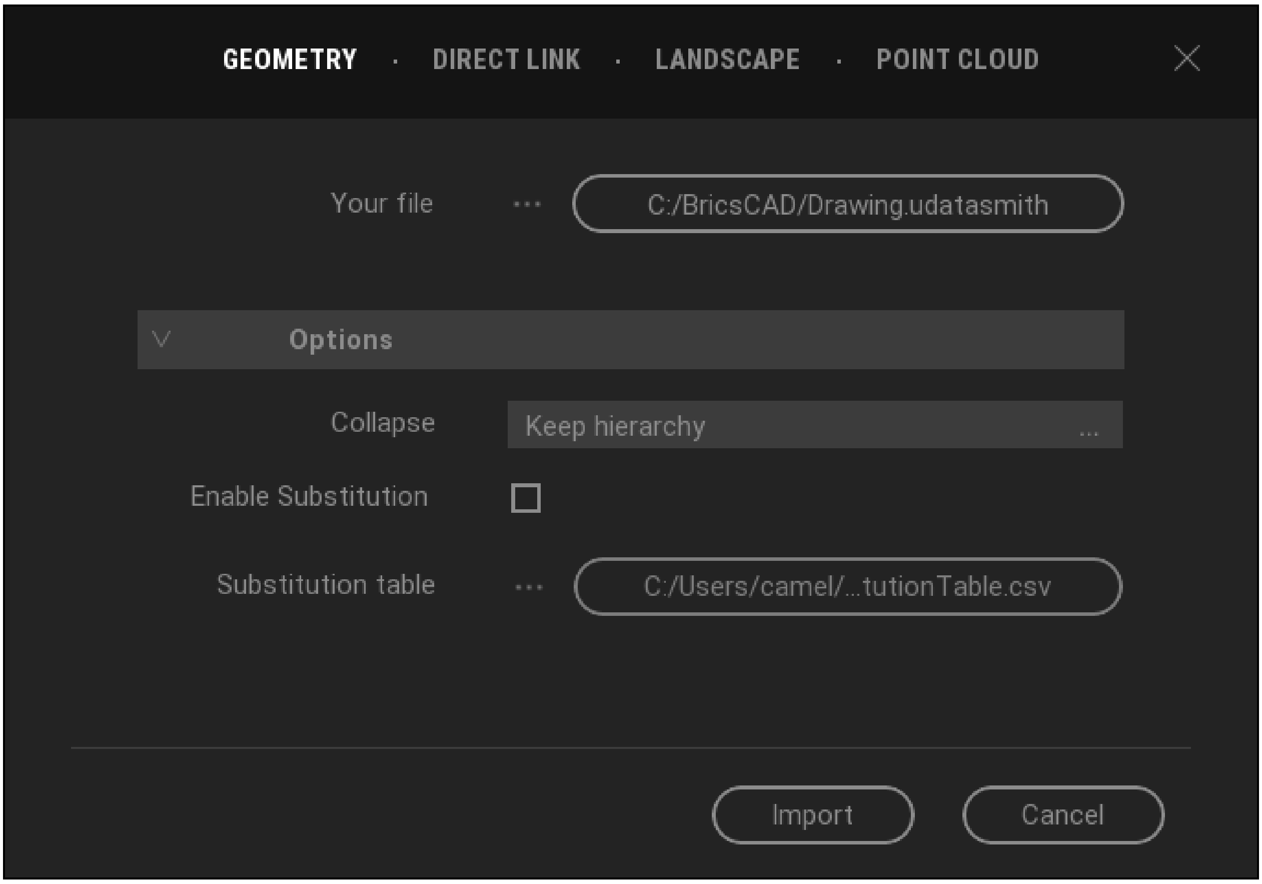
1 Launch Twinmotion.

2 Do one of the following:
  - Click the **+ Import** button.
  - Press **Ctrl + I**.

3 Make sure the **GEOMETRY** tab is selected, then click the **Open** button.



On the **Import** dialog, select a Unreal Engine (*.udatasmith) model file.



4 Click **Options** and set the **Collapse** option to **Keep Hierarchy**.

This structures the model the same way as the layers used in BricsCAD, or choose **Collapse by material** to merge all elements with same material as one object in Twinmotion.

5 Click the **Import** button to import the model.

17.1.4 Creating direct link with Twinmotion

Note: This procedure requires:

- BricsCAD V22 or above, with Pro, BIM, Mechanical, or Ultimate license.
- Twinmotion 2021 or above.
- Optionally Unreal Engine 4.26 or above.

1 Install Twinmotion 2021 or above.

1 ***Note:*** Direct link does not work with Twinmotion 2020.

2 In BricsCAD, run DATASMITHCONNECT or click the icon in the **View ribbon** tab to connect to Twinmotion.

Note: After establishing a connection to Twinmotion with Direct Link, the model gets synchronized the first time automatically.

3 In Twinmotion, on the **Import** panel, click the **+ Import** button.

4 In the dialog, go to the **DIRECT LINK** tab.

5 In the source list, your BricsCAD model should appear.

6 Choose **Collapse** > **Keep hierarchy** to keep the BricsCAD hierarchy, or choose **Collapse by material** to merge all elements with same material as one object in Twinmotion.

7 Click the **Import** button to import the model.

8 Whenever you want to update the model in Twinmotion, run DATASMITHSYNC in BricsCAD.

17.2 Rendering overview

Rendering is the process of generating an image from a 3D model. The model contains the geometry, viewpoint, texture, lighting and shading information. The result is a digital image which can be saved as a raster image file: BMP, JPEG or PNG.

Rendering in BricsCAD is based on the Redway3d rendering engine. By default the rendering engine will try to leverage the hardware on the graphics card. If experiencing problems, please switch off the RenderUsingHardware system variable. It is recommended to install the most recent drivers for your graphics card.

The RENDERPRESETS command allows you to create and edit render presets, and to set the current render preset via dialog box.

17.3 Rendering preferences

The RENDERPRESETS command allows you to create and edit render presets, and to set the current render preset through a dialog box.

17.3.1 Setting the render preferences

1 Open the **Settings** dialog box.

2 Under **Program Options**, expand the **Rendering** settings category.

Current material:

Specifies the material for new entities. By default (**ByLayer**), the **Global** material is used.

Lighting units:

- 0: No lighting units are used. Enables generic lighting.
- 1: Enables American lighting units: Foot-Candles
- 2: Enables International lighting units: Lux

Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 or 2.

Default lighting:

A distant light that follows the view direction. This setting can be different per viewport.

- 0: Default lighting is used if no lights are switched ON.
- 1: Default lighting is used instead of the light sources defined in the drawing.

Linear contrast:

Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light.

Linear brightness:

Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport.

Render using hardware:

Choose **User software only (slower)** if hardware rendering fails. It may be necessary to restart BricsCAD after changing the setting.

Real world scale:

Controls the rendering of materials with units set to real-world scale.

Sky status:

Specifies whether sky illumination is computed at render time. (Not implemented yet)

Texture map path:

Defines the search path for texture map images. In the BricsCAD program folder exist three subfolders under **Textures**, each containing a number of texture files of the same name. Images in folder 1 have a size of 256 x 256 pixels, folder 2 contains images of 512 x 512 pixels, images in folder 3 have a size of 1024 x 1024 pixels. If the **Diffuse map** setting of a material uses the image name only (not path), you can control the quality of a rendered image by setting the TextureMapPath user preference to folder 1, 2 or 3.

Tile mode light synch:

Controls the synchronization of lighting in all model space viewports. (Internal use only)

17.4 Rendering tools

Rendering tools are available in:

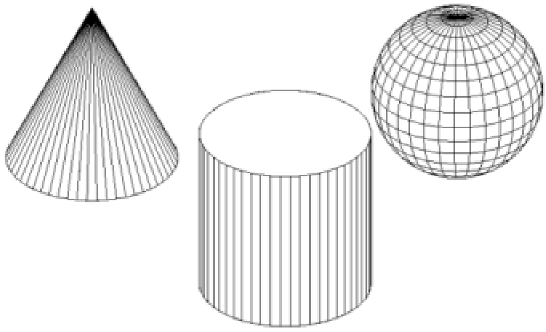
- The **Rendering** toolbar.
- In the **View** > **Rendering** menu.
- The **View** and **Visualize** ribbon tabs.
- In the **General** tab of the **No Selection** Quad.

Icon	Tool Name	Command	Description
	Render	RENDER	Photorealistic rendering using materials and lights

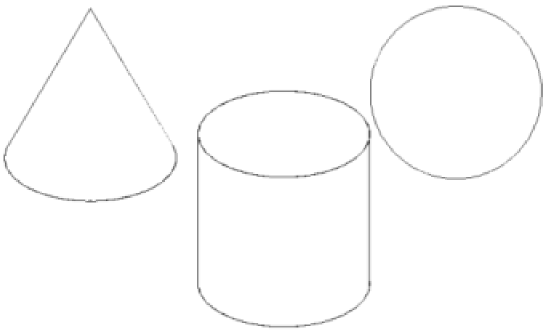
Icon	Tool Name	Command	Description
	Hide	HIDE	Hidden line view (*). If the previous shademode option was **2Dwireframe**, wireframe representation is restored if you zoom or pan. If the previous shademode option was one other than **2Dwireframe**, the hidden line representation is kept until a different shade mode is chosen.
	Shade	SHADE	Shades the entities between the polygon faces. Curved surfaces are segmented.
	2D Wireframe	SHADEMODE + 2	Wireframe representation
	3D Wireframe	SHADEMODE + 3	Wireframe representation
	Hidden	SHADEMODE + H	Hidden line view (*)
	Flat	SHADEMODE + F	Shades the entities between the polygon faces. Curved surfaces are segmented
	Gouraud	SHADEMODE + G	Shades the entities between the polygon faces. Curved surfaces are smoother and more realistic
	Flat + Edges	SHADEMODE + L	Combines flat shading and wireframe lines and curves
	Gouraud + Edges	SHADEMODE + O	Combines gouraud shading and wireframe lines and curves
	Materials	MATERIALS	Opens the **Drawing Explorer - Materials**
	Material Map	MATERIALMAP	Allows you to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.
	Light List	LIGHTLIST	Opens the Drawing Explorer - Lights
	Sun Properties	SUNPROPERTIES	Opens the Drawing Explorer - Lights

Icon	Tool Name	Command	Description
	Geographic Location	GEOGRAPHICLOCATION	Opens the **Geographic Location** dialog box.
	Animation Path	ANIPATH	Records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.
	Animation Editor	ANIMATIONEDITOROPEN	Opens the **Animation Editor** panel.

(*) Set the DISPSILH system variable On to display silhouette lines instead of a polygon mesh for curved surfaces.



Hidden line of a cone, a cylinder and a sphere with DISPSILH Off (top) and On (bottom).



17.5 Render materials

The MATERIALS command creates materials and edits their properties through the **Drawing Explorer**.

The MATBROWSEROPEN command opens the **Render materials** panel, which shows an overview of available high-resolution render materials.

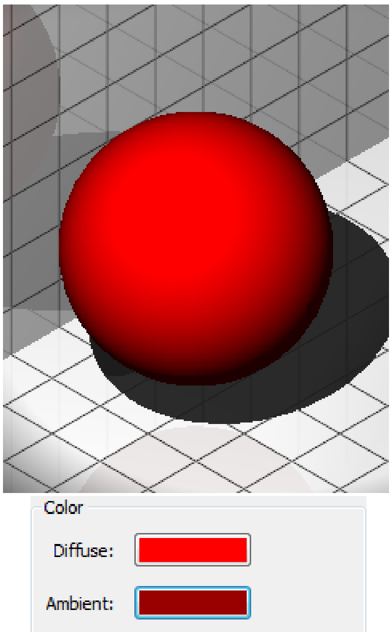
The MATBROWSERCLOSE command closes the **Render materials** panel.

The MATERIALMAP command allows you to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.

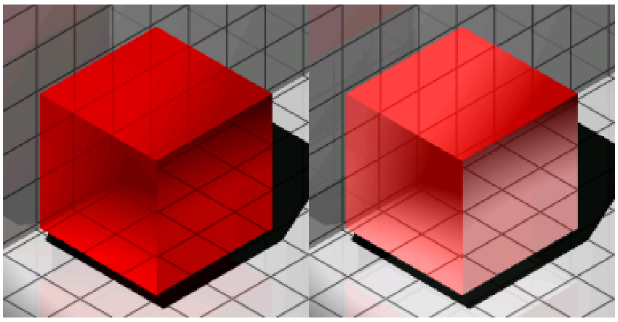
Material definitions are saved in the drawing. Each drawing contains the **Global** material, which cannot be deleted or renamed. The properties of the **Global** material can be edited.

17.5.1 Materials glossary

- **Diffuse color**: the color that the object reveals under pure direct, white light. It is perceived as the color of the object itself.
- **Ambient color**: the color of the object in the shadow, under ambient light rather than direct light.

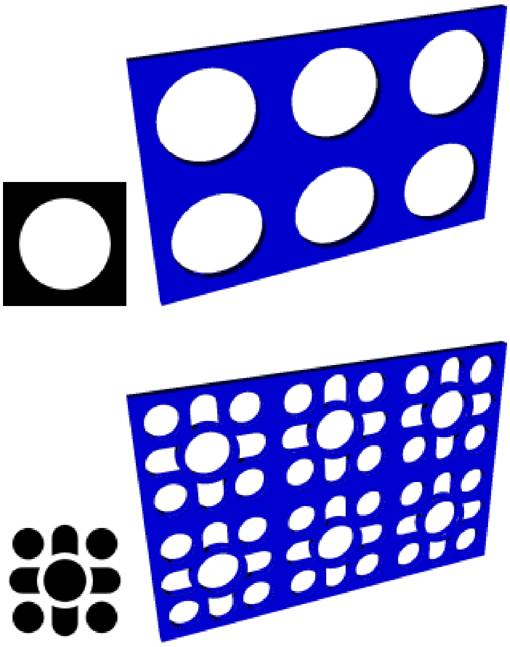


- **Highlights**: highly reflective highlights in the color of the material.

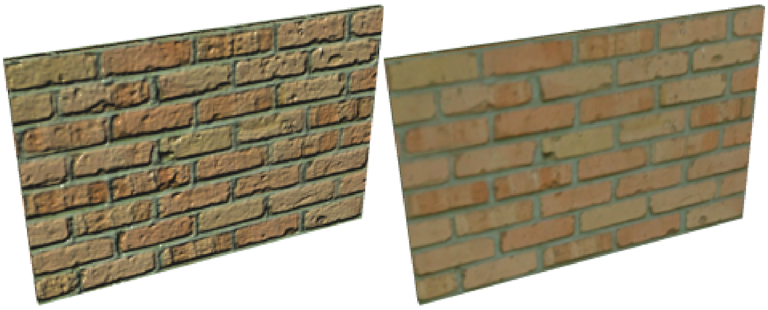


Metallic Highlights (left) and Non-metallic Highlights (right)

- **Shading**: how the color and brightness of a surface varies with lighting.
- **Texture-Mapping**: adds detail to a surface that is not included in the 3D model itself.
- **Diffuse map**: adds a structure or pattern to a surface to simulate a material.
- **Transparency map**: sets the transparency of a texture from 0% (black) to 100% (white) and anything between (grayscales). (Black=Transparency map)



- **Bump map**: simulates a certain roughness or bumpiness on surfaces. (left = with bump map)



- **Reflectivity**: the property of a surface to reflect light.
- **Transparency**: sharp transmission of light through solid objects.
- **Translucency**: scattered transmission of light through solid objects.
- **Refraction**: bending of light when passing through transparent objects.
- **Self-illumination**: it appears as if a surface is emitting its own light. No light is cast on other objects.

17.5.2 Defining render materials

See the MATERIALS command.

17.5.3 Adding a render material to the drawing

1 Select the material in the **Render Materials** panel.

2 Right-click and choose the **Add to drawing** option.

17.5.4 Adding your own materials

1 Download the textures you want to use.

Note: You can download a photo of your texture, but in order to get the most realistic view it is best to download a zip file that gives you different representations of the texture:

2 Launch the MATERIALS command.

3 The **Drawing Explorer – Materials** dialog box appears.

4 Click the **New** button.

5 Give your new material a name.

6 Check on the **Diffuse Map** and the **Bump Map**.

7 Edit the path of the diffuse map (...) to the colored image of your texture.

8 Edit the path of the bump map (...) to the normal image of your texture.

This will give your render material a more 3D and realistic look.

9 Edit the scale of your image until you get a realistic image in the preview.

10 The scale of the texture represents the number of drawing units covered by the full width of your image.

11 Right-click on your new material and choose **Add material(s) to library** in the context menu.

12 Close the **Drawing Explorer** dialog box.

13 Open the **Render Materials** panel and click on the folder **User Defined**.

You can find your new material in this folder of the library.

This folder is automatically generated under the following path: *C:\ProgramData\Bricsys\RenderMaterials\UserMaterials\User defined*.

17.5.5 Adding the Redway material library

See the Guide article **Adding the Redway material library**.

17.5.6 Assigning render materials

Materials can be assigned to a layer and to entities or subentities. If the **Material** property of an entity is **ByLayer**, the material assigned to the layer of the entity will be used.

By default, new entities get the **ByLayer** material setting.

17.5.7 Assigning a render material to entities

When the current visual style is **BIM**, **Modeling**, **Realistic** or **X-Ray**, entities and subentities are rendered using their material.

Method 1: Using the Properties panel

1 Select the entities or sub-entities (e.g. faces or 3D solids).

2 Under **3D Visualization** > **Material**, click the down arrow and select the material in the **Material** list.

Note: Materials in the drawing are available only.

Method 2: Using the Render Materials panel

1 Select the material, which launches the MATERIALASSIGN command.

1 The cursor turns into a paintbrush.

You are prompted: Select object to assign material to:

2 Click the entities or subentities to assign the material.

3 (option) Select a different material, then click more entities or subentities.

4 (option) Hold down the **Alt** key.

The cursor turns into a dropper.

You are prompted: Select material from object:

Click an entity or subentity to copy the material from, then click entities or subentities to assign the material to.

Note: The **Hotkey Assistant** appears at the bottom of the screen, indicating the current selection mode: **3D solids** or **3D solid faces**. Hold down the **Ctrl** key to invert the selection mode.

To assign a material to a selection set:

1 Select the entities.

2 Search the material in the **Render Materials** panel.

3 Click the material.

4 The material is assigned to all entities or subentities in the selection.

17.5.8 Assigning a material to a layer

1 Select the **Material** field of the layer in the **Layers** panel or the **Drawing Explorer – Layers** dialog box.

2 Click the down arrow and select the material in the list.

3 Materials in the drawing are available only.

17.5.9 Removing a render material

Assigning the **Global** material, removes a rendering material from an entity or subentity.

1 Select the entities and/or subentities.

2 Select the **3D Visualization / Material** field in the **Properties** panel and select the **Global** material.

Note: Clicking the **Erase Material** tile in the **Render Materials** panel sets the **Material** property to **ByLayer**.

17.6 Adding the Redway material library

Materials in the folder linked under **RENDERMATERIALSPATH** appear in the **Render** materials panel. To add the **Redway Material Library**, follow this procedure:

1 Download:

1 - the complete **Redway Material Library** here
 - materials based on **AmbientCG** here

2 Unzip the downloaded file and copy the path of the folder. The path should end with:

3 - *...\all_redway_materials*
 - *...\AmbientCG*

4 In the **Render Materials** panel, click the hamburger menu (⋮).

5 Select **Manage libraries** in the menu.

The **Settings** dialog box is displayed with the RENDERMATERIALSPATH system variable selected.

6 Click the Browse button (…).

The **Path List** dialog box appears.

7 Click the **Add Path** button.

An empty path field is added.

8 Do one of the following:
 - Paste the path you have copied in the empty path field.
 - Click the Browse button (…), then select the folder in the **Choose a folder** dialog box.

The downloaded materials appear in the **Render Materials** panel.

17.7 Lights

Lighting is the key to make the viewer believe to be looking at a realistic scene.

17.7.1 Lighting glossary

- **Ambient light**: light from an unspecified source.
- **Distant light**: light from an infinitely far source. This causes the complete scene to be lit from the same direction.
- **Fall-off**: the attenuation or decrease in brightness the further from the light source.
- **Point light**: light that shines in all directions from a central point.
- **Spot light**: focusable light that is aimed to one area.
- **Web light**: approximates real-world light distribution using a 3D representation of the light intensity. Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 (American lighting units) or 2 (International lighting units).
- **Attenuation**: the fall-off in brightness the further from the light source.

17.7.2 System variables

LIGHTINGUNITS (Lighting units):

- 0: Generic lighting - this obsolete option can no longer be set but may exist in older drawings.
- 1: Enables Photometric lighting using American units: Foot-Candles.
- 2: Enables Photometric lighting using International units: Lux.

DEFAULTLIGHTING (Default lighting): A distant light that follows the view direction. This setting can be different per viewport.

- 0: Default lighting is used if no other lights are switched ON.
- 1: Default lighting is used instead of the light sources defined in the drawing.

LINEARCONTRAST (Linear contrast): Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light.

LINEARBRIGHTNESS (Linear brightness): Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport.

LIGHTGLYPHDISPLAY (Light glyph display): Controls the display of light glyphs.

FBXEXPORTLIGHTS (Fbx Export Lights): Specifies whether to export lights from model space.

17.7.3 General properties

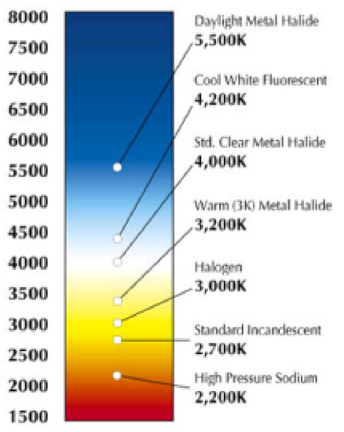
The general properties are common to all lights:

- **Name**: specifies a name for the light.
- **Type**: specifies the type of light: spot, point, web or distant.
- **On/Off**: specifies whether the light is turned on or off.
- **Shadows**: specifies whether the light casts shadows or not. Turning shadows off increases performance.
- **Intensity factor**: multiplies the **Lamp Intensity** property, the result is the **Resulting Intensity**.
- **Filter color**: specifies the color of the light.
- **Plot glyph**: if on, the light glyph is plotted.
- **Glyph display**: controls the display of the light glyph.

17.7.4 Photometric light properties

LIGHTINGUNITS = 1 (American units) or 2 (International units).

- **Lamp intensity**: defines the brightness of the light. The lamp intensity is expressed in candela (cd), which is the SI base unit of luminous intensity: the power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function).
- **Resulting intensity**: the product of the lamp intensity and and intensity factor.
- **Lamp color**: defines the inherent color of the light or color temperature in Kelvin or standards.

	D65White Fluorescent Cool White White Fluorescent Daylight Fluorescent Incandescent Xenon Halogen Quartz Metal Halide Mercury Phosphor Mercury High Pressure Sodium Low Pressure Sodium
color temperatures	lamp color list

- **Resulting color**: this is the final color of the light: a combination of the lamp color and the filter color.
- **Photometric web**: if the light type is **Web**, **Photometric Web** and **Web offsets** are available (not implemented yet).

17.7.5 Geometry properties

- **Position**: controls the location of the light.
- **Target**: defines the target point for point lights, spotlights and weblights.
- **Targeted**: switches the **target** property on/off.

17.7.6 Attenuation properties

In the real world, an object appears darker if it is farther away from the light source. The attenuation properties define how the light diminishes over distance. Attenuation does not apply to photometric lights.

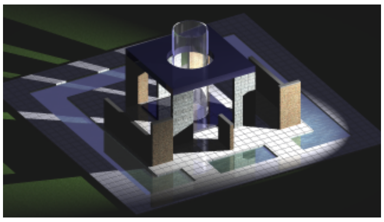
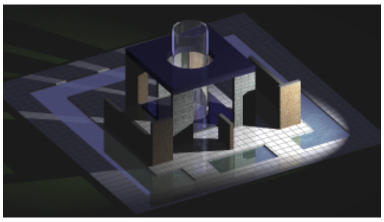
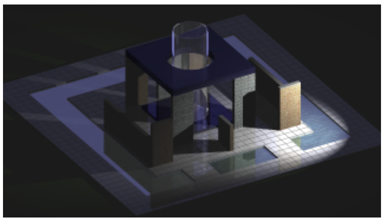
An alternative method to control the distance a light shines is the use of limits, which define the points from where a light starts to shine and where it stops. Using limits decreases the time needed to calculate the illumination of a scene.

Attenuation properties apply to spot lights and point lights.

- **Type**:
  - **None**: no attenuation. The distance to the light source has no influence.
  - **Inverse Linear** : the attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, the light is half as strong; at a distance of 4 units, the light is one quarter as strong.
  - **Inverse Square**: the attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, the light is one quarter as strong; at a distance of 4 units, the light is one sixteenth as strong.
- **Use Limits**: if this property is set to Yes, the **Start limit offset** and **End limit offset** fields become active.
  - **Start limit offset**: defines the point where the light starts to shine, measured from the center of

the light.

- **End limit offset**: defines the point where the light stops to shine, measured from the center of the light.

No attenuation	Inverse linear attenuation	Inverse square attenuation
		

Note:

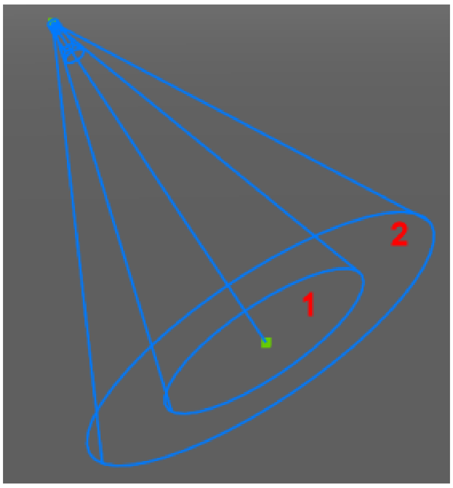
- The **Attenuation** properties apply to generic lights only (LIGHTINGUNITS = 0).
- Inverse square attenuation is applied automatically for photometric lights (LIGHTINGUNITS = 1 or 2).

17.7.7 Rendered shadow details

- Type: Sets the shadow type.
- - Sharp (raytraced): generates sharp edged shadows.
  - Shadow map (unsupported, using sharp): not supported
  - Map size: specifies the amount of memory for shadow calculation. Click the field, then select a value in the drop-down list (64/128/256/512/1024/2048/4096).
  - Softness: sets the softness of the shadow map. Enter a value between 1 and 10.
  - Soft (sampled): generates realistic shadows with soft edges.
  - Samples: sets the sampling size of the shadow
  - Visible in render: toggles the visibility in rendering
  - Shape: sets the shadow shape (linear. rectangle, disk, cylinder, sphere).

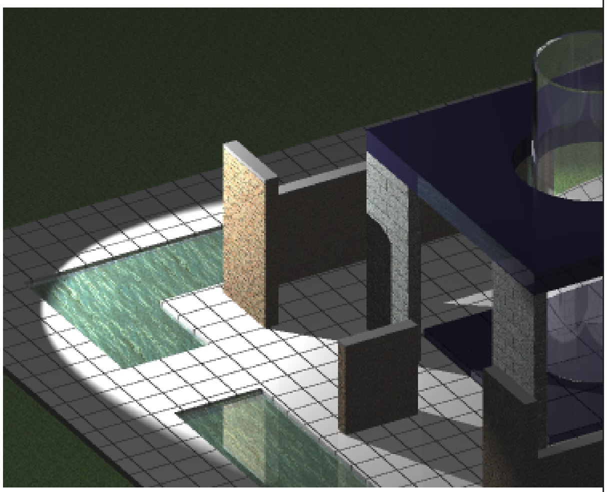
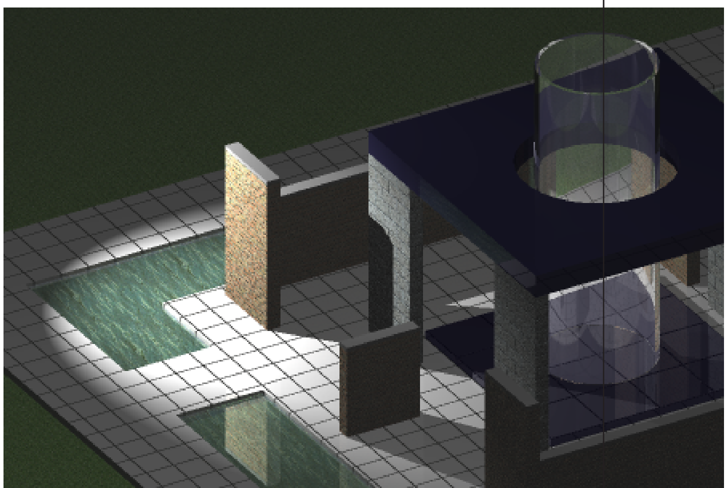
17.7.8 Spotlight hotspot and falloff properties

When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot = 1) that is surrounded by an area of lesser intensity (falloff = 2).



Hotspot angle: angle of the central light cone (defines the hotspot)

Falloff angle: angle of the full light cone

<table>
<tr><th>Hotspot angle = 55, Falloff angle = 60</th><th>Hotspot angle = 30, Falloff angle = 60</th></tr>
<tr><td></td><td></td></tr>
</table>

The difference between the hotspot angle and the falloff angle defines the area of lesser light intensity.

If the falloff angle and the hotspot angle are almost equal the edge of light cone is rather sharp (left).

The greater the difference between both angles, the softer the edge of the light cone (right).

17.7.9 Managing Lights

The LIGHTLIST command opens the **Drawing Explorer** > **Lights** dialog box.

Here you can:

- Create new lights.
- Delete lights.
- Select lights in the drawing.

- Switch lights On/Off.

17.7.10 Defining a light

1 Do one of the following:

1 - Launch the LIGHT command.

- Click the New ( ) button on the **Drawing Explorer** > **Lights** dialog box.

  You are prompted: Enter light type [Point/Spot/Web/Distant] <Web>:

  Specify the light type.

- Click a toolbutton on the **Lights** toolbar or the **View** or **Visualize** ribbon tab.
  - : spot light
  - : point light
  - : distant light
  - : web light

You are prompted: Specify source position <0,0,0>:

2 Specify a point in the drawing.

3 **You are prompted**: Specify target position <0,0,-10>:

4 Specify a point in the drawing.

5 **You are prompted**: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/Shadow/Attenuation/FilterColor/<eXit>:

6 Specify the properties of the light.

7 Choose **Exit** to conclude the command and create the light.

A light glyph is displayed in the model, if LIGHTGLYPHDISPLAY = On.

Note: It is not possible to define web lights if **LIGHTINGUNITS** =0 (No lighting units).

17.7.11 Editing lights

Lights can be edited in the **Properties** panel.

1 Do one of the following to select the light in the drawing:

- Click the glyph of the light(s).
- Select the light(s) in the **Drawing Explorer** > **Lights**, then right-click and choose **Select lights** in the drawing in the context menu.

The properties of the selected light(s) appear in the **Properties** panel:

2 Edit the properties.

3 Press the Esc key to stop.

17.7.12 Defining the sun properties

1 Launch the SUNPROPERTIES command.

1 The **Drawing Explorer** > **Lights** dialog box appears, showing the sun properties in the **Editor** panel.

2 Edit the properties.

Note: When LIGHTINGUNITS = 1 or 2, the **Color** property of the sun cannot be edited.

17.7.13 Defining the geographic location

1 Launch the GEOGRAPHICLOCATION command.

1 The **Geographic Location** dialog box appears.

2 (optional) Check the **Store geographic location information in drawing** option.

3 (optional) Check the **Use Map Grid coordinate system** option.

4 Do one of the following:

5 - Type the latitude and longitude of the location in the **Latitude** and **Longitude** fields and select the appropriate hemisphere for each setting.
  - Click the **Choose location...** button, then select a city and timezone and click the **OK** button.

6 (optional) If you did not select a time zone in the previous step, choose a time zone in the **Time Zones** list.

7 (optional) Specify the **Coordinates of the corresponding location in the drawing**.

8 Define the direction of the north.

9 Click the **OK** button.

17.8 Creating a rendered image

The rendered image appears in the current viewport in a separate window or can be exported to a file.

The background of a rendered view can be controlled by the Background property of the current named view.

The ANTIALIASRENDER system variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5.

Note:

- High anti-alias values incur a high calculation cost.
- The values of the lighting system variables influence the rendering result.

17.8.1 Rendering a viewport

1 Launch the RENDER command ().

The **Render** dialog box appears.

2 Click the **Viewport** option.

3 Click the **OK** button.

The view is rendered.

17.8.2 Rendering in a window

1 Launch the RENDER command ().

The **Render** dialog box appears.

2 Click the **Window** option.

3 Click the **OK** button.

The view is rendered in a popup window.

17.8.3 Saving a rendering to a file

1 Launch the RENDER command ().

The **Render** dialog box appears.

2 Click the **File** option.

3 Click the **Browse** button (), at the right hand side of the **File** field. The **Render output file** dialog box appears.

4 Select a folder and type a name for the output file in the **File name** field.

5 Click the **Save as type** button to select the file type.

The options are: **bmp**, **jpg** and **png**.

6 Click the **OK** button.

The **Render Output File** dialog closes.

7 Do one of the following:

- Type the desired resolution for the output size in the **Width** and **Height** fields.
- Check the **Same as viewport** size option to set the outsize to the size of the current viewport.

8 Click the **OK** button to create the file.

18. Collaboration with others

18.1 Using hyperlinks

18.1.1 About

Hyperlinks allows you to associate various kinds of files or web pages with entities in the drawing.

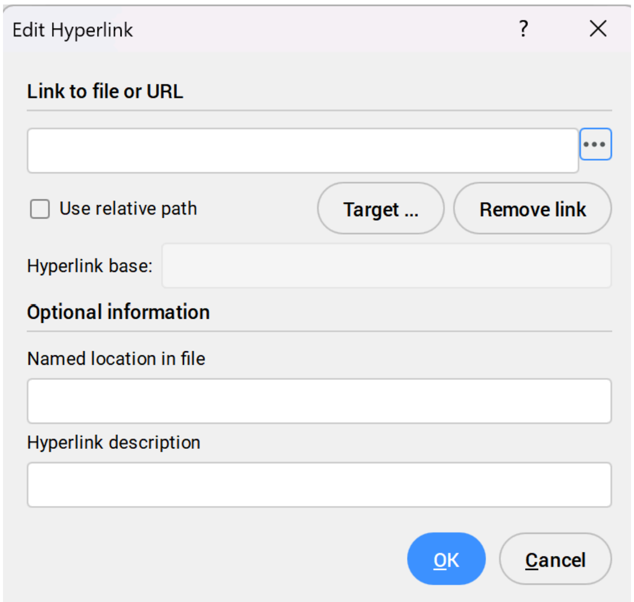
The HYPERLINK command allows you to add or edit a hyperlink through the **Edit Hyperlink** dialog box.

Hyperlinks can have full or relative paths:

- **Full hyperlink**: contains the full path to a file location.
- **Relative hyperlink**: contains a partial path to a file location.

18.1.2 Adding and editing hyperlinks to a DWG

1 Run the HYPERLINK command to open the**Edit Hyperlink**dialog box.



2 You can either add a link to a file or a URL to a web page.

3 **Adding a link to a file**

- Click the **Browse** button (...) to open the **Select file** dialog box.
- Select the file and click **Open** to add the link to the **Link to file or URL** field.

Adding a URL to a web page
Type a URL to a web page in the **Link to file or URL** field.

Note: To remove the attached hyperlink, click **Remove link** button.

18.1.3 Using relative path

When the **Link to file or URL** field is filled in, you can manually erase a part of the path (including the disk) to make it relative.

Note:

- The **Use relative path** option in the **Edit Hyperlink** dialog box is automatically ticked after shortening the file path.
- Relative paths only apply to file paths, not to web pages.

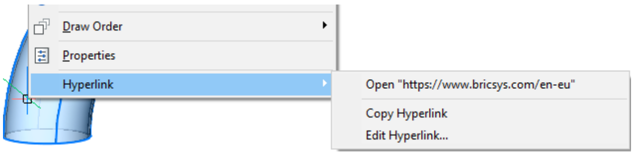
The HYPERLINKBASE system variable allows you to specify a path to use for all relative hyperlinks in the drawing.

Note: All the files linked with a relative path need to have the same location of a file or directory in relation to the HYPERLINKBASE system variable.

18.1.4 Displaying or accessing hyperlinks attached to an entity

To display or access hyperlinks use one of the following methods:

- **Accessing hyperlinks from the context menu**
  Access the context menu and expand the **Hyperlink** option.

  Note:

  - The context menu can be accessed when the SHORTCUTMENU system variable is set to 4.
  - The HYPERLINKOPTIONS command needs to be enabled to view the **Hyperlink** option in the shortcut menu. With this command, you simultaneously set the ENABLEHYPERLINKMENU and ENABLEHYPERLINKTOOLTIP system variables to ON.

  Open hyperlink
  Opens the file or web page associated with the entity.

  Copy Hyperlink
  Copies the hyperlink to the Clipboard.

  Note: The hyperlink can be pasted into the drawing or other documents with the Ctrl+V shortcut or PASTECLIP command.

  Edit Hyperlink
  Opens the **Edit Hyperlink** dialog box.

- **Accessing hyperlinks from the Properties panel**
  Select the entity and go to the **Properties** panel. Double-click the hyperlink in the **Hyperlink** field to open it.

  Note: Click the Browse button () to access the **Edit Hyperlink** dialog box.

19. User feedback

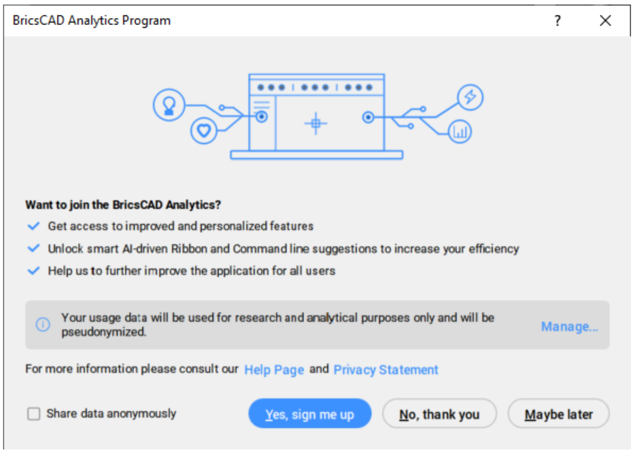
19.1 BricsCAD Analytics Program

The **BricsCAD Analytics Program** process allows you to improve BricsCAD by sending pseudonymized diagnostics and usage data to Bricsys. The Bricsys End User License Agreement (EULA) informs the user that sharing anonymous application data is **ON** by default during the trial period. During the trial period, the BricsCAD **Launcher** presents the user with additional data about **BricsCAD Analytics** and the sharing of anonymous application data with Bricsys. At any point during the Trial, the user can go to the **Analytics Settings** to change their preferences, or to stop sharing data with Bricsys. After the free Trial ends, if a BricsCAD license is activated via a valid license key, the user will be re-informed of the program, and that their installation is sharing anonymous application data with Bricsys. Licensed users can explicitly choose to join the **BricsCAD Analytics Program**, or not share usage data with Bricsys. You can decide at any time to withdraw your consent and to have your data removed from our databases. If possible, Bricsys can send updated AI models to the user.

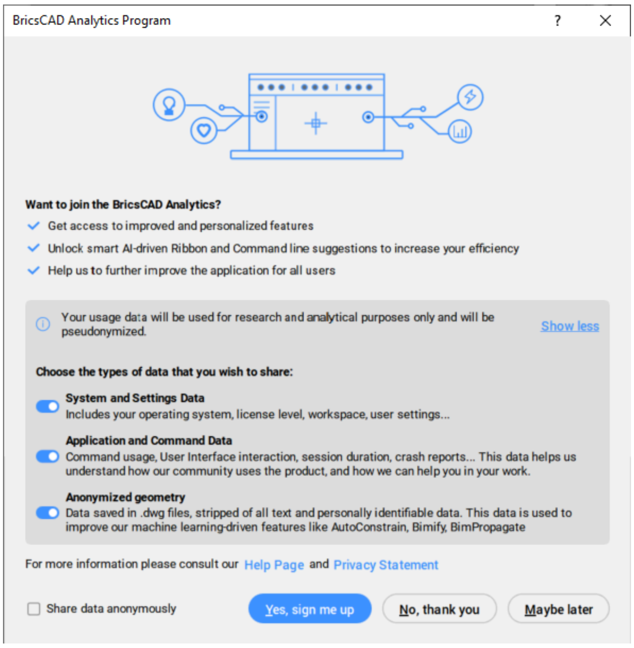
When transferring data:

- A unique, anonymous ID is used.
- The following data and information are removed:
  - All personal project information.
  - All texts and leaders.
  - All information that is stored in the **Settings** dialog box.

When you launch BricsCAD the second time or press **Sign Up** button from **AI Predict** ribbon tab, a dialog box displays asking you to participate in the Analytics Program.



By clicking the **Manage...** button, you are able choose the types of data you wish to share:

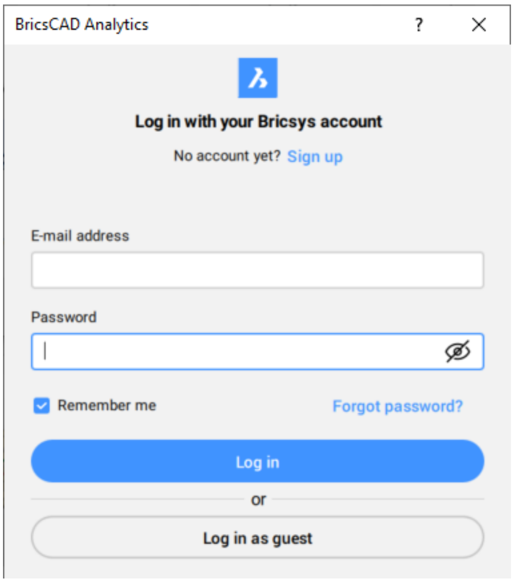


By choosing **No, thank you**, the **BricsCAD Analytics Program** will be switched off. When choosing **Maybe later** or by closing the dialog box, the dialog box will appear again next time you open BricsCAD.

If you tick the **Share data anonymously** checkbox and press **Yes, sign me up**, you will be logged in as a guest.

If the **Share data anonymously** is not ticked and you agree by clicking **Yes, sign me up**, the **BricsCAD Analytics** dialog box opens to sign in with your Bricsys account.

Note: You will not be able to sign up for the **BricsCAD Analytics Program** if all data sharing options are disabled.

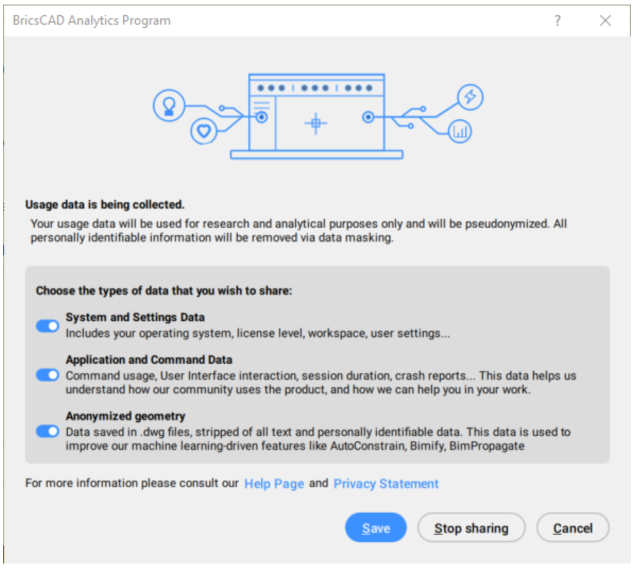


In order to log in, you need to enter your e-mail address and password and click on **Log in**. If you tick the **Remember me** checkbox, you will be signed in automatically each time you launch BricsCAD. A notification will appear if you entered the wrong credentials and the **Login** dialog box will stay open. If your login is successful, this will appear in the dialog box and it will close automatically.

If you do not have an account and would like to create one, you can choose to **Sign up**.

If you do not want to log in, you can also opt to log in as a guest by clicking on **Log in as guest**.

When you are logged in, the **Analytics Settings** button available in the **AI Predict** ribbon tab will open the **BricsCAD Analytics Program**:



By choosing **Save** the **BricsCAD Analytics Program** will collect the selected data.

By choosing **Stop sharing**, the **BricsCAD Analytics Program** will be switched off.

By choosing **Cancel**, the **BricsCAD Analytics Program** dialog box will be closed.

The status of your agreement is saved through the DATACOLLECTION system variable as indicated in the table below.

Value	Status	Result
-2	Not asked	When you close BricsCAD, the value is set to [-1] Not Set.
-1	Not set	When you restart BricsCAD the **BricsCAD Analytics Program** dialog displays, asking you to participate.
0	Off	The Diagnostics and Usage Data Collection process is switched Off.

Value	Status	Result
1	On	The Diagnostics and Usage Data Collection process is switched On. Data will be transferred when you are logged in only. To temporarily stop transferring data, choose **Logout...** in the **File** menu or launch the -LOGOUT command. To restart transferring data, choose **Login...** in the **File** menu or launch the -LOGIN command.

19.1.1 Stopping the process

1 In the **Settings** dialog box, under **User Preferences**, set DATACOLLECTION to **[0] Off**.

2 Close and restart BricsCAD.

19.1.2 Restarting the process

1 In the **Settings** dialog box, under **User Preferences**, set DATACOLLECTION to **[-1] Not set**.

2 Close and restart BricsCAD.

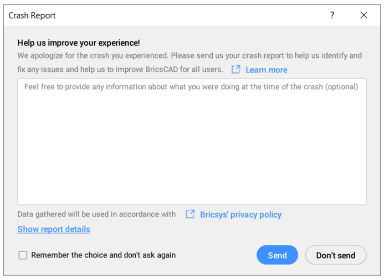
3 Click on **Manage...**, choose the types of data that you wish to share, press the **Yes, sign me up** button, then log in with your Bricsys account.

19.1.3 Removing your data from the database

To withdraw your consent and have your data removed from our databases send in a support request and select the **Data Collection** category under BricsCAD.

19.2 Sending crash reports via Crash Report dialog box

Next to the automatic sharing of crash report data for users that are part of **BricsCAD Analytics**, BricsCAD offers you the possibility to manually send crash information. You can set your preferences via the **Crash Report** dialog box.



Upon the first application startup after a crash, the **Crash Report** dialog box is displayed automatically. From here, you can inspect the crash report details by clicking **Show report details**. See the **Crash Report dialog box** article.

Buttons are offered to either send, or don't send the report of the latest crash.

For **Send** option, the following information is sent: crash report file, operating system, RAM usage during the crash, a unique identifier of the crash, the application uptime at the time of crashing. Optionally, you can provide more information about the context of the crash, for example:

- What you were doing when the problem occurred.
- What type of document you were using.
- Describe the effect of the **Safe mode** (see **Startup options** article).

  Note: If the problem goes away when running in **Safe mode**, it can be helpful to switch back to normal mode and disable or uninstall plugins and customization to narrow down the culprit.
- A repeatable scenario on how to reproduce the problem.
- ***Note:*** A repeatable scenario means to provide a step-by-step procedure together with a well-defined starting context. In general, to make a scenario repeatable the environment is best reduced to be as standard as possible. This is a process of simplification and elimination. If possible, a scenario should start with a default BricsCAD user profile, a new drawing, no third-party plugins or user customization, and a default CUI file.

If the **Remember the choice and don't ask again** check box is not ticked, the **Crash Report** dialog box will be displayed again after a crash occurs. If ticked, this dialog box will never be shown again and the sending preference will be set to automatic sending or never sending depending on whether **Send** or **Don't send** will be pressed.

Your choices are stored with the CRASHREPORTSENDING system variable. Its value can be changed anytime via the Command line or the **Settings** dialog box.

Note: Manual sending of crash reports can be suppressed by the data collection admin override. For more information, see **BricsCAD Analytics Program** and **DATACOLLECTIONENABLED system variable** articles.

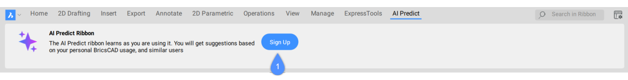
Note: The **Crash Report** dialog box is never shown for users who joined **BricsCAD Analytics Program**. In this case, the crash reports are sent automatically if the **Application and Command Data** sharing data option is enabled in the **BricsCAD Analytics Program** dialog box.

19.3 AI Predict

The **AI Predict** ribbon provides suggestions based on your personal BricsCAD usage, and similar users.
The **AI Predict** tab has two states: an empty state and a populated state.

You can add or remove the **AI Predict** ribbon tab by removing or adding it to the CUI file.

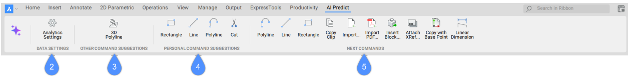
The **AI Predict** tab in the empty state:



Sign Up

The **Sign Up** button opens the **BricsCAD Analytics Program** dialog box.
The populated state becomes available when the user signed up for the **BricsCAD Analytics Program**.

The **AI Predict** tab in the populated state:



Analytics Settings

The **Analytics Settings** button opens the **BricsCAD Analytics Program** dialog box.

Other Command Suggestions

Displays commands used by other users with similar interest as yours.
You will get generic suggestions based on the workspace, and license level from the first day of usage.

Personal Command Suggestions

Displays the most important commands used.

Note: Panels will be populated from the first day of usage BricsCAD with commands based on your interaction with BricsCAD via Quad, command line, ribbon, menu, and toolbars. Also, not everyone will be eligible for all models, depending on how we categorize them.

Next Commands

Dynamically displays some predicted commands which could be used by the user after the current command.

20. Express tools

20.1 Express tools command overview

This is an overview of all the commands that are part of the Express Tools:

21. Getting started

21.1 Creating a BIM model

Do the following:

1 Start BricsCAD and choose the BIM workspace on the **BricsCAD Launcher**. In this workspace, all BIM tools and resources are available in the Ribbon, the Quad and various dockable panels.

2 On the Start page, select a **BIM** template.

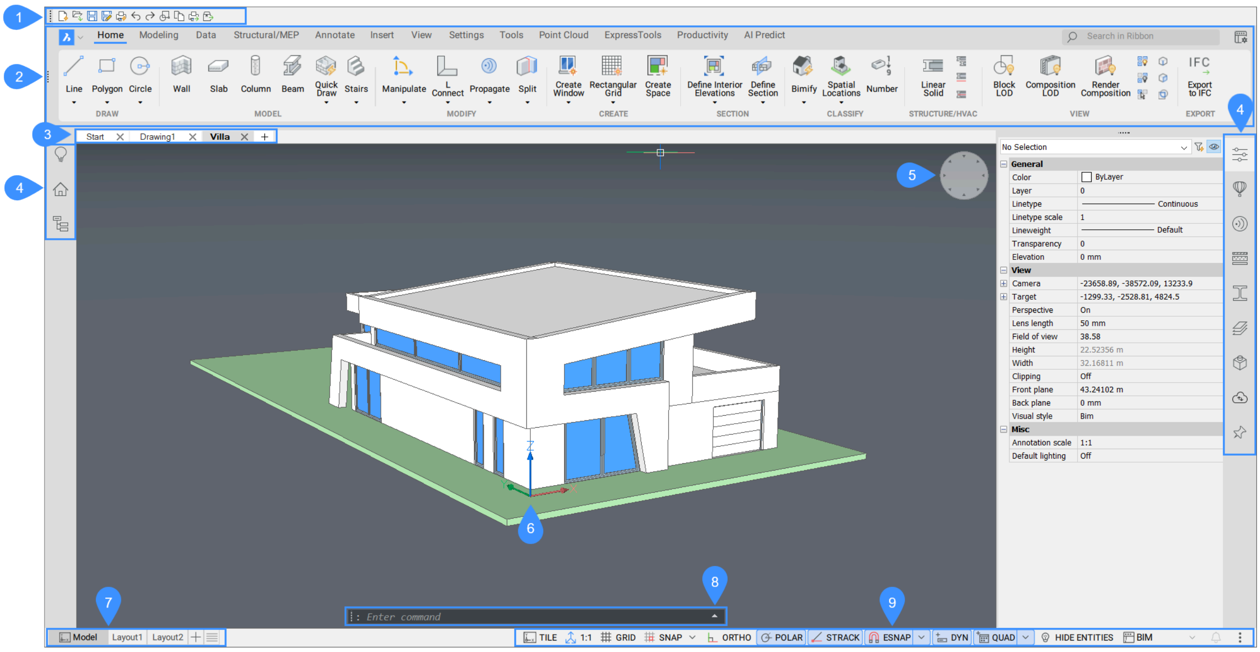
3 Click the **Create new drawing** button to create the model.

21.2 Exploring the BIM interface

21.2.1 User interface

The BricsCAD BIM interface includes all tools, commands, and settings required to create and edit building elements. Start by creating building elements, opening an existing drawing or importing a building model.

21.2.2 Interface elements



1 Toolbar

2 BIM Ribbon

3 Working with Document Tabs

4 Tool Panel

5 Look from Widget

6 UCS Icon

7 Model and Layout Tabs

8 Command line

9 Status Bar

21.2.3 Toolbar

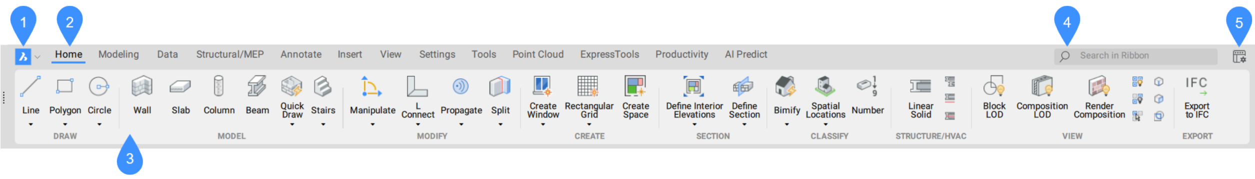
In BricsCAD, toolbars are arranged in collections of similar functions within a bar of tools. Each function is displayed with an icon and is categorized by topic. Toolbars are available for all command categories. For example, a BIM toolbar is one of the tool categories available in BricsCAD BIM. This toolbar includes the most used BIM tools for modeling. It is possible to customize existing toolbars or create your own custom toolbars.

By default, the **Access (Easy)** toolbar is shown on top of the screen.

Note: To open more toolbars, right-click it, choose **Toolbars** and check the topics you want.

21.2.4 BIM Ribbon

A ribbon consists of tabs (2) that group different tab panels (3). Each tab panel contains a series of command buttons and flyouts.



Note: The content of the ribbon depends on the current workspace (see **Editing In Workspace**).

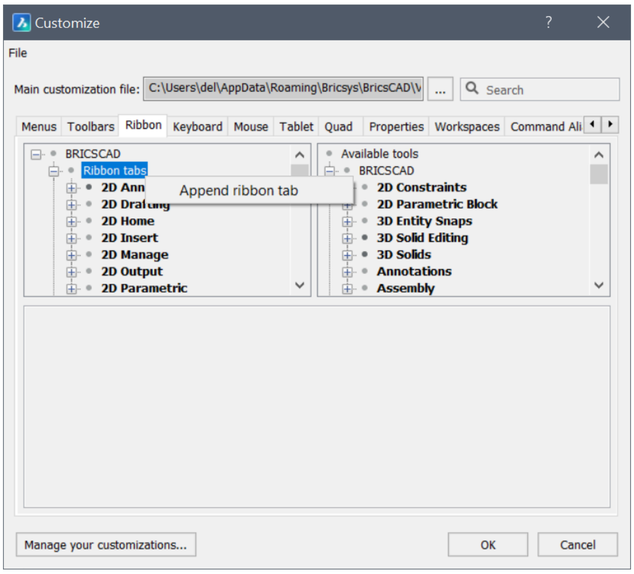
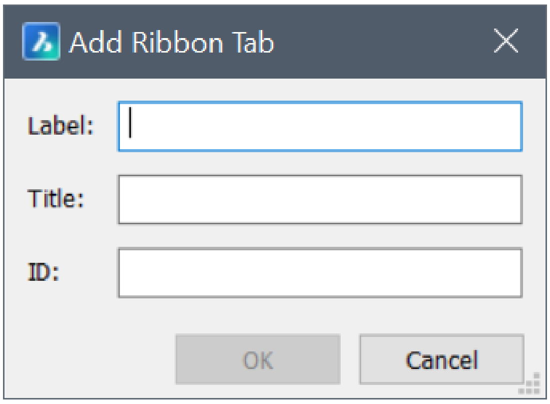
Note: Use the RIBBON and the RIBBONCLOSE commands to show or close the ribbon.

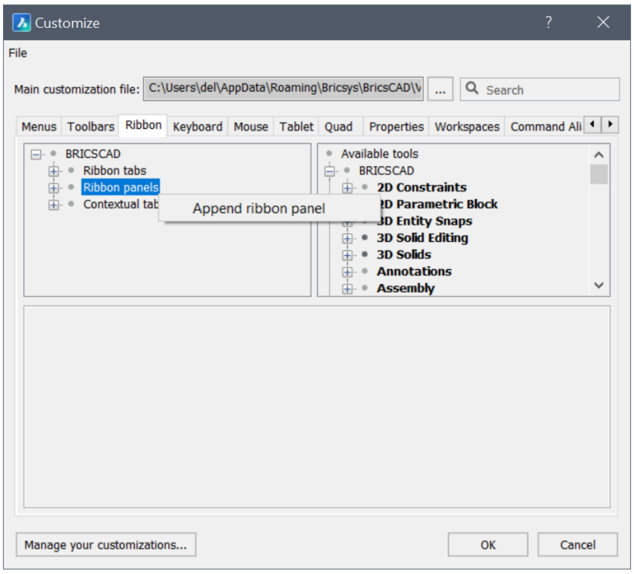
Click the **File** menu button (1) to access BricsCAD's **File** menu options.

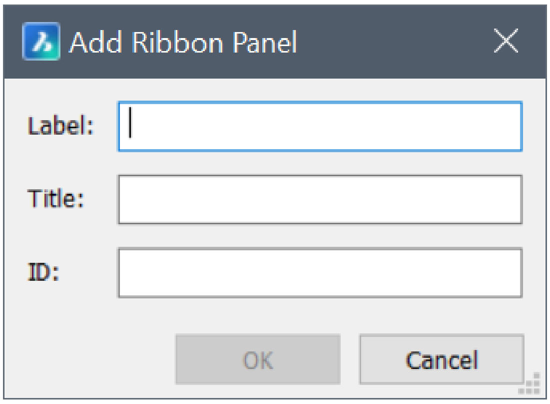
Type in a string in the **Search bar** (4) to locate tools in the Ribbon.

Click the **Interface Settings** button (5) located at the top-right of the Ribbon to easily modify the interface via the **Interface Settings** dialog box (see the **Interface Settings dialog box** related article).

Note: **Classic** is the default layout.

Creating a ribbon tab	Open the **Customize** dialog box (CUI command) and click the **Ribbon** tab. Right-click the **Ribbon tabs** group and choose **Append ribbon tab** in the context menu.  Fill out the fields in the **Add Ribbon Tab** dialog box:  **Label**: Caption on the ribbon tab. **Title**: Used in the **Select Ribbon Tab** dialog when selecting ribbon tabs for a workspace. **ID**: A unique name to identify the ribbon tab.
Editing a ribbon tab	Open the **Customize** Dialog box and click the **Ribbon** tab. Expand the **Ribbon tabs** group and you can edit each tab.

<table>
<tr>
<td>Creating a ribbon panel</td>
<td>
Open the Customize dialog box and click the Ribbon panels.
Right-click the Ribbon panels group and choose Append ribbon panel in the context menu.


Fill out the fields in the Add Ribbon Panel dialog box:


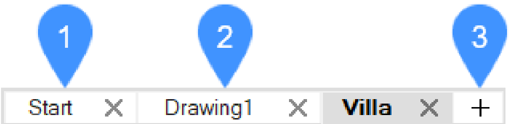
Label: Caption of the panel in the ribbon.
Title: Used in the Select Ribbon Panel dialog when selecting the panels for a ribbon tab.
ID: A unique name to identify the ribbon panel.
</td>
</tr>
<tr>
<td>Editing a ribbon panel</td>
<td>
Open the Customize dialog box and click the Ribbon panel.
Expand the Ribbon panels group and you can edit each panel.
</td>
</tr>
</table>

21.2.5 Working with Document Tabs

Document tabs allow you to:

- Easily switch between all open drawings by clicking on drawing tabs (2).
- Drag and drop the tabs to change their order.

- Click the **X** icon at the right-hand side of the tab to close it.
- Right-click a document tab to display a context menu.
- Click the **+** icon (3) at the end of the document tab to open a new drawing.
- Access the **Start Page** by clicking on the **Start** tab (1).



21.2.6 Tool Panel

In BricsCAD BIM the **Tool** panel is displayed on both sides of the workspace. The **Tool** panel consists of BIM panels. The BIM panels are dockable and thus they are movable. The panels can be moved to another docked panel using the drag and drop method.

By default, the **Tool** panel displays on the left-hand side the **Tips**, **BIM Project Browser**, and **Structure** panels, and on the right-hand side the **Properties**, **Library**, **Details**, **BIM Compositions**, **BIM Profiles**, **Layers**, **Mechanical Browser**, **Bricsys 24/7**, and **BIM BCF**.

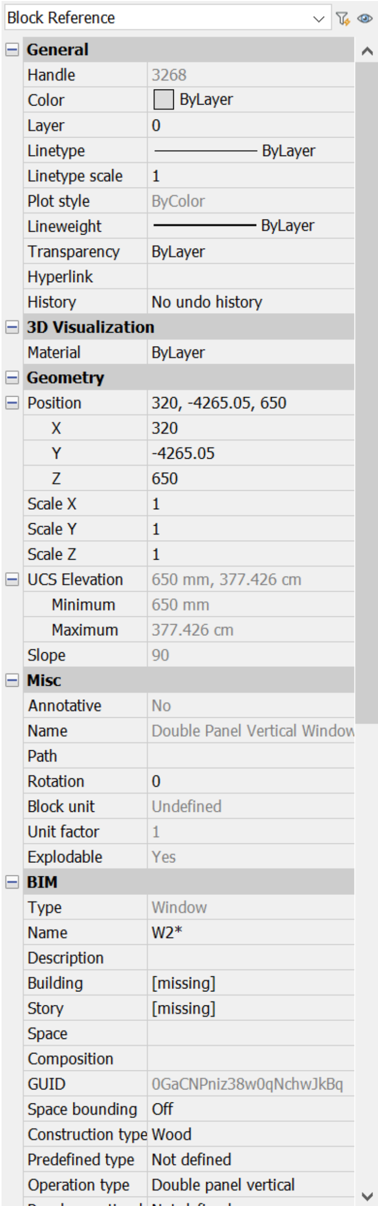
Note: You can always add panels in the Tool Panel by right-clicking and checking those you want.

These are the functions of the different dockable panels from the **Tool** panel:

Tips	The **Tips** panel gives general information about tools and commands. The tips can change depending on the active command. The tips can only be seen when the panel is open.
BIM Project Browser	The **BIM Project Browser** panel allows you to easily navigate through the entire content of a project: models, sections, drawings, and schedules. See The **BIM Project Browser** article on how to use this panel.
Structure	The **Structure** panel is an interactive tree that displays the entities in the current model. Using the configurable structure tree on the **Structure** panel, the BIM model can be organized in the way that you want to view the elements. Once you configured the tree, you can easily save this structured tree as a CST file. By default, the CST files are stored in the **Support Folder**. See the **Structure panel** article on how to use this panel.

Properties

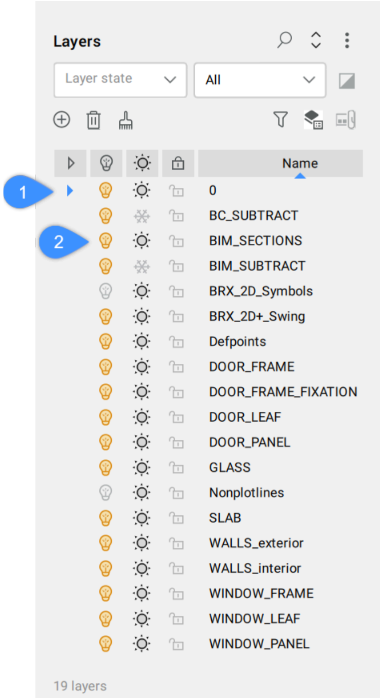
The **Properties** panel shows all the properties of a selected element. You always have some general info first, the other different tabs within this panel depend on the type of element that is selected.



For example, here the selected item is a window and is colored by layer. You can adjust the properties as you like.

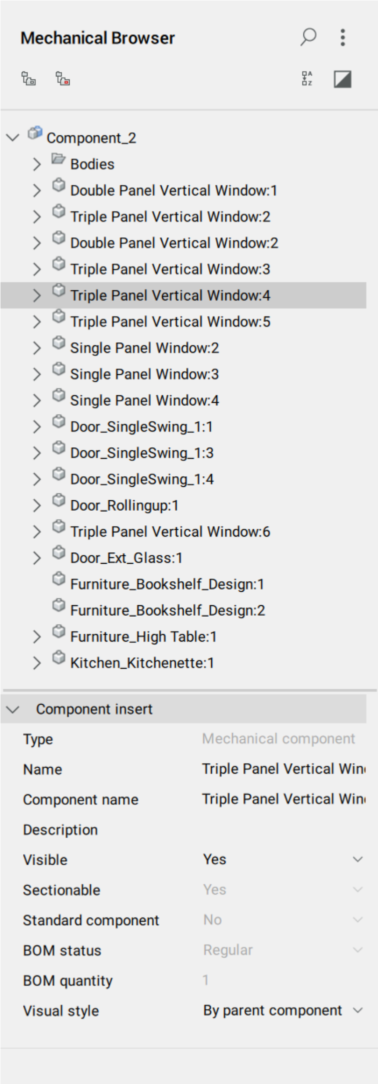
See the **Properties** article to see how to work with this panel in more detail.

Library	The **Library** panel gives a list of components that you have by default. You can simply click and drag these components into your drawing. At the top of the panel, you have the **Add** button that allows you to add a block to the library. For example, when you have made a window drawing, you can click this button to add it to the components library. This way, your window can be used as a component in all of your other drawings.
Details	In the **Details** panel you can: • View saved details. • Organize saved details. • Propagate details. • Open details and adjust them in the model space.
BIM Compositions	In the **BIM Compositions** panel, you find a library of predefined compositions you can assign to various solids of the BIM model by dragging them on. Assigning compositions to the elements in your model is a core step to increase the BIM model accuracy. Compositions can contain either multiple materials or a single ply. See the **BIM Compositions** article on how to use this panel in more detail.
BIM Profiles	The **BIM Profiles** panel shows a library of predefined profiles you can assign to linear solids by dragging them on. See the **BIM Profiles** article on how to work with this panel in more detail.

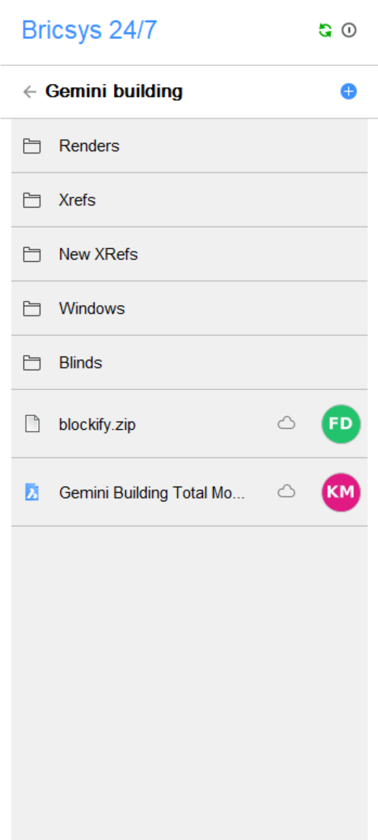
Layers	The **Layers** panel shows all the layers of your drawing. It shows you the current layer (1), you can change it by clicking left of another layer. It also shows if a layer is ON or OFF by the state of its light bulb icon (2). Other layer properties, such as color, linetype, etc., can be found and adjusted by scrolling to the right.  In this example the current layer is **DOOR_FRAME** and it is turned ON.

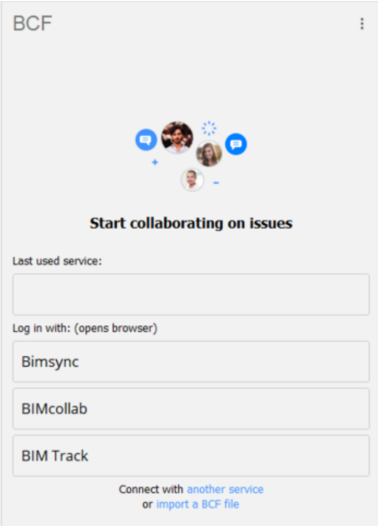
Mechanical Browser

The **Mechanical Browser** panel displays the hierarchy of components, features, arrays, parameters, and constraints for the current drawing.
They are displayed in a tree-like form: you can expand and collapse every node to see its child nodes (for example, nested components).



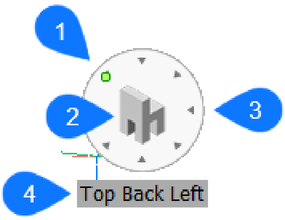
The **Mechanical Browser** panel allows you to edit properties of a particular node, displayed at the bottom of the panel and calls different tools depending on what type is available in the context menu by right clicking a particular node. For example, in the image above you could add a description for the selected window.

Bricsys 24/7	Login to your Bricsys 24/7 account here to collaborate on your project in the cloud. Bricsys 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible. When logged on, you can see an overview of your **in the cloud** Projects, Folders and Documents as shown in the image below:  See the **BricsCAD 24/7** panel article to see in detail how this panel works.

BIM BCF	Here you can import BCF (Bim Collaboration Format) files and connect to a BCF server. The BCF topics are listed in this panel.  The top part of the **BCF** panel shows all the topics and can look similar to the **Library** panel. The lower part of the **BCF** panel shows the currently active topic, with a bigger snapshot and the most common metadata: comment, author, status, etc. Double-clicking on an issue activates the topic.

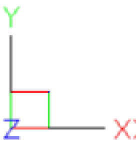
21.2.7 Look from Widget

The **Look From** Widget appears in the upper right corner of the drawing area. When you hover the cursor over the widget, a small green rectangle appears (1), as does the preview of a chair (2). Hovering the cursor over one of the triangle shapes (3) shows what the 3D view will look like (4). The default location of the widget is the top right corner of the drawing area.



21.2.8 UCS Icon

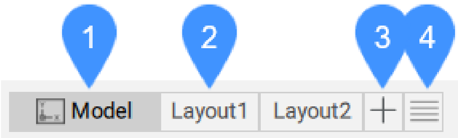
UCS stands for a User-defined Coordinate System. The UCS icon shows the nature of the current coordinate system.



The display of the icon is controlled through the UCSICON system variable and the position of the UCS icon is defined by the UCSICONPOSITION system variable.

21.2.9 Model and Layout Tabs

At the bottom of the graphic screen sit the Model and one or more Layout tabs which allow you to easily switch between model space and paper space.



Model Tab

When you start a drawing session, your initial working area is called **Model Space**. The **Model Space** is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or User Coordinate System (UCS). You view and work in model space while using the **Model** tab (1).

Layout Tab

Each drawing has at least one layout (2) and each layout is composed of one or more viewports. Each viewport can show a different part of the drawing at a different scale. The **Paper Space** is a work environment that provides the model space view at a given scale, depending on the size of the paper.

In each layout, you can add the entities needed to complete a printed copy of the drawing; title blocks, legends, frames, etc. These entities are only visible in the layout when you add. They are not visible in the other layouts or in the model space.

Click the **New Layout** button (3) to add a new layout.

Click the **Manage model and layouts menu** button (4) to display options to publish, create, copy and delete layouts.

Right-click a layout/model tab to open a context menu.

UCS in Layout Mode

In paper space of layout modes, a W letter appears in the UCS icon, which means the WCS is active.

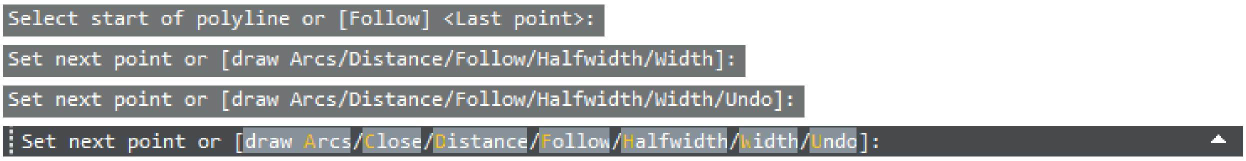
Note: **WCS** stands for World Coordinate System.

21.2.10 Command line

The Command line panel consists of two fields.

- In the lower field, you can type the commands and BricsCAD shows prompts, options and other information regarding the execution of commands here. If the Command line is closed, this information shows in the Status Bar.
- The command history displays in the upper field of the Command line panel.

By default, the Command line panel is floating (CMDLINEUSEMINIFRAME system variable = 1) and minimized to a single line. When executing a command, a number of prompt lines (set by the CLIPROMPTLINES system variable) are displayed temporarily and transparently on the graphic screen:



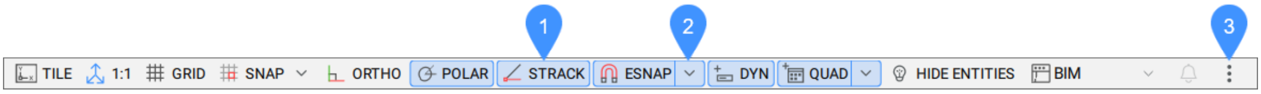
Click the up arrow at the right hand side to display the **Prompt History** window.

Press and hold the left mouse button on the edge of the right hand side, then drag to modify the length of the Command line panel.

Press and hold the left mouse button on the left hand side grip, then drag to modify the Command line panel's position.

21.2.11 Status Bar

The **Status Bar** sits along the bottom edge of the BricsCAD application window. It contains a lot of information about the settings in the current drawing. All fields in the **Status Bar** are optional.



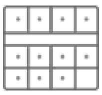
Click a field in the Status Bar (1) to toggle it ON/OFF. Right-click a field/ click the drop-down button (2) to display a context menu/settings for this field.

Click the **Status Bar Menu** button (3) to display the list of all fields. Click a field to toggle its display in the Status Bar.

21.3 Quad cursor

21.3.1 About

The **Quad cursor** menu, or **Quad** for short, is an alternative to the Command line or toolbars, offering a rich set of tools while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs. The rollover tips are a limited set of properties displayed when selecting or hovering over an entity.



To toggle the **Quad** on/off, do one of the following:

- Click the **Quad** field in the **Status Bar**.
- Press the **F12** function key.
- Edit the QUADDISPLAY system variable in the **Settings** dialog box.
- Type QUADDISPLAY in the Command line, then choose an option.

To toggle the **Rollover Tips** on/off, do one of the following:

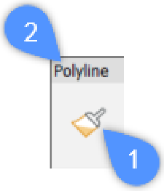
- Click the **RT** field in the **Status Bar**.
- Type ROLLOVERTIPS in the Command line then set a value.

21.3.2 Quad layout

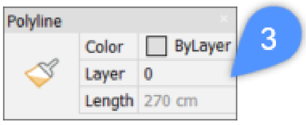
The **Quad** has a floating feature set that adjusts its content, depending on what you are or are not highlighting, and what you may have selected in the current workspace.

- Based on the highlighted entity type, the **Quad** displays the most recently used tool (1) with the entity name (2).

  Note: When multiple entities are under the cursor, hit the **Tab** key to cycle through the different entities.

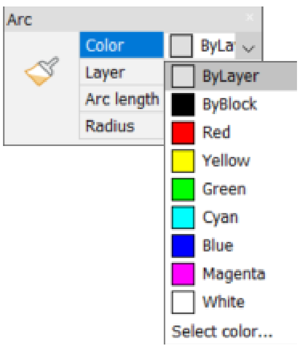


- When the **Rollover Tips** setting is on (and SELECTIONPREVIEW == 1 or 3), a limited set of properties (3) of the entity under the cursor display in the **Quad**.



The property values in the **Rollover Tips** can be edited.

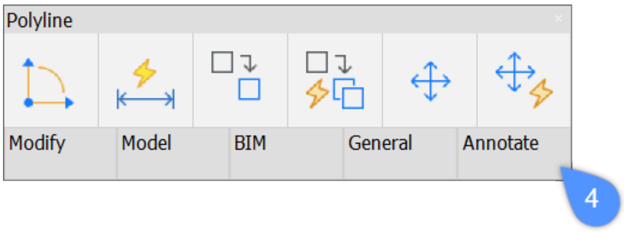
The following figure shows the color of an arc being changed using the **Rollover Tips**.



Note: To display the **Rollover Tips**, simply click the title bar of the **Quad** after expanding it.

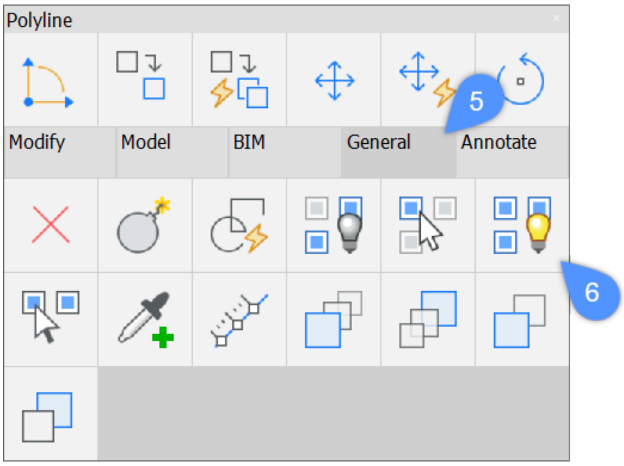
- When the cursor is moved on the **Quad**, a number of tool tabs display at the bottom (4).

  Note: The number of tool tabs is controlled by the **Quadwidth** user preference (default = 6).

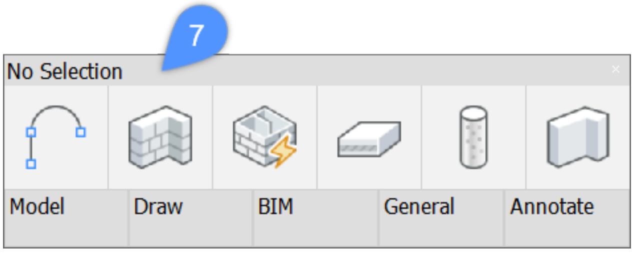


- When you move the cursor over the **General** quad tab (5), the tool groups expands under the cursor, which provides you with more tool options such as UNISOLATEOBJECTS command (6). Beside this command you can find ISOLATEOBJECTS command and HIDEOBJECTS command.

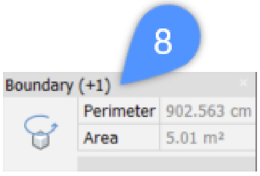
  Note: The number of available tool groups and the number of tools within a tool group depend on the entity type under the cursor.



- The **Quad** can also be accessed in the no selection mode. When the cursor is over an entity, hold down the right mouse button longer to display the **No Selection** Quad (7). Or when there is no selected/ highlighted entity in the drawing, right-click to display the **No Selection** Quad.



- When an entity is selected and another one is highlighted, next to the entity type (title bar on the top) a plus one (8) appears in the parentheses. The count, regardless of the type, increases each time you select an entity. To reveal the selected the entity types which are hidden in the parentheses, expand the Quad by moving your cursor on it.

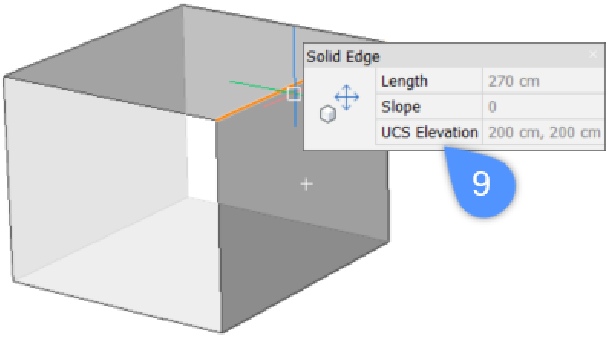
- 

  Note: The **Quad** also adapts to the type of entities that have been selected. If you select two surfaces, you will be provided with different tool options in the Quad, then if you selected a solid and a surface.

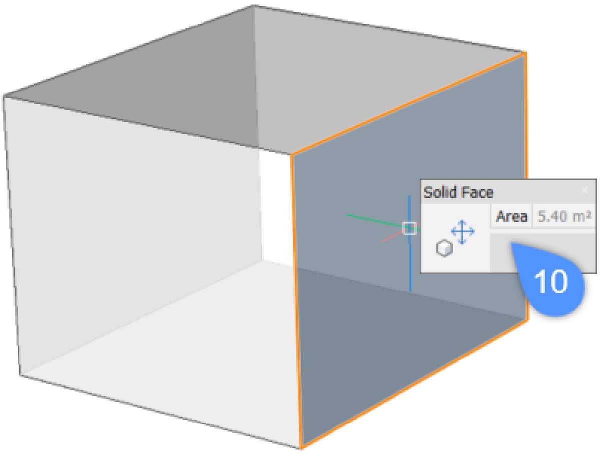
21.3.3 About: selecting entities

The **Quad** adapts to the type of entities that you have selected. If you select a surface, you will be provided with different tool options in the Quad, then selecting a solid or edge.

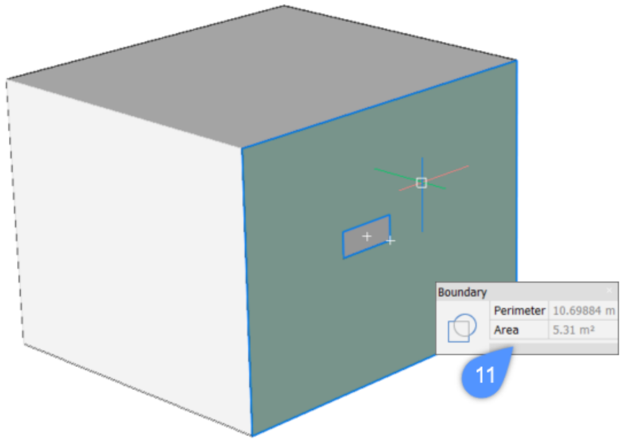
- Selecting an **edge**, the **Quad** appears showing the edge length, slope, and elevation (9).



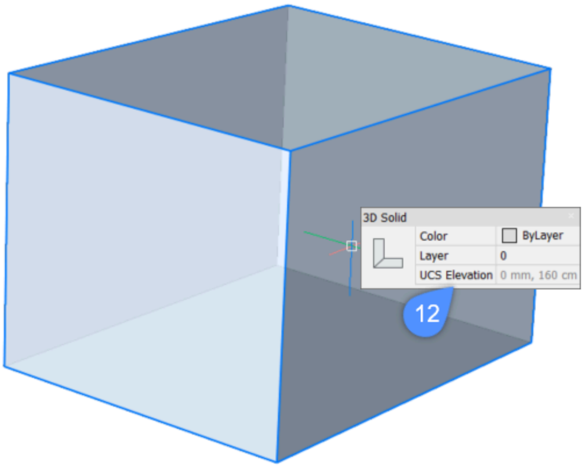
- Selecting a **solid face**, the surface area is shown in the **Quad** (10).



- Highlighting a **boundary**, the **Quad** displays the perimeter and area of this boundary (11).



- Highlighting a **solid**, the **Quad** displays the color, layer, and elevation of the solid (12).



21.3.4 Procedure: using the quad

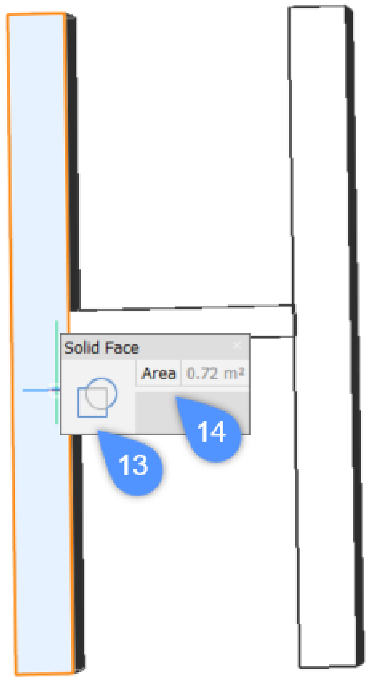
The following are general steps to use the **Quad** cursor.

Note: Make sure the **Quad** and **Rollover tips** are both enabled, and the SELECTIONPREVIEW == 1 or 3.

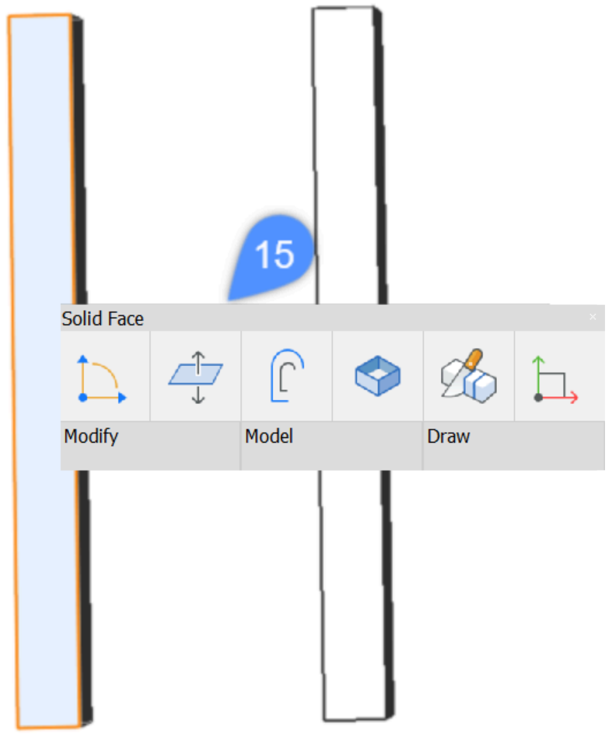
1 Hover the cursor over the entity you want to manipulate.

1 The entity under the cursor highlights.

The **Quad** displays a single icon (13), which is the most recently used tool with this entity type, and a limited set of properties (14) of the highlighted entity.



2 Right-click to launch the recently used tool. Or move the cursor to the single icon to expand the **Quad** (15), then select one of the available tools to manipulate the entity.



3 Move the cursor over the blue field to further expand the **Quad**.

All Quad tool groups for the current workspace display. When the first tool group is expanded, click an icon to use a tool or move the cursor to a different tool group.

21.4 Viewing entities

21.4.1 Commands

HIDEOBJECTS, ISOLATEOBJECTS, UNISOLATEOBJECTS

21.4.2 About

If you work with many entities in a drawing, everything that you have in the drawing is visible in the view. When you want to temporarily make only a few entities or a particular entity visible and manage them in a view, you can use the **Hide** or **Isolate** tools.

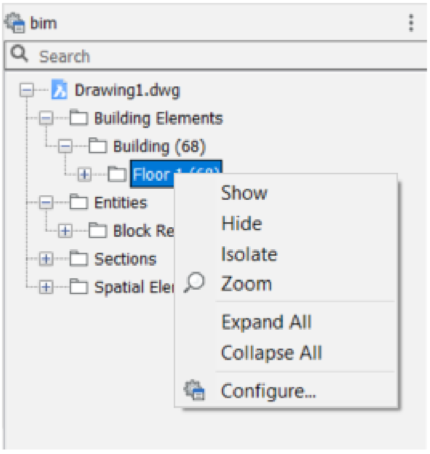
21.4.3 Hiding entities

The **Hide** tool temporarily hides the selected entities in the view.

For more information about this command, visit the Command Reference article HIDEOBJECTS.

(**Optional**) The **Structure** panel can also be used to hide and isolate selected entities as well as revealing all hidden entities in the model space. This is an easy way to quickly hide or isolate a certain category of objects or entities with a certain property value etc.

To do so, right-click on the entity in the **Structure** panel and choose one of the options in the context menu (see below image).



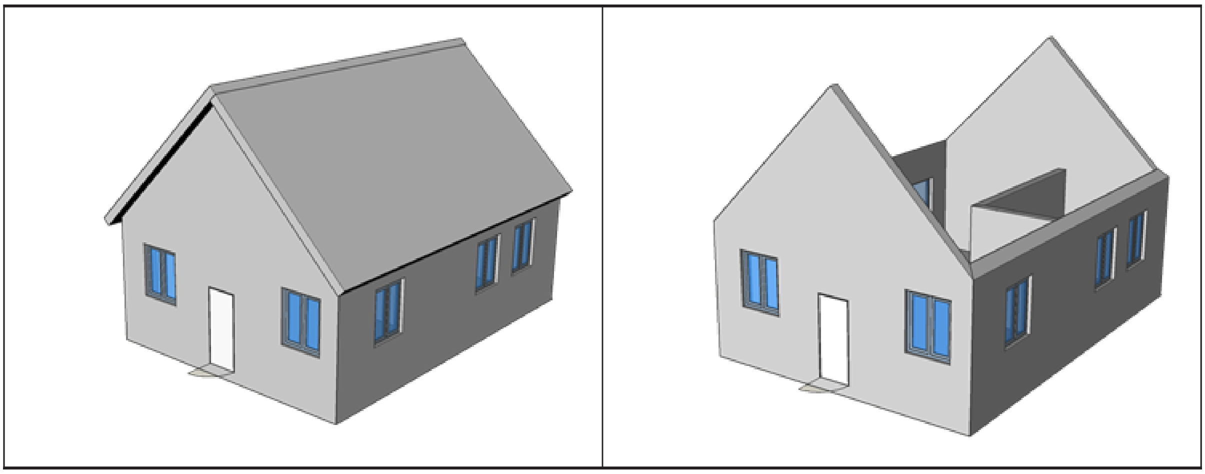
21.4.4 Procedure: hiding entities

1 Select one or more entities in the drawing area.

2 Select **Hide Entities** from the Quad.

All entities that have been selected are hidden in the view.

The following illustration shows an entire model on the left. In the second image, the roof was temporarily hidden.



21.4.5 Isolating entities

The **Isolate** tool temporarily hides all entities except those that have been selected.

For more information about this command, visit the Command Reference article ISOLATEOBJECTS.

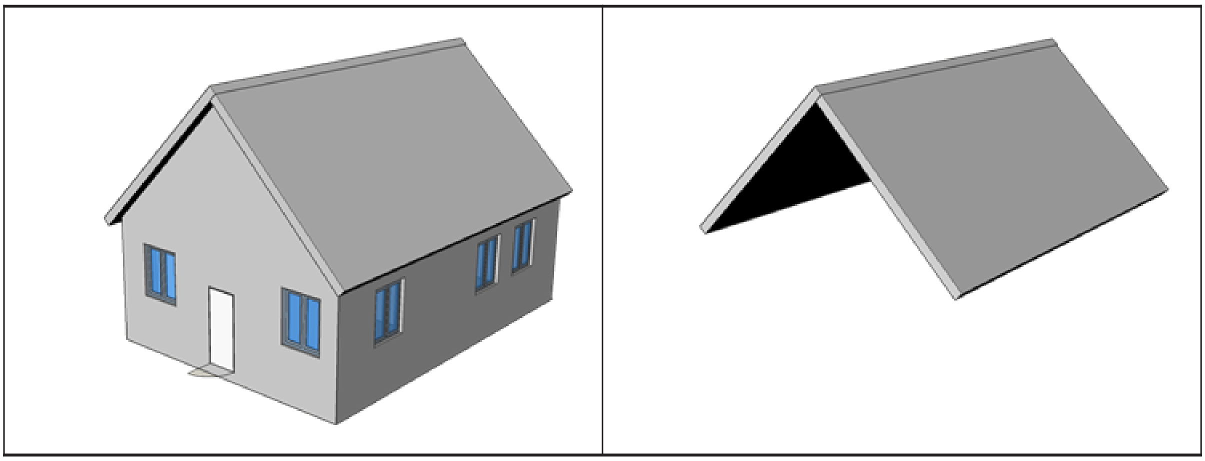
21.4.6 Procedure: isolating entities

1 Select one or more entities in the drawing area.

2 Select **Isolate Entities** from the Quad.

All entities that have been selected before are isolated in the view.

The following illustration shows the same building model from the below. In the second image, the roof was temporarily isolated.



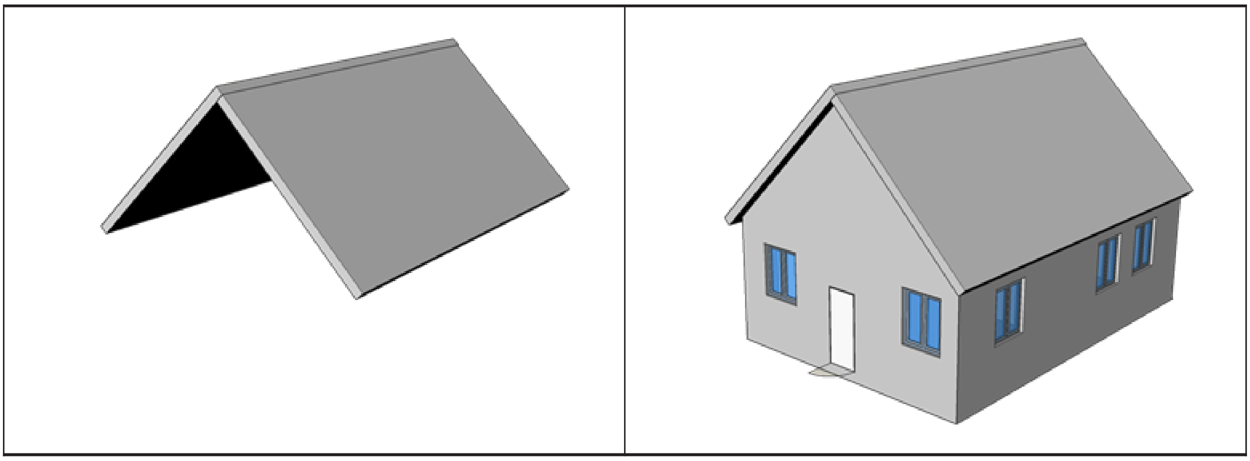
21.4.7 Showing entities

The **Show Entities** tool brings back all hidden entities in the view.

For more information about this command, visit the Command Reference article UNISOLATEOBJECTS.

If you have any hidden entities in your drawing, you can reveal them in the view using the show entities tool.

The following illustration shows the isolated roof elements from the above example. The second image shows the result after using the **Show Entities** tool.



21.5 Selecting entities

21.5.1 About

When you hover over an object in BricsCAD, the object is highlighted. When you left click a highlighted entity, it will be selected.

Highlighting: you only can highlight one entity at once.

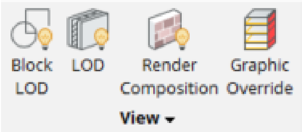
Selecting: you can select as many entities as desired.

You can perform most of the modeling process by highlighting the entity and using the Quad. For example, when you highlight a surface, the Quad will open. You can pick the command you want, without selecting the surface first.

21.5.2 Selection modes

The selection modes allow you to control which sub-entities (faces, edges, and boundaries) will highlight in the selection preview and can be selected.

- Ribbon: **Home tab** > **View panel** > **Selection modes**



Note: To display the **Ribbon**, right-click on a toolbar and choose **Ribbon** in the context menu.

- Settings: **Program options** > **Selection** > **Selection Preview** > **Selection Modes**

Selection modes	0x0006 (6)
1	☐ Select edges
2	☑ Select faces
4	☑ Select detected boundaries
8	☐ Select vertices

- Keyboard: SELECTIONMODES

This is a list of all the selection modes:

- **Enable detection of 3D solid edges** (): controls whether 3D solid edges are highlighted by selection preview and can be selected. If you enable this option, you can select the edges of 3D solids by clicking them once.
- **Enable detection of 3D solid faces** (): controls whether 3D solid faces are highlighted by selection preview and can be selected. If you enable this option, you can select the faces of the 3D solids by clicking them once.
- **Enable boundary detection** (): controls whether closed boundaries in XY-plane of the current coordinate system or on the face of 3D solids are detected. If you enable this option, you can select closed boundaries from the faces of 3D solids by clicking them once.
- **Enable selection of Sides and Ends** (): controls the bimSides and BimEnds visibility of all linear elements.
- **Enable vertex selection** (): controls whether 3D solid vertices are highlighted by selection preview and can be selected. If you enable this option, you can select the vertices of the 3D solids by clicking it once.

Note: BIM Sides/Ends are only available on classified linear elements with the attached library profile.

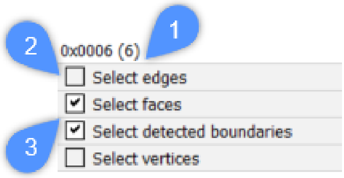
21.5.3 Changing selection modes setting

A sub-entity is a face, edge or boundary of a 3D solid. To control which sub-entities you select in your model, do one of the following:

Note: The default value of the SELECTIONMODES system variable is 6, which means that the detection of 3D solid edge is disabled, while the detection of 3D solid faces and boundaries is enabled.

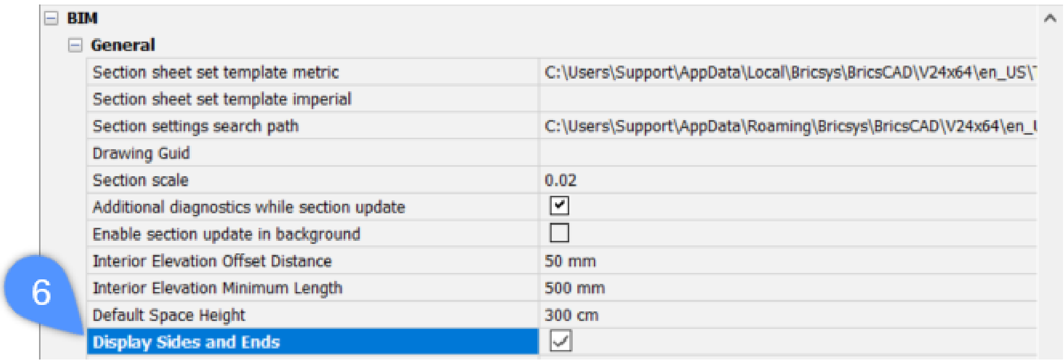
- Click the buttons on the **Settings** panel on the **Ribbon** and find **Selection Modes**.

The following illustration shows the sum of the values (1) of the SELECTIONMODES system variable when the **Select edges** and **Select vertices** options are disabled (2) after clicking the checkbox once; and the **Select faces** and **Select detected boundaries** are enabled (3).



- Type SELECTIONMODES in the Command line and change the current parameter.
- Use the Ctrl key on the keyboard to temporarily toggle between the different selection modes. These key functions enable you to select any type of entity.
- To display the BimSides/Ends for linear elements you can use the DISPLAYSIDESANDENDS system variable in the **Settings** dialog box or set the DISPLAYSIDESANDENDS system variable to 1 in the Command line.

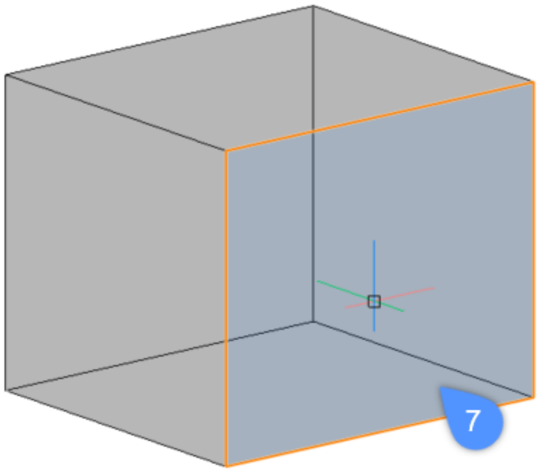
  The following illustration shows the **BIM** > **General** system variable when the **Display Sides and Ends**option is ticked (6) after clicking the checkbox once.



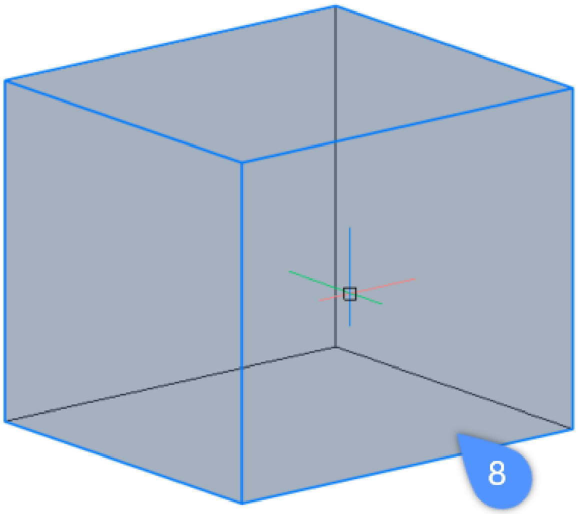
21.5.4 Selecting sub-entities

When **Select edges** is off, **Select faces** is on, **Boundary Detection** is on (default), do one of the following:

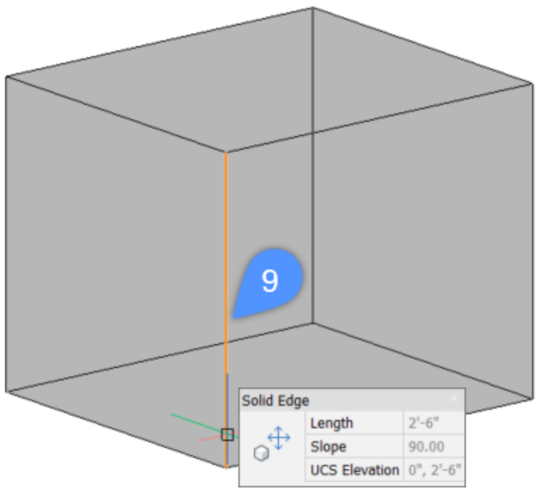
- To highlight a solid face, hover over the face with your mouse cursor. The highlighted face is displayed in orange (7). To select the face, left-click it when the face is highlighted. To select multiple faces of the solid, left-click them one by one.



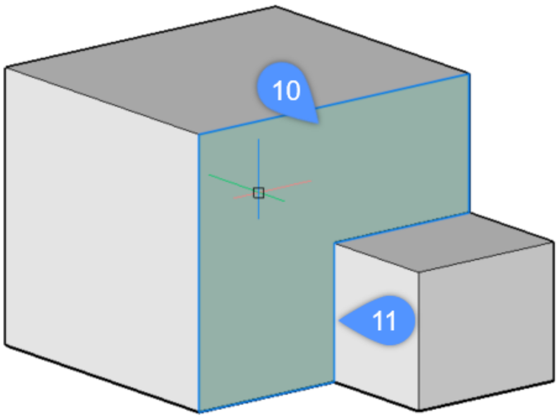
- To highlight a solid, hover over one of its faces while holding down the CTRL-key. The solid is displayed in blue (8). To select the solid, left-click it while the solid is highlighted. To select multiple solids left-click them one by one.



- To highlight an edge, hover over the edge of the solid while holding down the CTRL-key. The edge is displayed in orange (9). To select the edge, left-click it while the edge is highlighted. To select multiple edges left-click them one by one.



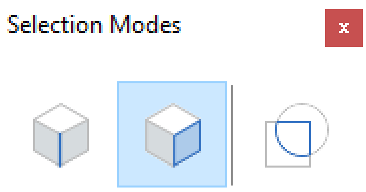
- To highlight a boundary, hover over the detected boundary on the face of the 3D solid.
- When the detected boundary is highlighted, it is displayed in a color. By default, inside of the boundary is displayed in green (10) and the outer edge of the boundary is displayed in blue (11).



Note: This setting can be changed by typing BOUNDARYCOLOR on the Command line (default value = 95).

21.5.5 Procedure: modifying the thickness of a solid

1 Make sure the **Select faces** option of the SELECTIONMODES system variable is selected.



2 Hover the cursor over the face of a solid.

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose **Push/Pull** in the Model command group of the Quad.

5 The selected face of the solid is dragged.

6 The dynamic dimension field displays the relative displacement of the face (= with respect to its original position).

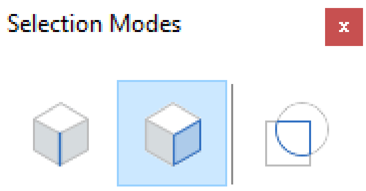
7 (**Optionally**) Press the Tab key to change the face selected as the reference face for the dynamic dimension.

Note: When a composition with locked thicknesses (non-variable thickness) is applied to a solid and if the thickness of the solid is not equal to the total amount of the composition thickness, the solid is displayed with red hatches. Execute the BIMUPDATETHICKNESS command to re-apply the composition to such solid.

21.5.6 Procedure: modifying the height of a wall

Modify the dimensions of a single wall

1 Make sure the **Select faces:** option of the SELECTIONMODES system variable is selected.



2 Hover the cursor over the top face of the wall.

3 If necessary, press the Tab key to highlight an obscured face.

4 When the face highlights, choose **Push/Pull** in the Model command group of the Quad.

The selected face moves dynamically with the cursor.

The dynamic dimension fields display the incremental height.

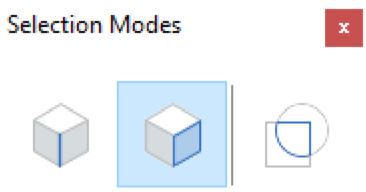
5 Do one of the following:

- Type a value in the dynamic dimension field and press Enter.
- Press Tab to use another reference face, e.g. the bottom face of the solid, then type a value in the dynamic dimension field.

Modify the height of multiple walls

Use the SELECTALIGNEDFACES command (**Quad** > **Select**:) to select the top face of all solids in the same horizontal plane.

1 Make sure the **Select faces** option of the SELECTIONMODES system variable is selected.



2 Move the cursor over the top face of one of the walls.

3 If necessary, press the Tab key to highlight an obscured face.

4 When the face highlights, choose SELECTALIGNEDFACES in the **Solid/Face Selection** command group in the Quad.

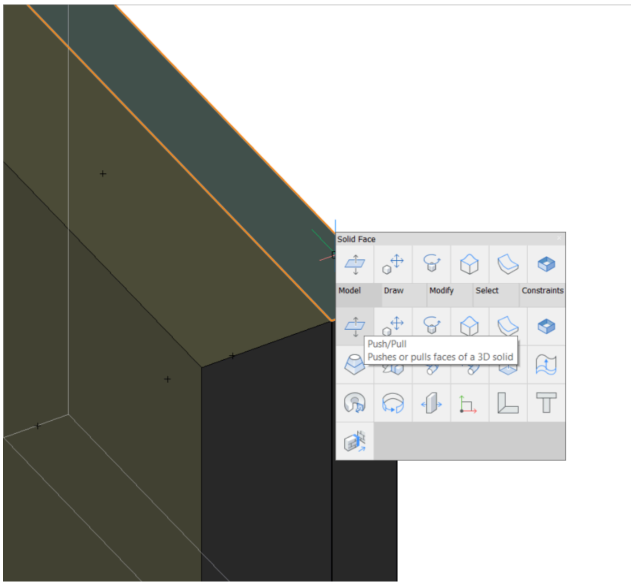
All faces lying in the plane of the selected face are selected.

5 Now follow the same steps as for a single wall.

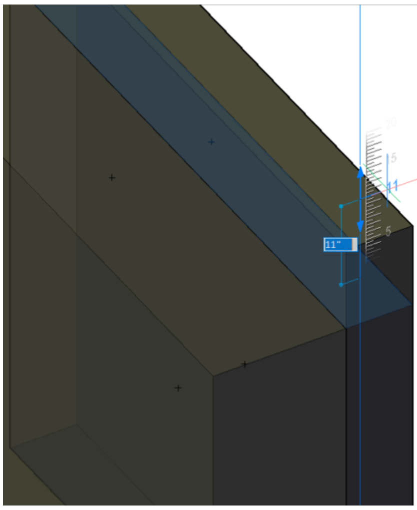
Modify the height ply-by-ply

When the **Display Composition** property of a solid is set (LEVELOFDETAIL system variable =2 and, optional, RENDERCOMPOSITIONMATERIAL system variable =1) and the **Select faces** option of the SELECTIONMODES system variable is selected, the height or length of the composition plies can be modified separately.

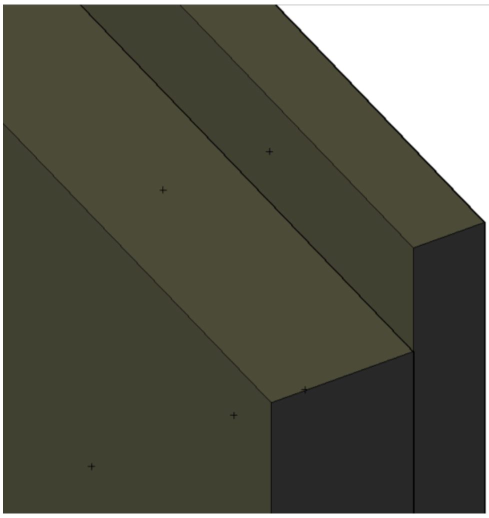
1 Move the cursor over the ply that you want to edit and choose **Push/Pull** in the **Model** command group of the Quad when the ply highlights.



2 Type value in the dynamic distance field.



3 Press Enter to accept the distance.



21.5.7 Selecting obscured sub-entities

In your drawing, some entities or sub-entities (faces, edges, and boundaries) might be hidden behind other entities. The Tab key enables you to cycle through sub-entities.

The following steps show how to highlight a hidden face that lies beneath the cursor.

When **Select edges** is off, **Select faces** is on, **Boundary Detection** is on (default).

1 Place your cursor on one of the solid faces.

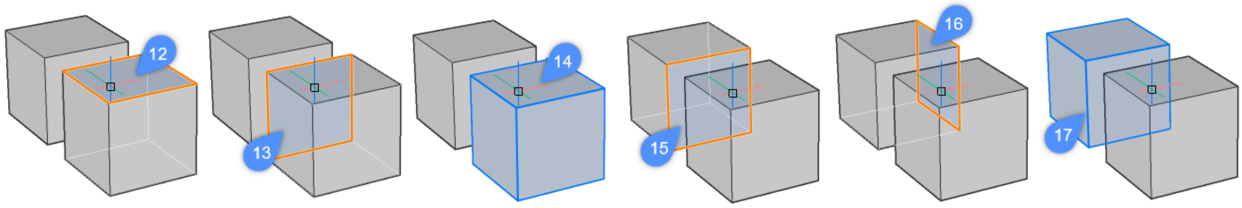
1 In the example below, the top face of the front solid is highlighted (12).

2 Press the Tab key to cycle through the hidden entity.

3 The back face of the front solid is highlighted (13).

4 Repeatedly pressing the Tab key while keeping the cursor at the same position allows you to highlight the entire solid in the front (14) and the front face (15) and right face (16) of the rear solid. Finally, the entire rear solid is highlighted (17).

5 When the entity or sub-entity that you want to select is highlighted, left-click it to select.

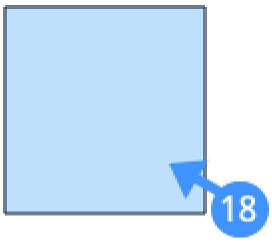


21.5.8 Selecting multiple entities using the selection boxes

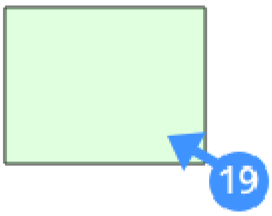
The selection boxes allow you to select one or more than one entity at once. To use a selection box, left-click on an empty space in the drawing area and move the cursor until the selection box covers the entities that you want to select.

There are two types of selection boxes in BricsCAD:

A **blue selection box** (18) appears when you create a box from left to right. The blue selection box will only select entities that are entirely covered by this selection box.



A **green selection box** (19) appears when you create a box from right to left. The green selection box will select all elements that are entirely covered by the selection box and all the elements that intersect with one of the edges of the box.



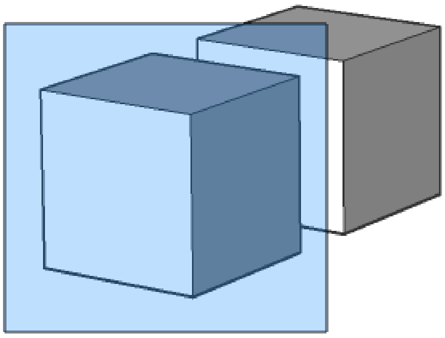
The following steps show how to select multiple entities using the blue selection box:

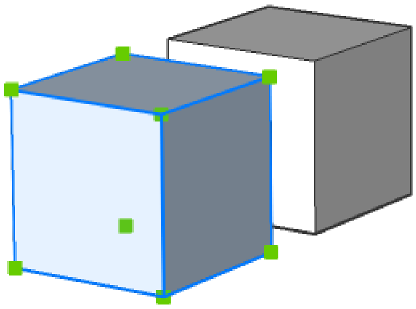
Note: By default, the selection box only selects entire entities (e.g. solids, lines, polylines, blocks, ...) but not sub-entities (e.g. solid faces, solid edges).

1 Click and move the mouse to the right to create a blue box around the entities you want to select. When the entity is completely inside the window, it will be added to the selection set.

1 The example below shows the blue selection box usage when the selection mode is set to **Solids**

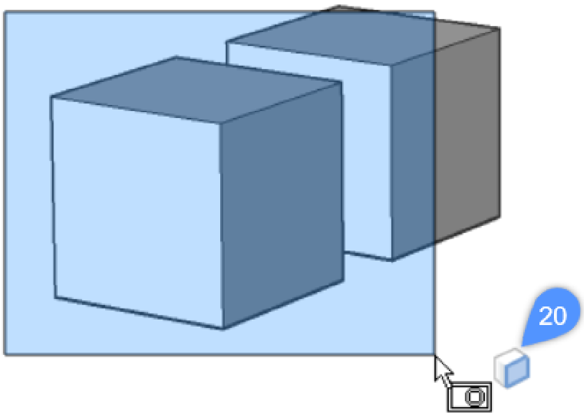
(default).

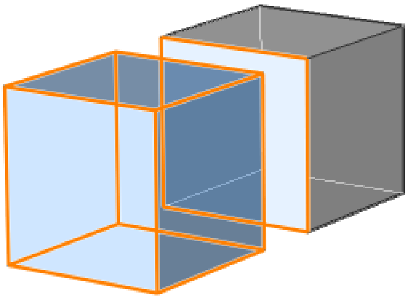




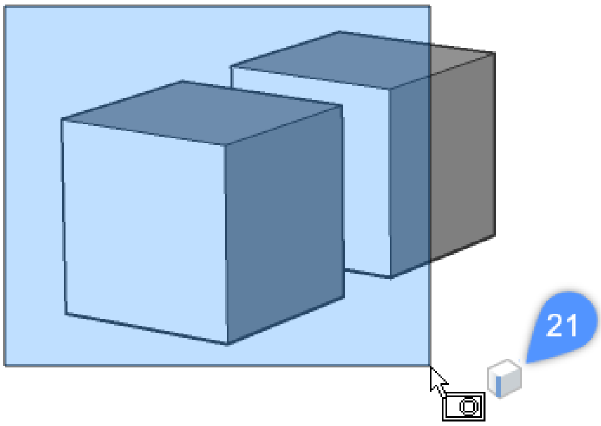
The blue selection box covers the left cube completely and therefore only this cube is selected in the model.

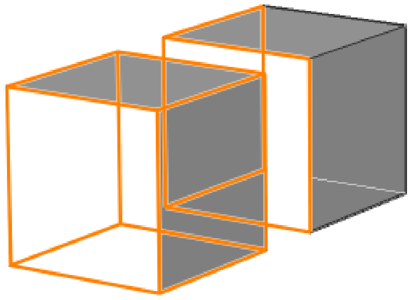
2 To select faces of 3D solids and surfaces, press the Ctrl key once during window selection. The icon (20) next to the cursor indicates the selection mode. The faces that are completely inside the window will be selected when the selection mode is set to **Faces**.





3 To select edges of 3D solids and surfaces, press the Ctrl key twice during window selection. The **Edges** icon (21) appears next to the cursor. The edges that are completely inside the window will be selected when the selection mode is set to **Edges**.

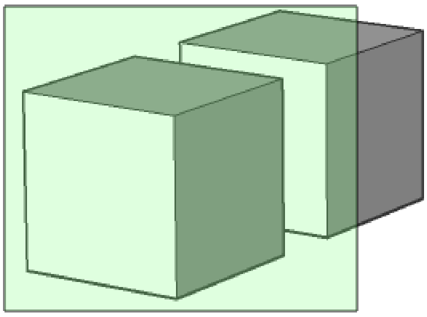


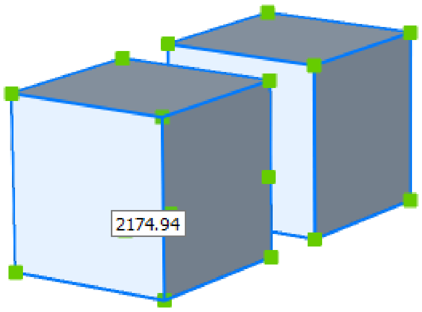


The following steps show how to select multiple entities using the green selection box:

4 Click and move the mouse to the left (crossing window) to create a green box around the entities you want to select. When the entities overlap with the window or are completely inside the window, they will be added to the selection set.

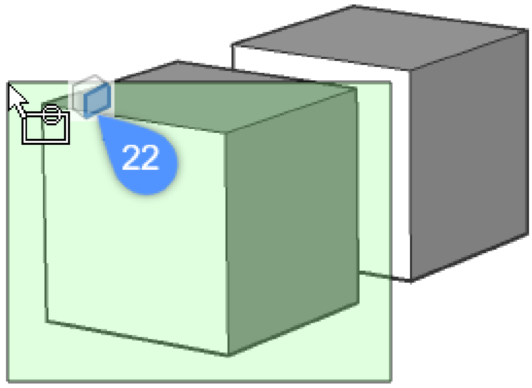
5 The example below shows the green selection box usage when the selection mode is set to default:

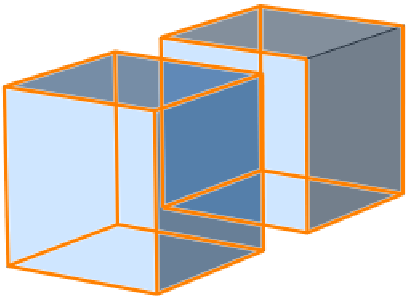




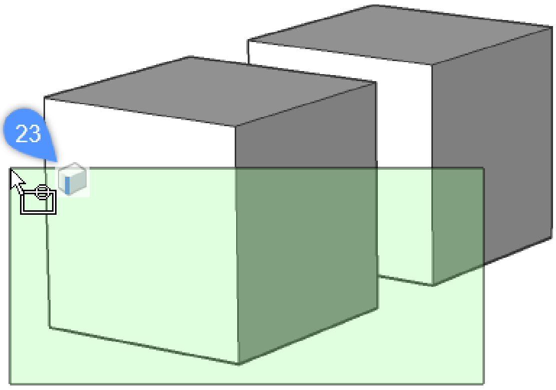
The green selection box selects both cubes in the model as the selection box overlaps with the two cubes.

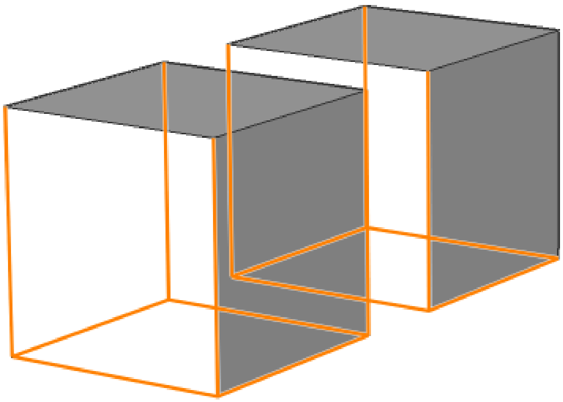
6 To select faces of 3D solids and surfaces, using of the green selection box, press the Ctrl key once during window selection. The **Faces** icon appears next to the cursor (22). Any faces that are completely or partially inside the crossing window will be selected.





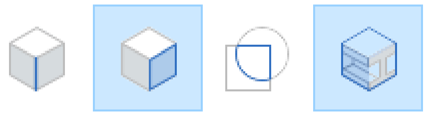
7 To select edges of 3D solids and surfaces, using of the green selection box, press the Ctrl key twice during window selection. The **Edges** icon appears next to the cursor (23). Any edges that are completely or partially inside the crossing window will be selected.



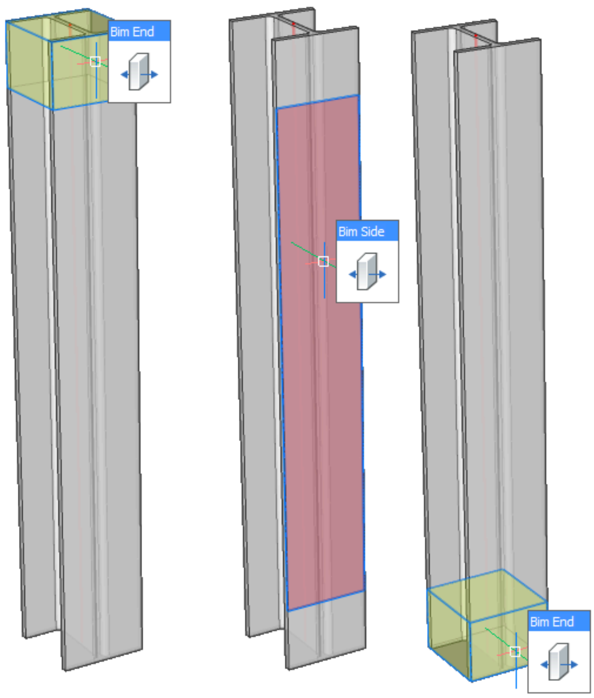


21.5.9 Selecting/highlighting BIM sides and ends on profiles

1 Click the **Enable selection of Sides and Ends** icon in the BIM Toolbar.



2 Hover over the linear solid in the model space when the cursor is on the top or on the bottom side of the linear solid. The **Bim End** face displays in yellow whereas the **Bim Side** face of the linear solid displays in red.



21.6 Level of detail

The RENDERCOMPOSITIONMATERIAL and LEVELOFDETAIL system variables control the display of materials and multiply compositions.

21.6.1 Level of detail

Quad: Bim

Ribbon: **Home** | **View**

Toolbar: **BIM**

Click the Icon to toggle the setting.

Option	Description
 [0] Low	Multiply compositions do not display.
 [2] High	The individual plies of the multiply compositions are visible and editable.

21.6.2 Render composition materials

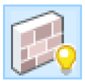
Quad: Bim

Ribbon: **Home** | **View**

Toolbar: **BIM**

Click the Icon to toggle the setting.

Option	Description
 [0] Off	Materials are not rendered.

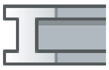
Option	Description
 [1] On	Materials are rendered. When Level Of Detail = 0 (Low), major faces of solids, that have a multiply composition attached, are rendered in the material of the underlying ply. Minor faces are rendered in the material of the exterior ply.

22. Modeling techniques

22.1 BIM linear solid

22.1.1 About BIM linear solid

The Linear Solid tool allows you to create structural members or flow segments. The tool can be made use of creating a variety of structural member profiles or MEP elements.



For more information about this command, visit the Command Reference article BIMLINEARSOLID.

For more information about **MEP modeling**, visit the help article **MEP Modeling** and **MEP Flow Connection Points**.

For more information about **Structural Steel**, visit the help article **Structural Steel**.

Before starting the process, make sure that Quad is active. Next, follow these steps:

1 Hover over the **Model** tab in the Quad and select the Linear Solid tool.

 Note: You can also launch the **BimLinearSolid** command by entering it in the command line.

 You are prompted: Set start point or [Follow/Rectangular/Circular/Library/select in Model/adJust profile] <Last point>:

 Note: The **Follow** and **Last point** options appear only if a linear entity has been previously created in the drawing.

2 Select a start point.

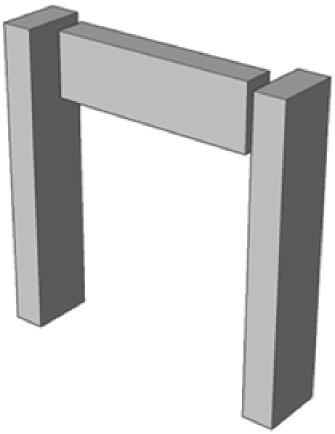
3 You are prompted: Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/adJust profile]:

4 Move the mouse cursor to draw the beam or column.

5 Click to specify the endpoint or enter a value for the distance using the dynamic dimension field.

6 Press Enter key to accept it.

7 The following illustration shows a beam and two columns with the default rectangular profile.

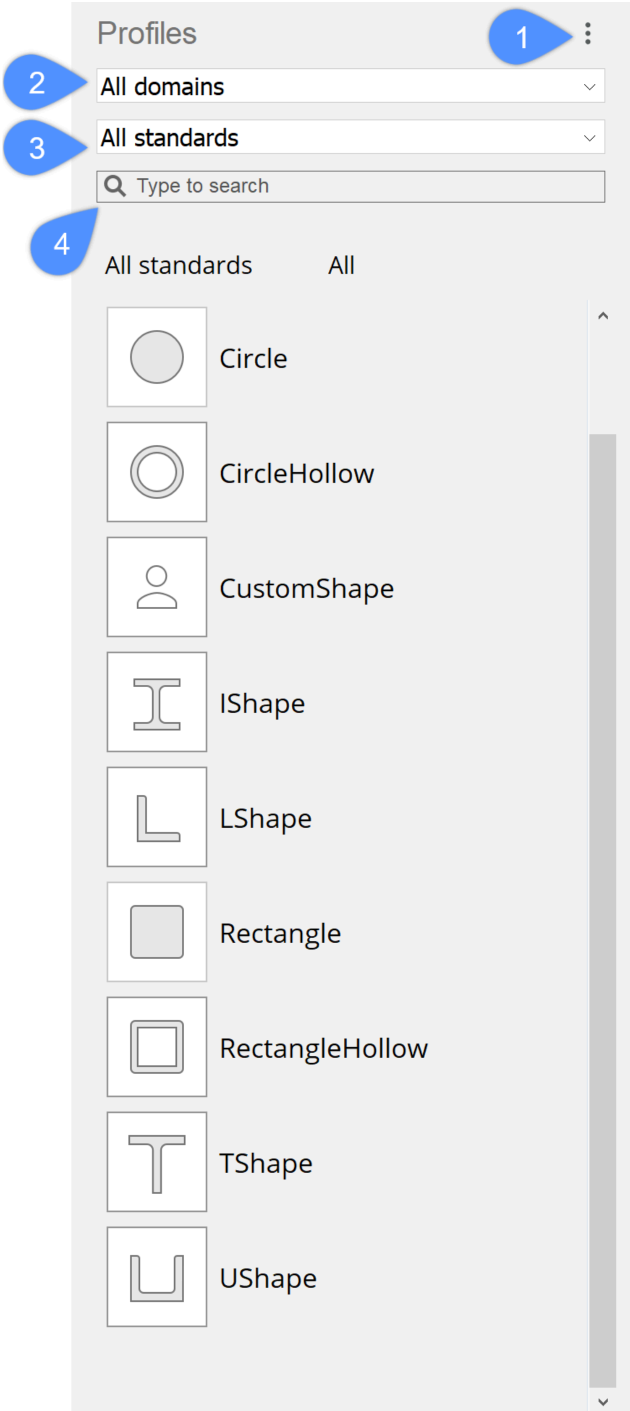


22.1.2 About BIM profiles

You can access the **BIM Profiles** panel by clicking the I-beam on the right-hand side of your screen. If the **BIM Profiles** toolbar is not displayed on the right-hand side of your screen, right-click on the **Tool** panel at the right-hand side of your screen, choose **Panels** in the context menu, then check **BIM Profiles**.

Note: You can also find the **BIM Profiles** panel by typing BIMPROFILES in the Command line.

The **BIM Profiles** panel will be displayed:



The drop-down box at the top (2) acts as a filter. There are different domains on which you can filter: **Generic**, **Structural Concrete**, **Structural Steel**, **HVAC**, **Piping** and **Electrical**.

Note: If you don't have any profiles in a certain domain, you will not be able to filter on it.

The second drop-down boxes (3) allow you to filter the profiles by the country standards.

The search box (4) allows you to filter by name.

The menu (1) on the right-hand side allows you to access the **BIM Profiles** dialog box. Clicking the menu icon displays a context menu with a couple of options.

Select **Open profile dialog** on the menu to open the **Profiles** dialog box.

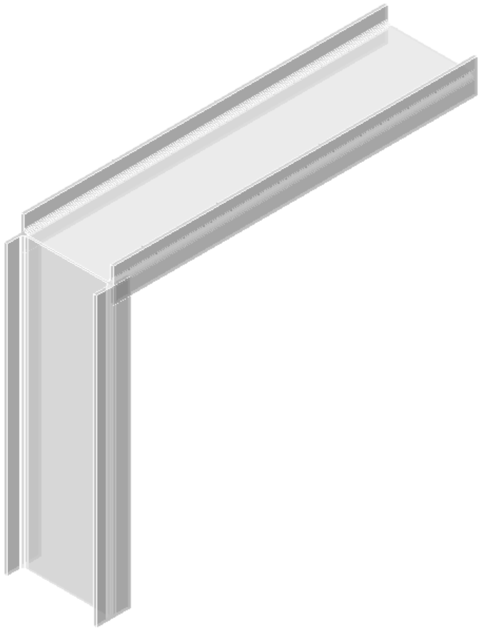
22.1.3 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the **BIM Profiles** panel to the model space.

Note: This command works in the same way as the LINE command.

2 To define the starting point and the following point(s) click in the model space or use the dynamic dimension fields.

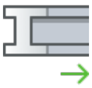
3 Press Enter to end the command.



Note: BricsCAD doesn't automatically generate the connections between the different segments because the chosen profiles are structural steel.

22.1.4 About BIMAPPLYPROFILE

The **Apply Profile** tool allows you to change the profile of linear entities and linear solids. Linear solids have a constant cross-section over a linear extrusion path. The tool recognizes clippings and openings. Using the **Apply Profile** tool, you can create columns, railings, beams, pipes, ducts and more.



For more information about this command, visit the Command Reference article BIMAPPLYPROFILE.

How to apply profiles to an existing line or solid: with the BIMAPPLYPROFILE command

1 Select the line or solid you want to apply a profile to and launch the BIMAPPLYPROFILE command.

2 Press Enter and the **Profiles** dialog box, appears.

3 Choose a profile and press the **Select** button in the bottom-right corner.

4 Press Enter to end the command. The profile is applied to the line or solid in your model.

How to apply profiles to an existing line or solid: with the BIM Profiles panel

1 Open the **BIM Profiles** panel.

Note: You can access the **BIM Profiles** panel by clicking the I-beam icon on the right-hand side of your screen.

2 Select all the entities you want to apply a profile to.

3 In the **BIM Profiles** panel, search for the profile you want apply to the selected entities.

4 Drag the chosen profile onto one of the selected entities.

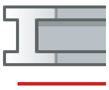
5 The profile is applied to the line or solids in your model.

Note: By applying a profile to a solid, the solid will be replaced by the chosen profile. In case of a line, the line will not be replaced, but a profile was added with his axis the same as the selected line.

6 Since the line is no longer needed, you can delete the line.

22.1.5 About BIMADDECCENTRICITY

BIMADDECCENTRICITY command allows you to control the relative position of the axis of a linear solid.



For more information about this command, visit the Command Reference article BIMADDECCENTRICITY.

1 Select an existing linear solid or profile in the drawing area.

2 Launch the BIMADDECCENTRICITY command by entering it in the Command line or select the command in the Quad (under the model tab) or in the Ribbon (in the **Structural/MEP** tab).

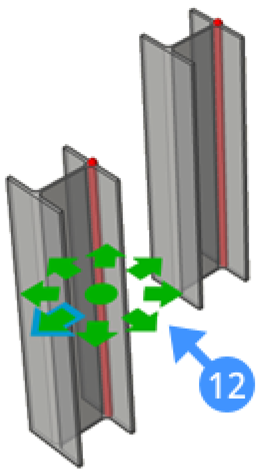
The eccentricity widget appears.

3 To set an eccentricity, click one of the nine locations from the widget, then right-click to confirm.

4 Optionally, type a location in the Command line.

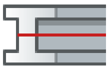
5 The options of the locations are: **TR**: Top Right, **TM**: Top Middle, **TL**: Top Left, **MR**: Middle Right, **C**: Center (resets the axes of the selection set to the default location), **ML**: Middle Left, **BR**: Bottom Right, **BM**: Bottom Middle, **BL**: Bottom left .

6 Press Enter to accept changes. The following illustration shows the Eccentricity widget (12) on the linear solid profile.



22.1.6 About BIM recalculate axis

The Recalculate Axis tool recalculates the axis of linear building elements and repositions it back to the centerline of a linear element.



For more information about this command, visit the Command Reference article BIMRECALCULATEAXIS.

Recalculate the axis of linear building elements

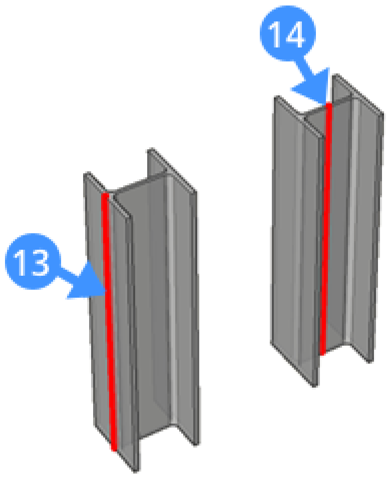
1 Select any linear building element in your model.

2 Launch the BIMRECALCULATEAXIS command.

3 Press Enter.

4 The axis of the element is moved to the default position.

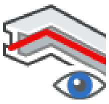
5 The following illustration shows the result of using BIMRECALCULATEAXIS on the linear solid. The axis (13) is moved to the centerline (14) of the linear element.



22.1.7 Helpful settings when modeling Linear Solids

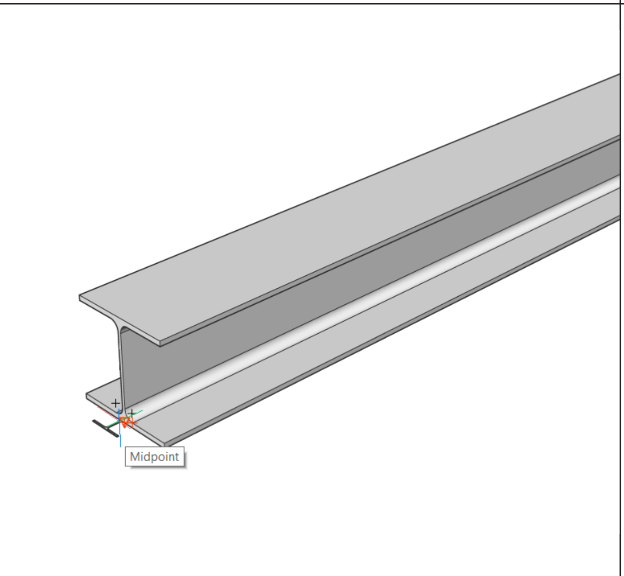
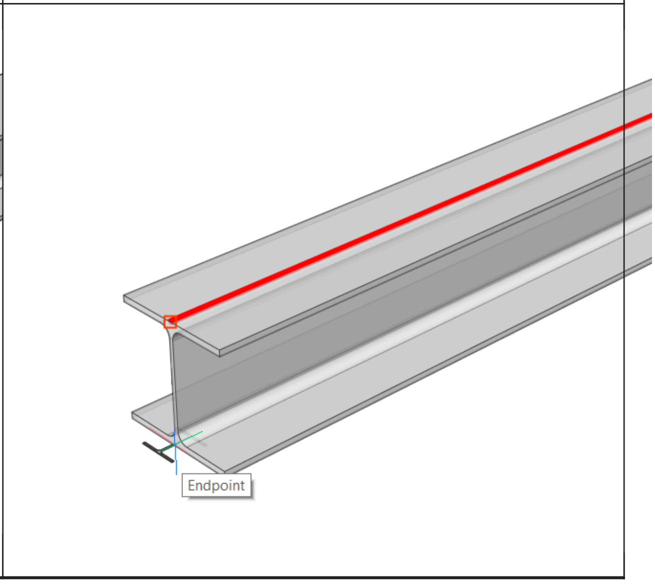
Display Axes

The DISPLAYAXES variable system can hide or view the Axes of linear solids in order to edit the structure more easily.



If DISPLAYAXES is toggled on, the axes of the linear solids are visible as a thick red line. In this case, you will only be able to snap the endpoint of the axis as a starting point to add more linear solids or profiles to your already existing structure.

When DISPLAYAXES is toggled off, no axis is visible in your drawing area and you're able to snap all specific points you have ticked in the context menu of ESNAP.

DISPLAYAXES OFF	DISPLAYAXES ON
	

BIMOSMODE

The BIMOSMODE system variable gives you the possibility to choose the level of detail you are snapping to.

1 Enter BIMOSMODE in the Command line or in Settings.

1 You are prompted: New current value for BIMOSMODE (0 to 3) <2>:

2 Choose a new value (type a number from 0 to 3).

Value	Snapping detail
0	Turns on all snapping.

Value	Snapping detail
1	Turns off all snapping to solids except for the axis of the solid.
2	Turns off all snapping to BIM Grid axes except for axis intersections.
3	Turns off all snapping to BIM Grid axes except for axis intersections. Turns off all snapping to solids except for the axis of the solid.

3 Press Enter

22.2 BIM window create

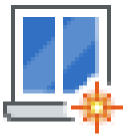
22.2.1 Commands

BIMWINDOWCREATE

22.2.2 About

In BricsCAD, the **Window Creator** tool allows you to turn a different set of design ideas into a window. You can use the tool on windows of all shapes and sizes, which enables you to utilize a significant amount of window styles in your BIM project.

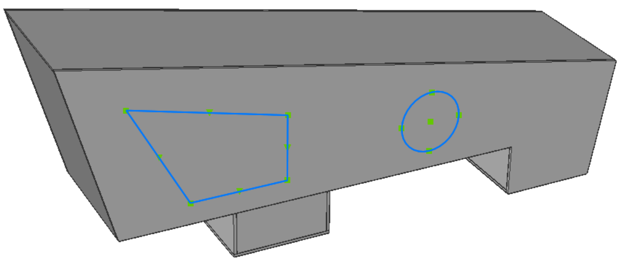
When you have a closed 2D entity or boundary on the face of a solid (wall), the window creator tool enables you to use these closed entities to make your window. Using the window creator tool, you can select any window styles in the dialog box and change the properties of this created window in a BIM model.



22.2.3 Creating a parametric window using boundaries

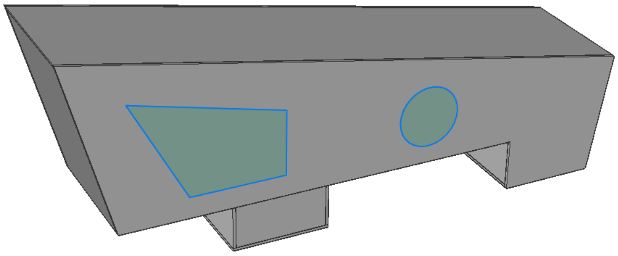
1 Draw a polygon on the face of a solid (wall).

1 Based on this base profile, the window opening will be created.

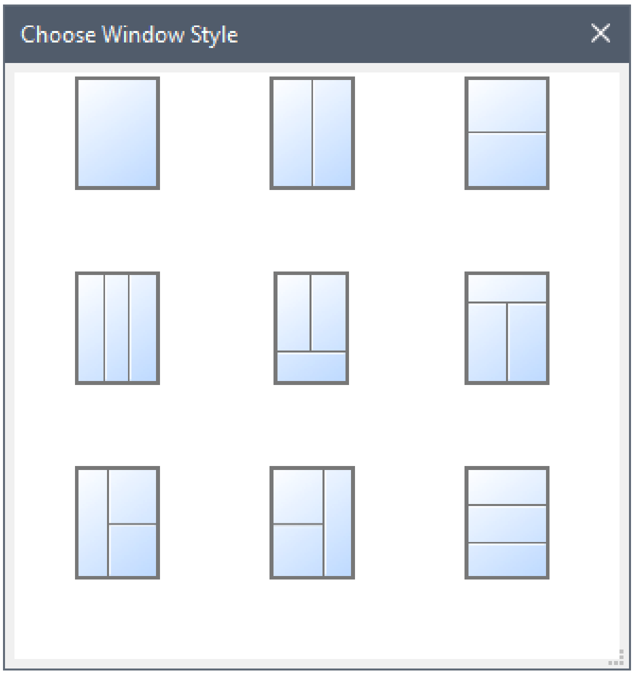


2 Make sure **Boundary Detection** (select detected boundaries) is enabled into SELECTIONMODES system variable. Hover over the inside of the polygon and select the **Create Window** tool from the

Quad.

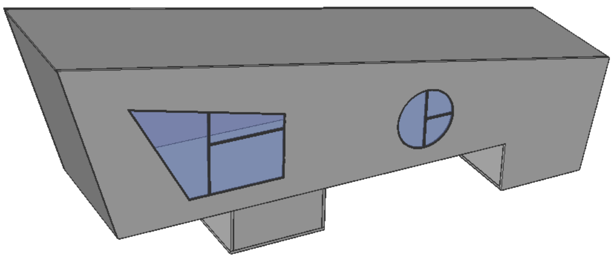


A **Choose Window Style** dialog box displays:



3 Select a window style in the dialog box.

4 The window is created on the wall with the selected style. The following illustration shows the triple panel left window style which was selected to create the windows on the face of the wall.



5 (**Optional**) You can refine the window design by changing the default values of the parameters. To do this, select the window and change the desired parameters in the **Properties** panel.

Instance properties	
Instance name	Triple Panel Left Window:4
Component file	
Parameters	
LiningThickness	60 mm
LiningDepth	60 mm
MullionThickness	60 mm
TransomThickness	60 mm
FirstMullionOffset	0.5
FirstTransomOffset	0.5
GlazingDepth	30 mm
GlazingOffset	30 mm
PlacementDepth	50 mm
LeftPanelThickness	60 mm
LeftPanelDepth	60 mm
BottomPanelThickness	60 mm
BottomPanelDepth	60 mm
TopPanelThickness	60 mm
TopPanelDepth	60 mm

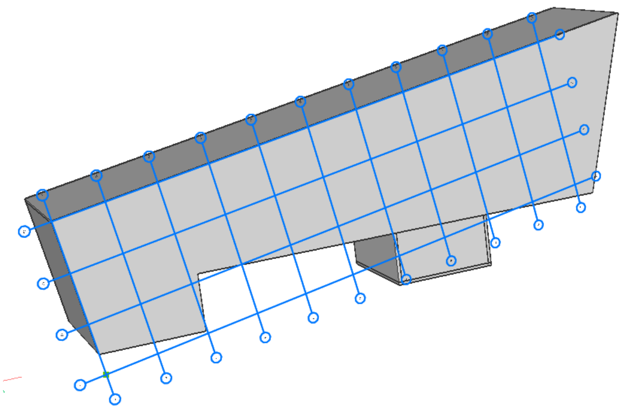
To change one of the default parameters of this window, type a new value in the property field.

Press **Enter** to accept the new value of the parameter. The window should be updated automatically.

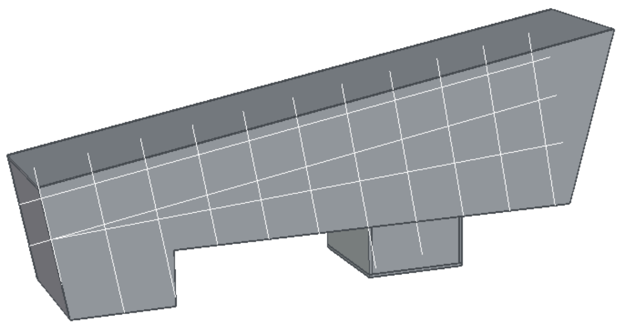
22.2.4 Create windows based on a grid

1. Use BIMGRID command to create a Grid on the face of the wall. Choose the basic dimensions in the Command line. Changing the **U-line** and **V-line** panel options allows you to specify the desired size and number of panels on the surface.

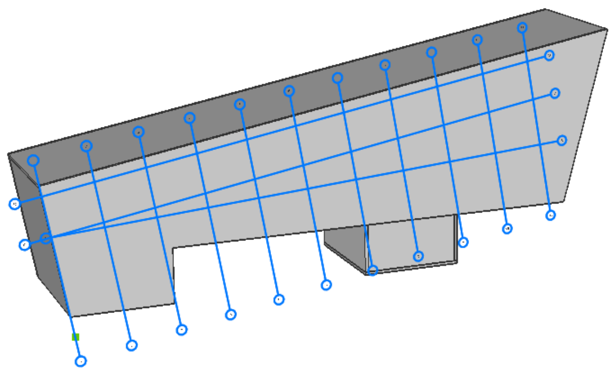
   Note: To draw on the face, make sure you first highlight the face and then click to start drawing. Otherwise, the Grid will be created in the XY-plane.



2. Use BEDIT command or REFEDIT command to edit the Grid.
3. Use the Manipulator to change the position of the gridlines on the wall.

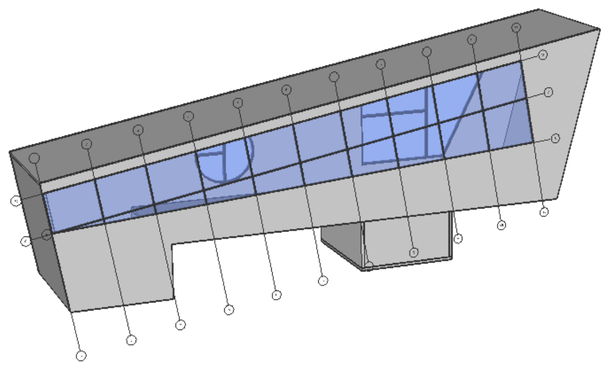


4 Select the Grid and then select **Create Window** in the Quad.



5 The window is generated based on the gridlines.

6

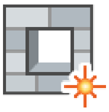


Note: You can edit the parameters in the **Properties** panel. Type a new value in the property field and press **Enter**. The window will automatically update itself.

Parameters	
FrameDepth	60 mm
GlazingDepth	30 mm
GlazingOffset	30 mm
PlacementDepth	50 mm

22.2.5 Creating window openings

BIM Opening Create creates an opening without window geometry. It uses the same command as BIMWINDOWCREATE. The command allows to create an opening at the location of the selected base profile. The base profile can be a boundary or a closed polyline.



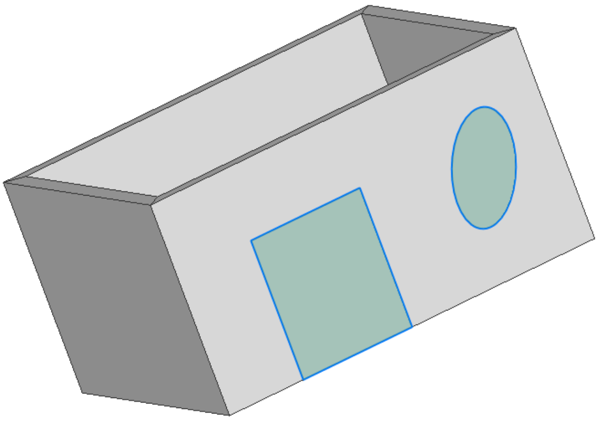
For more information about this command, visit the Command Reference article BIMWINDOWCREATE.

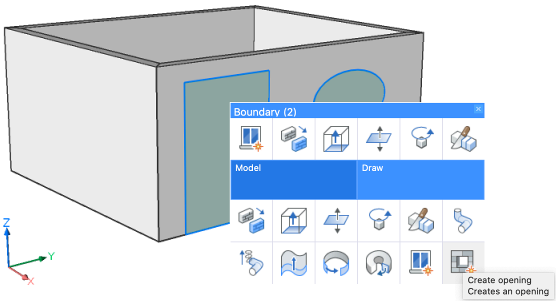
1 Draw one or more closed polylines on the face of a wall.

2 While boundary detection is on, hover over the polygon.

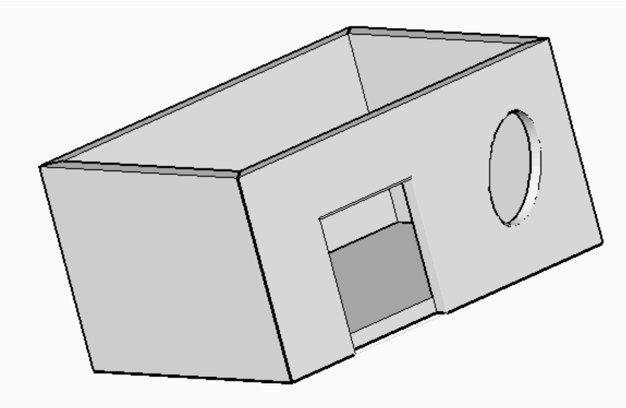
3 Or select the closed polyline.

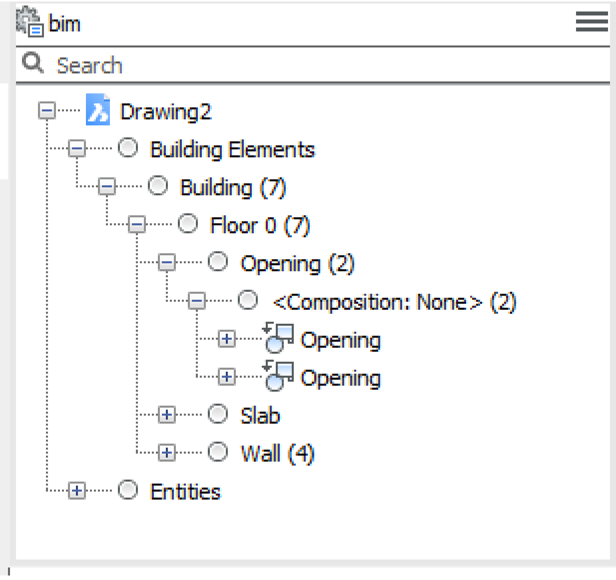
4 Select the **Create Opening** tool from the Quad.





5 The opening is created based on the closed contour. In the **Structure** panel, the opening is added as a block reference. The block reference is classified as an opening.





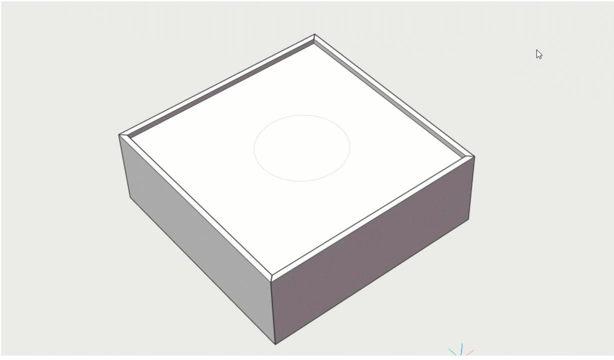
If you select the opening from the **Structure** panel, you will see a set of properties related to this opening element in the **Properties** panel.

BIM	
Type	Opening
Building Element Type	Opening
Name	
Description	
Building	Building
Story	Floor 0
Space	
Composition	
GUID	1CAb2$VRL1t9SusKzjHN25
Number	
Entity property sets	
Instance properties	
Mechanical component	
Component's material	
Parameters	
W	183 cm
H	215 cm

Note: If your closed contour has a rectangle shape, the opening element will include **Width** and **Height** parameters.

> ***Note:*** Opening elements contain subtractor solids. To display the subtractor solid, open the **Layers** panel and select the **BC_SUBTRACT** layer.
>
> ***Note:*** The associative tags can be generated using the BIMTAG command for the opening element in a sectioned drawing.

The following animated GIF file demonstrates how to create an opening based on a closed contour.



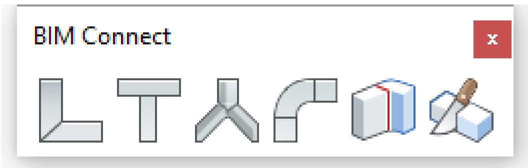
22.3 Connecting solids

22.3.1 Commands

LCONNECT, TCONNECT, BIMSTRUCTURALCONNECT

22.3.2 About

Walls, profiles, plies, etc. are all created as native 3D solids. So, any of the 3D Solids Editing commands can be used to modify them. Use **BIM Connect** commands to connect two solids in different ways.



The LCONNECT command creates an L-connection (bisector or parallel) between two solids.

The TCONNECT command creates a T-connection between two solids.

The BIMSTRUCTURALCONNECT command connects structural profile solids such as beams and columns.

The BIMFLOWCONNECT command creates a connection between flow segments, such as pipes or HVAC ducts.

The procedures below use the Quad cursor menu.

22.3.3 Procedure: creating L-connections

1. Make sure SELECTIONMODES system variable = 0.
2. Type **LConnect** in the Command line or select it via the Quad.
3. Select two solids or faces to connect.
4. The two selected solids are connected: the default connection type is L-bisector.

5 If HOTKEYASSISTANT system variable is ON (HKA is switched on), the Hotkey Assistant widget displays.

Use the CTRL key to cycle between the connection types.

You are prompted: Select base solid [Switch] <Accept>. Type **A** to use the selected connection, type **S** to switch to the desired connection type.

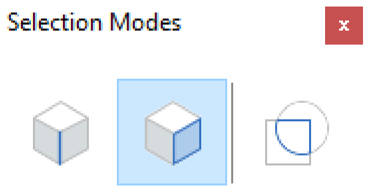
The 3D model updates dynamically when cycling the L-connection types.

Note: L-connections can also be created between non-vertical solids such as roof slabs.

22.3.4 Procedure: Creating T-connections

The following procedure allows connecting minor faces of a solid to the major faces of other solids, regardless of what the orientation of the faces is. As a result, you can use TCONNECT command to T-connect solids.

1 Make sure the **Select Faces** option of the SELECTIONMODES system variable is selected.

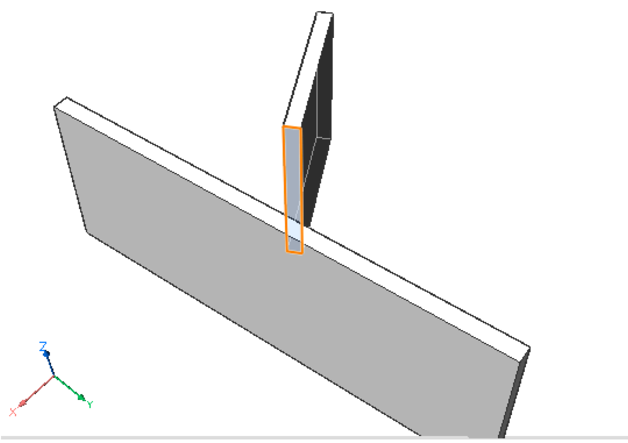


2 Type TCONNECT in the Command line or select it via the Quad.

3 Move the cursor over the minor face of the first solid at the end that needs to be connected.

4 Press Tab to highlight obscured faces.

Click when the face highlights.



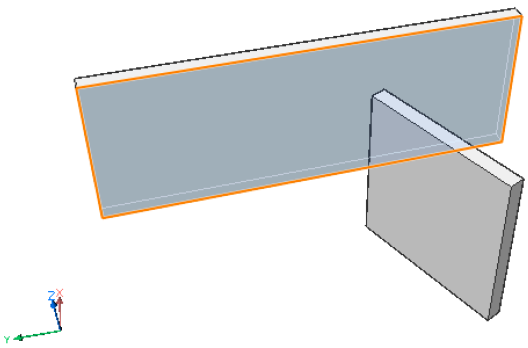
5 (**Optional**) Repeat the previous step to select more faces, then press Enter.

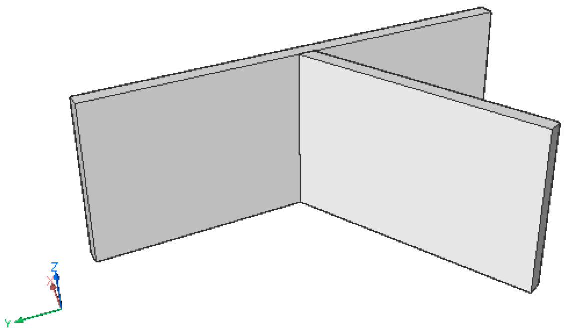
Tip: The SELECTALIGNEDFACES command in the **Select** command group in the Quad selects all faces in the 3D model that are in the same plane (coplanar) with a selected face.

6 You are prompted: Select entities to connect to or [Connect to nearest/Disconnect/selection options (?)] <Connect to nearest>:

7 Do one of the following:
- Press Enter to connect to the nearest face(s).
- Select one or more faces.
- If necessary, press Tab to highlight an obscured face; click when the face is highlighted.

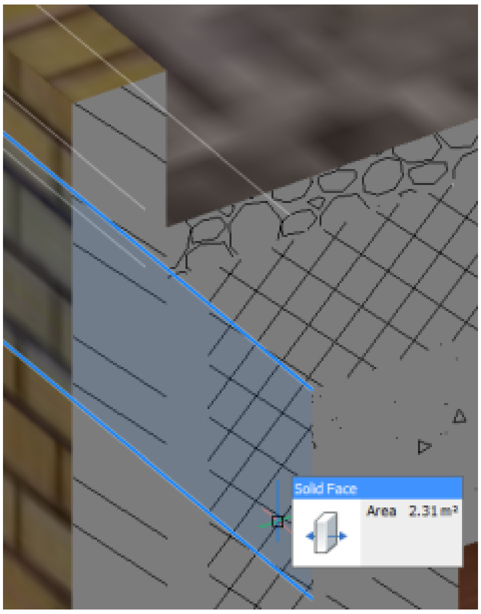
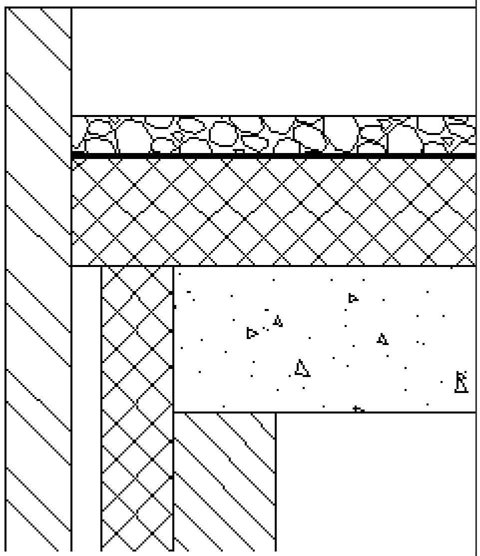
  Right click to stop selecting faces.





Tip: Hover over the minor face of the solid and choose **Connect with Nearest in the Model** command group of the Quad.

Creating T-connections ply-by-ply

	
3DModel	Calculated Section

To control the connection of composition plies between two building elements, you can **Push/Pull** a ply in a clipped BimSection (if Show Composition is ON). For more in-depth information, visit the BIM Ply Editing article.

To create a T-connection ply-by-ply in a section, do the following.

1 Place the cursor over the face of the ply you want to connect. If necessary, repeatedly press Tab to select an obscured face.

2 When the ply face highlights, choose **Push/Pull** in the Quad.

3 Specify to which point the selected ply should reach by moving the cursor or typing in the distance.

4 Right-click or press Enter to confirm.

5 The ply of the first solid is connected to the ply of the second solid. The connected ply is subtracted from the second solid.

6 Optionally, use the BIMPROPAGATEPLANAR command to apply the connection to similar locations in the model.

22.3.5 Procedure: Setting the distance between parallel solids

1 Select the solid of which the position must be fixed.

2 Select the solid which will be moved when defining the new distance.

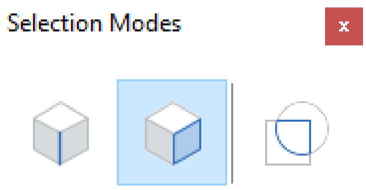
3 The nearest distance between the solids displays.

4 Type a value in the distance field and press Enter.

22.3.6 Procedure: Creating a copy of a solid

Creating a parallel copy of a solid

1 Make sure the **Select Faces** option of the SELECTIONMODES system variable is selected.



2 Move the cursor over a face of the solid to be copied.

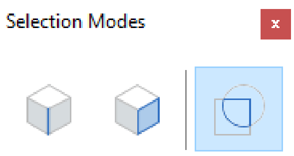
3 When the face highlights, choose **BimCopy** in the **Model** command group of the Quad.

A copy of the solid displays dynamically. The ruler indicates the current distance.

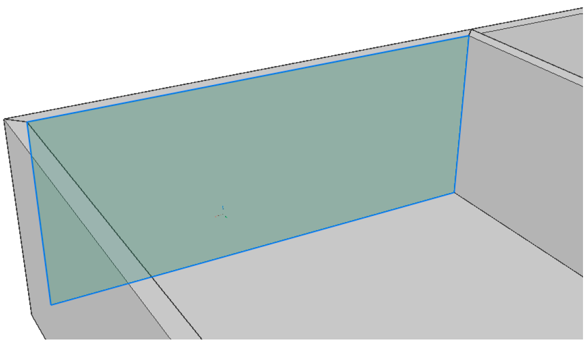
4 Type a value in the dynamic entry field and press Enter.

Creating a partial copy of a solid

1 Make sure the **Enable boundary detection** option of the SELECTIONMODES system variable is selected.



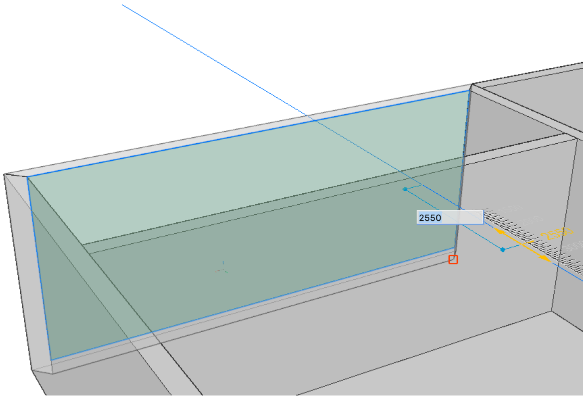
2 Highlight the part of the boundary for the part of the solid you want to copy.



3 When the face highlights, choose **BimCopy** in the **BIM** command group of the Quad.

A partial copy of the solid displays dynamically. The ruler indicates the current distance.

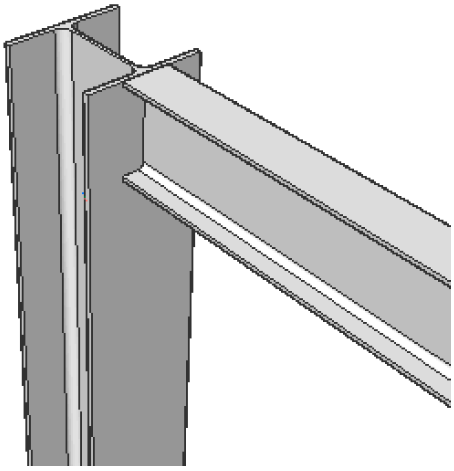
4 Type a value in the dynamic entry field and press Enter.



22.3.7 Procedure: disconnecting two connected solids

In some cases, you need to disconnect 2 solids of a node to do some further direct modeling.

You select two solids and it breaks the connection by separating the two solids physically and removes all end clippings at the side of the connection. As such, these solids can direct modeled upon again.



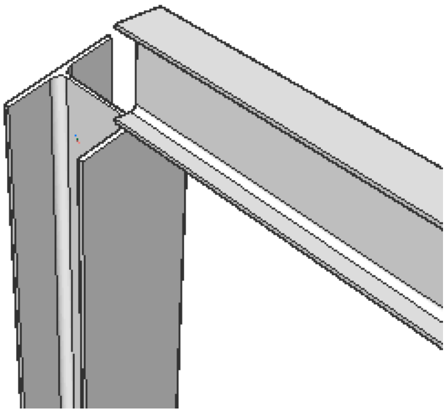
1 Type LCONNECT in the Command line.

2 Select the two solids.

3 The connection is automatically made, press CTRL until the disconnect icon is highlighted.

4 Press Enter.

5



Note: You can use Propagate to disconnect similar nodes.

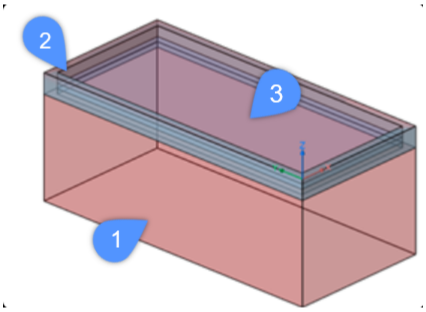
22.4 Creating a custom window or door

22.4.1 About

In BricsCAD, you can create any geometry or component from scratch. Once you have your geometry in 3D you can insert and use it in your BIM model. You can make changes or add more details to your window or door at any time.

This process is explained in 4 parts to make the steps of creating a 3D window or door geometry straightforward.

- Create the file
- Create the subtractor solid (1)
- Create the window or door frame (2)
- Create the (Glass) panes (3)



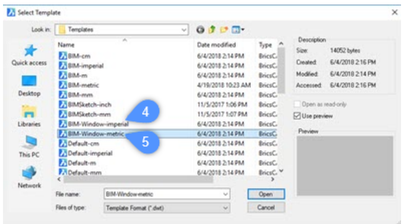
22.4.2 Creating a custom window

Step 1: Creating the file

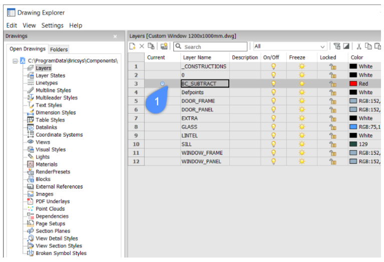
1 Click **New** in the **File** menu.

2 Select one of the pre-defined templates to create a simple window specifying the unit system of your drawing.

3 ***Note:*** The pre-defined templates contain the layers needed to create the geometry of the window or door. These DWT file format templates are **BIM-Window-imperial** (4) and **BIM-Window-metric** (5) which assure the required layers and properties are displayed in your drawing.

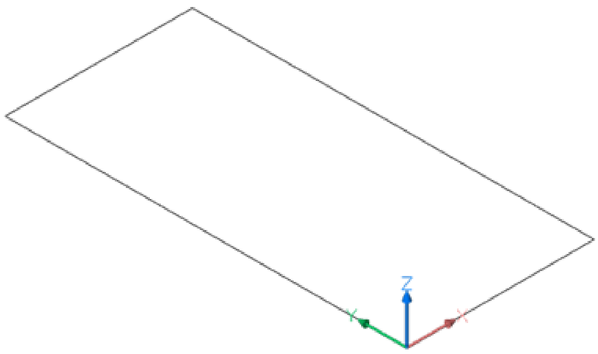


4 Set the **BC_SUBTRACT** layer current (6) either using the drawing explorer or layer field in the **Properties** panel or **Layers** panel on the right-hand side.



5 Create a closed polyline (e.g. a rectangle) that represents the form of the opening for the insert.

6



7 Save the file in any location. The library of user defined components is located in the Components folder. The folder's path is stored by the COMPONENTSPATH system variable.

8 By default this is: *C:\ProgramData\Bricsys\Components*. You can add more folders using the **Settings** dialog box.

Note: The library of predefined components can be found under *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support\en_US\Bim\Components*.

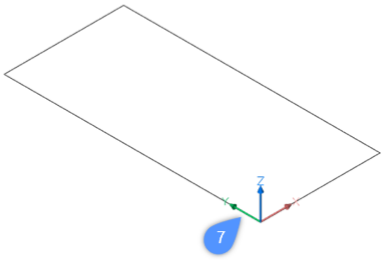
Step 2: Creating the subtractor solid

The subtractor solid will be used to create an opening.

Note: The **BC_SUBTRACT** layer must be set as the current layer.

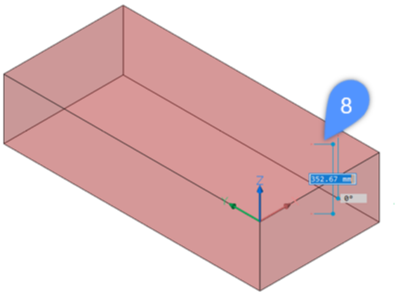
1 Use the outline of the solid which you have previously created (e.g. a rectangle).

1 The outline lies in the first quadrant of the XY plane of the World Coordinate System (WCS).



The lower left corner of the rectangle lies at the origin (0,0,0) of the WCS (7).

2 Select the **Extrude** tool and move the cursor downward (8) to extrude your solid outline in the negative Z direction.



3 Do one of the following to finish the subtractor solid creation.

- Click a point.
- Type a value in the dynamic dimension fields, then press **Enter**.

Note: Make sure that the height of the extrusion is greater than the width of the wall the component will be inserted into.

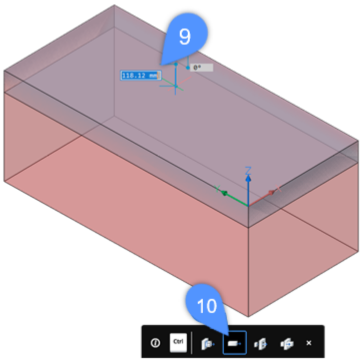
Step 3: Creating the fixed frame

The outline of the fixed frame coincides or is parallel (in case of a rebate) with the outline of the subtractor solid.

1 Set the **WINDOW_FRAME** layer current.

2 Highlight the top face of the subtractor solid.

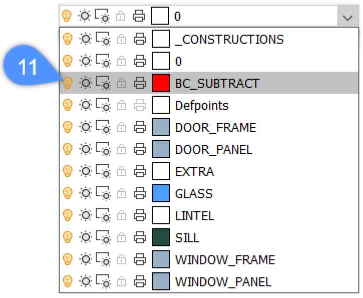
3 Next, extrude the top face of the subtractor solid in the negative Z direction (9) using the **Extrude** tool ‘**Create**’ (10) option by pressing the **Ctrl** key once.



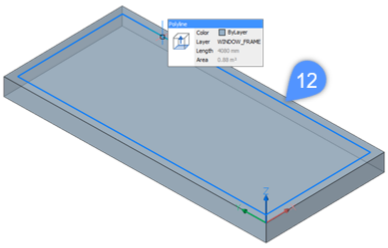
4 Type a value in the dynamic dimension field to specify the height of the fixed frame.

5 ***Note:*** The height of the extrusion will be the thickness of the fixed frame.

6 Turn off the **BC_SUBTRACT** layer by clicking the light bulb (11) next to the layer name.

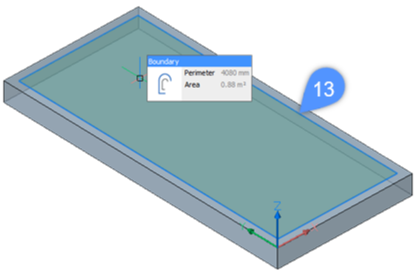


7 Use the **Offset** tool to create a parallel offset of the outline of the fixed frame (12).



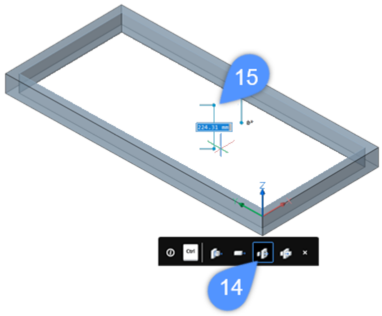
Note: The distance of the offset equals the width of the frame.

8 Hover over the inside boundary of the parallel offset (13) and select **Extrude** tool from the Quad.



9 When the extrude mode is on the **Subtract** option (14), move your cursor downward (15) to create an

opening in the fixed frame.



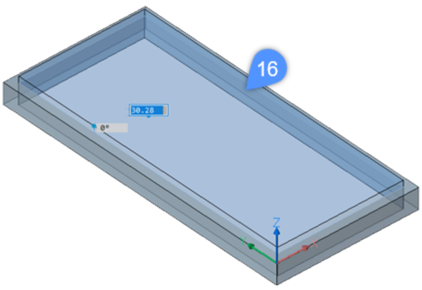
10 Do one of the following to finish the creation of the opening in the fixed frame.
- Click a point.
- Type a value in the dynamic dimension fields, then press **Enter**.

Step 4: Creating the glass panes

1 Set the **GLASS** layer current.

2 Use the DMEXTRUDE command to extrude the inside boundary of the window frame.

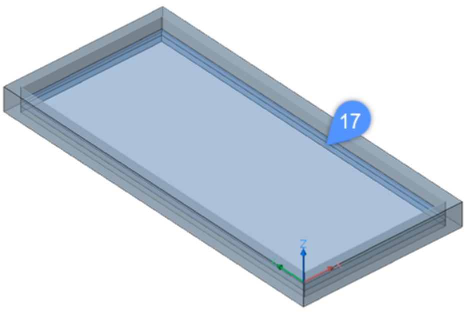
3 ***Note:*** The height of the extrusion equals the thickness of the glass pane. The following illustration shows the inside boundary of the window frame after using DMEXTRUDE command to create a window glass pane.



Note: To create a glass pane, the **Create** option should be selected from the Tooltip of **DMExtrude**.

4 Move the glass pane solid down in the negative Z-direction if the outside face of the fixed frame does not coincide with the outside face of the solid (wall) the insert will be placed in.

5 The following illustrations show the glass pane solid after moving it (17) down.



6 Optionally, before saving the drawing, you can classify the entire drawing as a BIM window element.

7 To do so, follow these steps:

- Type BIMCLASSIFY in the Command line.
- Press 'I' for window and press **Enter** to accept it.
- Next, press '**D**' for Drawing and press the **Enter** key until the command is executed.

The drawing is now classified as a window. Thus, it will automatically be recognized as a BIM Window element when inserted into a new drawing.

8 Save the drawing.

9 Optionally, you can assign more information to the window after inserting it into a new drawing. The following illustration shows the **Window Common** category of BIM properties from the **Properties** panel that becomes available after selecting the element. Information can be assigned about the thermal transmittance coefficient of the window materials, fire rating according to the national fire safety classification, and so on.

Window Common	
Reference	
Fire rating	
Acoustic rating	
Security rating	
Is external	Off
Infiltration	0 m^3/s
Thermal transmittance	0 $W/m^2 \cdot K$
Glazing area fraction	0
Smoke stop	Off

22.4.3 Adding 2D entities (door swing)

Optionally, windows and doors drawings can contain 2D symbols. These 2D symbols will either replace the actual **3D Solid** section of the window or door or be added to the **3D Solid** section, depending on their layer:

- Symbols on a layer with prefix "**BRX_2D_**" or " **BRX_2D+_**" will be added to the section block:
  - without being processed by analytical hidden line removal (HLR) when the section plane intersects the components.
  - after going through analytical hidden line removal (HLR) procedure when the section plane does not intersect the components.
- Symbols on a layer with prefix "**BIM_2D_SECT_**" or "**BIM_2D_SECT+_** " will be added *only* when the section plane intersects the components. All geometry goes through analytical hidden line removal (HLR) procedure.
- Symbols on a layer with prefix "**BIM_2D_BACK_**" or "**BIM_2D_BACK+_**" always will be added and all geometry goes through analytical hidden line removal (HLR) procedure.
- Symbols layers name
  - with "**+**": both the 2D symbol and the 3D solid section of the component are shown in the section result.
  - without the "**+**" only the 2d content is shown and all solids in the component are ignored for

section generation.

Note: **BRX_2D** is still available only for backwards compatibility.

To add a 2D symbol to your component follow the next steps:

1 Add a new layer '**BRX_2D+_***' and set it as the active layer. The 2D geometry added to this layer will be visible when you cut a section through this component and generate the section onto a sheet. The cut-through representation of the 3D geometry in the component will be visible as well.

2 Before starting drawing the 2D symbol of the component, make sure the Z axys is perpendicular to the bottom plane of the component and pointing upwards. You can do this either by switching your UCS (using UCS command, rotating it 90 degrees over the X-axis) or by enabling the **Dynamic UCS** (DUCS) feature.

3 Switch your view to the **Top** view by using the **Look From** widget.

4 Start drawing the 2D geometry on the '**BRX_2D+_***' layer.

Note: Use only **Polyline** entities, as these will be recognized when generating the component's section (while the **Line** entities will not be recognized).

5 Switch back to a 3D view of your model and check that the following are visible:

- the solids on the **BC_SUBTRACT** layer
- the component's solids
- the newly drawn 2D geometry

22.4.4 Parametrizing your component

You can do this first for the 3D geometry and then for the 2D geometry.

Note: For more information about parametrizing, see the Guide articles **Using parametrize** and **Working with parameters and constraints**.

1 Select all 3D geometry of your component, including the geometry under the **BC_SUBTRACT** layer (make sure this layer is turned **On** first).

2 Run the PARAMETRIZE command for the selected geometry and check the contraints that appear in the **Mechanical Browser** panel.

3 Optionally, you can rename the parameters as needed (for example, you can rename the '**Length_X**' user parameter as '**Width**', etc.). You can also change some of the constraints and parametric behavior previously defined.

4 Use the UCS command to change the UCS so that the 2D geometry lies in the XY-plane of the UCS.

5 Select the 2D geometry previously added to your component.

6 Run the PARAMETRIZE2D command for the selected geometry and check the contraints that appear in the **Mechanical Browser** panel.

7 Repeat step 3. Make sure to connect the newly created 2D parameter on the X-direction to the '**Width**' 3D parameter by typing this 3D parameter's name as the 2D parameter's value.

Note: The 2D constraining works differently than the 3D constraining, but you can use the same parameter names to control both 2D and 3D types of constraints together.

8 Switch back your UCS by selecting the **World** option of the UCS command. This will prevent undesired results when inserting the component into another drawing.

9 Turn the **BC_SUBTRACT** layer **Off**.

10 Save the drawing.

11 Insert the component into your model from the **Library** panel or by using the BMINSERT command. With the component selected, change the parameters from the **Properties** panel to check that everything behaves as expected.

22.5 Creating roofs

22.5.1 The roof creation workflow consists of the following steps:

1. Creating the roof footprint.
2. Extrude the footprint and create a hip roof (EXTRUDE command).
3. Adjust the slope of the roof faces if necessary (**Manipulator** widget).
4. Defining the thickness of the roof solid (**Body/Shell** option of the SOLIDEDIT command).
5. Create additional roof components.

22.5.2 Creating the roof footprint

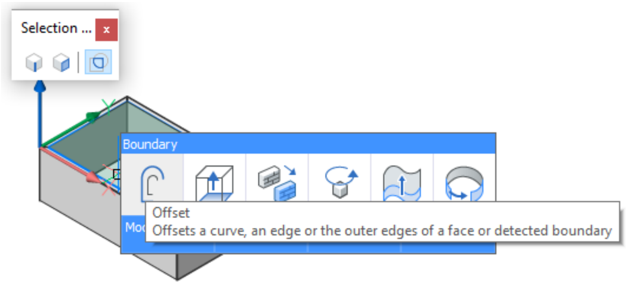
You can create the roof footprint:

- From the exterior walls boundary.
- Creating a closed polyline by clicking points or other 2D procedures.

To create the roof outline from the exterior walls boundary

1. Isolate the exterior walls. Isolating entities is easily done using the **Structure** panel. See also the ISOLATEOBJECTS command.
2. Set the coordinate system to the top face of the exterior walls:
   - Type UCS and press **Enter**.

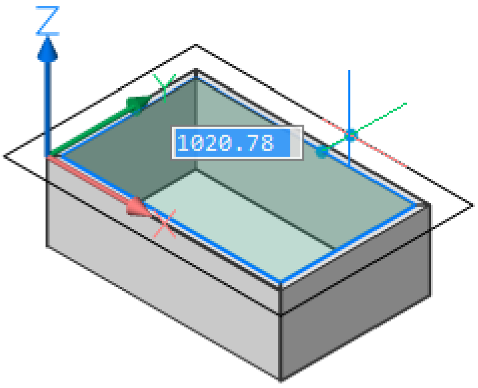
     You are prompted: Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:
   - Snap to a top corner of an exterior wall, then press **Enter**.
3. Set the **Enable boundary detection** ON in the **Selection Modes** toolbar or include the **Select detected boundaries** into SELECTIONMODES system variable.



4 Move the cursor to a void area inside the exterior walls.

5 The inside boundary highlights.

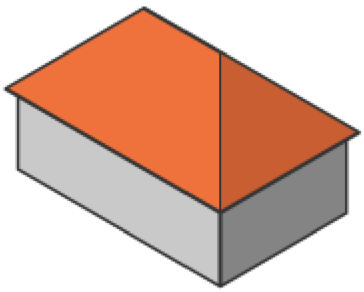
6 Choose **Offset** in the **Draw** tab of the Quad.



7 Do one of the following:

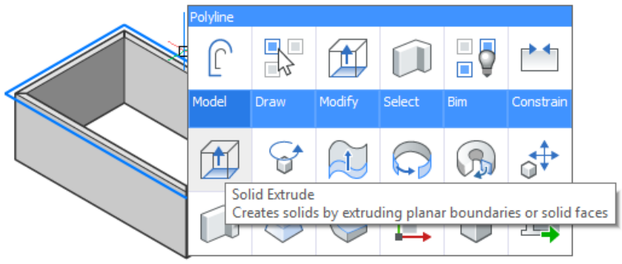
- Type a value in the dynamic distance field.
- Click to accept the value.

22.5.3 Creating a hip roof

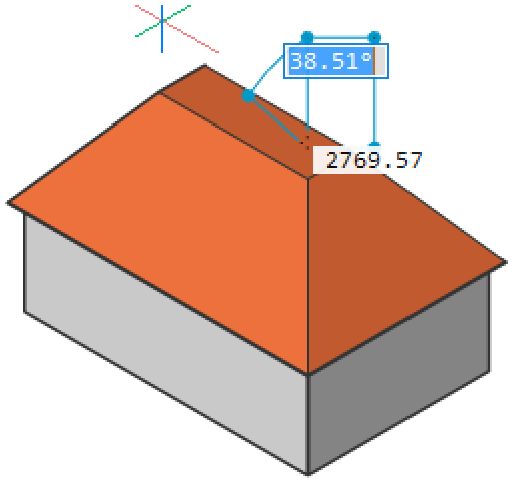


1 Create the roof footprint.

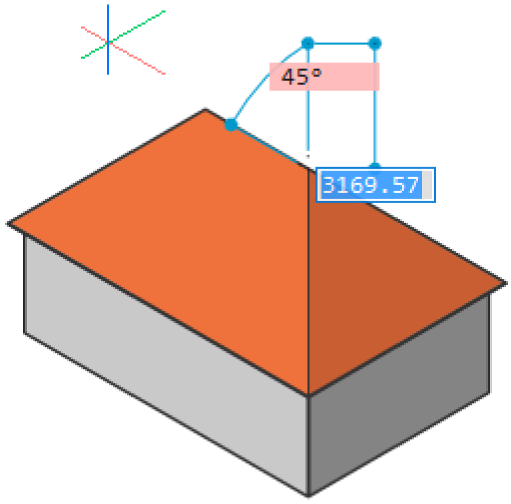
2 Move the cursor over the roof outline and choose **Extrude** in the **Model** tab of the Quad.



3 Move the cursor vertically, then hit the **Tab** key.



4 Type a value in the **Taper Angle** field and hit the **Tab** key.

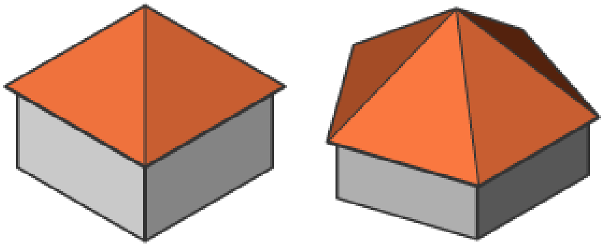
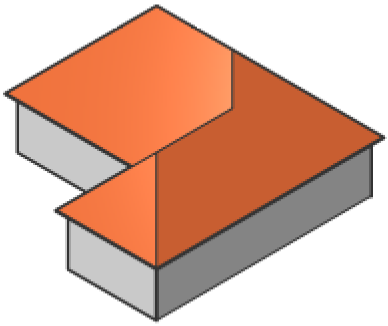
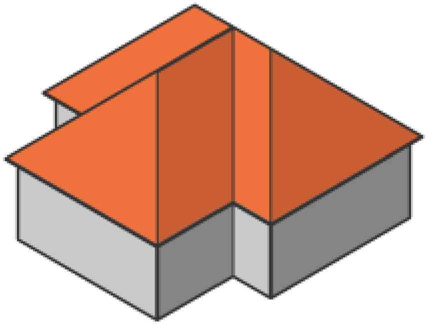


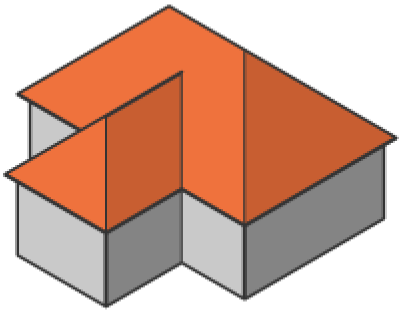
5 Do one of the following:

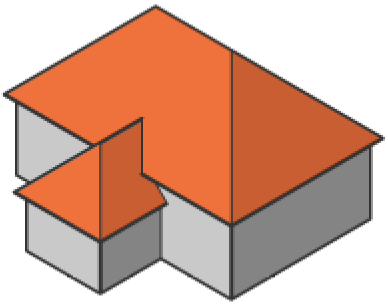
- Hit the **Tab** key and type another value in the **Taper Angle** field.

  The **Taper Angle field** is the complement of the inclination angle of the roof.
- Press **Enter** to create the roof.

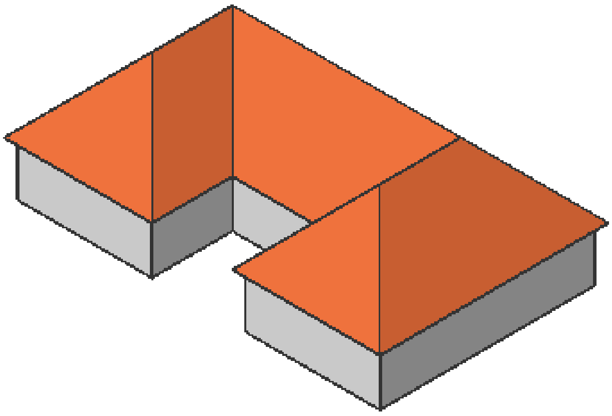
6 Depending on the shape of the roof outline, the result is:

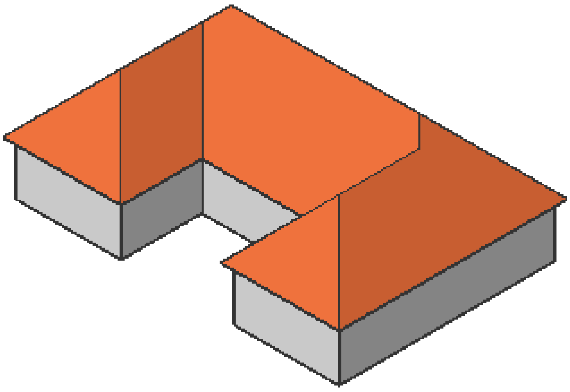
- Pyramid roof:
- 
- L-shaped hip roof:
- 
- T-shaped hip roof:
- 





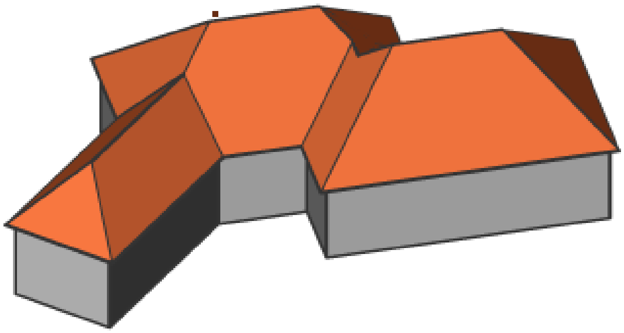
- U-shaped hip roof:
-





- Complex hip roof:

-



7 To define the thickness of the roof solid we will use the **Body** > **Shell** option of the SOLIDEDIT command.

Do the following:

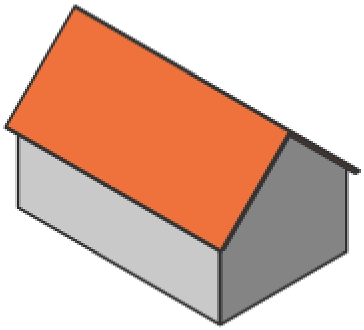
a Choose **Shell** in the **Modeling** > **SOLID EDITING** ribbon panel.
Choose **Shell** in the **3D Solid Editing** toolbar.

b You are prompted: Select 3d solid: - Select the roof solid.

c You are prompted: Select face to remove or [Add/Undo/ALL]: - Select the bottom face of the roof solid.

d You are prompted: Enter the shell offset distance: - Type a value to define the thickness of the roof solid.

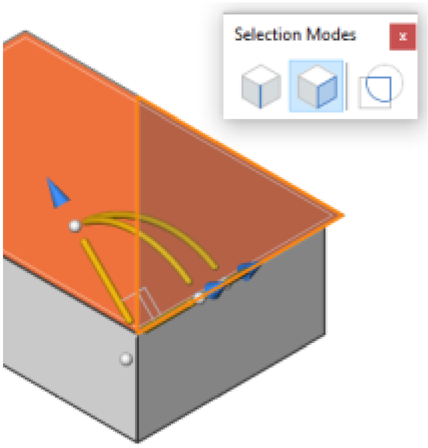
22.5.4 Creating a gable roof



1 Create a hip roof.

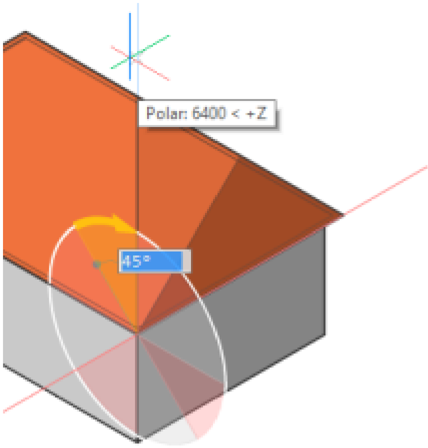
2 Make both end faces vertical:

- Move over the end face then hold down the left mouse button about half a second; the Manipulator widget displays (set the value of the MANIPULATOR system variable = 2).

- 

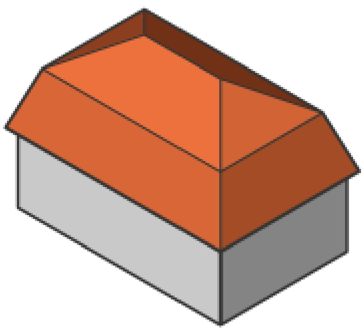
- Place the cursor over the vertical rotation arc. When the arc highlights, move the cursor.
- The selected face rotates dynamically.

  Click when the face is in a vertical position.



3 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

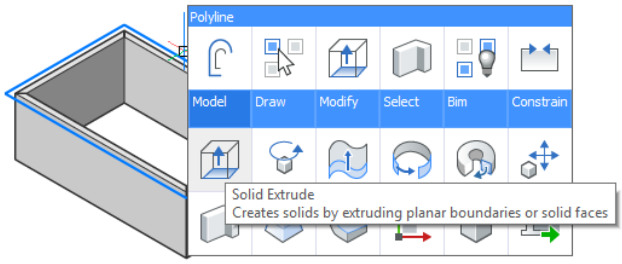
22.5.5 Creating a mansard roof



A mansard roof is composed of two solids, which we will create as two separate extrusions.

1 Create a roof footprint.

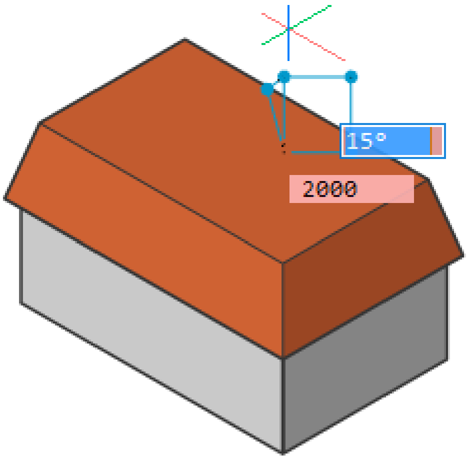
2 Move the cursor over the roof outline and choose to **Extrude** in the **Model** tab of the Quad.



3 Move the cursor vertically, then hit the **Tab** key.

4 Type a value in the **Taper Angle** field then hit the **Tab** key.

The **Taper Angle** field is the complement of the inclination angle of the roof.

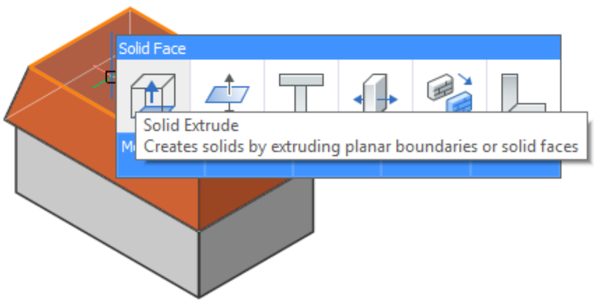


5 Type a value in the **Extrusion Height** field, then do one of the following:

- Hit the **Tab** key and type another value in the **Taper Angle** field.
- Press **Enter** to create the first solid.

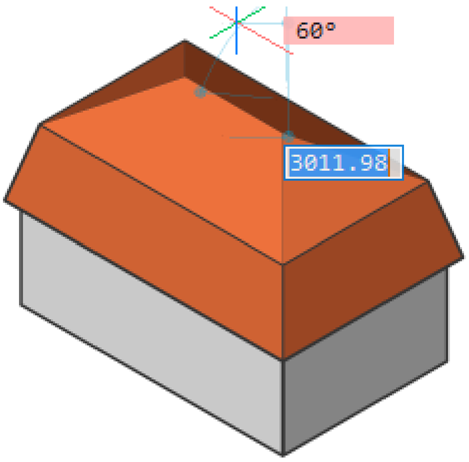
6 Optionally hit the **Tab** key to go back and forth between the entry fields and press **Enter** when you are satisfied with the result.

7 Move the cursor of the top face of the first roof solid and choose **Extrude** in the Quad.



8 Move the cursor vertically, then hit the **Tab** key.

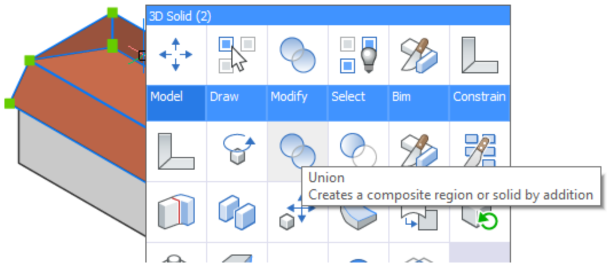
9 Type a value in the **Taper Angle** field then hit the **Tab** key.



10 Do one of the following:

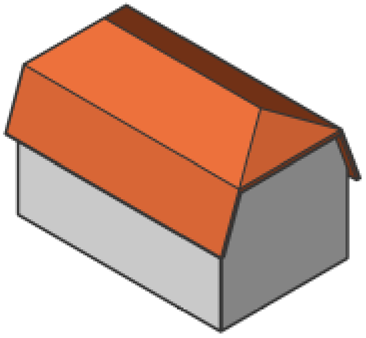
- Hit the **Tab** key and type another value in the **Taper Angle** field.
- Press **Enter** to create the first solid.

11 Select both solids and choose **Union** in the **Model** tab in the Quad.



12 Repeat step 7 of the Create a hip roof procedure.

22.5.6 Creating a jerkinhead roof



A jerkinhead roof is composed of two solids, which we will create as two separate extrusions.

1 Execute steps 1 to 6 of Creating a mansard roof procedure, to create the first solid.

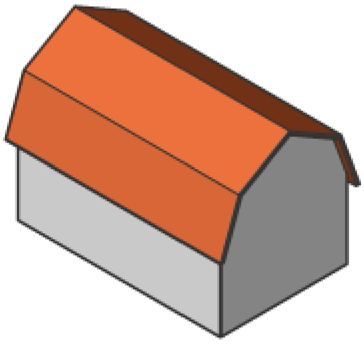
2 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

3 Execute steps 7 to 11 of the Creating a mansard roof procedure.

4 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

22.5.7 Creating a gambrel roof

A gambrel roof is composed of two solids, which we will create as two separate extrusions.



1 Execute steps 1 to 6 of Creating a mansard roof procedure, to create the first solid.

2 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

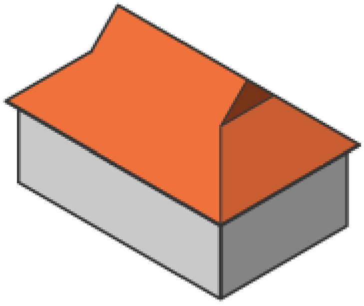
3 Execute steps 7 to 10 of Creating a mansard roof procedure to create the second solid.

4 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

5 Execute step 11 of Creating a mansard roof procedure.

6 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

22.5.8 Creating a dutch gable roof

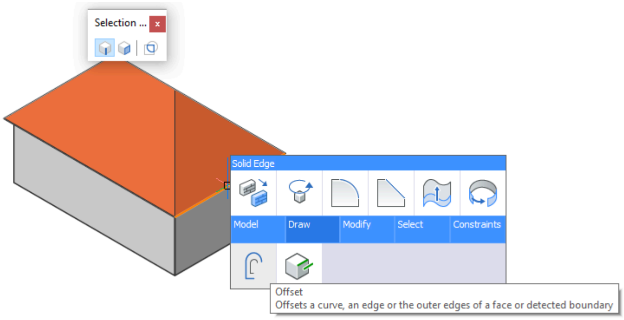


A Dutch gable roof is based on a hip roof.

1 Create a hip roof.

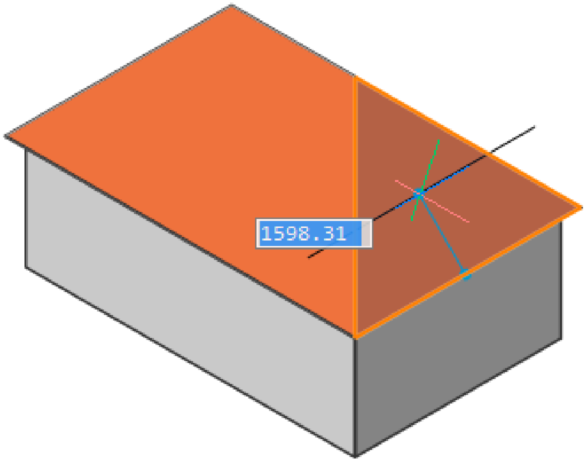
2 Set the **Select Edges** option of the SELECTIONMODES system variable ON.

3 Move the cursor over the bottom edge of the roof face, then choose **Offset** in the **Draw** tab in the Quad.



4 Move the cursor and type a value in the dynamic distance field or click a point.

5



6 Do one of the following:
- Choose **Imprint** in the **3D Solid Editing** toolbar.
- Run the IMPRINT command.

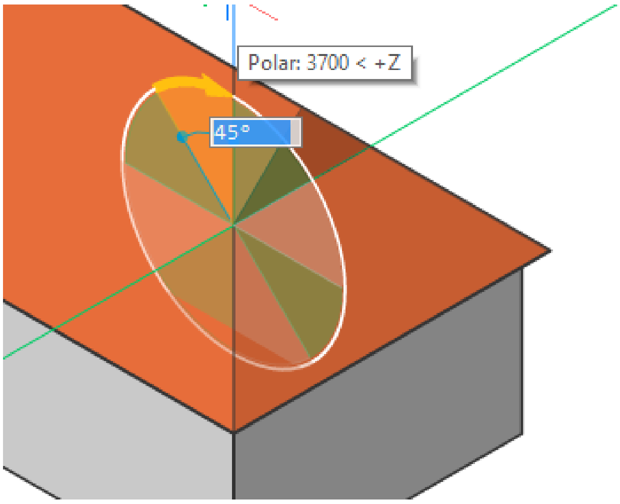
7 You are prompted: Select 3D solid, surface or region: Select the roof solid.

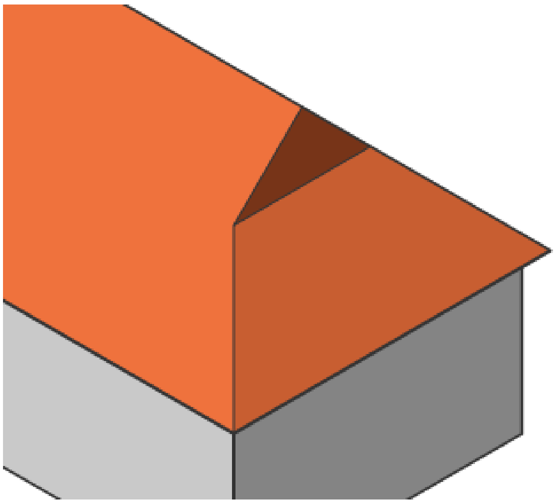
8 You are prompted: Select an entity to imprint: Select the offset roof edge.

9 You are prompted: Delete the source object[Yes/No] <No>: Choose Yes.

10 A triangular face is created.

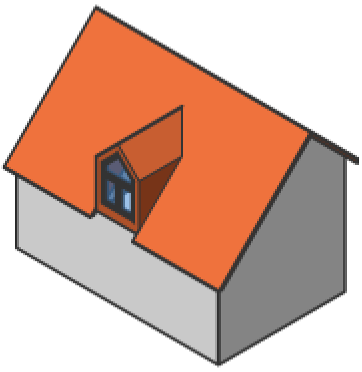
11 Execute step 2 of Creating a gable roof procedure to set the triangular face vertical.

12 



13 Execute step 7 of the Create a hip roof procedure to set the thickness of the roof solid.

22.5.9 Creating dormers



The procedure to create a dormer consists of the following steps:

- Create a box solid at the bottom edge of the roof plane and connect the box to the roof.
- Define the position and dimensions of the dormer.

- Create a dormer roof.

1 Right-click, then choose **Box** in the **Model** tab of the Quad.

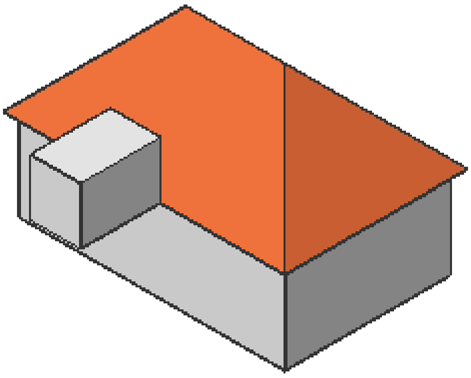
You are prompted: Set corner of box or [Center] <0.00,0.00,0.00>:

2 Click a point at the bottom edge of the roof plan.

3 You are prompted: Set opposite corner or [Cube/Length of side]:

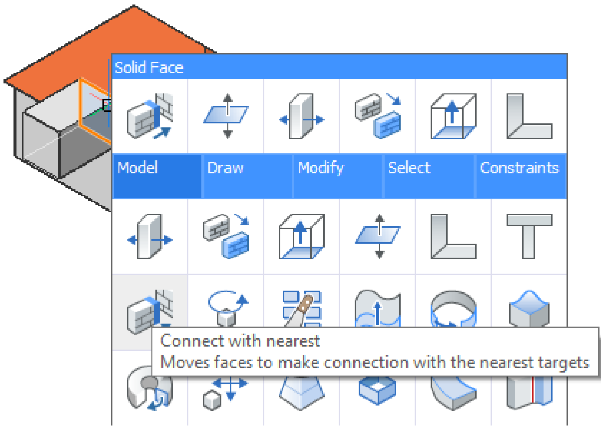
4 Click a point, then define the height of the cube: either click a point or type value.

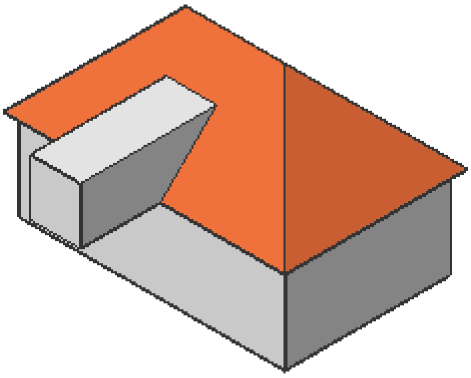
5



6 Select the back face of the cube: move the cursor over the cube at the location of the back face, then hit the **Tab** key until the face highlights (Selection Modes: Select Faces = ON).

7 Choose **Connect with nearest** in the **Model** tab in the Quad.

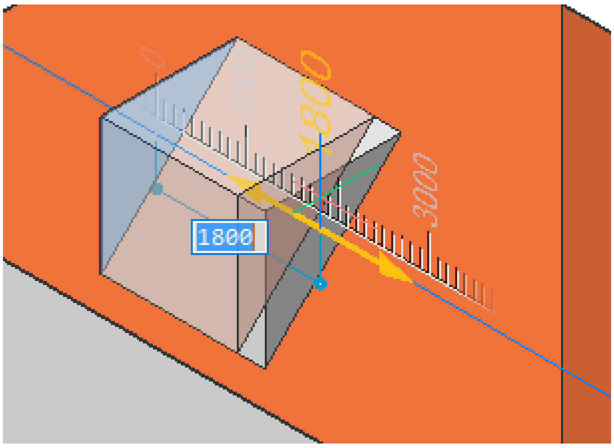




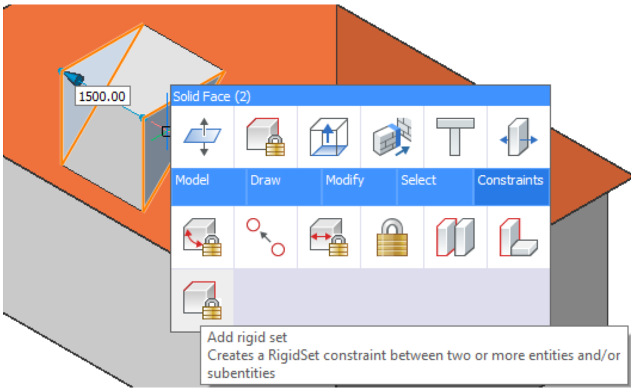
8 Select the front face of the box, choose **Push/Pull** in the **Model** tab in the Quad and move the face to the roof plane.

9 Use PUSHPULL command on the front, top and side face of the dormer solid to define the dimensions and the position of the dormer on the roof:

- Select a side face, then hit the **Tab** key to define the width.



- Select both side faces, then choose to **Add rigid set** in the **Constraints** tab in the Quad to fix the width.



- Drag one of the side faces to position the dormer. Do one of the following:
• Type a distance in the dynamic entry field.

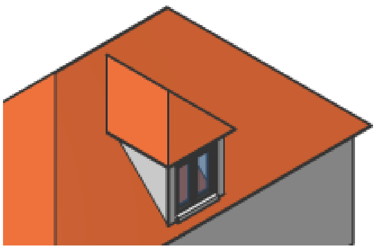
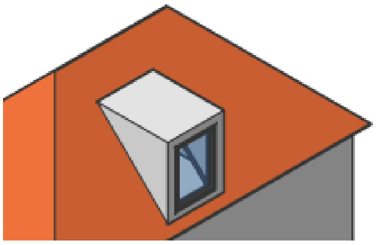
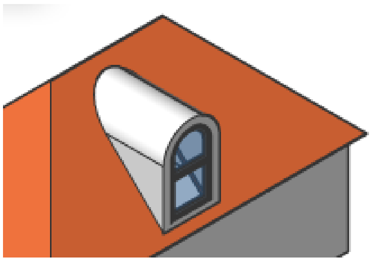
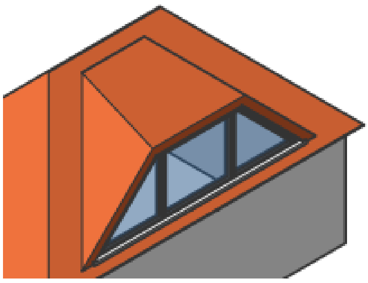
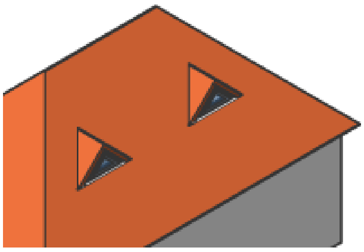
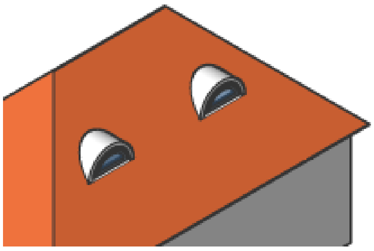
- Click to accept the value.
- Move the cursor over a face that is parallel to the face being dragged and hit the **Tab** key. The distance field now shows the distance to this face.

- Drag the top face to define the height of the dormer.
- Drag the front face to set the distance from the roof eave.

Move the cursor over the exterior face of the wall beneath the dormer, then hit the **Tab** key. The distance field now shows the distance to this face.

10 Modify the shape of the dormer:

- Add a hip roof.
- Add a hip roof, then rotate the front face vertical to create a gable roof.
- Add a vault.
- Rotate top and side faces.
- Starting from a gable roof move the front face of the dormer backward until the front face is triangular.

		
Hip	Flat	Vault
		
Special	Triangle	Eyebrow

22.6 Creating slabs

22.6.1 Commands

DMEXTRUDE, POLYSOLID

Note:

The DMEXTRUDE command allows to extrude a closed 2D entity such as polylines, circles, ellipses and extrude a detected boundary, enclosed by walls and/or 2D entities.

The POLYSOLID command allows to create pitched roofs.

22.6.2 Creating floor slabs between walls

Make sure that:

- The **Enable Boundary Detection** option of the SELECTIONMODES system variable is ON.
- The bottom faces of the surrounding walls are in the XY-plane of the current coordinate system.
- If necessary, move the origin of the coordinate system.
- Dynamic dimensions are ON.

1 Move the cursor inside the surrounding walls.

2 When the boundary highlights, choose **Extrude** in the **Model** command group in the Quad.

The slab is extruded dynamically.

3 Do one of the following:

   - Type the thickness of the slab in the dynamic dimension field.
   - Pick a point.

22.6.3 Creating a pitched roof slab

Make sure that:

- **Dynamic UCS** is ON.
- **Dynamic dimensions** (DYN) is ON; it is recommended to have the **Tracking dynamic dimensions** option of the DYNMODE system variable set.
- **Entity Snap Tracking** (STRACK) is ON.
- **Endpoint entity snap** is ON.

1 Launch the POLYSOLID command.

1 You are prompted: Start point or [Height/Width/Justification/Entity/Separate solids/Dynamic] <Entity>:

2 Move the cursor over the outside face of a wall which is perpendicular to the roof slab you want to create.

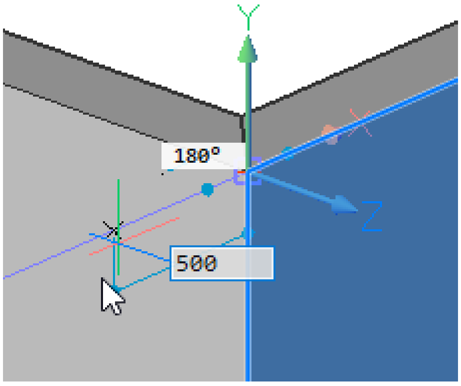
3 When the face highlights, hit the Shift key to lock the **dynamic UCS**.

4 Do one of the following:

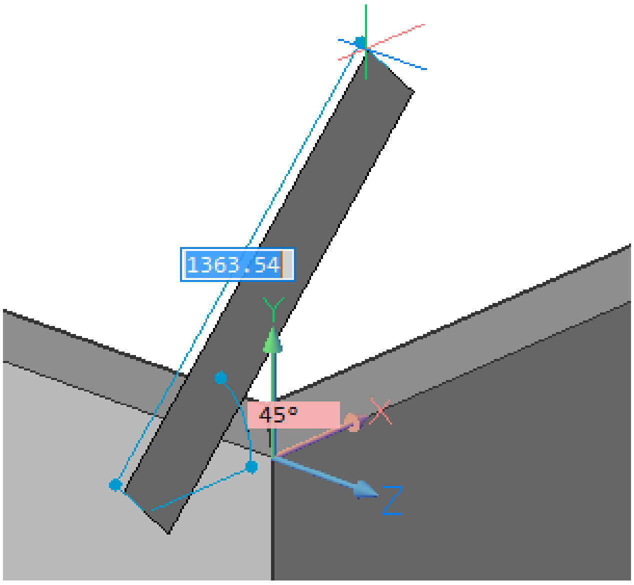
5 To create the roof at the top edge of the wall, snap to the exterior top corner of the wall.

To create a roof overhang (eave):

   - Move the cursor over the exterior top corner of the wall to require a snap tracking point.
   - When a small red cross indicates the tracking point is acquired, move the cursor to the outside and type the overhang distance in the dynamic entry field.



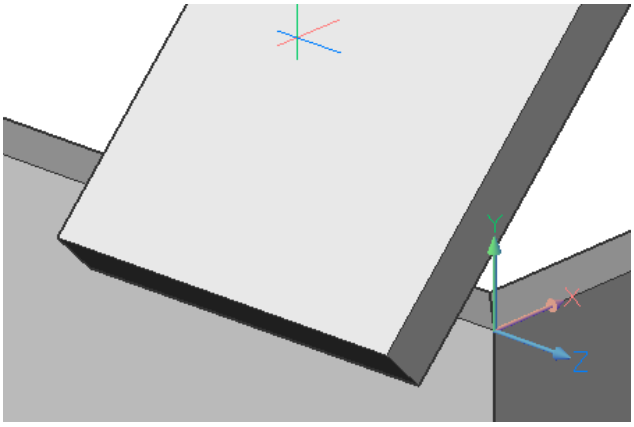
6 Hit the TAB key to activate the Angle field, then type the pitch angle and hit the TAB key again to jump to the Distance field.



7 Specify the height of the roof slab, then press Enter twice: the first Enter creates the footprint of the roof slab, the second Enter interrupts the creation of a second solid.

8 The roof slab is extruded dynamically.

9 Specify a point to define the length of the extrusion.



10 Repeat the previous steps to create more roof slabs.

11 Use CONNECTWITHNEAREST to connect the walls to the roof slabs.

22.7 Creating walls

22.7.1 Commands

POLYSOLID, LCONNECT, TCONNECT, BIMQUICKDRAW

22.7.2 About the Polysolid tool

The **Polysolid** tool allows you to create wall solids by selecting a start point, an endpoint and entering a height. Note that, in BricsCAD BIM, you always start by creating the geometry first and adding materials or compositions later. This way there is no need to worry about details when modeling a preliminary design.

The use of Dynamic Dimensions and Polar Tracking is recommended. Dynamic Dimensions can be used to specify the length of the wall and Polar Tracking controls the direction.

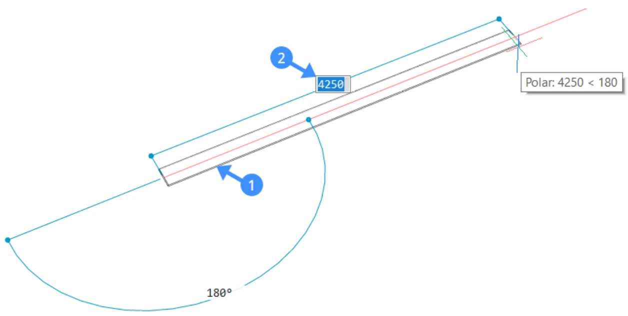
For more information about this command, visit the Command Reference article POLYSOLID.

22.7.3 Procedure: Create a simple wall

1 Launch the **Polysolid** tool.

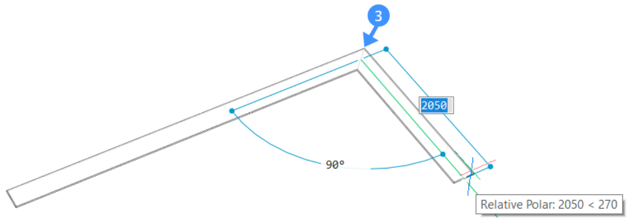
2 Select a start point, the footprint (1) of the wall is displayed automatically.

3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).

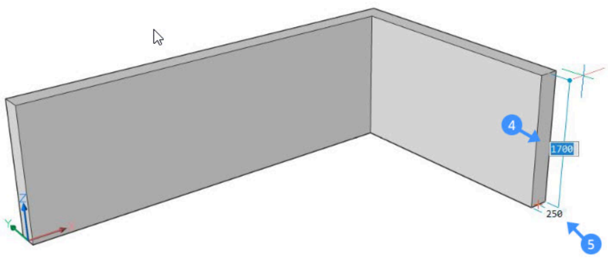


4 Select an endpoint or enter a value in the dynamic input field and press Enter. The footprint (3) of the

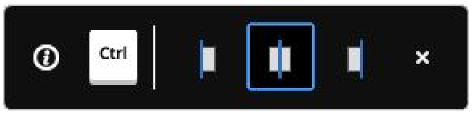
adjacent wall is displayed. Select a new endpoint or enter values in the dynamic input field to create adjoining wall segments.

5 

6 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field (4). Here, you can change the default height of the wall. To change the width of the wall, use the width field (5). You can switch between the two dynamic input fields by pressing Tab.

7 

8 To change the justification of the wall hit CTRL while the ’Polysolid justification Hotkey Assistant widget‘ appears at the bottom of the screen.

9 

Note: The width and height of the previous wall will be the new default values of the next Polysolid.

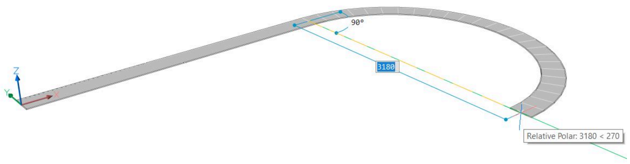
22.7.4 Procedure: creating a curved wall

1 Launch the **Polysolid** tool.

2 Select a start point and move the cursor to set the length or click to set the endpoint.

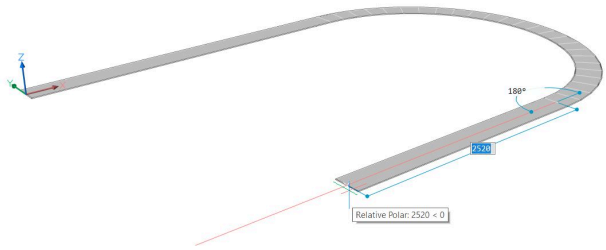
3 To create a curved wall, type A and press Enter to choose **Draw arcs** or select **Draw arcs** in the prompt menu.

4 Now move the cursor in the desired direction to curve the wall. Enter a value in the dynamic input field to set the degree and length of the arc. Press Enter.

5 

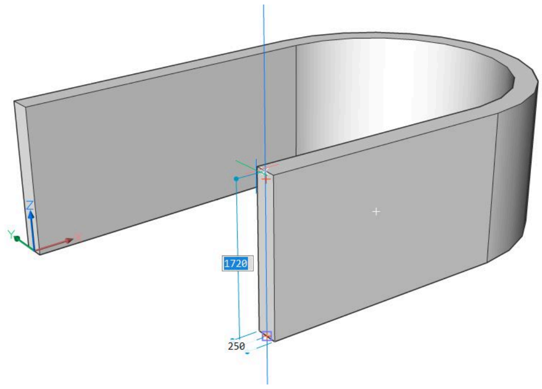
6 You can continue creating curved, adjoining wall segments by clicking or by entering values in the

dynamic input fields. To go back to drawing straight wall segments, type L and press Enter or select **Draw lines** in the options dialog.



7 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field. Type a value to change the default height of the wall. To change the width of the wall, use the width field. You can switch between the two dynamic input fields by pressing Tab.

8



22.7.5 Procedure: using an existing 2D plan to create walls

Using an existing 2D plan view enables you to create a wall element in 3D.

1 Click the **Polysolid** tool from the Quad and type E on the Command line or select **Entity** in the prompts menu. Using the **Entity** option in **Polysolid** enables you to create 3D geometry from a 2D layout.

2 Choose a 2D linear entity which will be the Polysolid base.

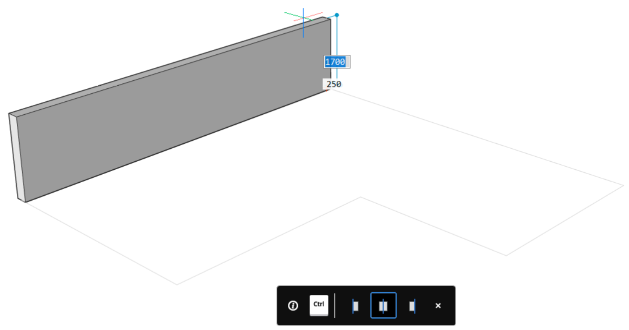
3 Lines, open and closed polylines, arcs, circles, ellipses, elliptical arcs, and splines are accepted as a polysolid base.

4 Move the solid up or down and left-click to set the height of the wall, enter a value in the dynamic input field or right-click to accept the default value.

5 The current width of the wall appears in the Width Field. Press Tab switch between the Height and the Width field.

6 The Hotkey Assistant appears and displays the possible justification options. Press CTRL to cycle through the wall justification options.

7

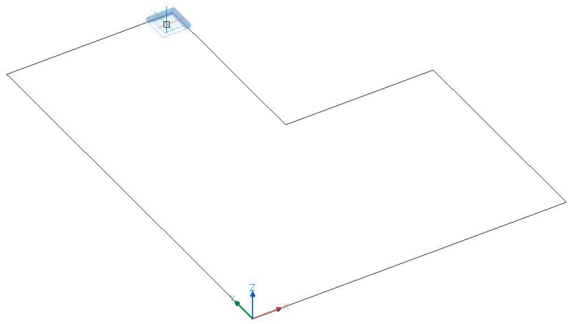


It is also possible to use an existing 2D plan to create walls using the **BimQuickDraw** tool:

1 Open the **Quickdraw** tool from the Quad or type BIMQUICKDRAW in the Command line.

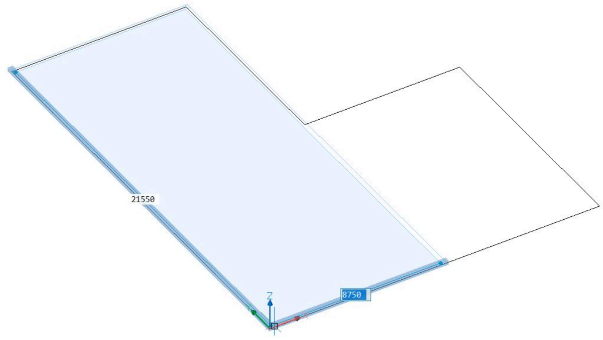
2 The cursor automatically snaps to the lines of your 2D drawing. Place your cursor where you want to place your first corner and left-click.

3



4 Hover your mouse to the opposite corner and left click again or type the values of the lengths of the sides in the respective fields and press Enter.

5

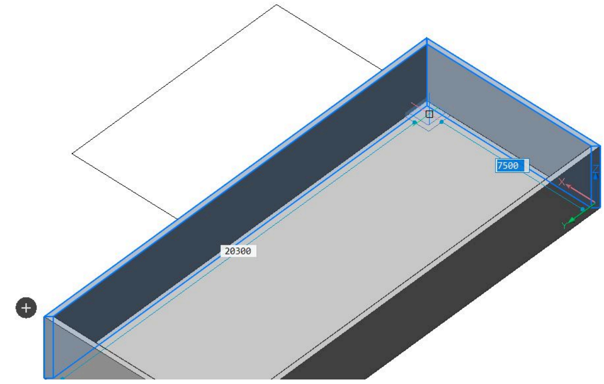


6 The **Quickdraw** tool always uses the default height and width settings.

You can change these values in the **Quick Draw** command context panel.

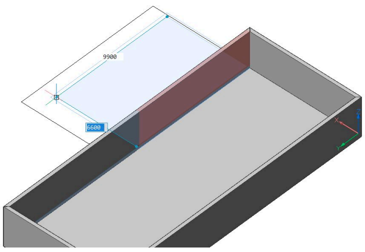
7 To finish the L-shaped form, you can draw a new rectangular on the second part of the figure. The common wall of the two rectangles will automatically be removed if you draw a second rectangular starting from the interior side of the first rectangular as indicated below:

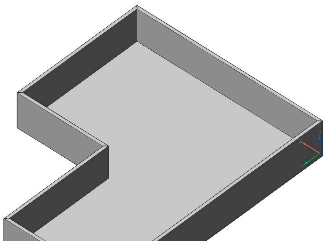
8



9 While drawing the second rectangular, you can see the wall that will be deleted is displayed in red.

10

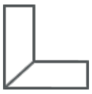




For more info about **Quickdraw**, please read the article **Using BIM Quickdraw**.

22.7.6 About the L Connect tool

The **L Connect** tool allows you to connect one wall to another element.

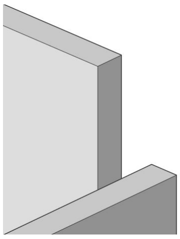


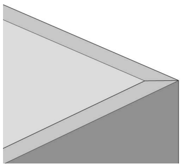
For more information about this command, visit the Command Reference article LCONNECT.

22.7.7 Procedure: connecting two walls

1 Select two wall elements and apply the **L Connect** tool.

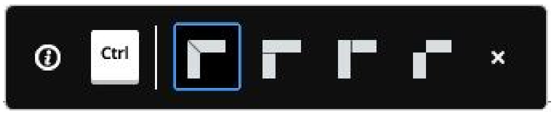
When two walls intersect, a mitered connection will be created by default.





2 Change the layout of the wall connections using the ‘LConnect HotKeyAssistant widget’.

3 The selected connection type is indicated by a blue frame.

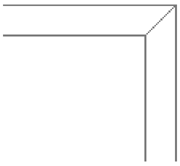


The connection can be (from left to right) a bisector L-connection , L parallel type 1 , L parallel type 2 or a disconnection.

Switch between the options by pressing CTRL.

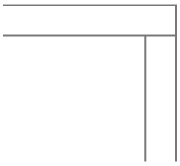
- **Bisector L-connection (miter)**

  When the **bisector L-connection** is selected, the walls are connected at an angle of 45°.

- **Parallel L-connection 1 Parallel L-connection 2**

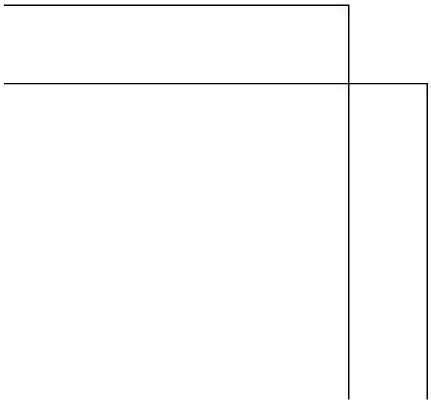
  When the L parallel type 1 or type 2 connections are selected, the walls are connected at an angle of 90°.

- **Disconnection**

When the disconnected option is selected, the existing connection between the walls is removed.



Press Enter to accept the current connection type or press CTRL to cycle through the possible connection options.

22.7.8 Connecting walls to other elements

The **Connect With Nearest** tool allows you to connect the minor face of a wall to the major face of another element, regardless of the orientation of the faces. This tool can be used to connect the top face of a wall to a roof, the bottom face to a slab or the side face of a slab to a wall.

To connect the face to the nearest object, select the face of the object that you would like to connect with the nearest object and choose the **Connect With Nearest** tool in the Quad.

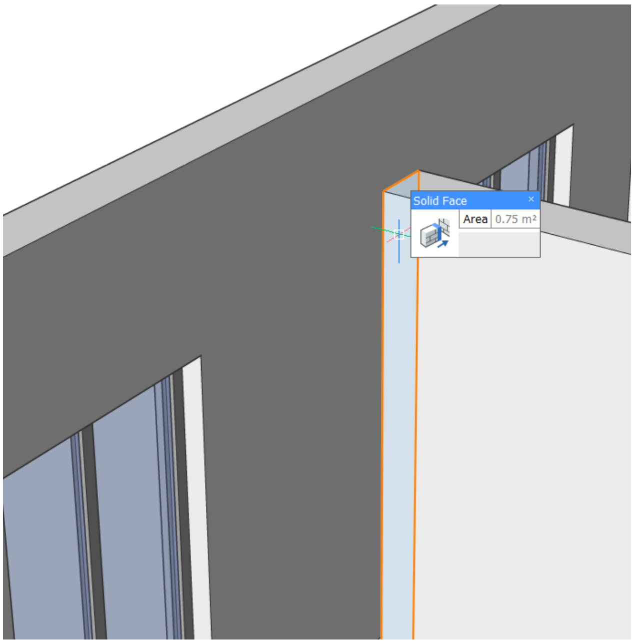
Note: Before using the **Connect With Nearest** tool, make sure the **Select Faces** option is enabled in the selection mode settings.

For more information about this command, visit the Command Reference article TCONNECT.

22.7.9 Procedure: creating T- connections between two walls

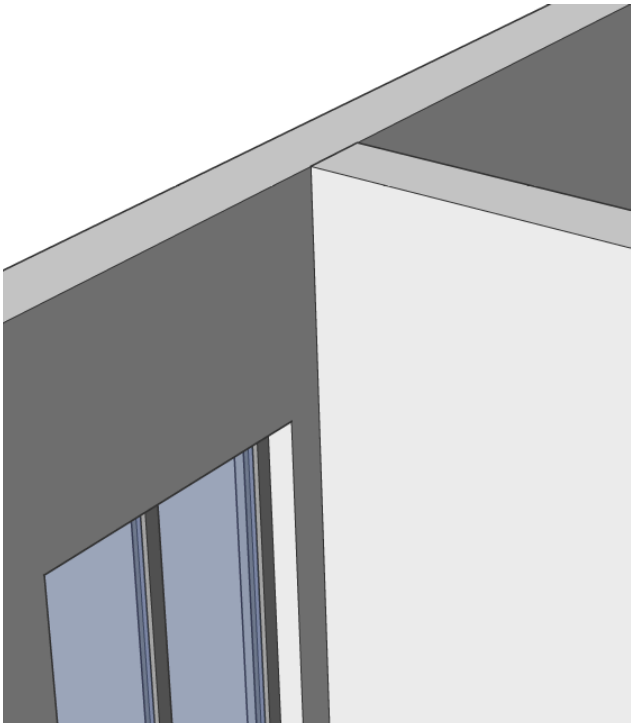
1 In the drawing area, highlight the face you want to connect to another wall.

1



2 Use the **Connect With Nearest** tool from the Quad.

The **Connect With Nearest** tool automatically makes a T-connection between the two walls by extruding the face of the first wall to the face of the other wall.



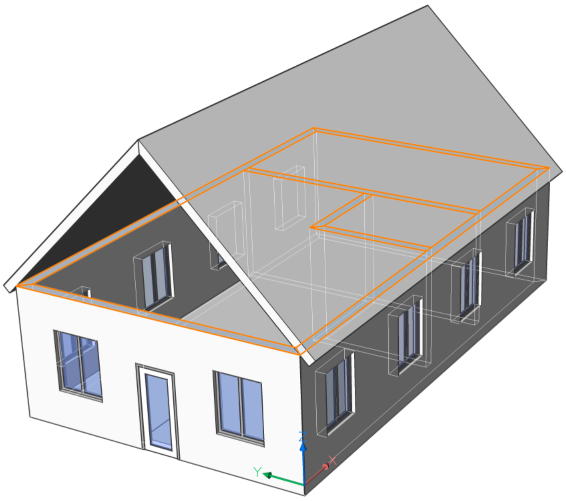
22.7.10 About the Select Aligned Faces tool

The **Connect With Nearest** tool can also be used to connect multiple faces of a wall to the nearest face of another element. To do this use the **Select Aligned Faces** tool. This tool enables you to highlight all faces that are coplanar with the current selection.

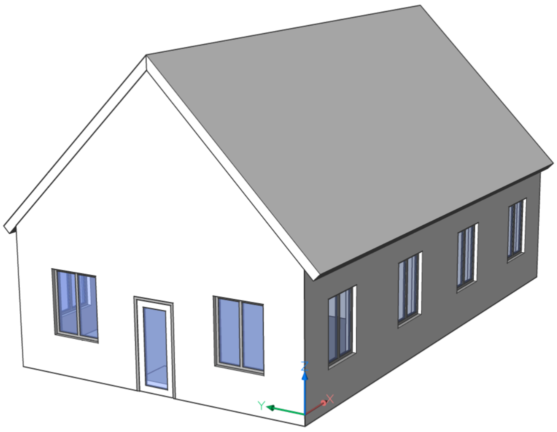
For more information about this command, visit the Command Reference article SELECTALIGNEDFACES.

22.7.11 Procedure: connecting multiple faces of the wall to the roof slab

1 To select all faces aligned to the current selection, highlight the top face of one of the walls and click the **Select aligned faces** tool in the **General** tab of the Quad.



2 Launch the **Connect With Nearest** tool. The top faces of the wall are attached to the roof.



22.8 Curtain wall

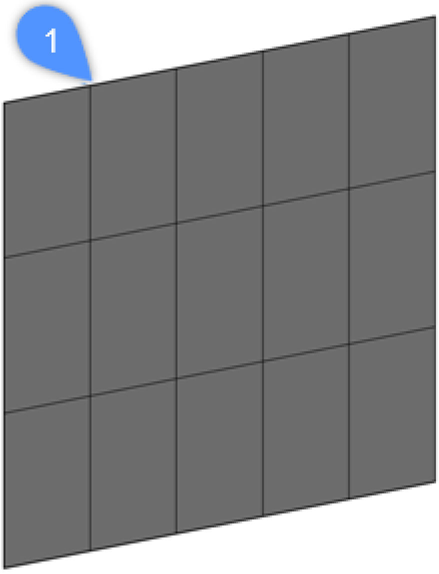
22.8.1 Commands

BIMCURTAINWALL

22.8.2 About

In BricsCAD, curtain walls are created from the face of a 3D solid or surface as a block by using the **Curtain Wall** tool.

- Curtain walls can be created from any planar and curved surface.
- Window Frames are placed on a BIM Grid when creating a Curtain Wall.
- Changing the U-line and V-line panel options allows you to specify the desired size and number of panels on the planar or curved surface.



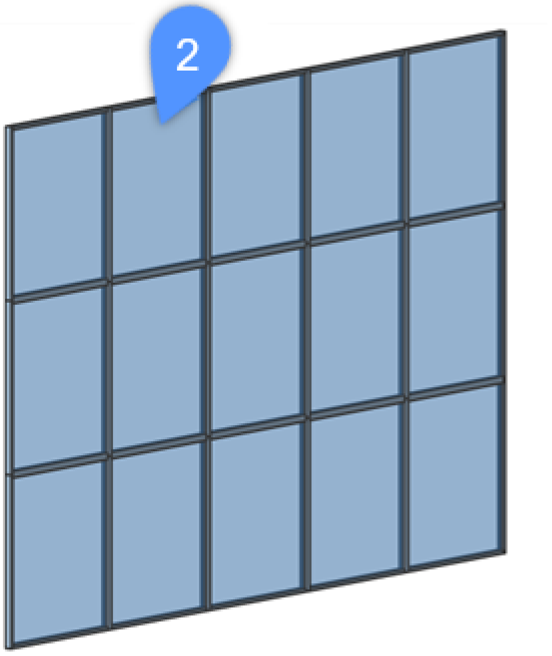


Figure 1, Curtainwall grid (1).

Figure 2, Curtainwall with glazing (2).

The curtain wall system consists of panels of glass. This is a panel infill for a curtain wall.

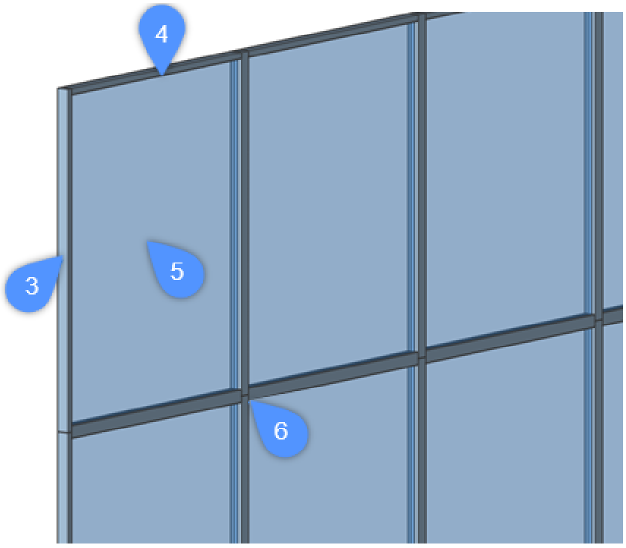
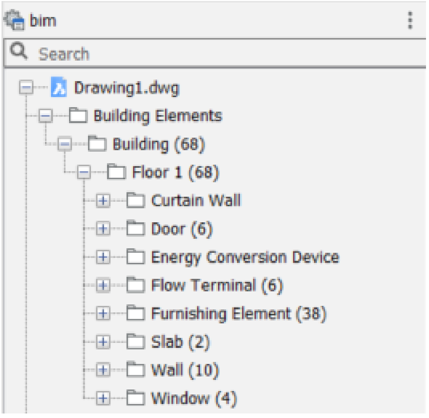


Figure 3, A curtain wall. The frame that coincides with the grid vertical (U) panels (3), the frame that coincides with the grid horizontal (V) panels (4), and glass in-fill (5) are shown. By default, the straight connection (6) is made between the beams.

Curtain walls are automatically classified as a **Curtain wall** in the **Structure** panel.

If you explode the Curtain wall, the sub-entities are classified as windows and as frames. The frames are linear solids. This allows you to calculate the length of a curtain wall.



22.8.3 Procedure: creating a curtain wall

1 Launch the BIMCURTAINWALL command in the Command line.

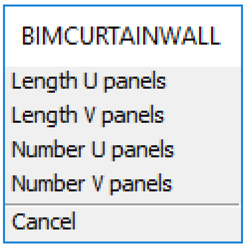
1 You are prompted: Select a face:

2 Select a face in the drawing area to create a curtain wall.

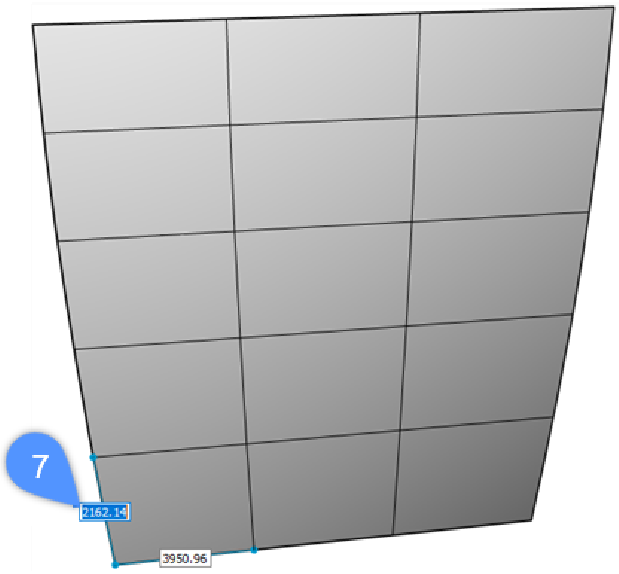
3 The curtain wall grid is automatically generated on this face.

You are prompted: Create grid or [Lenght U panels/Length V panels/Number U panels/Number V panels]:

4 Change the length of the U or V panels and number of U or V panels using the prompt menu and the Command line.

5 

Or, if Dynamic dimension (DYN) is on, the length of the panels can be changed using the DYN fields (7).

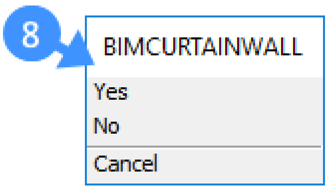


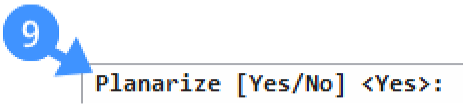
6 Hit the TAB key to switch between dynamic dimension boxes.

7 Or Press Enter to accept the current value.

Note: The maximum deviation from planarity is displayed in the command line.

Optional: If the value of the max deviation from planarity is different to "0", the prompt menu (8) will appear and the Command line prompts a message (9).





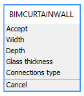
The prompt menu and the Command line allow you to planarize the grid cells.

Click "Yes" to planarize the grid cells. By default, the target deviation from planarity is set to "0.00001".

8 Press Enter to accept the default value.

9 You are prompted: Change parameters or accept [Accept/Width/Depth/Glass thickness/Connections type] <Accept>:

The prompt menu:



(**Optional**) To change the frame width, select the **Width** option in the prompt menu or type "W" in the Command line.

Default frame width = 60.000000

(**Optional**) To change the frame depth, select the **Depth** option in the prompt menu or type "D" in the Command line.

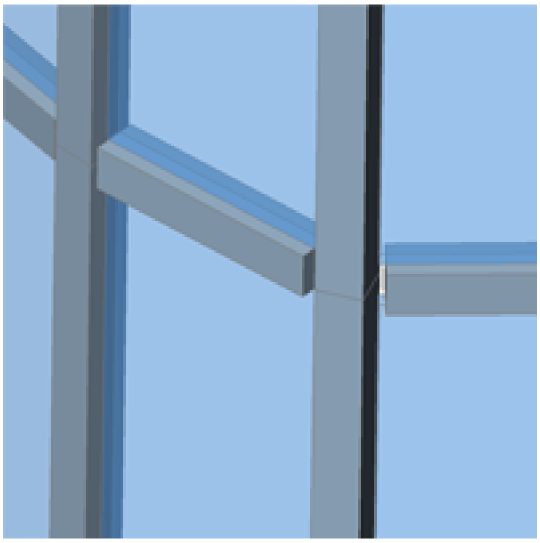
Default frame depth = 60.000000

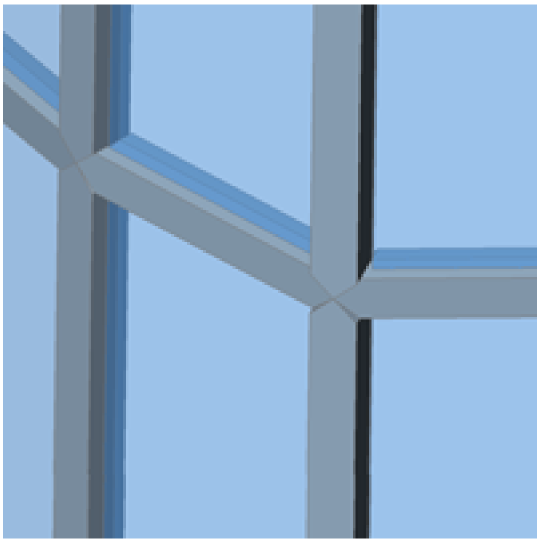
(**Optional**) To change the glass thickness, select the **Glass thickness** option in the prompt menu or type "G" in the Command line.

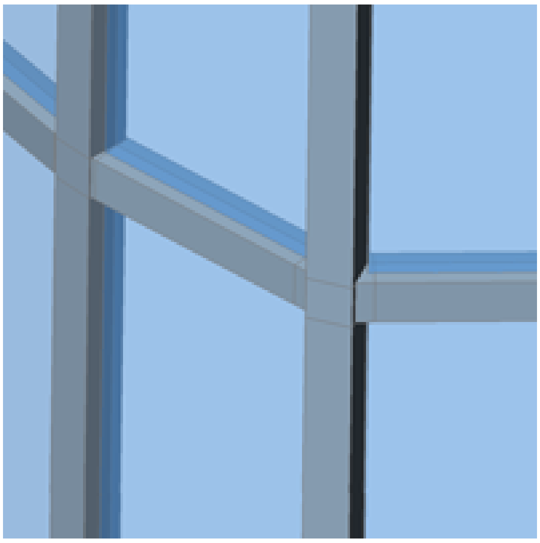
Default glass thickness = 20.000000

(**Optional**) To choose a connection type between, select the **Connections type** option in the prompt menu or type "C" in the Command line. For smooth connections, select the **Smooth connection** option or type "S" in the Command line. For nodes connection, select the **Nodes connection** option or type "N" in the Command line.

The default connection is straight.



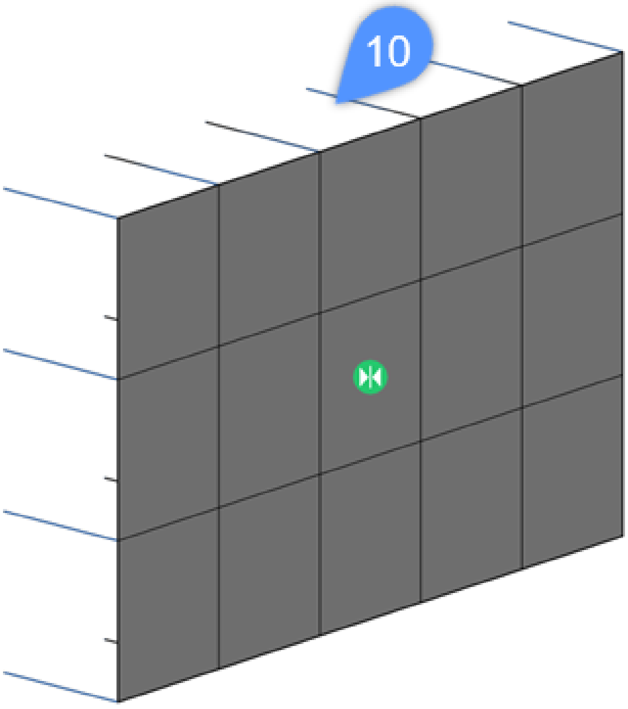




There are three types of connection. From left to right: Straight - Smooth - Nodes.

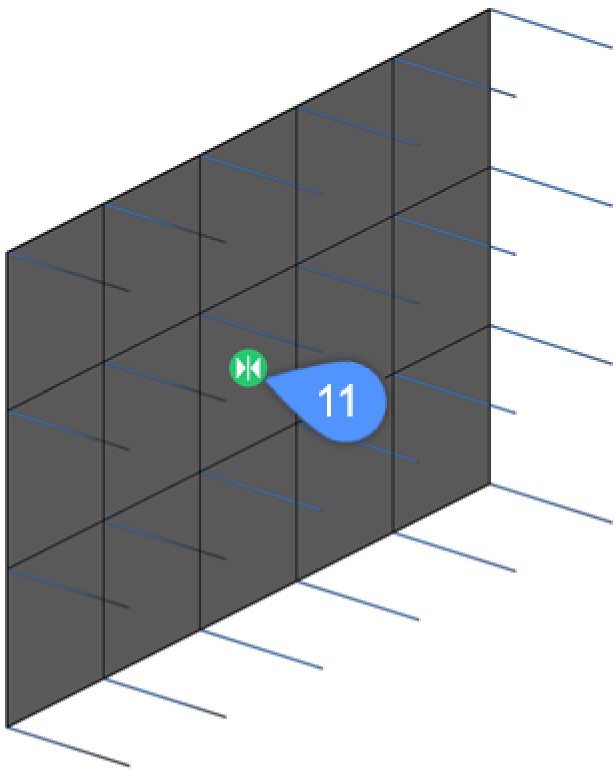
10 Blue lines (10) appear on the face to show the side a curtain wall will be created.

11



Note: Step 6 and step 7 are skipped when the face is selected from a solid. In this case the inside normal is chosen by default.

12 Use the widget (11) to flip the curtain wall.

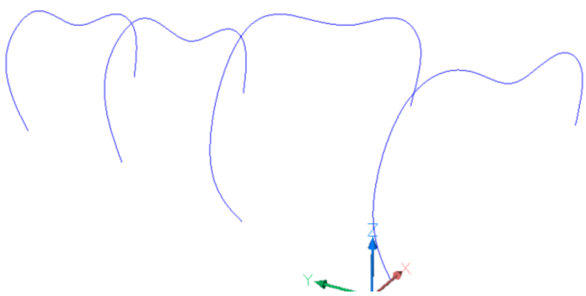
13 

14 Press Enter to accept.

15 ***Note:*** The surface or solid used to create the curtain wall will be deleted by default. If you don't want this, set the **DELOBJ** value to '0' in **Settings**.

22.8.4 Procedure: Creating a curtain wall from a free form surface

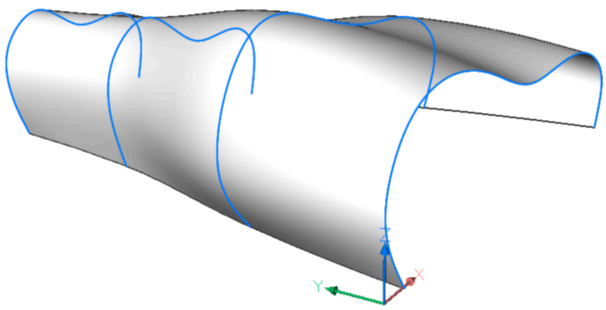
1 Start with 4 splines.

1 

2 Launch the LOFT command in the Command line.

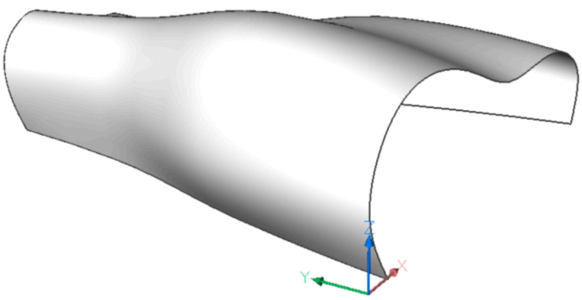
3 You are prompted: Select cross-sections in lofting order [MOde]:

Select the 4 splines in lofting order.



4 Press Enter twice to end the command.

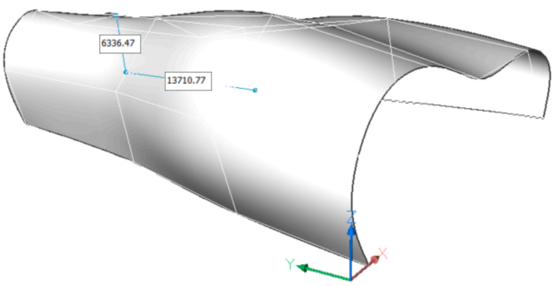
5



6 Launch the BIMCURTAINWALL command in the Command line.

7 You are prompted: Select a face

Select the lofted surface.



8 A curtain wall grid is automatically generated from this surface.

9 You are prompted: Create grid or [Lenght U panels/ /Length V panels/Number U panels/Number V panels].

Change the number of the U and V panels in the prompt menu or the Command line. In this example: U = 20 and V = 8.

10 You are prompted: Planarize [Yes/No] <Yes>

11 Click "Yes" to planarize the grid cells.

12 You are prompted: Enter target deviation <0.00001>:

13 By default, the target deviation from planarity is set to "0.00001".

Press Enter to accept it.

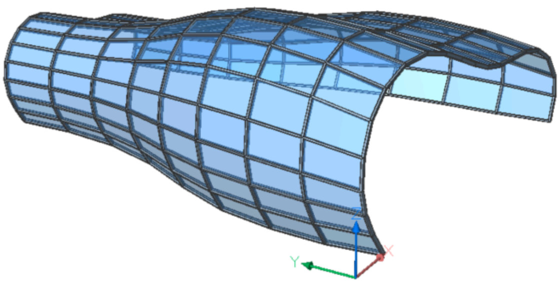
14 You are prompted: Change parameters or accept [Accept/Width/Depth/Glass thickness/Connections type] <Accept>:

15 Change the parameters to:

- Width: 150
- Depth: 300
- Glass thickness: 20
- Connection type: Smooth

16 Press Enter to accept. A curtain wall is created.

17



22.9 DmPushPull

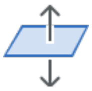
22.9.1 Commands

DMPUSHPULL

22.9.2 About

The **Push/pull** tool allows you to add and remove volume from solids by moving highlighted faces.

Note: **Push/pull** tool applies to planar, cylindrical, spherical, conical and toroidal faces of a solid.



For more information about this command, visit the Command Reference article DMPUSHPULL.

22.9.3 Procedure: using Push/Pull tool to add or remove volume to the solid

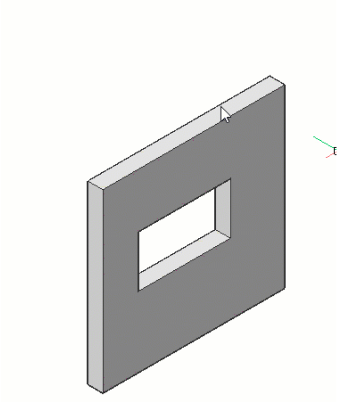
The following are general steps to add or remove volume from a solid(s) using the **Push/Pull** tool.

1 Highlight any face(s) of the solid(s) in the drawing.

2 Select the **Push/pull** tool in the **Model** tab from the Quad.

3 Move the cursor to pull the highlighted face.

4 The following movie shows the highlighted face behavior. The selected face moves dynamically, adding or removing volume.



5 Specify a point in the drawing or enter a value in the dynamic dimension field, then right click to accept changes.

22.10 Editing grids

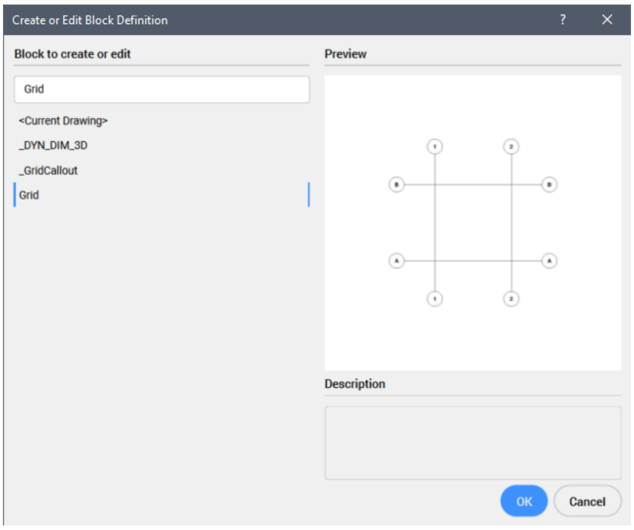
22.10.1 About

BIM grids are created as a block. The BEDIT (Block Edit) command allows to edit a BIM grids.

22.10.2 Editing a rectangular grid

1 Double click the grid.

1 The **Create or Edit Block Definition** dialog box displays.



2 Select the grid, then double click or click the OK button.

3 A **Block Edit** session is started.

4 Edit the grid axes.

5 You can move, rotate, copy, stretch, grip-edit or delete grid axes.

6 Renumber grid axes or edit axis labels if necessary (see the procedures below).

7 Click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

22.10.3 Renumbering the grid labels

Renumbering grid labels is needed:

- If you want a different numbering style.
- When grid axes are added (copied) or deleted.

Do the following:

1 Double click the grid to start a **Block Edit** session.

2 Select all vertical axes.

3 Start the NUMBER command and choose the appropriate number style.

4 Make sure to overwrite the existing numbers.

5 Repeat the previous steps for the horizontal axes.

6 Click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

22.10.4 Editing a grid label

1 Double click the grid to start a **Block Edit** session.

2 Double click an axis.

3 The properties of the axis entity (line or arc) display in the **Properties** panel.

4 Select the **BIM/Axis number** property.

5 Type a value or name.

6 Click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

22.11 Inserting and editing windows and doors

22.11.1 About

In BricsCAD, the **Insert BIM Component** tool allows you to insert windows, doors and other components, such as furniture, sanitary equipment, plants, etc., in any face of a 3D solid. It is typically used to insert windows and doors.

For more information, visit the Command reference articles BMINSERT and -BMINSERT.

The -BMINSERT command is launched automatically when a component is dragged from the **Library** panel to the 3D model.

Note: Make sure that **Dynamic UCS** (DUCS) and **Dynamic Dimensions** (DYN) are activated when inserting components.

You can insert components in a floor plan, such as windows and doors, or to add and trim walls from a 2D orthographic view as well as a 3D view.

22.11.2 Inserting a window (or door)

1 Open the **Library** panel.

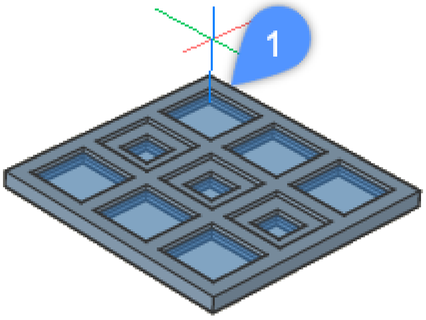
2 Make sure the **Bricsys Bim** library is selected in the **Library** menu.

3 Click the **Windows** folder.

4 Click the icon of the window you want to insert.

The **Properties** panel displays.

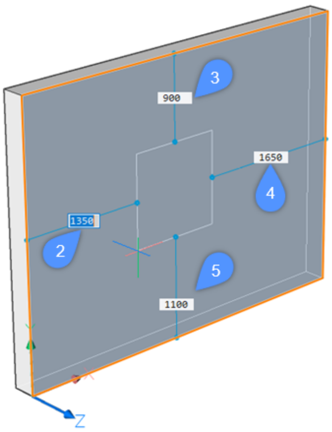
The window is attached to the cursor (1).



5 (option) Under **Inserted Components Properties**, edit the parameters of the window in the **Properties** panel.

6 Move the window over the face of a wall solid.

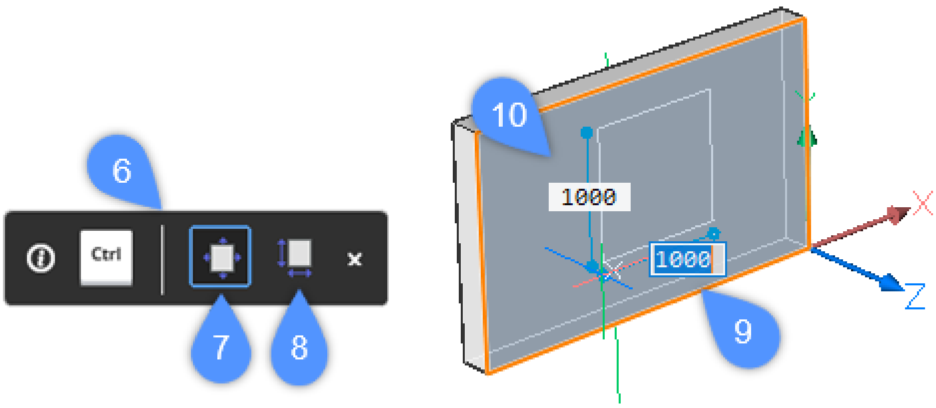
The solid face highlights and the distances from the insert to the nearest parallel edges of the parent solid (2, 3, 4, 5) are displayed:



7 (option) Press the **Ctrl** key to switch from the **Pick position** mode (7) to the **Change parameters** mode (8) in the **Hot Key Assistant** widget (6) and adjust the width and height of the window.

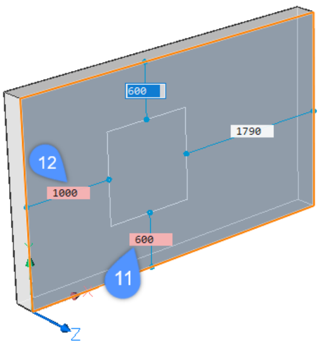
Press the **Tab** key to switch between the width (9) and height (10) fields.

Press the **Ctrl** key to confirm the dimensions. This will switch back the widget mode to **Pick position**.



8 Do one of the following:
- Specify a point to insert the window.
- Type a value in the desired dynamic dimension field and press the **Tab** key to jump to the next field. The dimension field turns red to indicate it is locked (11, 12).

  Press **Enter** to insert the window.



9 (option) Select the window to adjust its position in the wall. See **Moving an insert** section.

22.11.3 Inserting a copy of an insert

1 Hover over the insert and choose **BIM insert** in the **Modify** tab of the Quad.

The insert is attached to the cursor.

2 Proceed from step 6 in the previous procedure.

22.11.4 Editing the parameters of an insert

Parameters defined in the window or door source drawing allow you to edit each detail of a window/door separately.

1 Select the insert in the drawing. The current values of the various parameters are displayed in the **Parameters** section of the **Properties** panel.

Parameters	
W	130 cm
H	230 cm
FrameDepth	80 mm
FrameThickness	60 mm
LeafDepth	60 mm
LeafThickness	80 mm
PaneDepth	20 mm
PlacementDepth	90 mm
Rebate	0 mm
PaneOffset	20 mm
LeafOffset	20 mm
SubtractorDepth	1000 mm

2 Select a parameter and type a value in the settings field.

22.11.5 Moving an insert

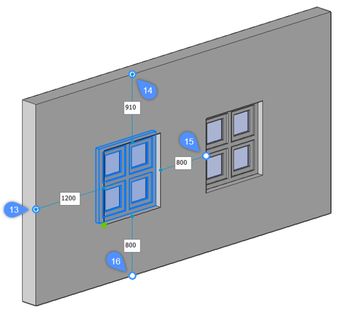
In BricsCAD, there are multiple methods to move an insert onto the face of a solid:

- Using distance fields and reference points of the selection dimensions.
- Using the DMPUSHPULL command.
- Using the Manipulator.

Using distance fields and reference points of the selection dimensions

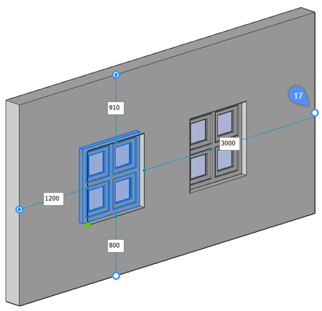
1 Select the insert in the model.

Distances from the insert to the nearest parallel edges of the parent solid are displayed, as well as their respective reference points (13, 14, 15, 16):



2 (optional) To show the distance from the insert to other points than those automatically detected, move the position of the reference points as needed (17).

3

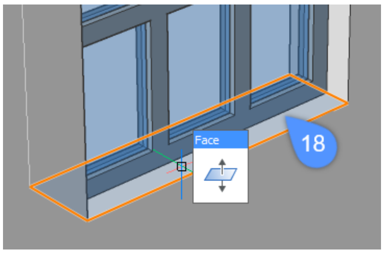


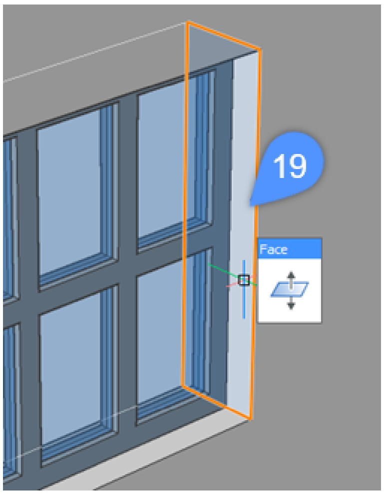
4 Click a distance field, type in a value and press **Enter**.

Using the DMPUSHPULL command

Note: Enable **Select faces** for SELECTIONMODE system variable.

1 Do one of the following:
  - To move the insert vertically: hover the cursor over the top or the bottom inner face of the opening (18)
  - To move the insert horizontally: hover the cursor over the left or the right inner face of the opening (19)





2 When the face highlights, select **PushPull** from the **Modify** tab of the Quad.

3 The insert moves vertically if the top or bottom inner face is highlighted.

The insert moves horizontally if the left or right inner face is highlighted.

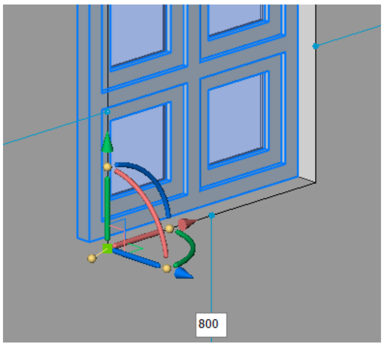
4 Do one of the following:
  - Specify a point.
  - Type a distance in the dynamic dimension field.
  - Use **Adaptive Grid Snap** of the Ruler.

Using the Manipulator

Note: Set the value of the MANIPULATOR system variable to 2.

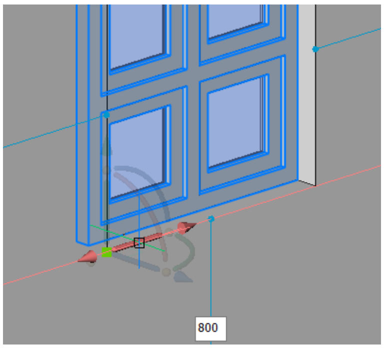
1 Select the insert by holding the left mouse button down a bit longer.

1 The Manipulator displays.



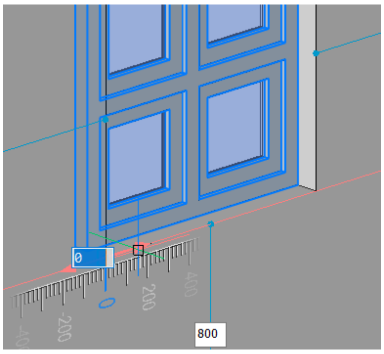
2 Hover over the vertical or horizontal axis trigger zone of the Manipulator.

The axis highlights and the rest of the Manipulator fades.



3 Click to start the movement operation.

The Ruler displays.



4 Do one of the following:
- Type a distance in the dynamic dimension field.
- Use **Adaptive Grid Snap** of the Ruler.

Note: Hold down the **Ctrl** key before starting the operation to make a copy of the insert.

22.11.6 Replacing an insert

Use the BMREPLACE command to replace an insert.

The command allows you to replace:

- An insert from a file.
- You can select a source file in a dialog box.
- A similar insert in the model.
- The command prompts you to replace either the selected insert only, or all similar inserts in the entire model. The **Parameter changes** option allows you to either keep the original values or to apply the values of the replacement insert.

22.12 Mass modeling with QuickBuilding

22.12.1 About

The **QuickBuilding** tool allows you to generate a BIM building just from one or more massing solids. BricsCAD creates a separate drawing and in this drawing walls, slabs and roofs, as well as spatial elements such as floors and spaces will be automatically created

The tool is useful in early design stages, volume studies and conceptual designs.

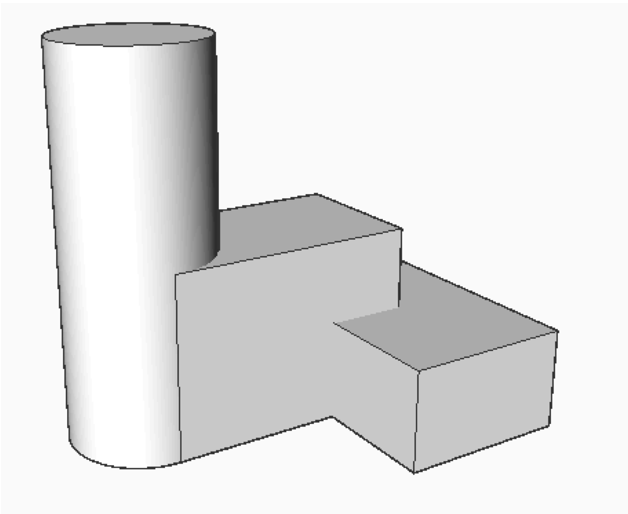
The tool will assign BIM properties and spatial information to each part of the new building.

For more information about this command, please visit the Command Reference article **BIMQUICKBUILDING command**.

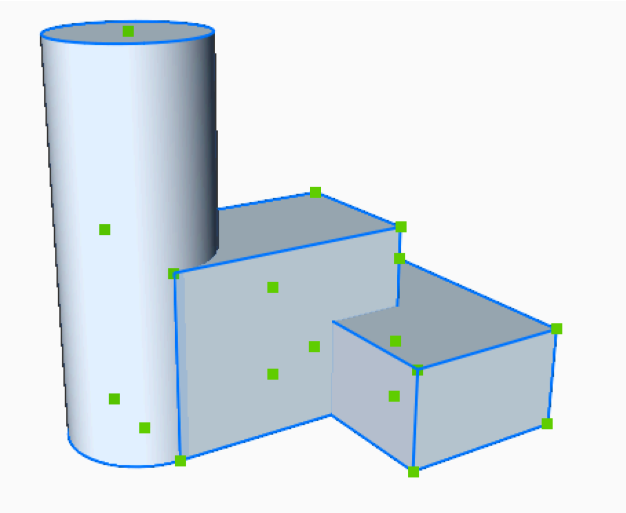
22.12.2 Procedure: top-down workflow

1 Draw one or more solids in your model space.

1 ***Note:*** The solids don't need to be unionized, the command will do it automatically.



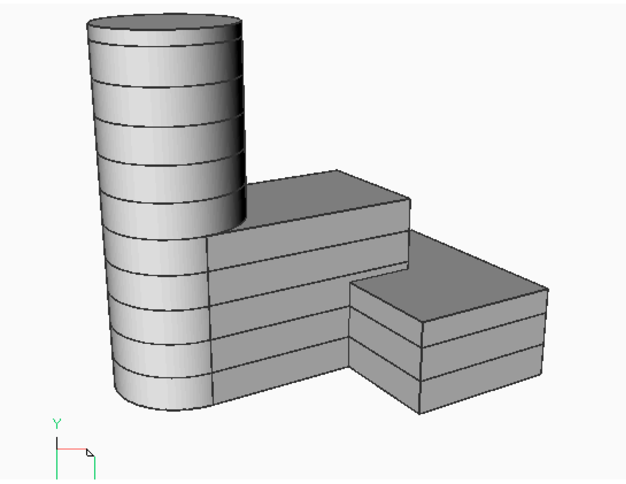
2 Type the command BIMQUICKBUILDING in the Command line or select it from the Quad and select the solids.

3 

4 The **Quick Building** command panel is opened. Choose between the **Buildings** and **Spaces** modes. Choose **Building** mode to continue with the top-down workflow.

5 You can add as many floors you want (**Add floor**) and choose the desired storey height for each. An automatic number of storeys are generated according to the elevation input (**Height**) value if **Fill** cell is checked.

6 Press **Apply** to generate the building.



Or, import the storey height values from a .csv file.

Enter BIMQUICKBUILDING in the Command line,

Choose **Import** and press **Enter**.

Next, search for the .csv file in the file folder and click **Open**.

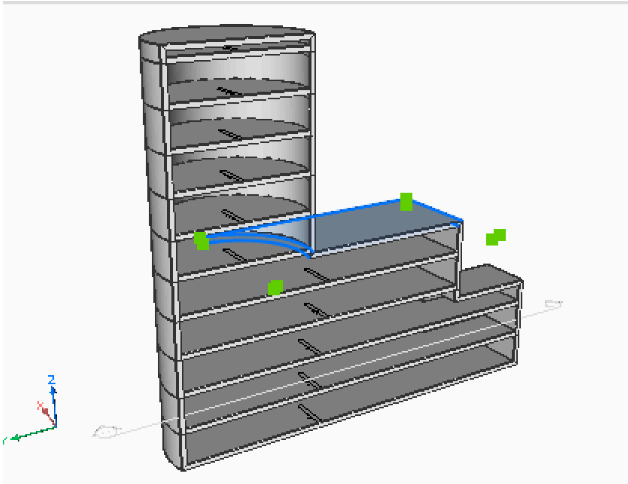
Example of a .csv file with parameters for your building:

```
Floor;Height- mm;Amount;Fill 0 - 10;3250.000000;11;Yes 12 -
 16;5000.000000;5;No
```

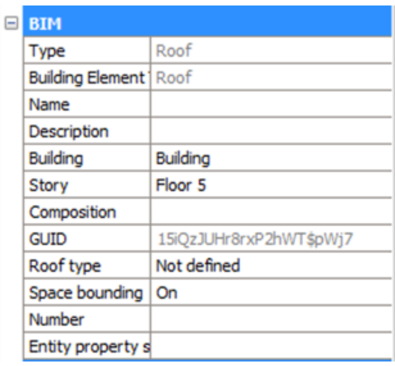
If the result of **QuickBuilding** does not satisfy the required storey heights this will be printed in the

Command line, see also the section about **Command Line Output** later in this article.

7 If you create a section, you can see the newly formed slabs and walls which each have individual BIM properties.

8 

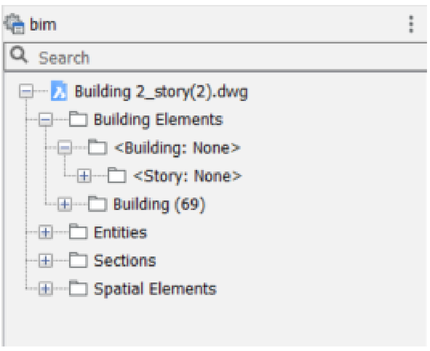
Select the roof and navigate to the **Properties** panel - **BIM** tab. You will see that this entity has **Roof type** property, as well as some other properties such as spatial locations, quantities or a **GUID**.



BIM	
Type	Roof
Building Element	Roof
Name	
Description	
Building	Building
Story	Floor 5
Composition	
GUID	15iQzJUHr8rxP2hWT$pWj7
Roof type	Not defined
Space bounding	On
Number	
Entity property s	

Note: All the elements are categorized, in the **Structure** panel you will see a list of building elements, sorted by **Story** and **Building** properties.

The different buildings elements will be placed in different layers according to their building type (**External wall/slab**, **Flat/pitched roof**, **Internal wall/slab**).



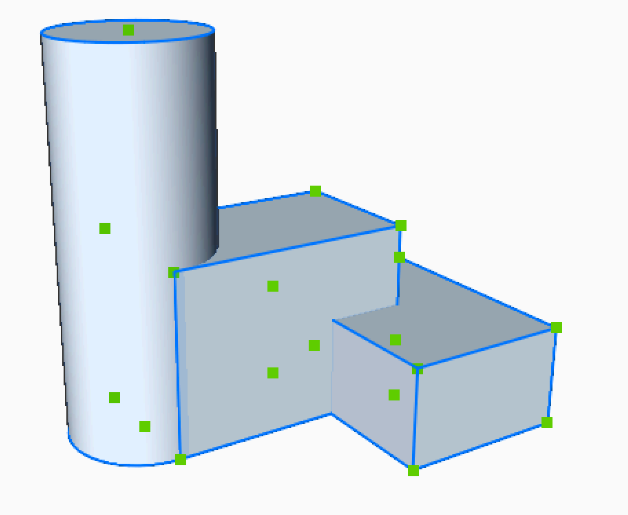
22.12.3 Bottom-up workflow

The bottom-up workflow creates a building out of solids that represent spaces. This approach allows you to first shape the spaces, then use **QuickBuilding** to create the shell and the new spaces.

1 Type BIMQUICKBUILDING in the Command line.

2 Select the solids.

3



4 The command panel **Quick Building** is opened. You have to choose between the **Buildings** and **Spaces** modes. Choose **Spaces** mode to continue with the bottom-up workflow.

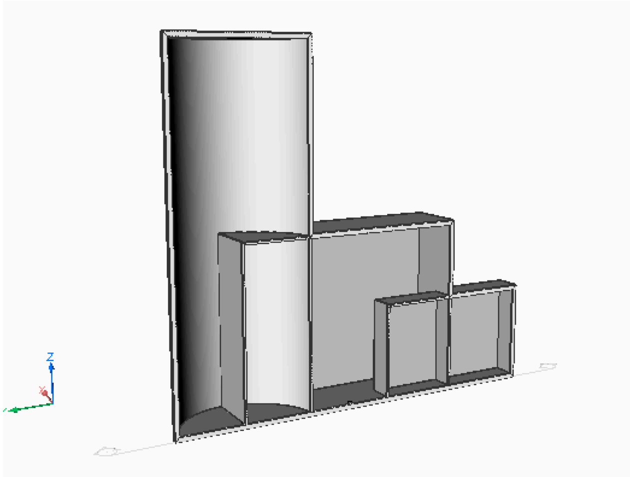
5 A building is created from the selected massing solids.

6 ***Note:*** BricsCAD takes these defined solids and converts them to BIM Spaces with walls and slabs.

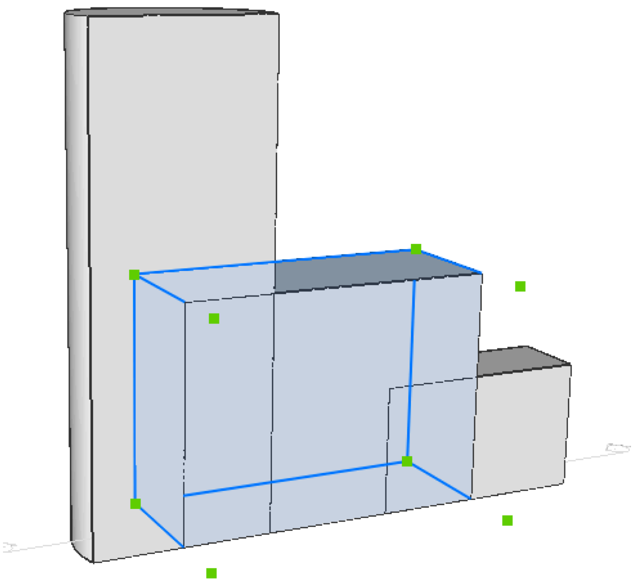
7 Press **Apply** to create the building.

8 (Optional) make a section to see the newly created spaces.

9



Note: The original solids are kept, **QuickBuilding** just creates a Building from it with the information that these input solids represent Spaces, so it knows that inner walls and slabs should be created based on those input spaces.



22.12.4 Command line output

After you have run the BIMQUICKBUILDING command, BricsCAD will create an overview with information on some basic building information. For instance, the total floor area of the building or other information for each floor.

```
Floor 6 :
    Floor area:  44.179 m²
    Side area:  53.564 m² [North: 53.564 m² , East: 0.000 m² , South: 0.000 m² , West: 0.000 m² ]
Floor 7 :
    Floor area:  44.179 m²
    Side area:  53.564 m² [North: 53.564 m² , East: 0.000 m² , South: 0.000 m² , West: 0.000 m² ]
Floor 8 :
    Floor area:  44.179 m²
    Side area:  53.564 m² [North: 53.564 m² , East: 0.000 m² , South: 0.000 m² , West: 0.000 m² ]
Floor 9 :
    Floor area:  44.179 m²
    Side area:  21.266 m² [North: 21.266 m² , East: 0.000 m² , South: 0.000 m² , West: 0.000 m² ]
Total Floor area: 1091.765 m²
Total Side area: 843.057 m²
```

Note: If you use a .csv file, and some requirements are not passed (e.g. not enough total area or floor area), you can see the problems in the output.

```
Total Floor area: 330.990 m² Requirement NOT passed, area should be >= 15000.000 m² m²
Total Side area: 1001.571 m²
Floor to Side ratio: 33.047 %
```

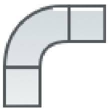
22.13 MEP modeling

22.13.1 Commands

BIMAPPLYPROFILE, BIMLINEARSOLID, BIMFLOWCONNECT, DRAG, BIMSTRETCH

22.13.2 About BIMFLOWCONNECT

It's also possible to connect different segments using the BIMFLOWCONNECT command.

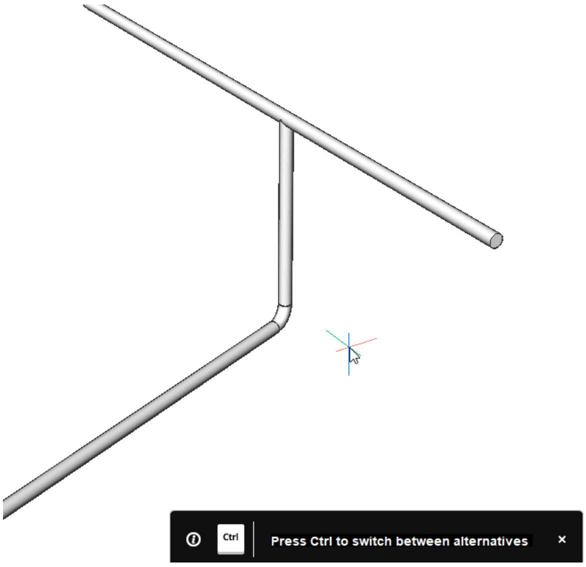


For more information about this command, visit the Command Reference article BIMFLOWCONNECT.

Connecting different segments to each other

Select the segments you want to connect and launch the BIMFLOWCONNECT command.

When the Hot Key Assistant is switched on (**HKA** on the status bar), the following widget appears on the screen. Press the CTRL-key to cycle through the different options.



End the command by pressing Enter, the segments are now connected.

22.13.3 Procedure: altering the position of segments

Before starting with the procedures: enable the selection of sides and ends:

Click the icon on the **Selection Modes** toolbar or set the DISPLAYSIDESANDENDS system variable ON.

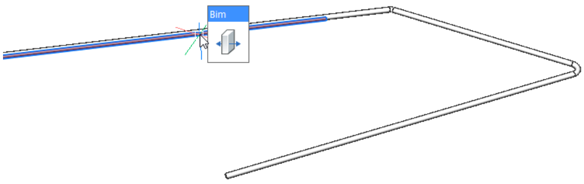
DRAG

The DRAG command makes it possible to alter a segment of a drawn profile without losing its connection to the other segments.

For more information about this command, visit the **DRAG command** article.

1 Hover over a segment, select a face perpendicular to the section of the profile.

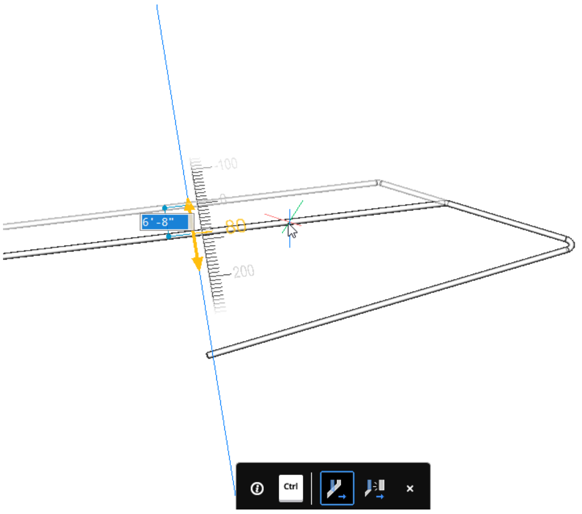
1



2 Launch the DRAG command.

3 Choose a new position for the segment or use the dynamic dimension field to enter a value for the displacement.

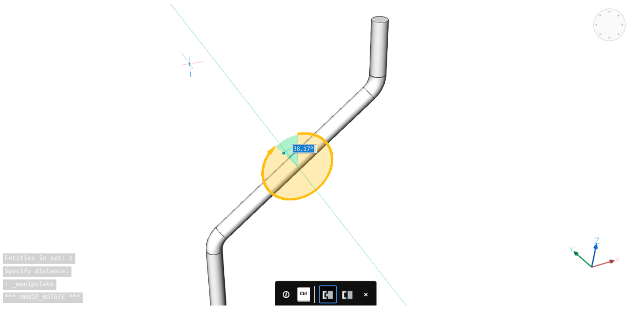
4 ***Note:*** The software automatically regenerates the connection between the segments.



Manipulator

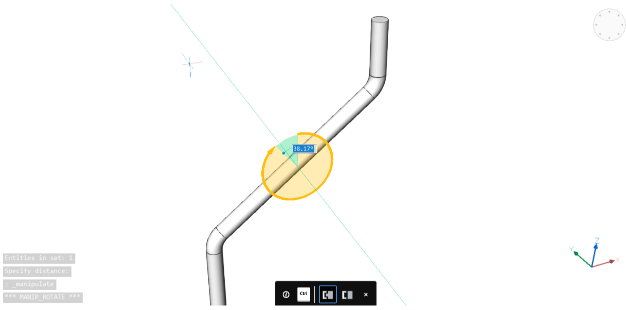
1 Hover over a segment, select a face perpendicular to the section of the profile.

2 Select **Manipulate**.



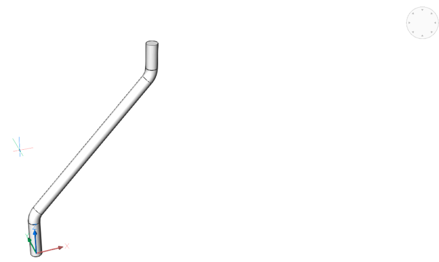
3 Choose a new position for the segment or use the dynamic dimension field to enter a value for the rotation.

4



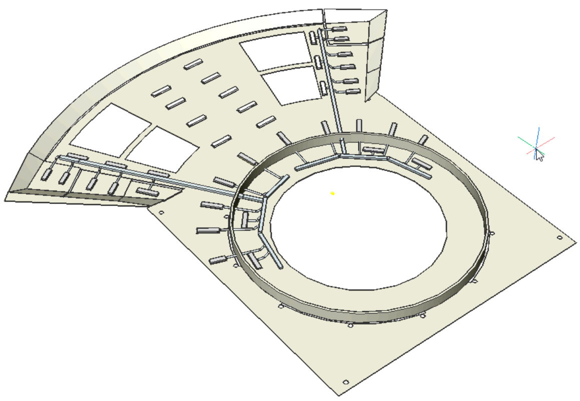
5 Click the new position or enter the rotation angle and hit Enter.

6



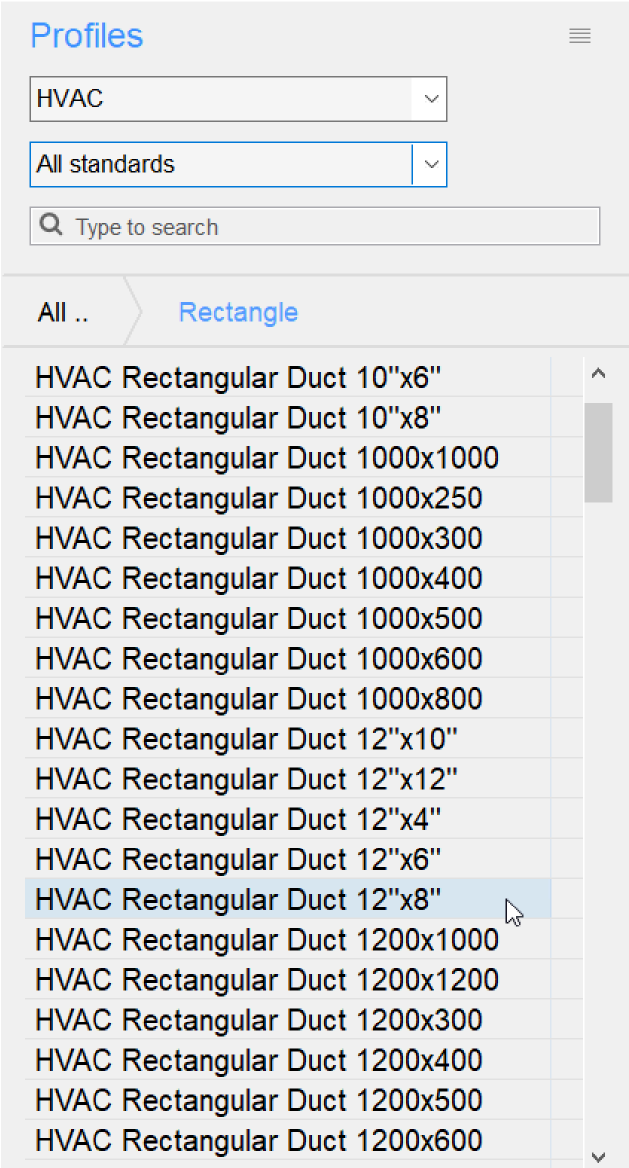
22.13.4 Procedure: using the MEP modeling tools to draw and connect profiles

1 Open the drawing that contains the MEP model. You can download this **MEP.zip**-file containing the drawings at the end of this article.



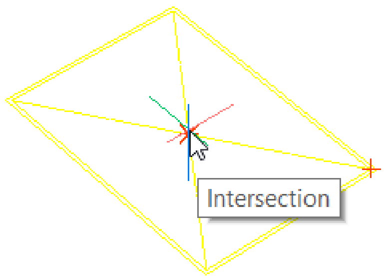
2 Open the **BIM Profiles** panel on the right-hand side of your screen (see Dockable Panels).

Filter on **HVAC** and select the Rectangular 12"x 8" profile, drag it to the model space.



3 Define the starting point of the profile.

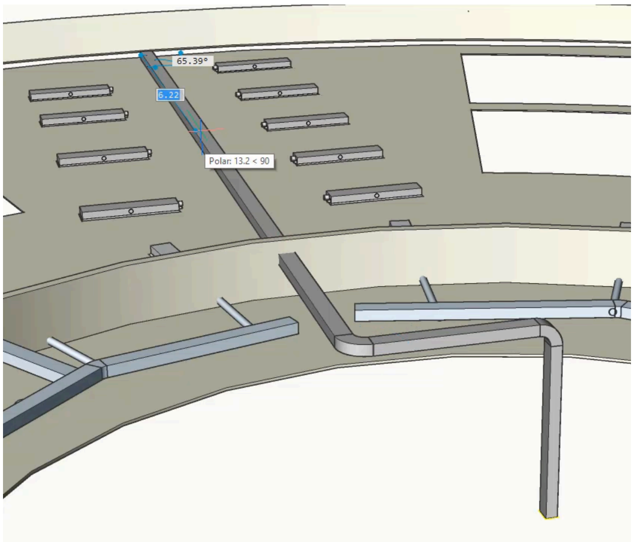
4 Snap to the geometric center of the yellow rectangle.



5 Turn **Ortho** on and define the next points of the profiles by using the dynamic dimension field.

Quarter turns the profile by entering Q in the Command line.

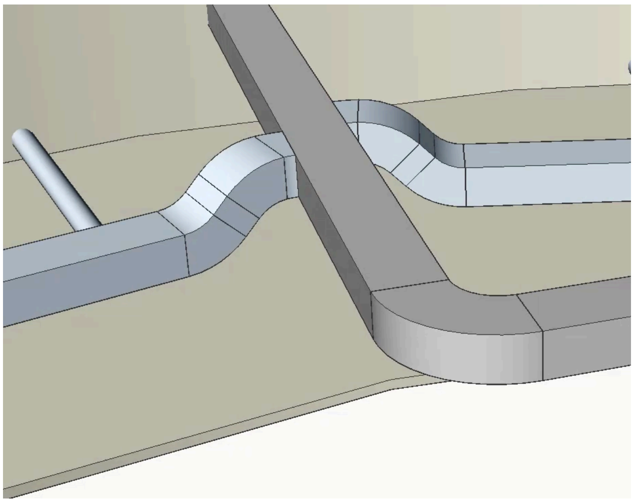
Go up 3.5, go 3.5 to the left and go 21 horizontally.



Note: If you had chosen to draw with a non-HVAC profile, you will not get the automatic classification as **Flow Segment**. You can manually classify the profiles through the BIMCLASSIFY command as **Building Service Elements** - **Flow Segments**.

6 Connect the other rectangular ducts on the side to the one we have just created.

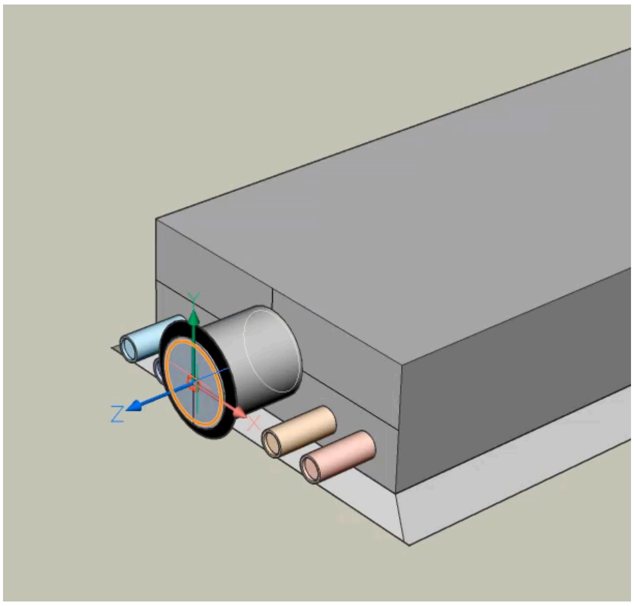
7 Select the ducts together with the main duct and launch the BIMFLOWCONNECT command. Your connection should look like the image below.



8 Create a duct from the flow terminal.

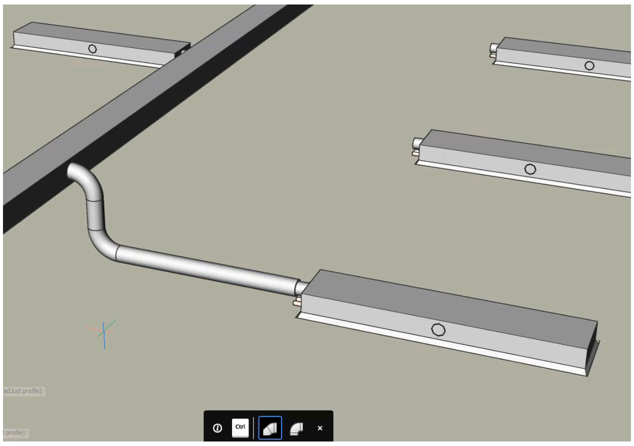
9 Launch the BIMLINEARSOLID command and select the flow connection point.

This connection point contains profile information, so the software automatically creates the predefined circular duct. Give the duct a length of 1.



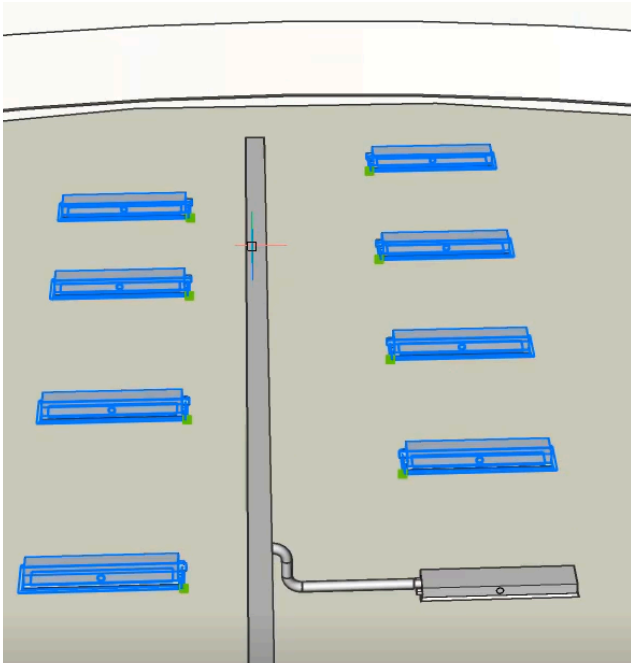
10 Connect the ducts from the flow terminal to the rectangular main duct.

11 Select the main duct and the duct from the flow terminal and launch the BIMFLOWCONNECT command.



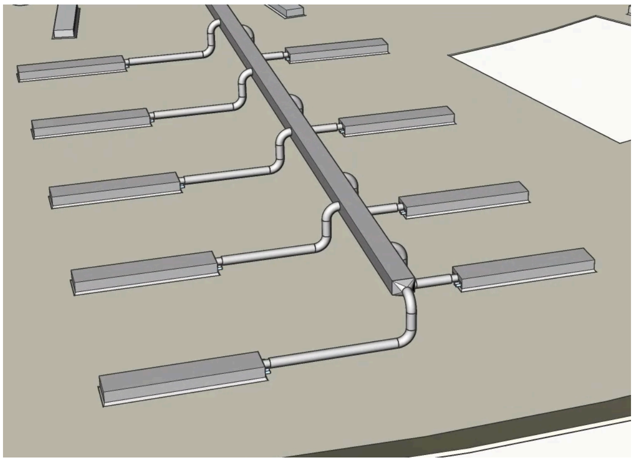
12 Connect all the other flow terminals to the rectangular duct at once.

13 Select the other flow terminals and the rectangular duct and launch the BIMFLOWCONNECT command.



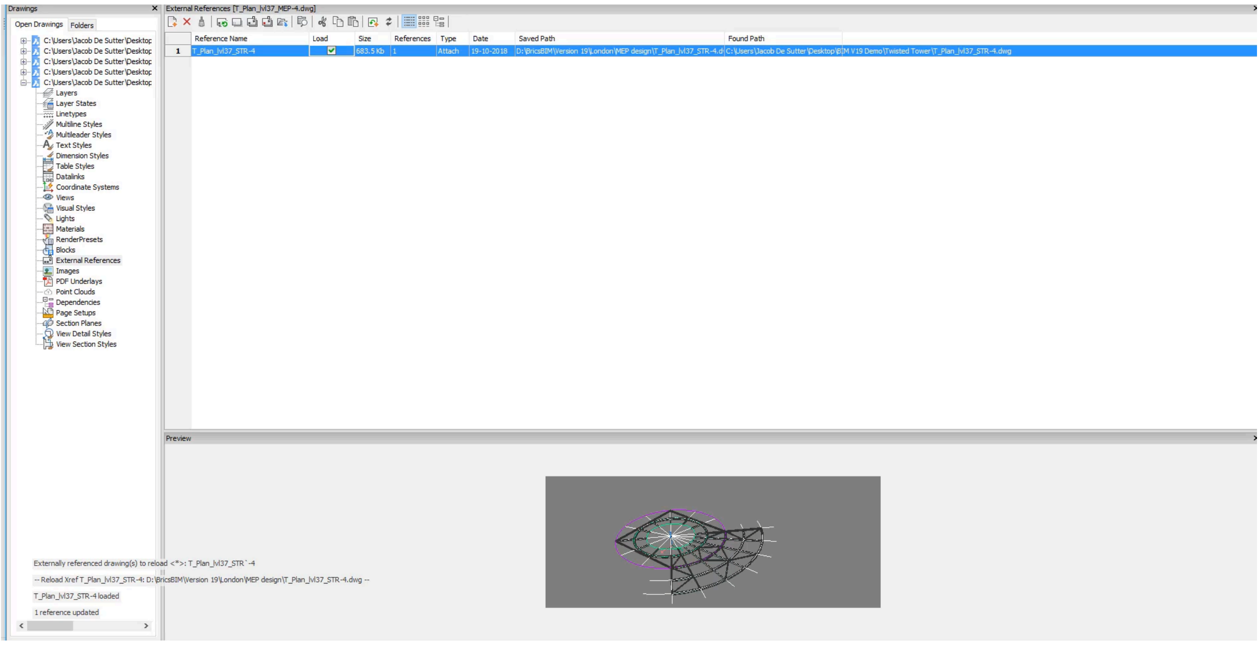
14 BricsCAD uses the information from the flow connection point to determine the type of duct used to connect with the main duct.

15 ***Note:*** The flow terminal on the back is connected to the back of the rectangular duct.



22.13.5 Procedure: Using the MEP Modeling tools to alter profiles

1 Type XREF in the Command line and check the load box next to the structural model to load the structural drawing.

1 

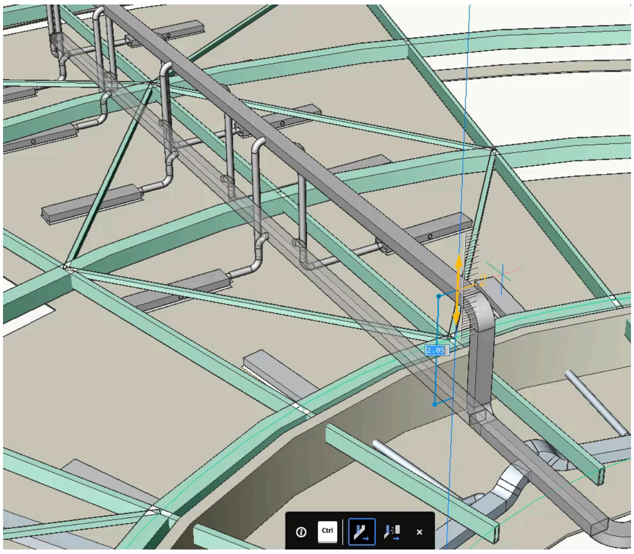
2 Use SLICE command and **Multislice** option to divide the rectangular duct.

Enter 3 in the dynamic dimensions field and press Enter to end the command.

3 Alter the positioning of the rectangular duct, so it doesn't interfere with the structural model.

4 Select the upper face of the duct and launch the DRAG command.

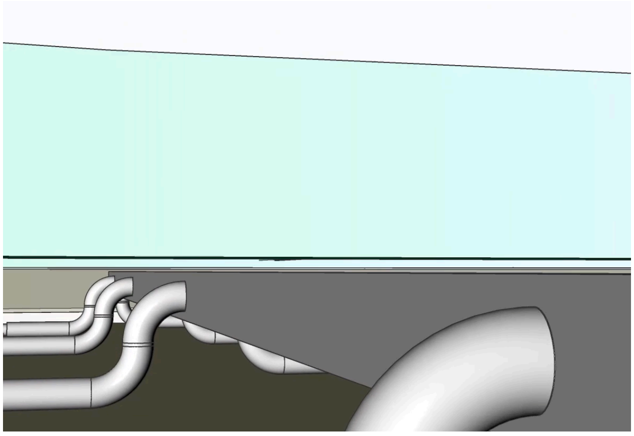
Note: BricsCAD preserves the connections with the circular ducts. You can change this by pressing the CTRL-key if you want to disconnect the ducts.



Move the duct down by 0.23.

5 The structural model no longer interferes with the MEP Model, while the connections between the ducts of the flow terminals and the main rectangular duct were preserved.

6 ***Note:*** When moving a **flow terminal** with the **Manipulator**, its connections with **flow segments** remain intact.



22.13.6 Procedure: using Bimstretch to modify flow segments

1 Open a new drawing.

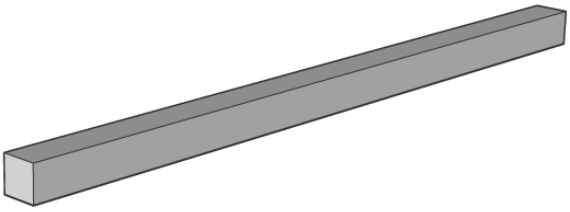
2 Open the **BIM Profiles** panel by clicking on it on the right-hand side of your screen.

Note: If the BIM Profile panel icon isn't visible yet, right-click the ribbon and check **BIM Profiles** in the context menu.

3 Filter on the **HVAC** standard. These profiles will be automatically classified as flow segments once they are dragged into the model space.

4 Search for the profile **HVAC Rectangular Duct 10"x10"** in the **BIM Profiles** panel and drag the profile in the model space.

5 Select a starting point and draw a profile. Press Enter to stop drawing with this profile. Your model should look like this:

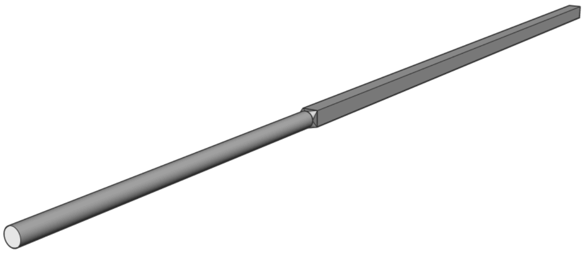
6 

7 Search for the profile **HVAC Round Duct 8"** in the **BIM Profiles** panel and drag the profile in the model space.

8 Click the end of the Rectangular profile, after hovering on its profile face.

9 ***Note:*** To easily snap on the end of the rectangular profile, turn on the **Display Axes** setting or set the DISPLAYAXES system variable to 1.

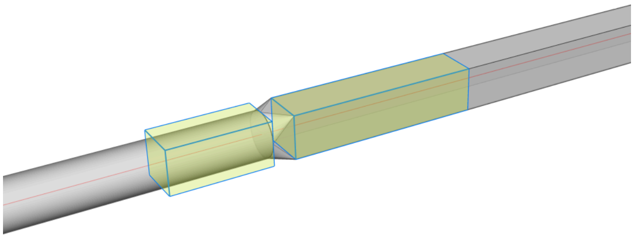
10 Make sure to draw the circular profile along the same axis as the rectangular one. After accepting the last point of the circular profile, it will be connected to the rectangular profile through an automatically generated reducer.

11 

Note: If you had chosen to draw with a non-HVAC profile, you will not get the automatic classification as Flow Segment and the connection will not be made immediately. You can still manually classify the profiles through the BIMCLASSIFY command and then use BIMFLOWCONNECT to connect the two through a reducer.

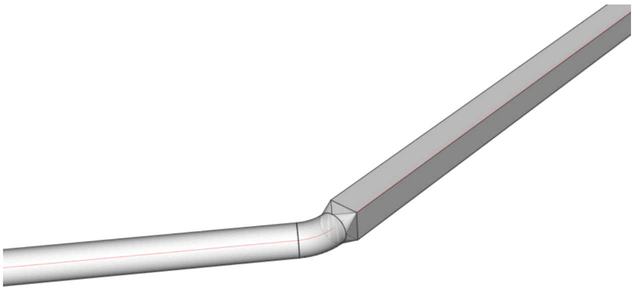
Note: Make sure the **Display Sides and Ends** and **Display Axes** is enabled.

12 Select the connecting ends of both segments.

13 

14 Launch the **BimStretch** tool.

15 Hover your cursor to one of the sides and click the desired location. BricsCAD automatically stretches the 2 Flow Segments in their new position and adds an extra curved Flow Fitting in the middle.

16 

[MEP example file.zip]

22.14 MEP flow connection points

22.14.1 Commands

BIMLINEARSOLID, BIMFLOWCONNECT

22.14.2 About

When creating flow elements like Flow Terminals and Flow Segments, you can add a 2D symbol and classify it as a **Flow Connection Point**. A flow connection point contains BIM data where you can predefine the profile you want to use with this flow component. When you select the flow connection point and use the commands BIMLINEARSOLID or BIMFLOWCONNECT, BricsCAD will automatically use a BIM Profile based on the given information.

See the related links for more information about BIM linear solids.

See the related links for more information about MEP modeling.

22.14.3 Procedure: creating a flow connection point

In this procedure, you will learn how to create your own HVAC components with flow connection points. As an example, we will create an air heat pump unit.

Creating a flow connection point in the Library panel

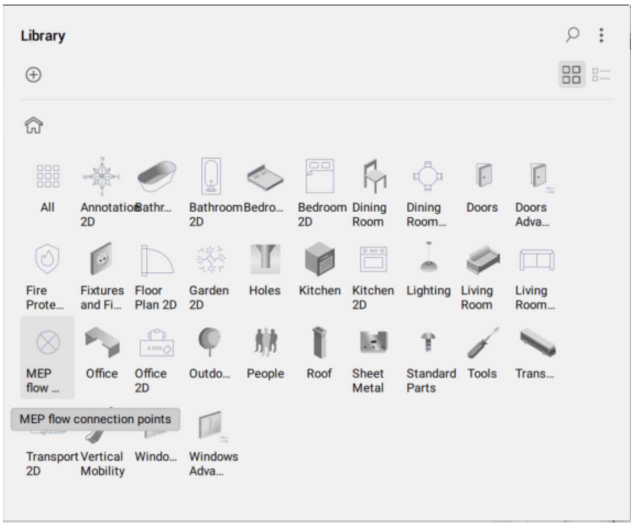
1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

2 Select the box and launch BIMCLASSIFY with the <Other> option or hover over the box and select **Classify Manually** in the **BIM** tab of the Quad.

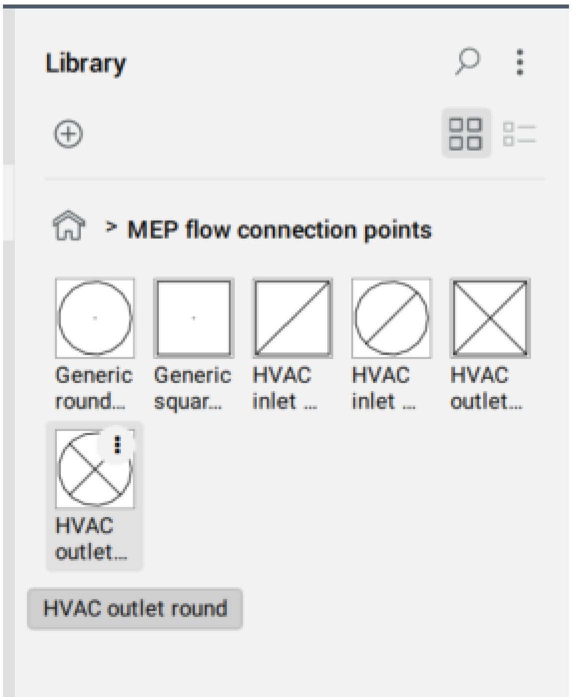
3 Classify the box as **Building Service Elements** > **Flow Terminal** and click **OK**.

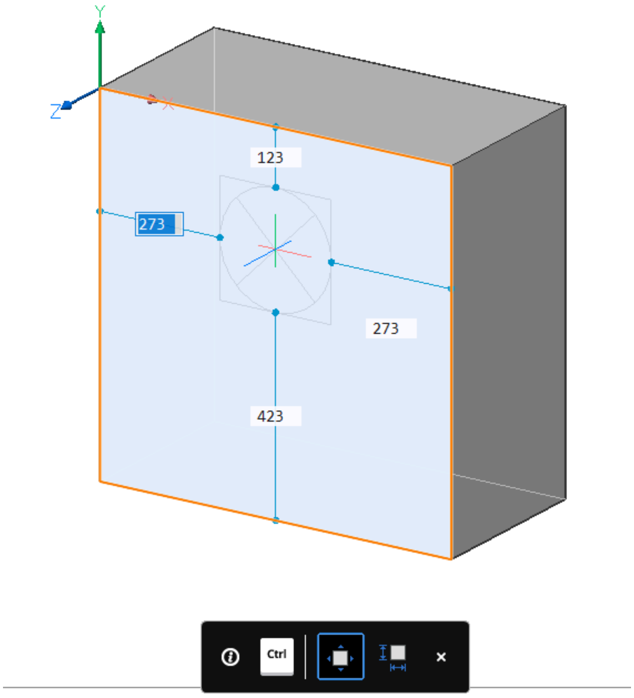
4 Open the **Library** panel.

5 Go to the **MEP Flow Connection Points**.

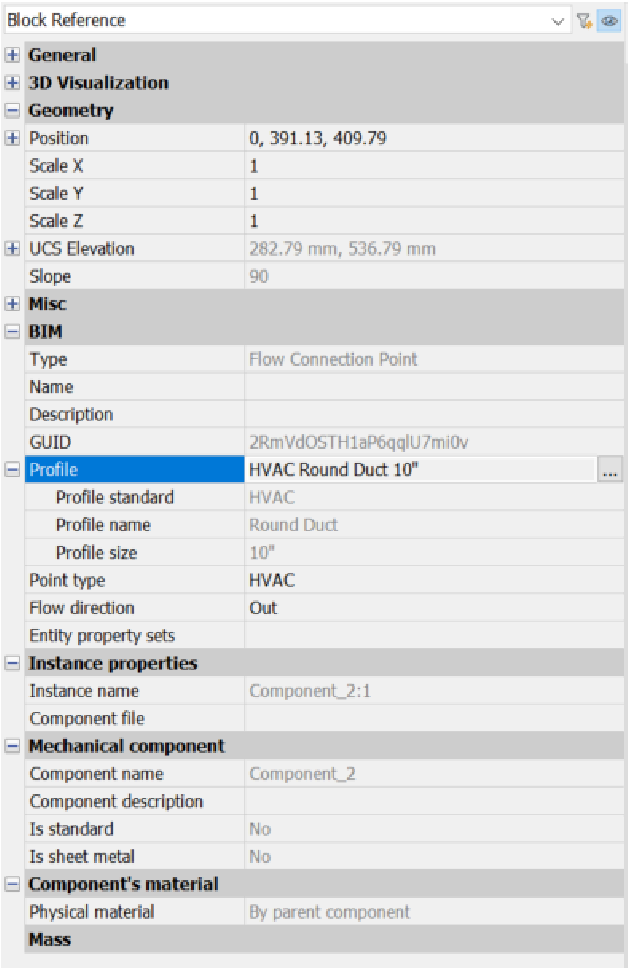


6 Place **HVAC outlet round** on the heat pump box.





7 Select the Outlet and click the **BIM**/**Profile** property in the **Properties** panel.

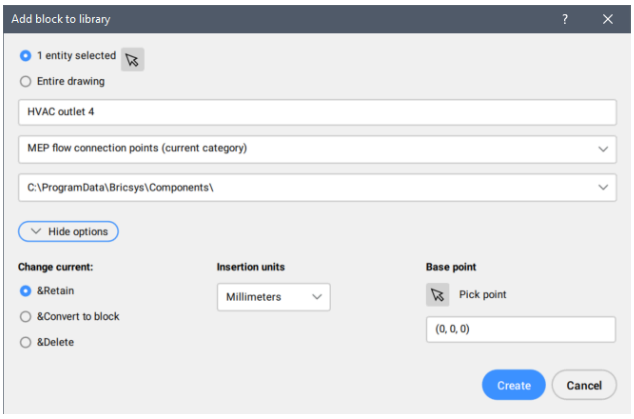


8 Click the browse (...) button.

The **Profiles** dialog box displays.

9 Choose the desired profile and click the **Select** button.

10 Go back to the **Library** panel and click the **Add block to library** button. The **Add block to library** dialog box opens.



11 Choose a name and category and click the **Create** button.

See the **Library** panel article for more information about adding a component to the library.

The drawing is now saved as a component and can be used in other drawings.

Creating a flow connection point from scratch

1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

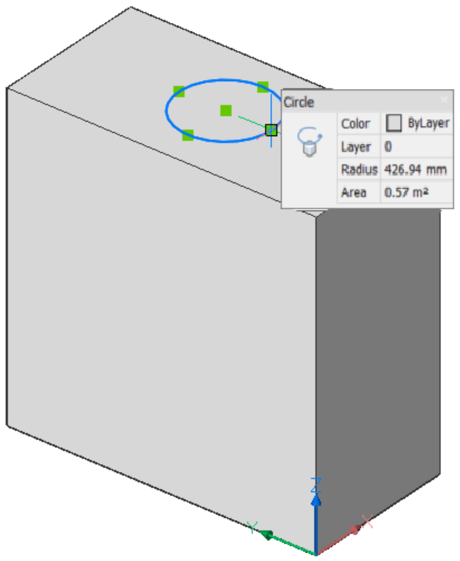
2 Select the box and do one of the following:

- Launch BIMCLASSIFY with the **Other** option.
- Hover over the box and select **Classify Manually** in the **BIM** tab of the Quad.

3 Classify the box as **Building Service Elements** > **Flow Terminal** and click **OK**.

4 Draw a circle on the top face of the box.

5

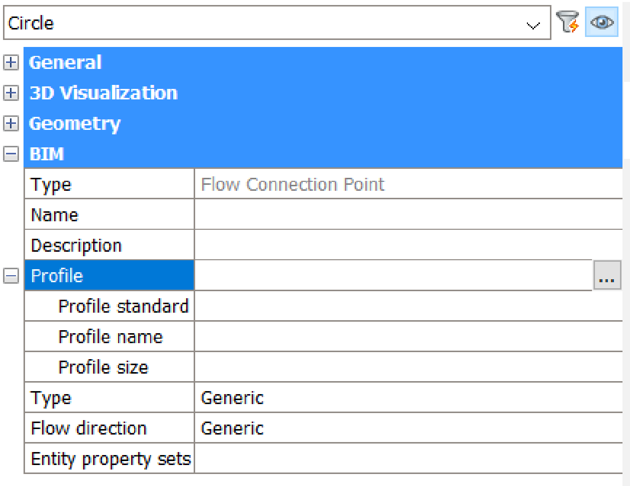


6 Select the circle and launch BIMCLASSIFY with the **Other** option.

7 Classify the circle as **Building Service Elements** > **Flow Connection Point** and click **OK**.

8 Select the circle.

9 In the **Properties** panel, select the **BIM/Profile** property and click the Browse (...) button.



The **Profiles** dialog box displays.

10 Set **Type** and **Standard** as **HVAC**.

11 Select the desired profile and click **Select**.

12 Go to the **Library** panel and click **Add block to library**. The **Add block to library** dialog box opens.

13 Choose a name and category and click **Create**.

The drawing is now saved as a component and can be used in other drawings.

22.14.4 Procedure: Creating an HVAC System

In this procedure, we will use flow connection points to create a ventilation system.

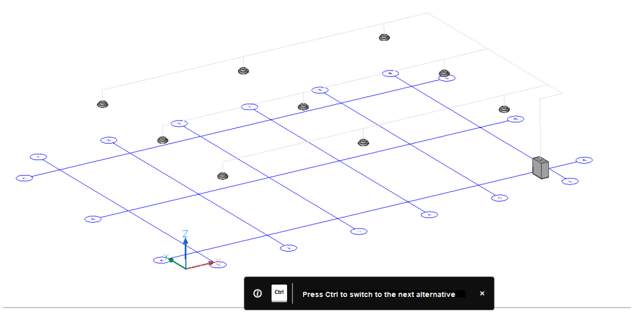
1 Let's start from a file with a series of flow terminals and a ventilation unit, e.g. the attached MEP.dwg file.

2 Select the flow terminals and the ventilation unit.

3 Launch BIMFLOWCONNECT or select **Connect Flow Segments** in the **Structural/MEP** ribbon tab.

4 Press the CTRL-key to switch between the alternative setups and press Enter.

5

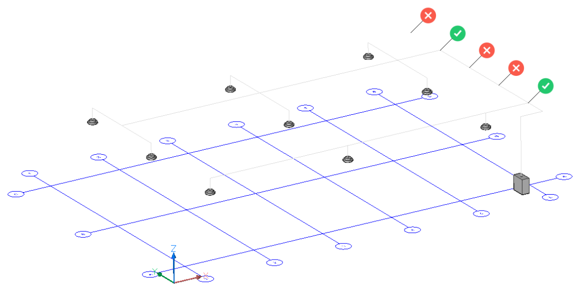


6 Click the **Tune** option or enter T in the Command line to modify the chosen setup.

7 Click the **Topology** option or enter T in the Command line.

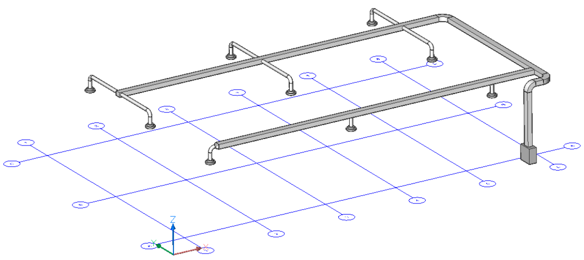
8 Select the check icons to modify the layout of the system.

9



10 Press Enter three times to finish the command.

11 ***Note:*** BricsCAD automatically uses the profiles that were assigned to the ventilation units. This information is stored in the **Flow Connection Points**.



[MEP.dwg](MEP.dwg)

22.15 Making parallel copies of 3D Solids

22.15.1 Commands

OFFSET

22.15.2 About

The **Offset** tool allows you to create copies of linear entities and align them parallel to the original entities at a specified distance.

Depending on the selected options in the **Selection Modes** toolbar, the **Offset** tool allows you to create a parallel copy of:

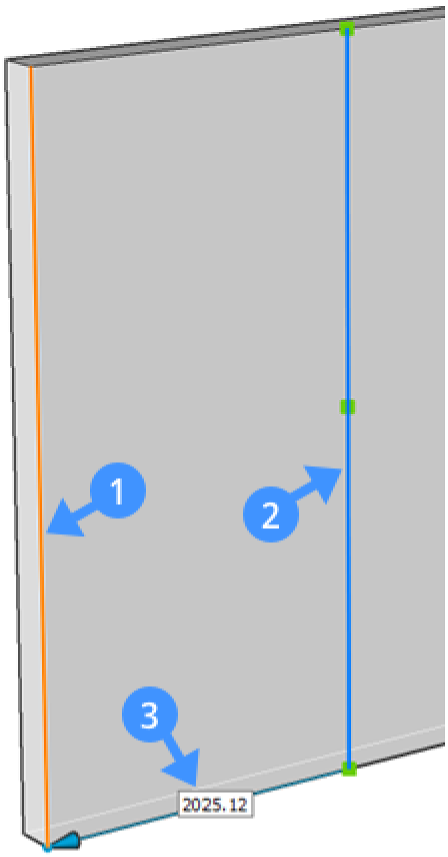
- An edge of a 3D solid if the **Select Edges** option ( ) is enabled or if the SELECTIONMODES system variable is set to 1.
- The edges of a 3D solid face if the **Select Faces** option ( ) is enabled or if the SELECTIONMODES system variable is set to 2.
- Detected boundaries if the **Select Detected Boundaries** option ( ) is enabled or if the SELECTIONMODES system variable is set to 4.

22.15.3 Procedure: make a parallel copy of a 3D solid edge

Before starting the process, make sure that Quad is active and the **Select Edges** option in the **Selection Modes** toolbar is on. Next, follow these steps:

1. Move the cursor over the edge of the solid to highlight the edge.
2. Choose the **Offset** tool from the Quad.
3. Specify the offset distance by dragging the edge of the solid face.

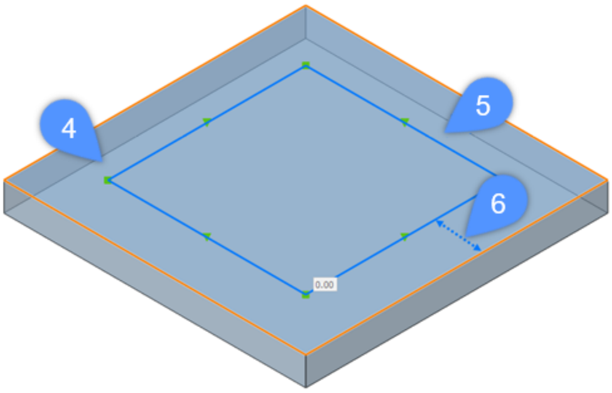
   Note: The distance of the offset displays in the dynamic dimension field. The parallel copy of the edge is created dynamically.
4. While dragging the edge to set a precise offset distance, type a distance in the dynamic entry field and press Enter or left-click to accept the offset.
5. The following illustration shows the offset edge on the 3D solid. The edge highlights in orange is the original edge (1) of the 3D solid, whereas the blue highlighted line with grips (2) is the offset copy of the original one. The line (3) between the original edge and copied edge gives the offset distance.



22.15.4 Procedure: Make a parallel copy of a 3D solid face

Before starting the process, make sure that Quad is active and the **Select Faces** option in the **Selection Modes** toolbar is on. Next, follow these steps:

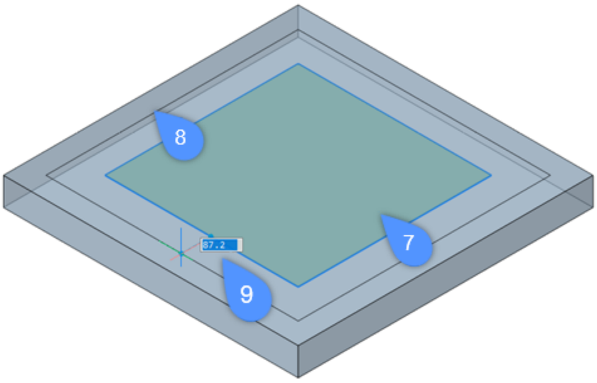
1. Move the cursor over the face of the solid to highlight the solid face.
2. Choose the **Offset** tool from the Quad.
3. Specify the offset distance by moving the cursor outside or inside of the solid.
4. The parallel copy of the face is created dynamically.
5. Enter a distance in the dynamic entry field and press Enter, or left-click to finish the offset.
6. The following illustration shows the offset of the 3D solid face. The highlighted face is the original 3D solid face (4), whereas the created polyline (5) is the offset from the original one. The line (6) between the original 3D solid face and copied polyline gives the offset distance.



22.15.5 Procedure: make a parallel copy of a detected boundary

Before starting the process, make sure that Quad is active and the **Select Detected Boundaries** option in the **Selection Modes** toolbar is on. Next, follow these steps:

1. Move the cursor inside the boundary to highlight the detected boundary.
2. Choose the **Offset** tool from the Quad.
3. Move the cursor to define the offset distance and left-click.
4. The parallel copy of a detected boundary is created dynamically.
5. Enter a distance in the dynamic entry field to set a precise offset distance and press Enter or left-click to accept the offset.
6. The following illustration shows the offset boundary on the face of a 3D solid. The highlighted face is the detected boundary (7), whereas the created polyline (8) is the copy from the original one. The line (9) between the detected boundary and copied polyline gives the offset distance.



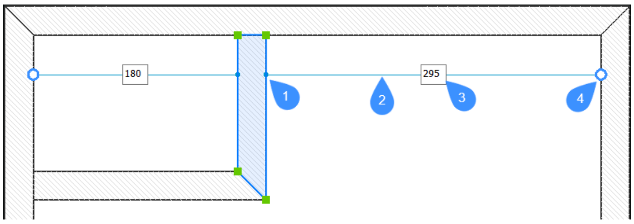
22.16 Selection dimensions

Selection dimensions provide an interface to easily reposition an existing BIM element (wall, column, window or door) by specifying the distances from the selected element to the reference points.

22.16.1 Selection dimension parts

To have access to selection dimensions, select an existing BIM element in the drawing by clicking it.

The four parts composing a selection dimension are:



1 Origin point

2 Line

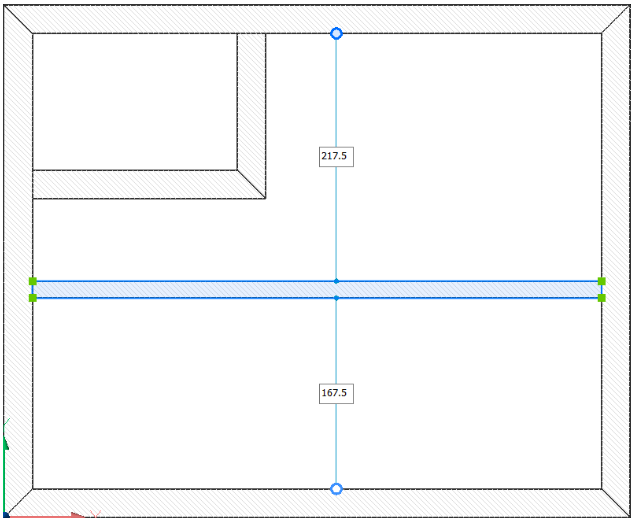
3 Distance field

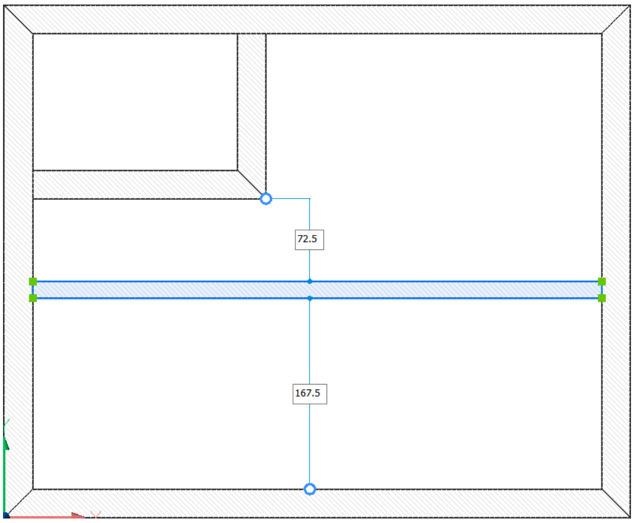
4 Reference point

22.16.2 Using selection dimensions when moving entities

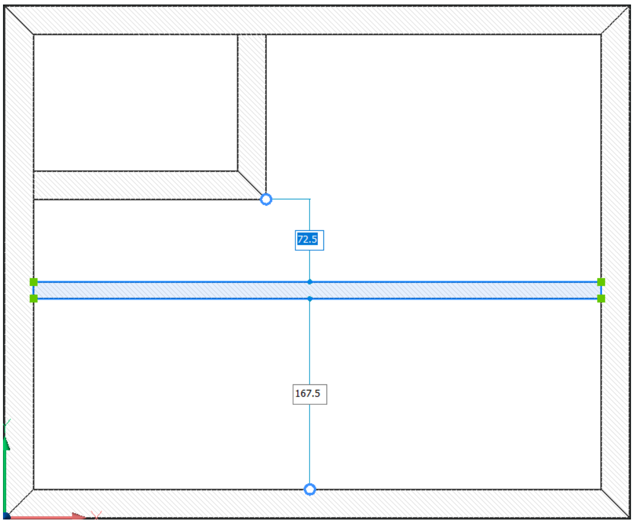
1 (Optional) Enable **Top View Mode** (**TVM**) by clicking a story disk ( ) in the **Story Bar** (see **The Story Bar** article).

2 Select a BIM element in the drawing. The corresponding selection dimensions are displayed.

3 (Optional) Choose a more convenient reference point with two clicks:
- Click a reference point to select it. The reference point becomes gray.
- 

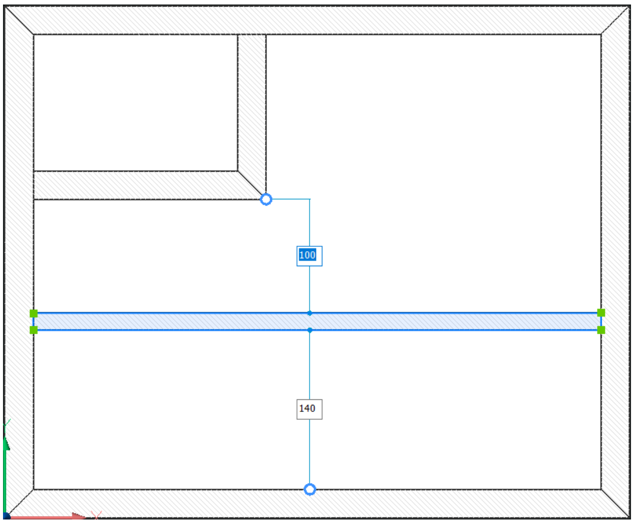
- Click a different point in the drawing to set it as the reference point. The distance fields update their value.
- 

4 Click a distance field.

5 

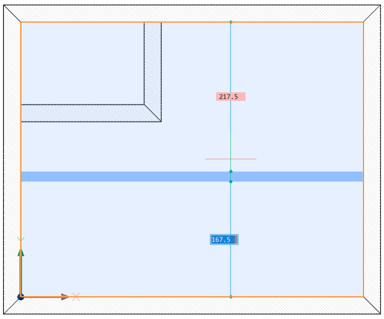
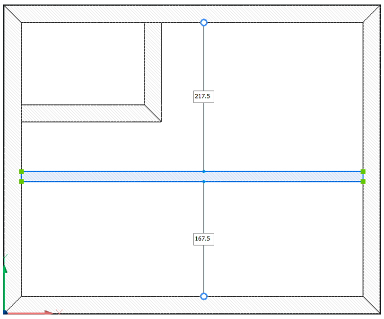
6 (Optional) Press the **Tab** key to jump to the next distance field.

7 Type in a value and press **Enter**. The BIM element is now repositioned accordingly.



8 Press **Esc** to deselect the BIM element.

Note: The differences between dynamic and selection dimensions relate to when they are displayed and the possibility to manually move automatically detected points. These are listed in the table below:

Dynamic dimensions	Selection dimensions
	
• Are displayed while placing a BIM element in the drawing. • Have origin points that automatically follow the cursor without giving the possibility to be manually moved.	• Are displayed after selecting a BIM element from the drawing. • Have reference points that give the possibility to be manually moved to other points than those automatically detected.

See also Dynamic dimensions article.

22.17 Splitting solids

22.17.1 About

The BIMSPLIT command either splits segmented solids into separated solids automatically or allows to split a solid using a selection of cutting faces.

22.17.2 To split a solid automatically

Make sure that the value of the SELECTIONPREVIEW system variable is 2 or 3.

1 a Do one of the following:

- Click the BimSplit tool button () on the **Model** tab of the ribbon.
- Choose **Split** in the BIM menu.
- Type BIMSPLIT in the Command line, then press Enter.

You are prompted: Select solid for auto-splitting or cutting faces for manual splitting:

2 Place the cursor over a face of the solid.

3 The face highlights.

4 Without moving the mouse, hit the TAB key until the complete solid highlights, then click.

5 You are prompted:

Entities in set: 1

Select solid for auto-splitting or cutting faces for manual splitting:

6 Right click.

7 The selected solid is split.

The number of parts is reported in the command window.

Using the Quad

Make sure the **Select Faces** option ( ) of the SELECTIONMODES system variable is switched off.

Place the cursor over a face of the solid.

When the solid highlights, choose Split ( ) in the BIM command group of the Quad.

The solid is split.

The number of parts is reported in the command window.

22.17.3 To split a solid using cutting faces

Make sure that the value of SELECTIONPREVIEW system variable is 2 or 3.

1 Do one of the following:
   - Click the **BimSplit** tool button on the **Model** tab of the ribbon.
   - Choose **Split** in the **BIM** menu.
   - Type BIMSPLIT in the Command line, then press Enter.

   You are prompted: Select solid for auto splitting or cutting faces for manual splitting:

2 Place the cursor over a face of the solid.

3 Hit the TAB key to highlight an obscured face.

   When the face highlights, click to select the face.

   You are prompted:

   Entities in set: 1

   Select solid for auto splitting or cutting faces for manual splitting:

4 (Optional) Repeat step 4 to select more faces.

5 Right-click to stop selecting faces.

6 The solid is split using the selected faces as cutting planes.

   The number of parts is reported in the command window.

Using the Quad

Make sure the **Select Faces** option of the SELECTIONMODES system variable is set.

1 Place the cursor over a face you want to use as a cutting plane.

1 When the face highlights, do one of the following:

2 Go to step 3.

3 Go to step 2 to select more faces.

4 (Optional) Place the cursor over a face you want to use as a cutting plane.

5 When the face highlights, do one of the following:

6 Go to step 3.

7 Repeat this step to select more faces.

8 Choose **Split** in the **BIM** command group of the Quad.

The solid is split using the selected faces as cutting planes.

The number of parts is reported in the command window.

22.18 Stair tool

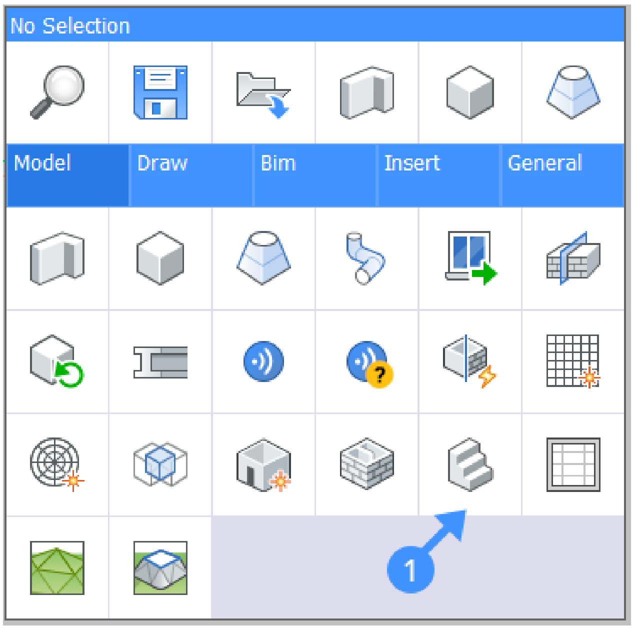
22.18.1 About

The **Stair** tool is a powerful tool that allows you to add a custom flight of stairs for your project in just two clicks. You can change the default stair settings as required. The **Stair** tool will automatically try to fit these requirements. It is possible to create single-flight straight, double-flight L-shaped, and double or triple-flight U-shaped stairs.

For more information about this command, visit the Command reference article BIMSTAIR.

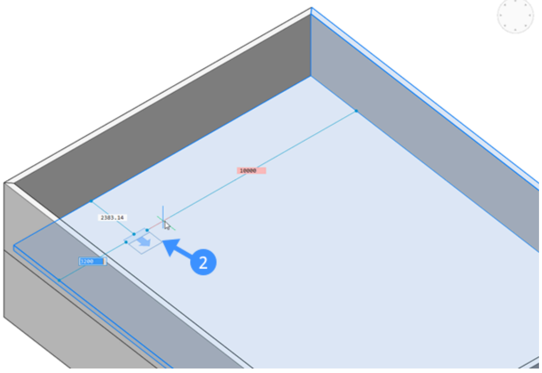
22.18.2 Creating a straight flight of stairs

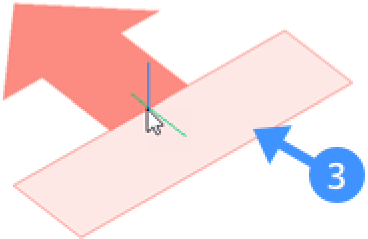
1 Select the **Stair** tool from the Quad (1).



2 Hover over a slab. The Stair Cursor will appear (2) and the slab on which the stair cursor hovers is highlighted. This indicates the position of the first step. If the position of the cursor is invalid, the cursor will turn red (3).

3





4 To flip the direction of the first step, choose the **Flip** option or type **F** in the command line.

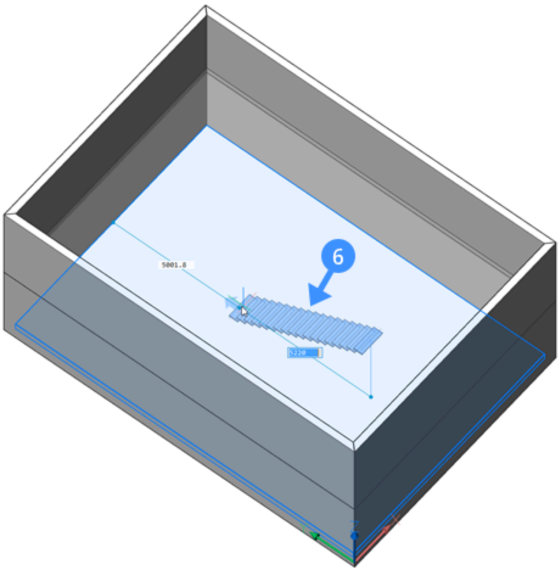
To turn the direction of the first step 90°, choose the **Quarter-turn** option or type **Q**.

To cycle through the different stair types, use the **Ctrl** key when the **Hotkey Assistant** is active.

Note: The cursor will automatically flip or quarter-turn when it is positioned close to a wall or the edge of a slab. This auto-quarter-turning/flipping behavior is only active when the **Automatically flip/ quarter-turn** option is activated.

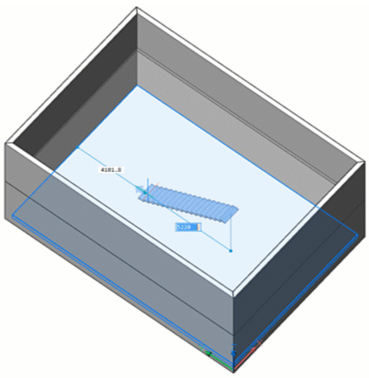
5 Once the cursor is in position, left-click with the mouse. If the upper slab is blocking the view, the slab will become transparent.

6

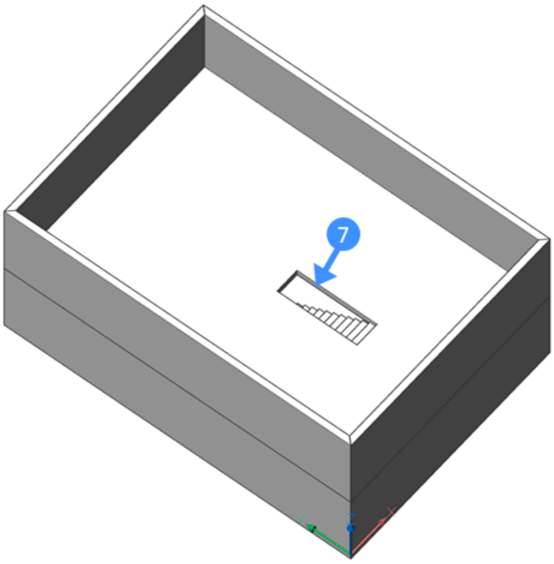


7 To create a straight flight of stairs, align the mouse perpendicular to the first step.

Note: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (6). The **Stair** tool will automatically snap to the configurations that correspond to the **Preferred tread length** setting. When the optimal configuration is satisfying, this preview will display in a darker shade of blue. The other settings are taken into account as well, but not to this specific snapping feature (where the stair preview turns darker blue).



8 Once you have the cursor in the correct position, left-click. A flight of stairs will be created. A hole is automatically generated in the upper slab, corresponding to the **Head room** setting (7).



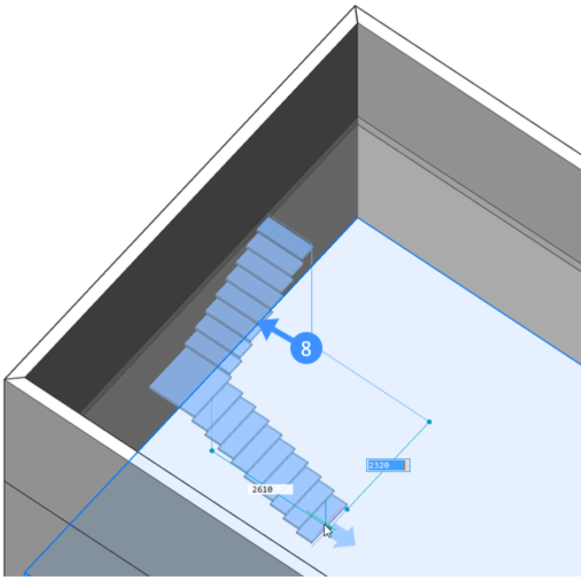
22.18.3 Creating an L-shaped staircase

1 Select the **Stair** tool from the Quad.

2 Hover over a slab. The Stair Cursor will appear. Left-click once the stair cursor is in the correct position.

3 Move the cursor away from the perpendicular to create an L-shape flight of stairs (8).

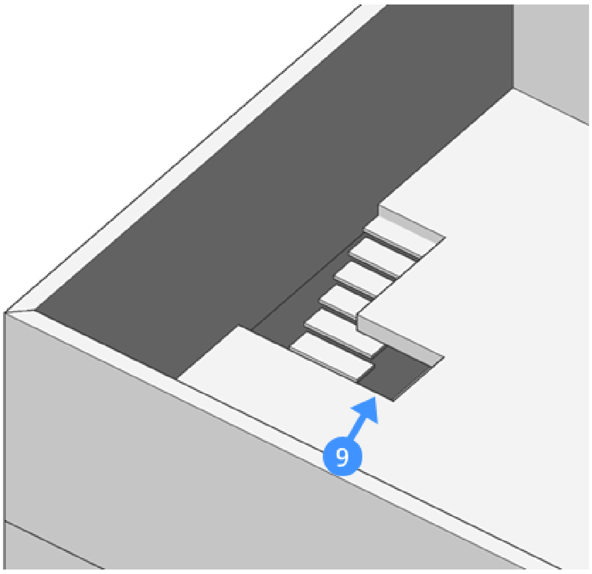
4



Note: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (8). The **Stair** tool will automatically try to find optimal configurations of the stair that correspond with the dimensions given in the **Stair** dialog. It will snap to the optimal configuration and display this preview in a slightly darker color.

5 When you are satisfied with the preview, left-click. A flight of stairs will be created and a hole will be automatically generated (9). This time, the hole can be L-shaped, depending on the length of the upper flight and the **Head room** stair setting. If the head room is so large it cannot be covered with only the

topmost stair flight, it will span also a part of the second stair flight and thus be L-shaped.



22.18.4 Creating a U-shaped staircase

1 Select the **Stair** tool from the Quad.

2 Hover over a slab. The Stair Cursor will appear. Left-click when the stair cursor is in the correct position.

3 Move the cursor away. Type **Q** and **Enter** to rotate the direction of the second cursor 180° with respect to the first cursor.

Or use the **Ctrl** key when the **Hot Key Assistant** (**HKA** field in the **Status** bar) is **ON**.

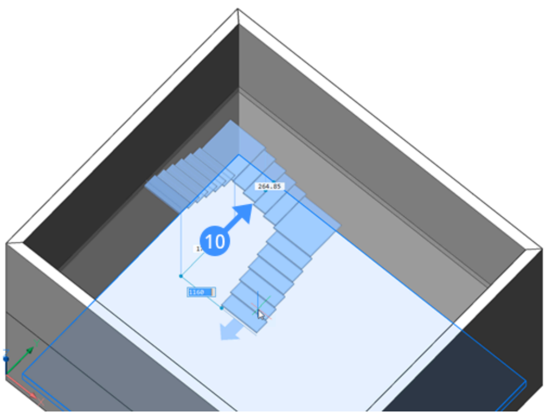
1 Select the **Stair** tool from the Quad.

2 Hover over a slab. The Stair Cursor appears. Left-click when the stair cursor is in the correct position.

3 Move the cursor away from the perpendicular to get an L-shaped stair.

4 Press the **Ctrl** key to switch between an L-shaped stair and a U-shaped stair.

Or align the last step with the first fixed step. When entering a strip parallel to the first step, the last step will automatically quarter turn.

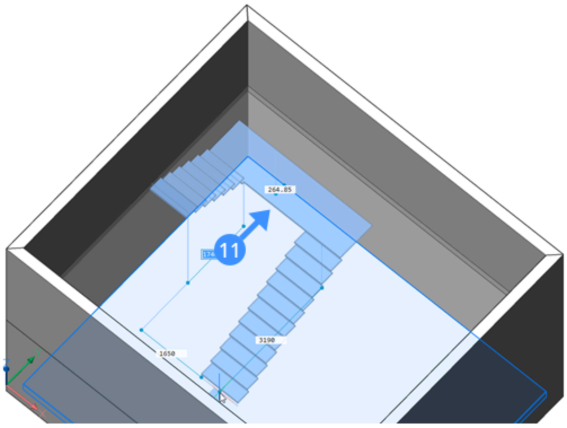


Note: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (10). The **Stair** tool will automatically snap to the configurations that correspond to the current settings. When the optimal configuration is satisfied, this preview will display in a darker shade of blue.

5 There are two possible types of U-shaped stairs you can switch between:
   - The middle section includes steps (10).
   - The middle section is a landing in between the two opposite stair flights (11).

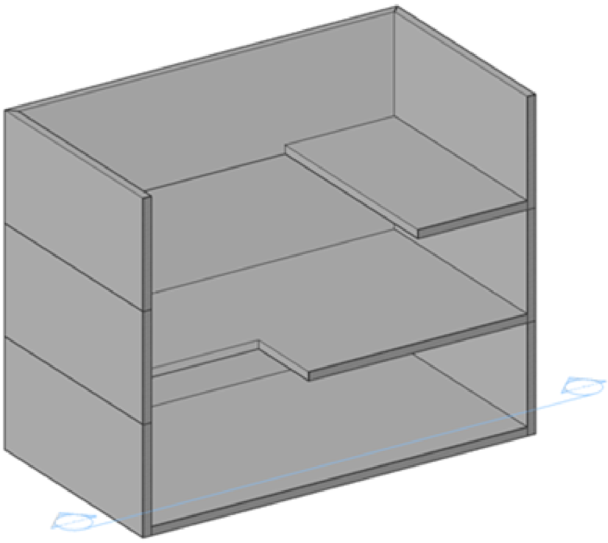
   It is possible to switch between these two modes by typing **U** and hitting **Enter**.

   Or press **Ctrl** to cycle between different stair variants.



22.18.5 Creating a flight of stairs in an existing opening

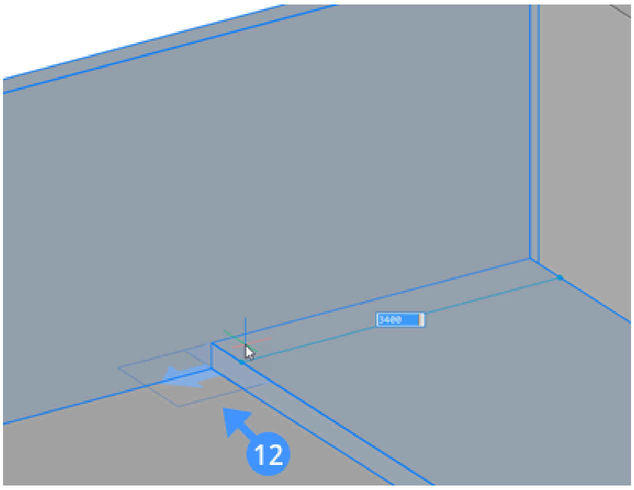
Sometimes it is necessary to create a flight of stairs starting from the edge of a slab, where an opening already exists.



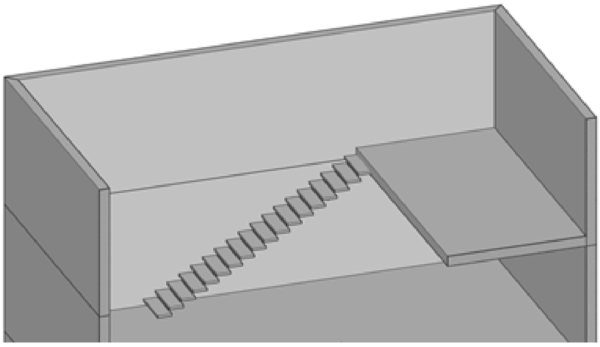
In the example above, different floors need to be connected with flights of stairs. However, an opening does not need to be created.

1 Select the **Stair** tool from the Quad.

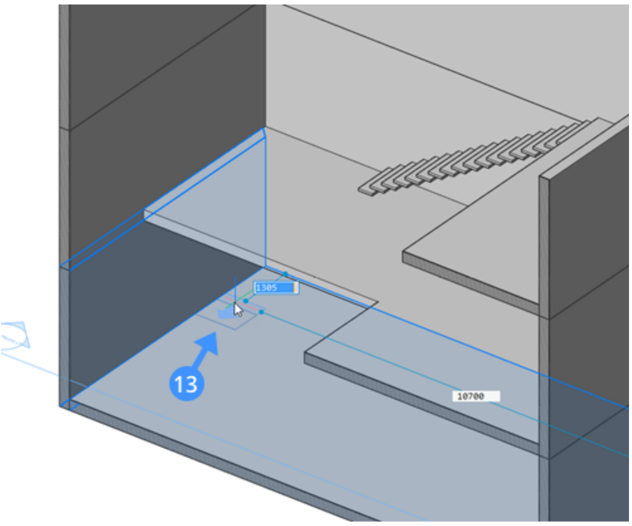
2 Hover the Stair Cursor near the edge of the slab (12). The cursor automatically snaps to the outer side of the edge.

3 

4 Create a flight of stairs as described above.

5 

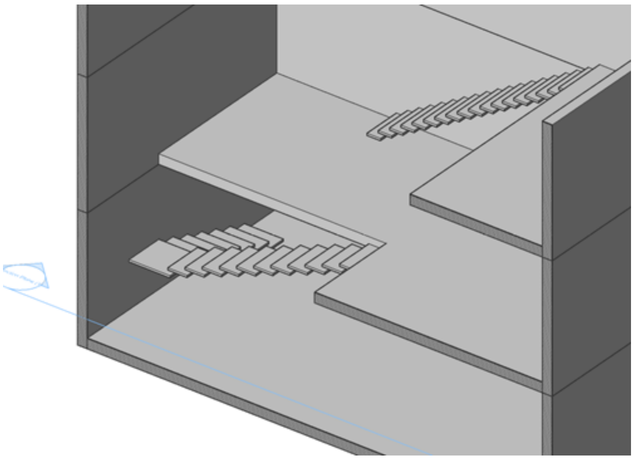
6 You can also make a flight of stairs by starting at the lowest point. Select a starting point on the first floor (13).

7 

8 Hover over the edge of the slab on the second floor. The Stair Cursor automatically snaps to the edge

of this slab. Left-click to create the flight of stairs.

9



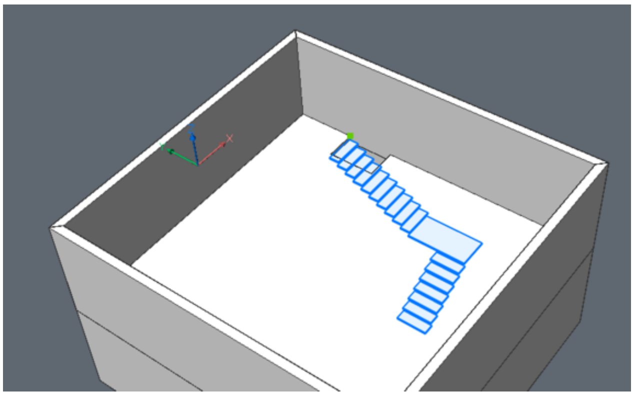
22.18.6 Editing a flight of stairs

You can adjust multiple parameters of a flight of stairs created with the **Stair** tool in the **Properties** panel. These parameters include:

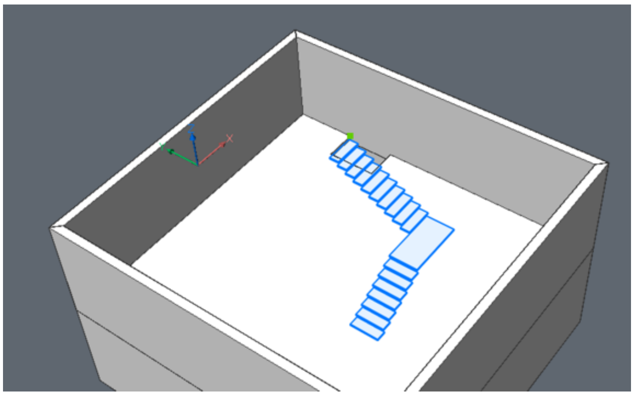
Parameters	
Height_Staircase	325 cm
Stair_Width	1000 mm
Ideal_Riser_Height	170 mm
Step_Thickness	50 mm
Nosing	50 mm
Length_StairFlight_0	145 cm
Length_StairFlight_1	174 cm
Length_StairFlight_2	145 cm

Height_Staircase	The overall height of the flight of stairs. The height is fixed at the top and is adjusted from the base.
Stair_Width	The width of the flight of stairs. When a flight of stairs is selected, a green dot appears on the upper slab. This dot indicates the insertion point of the flight of stairs. When this point is in the middle of the flight of stairs, the flight of stairs will widen from the center. When it is located at the side, for example when a stair is attached to a wall, the stair will only widen the side away from the dot.
Ideal_Riser_Height	The ideal riser height. When creating a flight of stairs, the real riser height will be calculated based on the ideal riser height and the height of the steps, so that the risers are equally divided as close to the ideal riser height as possible.
Step_Thickness	The step thickness. The upper face of the steps is fixed so that the thickness increases downwards.

Nosing	The horizontal overlap distance between subsequent treads.
Length_StairFlight_0	The length of the first flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs.
Length_StairFlight_1	The length of an optional, second flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs. This option is only available for L-stairs and U-stairs.
Length_StairFlight_2	The length of an optional third flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs. This option is only available for U-stairs.
Landing_Extension_Up	The length of the landing going down the first flight of stairs. The first flight is fixed so the second flight adjusts to the given parameter value.
Landing_Extension_Down	The length of the landing going down the second flight of stairs. The first flight is fixed so the second flight adjusts to the given value.



Setting: **Landing_Extension _Up**.



Setting: **Landing_Extension_Down**.

You can re-position a flight of stairs created with the **Stair** tool using the MOVE command.

For complex edits to a flight of stairs created with the **Stair** tool, use the BEDIT command to edit the block definition. When in the BEdit-mode, navigate to the **Mechanical Browser** panel. Here you can see and edit all the constraints that apply to a flight of stairs.

22.19 Structural steel

22.19.1 Commands

BIMAPPLYPROFILE, BIMLINEARSOLID, BIMSTRUCTURALCONNECT, BIMADDECCENTRICITY, BIMRECALCULATEAXIS, DRAG, BIMSTRETCH

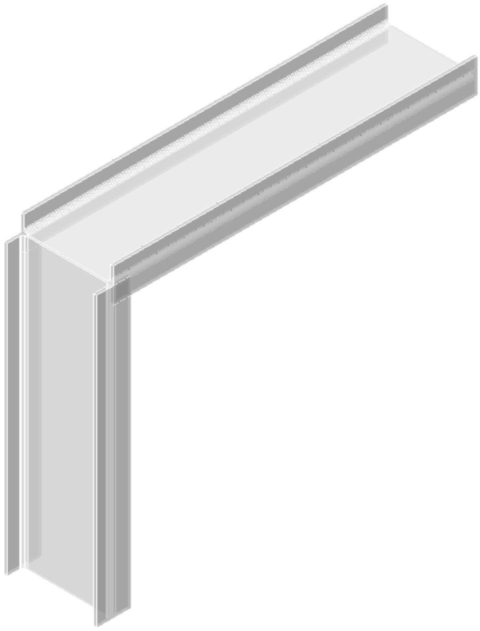
22.19.2 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the **BIM Profiles** panel to the model space.

  Note: This command works in the same way as the LINE command.

2 To define the starting point and the following point(s) click in the model space or use the dynamic dimension fields.

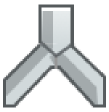
3 Press Enter to end the command.



Note: BricsCAD doesn't automatically generate the connections between the different segments because the chosen profiles are structural steel.

22.19.3 About BIMSTRUCTURALCONNECT

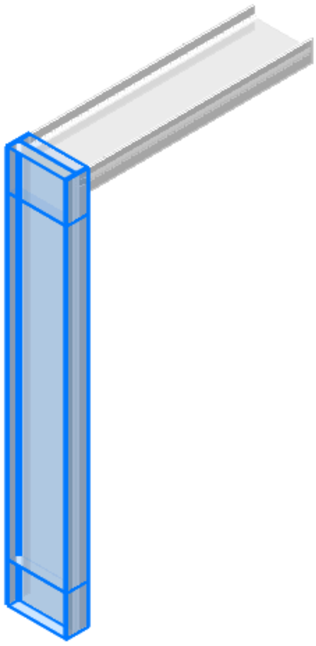
The BIMSTRUCTURALCONNECT command automatically generates connections for profiles.



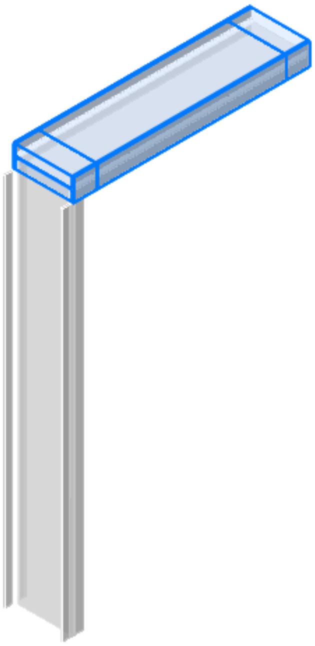
For more information about this command, visit the Command Reference article BIMSTRUCTURALCONNECT.

Making a structural connection to linear solids

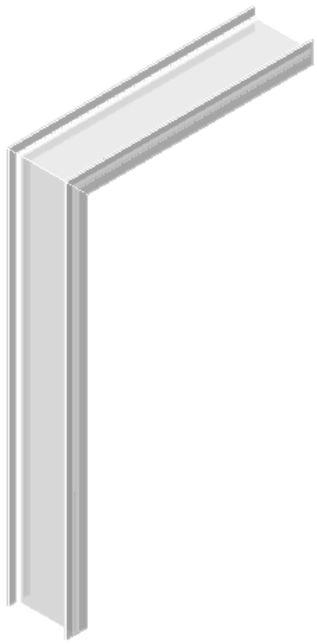
1. Select the profiles you want to connect with.
2. Launch the BIMSTRUCTURALCONNECT command by typing it in the Command line or select **StructuralConnect** in the **Model** tab in the Quad.
3. When the Hot Key Assistant is on (**HKA** on the **status** bar), a widget (as displayed below) appears on the screen. Tap the CTRL-key to cycle through the different options.




Ctrl




Ctrl



4 Press Enter to accept the selected option and end the command.

22.19.4 About DRAG

The DRAG command makes it possible to alter a segment of a drawn profile without losing the connection to other segments.

For more information about this command, visit the Command Reference article DRAG.

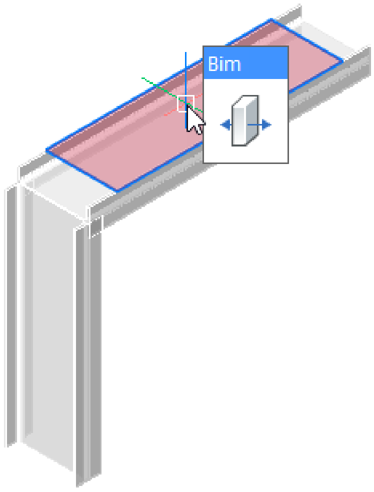
Altering the positioning of segments

Note: This only works if the connections were created with the BIMSTRUCTURALCONNECT command.

1 Set the value of DISPLAYSIDESANDENDS system variable to 1.

2 Select a face perpendicular to the section of the profile.

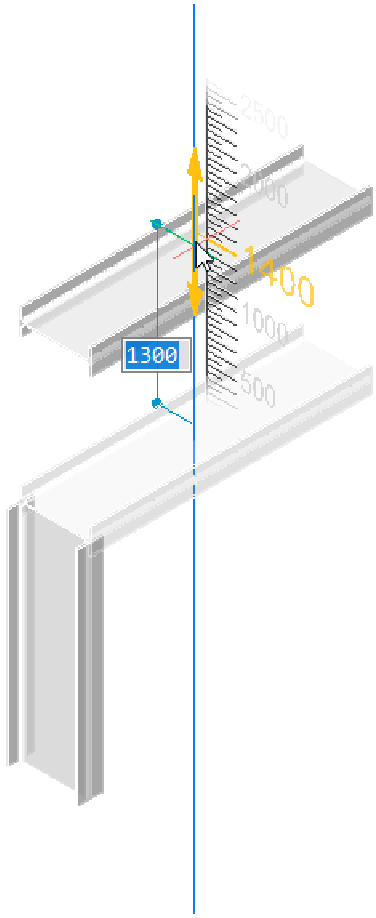
3



4 Launch the DRAG command.

5 Click the mouse to set a new position for the segment. Or use the dynamic dimension field to enter a value for the displacement.

6



22.19.5 Procedure: use modeling tools to create a structural steel model

1 Draw a BimGrid as a base.

2 Launch the BIMGRID command.

3 Click a starting point and enter values into the dynamic dimension fields.

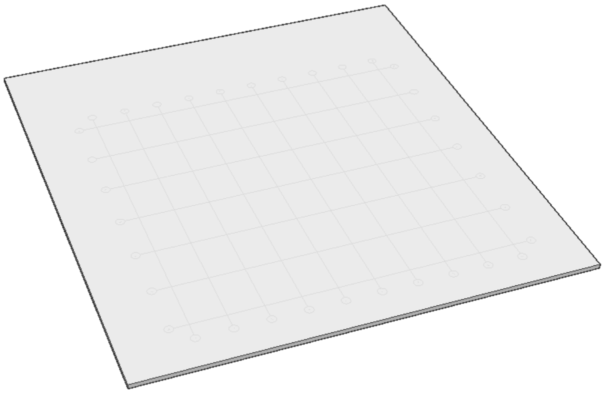
4

5 Draw a base slab underneath the grid.

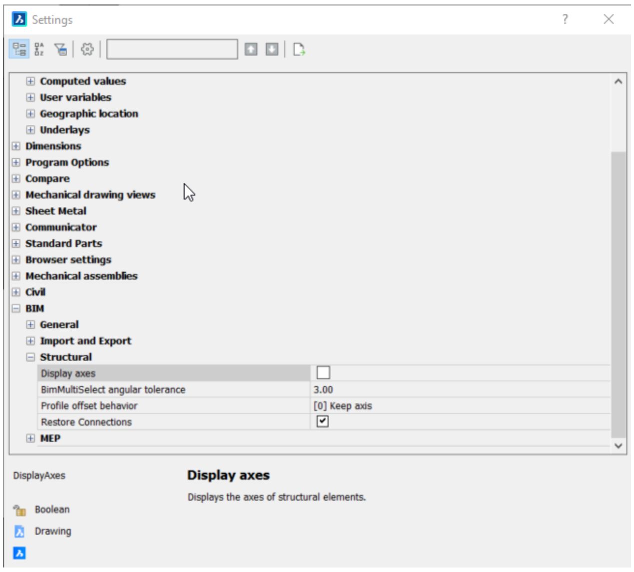
6 Create a slab using the BOX command.

7 Ensure that the grid is entirely on top of the slab and that the axes of your profiles are displayed.

8

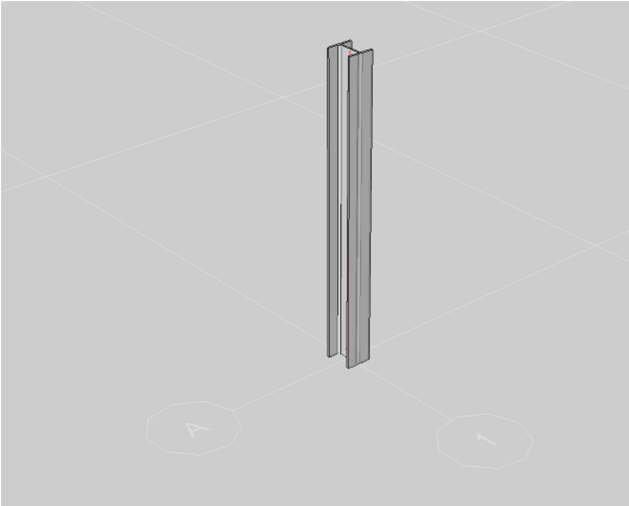


9 Go to the **Settings** dialog and under the **BIM** > **Structural** section. Ensure that **Display Axes** is checked.



10 Open the **BIM Profiles** panel and drag a structural steel profile into your model. Start drawing at the intersection point of two grid axes and draw vertically upward.

Note: To snap specific points, make sure you have toggled on the different types of points you want to snap in the context menu when right-clicking on ESNAP.



11 Launch the BIMPROPAGATE command.

12 Select the slab as base solid and press Enter.

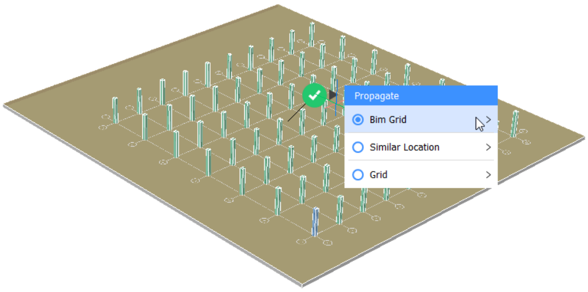
13 Select the column as your detail element and press Enter. BricsCAD will zoom into the 3D detail. Press **C** to copy the column as a solid, not as a block.

Note: The default setting for BIMPROPAGATE is to copy the object as a block.

14 Hit Enter.

15 ***Note:*** BricsCAD automatically copies the beam to every grid intersection. This is because you chose the floor slab with a BimGrid as your base solid.

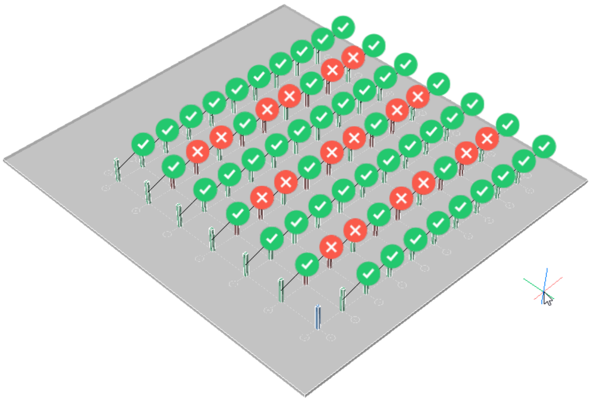
16 Hover over the green checkmark and click **Bim Grid** in the dialog box.



17 Click **Explode**. You can now manually toggle individual suggestions to create the pattern of your choice.

18 Press Enter to accept and end the command.

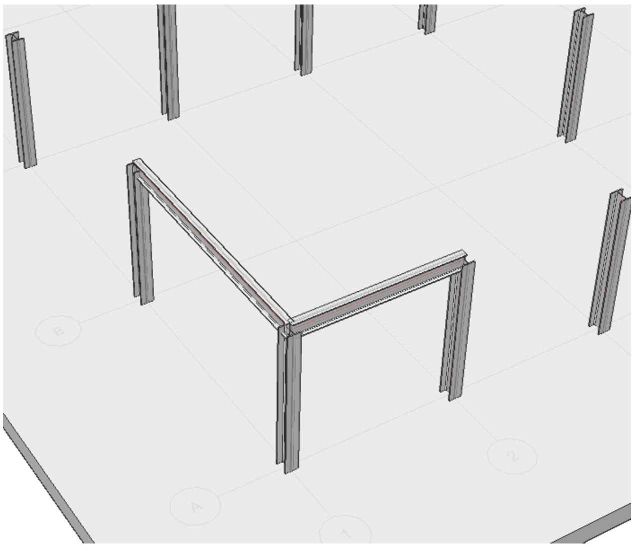
19



Create beams between the columns

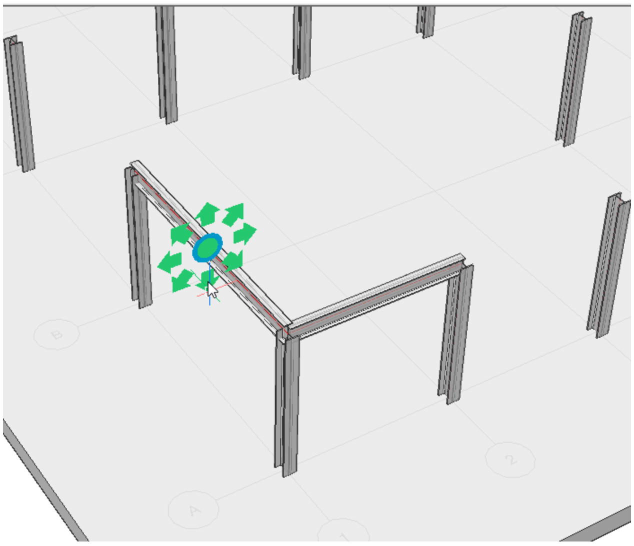
1 Open the **BIM Profiles** panel and drag a structural steel profile into your model. To set the start and endpoints, click the top of the column and then the next. Do this in both directions.

Note: If you want to turn your profiles to make sure the flanges are facing upwards, press Q (from Quarter turn). The profile will turn 90 degrees around its axis.



2 Drag and drop a beam profile from the **BIM Profiles** panel onto this beam. Give the second beam a large profile, as it has a longer span.

3 Select a beam and launch the BIMADDECCENTRICITY command to alter the relative position of the axes of the beams.

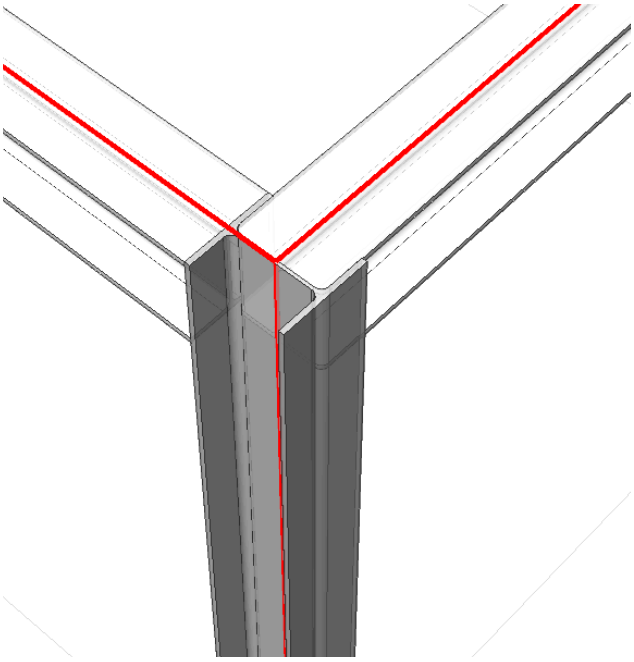
4 

5 Click the down arrow. The beam moves down in relation to its axis.

6 Press Enter to end the command.

7 Repeat steps 3-5 for the other beam.

8 Select the column and the two beams then launch the BIMSTRUCTURALCONNECT command to make a simple connection. Press Enter to accept the default option.

9 

10 Repeat step 7 for the other 2 junctions.

11 Select the two columns connected to the beam in the Y-direction and launch the BIMPROPAGATE command.

12 Select the beam in the Y-direction as the detail solid.

13 Press C to copy it as a solid, not as a block.

14 Press Enter.

15 Choose which suggestions to accept. Press Enter to accept and finish the command.

16 ***Note:*** Suggestions are made at the bottom of every column because the situation is symmetrical. You can toggle the suggestions off.

17 Repeat steps 9 - 13 to create beams on the X-direction.

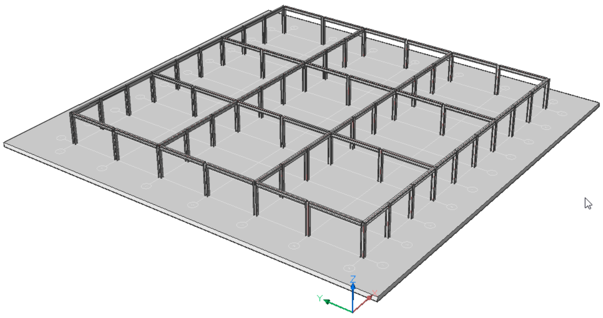
18 ***Note:*** You can use box selections to select all the lower beams at once.

Move a Beam

1 Set the value of DISPLAYSIDESANDENDS system variable to 1.

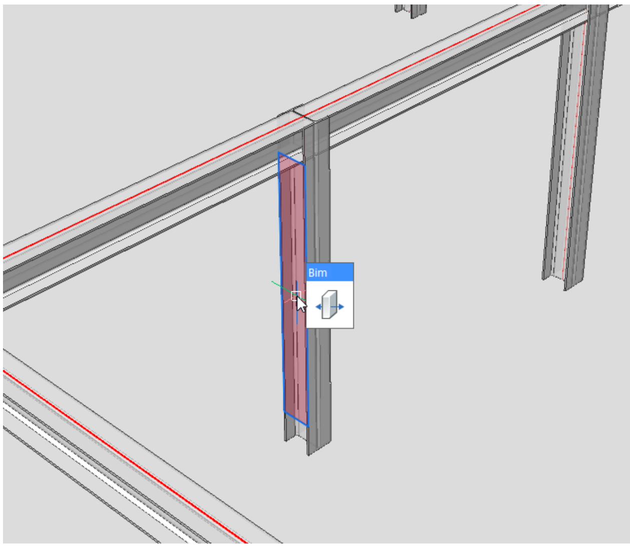
2 Highlight the side face of the desired column and launch the DRAG command.

3



4 Move the column to the desired position.

5



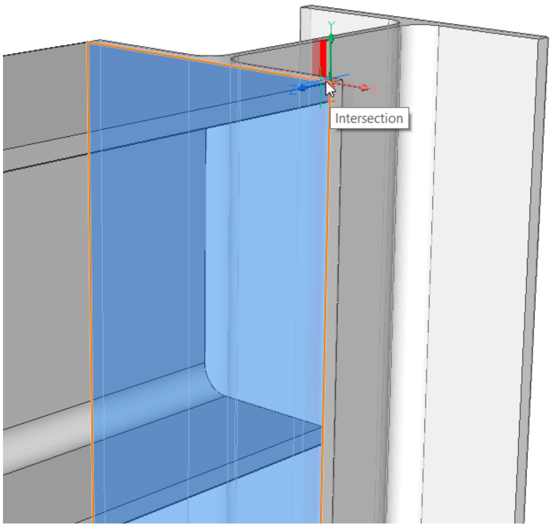
Note: The connections with the adjoining beams are maintained.

22.19.6 Procedure: use the modeling tools to create a more detailed connection

1 Set the value of DISPLAYSIDESANDENDS system variable to 0.

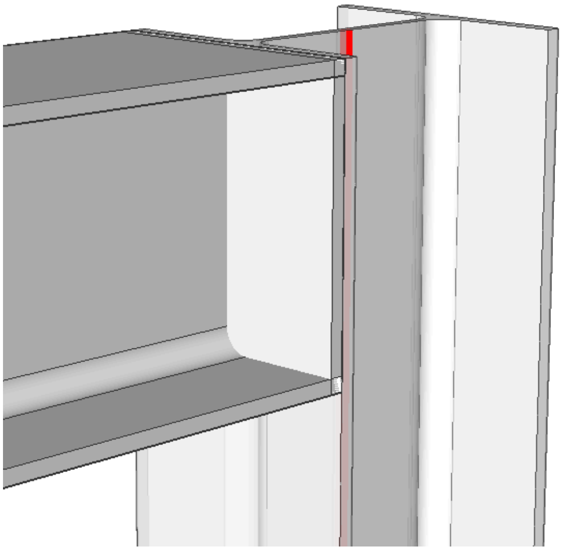
2 Launch the BOX command and hover over the side face of the column.

3 Press Shift once so that the face is highlighted in blue.

4 

5 Now draw a plate on the flange of the column.

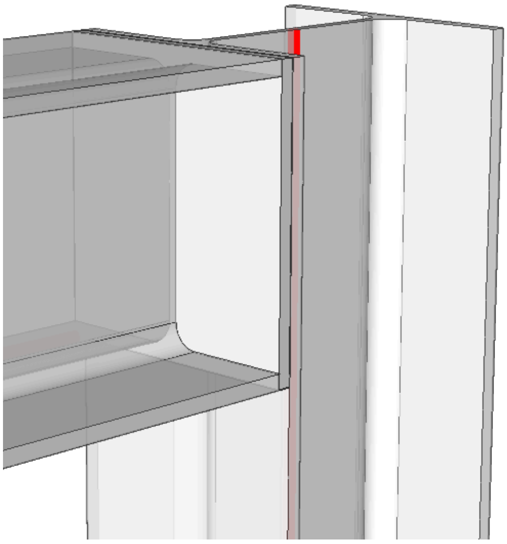
6



7 Select the beam and launch the SUBTRACT command.

8 Select the end plate and press Enter.

9

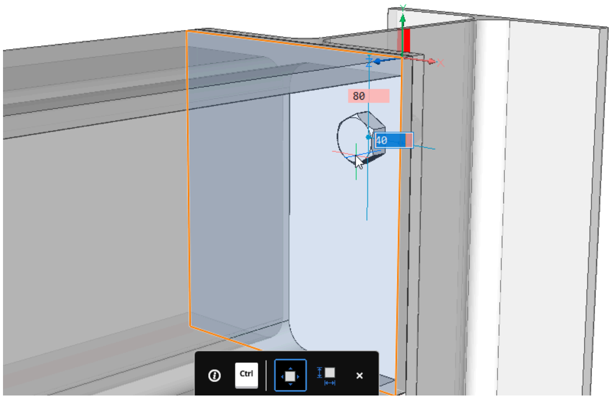


10 Select the face and launch the BIMIFY command from the Quad.

Note: You can also Classify the plate manually by Quad selecting **Classify Manually** in the **BIM** tab.

11 Open the **Library** panel and drag a bolt onto the plate.

Note: You can position it precisely by entering values in the dynamic dimension fields.

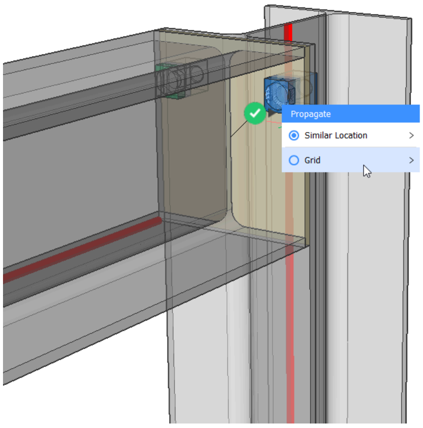


12 Select the plate and launch the BIMPROPAGATE command.

13 Select the bolt as the detail and press Enter.

14 Press Enter again and click on the blue question mark. It will turn into a green checkmark.

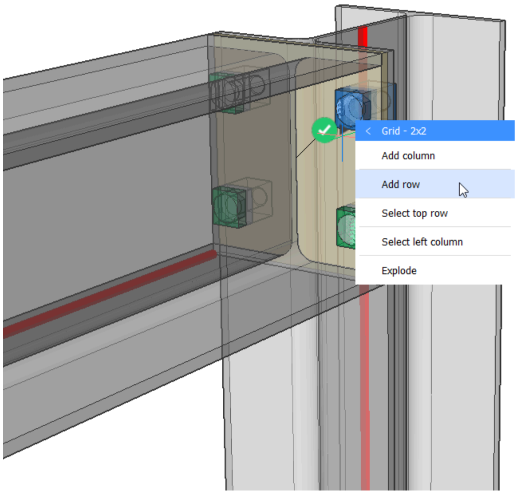
15 Hover over the checkmark and select **Grid**.



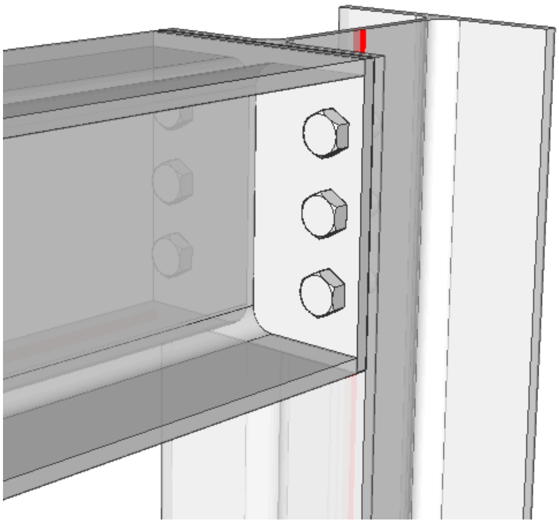
16 Add one row.

17 Press Enter to end the command.

18



The detailed connection has now been created.



22.19.7 Procedure: using BIMSTRETCH to modify structural steel

1 Open a New drawing.

2 Open the **BIM Profiles** panel by clicking it on the right-hand side of your screen.

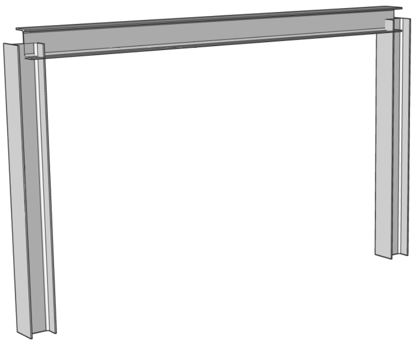
 Note: If the **BIM Profiles** panel icon isn't visible yet, right-click the ribbon and check **BIM Profiles** in the context menu under **Panels**.

3 Drag the profile you want to draw a structure with within the model space.

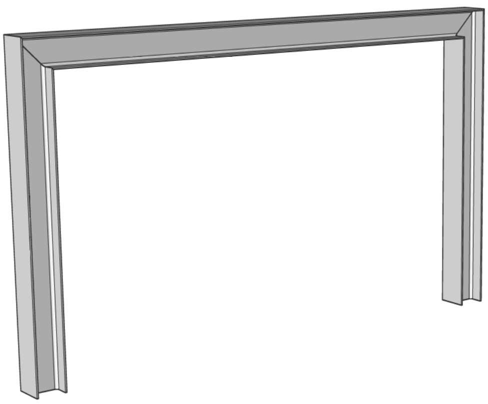
4 Draw a column with a height of 3000 mm. Don't exit the command and draw a connecting beam with

a length of 5000 mm horizontally. Now, add another column at the end of the beam. The height of this column is also 3000 mm.

5 ***Note:*** Make sure you turned the profiles in the right way.



6 Select the 3 beams and select **Connect Structural** in the ribbon or select the command from the Quad. You will be able to choose which connection you prefer. Choose the L-connection option and press enter. Your entities are now connected.



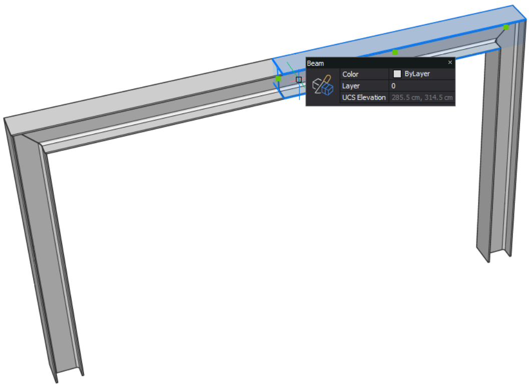
7 In the ribbon, make sure you enable the **Display Sides and Ends** and **Prioritize selection of Faces**. You can find them in the **Structural/MEP** ribbon tab or set the value of DISPLAYSIDESANDENDS system variable to 1 and SELECTIONMODES system variable to 2.

8 Check if the profiles are correctly classified as **Beam** and **Columns**.

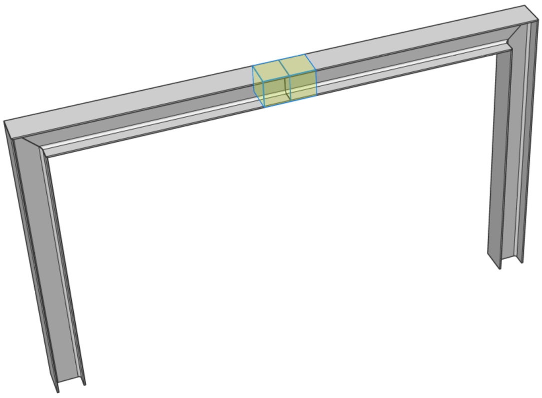
9 Launch the SLICE command and slice the beam in the middle.

10 You will be prompted: Slice along axis? Press Enter to accept the default option (Yes).

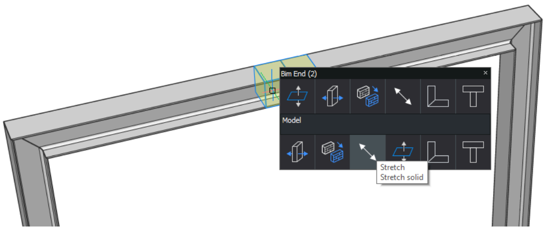
11 Select the midpoint of the beam and press Enter. Your beam is now sliced into two beams.

12 

13 Select the 2 ends of the beams in the middle of the structure. They will be highlighted in yellow.

14 

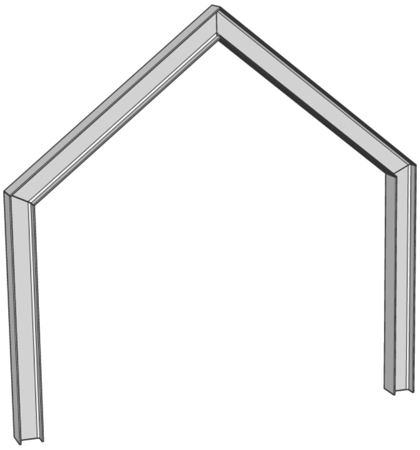
15 Select the **Stretch** tool in the Quad from the **Model** tab to run the BIMSTRETCH command.



16 Hover orthogonally upwards, enter a height of 2000 mm and press Enter to accept.

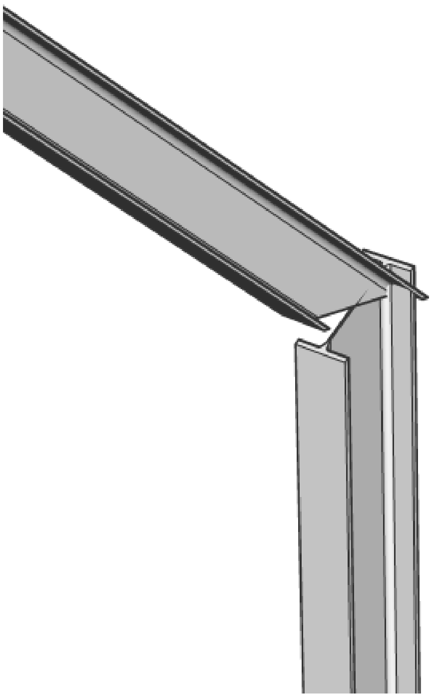
17 The result should look like this.

18



As you can see, the connections between the profiles are automatically adapted to the new angle between the entities.

Note: If you would only have selected one of the two side-ends in the middle of the beam, the connections on the side that was not selected won't be included in the stretch and will look like this:



22.20 Using BIM Quickdraw

22.20.1 About

The **Quickdraw** tool allows you to get started with conceptual modeling easily. It also automatically adds BIM data. You can use the **Quickdraw** tool to create a room or a whole building. Once in use, **Quickdraw** will remain active until you explicitly exit the command or until another command is executed.

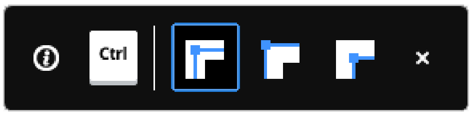
For more information about this command, visit the Command Reference article BIMQUICKDRAW.

22.20.2 Room and walls settings

The BIMQUICKDRAW command opens the **Quick Draw** command context panel. This allows you to set the room dimensions by directly typing the needed value or by using the field arrows. You can also set the wall placement configuration and snapping options, as well as adopt dimensions of nearby rooms by checking the **Auto adopt sizes** option.

To change the justification of the walls, press the **CTRL** key while the **Quickdraw** justification **Hotkey Assistant** widget is displayed at the bottom of the screen.

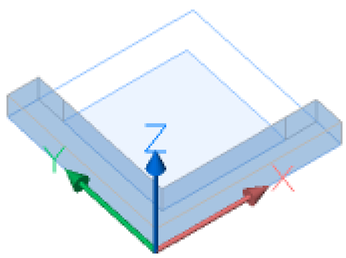
Note: The **Hotkey Assistant** widget is displayed if the HOTKEYASSISTANT system variable is set to 1 and the **Display Hotkey Hints for BIMQUICKDRAW options** checkbox is ticked in the **Hotkey Assistant Configuration** dialog box.

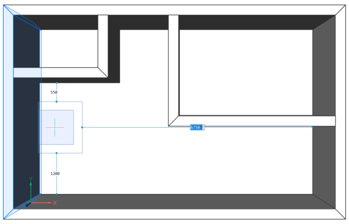
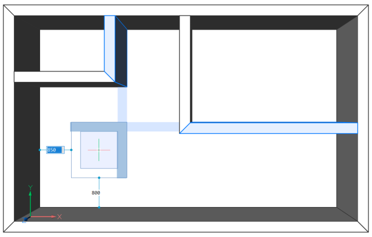


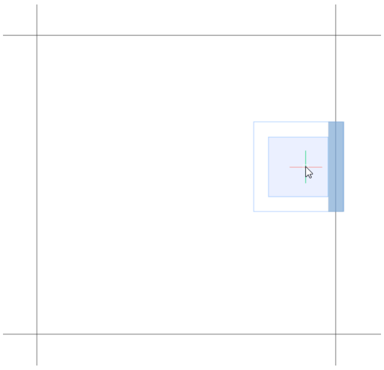
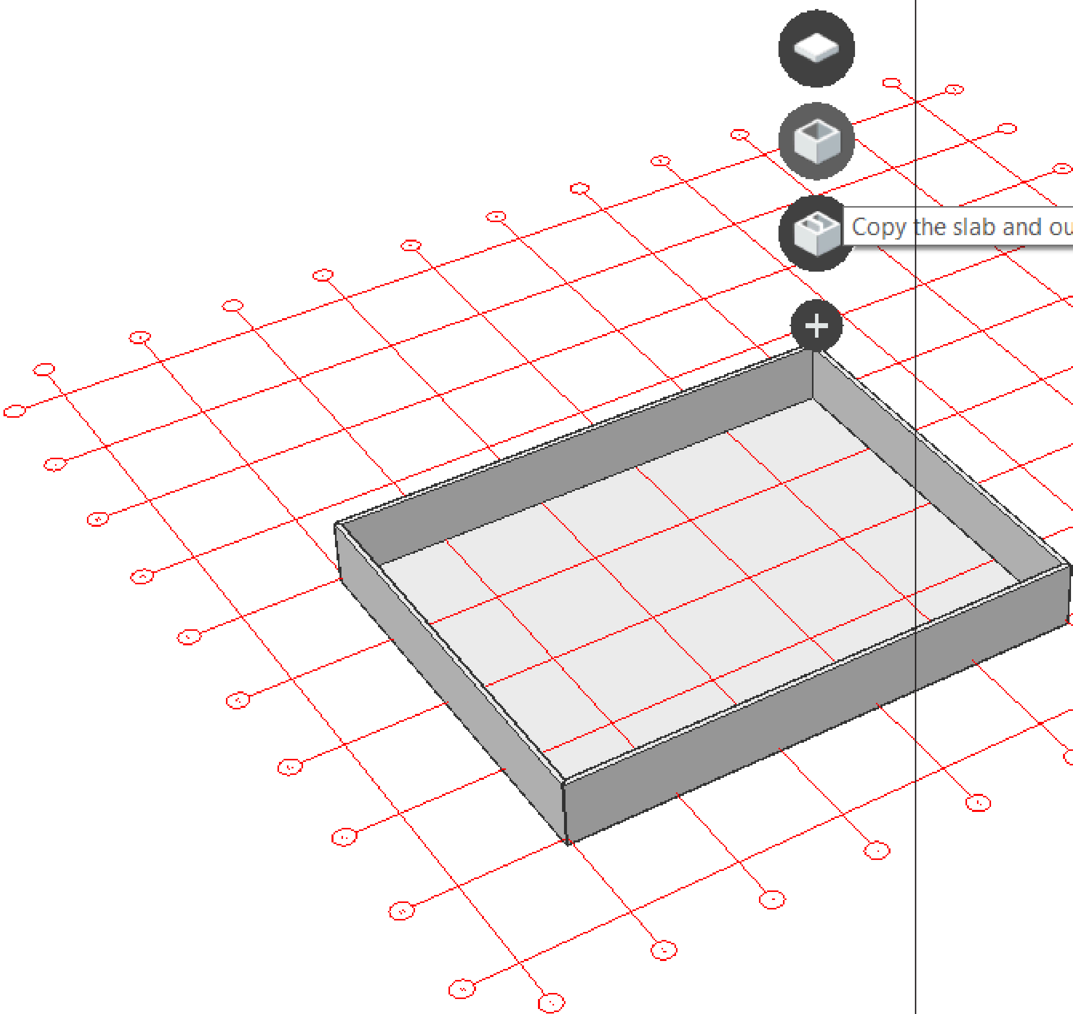
22.20.3 The magnetic snapping feature

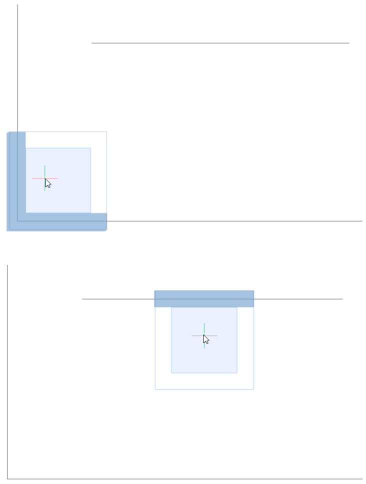
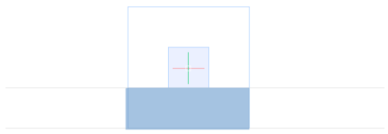
With the magnetic snapping feature, you can easily align the **Quickdraw** cursor. When the command is active, move the cursor close to the object you want to align to. The **Quickdraw** cursor automatically snaps to the object.

You have the following options:

1. Align to the origin	

2. Align to existing walls	To the wall itself 
	To the extents of the wall 

3. Align to lines	To grid lines  ***Note:*** If you use **Quickdraw** to copy a story, the gridlines will only be copied if the grid is assigned to the same spatial location of the ground floor slab and walls. If this is not the case, the gridlines that lie in the boundary of the ground slab will be projected on the slab of the top floor. This is only a visual help. The gridlines on the slab of the top floor will disappear when you exit the BIMQUICKDRAW command. 

	To X- and Y- aligned lines and line segments 
	To parallel lines 

22.20.4 Procedure: creating an L-shaped house

1 When the Quad cursor is in the **No Selection** state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the **Quickdraw** icon in the ribbon or type BIMQUICKDRAW in the Command line.

You are prompted: **Select first point:**

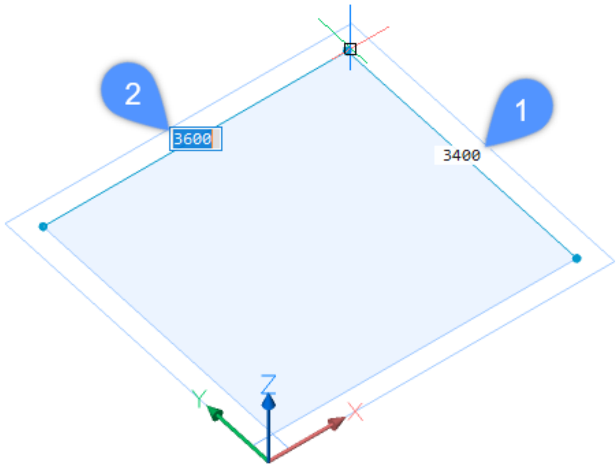
Note: Use the **Quick Draw** command context panel to change the settings before you click and then specify the first corner of a building.

2 Select any point in the drawing area to set the first corner.

3 ***Note:*** To start from the origin, use the magnetic snapping feature of **Quickdraw**.

4 Drag the mouse cursor to define the footprint of the building.

5 ***Note:*** If the **Dynamic dimension** (DYN) option is on, the current values of the room width (1) and length (2) are displayed in a dynamic dimension field.

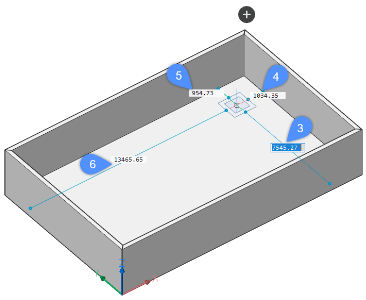


Specify the values of the room width and length in the **Dynamic dimension** field. Press the **Tab** key to switch between width and length and press **Enter** or just click freely. Press **Enter** or **Escape** to leave the QUICKDRAW command or you can continue to create more rooms.

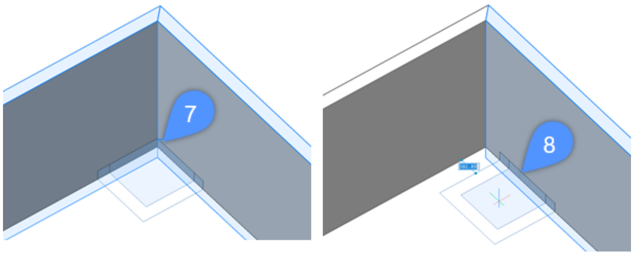
6 Place the **Quickdraw** cursor inside the existing building to create more rooms.

When you place the **Quickdraw** cursor inside the existing building, the dynamic dimension fields display the distances from the **Quickdraw** cursor to the adjacent walls (3,4,5,6).

When the **Auto adopt sizes** option is checked, the dimensions of the walls of the new room will be the same as the wall you snap to. If you don't want this to happen, uncheck this option in the command context panel.



If the cursor aligns with one or two existing walls in the model, the layout of the cursor will adapt and depict the alignment.



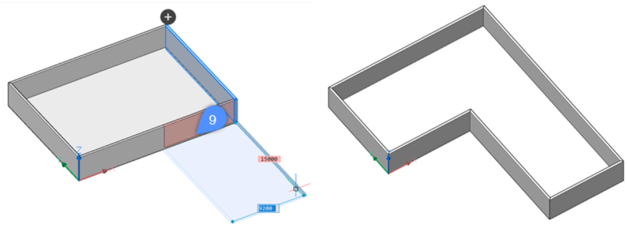
To actually draw new walls, choose the distances from the already existing walls to specify your first point. Move your cursor or type the values in the dynamic dimension fields to do this. Draw walls in the same way as explained in steps 2 and 3.

7 To remove a section of a wall from the existing room, first, align the **Quickdraw** cursor with an existing wall at the inner side of the building. Then click and drag the cursor outside of the existing building.

The outer wall will turn red (9) to indicate the part of the wall that will be removed.

8 Select a second point. The wall highlighted in red will automatically be deleted and an L-shaped room will be created.

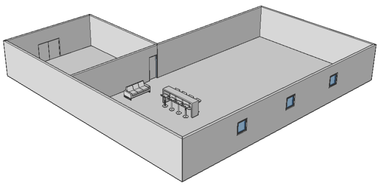
9



Note: You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

22.20.5 Procedure: creating a story and a roof

Start with a similar configuration:



Go through the following steps to create two extra stories, a floating part and a flat roof.

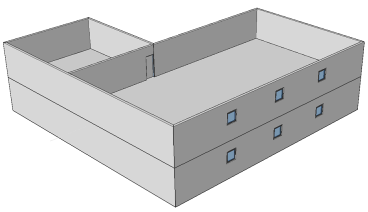
1 When the **Quad** cursor is in the **No Selection** state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab or type BIMQUICKDRAW in the Command line.

2 Click the ( ) icon available in the drawing area, next to the building, to see the options:

- Fully copy the top floor. All features will be copied (windows, doors, interior walls, and furniture, excluding doors on the outer walls of the ground floor).
- Copy the slab and the outer walls of the top floor.
- Finish the building with a flat roof.

3 For the first story, choose the first option (Full copy).

4



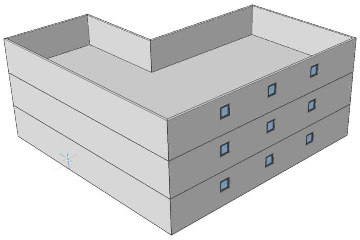
Note: The interior wall and interior door and the windows in the exterior wall are copied. The door in the exterior wall on the ground floor is not copied.

Note: The furniture is not copied in this case, because it is not assigned to the ground floor (see the **Properties** panel). If you want to copy the furniture, use BIMIFY command to assign the furniture to a spatial location. The new furniture will be automatically assigned to the new spatial location.

Note: Walls inherit the spatial location of the slab on which they are created.

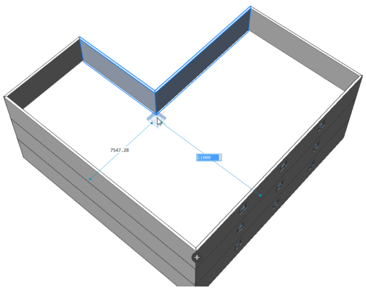
5 Repeat step 2. Choose the second option (Copy the slab and the outer walls) to create the top story.

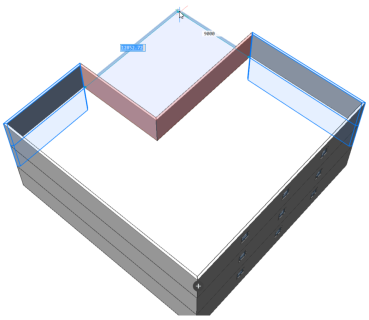
6



7 To make the overhang you should remove parts of the outer walls of the second floor. First, snap the **Quickdraw** cursor in the corner of the two outer walls, inside the building. Then click and drag the cursor outside.

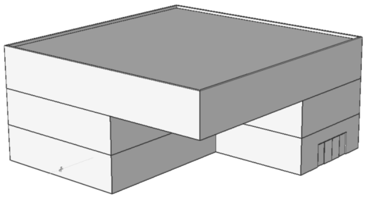
The outer walls will turn red to indicate that the walls will be removed.





8 Use the same method as in step 2, selecting option 3 (Finish the building with a flat roof).

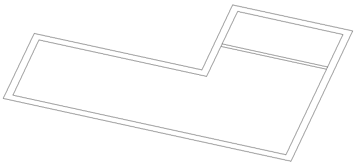
9



22.20.6 Procedure: using Quickdraw starting from a 2D layout

1 Open a 2D floorplan of a rectangular or L-shaped building.

1



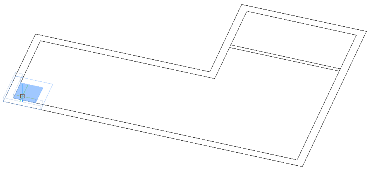
Note: It is also possible to draw a simple 2D floor plan with the (POLY)LINE command to start from. In this case, the default wall thickness of the BIMQUICKDRAW command will be used.

Note: The **Quickdraw** cursor now supports snapping to (double) lines inside XREFs and to entities using **ESNAP**.

2 When the **Quad** cursor is in the **No Selection** state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab, click the **Quickdraw** icon in the ribbon or type BIMQUICKDRAW in the Command line.

3 Hover over one of the outer walls with the cursor.

4

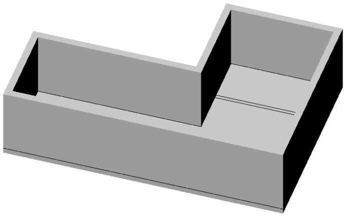


Note: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

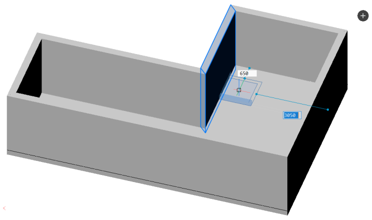
Note: If the lines in the (imported) 2D floorplan are not optimal (for example, not exactly X- or Y-aligned), the **Quickdraw** cursor will not snap to it (as it is perfectly X-Y-aligned). In that case, use the OPTIMIZE command to make sure all 2D lines exactly X- or Y-aligned.

5 Draw the outer walls, using the magnetic snapping feature to snap to the lines of the 2D drawing.

6

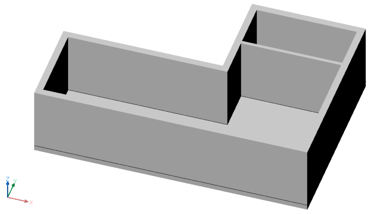


7 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.

8 

Note: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

9 The 2D plan has been transformed into a 3D model.

10 

Note: you can use the same method as in step 2 and 6 of the previous procedure to add floors or finish the building with a flat roof.

22.21 Using drag

22.21.1 Commands

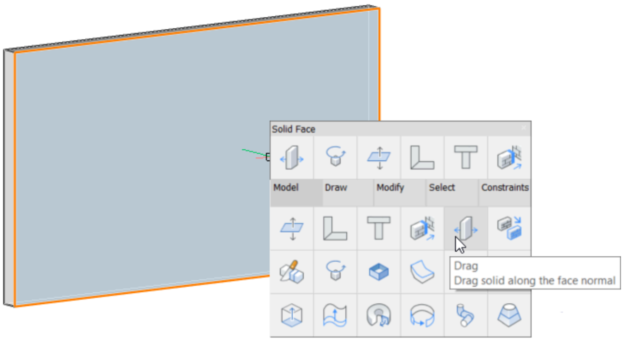
DRAG

22.21.2 About

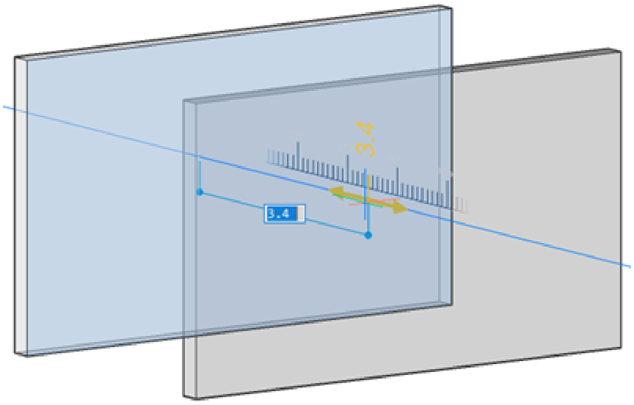
The **Drag** tool allows you to move walls, slabs or linear elements on the highlighted face of a solid. For instance, the **Drag** tool can be used to move linear solids by highlighting/selecting the end or side face of a linear solid. Depending on the modeling operation, the connection between solids can be either preserved while dragging one or more solids at a time or solid (s) can be moved independently by disjoining the connectivity to other solid(s).

22.21.3 Procedure: using drag on a 3D solid

1 Highlight/select any faces of the solid and select **Drag** from the **Quad** > **Model** tab.

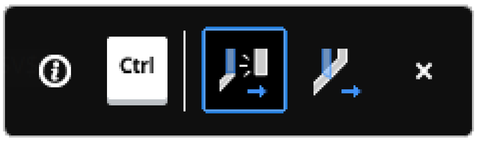


2 Move the cursor to adjust the position of the moving solid in the dynamic dimension field, then click or enter a precise value in the dynamic dimension field and press Enter.



Note: A ruler appears to visually represent the distance over which you will move the entity.

3 The Hotkey Assistant appears at the bottom of the screen showing the available options.

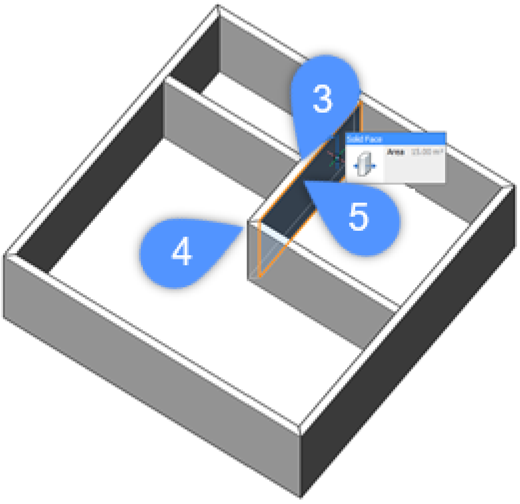


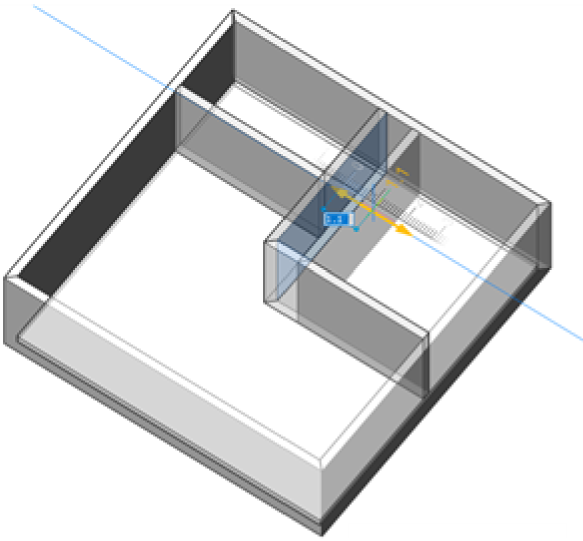
Note: The current mode is indicated by a blue frame.

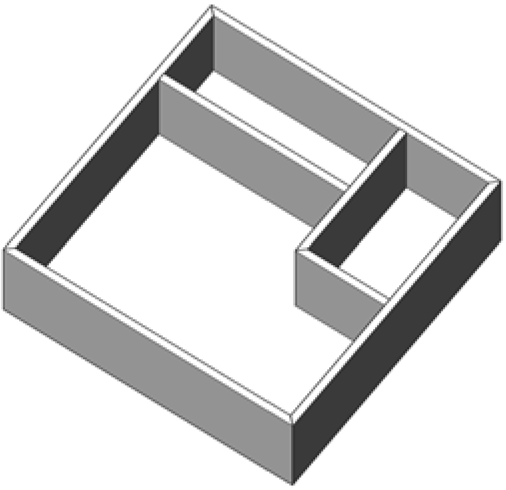
Hit the CTRL-key to toggle the connectivity mode. Alternatively, type D+Enter to disable connectivity or type E+Enter to enable connectivity.

- Select the **Enable connectivity mode** option, to move the solid while preserving the connections between solids.

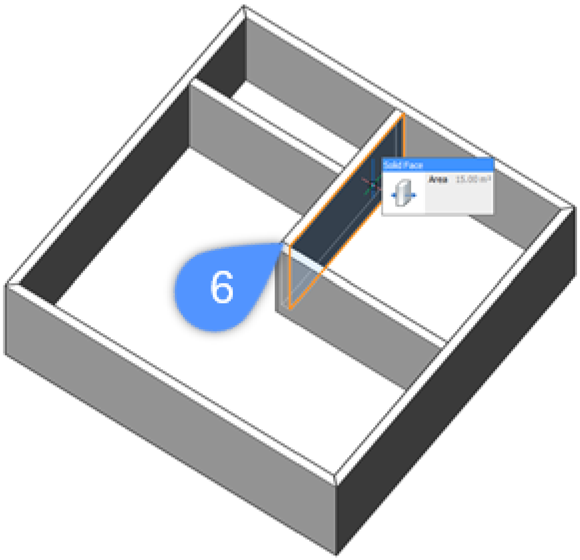
In the following illustration, the solid behavior is shown before and after applying the DRAG in **Enable connectivity mode**. The connectivity between walls, i.e. T- (3) and mitered (4) connections are preserved after dragging an inner wall surface (5).

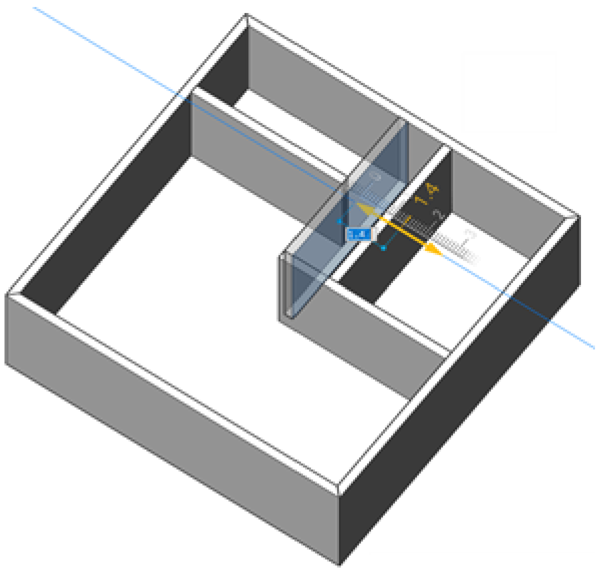


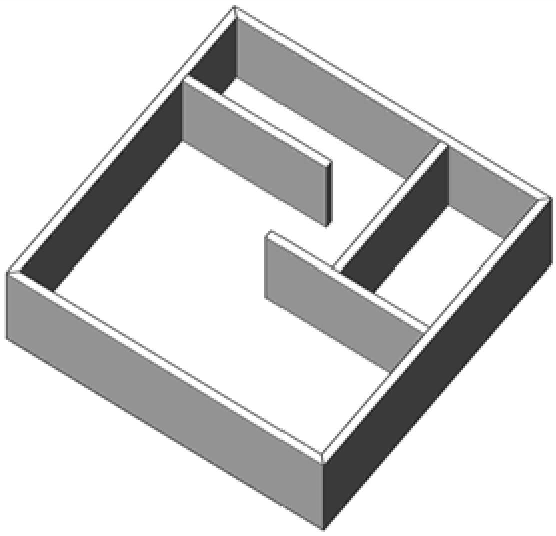




- Select the Disable connectivity mode (default) option to move the solid while breaking the connections between solids.
- In the following illustration, the solid behavior is shown before and after applying the Drag in "Disable connectivity mode". The connectivity between walls, i.e. T- (3) and L- (6) connections are disjointed after dragging an inner wall surface (5).







22.22 Using parametrize

22.22.1 Commands

PARAMETRIZE, PARAMETERSPANELOPEN

22.22.2 About

Parametrizing is a process that automatically adds constraints and parameters to a set of 3D solids, without any manual work. For each unique value, a specific parameter is created. The constraints and parameters are represented by their name and expression in the **Mechanical Browser**. Each parameter can be animated to show what the parameters correspond to in the model. The parametrized geometry becomes controllable in terms of shape and size. When one of the parameters is changed, the geometric model will be updated automatically.

22.22.3 Procedure: parametrizing a component automatically

1 Open the geometric model that represents your component.

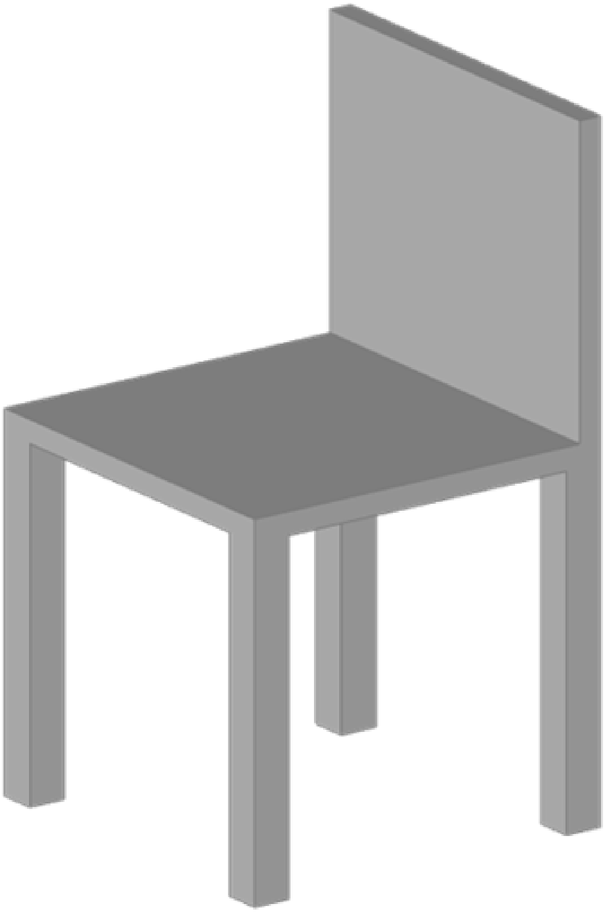
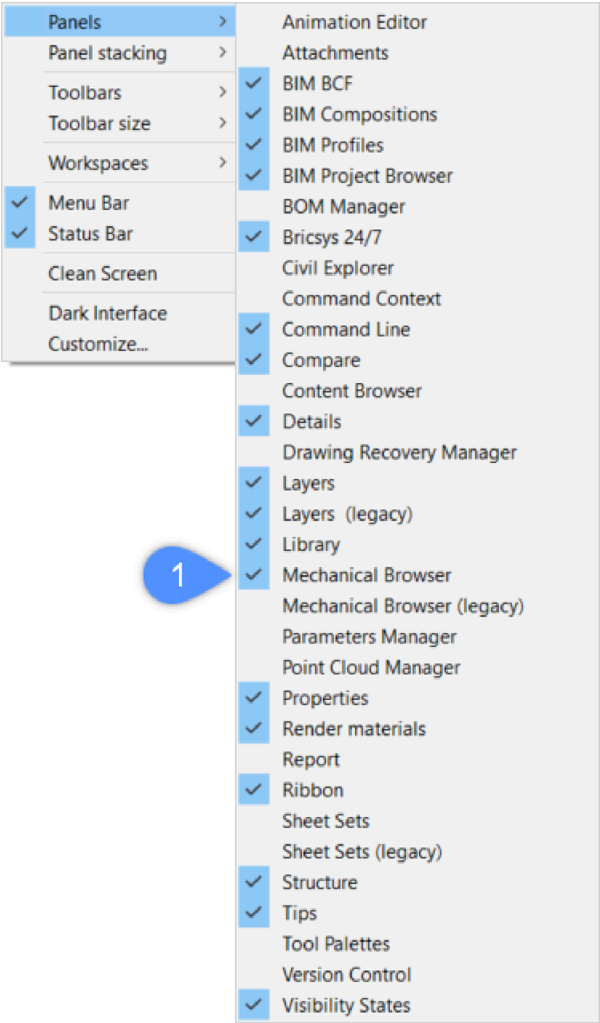


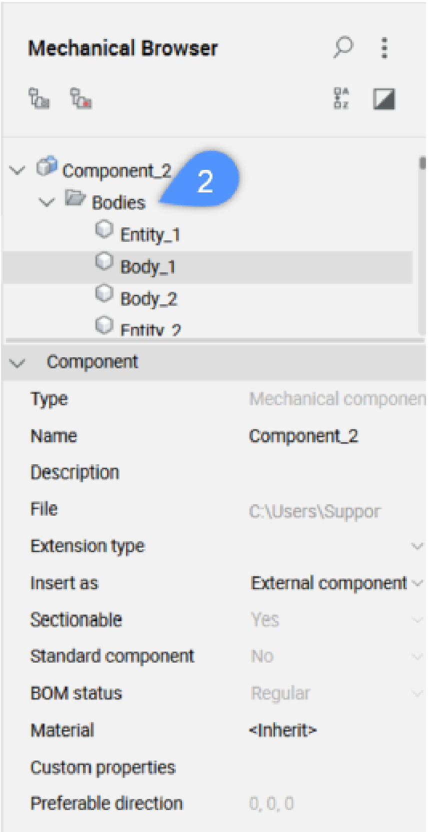
Figure 1, a geometric model of the sample chair.

2 Open the **Mechanical Browser** by clicking its icon in the **Toolbar** panel or type the MECHANICALBROWSEROPEN in the Command line.

Note: If the Browser or Parameters icons are not displayed in the **Toolbar** panel, place the cursor on a toolbar, then right-click to open **Panels** and choose **Mechanical Browser** (1) in the context menu.



If the model does not contain any parameters and constraint it will display only the 3D solids under the **Bodies** (2) section. If the model contains any constraint or parameter, the parameterized tool asks whether you want to reuse the existing ones.



3 Select the component geometry you want to parametrize.

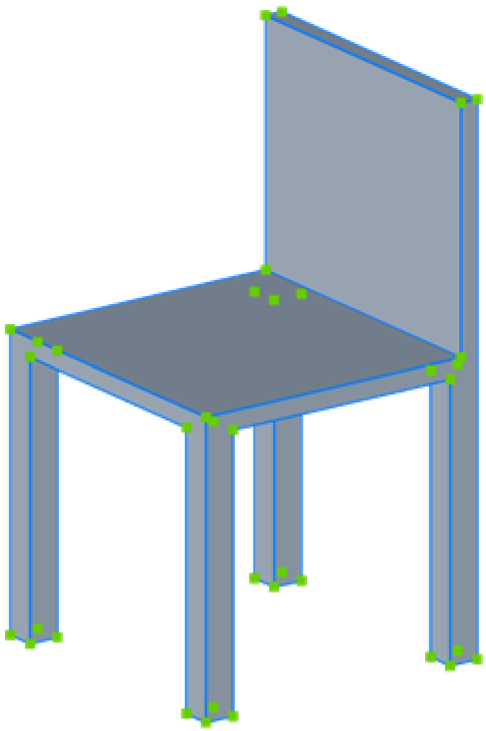


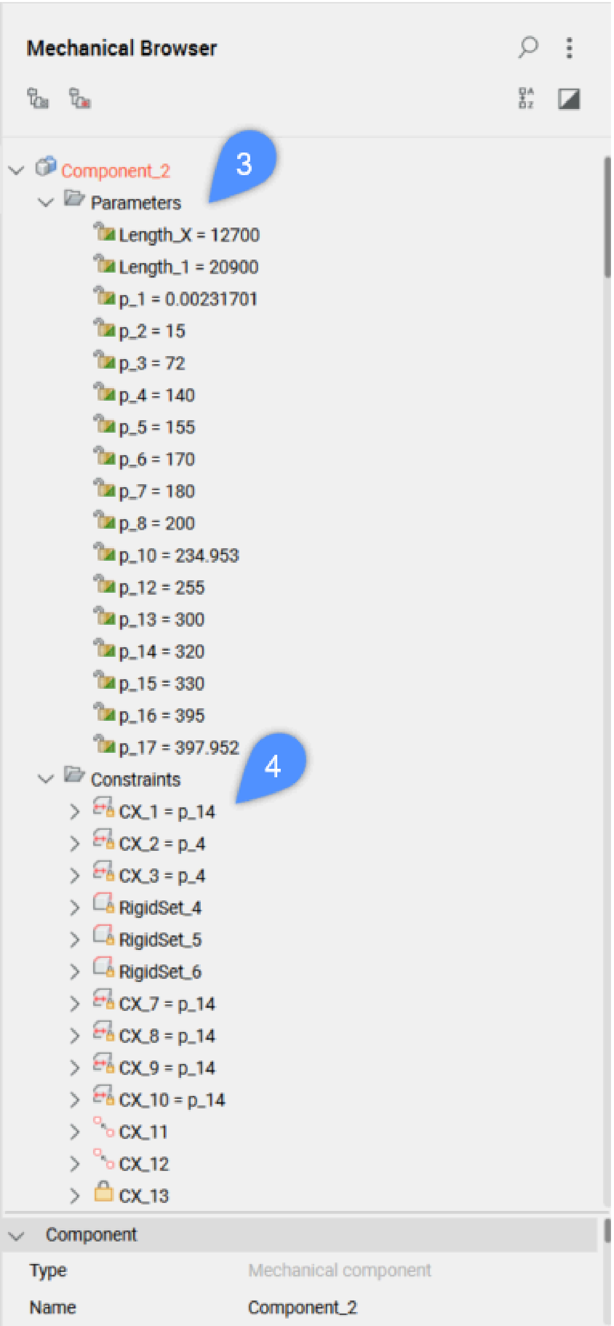
Figure 2, selected object.

4 Launch the **Parametrize** tool in the **Constraints** tab from the Quad.

Or type PARAMETRIZE in the Command line.

A set of parameters and constraints are defined automatically.

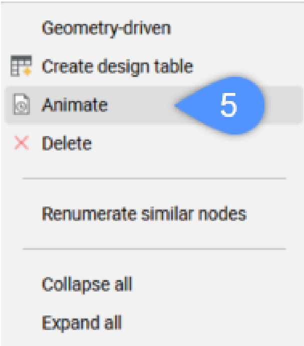
These parameters and constraints are shown in the **Mechanical Browser** (3 and 4).



Note: **Parametrize** creates Distance, Radius, Rigid Sets, Fix, Coincidence, Concentric constraints depending on the geometric model.

Note: Perpendicularity and tangency can be set (if they are applicable) by changing the DMRECOGNIZE system variable in the **Settings** dialog box.

5 To check easily which parameter corresponds to what, right-click one of the parameters, and choose **Animate** (5) in the context menu.



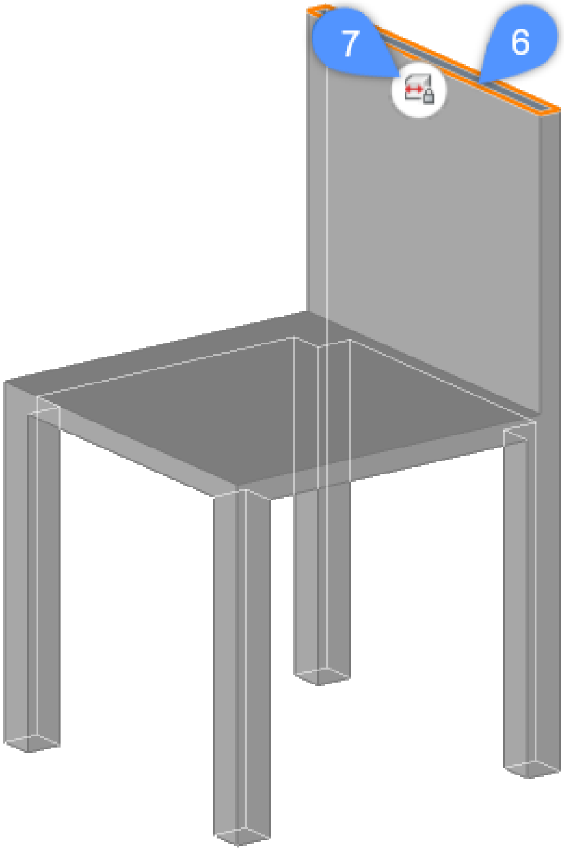
Note: In the attached .dwg file after parametrizing the chair geometry, parameters and constraints will be assigned and shown in the **Mechanical Browser**. Each parameter corresponds to one measurement in the chair geometry. For instance, **Length_X_Y** controls the width and depth, **Length_Z** controls the overall height, **p_1** controls the seat thickness **p_2** controls the leg thickness **ratio_1** controls the ratio between seat height and overall height.

22.22.4 Procedure: changing the parameters and constraints in the parameters manager

Note: After the auto parametrizing, it's possible to change what you do not find clear about the parameters and constraints using the parameters manager.

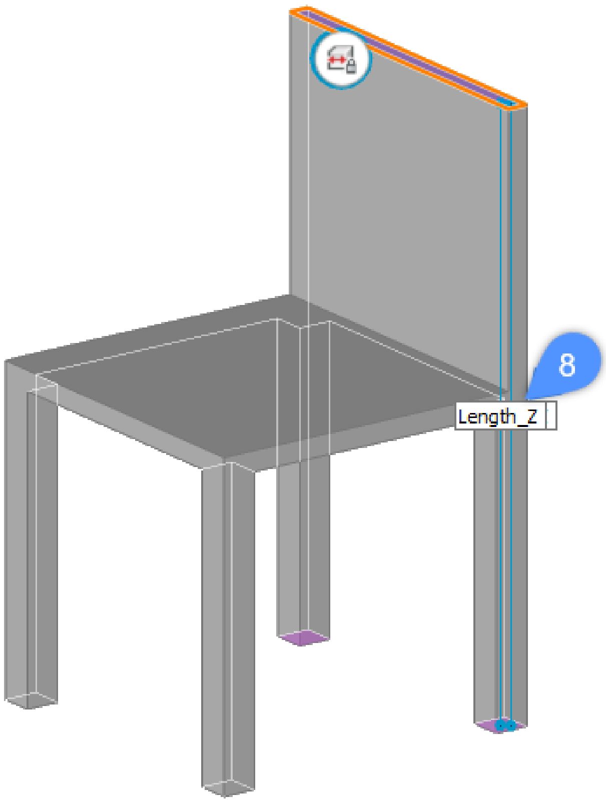
1 Select one of the faces (6) of your component. The constraint type (e.g. distance, fix) regarding the

face is displayed.

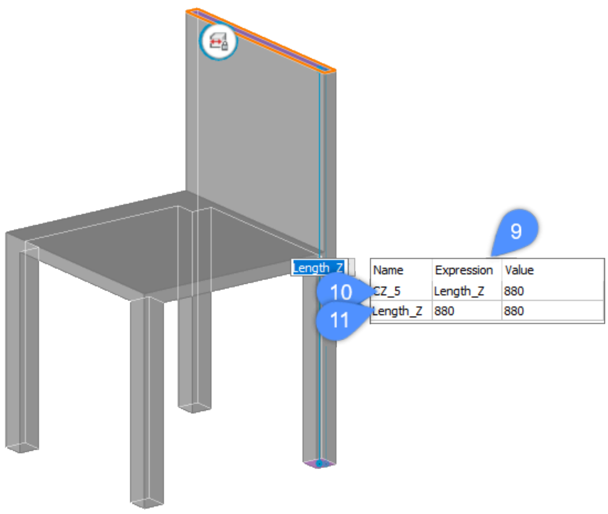


2 Click the distance constraint (7) icon on the element. The editable dimension(s) is shown inside the box.

3 Click the editable dimension box (8).



A parameter manager displays (9). The parameter manager shows the expressions of the distance constraint (10) and the parameter (11) depending on the selected face.



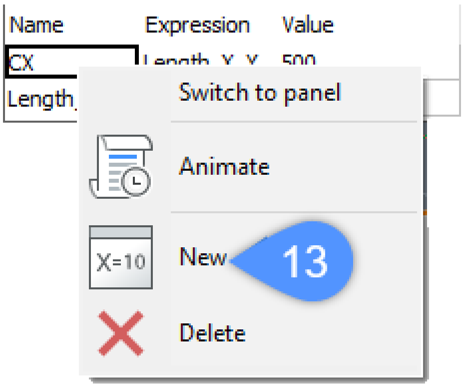
4 Change the parameter name by clicking its field (12) in the parameters manager.

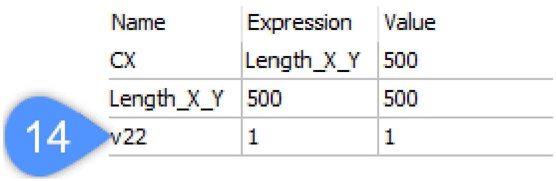
Name	Expression	Value
CZ_5	Height	880
Height	880	880

5 (**Optional**) A new parameter can be defined using the **New** option in the parameter manager's context menu.

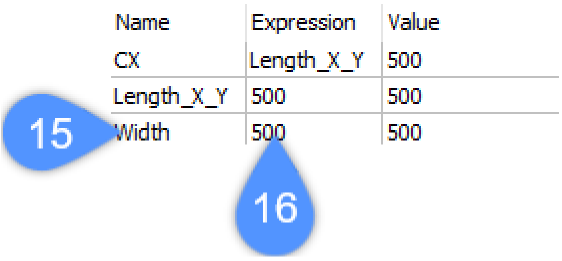
6 To do so, right-click the parameters manager and select **New** (13).

7 A new parameter is added to the list (14).

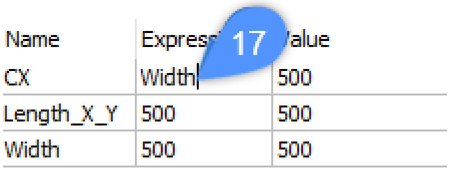




8 Specify a **Name** (15) for the newly created parameter and enter an expression using the **Expression** field (16).



9 To link the newly created parameter with an existing constraint, change the expression field of the constraint with the new parameter name (17).



Note: The above procedure can also be done using the **Mechanical Browser**.

22.23 Using the Library panel

22.23.1 About

The **Library** panel is a quick way to keep your BIM blocks at your fingertips or to access predefined ones. You can add any CAD model to the **Library**. You can also fully customize the name and category to make it easier to find. You can then insert a library block in your drawing by dragging and dropping. Keep your library clean by deleting the items you no longer need.

22.23.2 Content of the Library panel

You can control the content of the **Library** panel in the **Option** menu.

Note: When the COMPONENTSPATH system variable is empty (all paths deleted), the default *C:\ProgramData\Bricsys\Components* path is added automatically when you close and reopen the **Settings** dialog box.

22.23.3 Creating a library block

1 Open or create the drawing which contains the geometry for the new block.

2 Open the **Library** panel.

3 Do one of the following:

- Click the **Home** icon ( ).

  This allows you to select the destination category; or you can create a new category.
- Click the **Category** thumbnail.

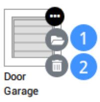
  The new item is created in the selected category or in a subcategory of the selected category.
- Press the **Add** button or launch the CREATELIBRARYBLOCK command.

  The **Add block to library** dialog box displays.
- Select the entities or choose the **Entire drawing** option.
- Type a name for the block.
- (**Optional**) Type a new category name or choose a category from the list.
- (**Optional**) Click **Show Options** and set the block creation options.
- Click the **Create** button.

22.23.4 Managing library blocks

1 In the **Grid view** display, hover over the block icon.

2 Click the **Menu** icon (...).



3 (**Optional**) Click the **File Open** icon (1) to open the block file.

Edit the file, then save the changes.

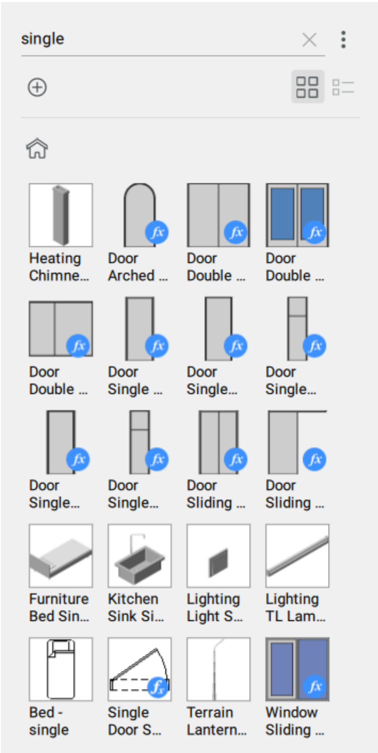
4 (**Optional**) Click the **Delete** icon (2) to delete the block file.

Click **Yes** to confirm.

22.23.5 Searching the block libraries

Type a search string in the search field.

Only matching library blocks in the current category display.



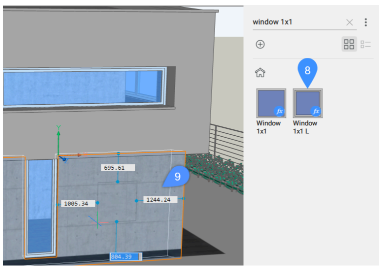
22.23.6 Inserting a BIM window into your 3D model

1 Click the **Windows** category tile in the **Library** panel.

2 Click a window (8).

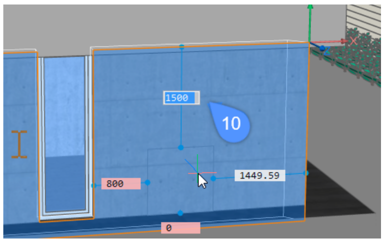
3 Drag and drop the window onto the wall (9).

4



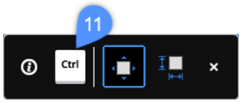
5 If Dynamic dimensions (**DYN**) is switched on, set the window location by entering the dimensions into the DYN fields (9). Press the Tab key to cycle through the DYN fields (10). A DYN field turns red to

indicate the distance is locked.

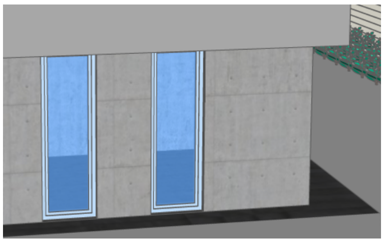


6 To edit the dimensions of the window press Ctrl (11) once and then use the Tab-key to cycle through the DYN fields (12).

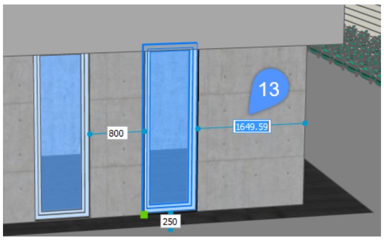
7



8 Press Enter. The window is created. This same technique can be used for most of the blocks in the **Library**.



9 (**Optional**) Once you have a window, you can adjust the position by selecting the window and then clicking on the dimension lines and editing the dimensions (13).



10 (**Optional**) To adjust the dimensions of the window after the window has been created, go to the **Properties** panel. Scroll down to the **Parameters** section and change the parameters as required. All windows and doors in the block library are parametric.

Parameters	
Name	Window_1x1_L:9
File	
Physical material	
W	800 mm
H	250 cm
Rebate	0 mm
PlacementDepth	90 mm
FrameThickness	60 mm
FrameDepth	50 mm
LeafThickness	45 mm
LeafDepth	60 mm
PaneOffset	20 mm
LeafOffset	20 mm

Note: A red preview appears while a solid with a subtractor, such as a window, is copied, moved, or manipulated, ensuring easy placement on the affected solid.

22.24 Working with parameters and constraints

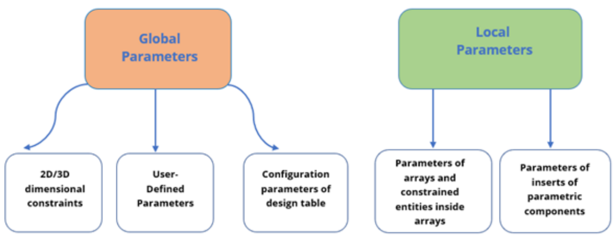
22.24.1 Parametric modeling

In BricsCAD, parametric modeling is done through the use of geometric and dimensional constraints. A rich toolset controls the 2D and 3D geometric model with a set of parameters. Each parameter will be displayed with a value in the **Mechanical Browser**. The value from the different parameters can be linked together via expressions. When one of the parameter values is changed, the geometric model is updated automatically. The toolset allows BricsCAD users to add parametric behavior to any geometry and easily explore design intent.

Note: In BricsCAD, there are no parent-child dependencies between geometric elements. For instance, if you change a sketch used to create an extruded 3D solid, the solid is not changed accordingly. Any kind of dependencies can be created using the parameters and expressions.

There are two types of parameters:

- Local parameters are attached to a particular entity.
- Global parameters are not attached to a particular entity.



22.24.2 Working with constraints

If you create some components (e.g. windows, doors,...) in your BIM model, you can parametrize them by applying 3D constraints. Defining constraints allows you to control the shape and size of the elements. Together with constraints, parameters determine the positions of entities through an expression.

There are two types of 3D constraints in BricsCAD: one specifies the size of the entities, the other locates their positions.

22.24.3 Geometric constraints

Geometric constraints allow you to control the position of 3D entities with respect to each other.

Toolbar: **Parametric** > **3D constraints**

Quad: **Constraints**

The following table shows the 3D geometric constraints.

Fix	Keeps solids, edges or faces of solids in-place in the drawing.
Coincident	Applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
Concentric	Keeps two cylindrical, spherical or conical surfaces centered.
Parallel	Keeps the two faces of a solid or of different solids parallel.
Perpendicular	Keeps the two faces of a solid or of different solids perpendicular.
Tangent	Keeps a face and a curved surface of different solids tangent.
Rigid set	Makes a set of entities or sub-entities a rigid body.

22.24.4 Dimensional constraints

Dimensional constraints allow you to control the sizes of 3D entities in drawings and the distances between them.

Toolbar: **Parametric** > **3D constraints**

Quad: **Constraints**

Ribbon: **Parametric Tab** > **3D constraints**

The following table shows the dimensional constraints.

Distance	Creates a distance between two sub-entities.
Radius	Creates a radius to cylindrical surfaces or circular edges.
Angle	Creates an angle between two faces of a solid or of different solids.

22.24.5 Setting an expression to a parameter

In BricsCAD, you can set an expression to any parameter, be it global or local. As an example, you can create a simple expression that only contains a number or a name of a global parameter and apply it to the related geometry in the drawing. You can also use more complex formulas that include the use of standard operators and functions.

Note:

- Local parameters cannot be referenced by names in expressions.
- The constants Pi=3.14... and e=2.72... can be used in expressions. The constants names are not allowed to be used as parameter name and name of 3D constraint.

The following table shows the operators that can be used in expressions.

Addition (+)	Produces the sum of numeric values.
Subtraction or Negative (-)	Subtracts the two numeric values.
Multiplication (*)	Multiplies the numeric values.
Division (/)	Divides two numeric values.
Exponentiation (^)	Calculates the exponential value of the given numbers.
Modulo or Remainder operator (%)	Gives the remainder after the division of one numeric value to another e.g. The expression "5 % 2" would evaluate to 1, because 5 divided by 2 leaves a quotient of 2 and a remainder of 1.

The following table shows the functions and the syntax which can be used in expressions.

Sine	sin(expression)

Cosine	cos(expression)
Tangent	tan(expression)
Arc sine	asin(expression)
Arc cosine	acos(expression)
Arc tangent	atan(expression)
Hyperbolic sine	sinh(expression)
Hyperbolic cosine	cosh(expression)
Hyperbolic tangent	tanh(expression)
Arc hyperbolic sine	asinh(expression)
Arc hyperbolic cosine	acosh(expression)
Arc hyperbolic tangent	atanh(expression)
Square root	sqrt(expression)
Signum function (-1,0,1)	sign(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Round down	floor(expression)
Round up	ceil(expression)
Absolute value	abs(expression)
Largest element in array	max(expression1;expression2)*
Smallest element in array	min(expression1;expression2)*
Degrees to radians	d2r(expression)

Radians to degrees	r2d(expression)
Logarithm, base e	ln(expression)
Logarithm, base 10	log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Power function	pow(expression1;expression2)*
Random decimal between 0 and 1	Random

* Use the list separator character as defined on your system: , (comma) or ; (semicolon).

22.24.6 Parametric blocks

Parametric Blocks refer to blocks that have parameters that determine the size and shape of the 3D geometry. Once the parametric block is created, it can be inserted as a block reference in the drawing. The parameters of your block can be changed after you insert it in your model. Thus, no need to modify the parametric block file individually. The parametric blocks also enable you to use the same block in different sizes and in different shapes in your model.

22.24.7 Parametrizing a custom component

The window that was created in the previous lesson is used to demonstrate how the parametric design works. This window consists of three solids: subtractor, fixed frame, and glass pane.

Step 1: Before starting to add constraints:

1 Open the **Mechanical Browser** with the MECHANICALBROWSEROPEN command, to control and manage the values of 3D constraints and parameters.

Note: The **Mechanical Browser** allows to navigate through all the constraints and parameters in the drawing and to edit dimensional constraints.

2 Make sure that the **Enable selection of 3D solid faces** () is toggled on in the Selection Modes To easily follow the steps, make sure Boundary Detection is toggled off.

Step 2: Applying fix constraints

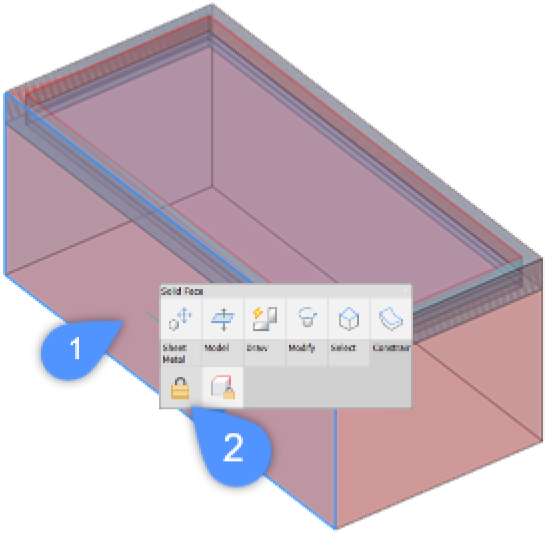
Command: DMFIX3D

1 Turn on the **BC_SUBTRACT** layer.

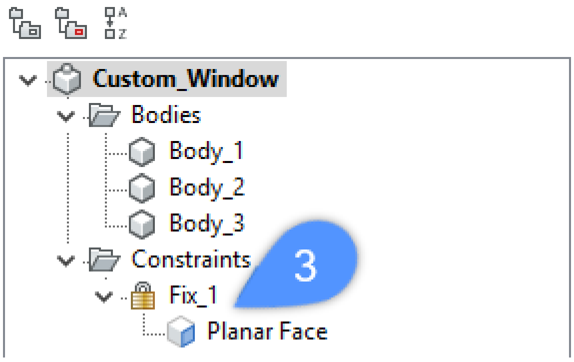
2 Select the entities (1) you want to add a fixation to.

3 (**Optional**) Press the TAB key to select obscured geometry.

4 Choose **Add Fixation** (2) in the 3D constraints command group in the Quad.



The fixation appears in **Mechanical Browser** as **Fix_1**.



Step 3: Applying rigid constraints

Command: DMRIGIDSET3D

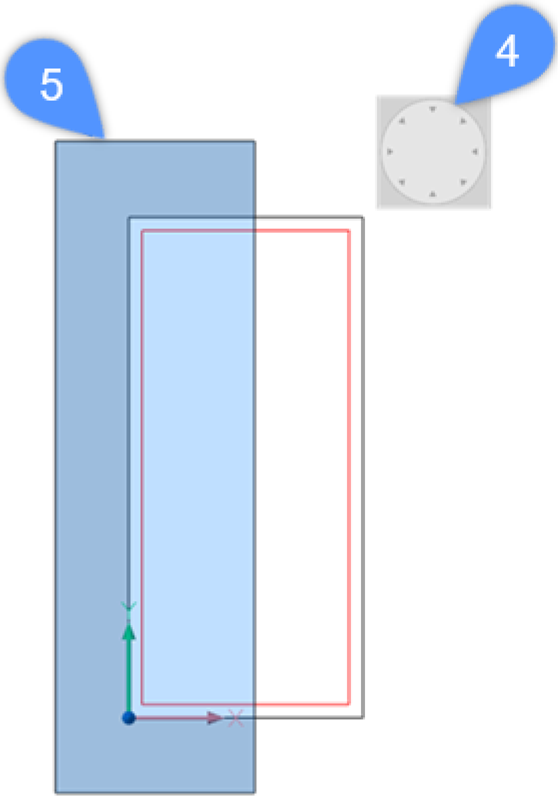
1 Select the entities to apply the **Rigid Set** constraints.

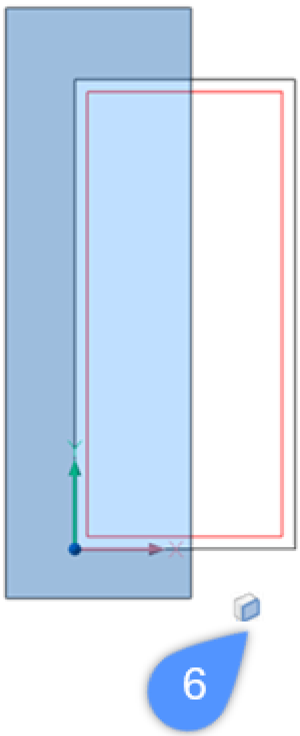
The entities remain in the same position with respect to each other when they are in a Rigid Set.

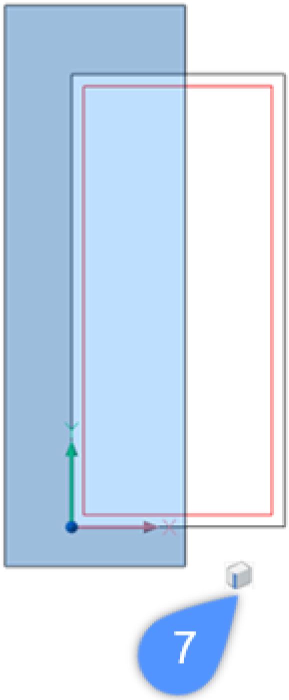
To select more than one sub-entity of the same type at a time, take the following steps:

- Use **Look From Widget** (4), by default it is at the top right of your screen, to change the view from 3D to TOP.
- Select the entities using the selection box (5). While drawing a selection box, press CTRL key to change the type of sub-entity selection. Every time you press CTRL key, the type of the sub-entity selection will change. The type of the sub-entity can be seen under your mouse cursor, for instance, faces (6) or edges (7).

Note: By default, the selection boxes see and select solids.


5
4


6



2 Choose **Add Rigid Set** () in the 3D constraints command group in the Quad.

3 Repeat the same process for the other sides: top, bottom, and right side.

Step 4: Applying distance constraints

Command: DMDISTANCE3D

Dimensions of the window and the thickness of the glass pane.

1 Turn off the **BIM_Subtract** layer.

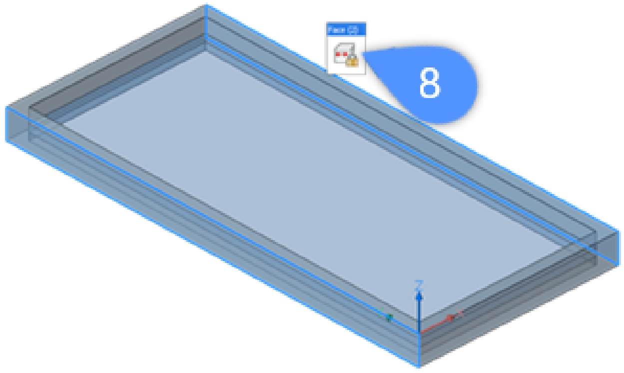
2 Select the first face.

3 The face highlights.

4 Hover over the second face.

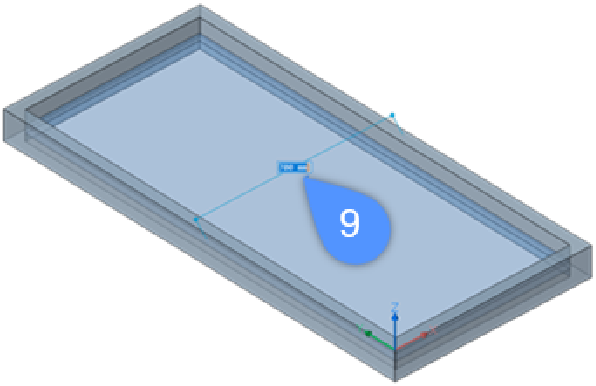
5 Hit the TAB key to highlight an obscured face.

6 When the second face highlights, choose **Add Distance** (8) in the 3D Constraints command group in the Quad.



You are prompted: Specify distance value or [Geometry-driven] <xxx.xx>:

7 Do one of the following:
   - Press Enter key or right-click to accept the current value.
   - The current value displays in the dynamic dimension field (9) when the **Dynamic Dimensions** (DYN) is active.



   - Type a value in the Command line.

8 The distance value between the two surfaces appears in the **Mechanical Browser**.

   Note: To remove any constraint in your model, select the constraint in the **Mechanical Browser**, e.g. fix, rigid set..., and press the Delete key or right-click and select the **Delete** option.

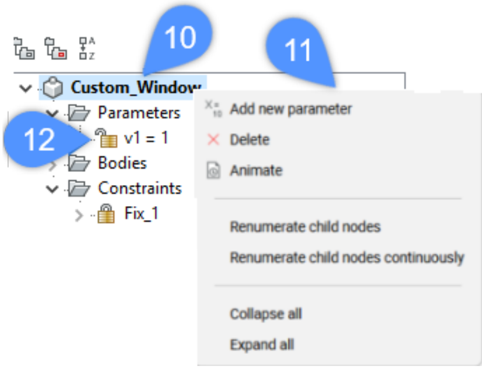
Step 5: Applying parameters

We will create a parameter that controls the width of the window.

1 Right-click the file name (10) in the **Mechanical Browser**.

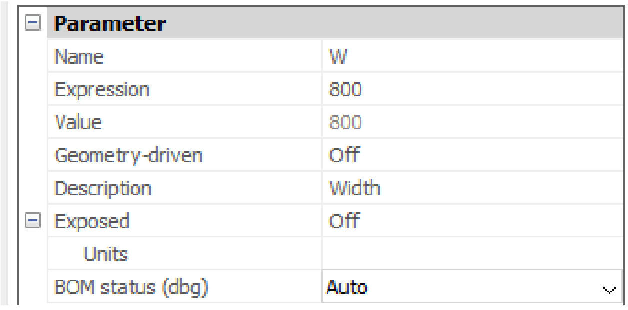
2 Select **Add New Parameter** (11) in the context menu.

   The parameter settings grid is created and the parameter is added in **Mechanical Browser** as **v1 = 1** (12).



3 Select the parameter, then edit its properties in the settings grid.

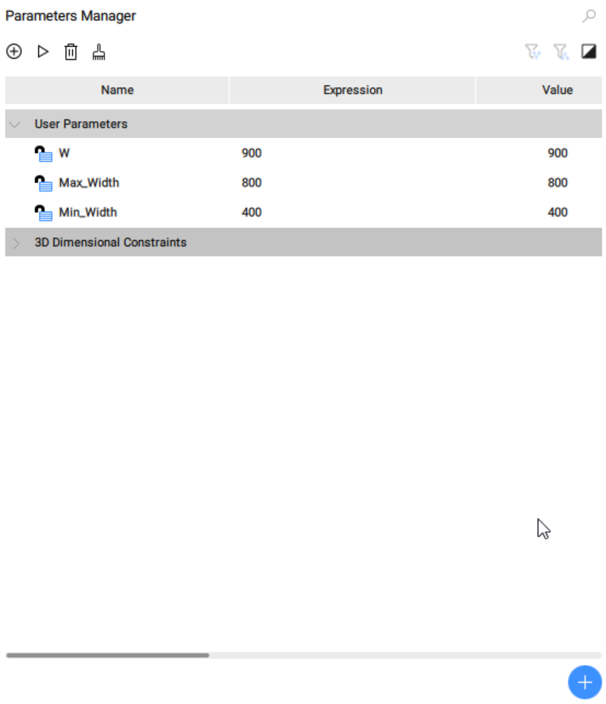
4 The following illustration shows the properties of a **Width** parameter of the custom window. The table below demonstrates each property.



Parameter	
Name	W
Expression	800
Value	800
Geometry-driven	Off
Description	Width
Exposed	Off
Units	
BOM status (dbg)	Auto

Name	Type a parameter name.
Expression	Type a value or formula.
Value	Shows the current value of the parameter.
Geometry-driven	If On, makes the parameter geometry-driven (disables the **Expression** field).
Description	Define an optional description of the parameter.
Exposed	Controls whether the parameter is available in the properties panel when the component is inserted into a model.
Units	Specifies whether the parameter is linear, areal or volumetric.
BOM status (dbg)	Specifies if the parameter will be included into BOM.

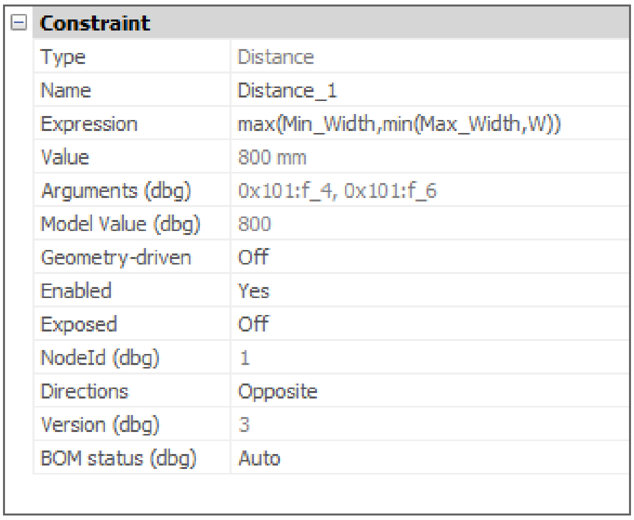
5 Similarly, also by using **Parameters Manager** panel, we can add parameters which will create upper and lower limits for window width by combining max(expression), and min(expression) expressions.



Parameters Manager

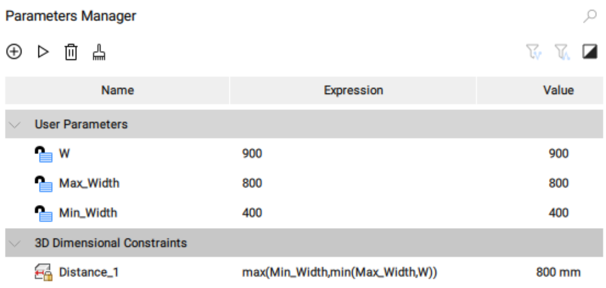
Name	Expression	Value
User Parameters		
W	900	900
Max_Width	800	800
Min_Width	400	400
3D Dimensional Constraints		

6 To link the parameters you just created with the dimensional constraint, select the distance constraint in the **Mechanical Browser** panel and use the parameters to formulate an expression in the **Expression** field in the settings grid of the constraint.



Constraint	
Type	Distance
Name	Distance_1
Expression	max(Min_Width,min(Max_Width,W))
Value	800 mm
Arguments (dbg)	0x101:f_4, 0x101:f_6
Model Value (dbg)	800
Geometry-driven	Off
Enabled	Yes
Exposed	Off
NodeId (dbg)	1
Directions	Opposite
Version (dbg)	3
BOM status (dbg)	Auto

For example, when W=900mm, the Distance_1 value will be 800mm:



7 **(Optional)** Change the expression value of the parameter and press Enter. The assigned distance constraint is changed accordingly.

*In your drawing, you can design as many constraints as possible. Note that the software will keep you away from applying more constraints than are necessary.

23. Building data

23.1 Analytical model

23.1.1 About the BIM analytical model

An Analytical Model a simplified wireframe model that only contains the load-carrying building elements. The linear building elements like columns, beams and members are represented by lines. The planar building elements like walls and slabs are not yet supported in the analytical model.

In BricsCAD, the BIMANALYTICALMODEL command allows you to generate an analytical model from a full 3D model of a structure. The command starts with an automatically generated proposal but allows you to further adapt the model. The automatic proposal will try to minimize rigid links (eccentricities) as much as possible by shifting, extending and even rotating some axis lines. The number of such changes can be controlled by the deviation settings.

You can modify the generated proposal afterward by using the options when you select an element (nodes or axes). To propagate the effect of a local change further you can use recalculate. This will launch a global solution that again tries to minimize rigid links but will respect the changes you made.

After editing the model, you can export it to an IFC file and a CIS/2 file. This analytical model can then be used for further structural analysis after import in a dedicated analysis software package.

It's important to emphasize that the input model has to be fully classified since the BIMANALYTICALMODEL command assumes that all the load-carrying elements are indicated in the building element properties.

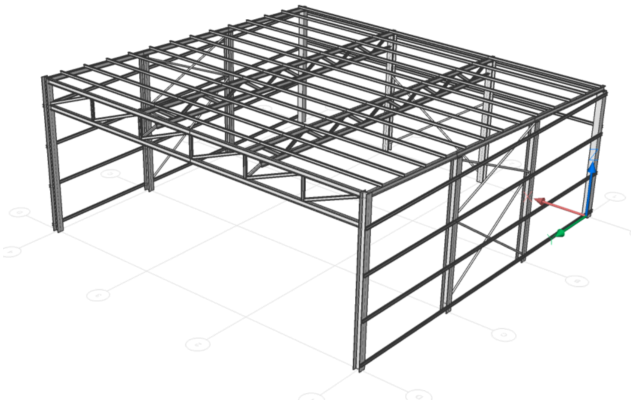
In the procedures, the analytical model is explained with an example model. You can find this model in the appendix.

23.1.2 Creating an analytical model with strict deviation settings

1 First, you have to classify your BIM model. This is necessary for using the BIMANALYTICALMODEL command.

1 Use the BIMIFY command on your model or classify elements manually.

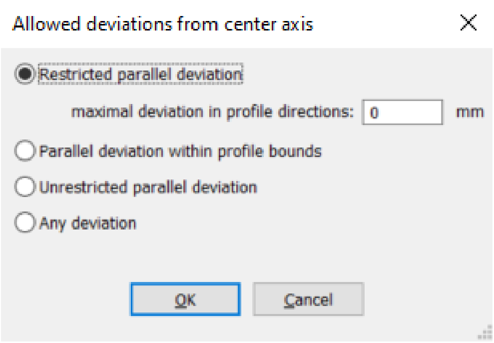
2 Launch the BIMIFY command. The **Bimify** command panel opens and allows you to analyze the model and run an automatic classification and a spatial location assignment for the entire model or for the selected entities.



3 Launch the BIMANALYTICALMODEL command in the Command line.

4 A pop-up window opens.

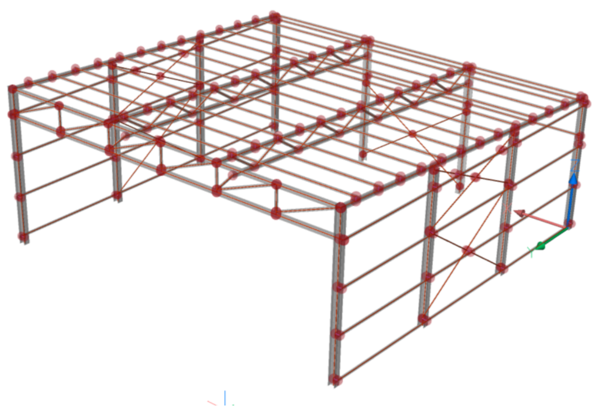
5



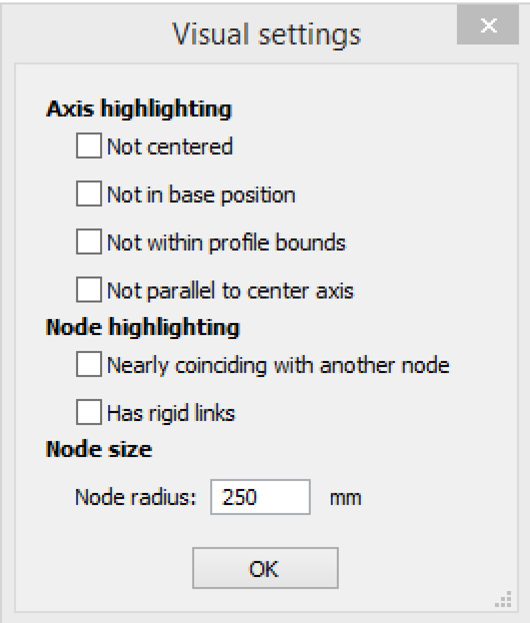
This window allows you to set the level of freedom used when generating an analytical model. When hovering over one of the options, a tooltip shows you what type of restrictions are imposed for that option. In this case, choose the **Restricted parallel deviation** with a maximal deviation in profile directions of 0 mm. No axis is allowed to deviate from its central position in the profile.

6 An interactive preview of the analytical model is constructed based on the chosen global deviation setting.

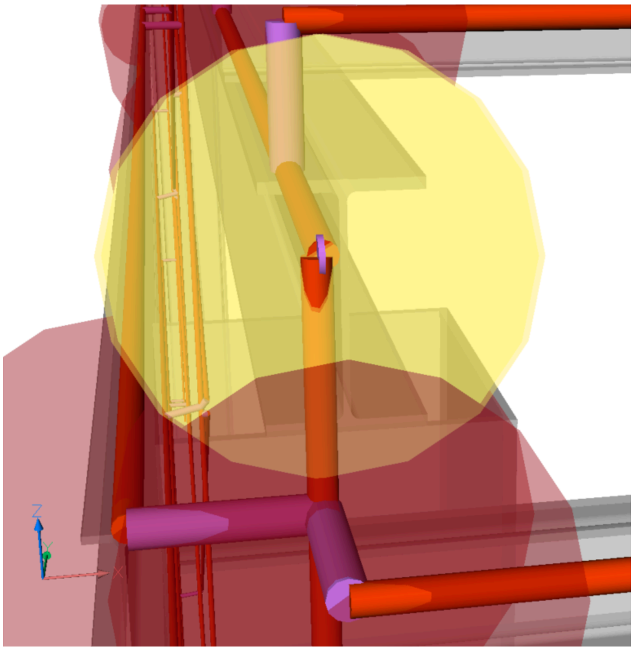
7



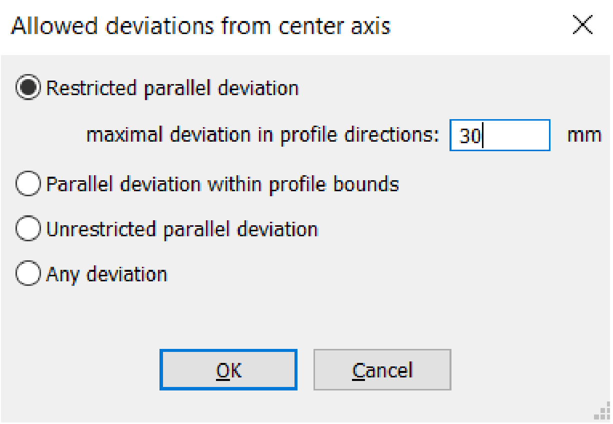
Also, the visual settings window appears. In this window, you can tick and untick different features of axes and nodes to highlight (in yellow) the axes and nodes which correspond to those features. This allows you to explore the current properties of the axis and nodes of the analytical model. Based on what you find you can determine if any adjustments need to be made. You can also change the size of the disks representing the nodes. This is useful in combination with the node highlighting, which can be used for example to find nearly coinciding nodes.



8 Select the **Nearly coinciding with another node** checkbox in the **Visual settings** window. You will see that the center axes at these nodes do not align perfectly. As a result, small rigid links (eccentricities) had to be introduced. This is because of the previous setting in step 4. These small rigid links are displayed in purple.

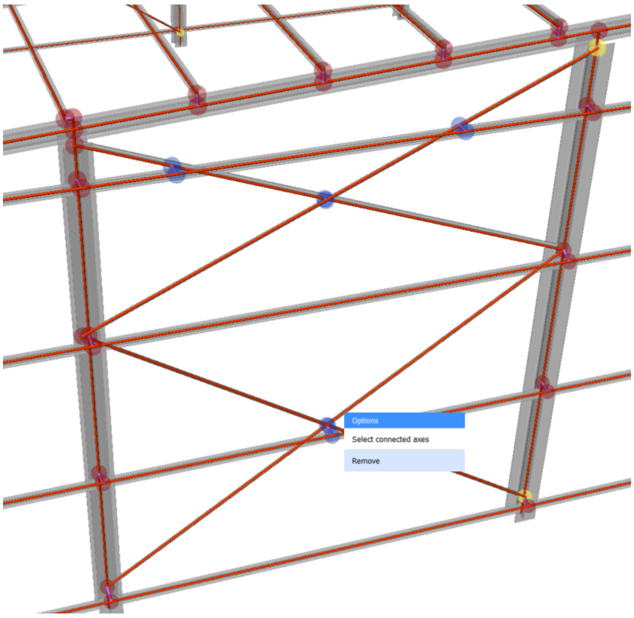


To avoid this, restart the command with a slightly more flexible setting. In the **Allowed deviations from center axis** window, choose again for **Restricted parallel deviation**, but this time with a maximal deviation in profile directions of 30 mm.

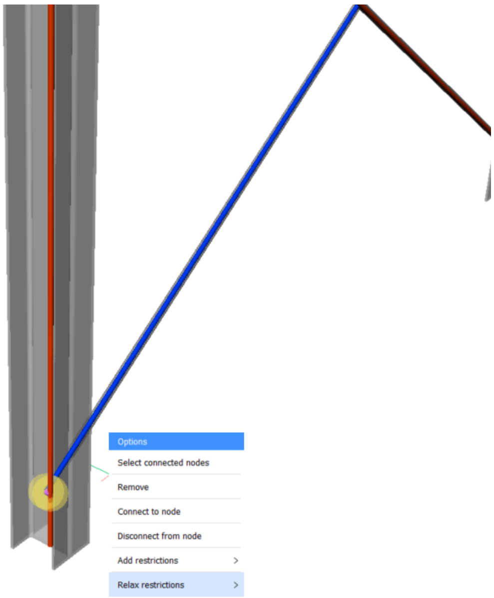


9 For the detection of nodes in the preview of the analytical model, the proximity of structural elements is used. Therefore, it is possible that nodes arise in the model where tie rods cross. You can easily remove these nodes by clicking on the node and selecting **Remove**.

Note: You can select several nodes at once using box selection.

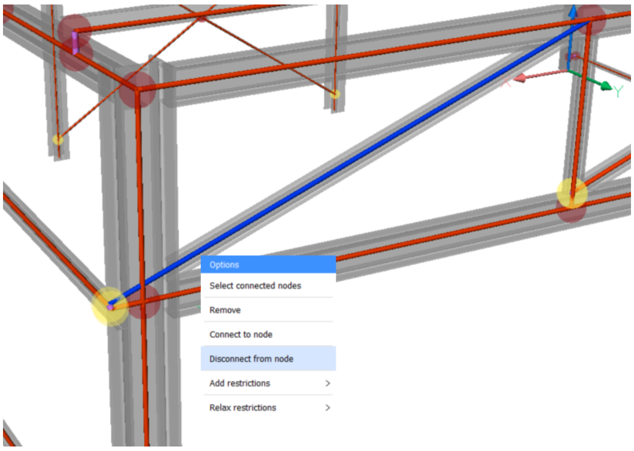


10 If you take a look at the remaining coinciding nodes, you see that the tie rods are not aligned perfectly with the center lines of the columns and small rigid links are formed. You can remove these rigid links by relaxing the position of the axes of the tie rods to a non-parallel position. You can do this by clicking on the tie rod, click **Relax restriction** and click **Non parallel**. The extra freedom will be used to remove the rigid links by moving the axes of the tie rods a bit.



11 The only coinciding nodes that are left are at the front truss of the hangar structure. If you take a look at the side of the front truss, you see that a tilted member is connected to the horizontal beam rather than the column.

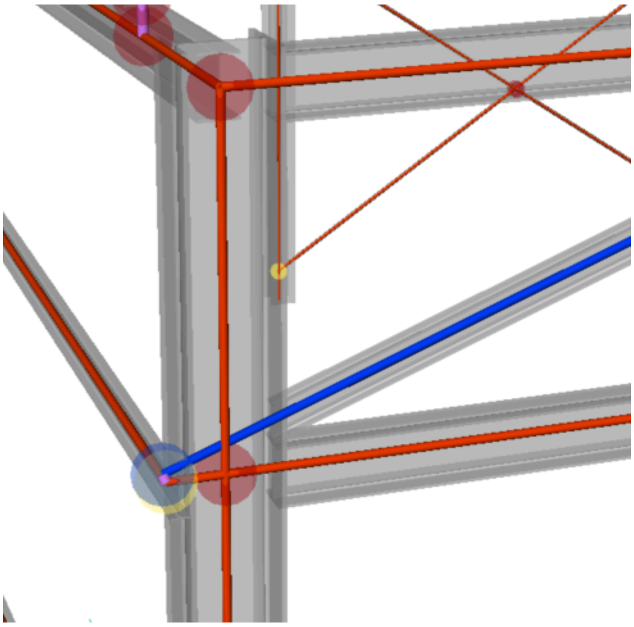
12 You can disconnect the tilted member by clicking on it and choose **Disconnect from node**.



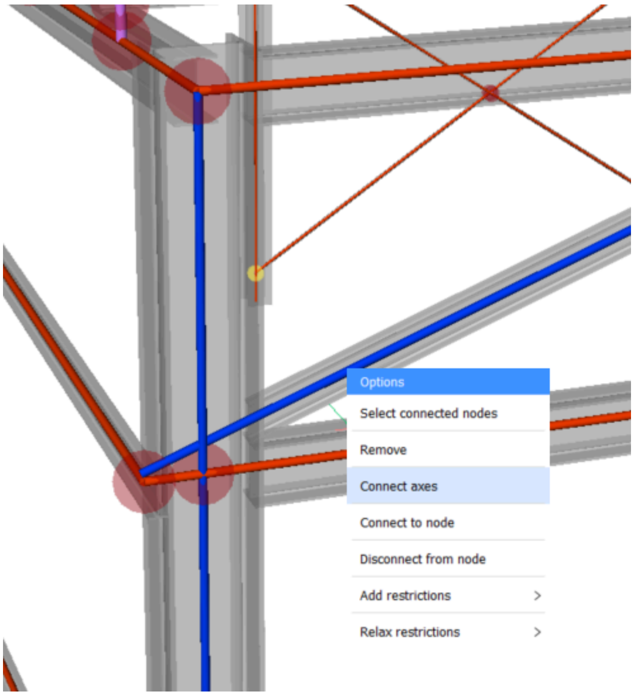
You are prompted: Select node to disconnect from axes [Ok]

Select the node you want to disconnect from.

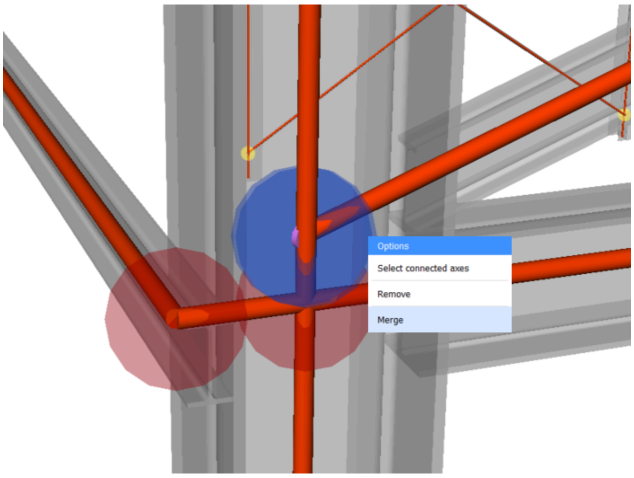
Type **O** to choose Ok.



Now select the two axes that you want to connect and click **Connect axes**.

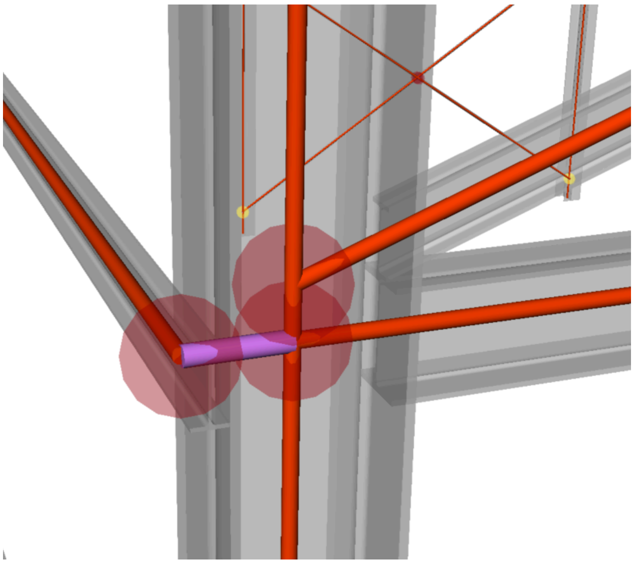


With the global deviation setting, you chose this can only be done by introducing a small rigid link again. You can remove this by merging the two nodes. Select both nodes and click **Merge**.



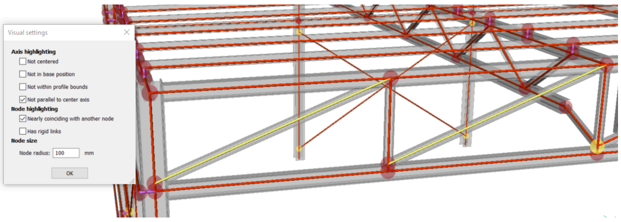
13 You see that the side member also is attached to the front beam instead of the column. Repeat step 9 for the side member.

14



15 You can remove the last coinciding nodes in the front truss by merging the nodes. This will shift the axes of the tilted members and introduces rigid links on the other end.

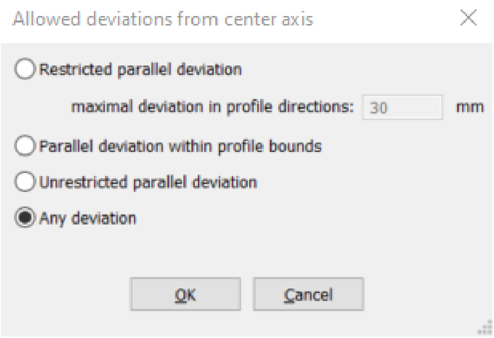
16 ***Note:*** If you take a look at the axis **Not parallel to center axis** by using the **Visual settings** window, you will see that the tilted member in the front truss has been allowed to be non-parallel to the center axis. So, you could also remove the small rigid links by relaxing the tilted member to a non-parallel position like as we did for the tie rods.



23.1.3 Creating an analytical model with the least restrictive deviation settings

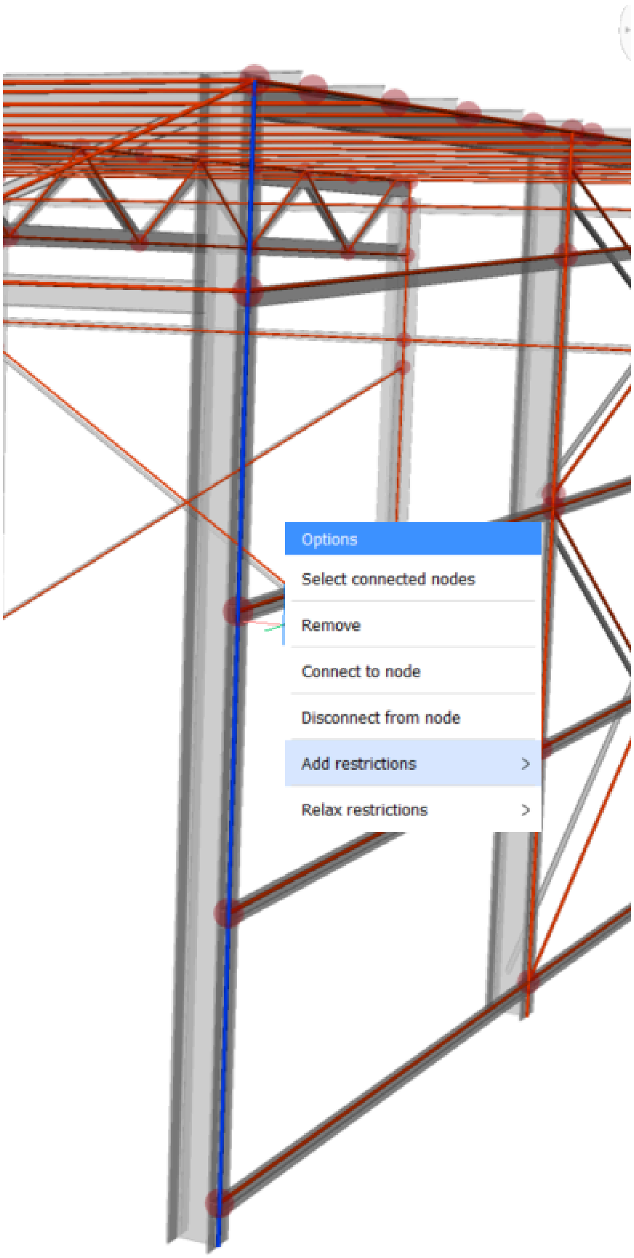
1 Repeat the steps from the previous procedure until step 4.

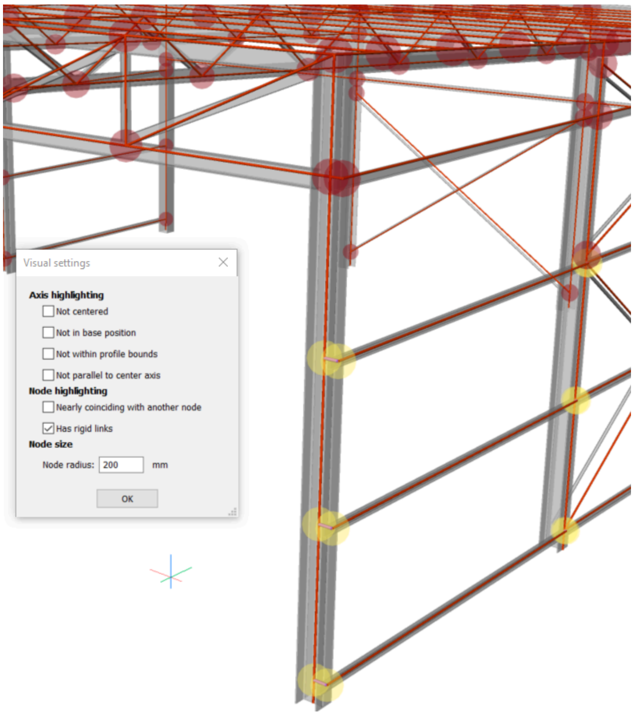
2 Choose **Any deviation** in the **Allowed deviations from center axis**. This allows all kinds of freedom to avoid any rigid links. In the preview of the analytical model, you will see that some axes have been tilted. For example, the small members in the trusses are not centered and the column axis is moved towards the profile bounds to find connections with smaller side members there.



3 Remove the unwanted tie rod nodes as in step 7 of the procedure with a strict deviation setting.

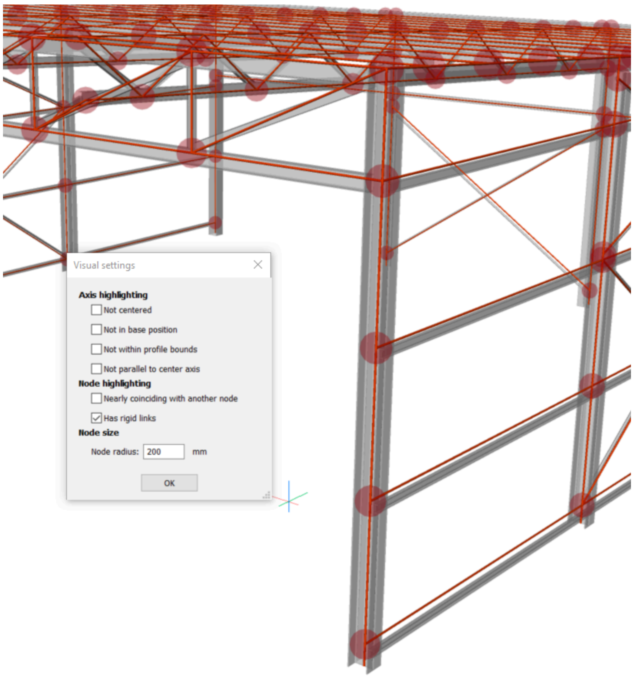
4 In the interactive preview of the analytical model, the column axes were moved towards the profile bounds. In this step, you will center the axis of the columns. You can do this by clicking on the axis of the column, click **Add restrictions** and click **To center**. This introduces rigid links very locally.


Options
Select connected nodes
Remove
Connect to node
Disconnect from node
Add restrictions
Relax restrictions



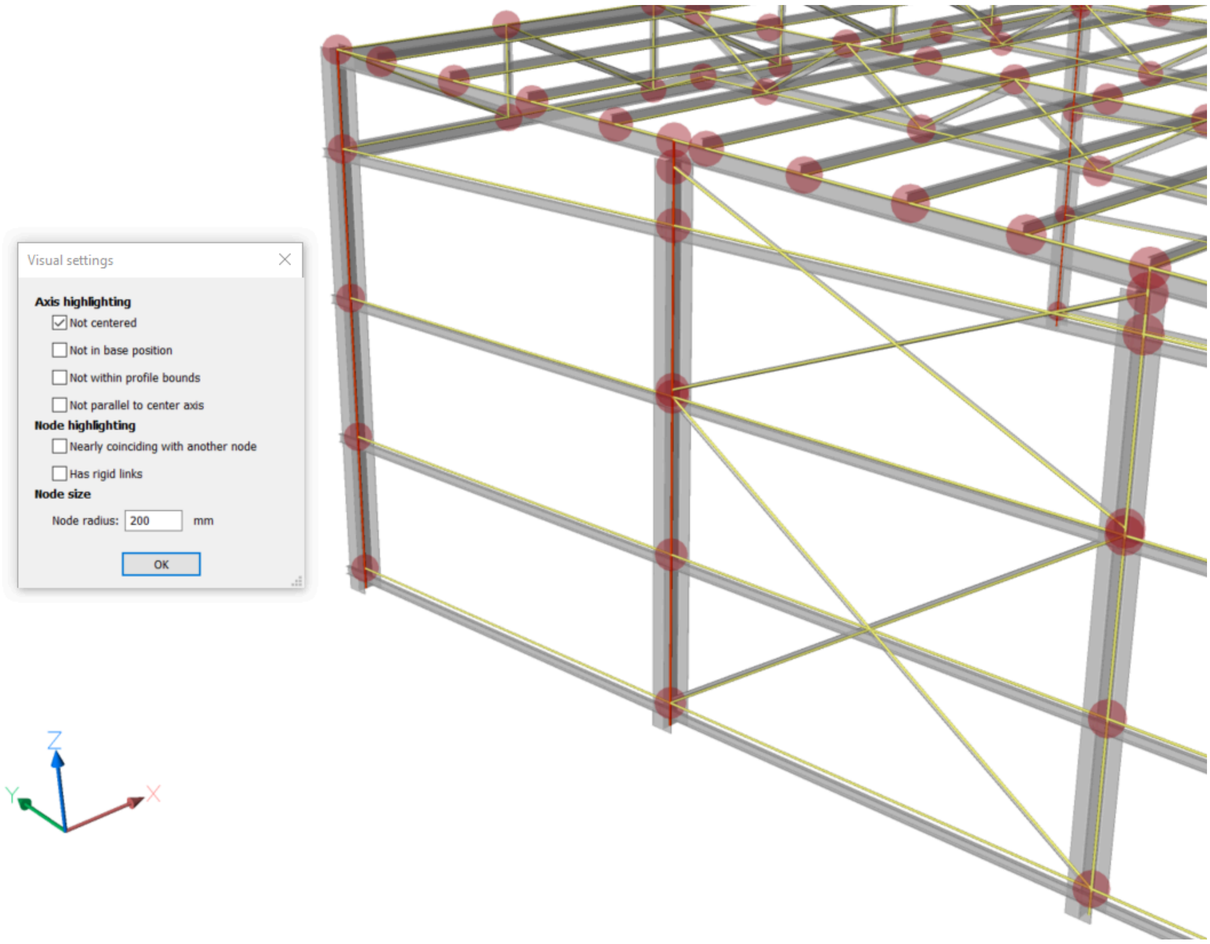
5 In the visual settings window, select the checkbox **Has rigid links**.

Type **R** in to choose **Recalculate**.



6 In the visual settings window, select the checkbox **Not centered**.

You will see that one of the columns is still not centered because the connection at its top includes not only the larger beams but also one of the smaller ones. You change this by imposing a center restriction on this column as well.



23.1.4 Exporting an analytical model

1 Launch the BIMANALYTICALMODEL command.

1 Change the analytical model to your desire.

2 When you have finished the model, type **E** to choose **Export and exit**.

3 The document window appears where you can name and save the file in the desired folder. You can also choose the file type to export. It can either be exported to IFC4 (*.ifc) or CIS/2 (*.stp) file.

4 ***Note:*** There are some remarks about the exported result.

- The **node concept** used by BIMANALYTICALMODEL does not completely coincide with **CIS/2** and **IFC** node concept:

  In the BIMANALYTICALMODEL, only connections between linear elements are modeled with a node. For example, columns will not have a floor node if there is no connection to another element there. During export, missing end nodes are added to be compatible with CIS/2 and IFC export.

  In the BIMANALYTICALMODEL, a node always lies exactly on every connected axis. Eccentricities are modeled with explicit visual rigid links (purple bars in preview). During export, each cluster of BricsCAD nodes that are interconnected with rigid links is translated into a single connection node with eccentricities in **CIS/2** and **IFC**.

- The **axis concept** used by BIMANALYTICALMODEL does not always coincide with the **CIS/2** and **IFC** axis concept:

  In the BIMANALYTICALMODEL, there will be one single axis for each physical element. An axis can be attached to more or less than two nodes. During export, an axis that has multiple nodes along its length gets split up into several analysis members, each with one end and one start node, for the **CIS/2** and **IFC** export.
- The concept of **Base Position** in BricsCAD is similar to **cardinal points**. Base positions in BricsCAD form a subset of the cardinal points specified in the **CIS/2** standard. For now, only cardinal positions 1 to 9 are supported.
- In the analysis model, as calculated by BricsCAD, an axis can shift away from the center of the physical entity, essentially to any point within certain bounds. This means that the final position of the analysis axis does not necessarily coincide with a cardinal point or base position of the profile. During export, the section profile is linked to the analysis axis using the closest base position of the profile to the final analysis axis, using the cardinal point concept in **CIS/2** and **IFC**. This means that if you visualize the structure in a **CIS/2** or **IFC** viewer that shows physical members as extrusions of the profile section along the analysis axes, you will see that the structure shown there can slightly deviate from the original structure as modeled or imported into BRICSCAD. The structure as described by the **exported analysis model** is therefore an **approximation of the actual structure**.

Analytical model.dwg

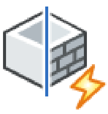
23.2 AutoComplete BIM data

23.2.1 Commands

BIMAUTOMATCH

23.2.2 About the AutoComplete BIM data

Instead of assigning compositions to every element manually, the BIMAUTOMATCH command detects the difference between interior and exterior walls and their respective composition and automatically assigns it to other walls. It can also make distinctions between different floors, different sides of the building, walls with a balcony, etc.



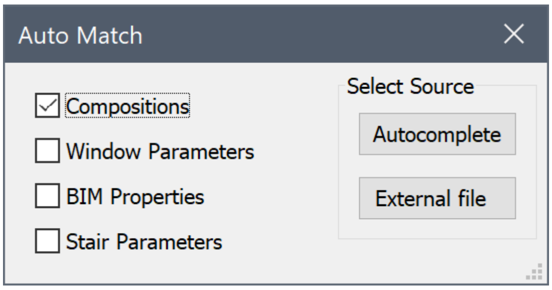
Note: Before you use the BIMAUTOMATCH command make sure that you have Bimified ( ) everything. AutoMatch works best with a fully Bimified model.

23.2.3 AutoComplete BIM data in one file

1 Do one of the following:

- Select **Automatch** from the **Classify** tab in the Ribbon.
- Select **Model** > **Automatch** in the Quad.
- Type BIMAUTOMATCH and press Enter.

You are prompted: Compositions/BIM Properties/Window Parameters/Stair Parameters:



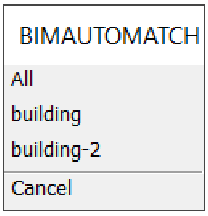
2 Select one of the following:

- **Compositions**: Copy compositions.
- **Window Parameters**: Copy window parameters.
- **BIM Properties**: Copy BIM Properties.
- **Stair Parameters**: Copy Stair Parameters.

3 Under **Select Source**, choose **Autocomplete**.

4 If the file contains multiple buildings, the BIMAUTOMATCH command asks you to define the source building.

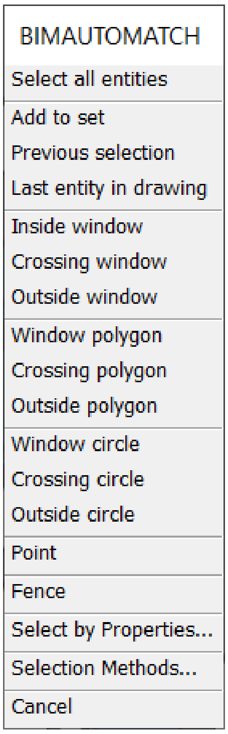
5 **You are prompted:**



Select the building from the menu.

6 If BIMAUTOMATCH finds a logical distinguishing feature between objects it will automatically use this. When it can not, the BIMAUTOMATCH command asks you to define the source composition or window.

7 **You are prompted:**

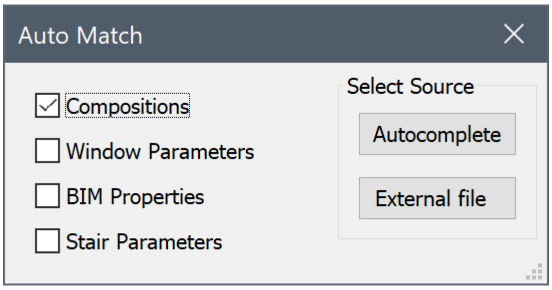


23.2.4 AutoComplete BIM data with an external file

1 Do one of the following:

- Select **Automatch** from the **Classify** tab in the Ribbon.
- Select **Model** > **Automatch** in the Quad.
- Type BIMAUTOMATCH and press Enter.

You are prompted: Compositions/BIM Properties/Window Parameters/Stair Parameters:



2 Select one of the following:

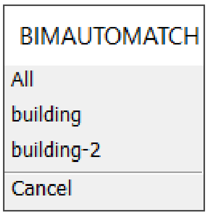
- **Compositions**: Copy compositions.
- **BIM Properties**: Copy BIM Properties.
- **Window Parameters**: Copy parameters of windows.
- **Stair Parameters**: Copy properties of stairs.

3 Under **Select Source**, choose **External file**.

4 Select source file and click **Open**.

5 If the file contains multiple buildings, the BIMAUTOMATCH command asks you to define the source building.

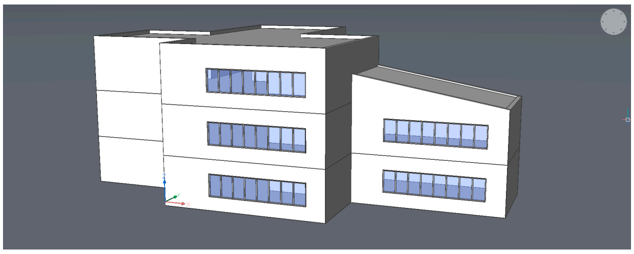
6 **You are prompted:**



7 Select the building from the menu.

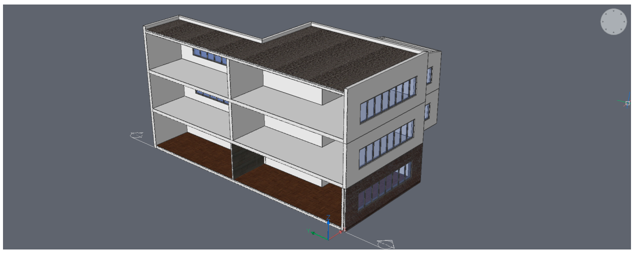
23.2.5 Procedure: using autocomplete in one file

1 Open up a file to start from. Here we use a basic rectangular building that looks like this:

1 

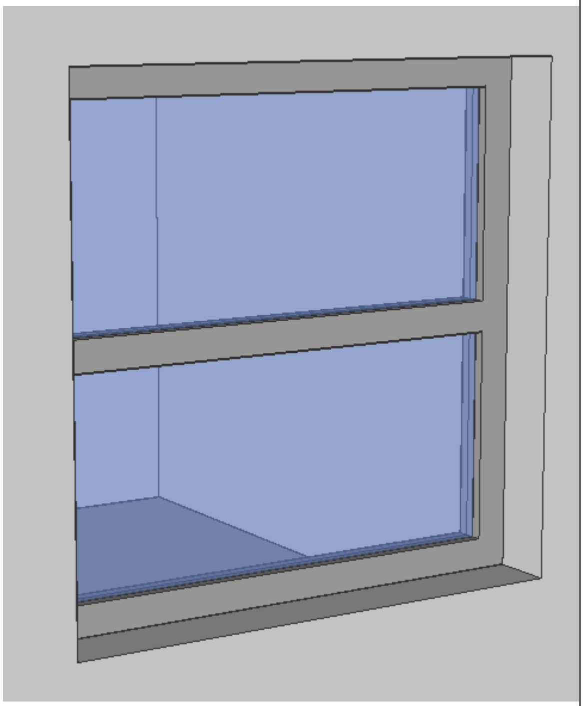
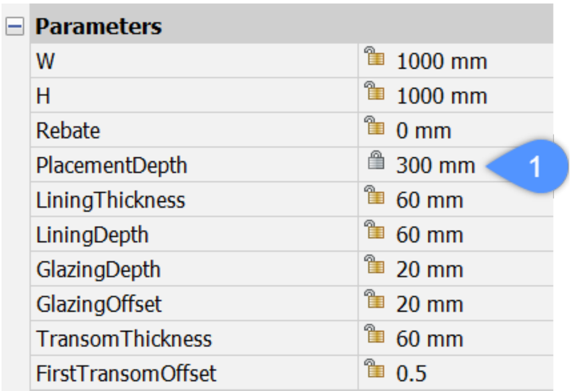
Note: No BIM data or compositions should be added yet.

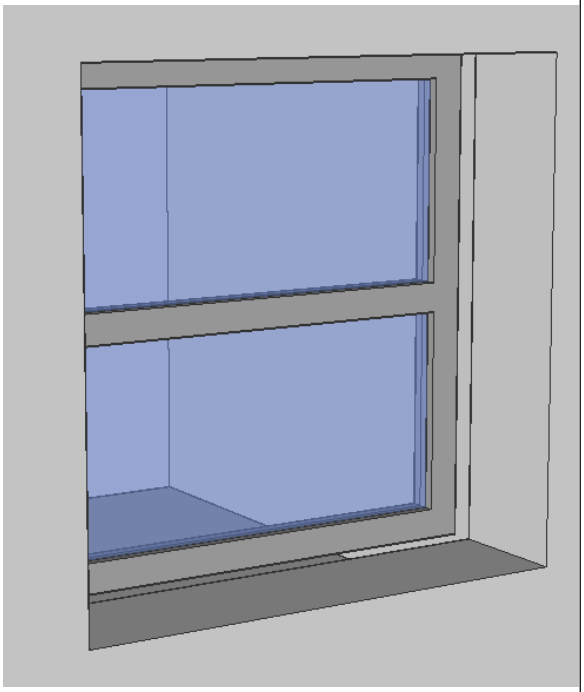
2 Add the compositions that you want. Don't do this for all your elements, but only for one element of each type. In our example we give a composition to 1 outer wall, 1 inner wall, 1 floor slab and the roof:

3 

Note: It could be that you have different types of outer walls, for example the wall facing north that needs more insulation. In this case, you give one of these northern walls a different composition.

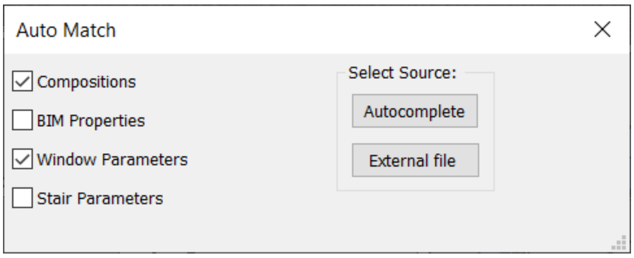
4 In this example, we also want to adjust the parameters of our windows. Select only 1 of the windows and change the parameter you want in the **Properties** panel.

<table>
<tr><td></td><td>The window before changes.</td></tr>
<tr><td></td><td>Changing the PlacementDepth in the Properties panel of the window.</td></tr>
</table>

	The window after the changes.

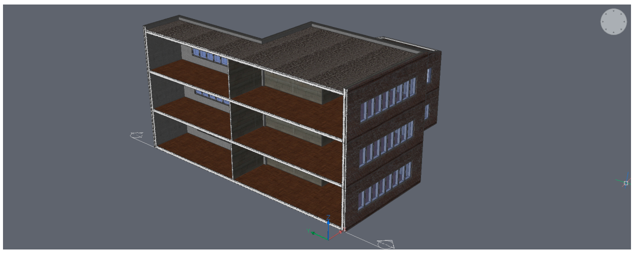
5 Select **Automatch** from the **Classify** tab in the Ribbon **or** select **Model** > **Automatch** in the Quad **or** type BIMAUTOMATCH and press Enter.

The following box appears:



We want the compositions and the window parameters that we chose to be completed in our entire drawing, so we check the boxes from **Compositions** and **Window Parameters**.

6 Select **Autocomplete**. The compositions and window parameters within your file are automatically copied. The drawing looks like this:

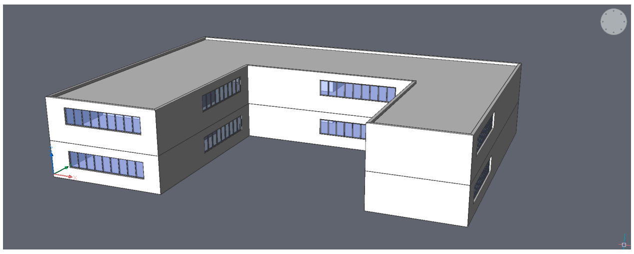


By using BIMAUTOMATCH command, we very quickly have all our compositions and window parameters as we want them, without having to worry about forgetting anything.

Note: You can see that different types of outer walls are no problem. All the walls facing north have been given the correct adjusted composition.

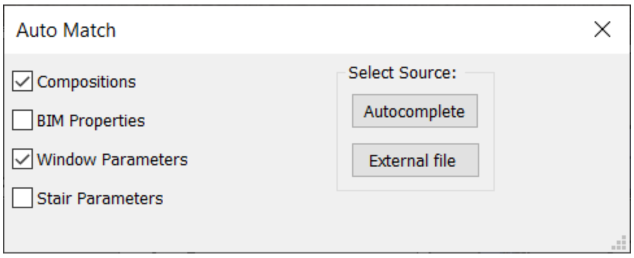
23.2.6 Procedure: using autocomplete with an external file

1 Open up a project that is similar in style to another project that you already finished. In this case, we take another rectangular building, similar to the one we looked at before:

1 

2 Select **Automatch** from the **Classify** tab in the Ribbon **or** select **Model** > **Automatch** in the Quad **or** type BIMAUTOMATCH and press **Enter**.

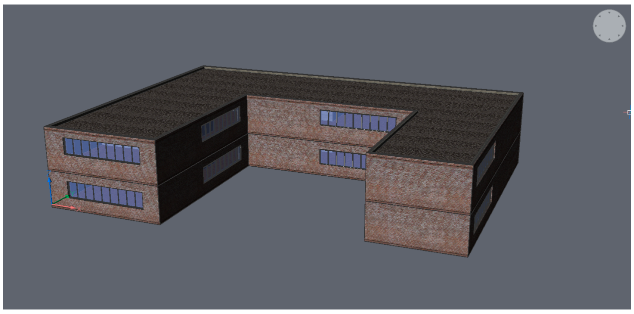
The following window appears:

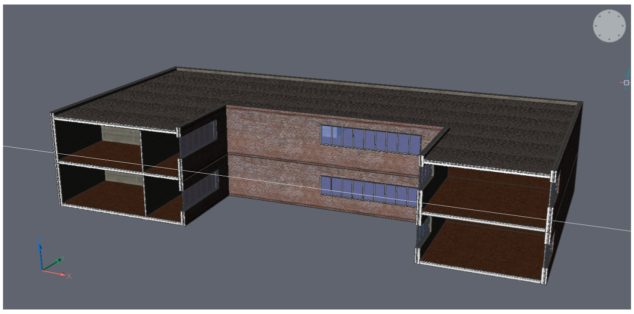


We want the compositions and the window parameters that we chose to be completed in our entire drawing, so we check the boxes from **Compositions** and **Window Parameters**.

3 Select **External file**.

4 Select the file that we made in the previous procedure. The compositions and window parameters from the external file will be copied to your new file. The drawing looks like this:

5 



Note: It can be handy to make a library of building styles with their typical compositions, in order to use the BIMAUTOMATCH command with external files in a very efficient way.

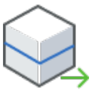
23.3 Attaching compositions

23.3.1 Commands

BIMATTACHCOMPOSITION, BLCOMPOSITIONS

23.3.2 About BIM attach composition

The BIMATTACHCOMPOSITION command allows you to easily apply a composition to a selection of solids. The assigned composition can be flipped or redefined to a different reference face. Optionally, you can align solids based on a reference face or the center of a composition.



See BIMATTACHCOMPOSITION for more information about attaching a composition.
See BIM Compositions for more information about managing compositions.

23.3.3 Attaching a composition

To attach a composition, you can:

- Use the BIMATTACHCOMPOSITION command.
- Drag a composition from the **BIM Compositions** panel on the solid(s).

Using the BIMATTACHCOMPOSITION command

1 Launch the BIMATTACHCOMPOSITION command.

1 **You are prompted:** Enter the composition name or [Dialog/Entity] <Dialog>.

2 Do one of the following:

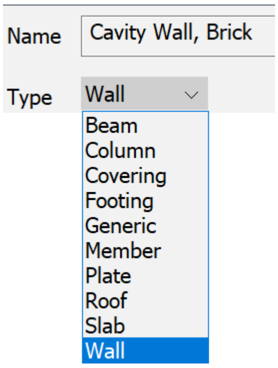
  - Type a composition name in the Command line and press Enter.

    Note: Composition names are case sensitive.

  - Press Enter to display the **Compositions** dialog and click on a Composition in the **In Project** or **In Library** list to open the menu of the composition.

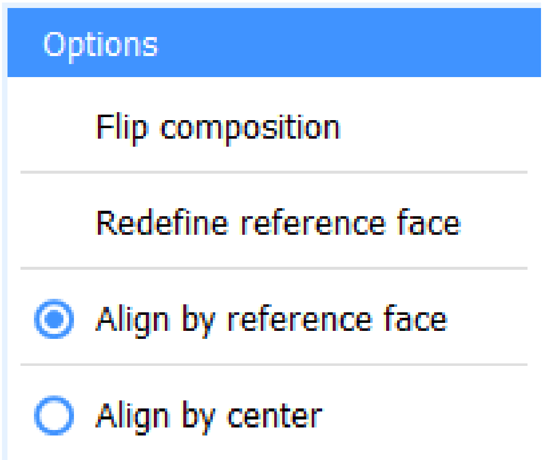
    If you want to select the composition without opening the menu, double-click it.

3 Choose a Composition **Type** from the drop-down menu and press **Select** to accept.



4 Select entities to attach the composition to and press Enter.

5 Choose the reference face. If you hover over the widget, a submenu displays showing different options. For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.



6 Press Enter to accept the reference face.

7 BricsCAD responds: The composition has been assigned to <xxx> element(s).

Note: Using Bimify building elements are automatically classified. This makes it easy to select the right entities.

Dragging a composition on a solid

1 Select the composition in the BIM Compositions panel ( ). See Dockable panels.

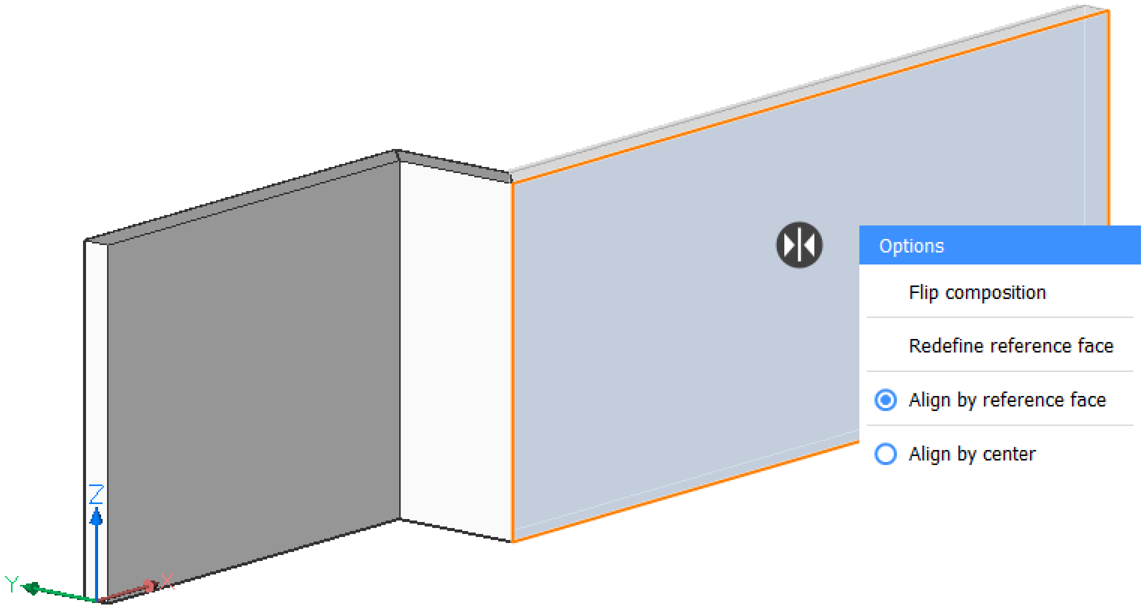
2 Hold down the left mouse button and drag the composition on the solid.

3 The solid face under the cursor highlights, indicating that this face is the reference face to attach the composition.

4 Release the left mouse button to attach the composition.

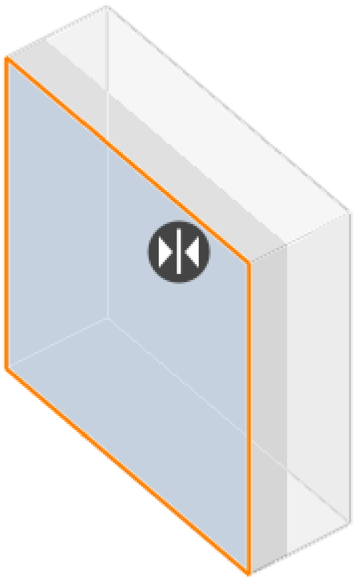
5 A widget appears that allows you to choose the reference face.

6 The reference face is fixed, while the opposite face moves if the thickness of the composition does not fit the current thickness of the solid.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

7 A darker volume indicates the volume and position of the solid after attaching the composition.



8 Press Enter to accept the current reference face and attach the composition.

Note: When a single ply composition with a variable thickness is selected, it is not needed to select a reference face. The composition is attached to the selected solid.

Dragging a composition on a selection of solids

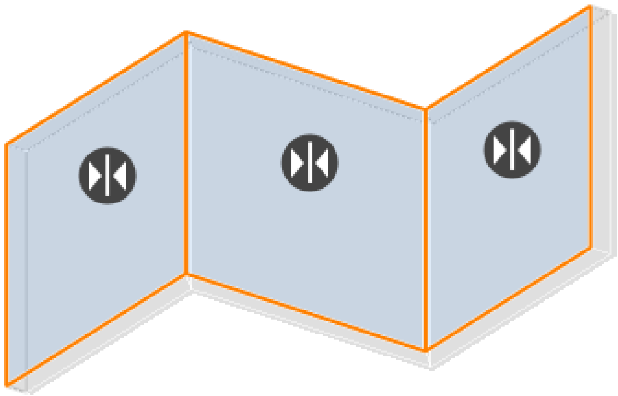
1 Select the solids you want to apply the composition to.

2 Select the composition in the **BIM Compositions** panel.

3 Hold down the left mouse button and drag the composition on any of the selected solids.

4 Release the left mouse button to attach the composition.

5 A widget appears which allows you to choose the reference face for every segment separately.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

6 Press Enter to accept the current reference face and attach the composition.

Note:

- BIM compositions are copied to the Project database upon attachment.
- Attaching a **Wall** composition to a solid, automatically classifies the solid as a wall.
- Attaching a **Slab** or **Roof** composition to a solid, automatically classifies the solid as a slab.

23.3.4 Copy compositions from other solids

1 Launch the BIMATTACHCOMPOSITION command.

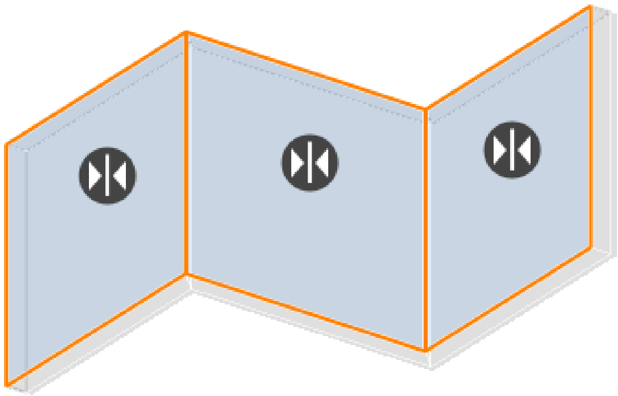
2 Select the **Entity** option.

Prompts you: **Select other entity to get composition:**

3 Select the solid to copy the composition from.

4 Select the solids(s) to attach the copied composition to.

5 A widget appears which allows you to choose the reference face for every segment separately.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

6 Press Enter to accept the current reference face and attach the composition.

23.3.5 Removing compositions from solids

The **Unclassify** option of the BIMCLASSIFY command removes a composition from a solid.

1 Select the solid(s).

2 Choose **Unclassify** from the **BIM** command group in the Quad.

23.4 BIM classify

23.4.1 Commands

BIMCLASSIFY

23.4.2 About

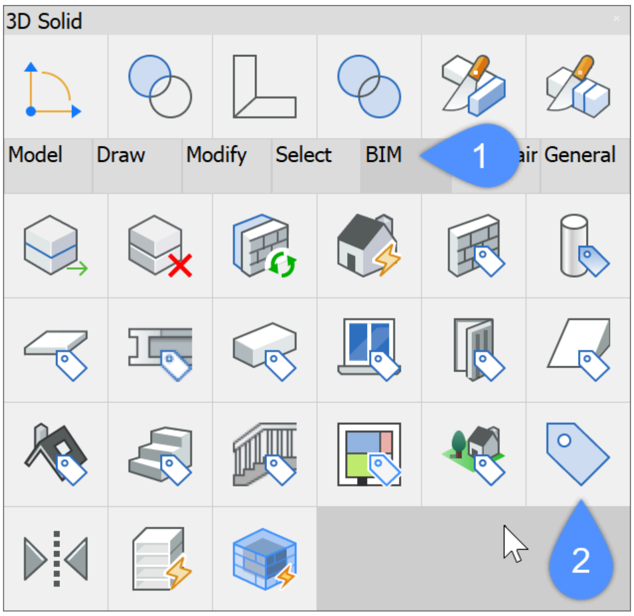
If you have a conceptual model without any classification, you can turn these elements into classified BIM elements using BIMCLASSIFY.

For more information about this command, visit the Command Reference article BIMCLASSIFY.

23.4.3 Classifying entities

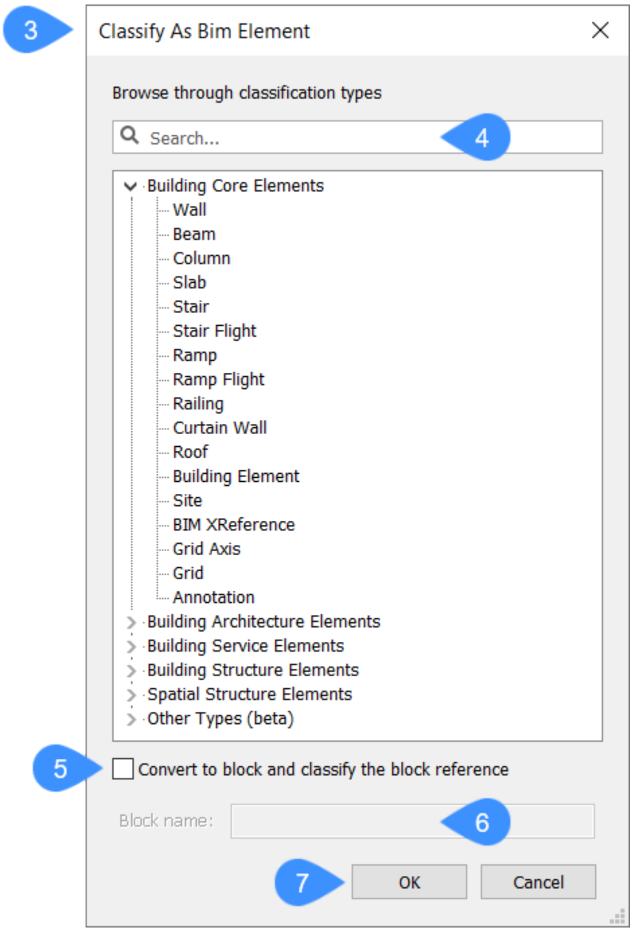
1 Select an entity in the drawing to classify.

2 Choose a classification type (1) in the **BIM** command group in the Quad.



The BIM data is assigned to the selected entity.

3 **(Optional)** Choose the **Classify Manually** option (2) from the Quad to access the BIM Element classification types. This displays the **Classify As BIM Element** dialog box (3):



Note: The classification types are grouped into the following categories:

- **Building Core Elements**
- **Building Architecture Elements**
- **Building Service Elements**
- **Building Structure Elements**
- **Spatial Structure Elements**
- **Other Types (beta)**

Do one of the following:

- Enter a specific name of the BIM Element type in the **Search** field (4) to filter the available element types.
- Select a Building Element type from the list.
- (**Optional**) Check the **Convert to block and classify the block reference** option (5) and enter a name in the **Block name** field (6).

  This option will convert the selected entity to a block and classify the block reference to the defined Building Element type.
- Click the **OK** button (7) to accept it.

4 **(Optional)** Use the **Auto** ( ) option from the Quad to classify the selected entities automatically.

This automatic classification is done based on the geometrical features of the object. Always check whether it was correctly classified.

5 To remove the assigned BIM data from the entity, select the entity in the drawing and choose the **Unclassify** ( ) option from the Quad.

The BIM data is removed from the selected entity.

23.4.4 Properties of the classified entities

When you classify an entity, a number of properties with regard to the building element type is displayed in the **Properties** panel.

The following illustrations show the properties that are available for windows and doors:

Door Window Glazing Type	
Glass layers	0
Glass thickness 1	0 mm
Glass thickness 2	0 mm
Glass thickness 3	0 mm
Fill gas	
Glass color	
Is tempered	Off
Is laminated	Off
Is coated	Off
Is wired	Off
Translucency	0
Reflectivity	0
Beam radiation transmittance	0
Solar heat gain transmittance	0
Thermal transmittance summer	0 W/m²·K
Thermal transmittance winter	0 W/m²·K

BIM	
Type	Window
Building Element Type	Window
Name	
Description	
Building	
Story	
Space	
Composition	
GUID	0shXs2TdfFB9RulemExNT4
Space bounding	Off
Construction type	Not defined
Operation type	Not defined
Window inset	
Overall height	0 mm
Overall width	0 mm
Sill height	0 mm
Head height	0 mm
Assembly code	
Assembly subtype	
Number	
Entity property sets	

Door Window Shading Type	
External shading coefficient	0
Internal shading coefficient	0
Inset shading coefficient	0

Window Common	
Reference	
Fire rating	
Acoustic rating	
Security rating	
Is external	Off
Infiltration	0 m³/s
Thermal transmittance	0 W/m²·K
Glazing area fraction	0
Smoke stop	Off

23.5 BIM compositions

23.5.1 Commands

BLCOMPOSITIONS, BIMCLASSIFY, BLMATERIALS, BIMATTACHCOMPOSITION

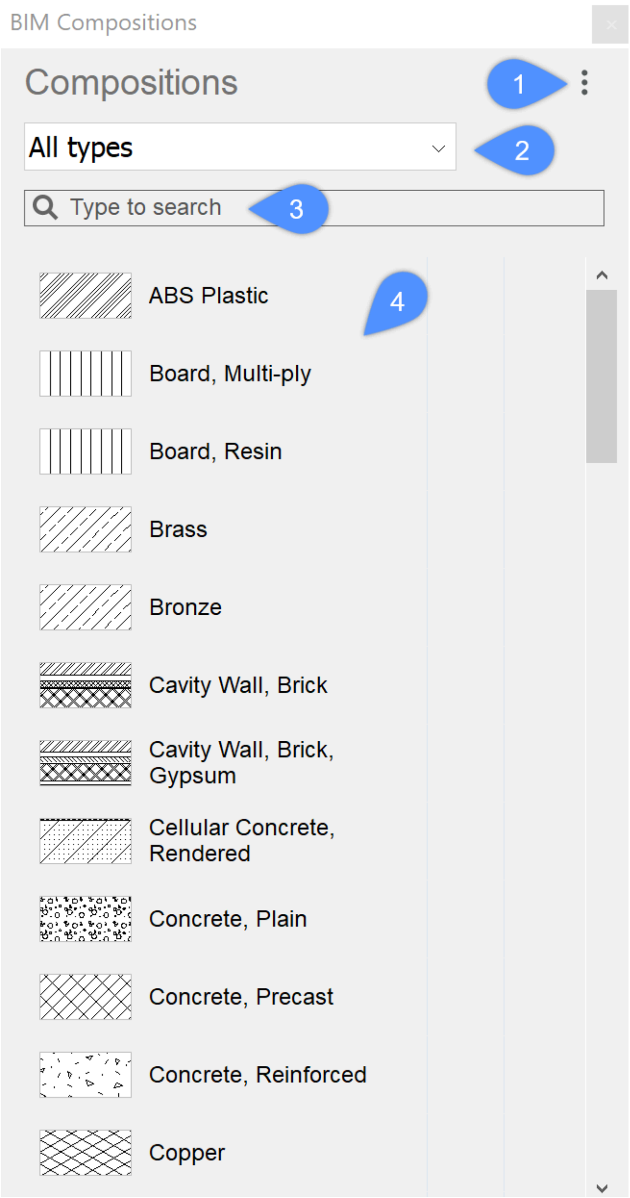
23.5.2 About

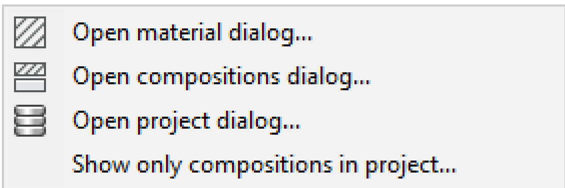
BIM compositions:

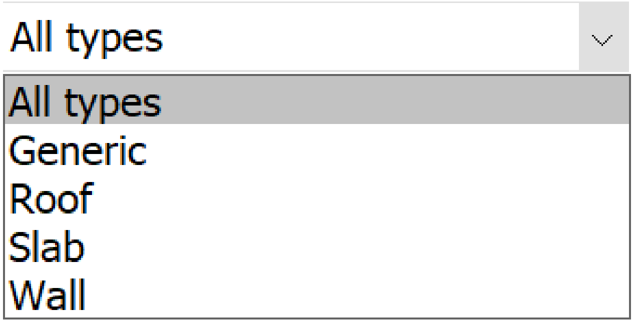
- Are stored in the central database.
- Are either multiply or single-ply.
- Ply materials are managed in the **Physical Materials** editor ( ).
- Can be assigned to any solid in a BIM model.
- Are listed in the **BIM Compositions** panel ( ).
- Are managed in the **Compositions** editor ( ).

For more information about Attaching Compositions, see the procedure article Attaching Compositions.

23.5.3 Using the BIM compositions panel


BIM Compositions
Compositions
1
All types
2
Type to search
3
ABS Plastic
4
Board, Multi-ply
Board, Resin
Brass
Bronze
Cavity Wall, Brick
Cavity Wall, Brick, Gypsum
Cellular Concrete, Rendered
Concrete, Plain
Concrete, Precast
Concrete, Reinforced
Copper

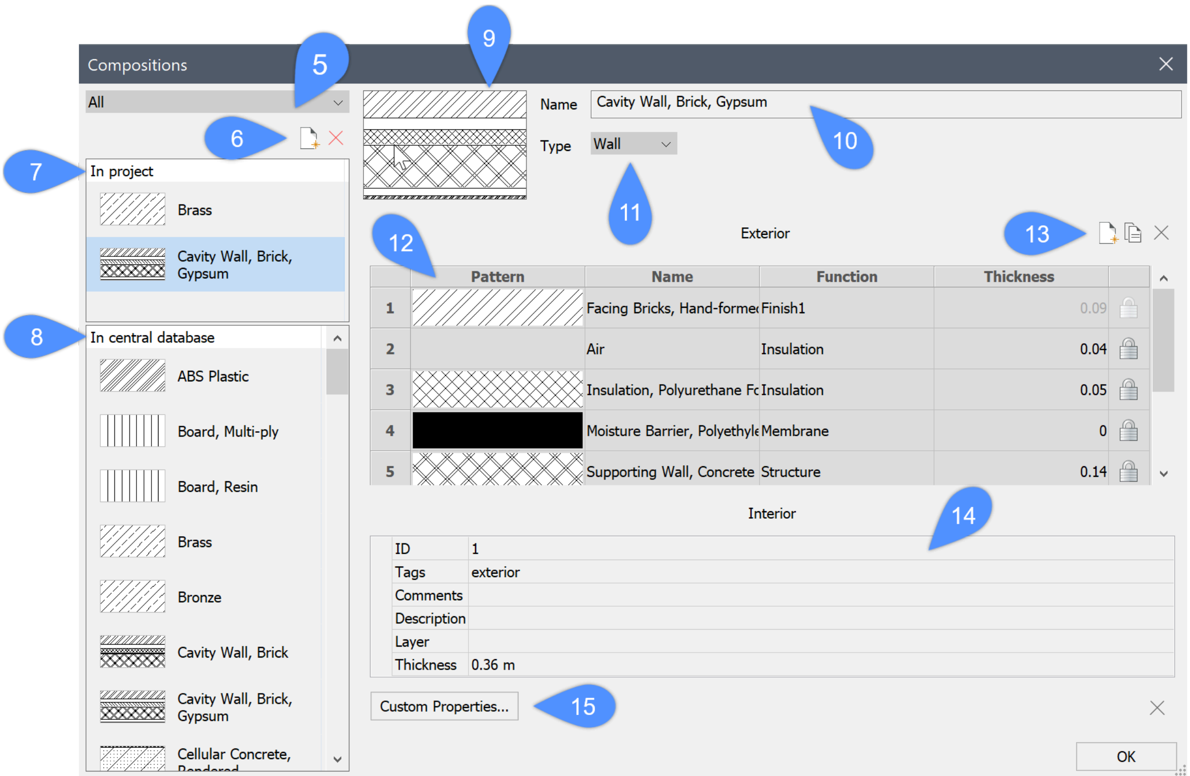
(1) Menu	 Open material dialog... Open compositions dialog... Open project dialog... Show only compositions in project...

(2) Composition types list	
(3) Search field	Filters the list of the compositions. Only compositions that contain the current search term are displayed.
(4) Available compositions	All compositions of the selected type in both the **In Central Database** and the **In project** database display, unless the **Show only compositions in project...** menu option is selected.

23.5.4 Using the BIM compositions editor

The BIM **Compositions** editor is a dialog box that contains options to create, modify and delete compositions. Using the **Compositions** editor, you can also sort the compositions by category and view the properties of the composition.

The BLCOMPOSITIONS command opens the BIM **Compositions** Editor:



(5) Composition category selection	Filters by category: **Wall**, **Slab**, **Roof**, or **Generic**.
(6) New composition	Creates a new composition.
(7) In project	Displays the compositions of the selected category (**wall**, **slab**...) in the Project database.
(8) In Central Database	Displays the compositions of the selected category (**wall**, **slab**...) in the Central Database.
(9) Preview pane	Displays the currently selected composition.
(10) Name field	Displays the selected composition name. You can change the default name.
(11) Category drop-down box	Displays a category for the new composition.
(12) Structure grid	Displays the material and thickness of the composition plies. The ply on top (exterior) is applied to the reference of the solid.
(13) Add ply	Opens the physical materials editor which allows you to choose a material from the central database or project database. Here you can also create a new material. Inserts a copy of the selected ply.
(14) Properties grid	Shows the properties of the selected composition.
(15) Custom properties...	Opens the **custom properties** dialog box.

23.5.5 Creating a new composition

You can create a new composition in either the **Project** or the **Central** database.

1. To open the BIM **Compositions** editor, do one of the following:
   - Select the **Open the compositions dialog...** in the menu on the **Compositions** panel.
   - Type BLCOMPOSITIONS in the Command line.
2. Click the **New** icon ( ) on the left of the compositions toolbar.

   A new (empty) composition is added to the selected database.
3. To add and edit plies, go to **Edit a composition**.

23.5.6 Editing a composition

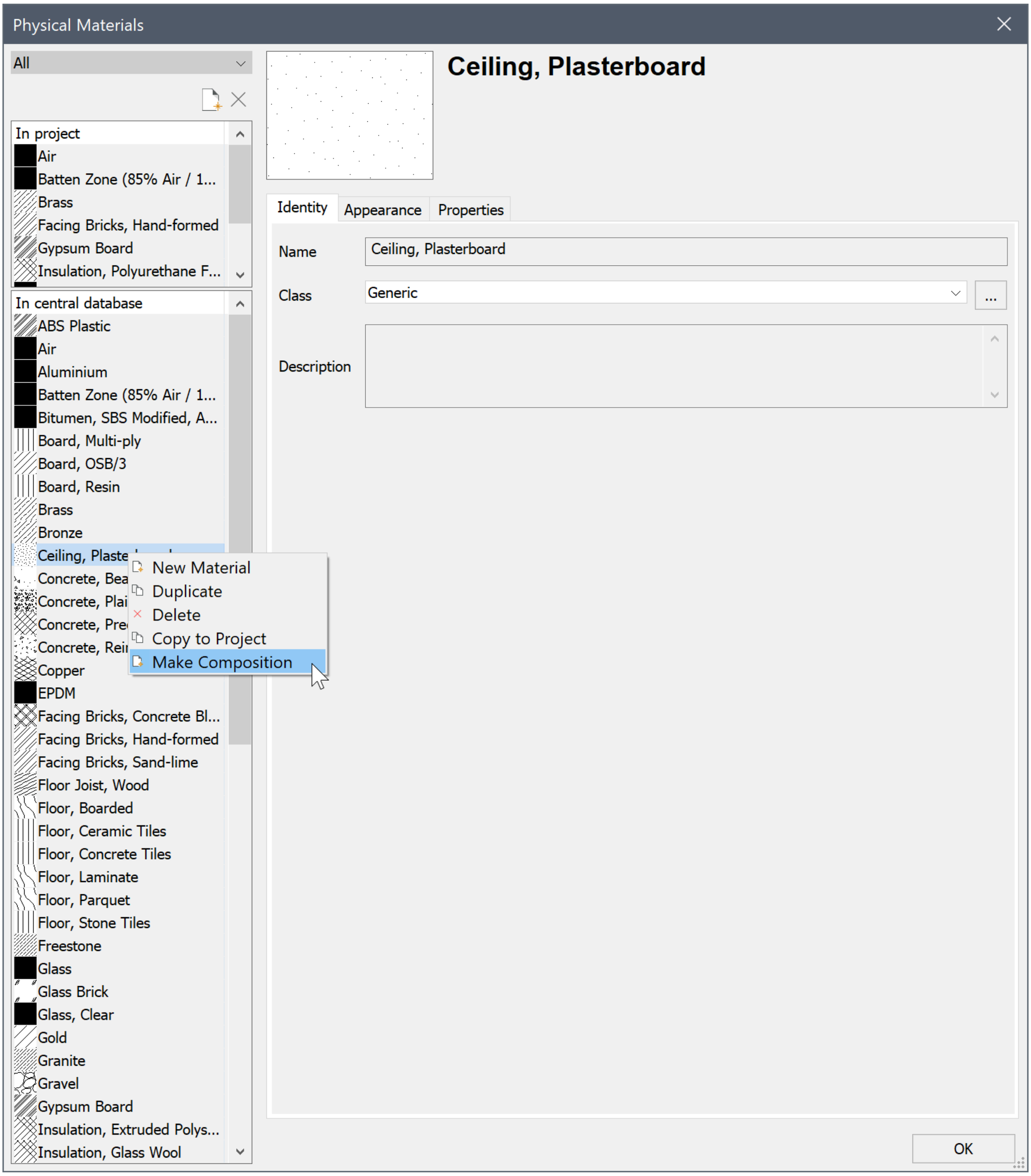
Step 1: Changing name and type

1. Select the new composition.
2. Type a new name in the **Name** field for the composition.
3. Use the Type drop-down box to specify the composition type: **Wall**, **Slab**, **Roof** or **Generic**.

Composition categories are used to filter the list of the compositions and the **Compositions** dialog box.

Step 2: Changing the plies

- **Adding, editing and deleting plies**
    a. Click the Add Ply button ( ) on the right of the compositions editor.
    The **Physical Materials** dialog box displays:



b Select a material either in the Project **or** the Central database and double-click **or** press the **OK** button to assign the material to ply in the composition.

c Double-click on the **Pattern** or **Name** of the ply to edit the material. See the **Physical Materials** article for more information.

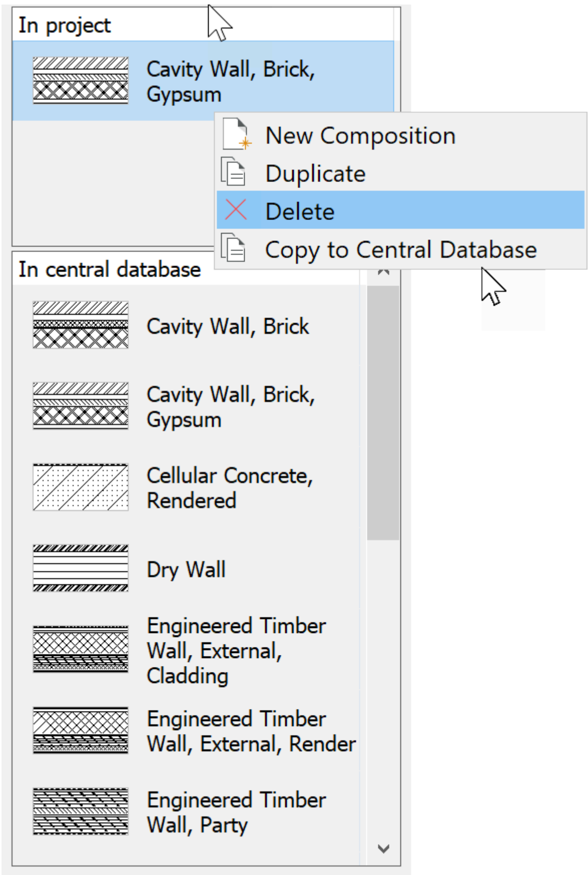
d To choose another material, pick a material in the Central **or** Project list and press **OK**.

e To delete a ply, select the ply and click on the **Delete** tool button.

f (Optional) Right-click the material and choose **Make Composition** (16) in the context menu to create a single-ply composition.

- **Modifying the thickness and changing the ply order**

  a Click the **Thickness** field (17) to modify the thickness of a ply in the Structure grid.
  You can only modify the ply thickness if the **Allow Custom Thickness** property of the material is set to **Yes**.



  b Enter a value in the field (18) and press Enter to accept it. The overall thickness of the composition is updated accordingly.

  c The plies of a composition are ordered from Exterior (top) to Interior (bottom). Drag the ply number (21) on the desired position to change the ply order.

Note: When a composition has been modified, the solids that have this composition attached are updated automatically. Use the BIMUPDATETHICKNESS command apply the changes in the model.

- **Locking/unlocking the composition thickness**

  The overall thickness of a composition is equal to the sum of the thicknesses of the composing plies.

  The thickness of a ply can be either locked or unlocked (see image above).

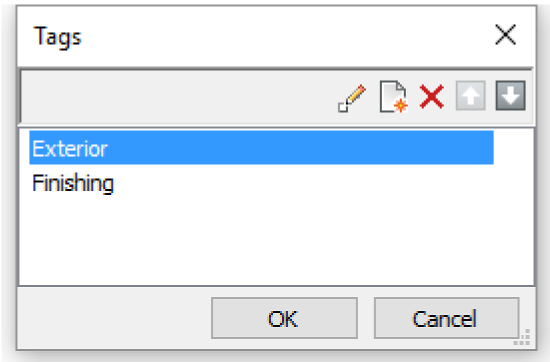
  - Locked thickness (19): the thickness of the ply is fixed.
  - Unlocked thickness (20): the thickness of the ply is variable.

Note: Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

- **Fixed:** all plies have locked thicknesses.
- **Minimal:** the composition contains at least two plies and one ply has an unlocked thickness. The minimum thickness is equal to the sum of the thicknesses of the locked plies.
- **Free:** a single ply composition has an unlocked thickness.

The thickness of a ply can only be unlocked if the **Allow Custom Thickness** property of the material is set to **Yes**.

- **Adding Tags**
  a Select the **Tags** field.
  b Type the tags, separated by commas. Tags are case sensitive.
  c Click the **Browse** button ( ) next to the tags field in the **Compositions** dialog box to open the **Tags** dialog.



  d Click the **New Item** button ( ) to add a new tag at the bottom of the list.
  e Select a tag in the list and click a tool button:

- 
- 
- 
- 

  f Click the **OK** button to save the changes.

Step 3: Creating new materials

You can create new materials from scratch or copy them from an existing material. See **Physical Materials** for more details.

23.5.7 Displaying compositions

Whether the various plies of a composition show in the 3D model is controlled through the LEVELOFDETAIL user preference. The RENDERCOMPOSITIONMATERIAL system variable controls whether the ply material is rendered or not.

Click the icons in the **Home/View** panel in the ribbon or the **BIM** toolbar to toggle the status of the composition display:

- Level of Detail: 
- Render Composition Material: 

When LEVELOFDETAIL system variable is 0, the major faces of a solid are rendered using the material of the outside plies. The minor faces (for example top, bottom, start and end face of a wall) are rendered using the material of the exterior ply.

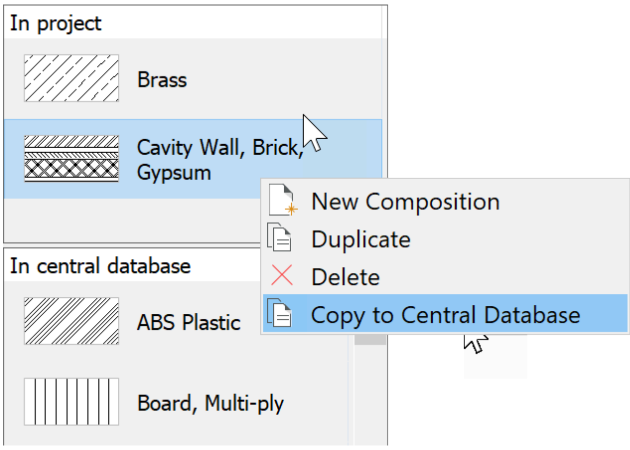
23.5.8 Copying compositions

You can use the following procedure in both the **BIM Compositions** panel and the **Compositions** editor:

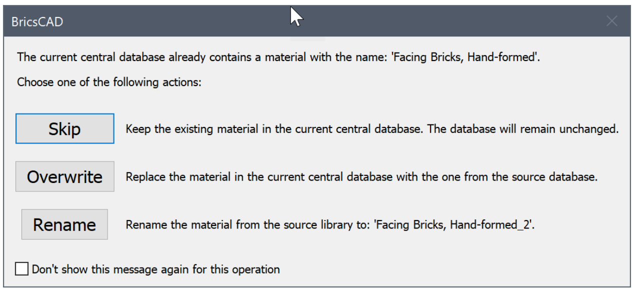
1 Select the composition(s) in the source database. Press and hold the Ctrl key to select multiple compositions.

2 Do one of the following:

- Drag the selection to the other database.
- Right-click and choose **Copy to Central Database** or **Copy to Project** in the context menu.



A dialog box displays if the composition already exists in the target central database. You can choose to skip, overwrite or rename existing compositions.
(Optional) Check the **Don't show this message again for this operation** option.



23.5.9 Deleting compositions

1 Select the compositions in the **In Project** or **In Central Database** list on the **BIM Compositions** panel or

the **Compositions** editor.

2 Do one of the following:

3 - Right-click and choose **Delete** in the context menu.
 - Click the **Delete** button (**Compositions** editor only).

Note: Compositions that are used in the project cannot be deleted from the Central Database.

23.6 BIM data structure

23.6.1 Project database and central database

A BIM Project consists of dwg entities, classified as **Building Elements**, to which Compositions can be attached. All Compositions being used in a project, are stored in the Project database. The Project database can be stored inside a dwg file, or in a separate file with extension **.bsyslib:**

- When the Project database is embedded in a dwg file, that dwg file contains the complete BIM Project.
- When the Project database is stored as an external file, the BIM Project can consist of multiple dwg files, all using the same Project database.

To reuse compositions and building materials across multiple BIM Projects, a Central database can be specified, in addition to the Project database. The Central database is a **.bsyslib** file, usually stored in a location which is common to several projects. When Compositions and Building Materials from the Central database are used in a BIM Project, they are automatically imported in the Project database. To maintain and extend BIM databases, users can drag Building Materials and Compositions from one database to the other, provided that they have write access to the destination database.

Note	Sample databases are installed in the **Support** subfolder in the **Roamable** root folder. When a BricsCAD update is installed, the **BricsCAD User File Manager** will prompt you whether to overwrite the sample databases or not. Therefore, it is recommended to rename your central databases and/or save them at a different location. The location of the central databases is defined in the **Central Database** tab of the **BIM Project Info** dialog box.

23.6.2 Materials and compositions

A BIM database consists of Building Materials and Compositions. A Building Material contains information about one specific material: its appearance, cost, manufacturer, etc. A Composition contains information about the structure of a building element, by defining Composition layers. Each Composition layer refers to a Building Material and has a certain thickness.

Flexibility

The same flexibility you have in modeling is available in attaching building information. Although Compositions are grouped into **Wall**, **Slab**, **Roof** and **Generic** Compositions, any type of Composition can be attached to any dwg entity. When attached to a 3D Solid, with specific geometrical properties, the thickness of the Composition is used to define the thickness of walls and slabs, and the cross section will show the wall or slab structure by applying the hatch patterns from the various layers on the section geometry. When a 3D Solid is assigned a Composition with a fixed thickness or a minimal thickness, and this thickness can not be applied to the geometry, the section appears in red to indicate the incompatibility between the selected Composition and the geometry of the solid.

BIM Compositions panel

All BIM database features are accessible from the dockable **BIM Libraries** panel.

Materials

A Material definition describes the appearance and properties of a Building Material.

Composition layers

A Composition describes the structure of a building element by an ordered set of composition layers. Each composition layer has the following fields:

- Pattern and Material **Name**: The Building Material for this layer.
- Function: One of (**None**, **Structure**, **Substrate**, **Insulation**, **Finish1**, **Finish2**, **Membrane**). Currently, this field serves for information purposes only.
- Thickness: Defines the thickness of the layer.

23.7 BIM ply editing

23.7.1 About

The **BIM Ply Editing** function allows you to manipulate all the composition plies in your model. In BricsCAD, each drawing has settings stored inside to control the composition plies, called LOD (LEVELOFDETAIL), and the render materials, called RENDERCOMPOSITIONMATERIAL. These two settings can be turned on/off separately.

23.7.2 Composition plies and render materials

Composition plies display only when LEVELOFDETAIL system variable is 2 (high).

Building elements do not need to be selected, they will be automatically adjusted. When LEVELOFDETAIL system variable is 0 (low) the material definition of the solid is used.

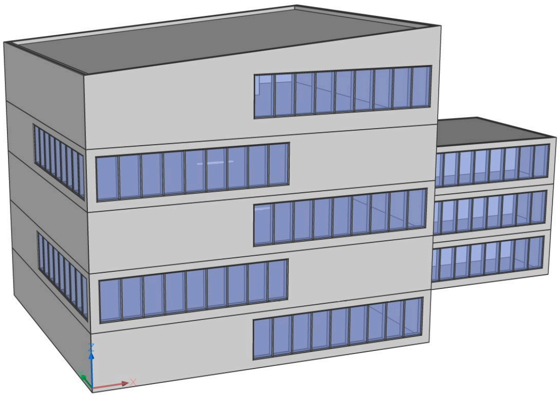
To toggle the value of the LEVELOFDETAIL system variable, click the icon in the ribbon under the **Home** tab or in the quad under the **BIM** tab:

For more information about this setting visit the article LEVELOFDETAIL.

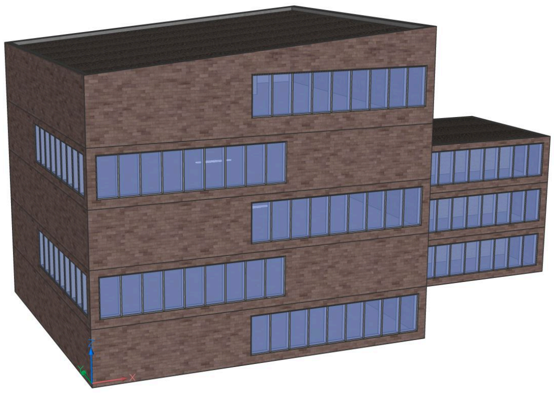
Render materials display only when RENDERCOMPOSITIONMATERIAL system variable is 1 (on). The render material of each ply is defined in the 3D Render Material setting of the material definition.

To toggle the value of the RENDERCOMPOSITIONMATERIAL system variable, click the icon in the ribbon under the **Home** tab or in the quad under the **BIM** tab:

For more information about this command, visit the article RENDERCOMPOSITIONMATERIAL.



RENDERCOMPOSITIONMATERIAL = 0 (off)



RENDERCOMPOSITIONMATERIAL = 1 (on)

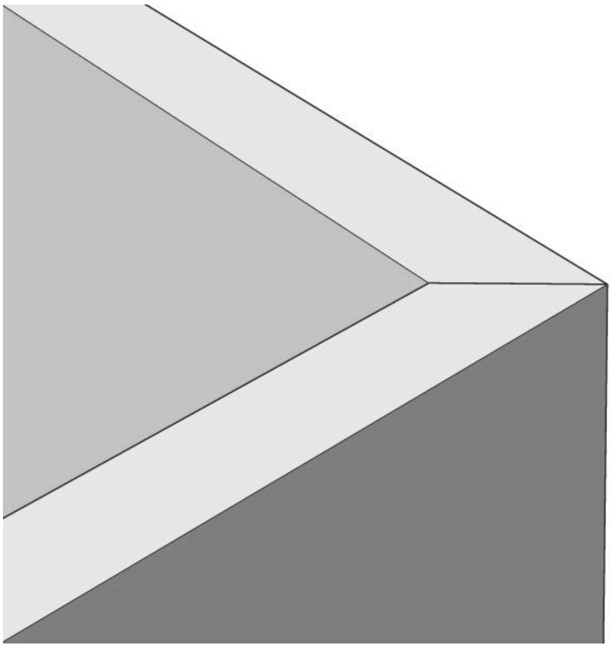
> ***Note:*** Only when Visual Styles have their **Face Settings / Material Display settings** set to **Materials and Textures** (e.g. **BIM**, **Modeling**, and **Realistic**), a visual representation of materials is displayed. Composition plies are displayed in the model as well as render materials.

> ***Note:*** If the render materials of the selected composition are not downloaded or the composition is incorrect, the entity will be displayed in red.

There are different combinations possible with LOD and RENDERCOMPOSITIONMATERIAL system variables turned on or off.

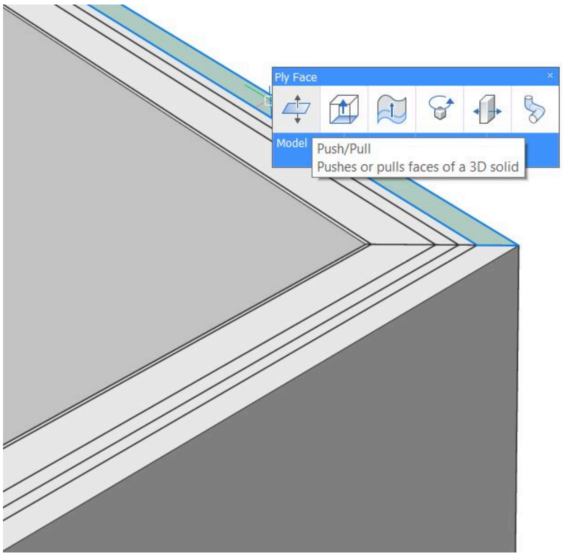
- **LOD =0, RENDERCOMPOSITIONMATERIAL = 0**

The material definition of the solid is used and no materials are displayed.



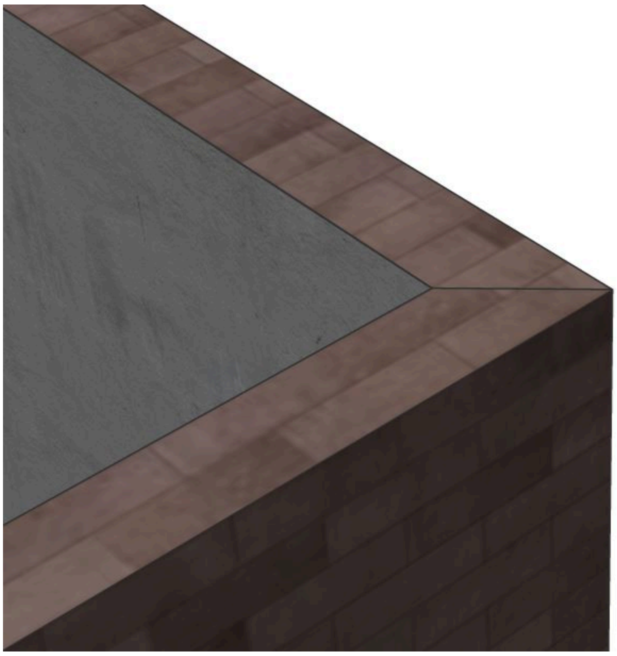
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 0**

  The solid is subdivided into plies, no materials are displayed. When **face detection** is turned on, you can select the faces of different plies to **push/pull** for example.



- **LOD =0, RENDERCOMPOSITIONMATERIAL = 1**

  The material definition of a solid is used. The materials of the exterior and interior face are displayed and the material of the exterior layer will be wrapped around the object.



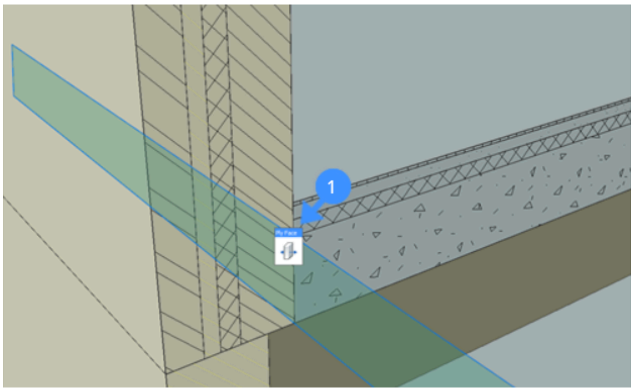
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 1**

  The solid is subdivided into plies. Each ply will be displayed with its own material. When face detection is turned on, you can select the faces of different plies to push/pull for example.

23.7.3 Manipulating the composition plies

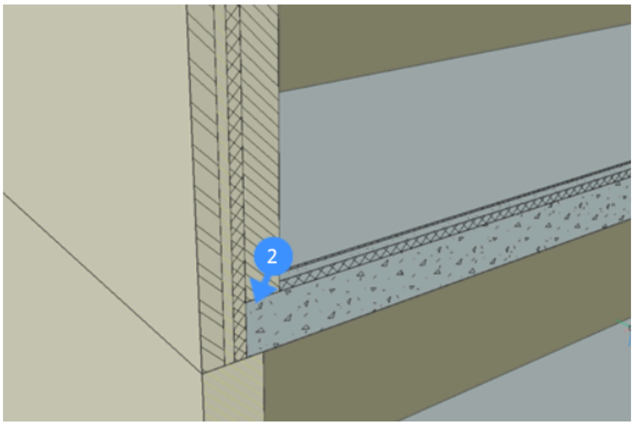
1 Make sure the value of **LEVELOFDETAIL = 2 (High)**.

2 Place your cursor near the composition ply you want to edit and press the Tab key until the **Ply face** (1) highlights.



Note: The highlighted ply face is displayed in the detected boundary color (default = 95). Face detection must be enabled to do this.

3 Select **Push/Pull** from the Quad and drag the ply to a new position (2).



Note: Make sure DMPUSHPULLSUBTRACT system variable = 1. This ensures that plies being push/pulled are subtracted from any of the other 3D solids they intersect.

Note: Make sure ESNAP is turned on to snap to the edge of a ply.

23.7.4 Procedure: creating a rooftop detail

It is also possible to create more complex details with extra solids added apart from the existing compositions. As an example, a rooftop detail will be created, starting from a rooftop edge.

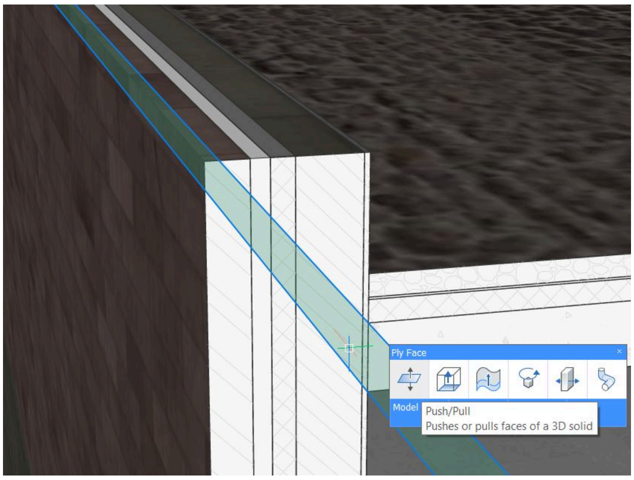
1 Start with this roof-wall connection.

2 Click the **Face detection** icon in the ribbon to enable face detection.

3 Turn on LOD to be able to select ply faces. Click the icon in the ribbon **or** type LEVELOFDETAIL in the Command line, type **2** and press Enter.

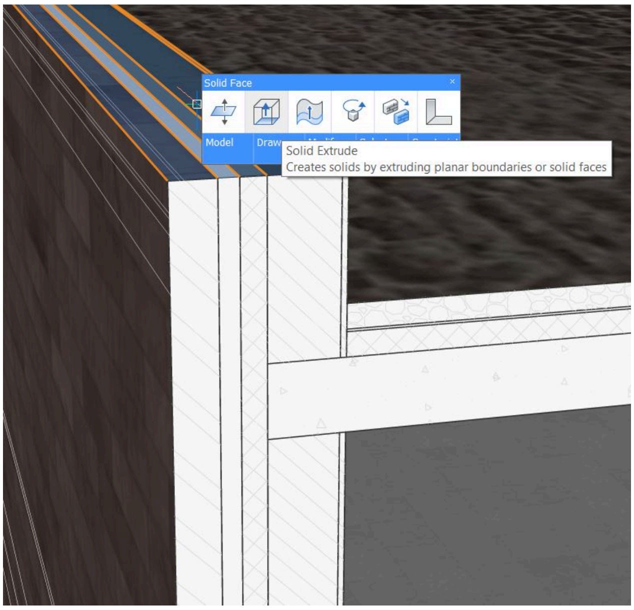
4 Hover over the edge of the concrete ply of the floor slab and press Tab to select the face of the ply.

5 Choose **Push/Pull** in the **model** tab of the Quad.

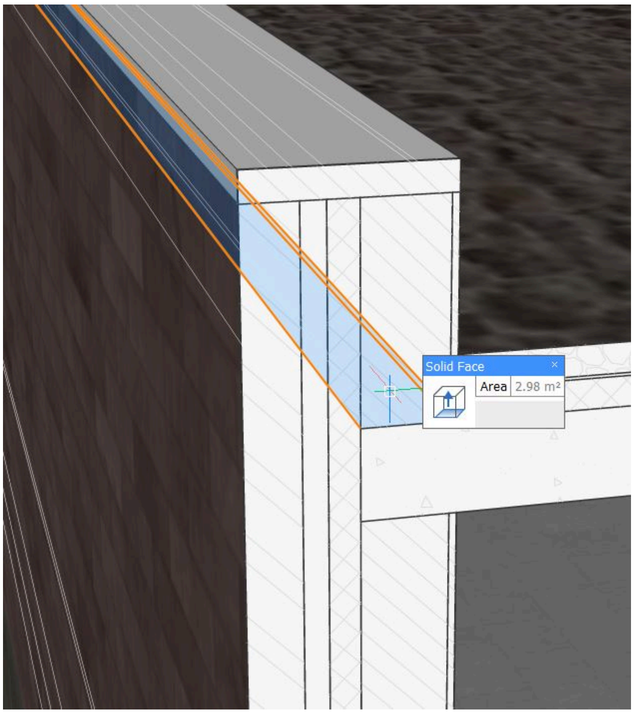


6 Push/Pull the face to the next ply of the wall.

7 To create a wall cap, select the top surface of the wall by pressing Tab and extrude the surface up for 50mm. The created cap isn't classified yet.



8 Delete the part where the insulating building block will come. Hover over the surface of the supporting wall and the gypsum board and press tab to select the surface. Then Extrude till the ply of the wall cap.



9 **Extrude** the same surface again, now a new solid will be created.

23.8 BIM project info

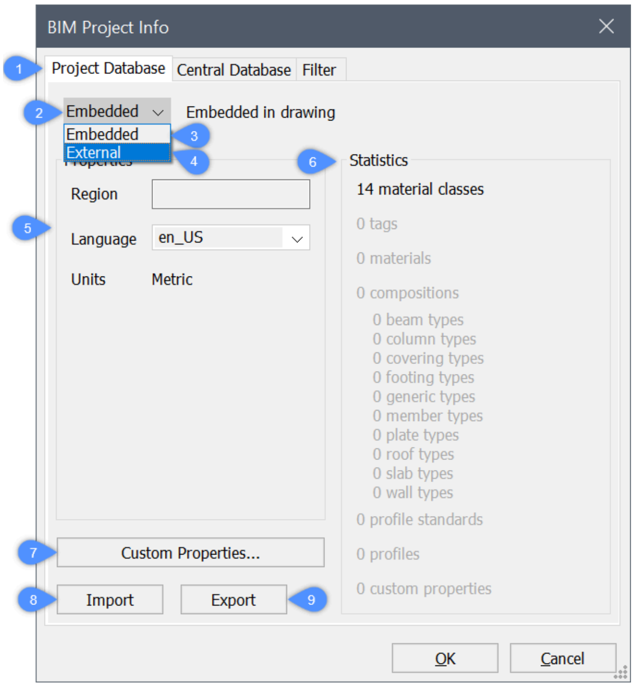
23.8.1 About

The **BIM Project Info** dialog box shows information about properties, materials, compositions, profiles, and spatial structures of the Project and Central databases. It allows you to import or export the database and to set the project database as embedded or external. When you launch the BIMPROJECTINFO command in BricsCAD, the **BIM project info** displays in a dialog box that includes components and tabs. You can change project and central databases of BIM models using this dialog box.

For more information about the Project and Central databases, see the article **BIM data structure**.

23.8.2 Setting the project database

1 Open the **BIM Project Info** dialog box by typing BIMPROJECTINFO in the Command line and select the Project Database tab:



Project Database (1)	Shows the BIM project info.
Project database drop-down list (2)	Choose between **Embedded** or **External**.
Embedded (3)	Saves the project database in the drawing file.
External (4)	Saves the project database in a BIM database file (BSYSLIB).
Properties (5)	Shows the properties of the BIM project e.g. region, language, and units.
Statistics (6)	Shows the properties, materials, compositions, profiles, and spatial structures (the amount of buildings, stories, etc.) of the current project library of the BIM model.
Custom Properties... (7)	Adds custom properties to materials and compositions.
Import (8)	Imports the selected XML and CSV library files which may add materials, compositions or profiles to the project database.
Export (9)	Saves XML library files in the defined location.

2 To change the project database from **Embedded** to **External** or vice versa, select the desired option in the Project Database drop-down list.

Note: The external database keeps the database independent from the current drawing. This is especially useful if you want to share the database between multiple models, e.g. when a model consists of several Xref drawings.

- When switching from **Embedded** to **External**, the **Select or create a Bricsys Library** dialog box displays.

  Do one of the following:

- Type a name in the **File Name** field to export the content of the project database to a new BSYSLIB file, then click the **Open** button.
- Select an existing BSYSLIB file, then click the **Open** button.

  The current content of the project database is merged with the selected BSYSLIB file, including the imported libraries.

- When switching from **External** to **Embedded**, the content of the external database is copied to the embedded project database.

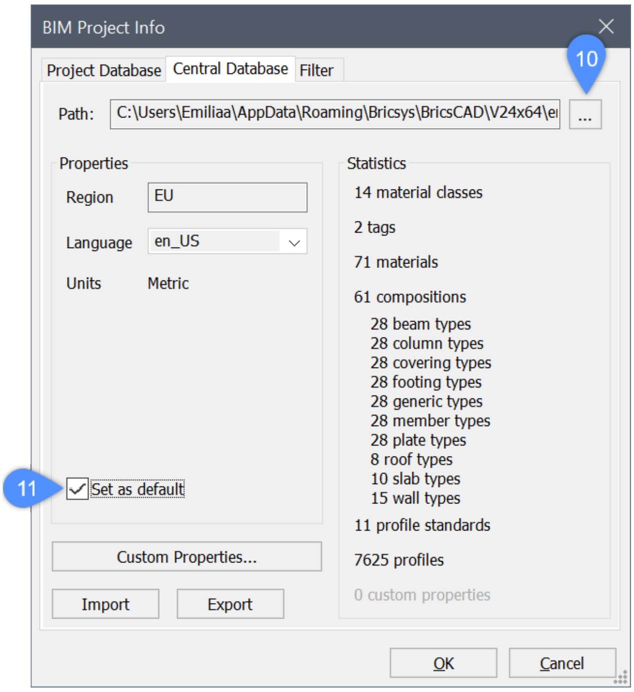
3 To specify the Project Database properties such as Region and Language, fill out the **Region** field and choose one of the language options in the drop-down list.

4 Optionally, use the custom properties button to add material and composition custom properties in the Project Database.

5 Press the **OK** button to accept it.

23.8.3 Setting the Central database

1 Click the **Central Database** tab in the **BIM Project Info** dialog box:

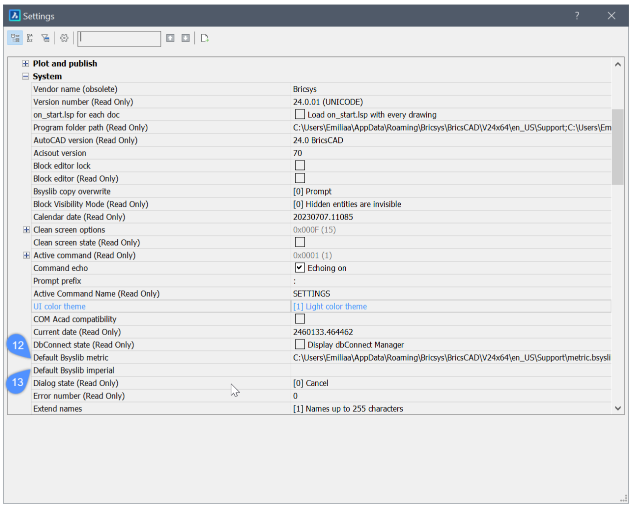


2 To select a BIM database file (BSYSLIB), click the **Browse** (10) button of the **Path** field, then choose a

database file in the **Select or create a Bricsys Library** dialog box.

3 Optionally, tick the **Set as default** (11) option to set the selected database as default library database for next projects.

Note: You can edit the default library settings (12, 13) in the **Settings** dialog under **Program options** > **System**.



23.8.4 Adding profiles by importing XML or CSV files in the project database (BSYSLIB)

1 In the **Project Database** tab, press the **Import** button that opens the **Select xml/csv file to import** dialog box.

2 Choose the XML or CSV Library files and press the **Open** button.

The CSV Library files that add profiles must respect the below format:

The first row indicates the profile shape name (AsymmetricIShape, CShape, Circle, CircleHollow, CraneRailAShape, CraneRailFShape, Ellipse, IShape, LShape, Rectangle, RectangleHollow, RoundedRectangle, Trapezium, TShape, UShape, ZShape).

The second row must contain the names of the profile properties. The first 5 names are general profile properties (standard, name, type, size, material). The next names are the specific parameters for each shape of profiles.

// or # - can be used to add comment to the CSV files, lines starting with these characters will be ignored.

For example, a CSV file for import C-shape profiles should contain at least the following:

CShape

standard	name	type	size	material	depth	width	wallThickness	girth

General columns for all shapes are:

standard	name	type	size	material
		Available types: Electrical, Generic, HVAC, Structural Steel, Structural Concrete, Piping		Available materials: Only existing in project or in library (use BLMATERIALS to check). Mostly used - Steel.

Note: Type and material names are case sensitive.

Note: Empty fields are not yet accepted.

Right of the general columns are the columns that indicate the specific parameters for each shapes of profiles.

Below are the shapes of profiles with the specific parameters (in bold) necessary to be indicated for a successful import. The meaning of each parameter can be better understood from the profiles preview (BIMPROFILES/New profiles).

Shape of profile	parameters		Shape of profile	parameters
AsymmetricIShape	**overallWidth** **overallDepth** **webThickness** **flangeThickness** filletRadius **topFlangeWidth** topFlangeThickness topFlangeFilletRadius centreOfGravityInX		LShape	**depth** **width** **thickness** filletRadius edgeRadius legSlope centreOfGravityInX
CShape	**depth** **width** **wallThickness** **girth** internalFilletRadius centreOfGravityInX		Rectangle	**xDim** **yDim**

Shape of profile	parameters		Shape of profile	parameters
Circle	**radius**		RectangleHollow	**xDim** **yDim** **wallThickness** innerFilletRadius outerFilletRadius
CircleHollow	**radius** **wallThickness**		RoundedRectangle	**xDim** **yDim** **roundingRadius**
CraneRailAShape	**overallHeight** **baseWidth2** **radius** **headWidth** **headDepth2** **headDepth3** **webThickness** **baseWidth4** **baseDepth1** **baseDepth2** **baseDepth3** centreOfGravityInY		Trapezium	**bottomXDim** **topXDim** **yDim** **topXOffset**
CraneRailFShape	**overallHeight** **headWidth** **radius** **headDepth2** **headDepth3** **webThickness** **baseDepth1** **baseDepth2** centreOfGravityInY		TShape	**depth** **flangeWidth** **webThickness** **flangeThickness** filletRadius flangeEdgeRadius webEdgeRadius webSlope flangeSlope centreOfGravityInY

Shape of profile	parameters		Shape of profile	parameters
Ellipse	**semiAxis1** **semiAxis2**		UShape	**depth** **flangeWidth** **webThickness** **flangeThickness** filletRadius edgeRadius flangeSlope centreOfGravityInX
IShape	**overallWidth** **overallDepth** **webThickness** **flangeThickness** filletRadius		ZShape	**depth** **flangeWidth** **webThickness** **flangeThickness** filletRadius edgeRadius

An example of such a CSV file (templates_of_mostly_used_profiles.csv) for the most used profiles is stored in your **Support** folder in the following path:

C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\Bim\Profiles.

The **Appdata** folder is a hidden folder by default, you may have to unhide folders first. This path can also be accessed by the SUPPORTFOLDER command.

23.8.5 The project filter

You can attach any number of tags to each Physical Material or Composition. Any string can be entered as a tag.

Each of the tags can be used as a filter on the database. There is a button at the top of each list ( ), to switch the filter on or off. If the filter is on, only Compositions and Physical Materials that match at least one of the filter tags will be listed.

The **Filter** button has three states:

- If the project filter is empty, the button is not active (**BIM Compositions** panel) or is not available (**Materials** and **Compositions** editors).
- The project filter is defined, but not active.

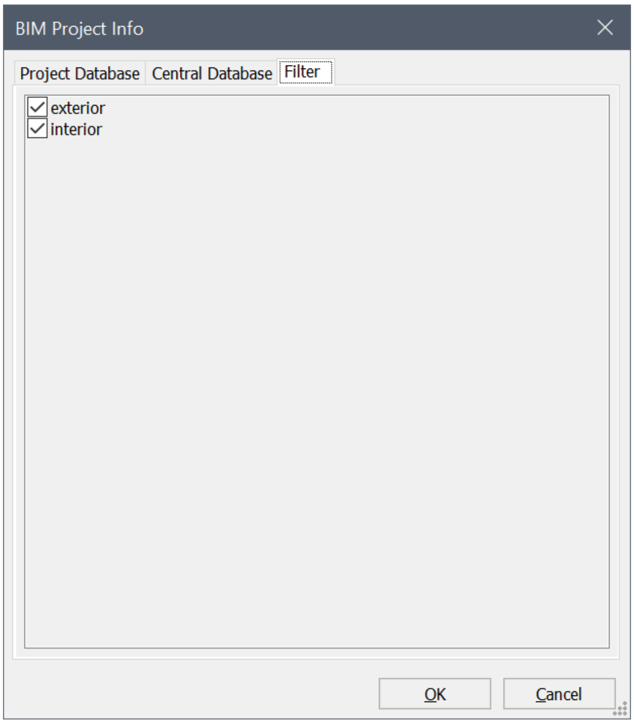
- The project filter is defined and active.

A tooltip shows the current project filter.

23.8.6 Setting the project filter

1 Select the **Filter** tab page on the **BIM Project Info** dialog box.

2 Check the tags you want to use in the filter.



23.9 Using Python scripts

23.9.1 Commands

BIMPYTHON

23.9.2 About BIMPYTHON

The BIMPYTHON command enables you to query and manage data from a model within BricsCAD BIM, with a suitable .py Python script.

The Python scripts can be as simple as obtaining quantities and associated properties of objects and BIM entities, or as elaborate as implementing a series of calculations based on the model parameters available.

BricsCAD does not ship with a Python Shell, so the scripts would need to be prepared in a text or code editor application.

The Python Programming Language, together with its standard libraries, is embedded within BricsCAD BIM so there is no need for you to install them separately unless you have custom packages and libraries which you wish to use in your scripts.

Several Python scripts samples are available in the BricsCAD BIM installer.

23.9.3 Procedure: setting up a Python script with the BIMPYTHON module

1 Before gaining access to the API, import or 'call' any desired modules you wish to have in the script.

1 If they are not part of the Python Standard library, make sure they have been installed separately beforehand.

You can "import" a standard module, such as math and "import as" an external one, for example Pyplot, which is a collection of functions in the Matplotlib package:

```
import math
import matplotlib.pyplot as plt
```

2 With your desired libraries "called" in, the next step is to gain access to the API in order to query the elements within the BricsCAD model. This is the range of bim objects and serves as an entry point to the model.

Import the current_model from bricscad module:

```
from bricscad.bim import current_model
```

3 Now, you can query the model with these example statements.

```
# Display info about the lengths of walls in the model
lengths = [wall.prop('Length') for wall in
 current_model().filter(Type='Wall')]
print(f'wall lengths. max: {max(lengths)}, avg: {sum(lengths)/len(lengths)}')
```

```
# Create a selection and print the objects
current_model().filter(Type='Wall', IsExternal=True).select()
for wall in bim_model.filter(Type='Wall', IsExternal=True,
 Length=max(lengths)):
    print(wall)
```

The mappers of the bricscad.bim.Objects are chainable. This is an example to get the parts of the roof that are close to a wall:

```
# Get all parts of roof within 40cm range of all walls
roof_parts = current_model().filter(Type='Roof').parts()
roof_parts_close_to_wall = current_model().filter(Type='Wall').within_distance(40,
 'cm', search_range=roof_parts)
```

Whereas this is an example of a function statement to filter:

```
# Filter roof parts longer than 50 project units
def is_long(obj):
    return obj.prop('Length') > 50
roof_parts.filter(is_long)
```

You can also export and show data in various formats:

```
# create a dictionary list
wall_info = [
    {   'Handle': wall.get_property('Handle'),
```

```
        'Length': wall.get_property('Length'),
        'Height': wall.get_property('Height')
    } for wall in current_model().filter(Type='Wall')]
```

```
# export to .json
import json
file = open('path/to/file.json', 'w+')
file.write(json.dumps(wall_info, indent=4))
file.close()
```

```
# plotting a histogram
import matplotlib.pyplot as plt
import pandas as pd
df = pd.DataFrame(wall_info)
df.hist();
plt.show()
```

```
# export to .csv
df.to_csv(r'path/to/file.csv', index = False, header=True)
```

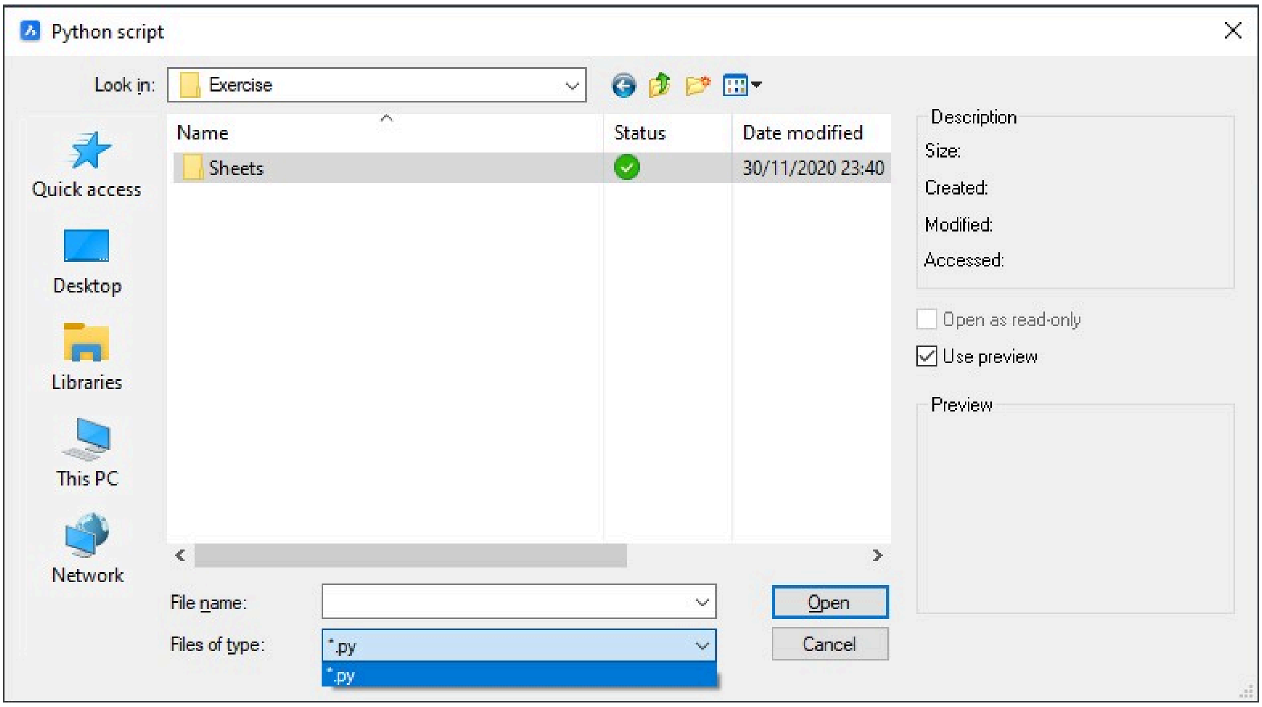
For more information on the BricsCAD API and its various classes, please visit the API chapter below.

23.9.4 Procedure: executing the Python script

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Type BIMPYTHON in the command-line and press Enter.

3 A dialog box is displayed where you can only select a Python script file (*.py). Select it and click **Open** to execute the file.



4 Unless you have specified the data to be exported or displayed in an external program outside BricsCAD, BricsCAD reports the output in the Command line.

23.9.5 Reference

Classes and syntaxes

get_property(*prop_name*)
Returns the property value of the Object with the given name.

set_property(*prop_name, value*)
Sets a value for the property name (prop_name).

distance_to(*other_obj, units='mm', distance_mode='exact'*)

- Calculate the distance between two Objects.
- Options for argument *units*: any units value form insunits ('Centimeters', 'Feet', 'Parsecs' etc.) and these abbreviations: 'mm', 'cm, 'm', 'km', 'ft'.
- Options for argument *distance_mode*: 'bbox_center', 'bbox', exact'.

parts()
Return the sub-elements of this object.

parent()
The opposite of parts(), returns the parent object of this sub-element.

plies()
Returns the associated **Plies** iterable object.

within_distance(*distance, unit='mm', distance_mode='exact', search_range=bim_model*)

- Return the objects that are within the distance of argument *distance*.
- Options for argument *units*: see distance_to.
- Options for argument *distance_mode*: see distance_to.

openings()
Returns the associated openings of this object.

spaces()
Returns the associated spaces of this object.

bounding_elements()
Returns the associated bounding elements of this (space) object.

select()
Add Object to selection.

deselect()
Remove Object from selection.

__eq__() and __hash__()
Makes Object interoperable with e.g. python set or dictionary.

***class* bricscad.bim.Objects**
Defines collection of BricsCAD BIM Objects

filter(*function*)
Filtering this range with a function parameter.

filter(*****conditions*** **)**
Filtering this range with conditions given as keyword arguments.

parts()
Return the parts of all the elements in this range.

parents()
Return the parent objects of elements in this range.

within_distance(***distance, unit='undefined', distance_mode='exact', search_range=bim_model*** **)**
Returns those objects that are within distance to any object in this range.

openings()
Returns the associated openings of these objects.

spaces()
Returns the associated spaces of these objects.

bounding_elements()
Returns the associated bounding elements of these (space) objects.

select()
Adds Objects to selection.

deselect()
Removes Objects from selection.

__len__ ()
Returns the number of Objects in this range.

bricscad.bim.list_properties(*obj*)
Returns the list of available properties of the passed entity or ply.

bricscad.bim.current_model()
Returns the Objects in the modelspace of the active document.

***class* bricscad.bim.Plies**
Indexed, sliceable container of **Ply** objects.
__iter__()
__getitem__(***index*** **)**
__getitem__(***slice*** **)**
__len__()

***class* bricscad.bim.Ply**
Individual ply object.

get_property(*prop_name*)
Returns the property value of the **Ply** object with the given name.

23.10 Classification codes

23.10.1 About

Assigning classification codes organizes the BIM and increases the level of detail. For instance, they allow you to organize library materials and project reports, estimate project costs and define product specifications. Currently, **GUBIMClassCA**, **GUBIMClassES**, **Master Format**, **NLSfb**, **OmniClass**, **UniClass2015**, **Uniformat**, and **VMSW** are available classification systems in BricsCAD.

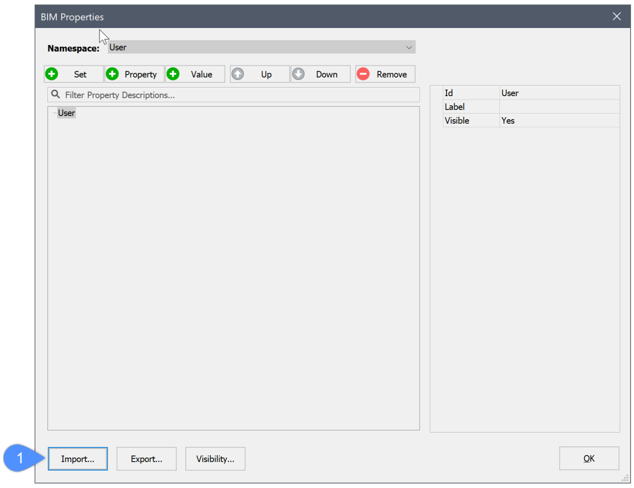
For more information, visit the Command Reference article BIMPROPERTIES.

23.10.2 Assigning classification codes to the BIM

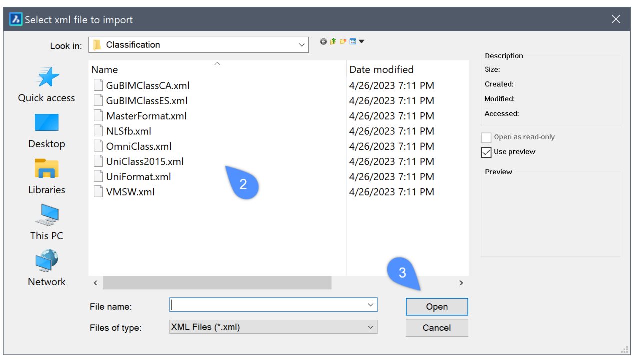
1 Click the **Properties** button from the Quad (**Bim** tab) to open the BIM Properties dialog box.

Note: The Quad should be in **No Selection** state.

A dialog box appears:



2 Click the **Import** (1) button at the bottom-left corner of the dialog box. A file dialog displays:



3 Choose the classification system on the list (2), and click **Open** (3) to import it into the BIM.

Note: If the file dialog does not show the **Classification** folder, you should manually enter this path in the address bar: *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD V25x64\en_US\Support\Bim\Classification*.

Available classification systems:

GUBIMClassCA	Classification system for the BIM Users group of Catalonia.
GUBIMClassES	Unified classification system of building elements designed for the AEC industry in Spain.
MasterFormat	Standards for commercial and institutional building projects in the US and Canada.
NLSfb	Standards for the AEC industry in the Netherlands.
OmniClass	Classification system for the construction industry.
UniClass2015	Standards for all sectors of the UK construction industry.
Uniformat	Standards for classifying building specifications, cost estimating, and cost analysis in the US and Canada.
VMSW	Standards for the AEC industry in Belgium.

4 The classification system is added to the **namespace** (4), the selected classification system is added as a property set (5), and the classification code (6) is attached to this property set.

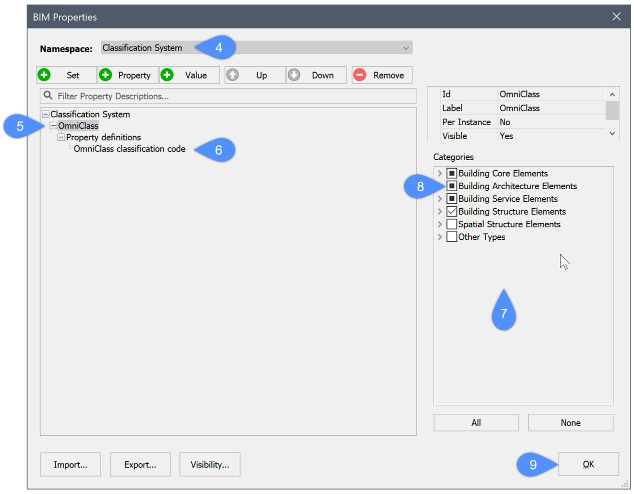
By default, the classification system is applied to all sorts of building elements (7).

Optional: Deselect the categories of building elements to which the classification system is applied.

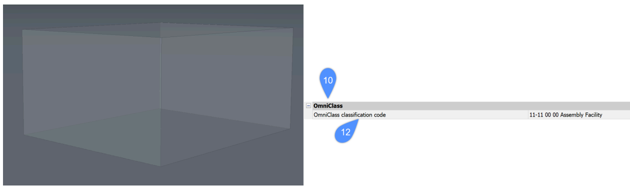
To do so, tick the checkbox next to the category name (8) (all elements in this category will be excluded), or expand the list and check the box next to the element type only.

Click the **OK** button (9) to finalize the procedure.

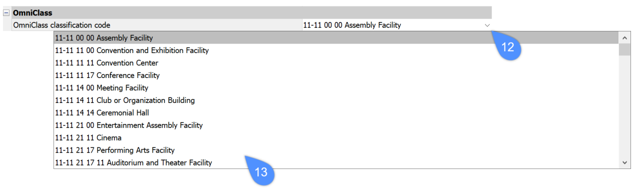
Or import another classification system following the previous steps.



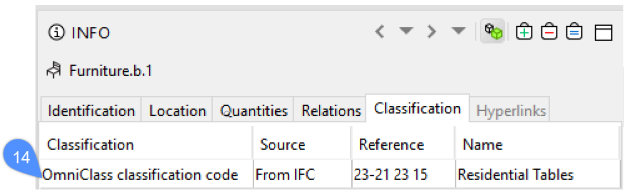
5 Select the element to which you want to assign a classification code, and open the **Properties** panel. The classification system (10) and code (11) should be available in the **Properties** panel.



6 Click the arrow icon on the right-hand side (12) and select the classification code for the selected element in the context menu (13).

7 

Note: Exporting the BIM to an IFC file contains the classification system for each element with assigned classification codes. These properties are exported with proper labels and descriptions. The following image shows the OmniClass Classification (14) in Solibri after exporting and importing BIM with the **OmniClass Classification code** from Bricsys to Solibri.



23.11 Creating and editing BIM spaces

23.11.1 Commands

BIMSPACE, BIMUPDATESPACE, BIMIFY

23.11.2 About

You can create BIM Spaces from enclosed spaces. When the wall elements enclose a space in your model, you can use the **BimSpace** tool to define a BIM Space. These spaces are solids so they can be easily manipulated. The 3D space geometries added to the current model are associated with space tags. The space tag is a block that contains a polyline and attributes. The default attributes of the space tag include a space **Name** and a space **Number**. The values of each attribute can be changed using the **Properties** panel.

For more information about this command, visit the Command Reference article **BIMSPACE command**.

23.11.3 Defining a BIM space

1 Select the **Space** tool under **Home** > **Classify** > **Space** or type BIMSPACE in the Command line.

You are prompted:*Pick a point inside the space or [Edit];*

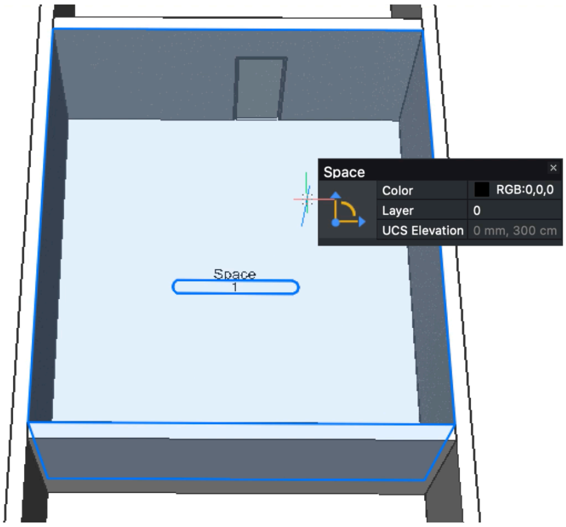
2 To define a BIM Space, pick a point inside the space area in your BIM model.

3 Move the cursor inside the space area to pick a point, the **Dynamic UCS** enables you to define the bottom plane of the space. A green checkmark will appear next to the cursor to indicate a possible location. A red cross will appear if the plane isn't available.

4 In this plane, an area enclosed by wall elements or any space bounding entities is found and a space tag is placed in the center of the space.

5 ***Note:*** Space bounding entities are 3D solids or linear 2D entities, which are classified as a BIM entity and having **Space bounding** property set to **On**. If the **Centerline** property of a space bounding solid is set to **On**, the center of the solid is used as the space boundary.

Space bounding entities can be walls, curtain walls, slabs, columns, but also hatches and polylines can be counted as space bounding entities.



6 Edit the properties of the BIM Space in the **Properties** panel.

Change the **Representation** property from **Solid** to **Footprint** for a less notable visualization.

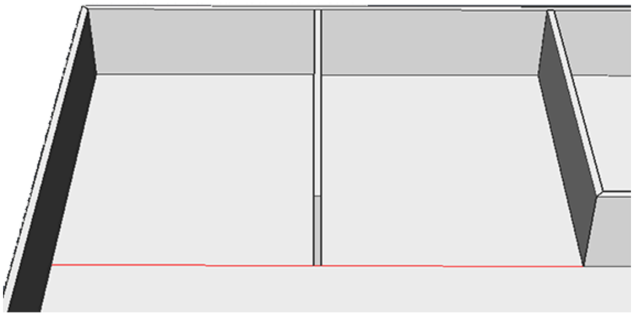
Note: Editing the **Building** and **Story**properties of the BIM Space overrides the corresponding properties of the elements that are contained in that space (that is have that BIM Space set as their **Space** property).

23.11.4 Procedure: creating BIM spaces from virtual boundaries

There are cases in which a virtual boundary is needed. For example, separate parking spaces, large auditorium spaces for energy analytics, etc.

1 Divide the Space with linear entities. This can be hatches or lines.

1



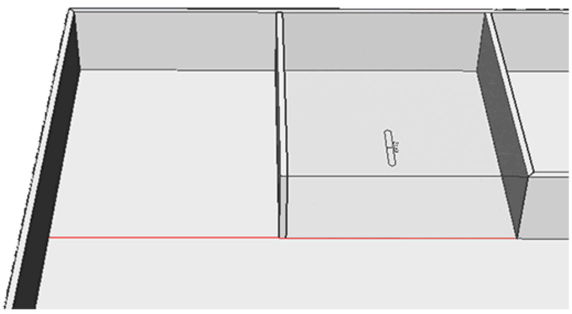
2 Select the dividing entities (the red line).

3 Classify them as a space bounding entity: **Home** > **Classify** > **Wall**.

Or in the Quad under **BIM**.

4 Type BIMSPACE in the Command line and pick a point inside the space.

5



Note: If the line doesn't divide the space, go to the **Properties** panel and then to **BIM**. Check if **Space bounding** property is turned **On**.

23.11.5 Creating BIM spaces automatically

BIMIFY and BIMQUICKBUILDING create BIM Spaces and automatically assign information to them (e.g., the story and building of the space; see BIMSPATIALLOCATIONS). The commands will search for space bounding entities and create spaces wherever possible.

These spaces act like manually made spaces and can be edited according to your own needs.

23.12 Custom properties

23.12.1 About

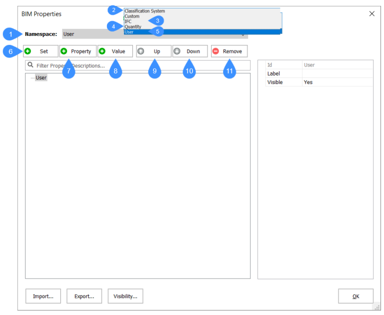
BricsCAD provides a way to assign your own properties to desired building elements in a single command. This is called assigning custom properties. The approach is first to create a custom property set, then assigning it to the building elements to which the property set is to be applied. The method is carried out by the BIMPROPERTIES command, which allows you to create, edit, and delete the custom property set in a dialog box. This dialog box also allows you to add properties to the currently selected property set. Important to note is that these custom properties are also included in IFC export.

For more information about this command, visit the Command Reference article **BIMPROPERTIES command**.

To learn more about creating user-defined quantity definitions, visit the Procedure article for **Customizing BIM Quantities**.

23.12.2 Using the BIM properties dialog box

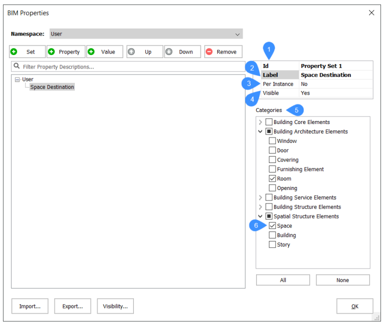
1 Open the **BIM Properties** dialog box by typing BIMPROPERTIES in the Command line. The dialog box displays for specifying and editing properties of entities in BIM projects.



(1) Namespace	Groups property sets in namespaces.
(2) Classification System namespace	Shows the classification system namespace.
(3) IFC namespace	Shows the IFC2x3 and IFC4 properties that are merged.

(4) Quantity namespace	Shows some hardcoded quantities
(5) User namespace	Shows the custom properties.
(6) Add set button	Creates a new property set.
(7) Add property button	Adds a new property to the currently selected property set.
(8) Add value button	Creates a new value definition for the currently selected property set.
(9) Up button	Moves the selected property or value up the list.
(10) Down button	Moves the selected property or value down the list.
(11) Remove button	Removes the selected property set, property definition or value definition.

2 To create a custom property set, select the **User** option in the namespace drop-list, then click the **Add Set** button (6). A new property set is added to the **User** space. To edit the property set do the following:



- Type a name in the **Id** (1) field.
- Type a name in the **Label** (2) field.

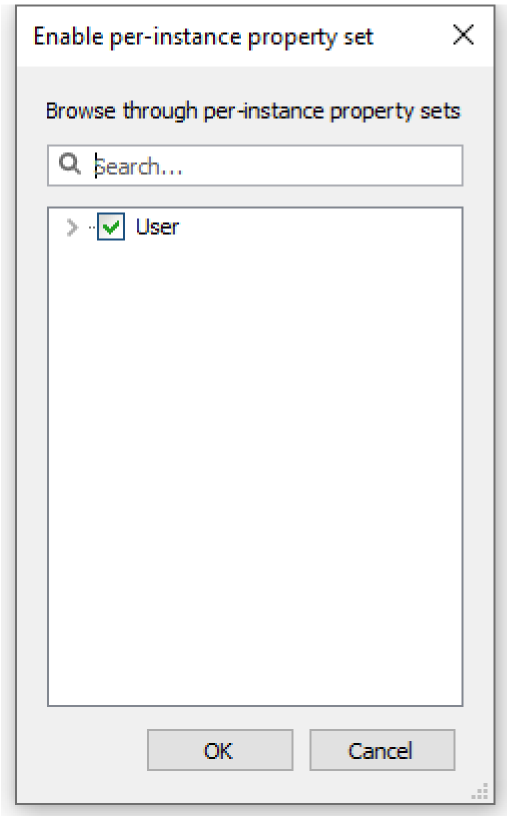
The **Label** name appears in the properties tree. If the **Label** name is not defined, the **Id** name displays instead. The **Id** must be unique.

Optionally, you can choose to have the custom property be only applied to specific instances (3), and choose the visibility (4).

Note: To enable the **Per Instance** property, do the following:

- Click the **BIM/Entity property sets** property in the **Properties** panel.
- Click the **Browse** button.

The **Enable per-instance property set** dialog box displays:



- Check the **User** namespace, then click the **OK** button

Select the building element categories (5) the property set applies to.

Expand one of the categories to see the different sub-categories of building elements. Check the box next to desired one (6) to associate it with the property set.

3 To add a property to the currently selected property set, first, select a property set, then click the **Add Property** button.

To edit the property do the following:

- Type a name in the **Id** (1) field.
- Type a name in the **Label** (2) field.
- Optionally, type a description in the **Description** (3) field.
- Select the property **Type** (4). The options are

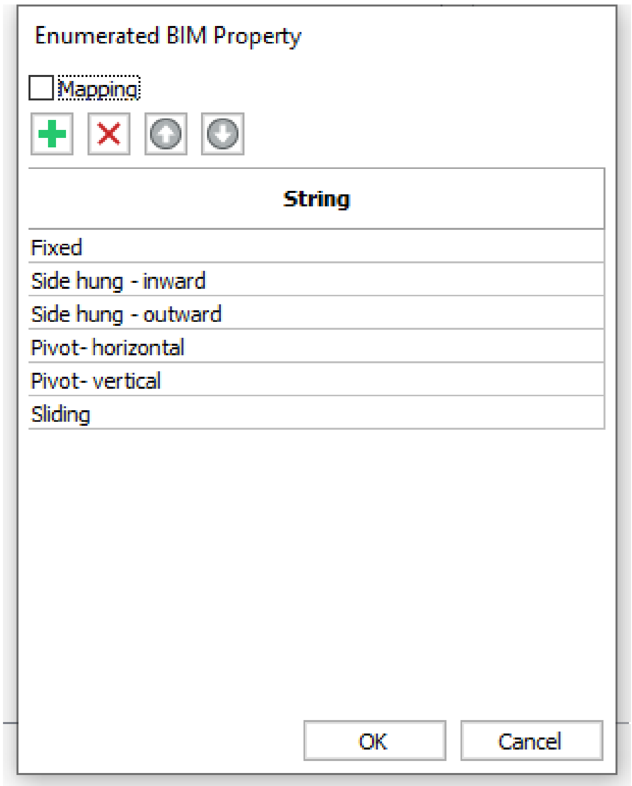
  Boolean: an ON or OFF value.

  Integer: a whole number between -2147483648 and 2147483647.

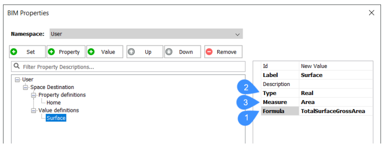
  Real: an approximation of a real number.

  String: a sequence of characters, can be both numbers and letters.
- Optionally, define enumerated values (5), that is a fixed number of possible values for custom BIM properties. These values can be selected in a drop-down list in the **Properties** panel.

Optionally, check the **Mapping** option to add a description for each value.



4 To add a new value to the currently selected property set, first, select a property set, then click the **Add Value** button.



You can edit the **Formula** (1) property by typing a value, according to the selected value type. You can use the syntax of the **Formula** field of the corresponding BIM entity type in the **Quantity** namespace to compose a formula.

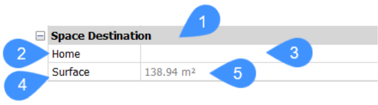
Note: When the **Type** property (2) is **Real**, **Measure** property (3) becomes available.

- **Measure**: Sets the dimension of the value: **Length**, **Area**, **Volume** or **Mass**.

5 To remove the property set/property, select a property set/property in the tree, then click the **Remove** button.

6 Press the **OK** button to accept it.

Now if you select a building element which is associated with the previously set category/sub-category and open the **Properties** panel, the new property set is displayed (1) with its newly assigned property definition (2) with no value (3) and its newly assigned value definition (4) with its value (5).



To assign a value to your property definition, select it in the **Properties** panel, then type a new value in its field (3).

23.13 Customizing BIM quantities

23.13.1 About

Entities have a set of properties which can be viewed in the **Properties** Panel. If an entity has been assigned a certain **BIM classification** or **Type** (e.g. **Wall** or **Slab**), additional properties will be displayed such as IFC Common properties and Base quantities. Which properties are displayed in the panel is managed in the **BIM Properties** dialog box, which is opened by the BIMPROPERTIES command.

23.13.2 Namespaces

In the BIM Properties dialog box, by default three namespaces are available: **IFC** (1), **Quantity** (2) and **User** (3). The **IFC** namespace contains properties that are defined by the IFC2x3 and IFC4 schemes, that are merged automatically in background during IFC Import or Export. This contains both common properties (e.g. Wall Common properties, such as **Fire Rating** and **Acoustic Rating**), and Quantities (e.g. **Wall Quantities** > **Nominal Height** (4)). These Quantities are based on so-called Core quantities (5) of the solid. See Core quantities for more information.

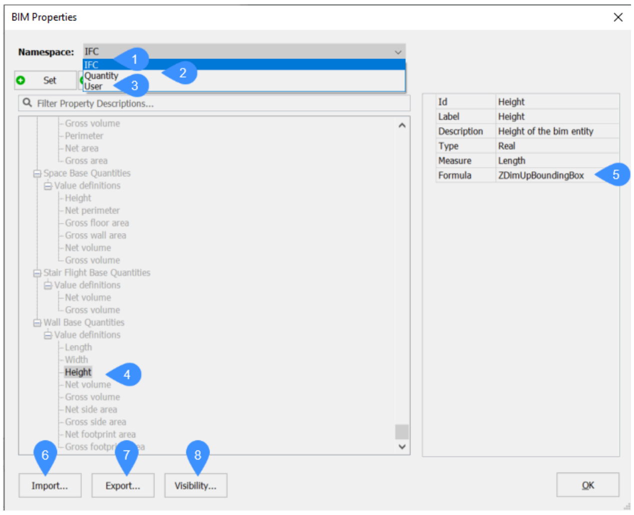
The **Quantity** namespace contains some hard-coded quantities of BIM objects also based on the Core quantities, much like in the **IFC** namespace.

Note: This **Quantity** namespace is deprecated. It has been replaced by the Quantities in the **IFC** namespace, however it is still available in the **BIM Properties** dialog for backward compatibility with versions V20 and older.

The **User** namespace exists for the user to create their own property definitions. See also the Procedure article Custom Properties for more information on custom properties.

You can export all property sets (properties/quantities) defined in the drawing to an XML file by clicking the **Export** button (7) on the bottom left. You can then import this file in another drawing.

The visibility of namespaces or attribute sets can be toggled on and off according to your preferences by clicking the **Visibility** button (8).



Note: Using this method the namespace will only be imported for this particular project. If you want namespaces to be imported over all your projects, open the **Settings** dialog and under the **BIM** > **General** > **Default properties path** you can add a new path. By default, it should take you to *C:\Program Files \Bricsys\BricsCAD V25 en_US\Support*, where you can find for instance the **bimproj_IFC.xml** containing the **IFC** namespace.

23.13.3 Core quantities

Core quantities are quantities of solids that are calculated based on the geometric features of that solid, and that can be used in user-defined quantities. These core quantities are also the basis for the quantities defined in the **IFC** properties as described in the previous section.

For example, under **Wall Base Quantities** in the **IFC** namespace, the **Nominal Height** of a wall is defined as the **ZDimUpBoundingBox** of the solid, which is a Core quantity. A full list of Core quantities can be found below.

Core Quantity	Description	Measure	Type
NumberOfPlies	Number of plies in the attached composition.		**Integer**
XDimUpBoundingBox	X dimension of the bounding box found with global Z axis rotation freedom.	**Length**	**Real**
YDimUpBoundingBox	Y dimension of the bounding box found with global Z axis rotation freedom.	**Length**	**Real**
ZDimUpBoundingBox	Z dimension of the bounding box found with global Z axis rotation freedom.	**Length**	**Real**

Core Quantity	Description	Measure	Type
XDimFreeBoundingBox	X dimension of the bounding box found with 3D rotation freedom.	**Length**	**Real**
YDimFreeBoundingBox	Y dimension of the bounding box found with 3D rotation freedom.	**Length**	**Real**
ZDimFreeBoundingBox	Z dimension of the bounding box found with 3D rotation freedom.	**Length**	**Real**
DistanceBetweenMajorSurfaces	Distance between major surfaces.	**Length**	**Real**
FirstMajorSurfacePerimeter	Perimeter of the first (largest) major surface.	**Length**	**Real**
SecondMajorSurfacePerimeter	Perimeter of the second major surface.	**Length**	**Real**
LinearAxisLength	Length of the linear element's axis.	**Length**	**Real**
VariablePlyThickness	Thickness of a variable ply.	**Length**	**Real**
FirstMajorSurfaceNetArea	Area of the first (largest) major surface after subtracting openings.	**Area**	**Real**
FirstMajorSurfaceGrossArea	Area of the first (largest) major surface before subtracting openings.	**Area**	**Real**
SecondMajorSurfaceNetArea	Area of the second major surface after subtracting openings.	**Area**	**Real**
SecondMajorSurfaceGrossArea	Area of the second major surface before subtracting openings.	**Area**	**Real**
IsMajorSurfacesParallel	Indicator if the major surfaces are parallel.		**Boolean**
TotalSurfaceNetArea	Area of the solid's surfaces after subtracting openings.	**Area**	**Real**
TotalSurfaceGrossArea	Area of the solid's surfaces before subtracting openings.	**Area**	**Real**

Core Quantity	Description	Measure	Type
FootprintNetArea	Area of the bottommost solid's surfaces after subtracting openings.	**Area**	**Real**
FootprintGrossArea	Area of the bottommost solid's surfaces before subtracting openings.	**Area**	**Real**
ProjectedNetArea	Area of the solid's surfaces projected to global XY plane after subtracting openings.	**Area**	**Real**
ProjectedGrossArea	Area of the solid's surfaces projected to global XY plane before subtracting openings.	**Area**	**Real**
CrossSectionNetArea	Area of the cross section for linear elements after subtracting openings.	**Area**	**Real**
OuterSurfaceNetArea	Area of the outer surface for linear elements after subtracting openings.	**Area**	**Real**
NetVolume	Volume of the solid after subtracting openings.	**Volume**	**Real**
GrossVolume	Volume of the solid before subtracting openings.	**Volume**	**Real**

23.13.4 User-defined quantities

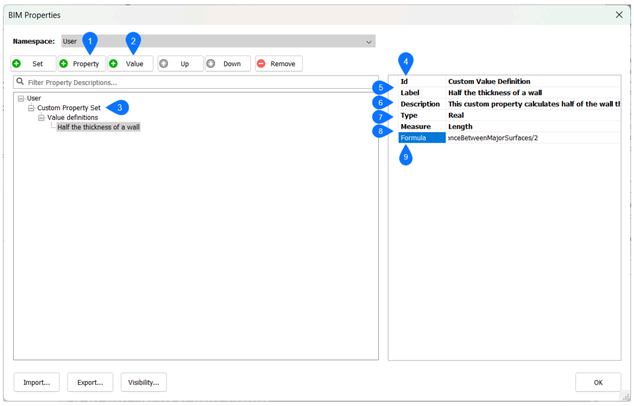
In the **User** namespace it is possible to create custom properties inside a custom property set using the **Property** (1) button, see the **Procedure** article > **Custom Properties**.

It is also possible to create user-defined quantities, using the **Value** (2) button. In the image below, a user-defined quantity was created in a **custom property set** (3). This user-defined quantity (or Value Definition) has the following attributes:

- **Id** (4): a unique identifier.
- **Label** (5): the name of quantity which will show up in the properties panel. This can be the same as the ID, however **the ID must be unique**.
- **Description** (6): a description of the quantity.
- **Type** (7): the type of the quantity. Typically, a quantity will be defined as a **Real** value.
- **Measure** (8): the measure of the quantity, i.e. is it a **length** value, an **area**, **volume** or **mass**?

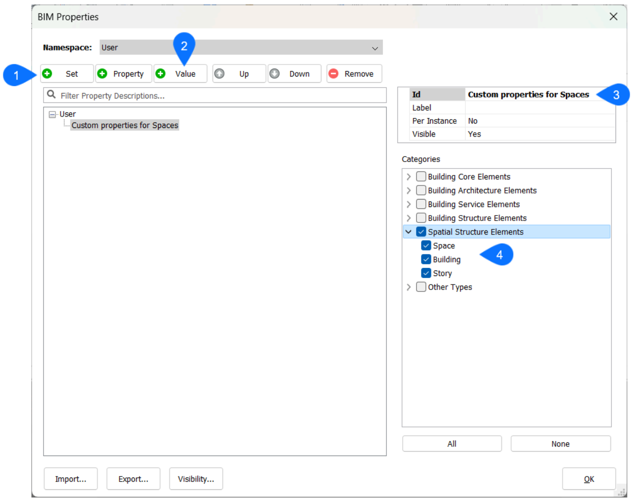
- **Formula** (9): the actual expression of the quantity. Here Core quantities can be used, as well as mathematical expressions as defined in the Procedure article Working with Parameters and Constraints.

In this example, a user-defined quantity is defined that displays half the thickness of a wall, which is calculated as **DistanceBetweenMajorSurfaces/2**.



23.13.5 Procedure: creating a user-defined quantity

In this part you will learn how to make user-configurable quantities. By way of example, we will define a quantity of Spatial objects such as a room. The quantity will be the **Compactness** which can be calculated as the volume of the space divided by the sum of the areas on all sides of the space.



1 Open the **BIM Properties** dialog box by typing BIMPROPERTIES in the Command line.

2 Set the **User** namespace as current namespace.

3 Create a new Property Set by clicking the **Set** button (1), and give it a name (3).

4 Assign the Property Set to one or more categories (4). In this example we will assign it to spatial elements.

5 Create a new Value Definition by clicking the **Value** button (2).

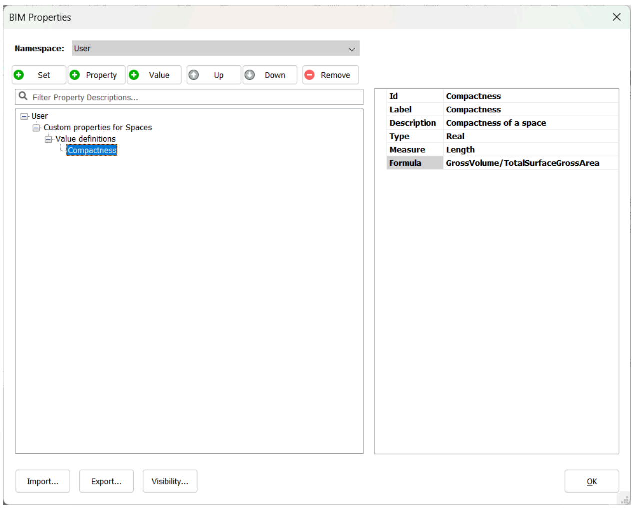
6 Give this Value Definition an **Id** and a **Label**, e.g. **Compactness**.

7 **(Optional)** Give this Value Definition a description.

8 Assign the correct **Type** to this Value Definition. Because our Value Definition will be based on a calculation of two **Real** values, this will also be a **Real** value.

9 Assign a **Measure** to the Value Definition. Because the result of the calculation will be a volume divided by an area, we will end up with a **Length** measure.

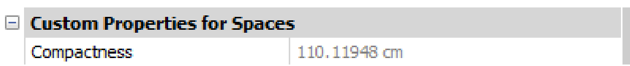
10 Fill in an expression in the **Formula** field. The volume of a space is a Core quantity with the name **GrossVolume**. The sum of areas of the space on all sides is also a Core quantity with the name **TotalSurfaceGrossArea**. Hence, we can define the **Compactness** = **GrossVolume/ TotalSurfaceGrossArea**.



11 Click **OK** to close the **BIM Properties** dialog box.

12 Create a volume that represents a space, either by creating surrounding elements like walls and a slab, and by defining a space inside using the BIMSPACE or BIMIFY commands, or by creating a solid and using BIMCLASSIFY to classify it as a **Space**.

13 Select the space. In the **Properties** panel, you should now see your user-defined property under a separate property set:



23.14 Load bearing direction for slabs

23.14.1 Commands

BIMSETLOADBEARINGDIRECTION

23.14.2 About BIMSETLOADBEARINGDIRECTION

In BricsCAD BIM, the BIMSETLOADBEARINGDIRECTION command allows you to define 0, 1, or 2 load bearing directions for a slab.

After defining the load bearing direction in the 3D model, you can indicate the load bearing direction in the floorplan using the **Manual** option of the BIMTAG command.

23.14.3 Specifying the load bearing direction(s)

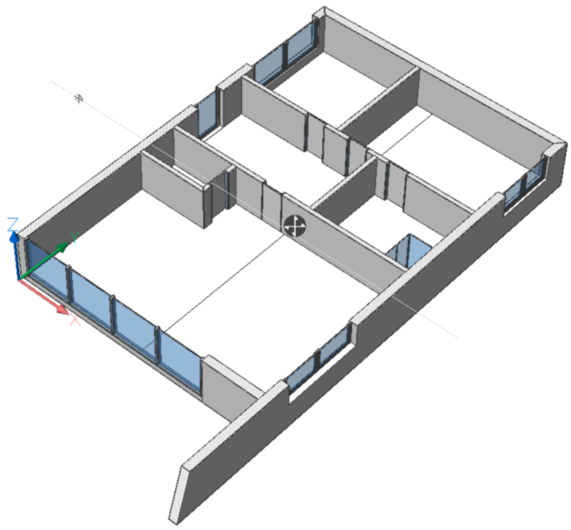
Only solids which are classified as a **BIM Slab** can be processed by the BIMSETLOADBEARINGDIRECTION command. Use the BIMIFY or BIMCLASSIFY command to classify a solid as a **BIM Slab**.

1 Launch the BIMSETLOADBEARINGDIRECTION command.

1 **You are prompted:** Select a slab

2 Click the slab. It may help to set the **Clip Display** property of a plan section to select the slab.

A widget displays at the center of the slab.



3 Repeatedly click the widget. Press Enter or right-click to confirm the loadbearing direction(s).

23.14.4 Adding a load bearing symbol in plan section

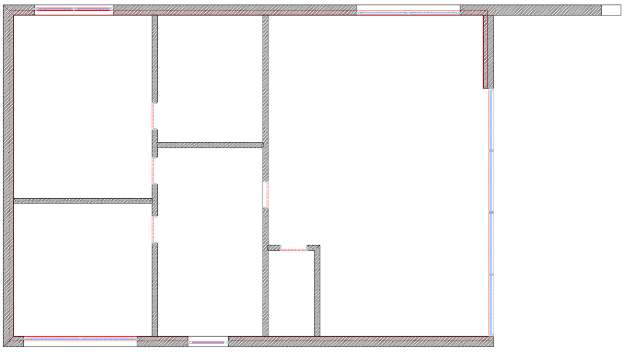
1 When you insert the load bearing tag in a plan section, you need to select a line, which is generated by the slab. Since slabs usually overlap with walls, you need to set the **Hidden Lines** > **Show** property of the **BIM Section** entity to **Yes**.

Hidden Lines

Show	Yes
Color	Red
Layer	0
Linetype	Continuous
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer

Next, execute the BIMSECTIONUPDATE command of the plan section.

2 Open the **Plan** section.



3 Launch the BIMTAG command.

4 Choose the **Manual** option.

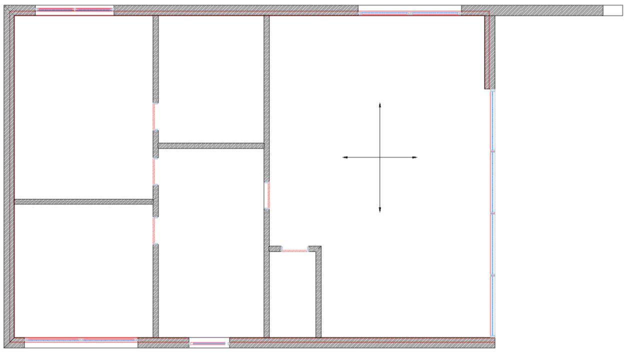
5 Choose the **Change Tag Type** option.

6 Choose the **Load-Bearing Direction** tag type.

7 Click on a point on the edge of the slab.

8 The tag displays, attached to the cursor.

9 Click to place the tag.

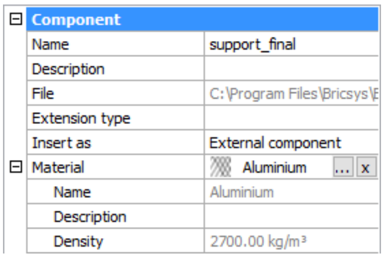


23.15 Physical materials

23.15.1 Opening the physical materials editor

Do one of the following:

- Launch the BLMATERIALS command.
- Use the hamburger menu (Open material dialog) in the compositions panel
- Click the **Browse** button ( ) of the **Material** property of a component in the **Mechanical Browser**.



23.15.2 Creating physical materials

You can create physical materials in either the Project or the Central database. You can create new materials from scratch or copy them from an existing material.

Creating a physical material from scratch

Select a material in either the **In Project** or the **In central database** material list.

Do one of the following:

1. Click the **New Material** button ( ) in the **Physical Materials** dialog box.
2. Right-click and choose **New Material** in the context menu.

A material named **New** is added to the selected database. Now you can edit the new material.

Adding a physical material as a copy of an existing material

Select a material in the **In Project** or in the **In central database** material list. Do the following:

1. Right-click on the material you want to copy and choose **Duplicate** in the context menu.
2. A material named **<existing_material>-2** is added in selected database.
3. Edit the copy of the material.

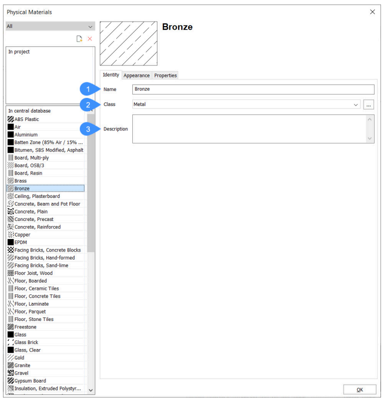
23.15.3 Editing a physical material

The various settings of a physical material definition are divided into three groups under three tabs on the **Physical Materials** dialog:

- **Identity**: **Name**, **Class** and **Description** of the material.
- **Appearance**: the BIM Materials layers the composition is composed of.
- **Properties**: properties of building material such as **Cost**, **Manufacturer**, **Thickness**...

Select the material you want to edit in the **In Project** or in the **In central database** list on the **Physical Materials** dialog box.

Identity tab page

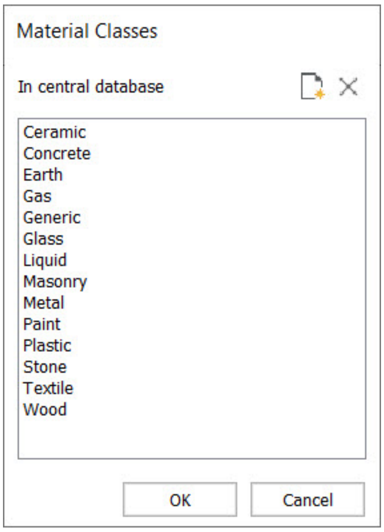


1 Type a **Name** for the material.

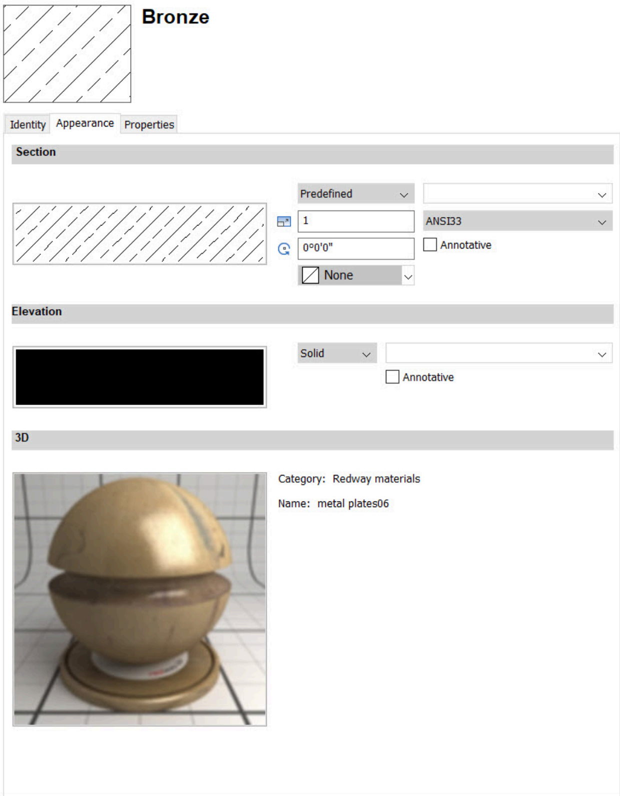
2 Select a **Class** from the drop-down list*.

3 **(Optional)** Type a **Description**.

* Click the **Browse** button ( ) to open the **Material Classes** dialog box where you can create new material classes.



Appearance tab page



- **Section settings**

  The **Section** settings define the fill of the section boundary lines in generated 2d sections.

  a Click the **Hatch Type** button and select the type of the hatch: **None**, **Solid**, **Linear**, **Predefined** or **Custom**.

  b Do one of the following to select a predefined or custom hatch pattern:

  - Click the **Hatch** preview and select the pattern in the **Hatch Pattern** dialog box.
  - Click the **Hatch Pattern Name button** and select the pattern in the list.

  c Define the **Scale**.

  The final pattern scale in a section is the product of the **Scale** setting of the physical material and the value of the **Scale** property of the **BIM** section. As a result, the printed scale of the hatch pattern is independent of the section plane scale.

  d Set the **Rotation** angle.

  e In the **Layer** field, either type a name or click the arrow button and select a layer in the list.

  f Optionally, check the **Annotative** option.

  g Optionally, choose a **Hatch Background Color** for the selected hatch pattern.

- **Elevation settings**

  The **Elevation** settings define the elevation fill of the section background lines in generated 2d sections. The **Generate elevation fill** setting needs to be active on the viewport of the 2d section to use elevation fills.

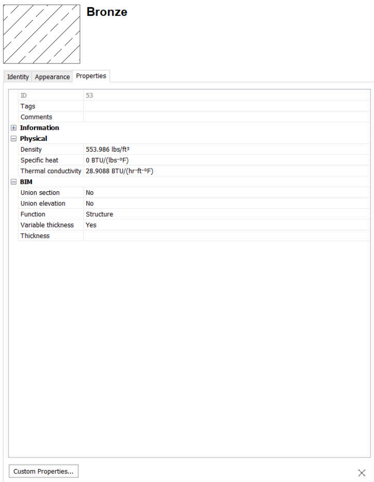
See the **Section** settings above, except for the **Scale** setting. The elevation hatch is independent of the **Scale** property of the **BIM** section.

- **3D settings**

  The **3D** settings define the rendering material that will be used in the 3d model.

  Double-click the preview image and select a material in the **Select Render Material** dialog box.

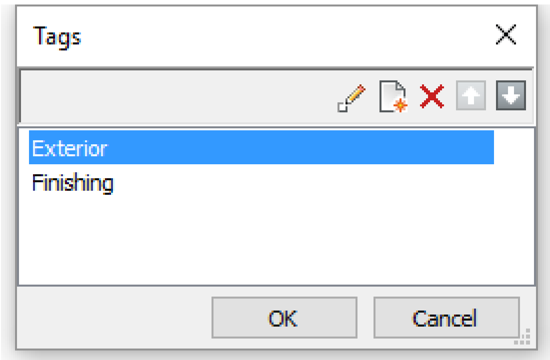
Properties tab page



1 Edit the **Tags** field: tags are separated by commas.

  Tags are case sensitive.

  Click the **Browse** button ( ) to open the **Tags** dialog box.



2 Click the **New Item** button ( ) to add a new tag at the bottom of the list.

3 Select a tag in the list and click a tool button:

  - : Edit the selected tag.
  - : Delete the tag.
  - : Move the tag up.
  - : Move the tag down.

4 **(Optional)** Expand the **Information** properties and fill out the necessary fields.

5 **(Optional)** Expand the **Physical** properties and specify the physical properties of the material.

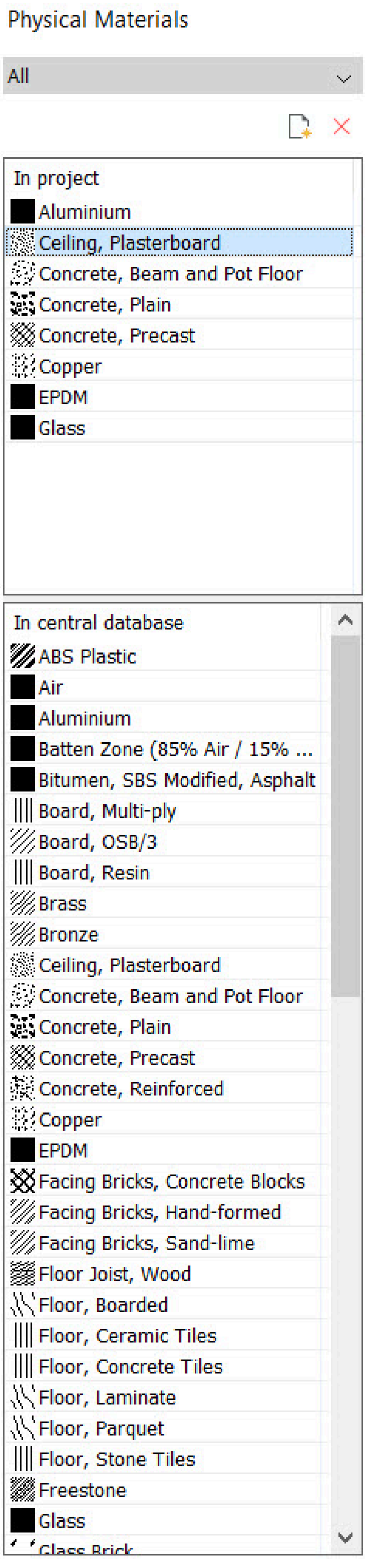
6 Expand the **BIM** properties.
   - **Union Section**: Select **Yes** to remove connection lines in sections between different solids of this material. Select **No** to keep the connection lines.
   - **Union Elevation**: Select **Yes** to remove connection lines in elevations between different solids of this material. Select **No** to keep the connection lines.
   - Select a **Function** from the drop-down list.
   - **Variable Thickness**: select **Yes** to allow a variable thickness. When applying a composition that contains a material with variable thickness to a solid, the ply thickness will adapt so the total composition thickness matches the thickness of the original solid. Select **No** to not allow a variable thickness. When applying a composition where all plies have a fixed thickness to a solid, the thickness of the solid will be adapted to match the thickness of the composition.
   - Type a default thickness in the **Thickness** field.

7 Click **OK**.

23.15.4 Copying materials between the project and library databases

1 Select the material(s) in the source database.

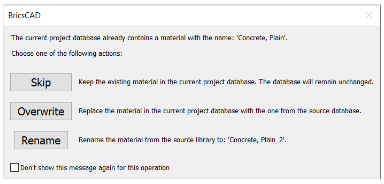
1 Press and hold the Ctrl key to select multiple materials.



2 Drag the materials to the target database.

3 The selection set is copied.

If one or more materials already exist in the target database, an alert box displays:



Click the appropriate button to either overwrite, duplicate or skip the existing material. If the **Apply to all materials in this operation** option is selected, the option applies to all existing materials. Duplicated materials have an index number added to their name.

23.15.5 Deleting physical materials

1 On the **Physical Materials** dialog box, select the material(s) in the Project or Library database.

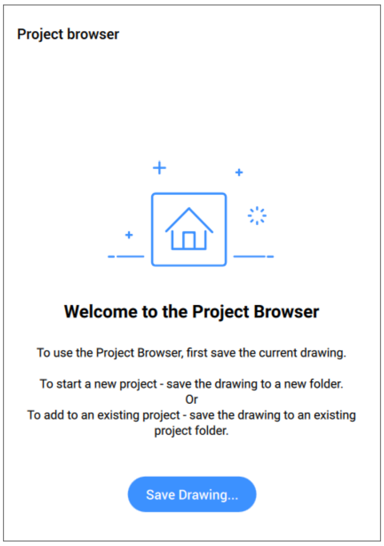
2 Click **Delete**.

Note: You cannot delete materials that are used in a composition or a mechanical component.

23.16 Using the Project Browser

23.16.1 The Project browser panel

You can find the **BIM Project Browser** panel on the left-hand side of the screen.



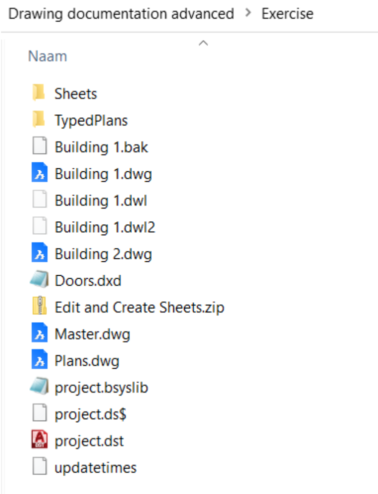
Note: If the **BIM Project Browser** doesn't display in the panel, right-click in the panel and select **Panels** > **BIM Project Browser**

23.16.2 Create a BIM project

To create a BIM project, you must save the model drawing and then:

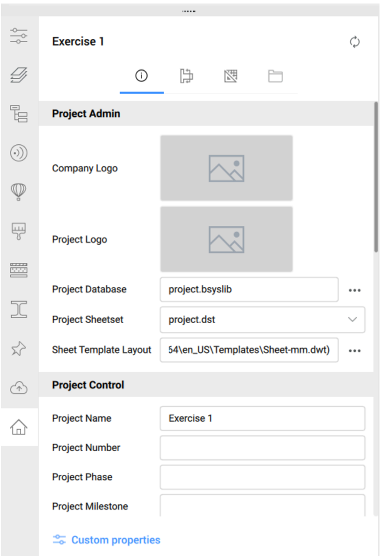
1 Click the **Create Project** button in the **BIM Project Browser**.

All external references files of the master are included in the project.

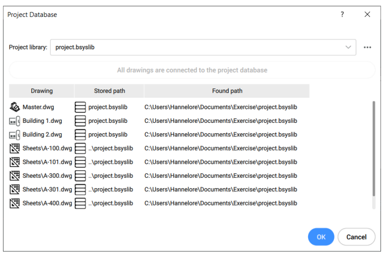


Other drawings can be added to the project by simply saving them in the project folder and then clicking **Add to Project**.

2 The **Info** tab of the **BIM Project Browser** panel is opened:



3 A **Project Database** is created. The **Project Database** dialog box opens by pressing the **Browse** button (…) on the right side of the **Project Database** field.



Make sure all drawings are connected to the same project database. That means all models are referred to the same single definition of materials, compositions, profiles, story and building.

Note: For more information about the project database, launch BIMPROJECTINFO command that opens the **BIM Project Info** dialog box.

4 The **Project Sheetset** field shows the used sheetset. You can open the drop-down menu to select another Sheetset that is located in the project folder.

5 Sheet Template Layout shows the path of the current template. Click the **Browse** button (…) to change the template.

6 **(Optional)** Set the Project info in the **Project Control** section:

Select a **Company Logo** file from the **Select a Company Logo** dialog box that opens when you click on the picture field.

Select a **Project Logo** file from the **Select a Project Logo** dialog box that opens when you click on the picture field.

Type a name in the **Project Name** field (the default project name is the name of the parent folder).

Sheetset Custom Properties and **Default Sheet Custom Properties** will appear in the title blocks of the sheets.

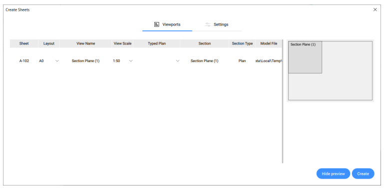
23.16.3 Refresh the project

Click on the **Refresh** button ( ). The **BIM Project Browser** is refreshed to match the changes in the model.

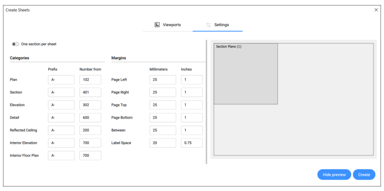
23.16.4 Create Sheets

Go to the **Sections and views** tab of the **BIM Project Browser** panel and right click a section. Select **Create Sheets** to open the **Create Sheets** dialog box.

1 The **Viewports** tab lists all sheets that will be created with their sheet name, layout, view name, view scale, typed plan, section name and type and model file name.

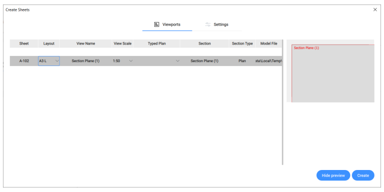


2 The **Settings** tab lists the settings of the sheets that will be created. The naming of the sheets can be changed in the **Categories** tab section. The margins of the sheets can be set in the **Margins** tab section. You can also toggle **One section per sheet** on and off to allow multiple sections per sheet.



3 A preview of the created sheet is shown on the right side of the dialog box. Click **hide preview** to toggle the preview off.

Note: When the viewport is larger than the layout, the preview is displayed in red.



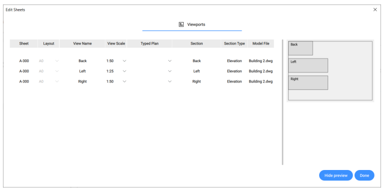
4 Click **Create** to create the sheets. The sheets are then visible in the **Sheets** tab of the **BIM Project Browser** panel.

23.16.5 Edit Sheets

Go to the **Sheets** tab of the **Project Browser** panel and right-click a sheet/sheet view. Select **Edit Sheet** to open the **Edit Sheets** dialog box.

1 The **Viewports** tab lists all sheet views of the selected sheet. You can edit the scale and select a typed plan for the selected sheet view.

Note: A sheet's views position on the layout cannot be changed in the **Edit Sheets** dialog box. To do that, go to the **Sheets** tab and double-click the sheet's name to open the sheet's drawing.



2 A preview of the sheet is shown on the right side of the dialog box. Click the **Hide preview** button to toggle the preview off. When a sheet view size is larger than the sheet's layout, the layout preview is displayed in red. In this case you can adjust the view scale to fit the layout.

Note: The layout size of an existing sheet cannot be changed from the **Edit Sheets** dialog box. To have a different layout size, you need to create a new sheet with the desired size.

3 Click **Done** to save the changes.

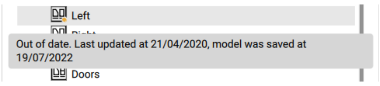
23.16.6 Auto Background Updating

Go to the **Sheets** tab in the **Project Browser**. At the bottom you can toggle **Auto background updating** option On or Off. When enabled, section results are calculated automatically without interrupting the user workflow.

Note: In the **Sheets** tab of the **Project Browser** panel, the drawings with an orange fill circle icon are out of date or not yet generated. Right-click on them and select **Update Now** or **Update in Background** to match changes in the models included in the project.

Note: The drawings with a red fill circle icon have an error. This occurs when the section is deleted or not stored at the correct location.

Note: When you hover over a sheet that is out of date, the last update time of the sheet and the last time the model was saved are displayed on a tooltip.



23.16.7 Creating a BOM (Bill of Materials)

In the **BIM Project Browser** panel, go to the **Files** tab and click the **Add** button. Select **New BOM** option to create a new Bill of Materials or **Import BOM** option to add an existing one. The BOM files are listed in the **Schedules** folder.

Note: The **BIM Project Browser** panel supports the old schedules workflow. You can add DXD files manually to the **Schedules** folder. When a DXD file is present in the **Schedules** folder, its context menu displays the same options as for a BOM file. All context menu options are listed in the **BIM Project Browser panel** related article.

To place a BOM on an opened sheet, go to the **Files** tab tree and find the BOM file in the **Schedules** folder. Drag-and-drop it from here on the sheet's Paperspace. This generates a table with a datalink to the BOM file.

To edit a BOM file, use the BMBOMTEMPLATEEDIT command and update the corresponding placed schedule(s) using the DATALINKUPDATE command.

Note: For more information about creating and editing a BOM, see the **BMBOMTEMPLATEEDIT command** and **BOM Manager panel** articles.

23.17 Spatial locations

23.17.1 Commands

BIMATTACHSPATIALLOCATION, BIMSPATIALLOCATIONS

23.17.2 About

Spatial locations allow you to specify building and its stories in a project. In BricsCAD, a BIM model file can contain one site and multiple buildings and a building can contain multiple stories. When the entities are classified as a building element in the project, these classified entities have a **Building** and a **Story** property. Every building element in the BIM model resides in a particular story within a particular building.

For more information about this command, visit the Command Reference article **BIMSPATIALLOCATIONS command**.

23.17.3 Procedure: assigning a building and story to a building element

1 Launch the BIMSPATIALLOCATIONS command.

2 Click the **New Building** icon ( ) to add a new building to the model.

3 Fill out the properties grid to specify a new name or value of the existing property.

4 Click the **New Story** icon ( ) to add a new story in the selected building.

5 Fill out the properties grid to specify a new name or value of the existing property.

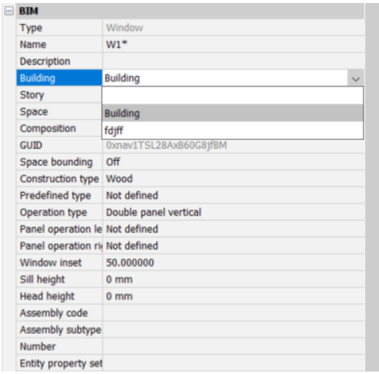
6 **(Optional)** Click the **Delete** icon ( ) to remove the selected building or story.

7 **(Optional)** Click **Import Spatial Locations** button ( ) to import a spatial location from a .txt file.

8 Select building element(s) in the drawing area to associate them with the appropriate spatial locations.

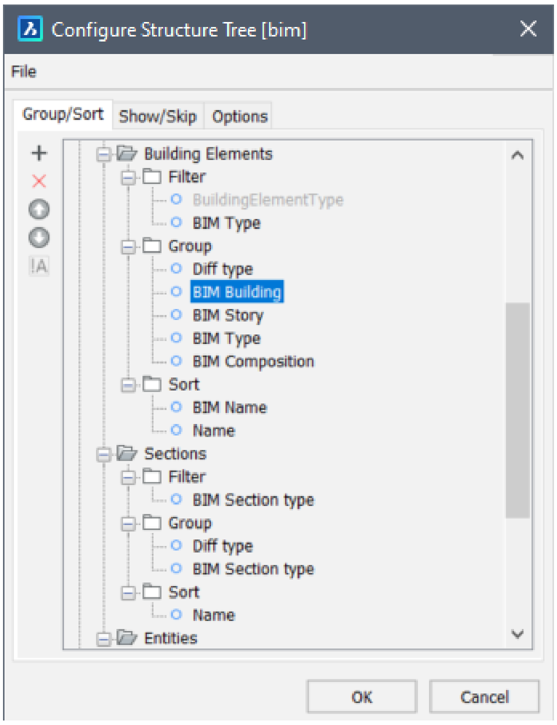
9 The properties of the selected building elements display in the **Properties** panel.

10 Do the following to assign a building and story to these building elements:
- Expand the **BIM** section in the **Properties** panel.
- Click the **Building** field, then use the arrow to select a building.
- Click the **Story** field, then use the arrow to select a story.



23.17.4 Viewing the spatial locations in the Structure panel

The **Structure** panel allows to view all of the entities in a drawing. By default, the structure of the building elements is organized in a particular form. Building elements are grouped by **Building**, **Story**, **BIM type**, and by **Composition** respectively. This organization can be fully configured. Right click in the **Structure** panel and click **Configure**, a new window will open. For more info, see the article **Working with the Structure panel**.



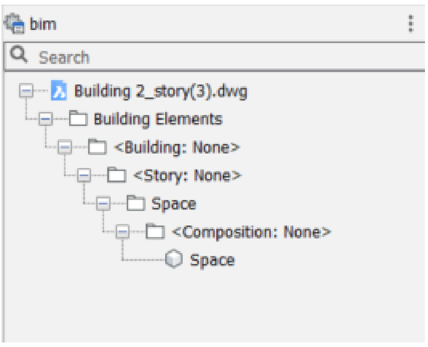
If there is no **BIM Building** or **BIM Story** in the **Structure** panel, you can add them manually. See **Working with the Structure panel** article for more info.

23.17.5 About BIMATTACHSPATIALLOCATION

Assigning new **Space** can be done using the BIMSPACE command. Spaces are not always assigned to the according buildings because they don't have a spatial location yet. Use the BIMATTACHSPATIALLOCATION command to assign a spatial location to spaces.

Note: Assigning a new**Space** to a building element overrides the element's **Story** and **Building** properties with those of the space.

For more information about this command, visit the Command Reference article BIMATTACHSPATIALLOCATION.



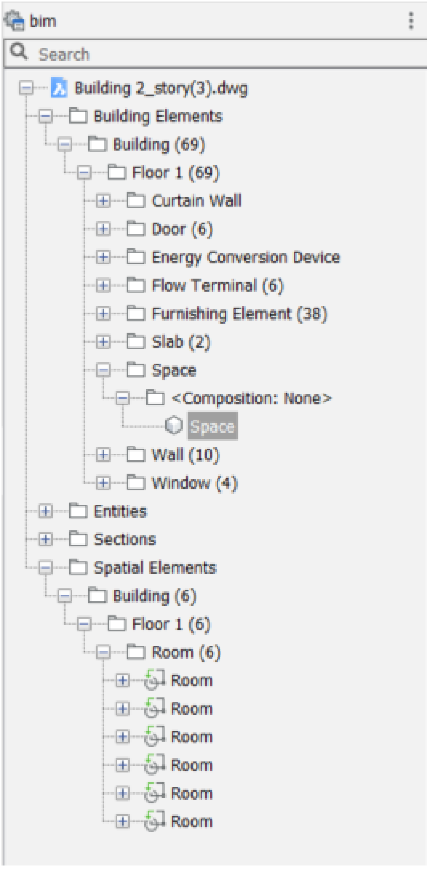
23.17.6 Procedure: attaching spatial locations

1 Launch the BIMATTACHSPATIALLOCATION command.

2 **You are prompted:***Enter location number or [Auto attach locations/Unattach current location]:* Press **A** to auto attach the spaces to their according building and floor level.

3 Select the spaces to update and press **Enter**.

4 **(Optional)** Press **U** to unattach the spatial location.



23.18 Structure panel configurations for BIM

23.18.1 Commands

STRUCTUREPANEL

23.18.2 About

The **Structure** panel is an interactive tree that displays the entities found in the current model. The way the entities are displayed in the panel depends on the selected configuration. For BIM, the preselected configuration is the predefined **bim** CST file. You can edit this configuration using the **Configure Structure Tree** dialog box. Once you configured the tree, you can easily save this configuration as a new CST file. By default, the CST files are stored in the **Support Folder**.

Place the cursor over a toolbar or ribbon panel, then right-click and choose **Structure** in the context menu.

Or

Type STRUCTUREPANEL in the Command line, then press **Enter**.

23.18.3 Procedure: explore the structure panel

1. Open the **Structure** panel.

   By default, the **Structure** panel displays on the left-hand side on your screen. With a **Search** field at the top of the panel (1).

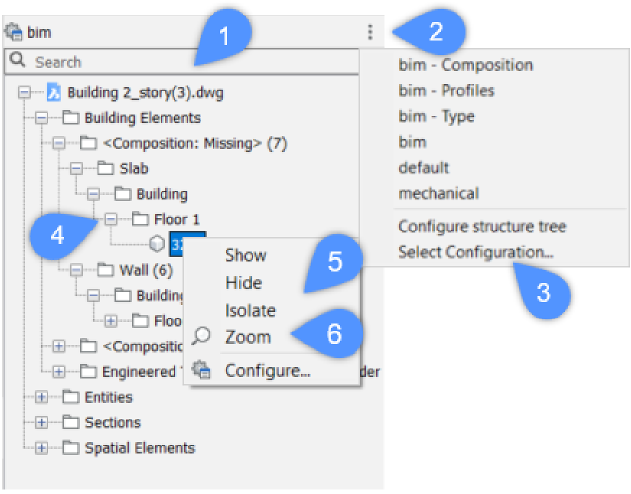
2. Click the three dots button (2) to load a configuration file in the list. Click **Select Configuration...** to select a CST file, which specifies a configuration (3). The current configuration displays on the left. Based on the selected file the BIM model will be organized in a tree-structure.

3. Click the expand/collapse icon (4) of a node to expand or collapse the node. Right-click a group name to select **Expand All** or **Collapse All** and expand or collapse all the subnodes at once.

   You can **Show**, **Hide** or **Isolate** the entities, selected in the **Structure** panel, in your model space, by right-clicking the selected names (5).

   This context menu also provides you to **Zoom** (6) into a specific entity in the model.

   The following illustration shows the **Structure** panel.



23.18.4 Procedure: query in the structure panel

1. Typing something in the **Search** field at the top of the **Structure** panel will issue a simple Text search: for example, if you type ‘wall’ in the search field and press **Enter**, the **Structure** panel will be filtered to everything with 'wall' in its name, including walls and curtain walls.

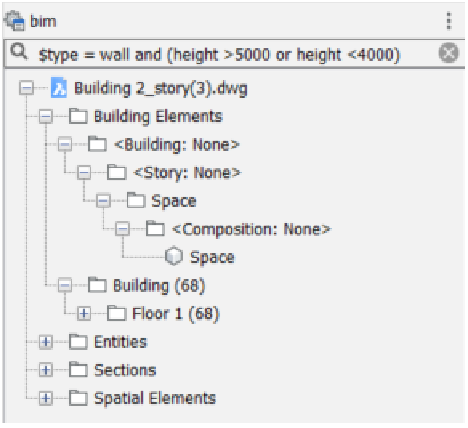
2. Text searches can be useful for a quick search, but more powerful queries are possible using

expressions.

3 An expression starts with **$**. For example, if you enter *$type = wall* and press **Enter,** the panel will be filtered on entities that have the BIM type = wall. Thus, using this expression, the curtain wall doesn't show up in the panel anymore.

4 Expressions can be more complex, such as filtering on multiple properties. For example, you can enter *$type = wall and Height > 5000* to filter on all walls with a height over 5000 mm.

5 Logical expressions can be nested using brackets: *$type = wall and (Height > 5000 or Height < 4000).*



23.18.5 Procedure: manage the tree-structure

The tree structure of the **Structure** panel represents the hierarchical set of rules in a graphical form. Each rule has a filter property, a grouping property, and a sorting property. Organizing the properties in the **Structure** panel enhances the performance of your project while querying the entities among others. When you have a tree for your BIM model you can establish a hierarchy across its building element types, buildings, stories, etc. You can use the hierarchy to manage the properties at various levels of the data display.

Do the following to configure the tree-structure:

1 Right-click the name of your drawing in the **Structure** panel, then choose **Configure** in the context menu.

The **Configure Structure Tree** (7) dialog box displays.

2 Select any existing rule in the structure tree, then click the **Add icon** (8) or right-click and choose **Add Rule** in the displayed context menu. This menu can also be used to delete and rename the existing rule.

The new rule is added below the previously selected rule. Type a new name for the newly created rule by clicking its name field (9).

3 Select the **Group** (10) node of the rule, then use the Add icon or right-click and choose **Add Grouping Property** (11) in the context menu.

The **Select Property** dialog box displays. Select a property on the list that you want to use to group the properties in your **Rule** setting. Then, click the **OK** button. The new property is added to the **Group** node.

If you use the **Filter** node to add a property, the selected property will be added in this node.

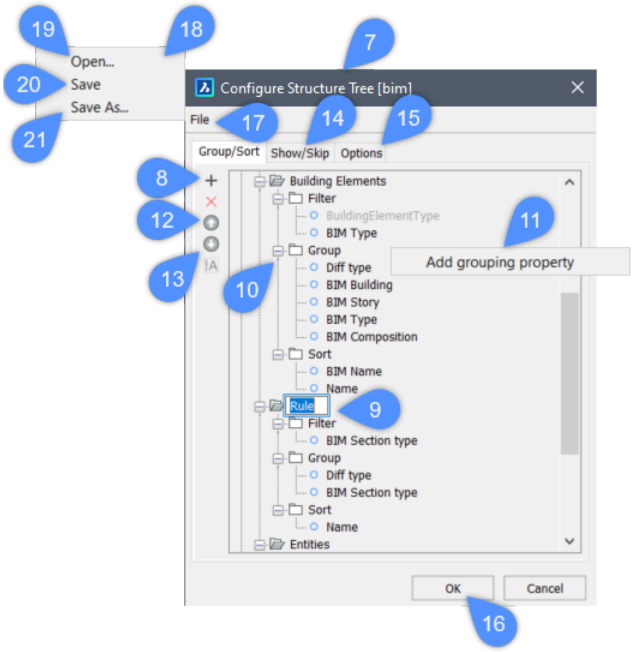
4 To move an item in the tree, select the item, then click the **Move Up** (12) or **Move Down** (13).

5 Select the **Show/Skip** (14) tab to sort the entity types or exclude them on the **Structure** panel.

6 Select the **Options** (15) tab to set the entities action in the model using the structure tree. You can here specify whether or not selecting an entity in the **Structure** panel also selects the entity in the model space.

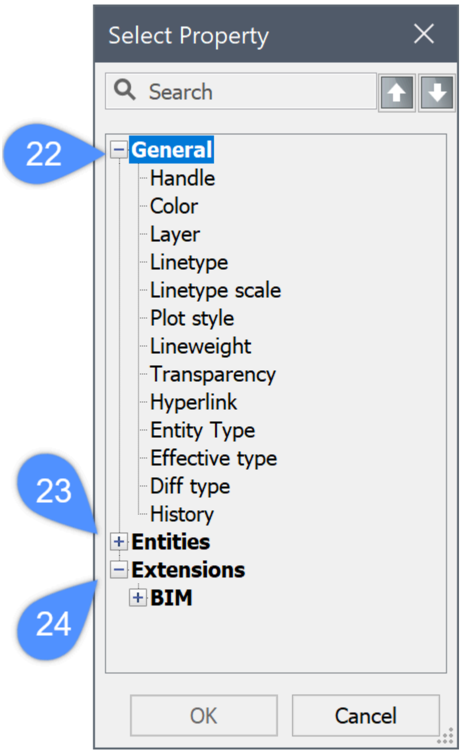
7 Do one of the following to save the configured tree:
   - Click the **OK** (16) button.
   - Click the **File** (17) menu in the dialog box and choose one of the available options on the list (18).

The following illustration shows the **Configure Structure Tree** dialog box.



Open (19)	Opens a file dialog to select another .cst configuration file.
Save (20)	Saves the current configuration and keep the file dialog open.
Save as (21)	Saves the current configuration under a different name and keep the file dialog open.

The following illustration shows the **Select Property** dialog box.



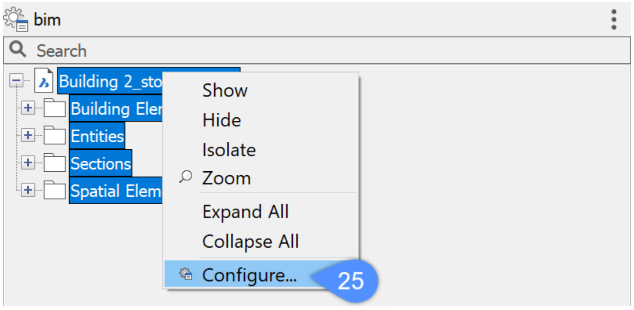
General (22)	Contains general properties.
Entities (23)	Contains various types of entity properties.
Extensions (24)	Contains various types of properties for **BIM** and **Quantity** extensions.

23.18.6 Procedure: manage the tree-structure with custom properties

In BricsCAD, the user-defined/custom properties are available to add in the **Select Property** dialog using the GUI. The **Select Property** dialog provides a list of dynamic properties for the drawing based on the current workspace. Note that custom properties, through the **Select Property** dialog box, can be used to group the tree structure.

Do the following to group the tree-structure using the existing custom properties:

1 Right-click the name of your drawing in the **Structure** panel, then choose **Configure** (25) in the context menu.



The **Configure Structure Tree** dialog box displays.

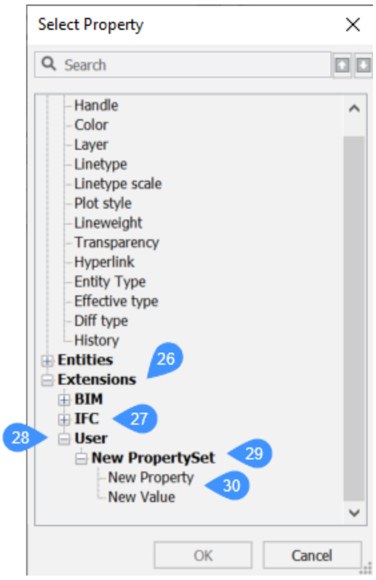
2 Select the **Group** node of the rule, and click the **Add rule or property** (+) icon.

Or right-click the Group node and choose **Add Grouping Property** in the context menu. The **Select Property** dialog box displays (see below image for the **Select Property** dialog). If you have custom properties, these properties will be shown under **Extensions** (26) and grouped by its namespace.

3 Click the expand/collapse icon of a node to expand and collapse the node.

4 The following image shows the **IFC** (27) and **User** (28) namespaces in the **Select Property** dialog.

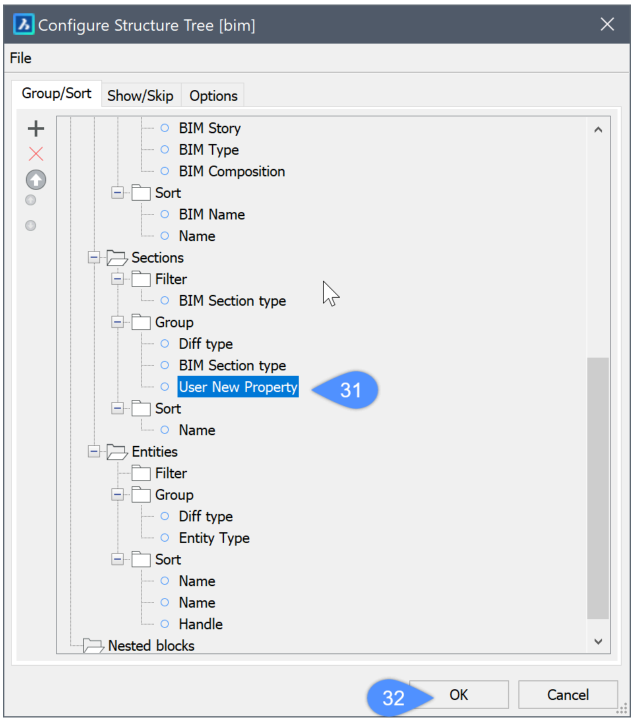
For example, the **User** namespace is expanded to display the property set (29) and properties (30) that are in this node.



5 Select the property to use for grouping in the tree structure, and press **OK**.

The property is added (31) to the **Group** node.

6 Press **OK** (32) to use the property for grouping entities in the **Structure** panel.


Configure Structure Tree [bim]
File
Group/Sort
Show/Skip
Options
BIM Story
BIM Type
BIM Composition
Sort
BIM Name
Name
Sections
Filter
BIM Section type
Group
Diff type
BIM Section type
User New Property
31
Sort
Name
Entities
Filter
Group
Diff type
Entity Type
Sort
Name
Name
Handle
Nested blocks
32
OK
Cancel

24. Design assistance

24.1 Using Bimify

24.1.1 Commands

BIMIFY

24.1.2 About BIMIFY

BIMIFY analyzes a model and automatically classifies 3D solids to building elements and assigns spatial locations, spaces, buildings, and stories. It also detects and classifies the external and internal walls.

BIMIFY can automatically detect whether a model is an Architectural, Structural or MEP model. BricsCAD uses this information during the auto-classification process. You can define the discipline of the drawing manually as well, in the **Bimify** command panel, to improve the accuracy of the automatic classification.

If solids are auto-classified as columns, beams or members and their profiles match a definition in the profile library, the profile name will be automatically added to the entity's properties as metadata. If no matching profile definition is found, a new one is created in the library.

BIMIFY can be used in a model that has both classified and unclassified entities. The classified entities are also linked to the appropriate spatial locations.

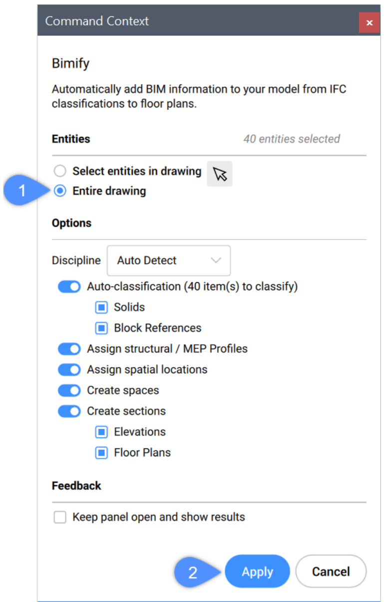
For more information about this command, visit the Command Reference article BIMIFY command.

24.1.3 Procedure: Bimify your entire drawing

1 Choose Bimify from the **Quad**, **BIM Data** > **CLASSIFY** ribbon tab, **BIM Classify** toolbar, or type BIMIFY in the Command line.

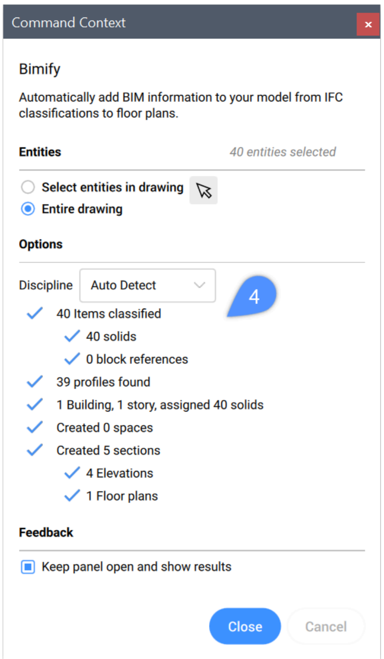
The **Bimify** command panel opens.

2 Check the **Entire drawing** (1) option and click the **Apply** button(2).

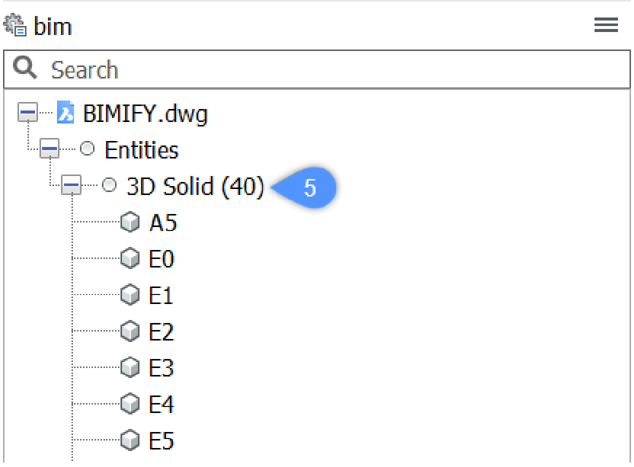


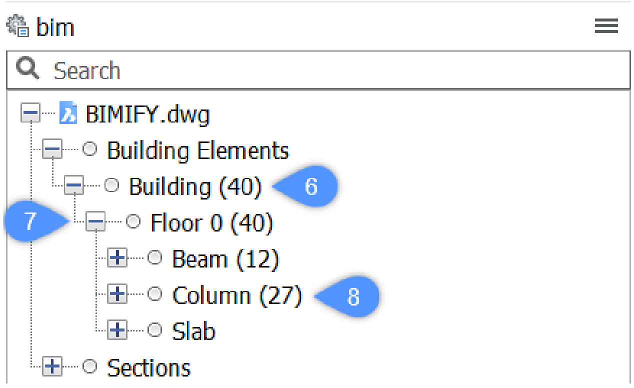
When the BIMIFY process is completed, an overview of the result (3) is provided in the **Prompt History** panel.

Note: If the **Keep panel open and show results** option is checked, the **Bimify** command panel will show the progress and results while calculating (4).



Bimify creates new spatial locations if necessary.



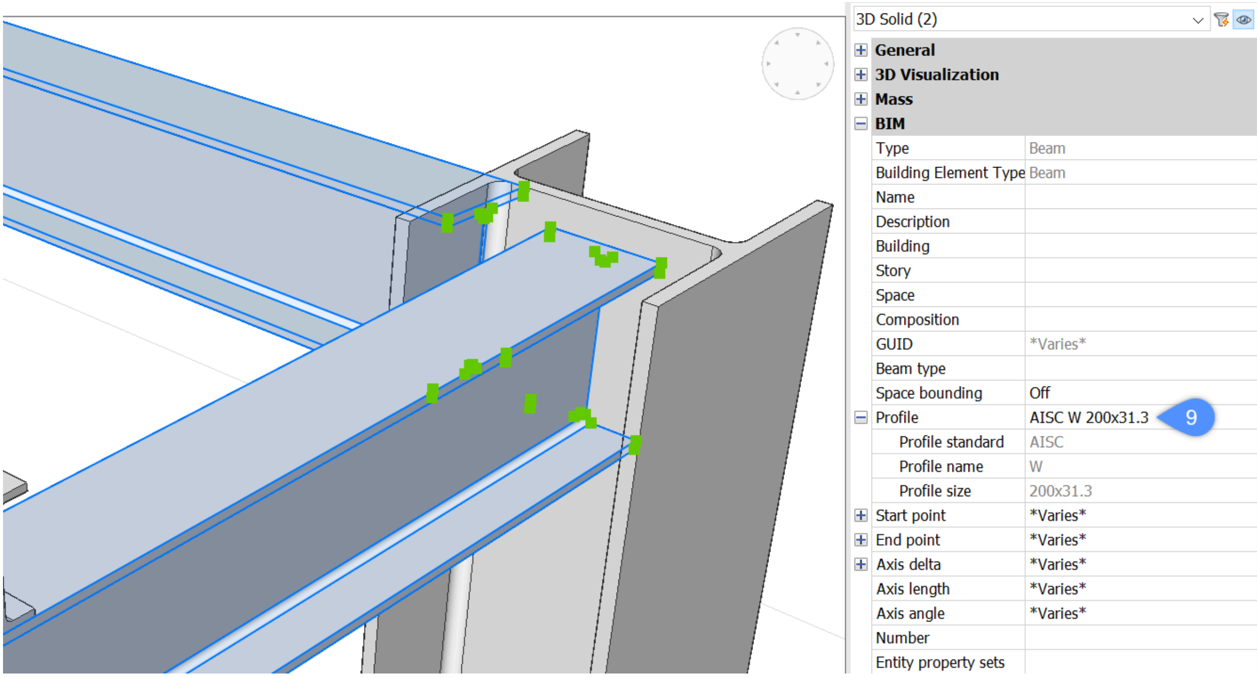


Left to right:

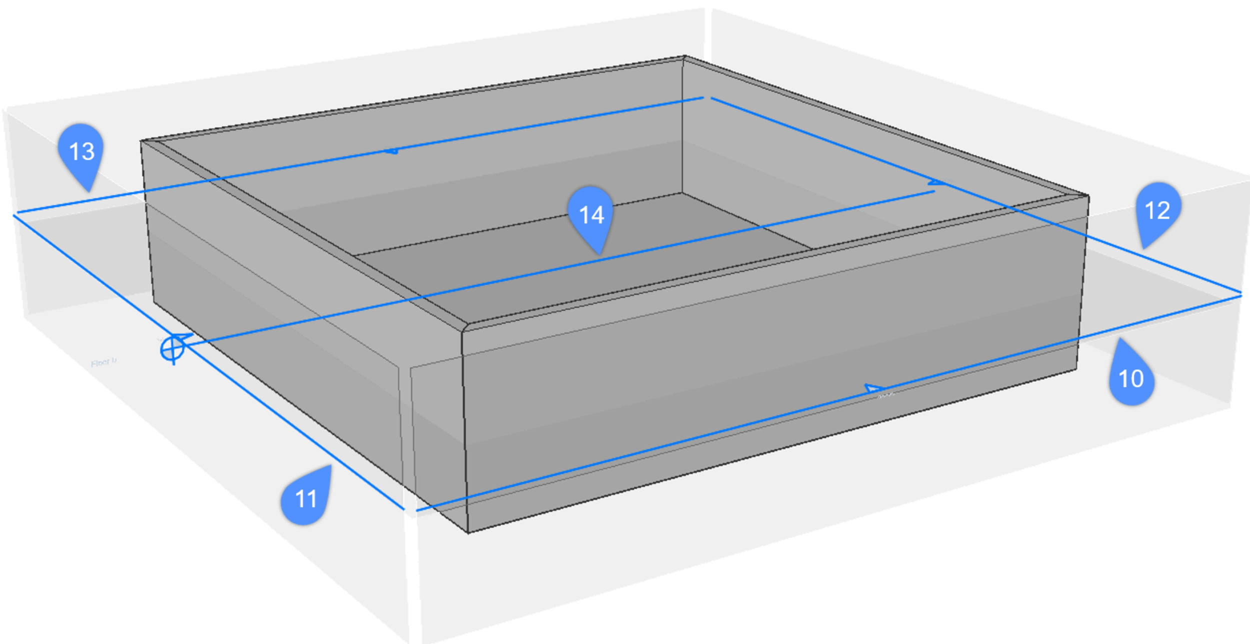
- ***Before Bimify:*** *The structure tree displays only 3D solids (5).*
- ***After Bimify:*** *The entities are sorted into their corresponding building (6), story (7) and building element type (8).*

Note: The **Structure** panel can be customized to control the content of the tree.

BIMIFY recognizes the profile of building elements as structural members and detects the axis location of the profile. The profile data of a linear solid can be seen in the **Properties** panel (9).

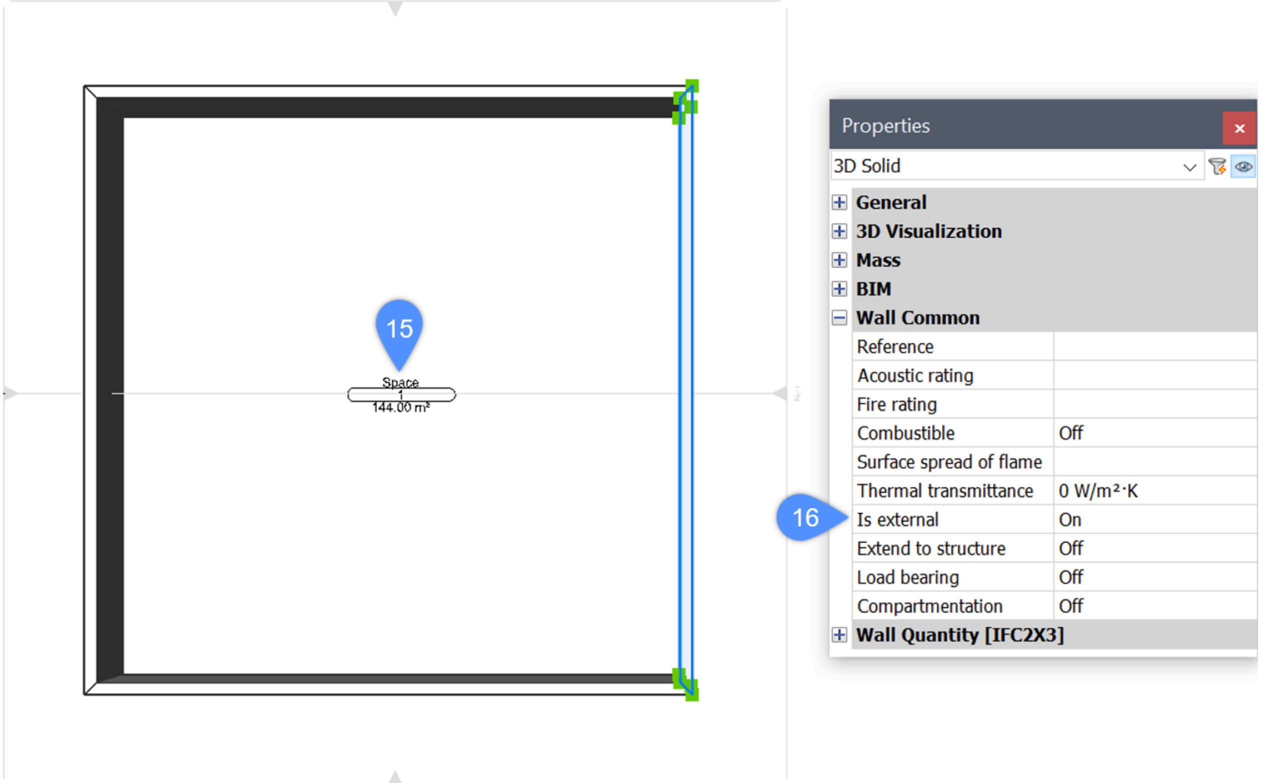


The elevation views and floor plan sections are created last.



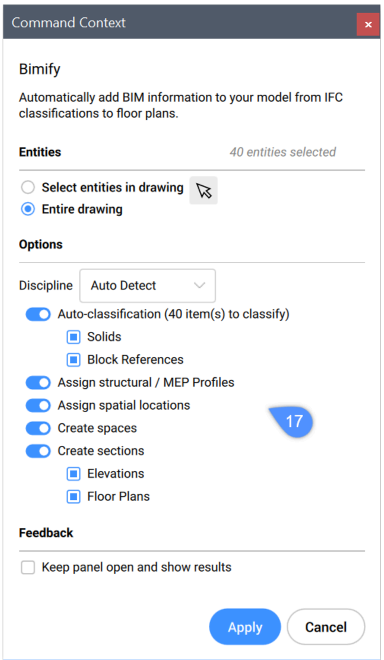
This building contains Front (10), Left (11), Right (12) and Back (13) views as well as a floor plan (14).

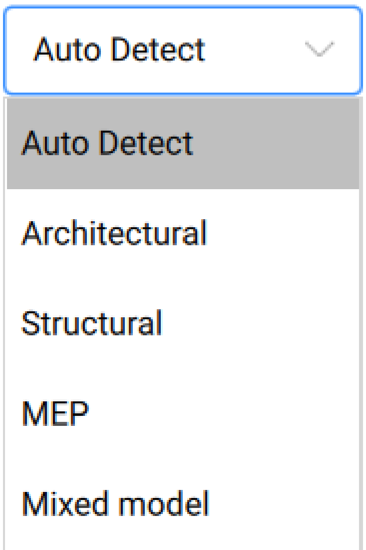
For this simple building, one BIM Space and 4 external walls have been detected.



*In the image above, the BIM Space is illustrated (15) and the **Is External** (16) property under the **Wall Common** in the **Properties** panel.*

You can control the functionality of the BIMIFY command by changing the options (17) in the **Bimify** command panel:

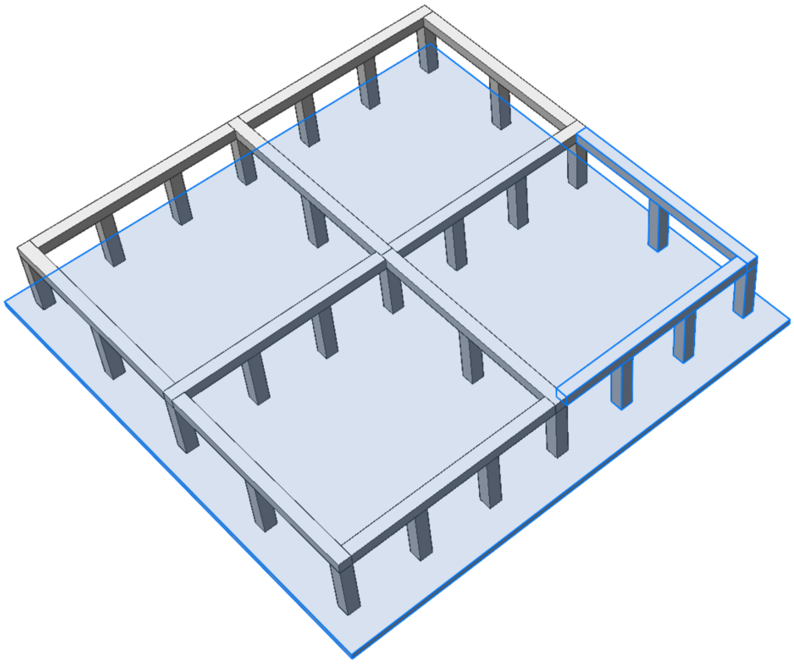


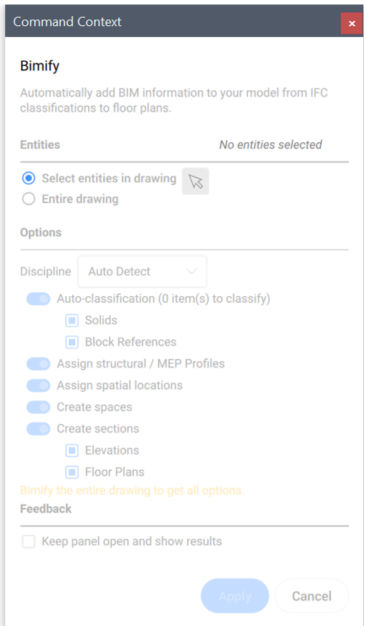
Discipline	 Defines the discipline of the drawing manually. This information helps improve the accuracy of Bimify.
Auto-classification	Classifies **Solids** and **Block References** automatically.
Assign structural/MEP Profiles	Assigns columns, beams, members or flow segments profiles if available in the **BIM Profile** panel. This option is available only when the **Auto-classification** option is selected.
Assign spatial locations	Assigns spatial locations such as buildings and floors, detects spaces and identifies internal and external walls. This option is available only when the **Auto-classification** option is selected.
Create spaces	Detects external walls and sets the property **Wall Common/is External** to **On** and also detects spaces. See the BIMSPACE command. This option is available only when the **Assign spatial locations** option is selected.
Create sections	Creates an elevation view and a plan section per floor. This option is available only when the **Assign spatial locations** option is selected.

Note: To disable the **Auto-classification** option for solids or blocks, simply uncheck the related checkbox. By default, all checkboxes are ticked. However, BricsCAD remembers your previous settings. In order to have access to **Assign spatial locations** options, **Auto-classification** option must be set to **On** and **Solids** checkbox must be ticked.This information is needed to detect the other elements.

24.1.4 Procedure: Bimify a selection of your BIM Model

1. Type BIMIFY in the Command line.
2. Click the icon to select the part of your drawing you want to Bimify.
3. Select the entities in the drawing you want to Bimify.





4. Press **Enter** to accept the selection.

   Note: Not all the options are available in the **Bimify** command panel when using BIMIFY on a selection of your BIM Model.

5. Click the **Apply** button to complete the Bimify process.
6. When the Bimify process is completed, an overview of the result is provided in the **Prompt History** panel.

24.2 Using propagate

24.2.1 Commands

BIMPROPAGATE

24.2.2 The five flavors

Before getting into the BIMPROPAGATE workflow it is important to note that there are five variants of BIMPROPAGATE, each working on a limited set of situations. These five variants (or flavors) were designed to guide new users through the process of BIMPROPAGATE. The tool has so many different uses that it can seem overwhelming to a new user; these variants help keep the tool more clear and concise, as they are designed for more specific use cases. However, they are still distillates from the original BIMPROPAGATE command, so every problem you can solve with a particular flavor you can **also** solve using the standard BIMPROPAGATE command. A more advanced user will even be able to solve problems that can not be solved by any of the particular flavors.

The BIMPROPAGATE variants are:

- BIMPROPAGATEPLANAR: Propagation of connections between planar elements such as walls, slabs and roofs.
- BIMPROPAGATELINEAR: Propagation of connections between linear elements such as beams, columns, pipes and ducts, and maximum one planar element.
- BIMPROPAGATEPATTERN: Propagation of a single element or group of elements on a flat surface to multiple locations and grids; can be used to propagate light fixtures, light switches, windows, air diffusers, columns on a grid.
- BIMPROPAGATEEDGES: Propagation along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc.
- BIMPROPAGATECORNER: Propagates details connected to multiple planar reference solids coming together in one node.

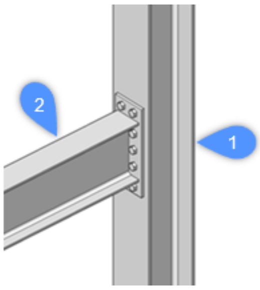
The general workflow of BIMPROPAGATE consists of two major steps: defining the detail volume, and choosing where to apply this detail.

24.2.3 Defining the detail volume

You are prompted:

Select reference solids. These will not be copied, but used to match other locations. [selection options (?)]:

The first step of every propagation is specifying the **reference solids**. These are the solids that define the situation where this particular detail should be applied. For instance, the image below shows a column-beam connection using an end plate and several bolts. This detail was modeled for this particular beam connecting with this particular column, and now we want to propagate this detail for every situation where a column and beam of these types come together. Thus, the reference solids in this case will be the column (1) and the beam (2). Later in the process, BIMPROPAGATE scans the model for situations where similar solids occur, and tries to map the detail onto these locations.



Note: This is an example of a situation that can be solved by BIMPROPAGATELINEAR.

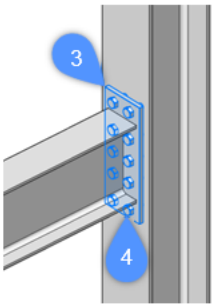
Press **Enter** to stop selecting reference solids.

The second step is specifying the detail objects or sub entities. These are the objects that you wish to propagate or copy throughout the model. They can be solids, block references or even faces (e.g. a hole in the reference solid can be propagated by selecting its faces).

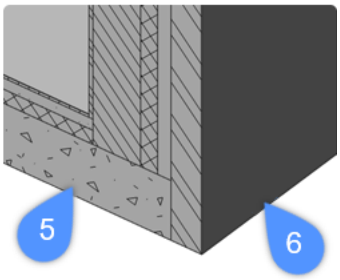
You are prompted:

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied. [selection options (?)]:

In the example at hand the detail objects will be the end plate (3) and the bolts (4).



Note: In some cases detail objects are not even required. Let's say we want to propagate the connection between the floor slab and the wall shown in the image below. Our reference solids are then the floor slab (5) and the wall (6), and no detail objects are required. In this case you can skip this step by pressing Enter or right click.



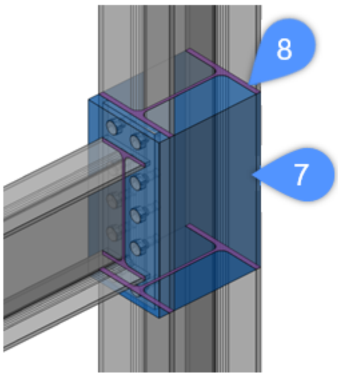
If we were to model extra solids in this connection (e.g. an insulation block or a skirting board), these extra solids would then be detail objects.

Note: This is an example of a situation that can be solved by BIMPROPAGATEPLANAR.

Once the **reference solids** and the **detail objects** are specified, a detail volume is defined automatically. The visual style is temporarily set to X-ray, and the view is automatically zoomed in onto the detail volume. Everything that is inside this volume and is either a detail object or part of a reference solid will be copied to the other locations when executing BIMPROPAGATE.

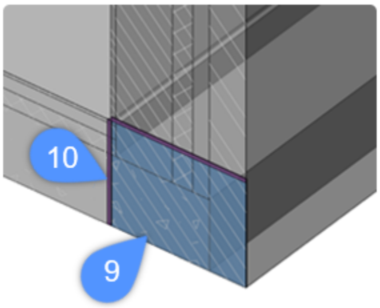
A detail volume can be either a **3D volume** or a **2D slice through a linear detail**.

In our example of the column-beam connection, the detail volume will be as shown in the image below. This is an example of a 3D volume. The connection of the reference solids to the detail volume is highlighted in purple (8).



Something interesting to note: In this particular detail, there are holes in the column where the bolts are bolted into the column. These holes are not necessarily detail objects, but since they are part of a reference solid and they are inside the detail volume, they will be copied as well. Thus, in this respect, BIMPROPAGATE does more than a 'dumb' copy of objects.

The detail volume of our wall-slab connection will look different. As shown in the image below, the detail (9) will be planar, and displayed in a section view. This is because the detail can be propagated as an extrusion of this 2D detail slice. Here as well, the locations where the reference solids connect to the detail volume are highlighted in purple (10).



Note: The detail of BIMPROPAGATEPLANAR and BIMPROPAGATEEDGES will always be a 2D slice through a linear detail.

You are prompted:

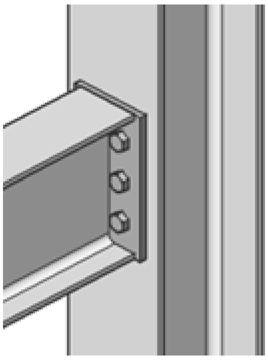
Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

You are prompted: Propagate detail as an extrusion of this 2D section? [Yes/No/Inflate first] <Yes>:

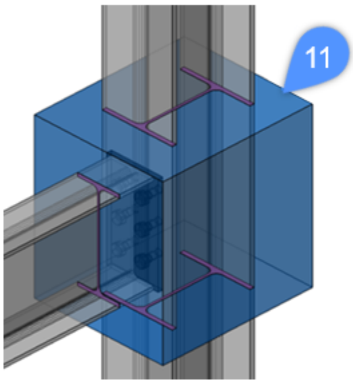
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Usually, the detail volume as it is proposed by BIMPropagate will be sufficient for what the user is trying to achieve. However, it is possible to inflate the detail to include a larger volume. This feature has two particular use cases:

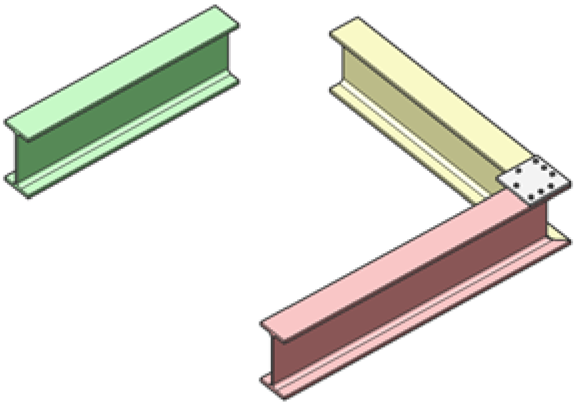
- If you want to replace an already existing detail with a new detail that is **smaller** than the original one. For example, we want to replace the previously proposed column-beam connection by the following:

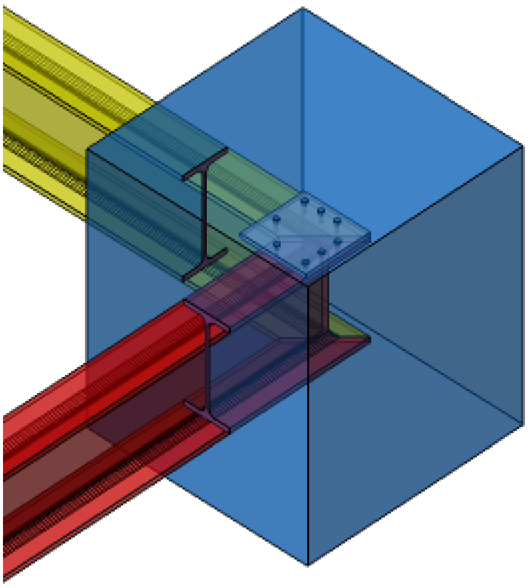
  In this case, we will have to inflate the detail volume (11) so that it is larger than the original detail:

- You might want to include a larger part of the reference solid, for example to connect with target solids that are further apart. In the image below, we want to propagate the connection between the **red** and the **yellow**beam, so that the **yellow** and the **green** beam have a similar connection.



To achieve this, we need to inflate the detail volume so that it overlaps the end faces of the red and the yellow beam.



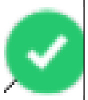
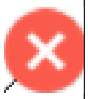
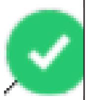
Once the detail volume includes everything you want to propagate, you can go over to the second step: choosing where to apply the detail. For 3D detail volumes there is one last choice to make: Propagate the detail as a **Block** or as a **Copy**. If you choose to propagate the detail as a block, then it will group all detail objects into a new **Block Definition** and copy around references to this **Block Definition**.

24.2.4 Choosing where to apply the detail

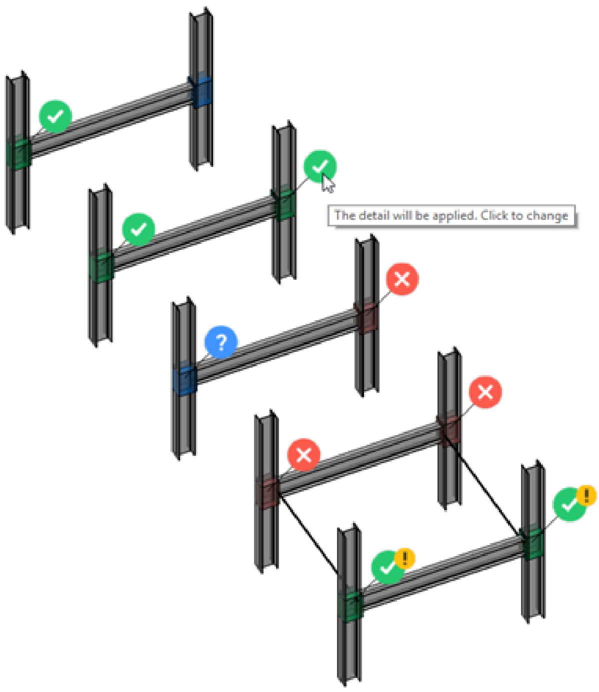
You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:

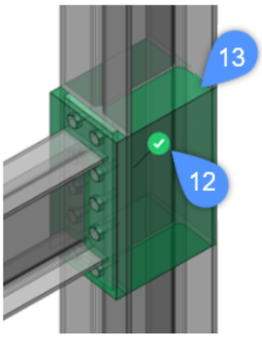
In this step the view zooms out showing all the possible suggestions on screen. Each suggestion is a location where this detail could be applied, and is indicated by one of the following icons (or widgets):

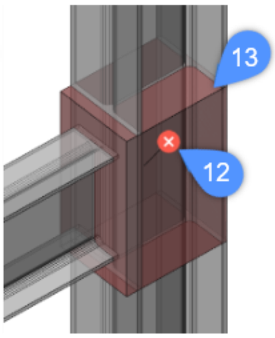
	The detail will be applied.
	The detail will not be applied; it was manually toggled off by the user.
	The detail will not be applied, because of one of several possible reasons. Hovering over the icon will provide more information.
	The detail will be applied, but interferences were detected.
	The detail will be applied, but there are several alternatives. Hovering over the icon will allow you to cycle between alternatives.

Hovering over a widget reveals some more information. Clicking a widget changes its status: you can toggle suggestions individually by clicking their respective widgets. Pressing Enter will then apply all the widgets as they are shown at that point: suggestions with a green checkmark will be applied, others will not be applied.



A suggestion also shows a preview of what the detail will look like. In the images below, you can both see the widgets (12) and the previews (13).





Widgets can also be **selected** using a window selection. A selected widget is highlighted by a blue border. Once one or more widgets are selected, pressing Enter will no longer apply **all** the suggestions, but **only those that are currently selected:**

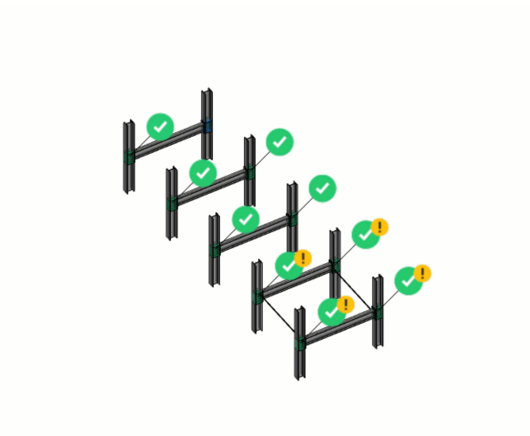
You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:

This procedure can be used to gradually solve all the suggestions.

In the video below, the following steps are taken:

- Four widgets are selected and toggled off
- Enter is pressed: the four selected suggestions were all toggled off, so they are not applied, and the four suggestion are removed from the view
- Four more suggestions are selected
- Enter is pressed: the four selected suggestions are applied, and removed from the view
- There is one suggestion left; Enter is pressed to accept this suggestion, and the BIMPROPAGATE command is terminated.

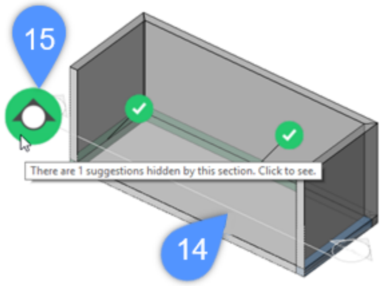


It is important to emphasize that BIMPROPAGATE only processes entities that are currently visible. As a result, hidden entities, or entities on a hidden or frozen layer are not included when BIMPROPAGATE is scanning the model for locations where the detail can be applied. This can be a useful when executing

BIMPROPAGATE in a large model; e.g. when you want to propagate a steel connection, only turn on those layers that contain steel members. This will improve the propagation performance.

It is possible to activate or move existing section planes during BIMPROPAGATE.

- A section plane (14) can be activated by double-clicking it.
- Left-click it once to move it.
- When a section plane is activated, a widget (15) might appear next to it indicating that there is one or more suggestion currently hidden because of the active section. Click the widget to deactivate this section and show the hidden suggestions.



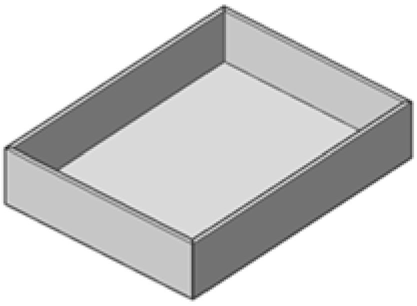
24.3 Propagate planar

24.3.1 Commands

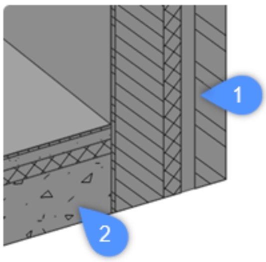
BIMPROPAGATEPLANAR

24.3.2 Procedure: propagating a wall-slab connection

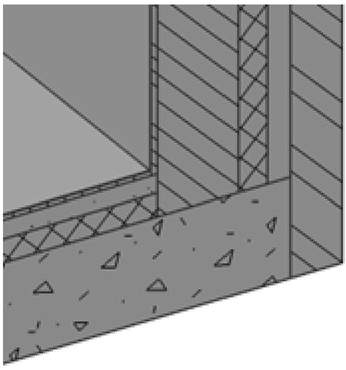
1 Consider the following situation of four walls and one slab, created with **QuickDraw**.



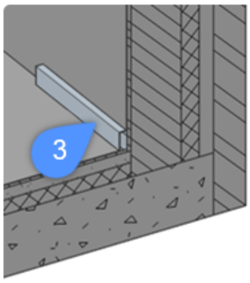
2 The four walls were given a composition, as well as the floor slab. Without any further editing or modeling, creating a vertical section through the building will show the connection one of the walls (1) and the slab (2) to be something similar to the following image.



3 While this section is active, we can **Push/Pull** the different ply faces to obtain the desired connection between the wall and the slab:



4 We can detail this connection even further, for example by locally adding a skirting board (3). This was done by simply drawing a box and classifying this solid as a **Building Element**.



5 Now that we have this detailed connection, we want it to be applied to every location where a wall and slab of this type come together. Let's deactivate the section and launch BIMPROPAGATEPLANAR.

6 **You are prompted:**

Select planar reference solids that form the connection. The connection detail will be copied to similar solids. [selection options (?)]:

Select the wall and slab, and press Enter.

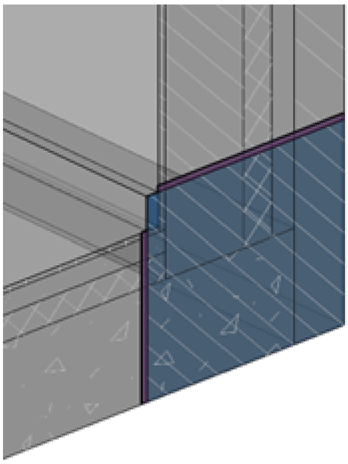
7 **You are prompted:**

Optional. Select detail objects to be copied as part of the connection. [selection options (?)]:

Select the skirting board solid, and press Enter.

8 **You are prompted:**The detail section varies too much along the extrusion direction.

In this particular case, BIMPROPAGATEPLANAR cannot find a solution: because the skirting board is not modeled along the entire length of the wall, BIMPROPAGATEPLANAR finds contradicting detail volumes. This problem can be solved easily by activating a section where the skirting board is present (e.g. the section we used earlier to create this detail) and trying again. If we now follow the same steps as before, BIMPROPAGATE finds a detail volume as shown in the image below.



9 **You are prompted:**

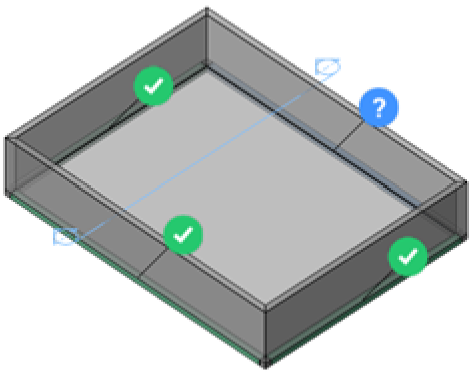
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command. Press **Enter** to accept.

10 **BimPropagatePlanar** now scans the entire drawing for similar locations. When this is done, all sections are deactivated and the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed:

You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:



A green checkmark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information.

In this case, it says **Location of original detail. Click to replace original detail by adapted extrusion**. Because we modeled the skirting board only locally, we should replace the original detail by an adapted extrusion so that it covers the entire length of the wall. Clicking the widget will turn it into a green checkmark.

11 Press **Enter** to complete the command.

Now, all four walls should have this particular connection with the floor slab, including the skirting board.

24.4 Propagate linear

24.4.1 Commands

BIMPROPAGATELINEAR

24.4.2 About

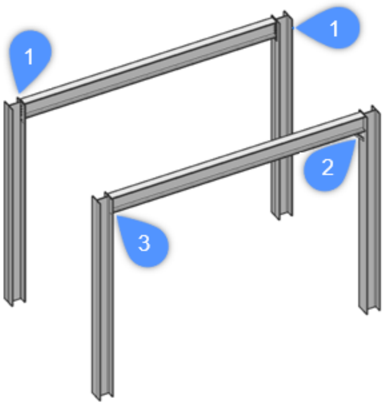
BIMPROPAGATELINEAR is one of the five Propagate flavors. It can be used to copy detailed connections between two or more linear solids. Common examples are column-beam connections or column-slab connections.

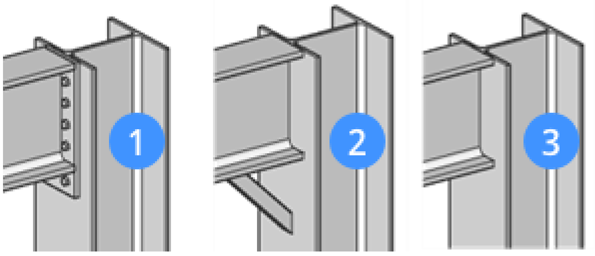
For a more in-depth explanation of the general principles of Propagate, read the Using Propagate article.

For more information about this command, visit the Command Reference article BIMPROPAGATELINEAR.

24.4.3 Procedure: propagating a column-beam connection

1 Consider the following situation of four columns and two beams. Two of the connections have been designed with a plate and bolts through the column flanges (1). One connection has been designed with a diagonal member for additional rotational stiffness (2). The last connection has not been designed yet (3).





2 We can now use Propagate Linear to copy one of the details to the other locations. In this example we will copy connection 2 to all four locations.

3 We will launch BIMPROPAGATELINEAR.

4 **You are prompted:**

Select the linear or planar reference solids that form the connection. The connection detail will be copied to similar solids. [selection options (?)]:

At least one should be linear.

Select the column and the beam that are connected by the diagonal member, and press **Enter.**

5 **You are prompted:**

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied as part of the connection. [selection options (?)]:

Select the diagonal member and press **Enter**. A detail volume is created that completely encompasses the diagonal member and parts of the column and beam.

6 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

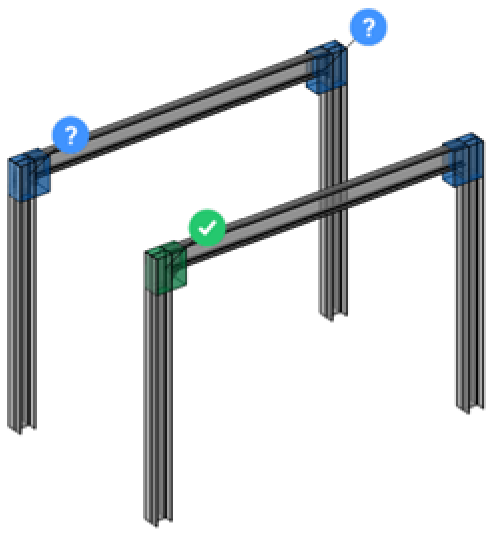
You can choose to propagate this detail as a block to keep the file smaller and more manageable, or as a copy. Inflating the detail is not necessary in this case. For more information on inflating, read the Using Propagate article. Choosing **No** will exit the command.

Press **Enter** to accept.

7 BIMPROPAGATELINEAR now scans the entire drawing for similar locations. When this is done, the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed:

8 **You are prompted:**

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:



A green check mark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information. In this case it says **Existing details detected. Suggestion will not be applied. Click to change**. We want to replace the bolted connection by this diagonal member, so we click both the blue widgets so they turn into green check marks.

9 If we now press **Enter**, all connections should be replaced by the diagonal member.

24.5 Propagate pattern

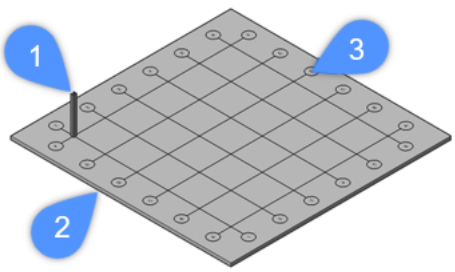
24.5.1 Commands

BIMPROPAGATEPATTERN

24.5.2 Procedure: propagating columns on top of a slab

1 Consider the following situation: a column (1) is placed on top of a slab (2). There is also a grid drawn on top of this slab (3).

1



2 We want to place this column on specific axis intersections. We could manually copy the column but, Propagate allows us to do this faster.

3 Let's launch BIMPROPAGATEPATTERN.

4 **You are prompted:**

Select planar reference solid to which the detail is related. The detail will be copied to similar solids.

[selection options (?)]:

Select the slab, and press **Enter.**

5 **You are prompted:**

Select detail objects (solids, block references, faces, edges,...) to be copied. [selection options (?)]:

Select the column, and press **Enter.**

6 A 3D detail volume is created.

7 **You are prompted:**

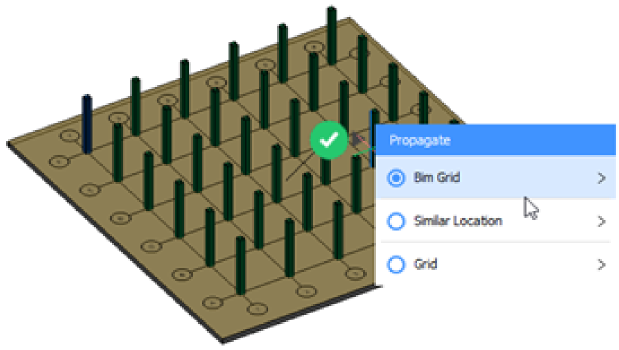
Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command.

Press **Enter** to accept.

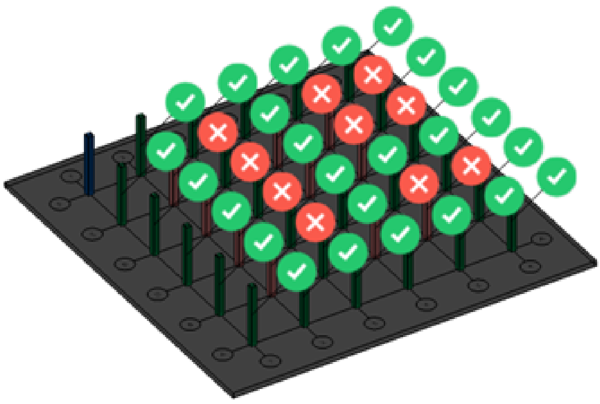
8 A widget appears. Hovering over this widget reveals a flyout menu with several options:

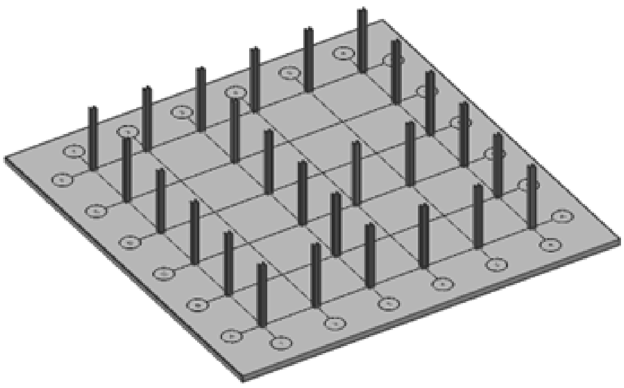
9



- **Bim Grid**: this is the grid that was drawn on top of the slab. Choosing this option and pressing Enter will copy this column to every axis intersection of this grid.
- **Similar Location**: this will copy the column to similar locations on this slab; in this case the four corners of the slab.
- **Grid**: this allows you to create your own grid of columns on top of this slab. You can choose to add or remove rows and columns. These rows and columns will always be evenly spaced.

10 We'll choose the **Bim Grid** option. Clicking this will expand the menu further; we now have the option to **Explode**. This will create a separate widget for each instance of this column, giving you more control over which suggestions you wish to apply (in this case which axis intersection you wish to place a column). We can now manually turn off suggestions:





24.6 Propagate edges

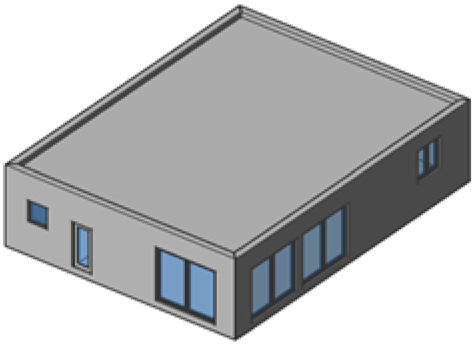
24.6.1 Commands

BIMPROPAGATEEDGES

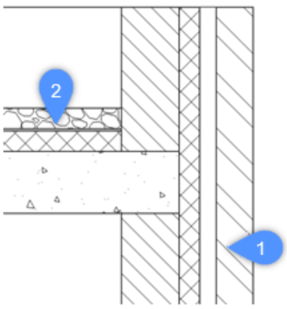
24.6.2 Procedure: propagating a wall cap

1 Consider the following situation of a one-story building with a flat roof.

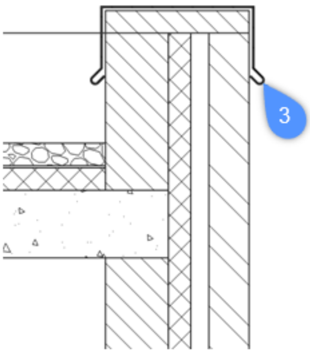
1



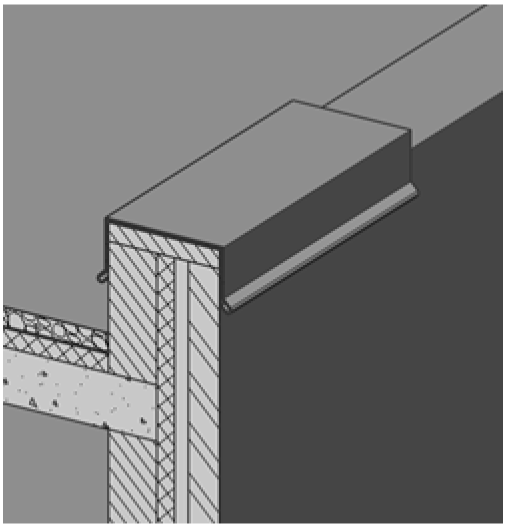
2 Let's create a vertical section plane to cut through our building and take a look at where the wall (1) and the roof slab (2) come together.

3 

4 We want to create a capping (3) on top of this wall to get the following result:

5 

We can start by creating this locally on top of one wall, either by using direct modeling tools or inserting a block from somewhere else and exploding it. In our section it might look like this:



6 Now we can use the BIMPROPAGATE command to copy this detail over to all applicable locations. Let's deactivate the section and launch the BIMPROPAGATEEDGES command.

7 **You are prompted:**

Select planar reference solid to which the edge detail is related. The detail will be copied to edges of similar solids. [selection options (?)]:

Select the wall on top of which the capping was modeled, and press **Enter.**

8 **You are prompted:**

Select detail objects to be copied. These should be solids or faces that are parallel to an edge of the reference solid. [selection options (?)]:

Select the objects that make up the capping, and press **Enter.**

9 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command. Press **Enter** to accept.

10 **You are prompted:**

Propagated to edges with similar orientation only? [Yes/No] <Yes>:

Choosing **Yes** will only apply this detail to edges with a similar orientation, in this case wall edges that point upwards. Press **Enter** to accept.

11 **You are prompted:**

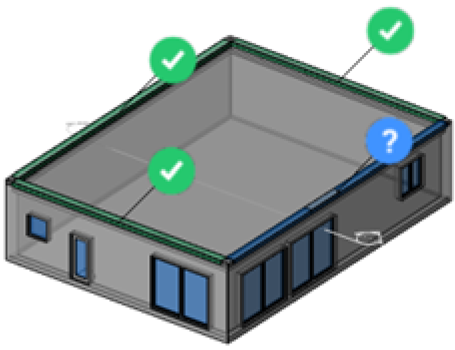
Choose location of detail relative to edge: [Outside reference solid/Inside reference solid] <Outside reference solid>:

This option controls whether the detail will be placed inside or outside (on top of) the base solids. When placed inside, the detail is subtracted from the base solids.

12 BIMPROPAGATEEDGES now scans the entire drawing for similar locations. When this is done, the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed:

13 **You are prompted:**

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:



A green check mark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information. In this case it says **Location of original detail. Click to replace original detail by adapted extrusion**. Because we modeled the wall cap only locally, we should replace the original detail by an adapted extrusion so that it covers the entire length of the wall. Clicking the widget will turn it into a green check mark.

14 Press **Enter** to complete the command.

Now, all four walls have this wall cap and the corner connections are created automatically.

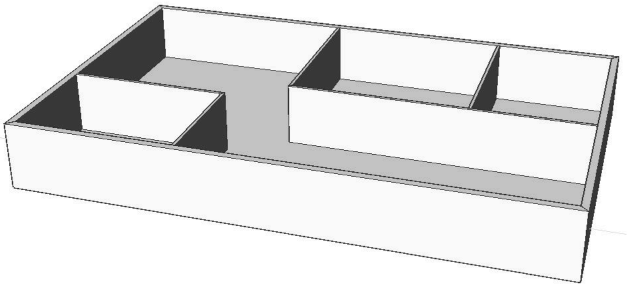
24.7 Propagate corner

24.7.1 Commands

BIMPROPAGATECORNER

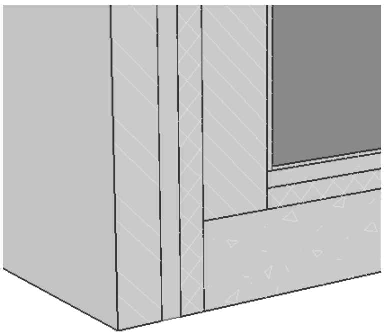
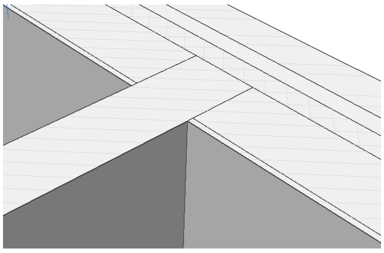
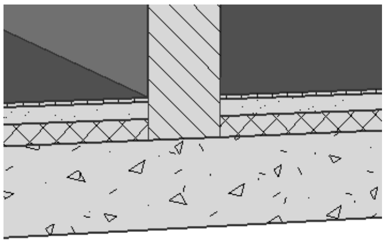
24.7.2 Setting the foundations

In this example the 4 exterior walls, 5 interior walls, and floor slab have already been assigned a composition.

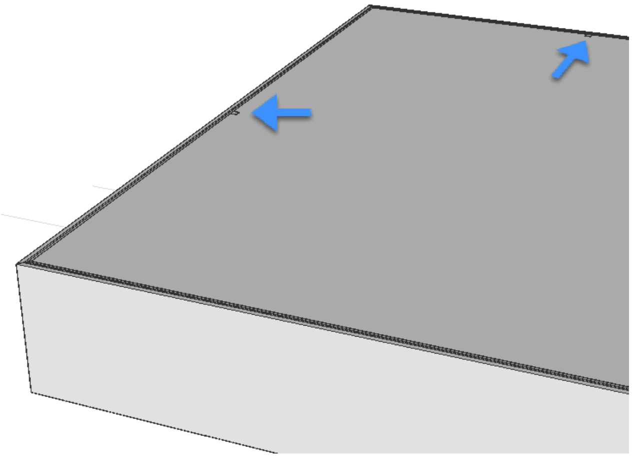


First, you must connect the 3 planar types of connections separately. For a detailed explanation of how to do this, read the article Propagate planar.

You should now have the following connections:

		
Exterior wall - Floor slab	Exterior wall - Interior wall	Interior wall - Floor slab

The nodes where the exterior wall, interior wall, and floor slab come together can't be correctly propagated using BIMPROPAGATEPLANAR. Instead, use BIMPROPAGATECORNER. BIMPROPAGATECORNER can propagate details connected to three planar base solids (i.e. corners).



24.7.3 Propagate corner

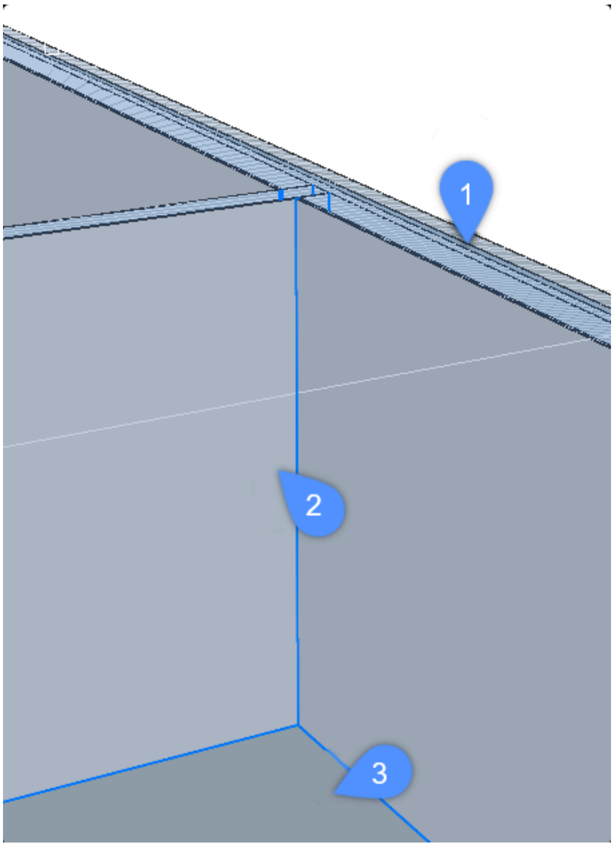
1 Launch BIMPROPAGATECORNER.

1 **You are prompted:**

Select at least three planar reference solids that form a 3D corner. The detail will be copied to similar corners. [selection options (?)]:

2 Select the exterior wall (1), the interior wall (2), and the floor slab (3).

3 Press **Enter** to accept.



You are prompted:

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied as part of the corner detail. [selection options (?)]:

If you have extra solids in your detail, i.e. an insulation block, you can select this as an extra detail object and it will be copied to the other nodes when using BIMPROPAGATECORNER.

4 Press **Enter** when ready.

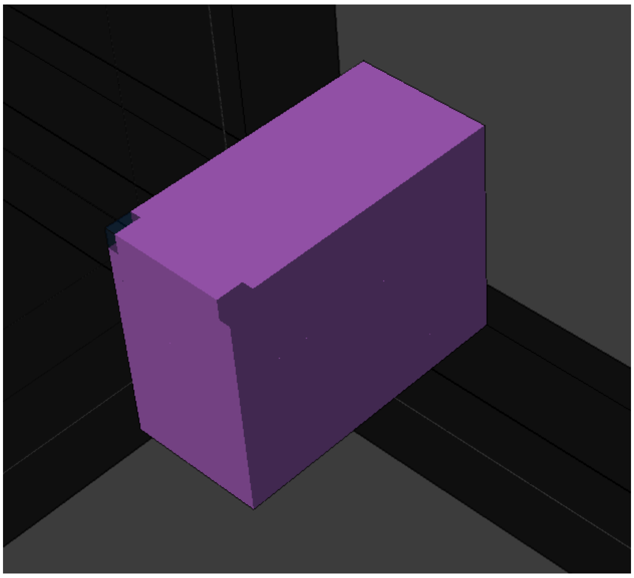
Note: If you don't have any extra details, just press **Enter** to continue.

5 The volume that will be propagated is encased in a purple box.

6 **You are prompted:**

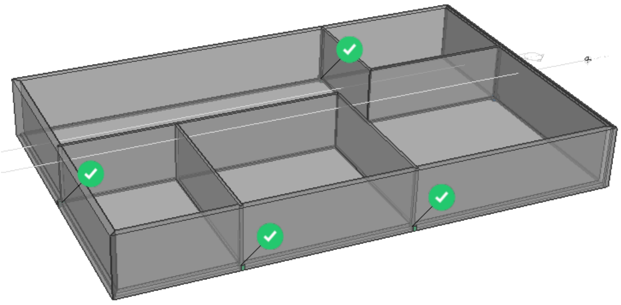
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

7 Press **Enter** to accept the detail.



Note: BIMPROPAGATECORNER shows a 3D volume instead of a 2D section.

8 BIMPROPAGATECORNER now scans the entire drawing for similar locations. When this is done, the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed. A green checkmark means that this suggestion will be applied when pressing **Enter**.



You are prompted: Choose next action [Apply all/Cancel] <Apply all>:

9 Press **Enter** to complete the command.

The nodes where the exterior wall, interior wall, and floor slab come together should now be propagated correctly.

24.8 Parametrize details

24.8.1 Commands

BIMPARAMETRIZEDETAIL

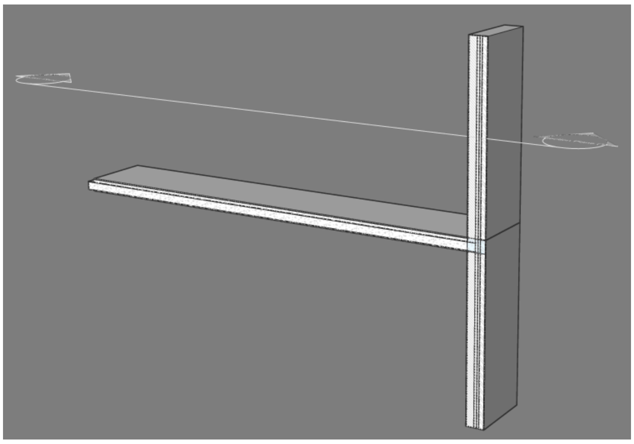
24.8.2 About BIMPARAMETRIZEDETAIL

Use the BIMPARAMETRIZEDETAIL command to automatically generate parameters for a saved detail. Standard parameters, such as angles, offsets in different directions and tolerance parameters are created. These parameters mean you can propagate details to target situations similar to the example below.

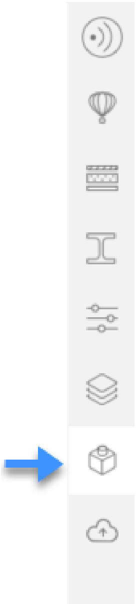
The automatically generated parameters form a framework that can control the position and shape of the reference solids. You can extend this framework adding custom constraints. You can evaluate the effect of these changes by animating the parameters from the **Mechanical Browser** panel. You can add custom constraints to link detail objects to the reference solids and to each other. You can also use the framework parameters in the expression of those constraints.

24.8.3 Parametrize and edit a wall-slab connection detail

1. To open the DWG file of the detail you want to parametrize click on the detail in the **Details** panel. The **View Detail** dialog box displays.
2. Click **Parametrize**, or click **Open** and run BIMPARAMETRIZEDETAIL. The detail DWG file will open and the detail will be parametrized.



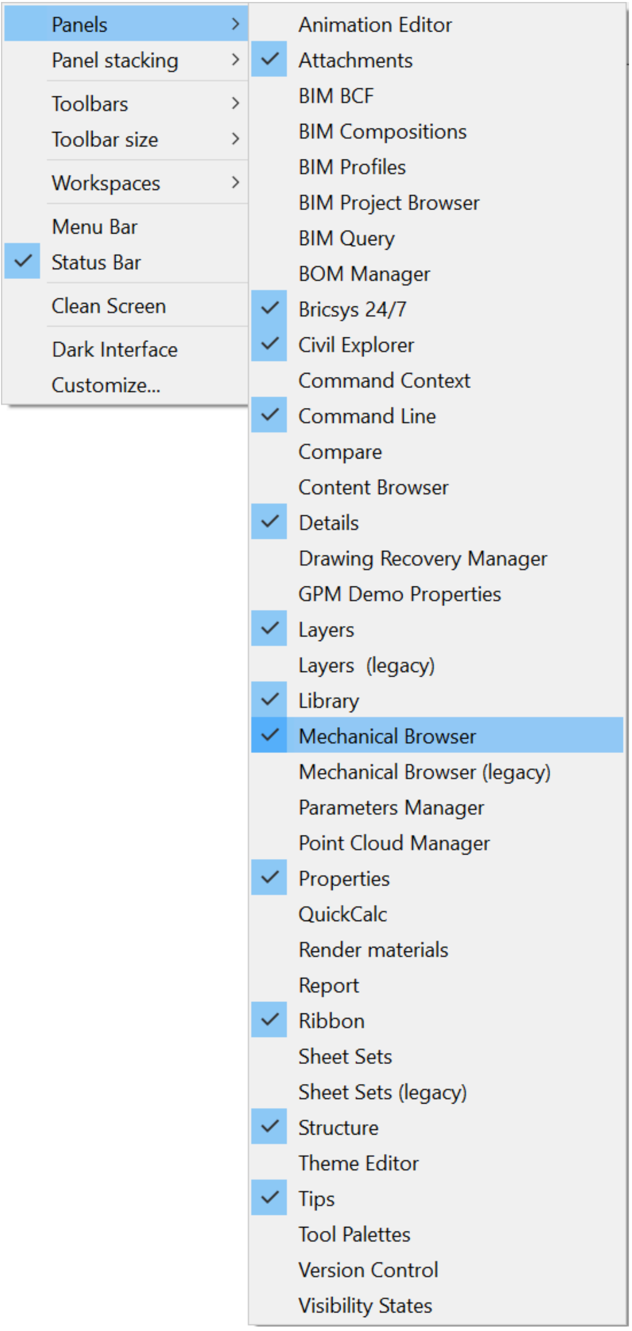
3. Open the **Mechanical Browser** panel.
4. Click the **Mechanical Browser** button in the tool panel set.



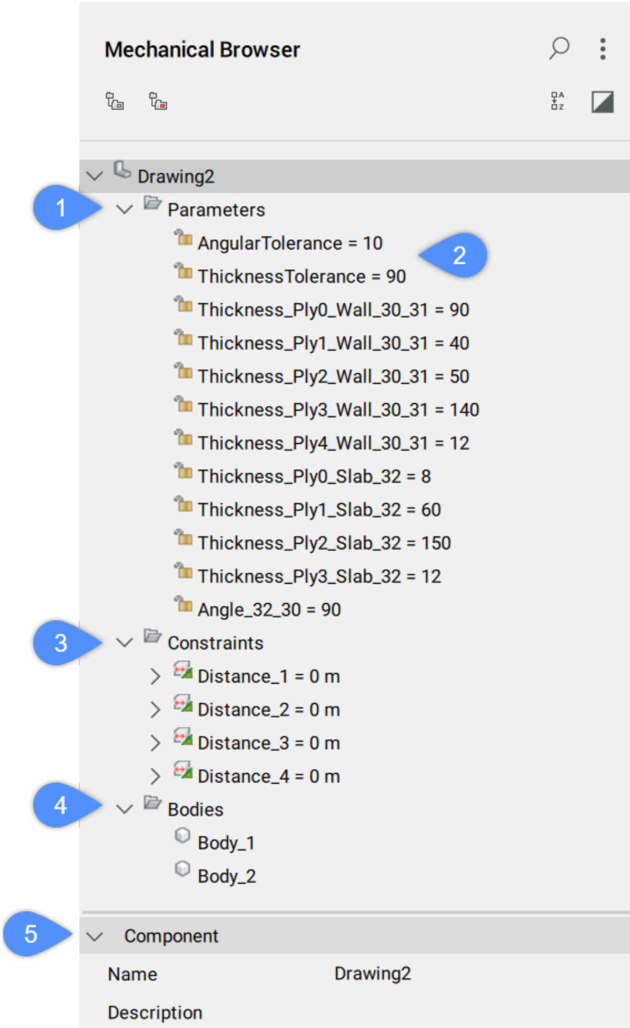
5 If the **Mechanical Browser** button isn't visible yet, move the cursor to the toolbar or ribbon panel and right-click. A context menu displays.

Expand the **Panels** option.

Select **Mechanical Browser** in the context menu.

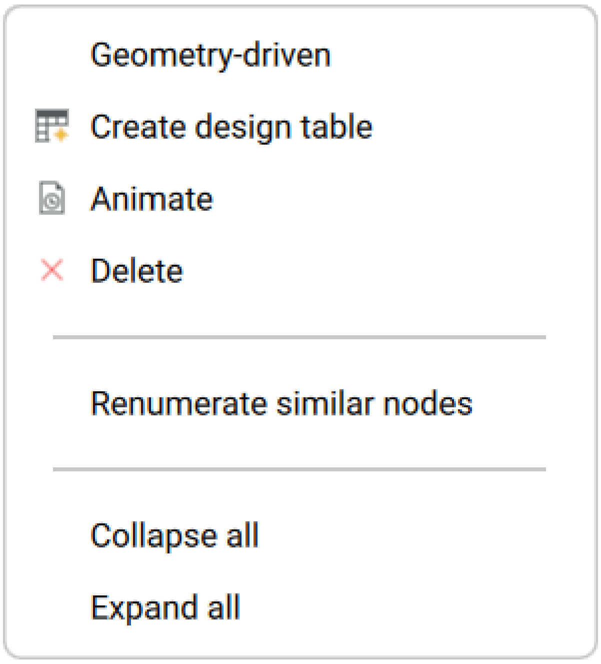


6 Once you have clicked on the **Mechanical Browser** button, the **Mechanical Browser**panel displays:



Note: You can also use the MECHANICALBROWSEROPEN command.

Parameters (1): Right-click a parameter to open a context menu:



- **Geometry-driven**: if checked, it makes the parameter geometry-driven.
- **Create design table**: creates a design table to drive parametric block parameters.
- **Animate**: animates models by means of parameters.
- **Delete**: deletes the selected parameter.
- **Renumerate similar nodes**: renumerates all nodes of the same type and the same level.
- **Collapse all**: collapse the whole **Mechanical Browser**.
- **Expand all**: expand the whole **Mechanical Browser**.

Tolerance Parameters (2): displays the maximum and minimum variation of an angle, thickness or dimension from the original parameter value.

Constraints (3): contains constraints that you can delete, modify or add.

Note: Some default constraints are generated that link the detail volume and the detail objects to the reference solids. You can delete and replace them with other constraints.

Bodies (4): displays the bodies in the model.

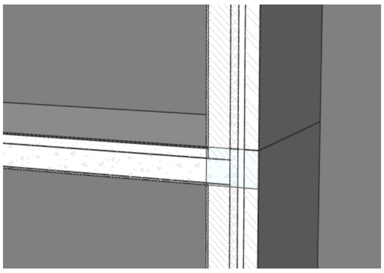
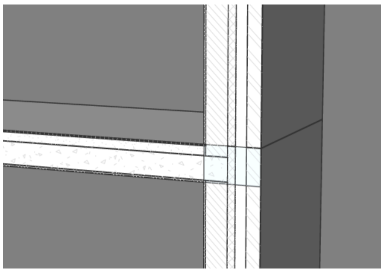
Component (5): Displays the properties of the detail file.

7 Edit the parameters.

8 The parameters that were generated have specific values to ensure the detail looks exactly as it was created. If you change these values, the detail will change.

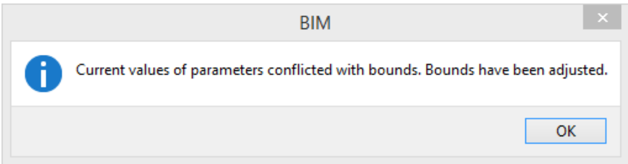
For example:

- Change the **Thickness_Ply1_Wall_30_31** parameter from 40 to 70 (in the **Expression** field). The new detail is visibly different in the model space:

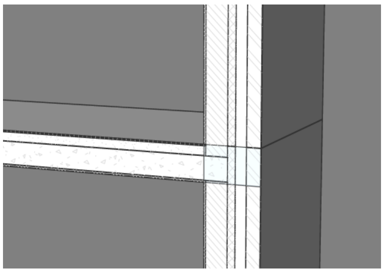
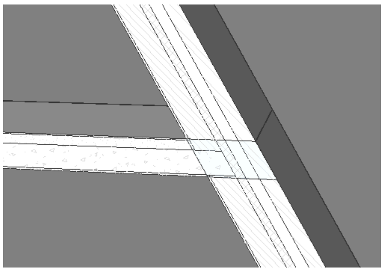
Before	After
	

- Change the **Angle_32_30** parameter from 90 to 120. The new angle between the wall and the slab is different in the model space.

  A warning pops up:

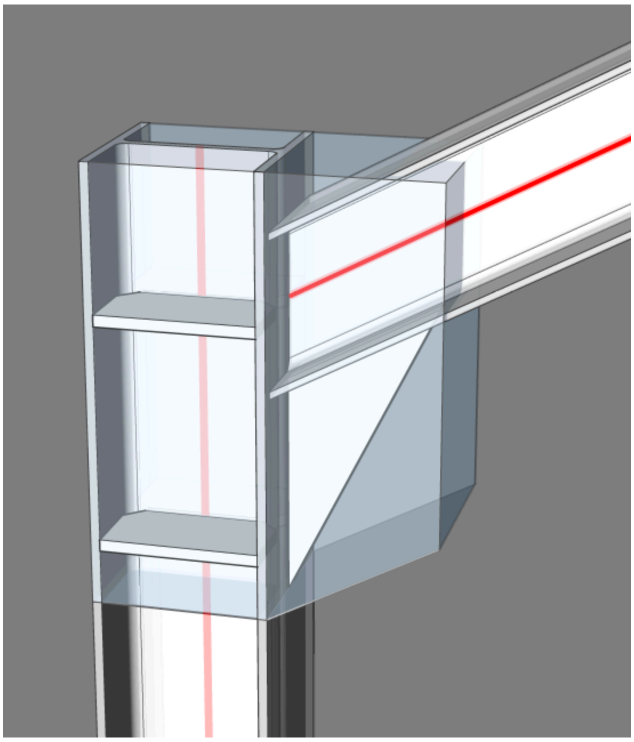
  This is because the **AngularTolerance** parameter was set to 10. The new **AngularTolerance** parameter automatically changes to 30 because you changed the **Angle_30_31** parameter by 30 degrees.
- Click **OK**

Before	After
	

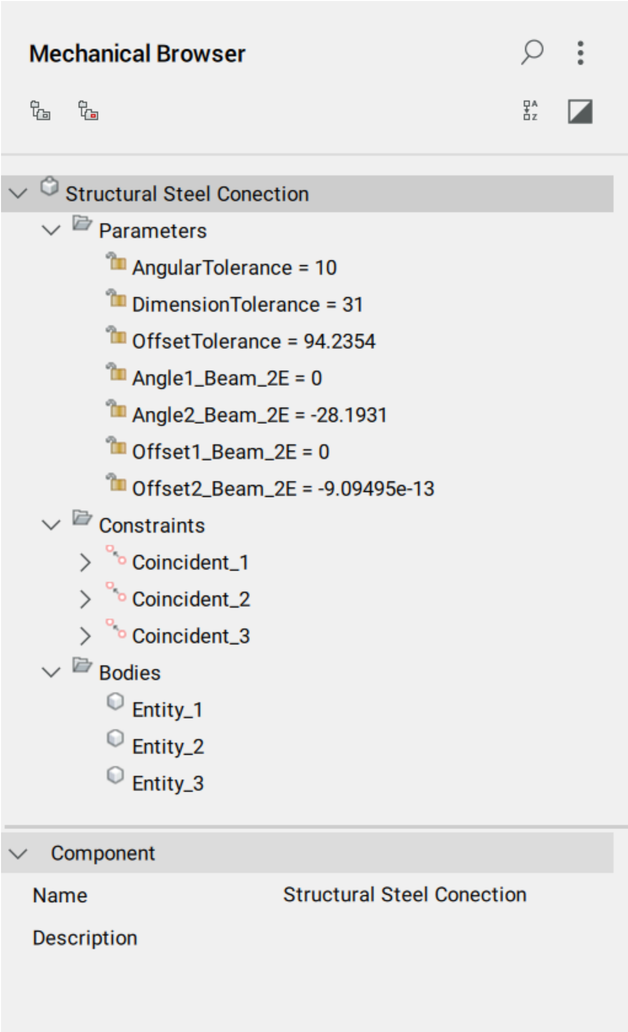
24.8.4 Parametrize and edit a structural steel connection

1. To open the DWG file of the detail you want to parametrize, click on the detail in the **Details** panel. The **View Detail** dialog box displays.
2. Click **Parametrize**, or click **Open** and run BIMPARAMETRIZEDETAIL.

   The detail DWG file will open and the detail will be parametrized.



3 Open the **Mechanical Browser**:



Note: For connections between linear reference solids, only the framework parameters are automatically generated. You must manually add constraints to specify the behavior of detail objects in relation to the reference solids. In this example, three coincidence constraints have been manually added: two to make sure that the connection planes of the triangular steel plate maintain coincident with the flanges of the steel column and girder, and one to make sure that the end section of the girder maintains coincident with the column flange. You can add coincidence constraints before or after the parametrize tool. If you add the coincidence constraints after you can see how the reference solids change with new parameters. This makes it clear which constraints have to be added to make all detail objects move correctly.

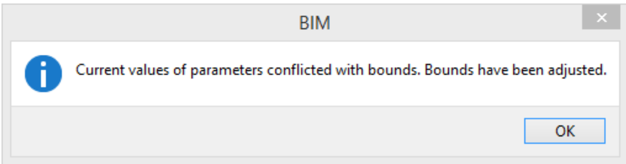
4 Edit parameters.

5 The parameters that were generated have specific values that make sure the detail looks exactly as it was created. If you change these values, the detail will change.

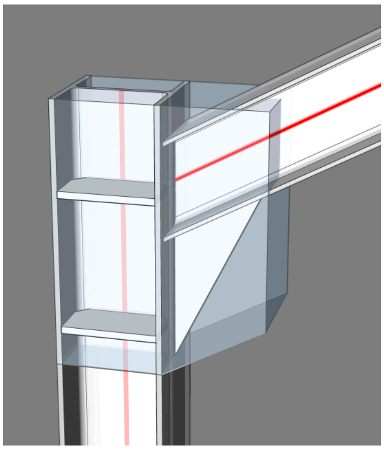
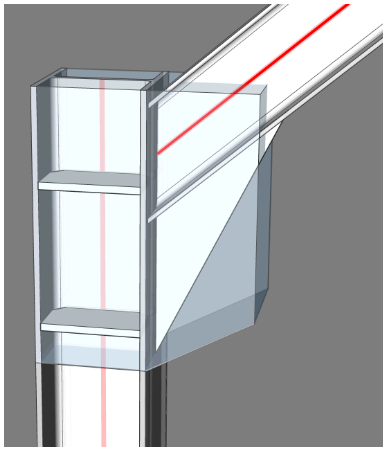
For example:

- Change the **Angle2_Beam_2E** from -28.1931013282 to -40. The column and the girder change in the model space.

A warning pops up:

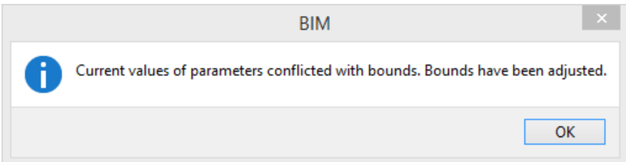


This is because the **AngularTolerance** parameter was set to 10. The **AngularTolerance** parameter automatically changed to 11.8069 because you have changed the **Angle2_Beam_2E** by 11.8069 degrees. Click **OK**.

Before	After
	

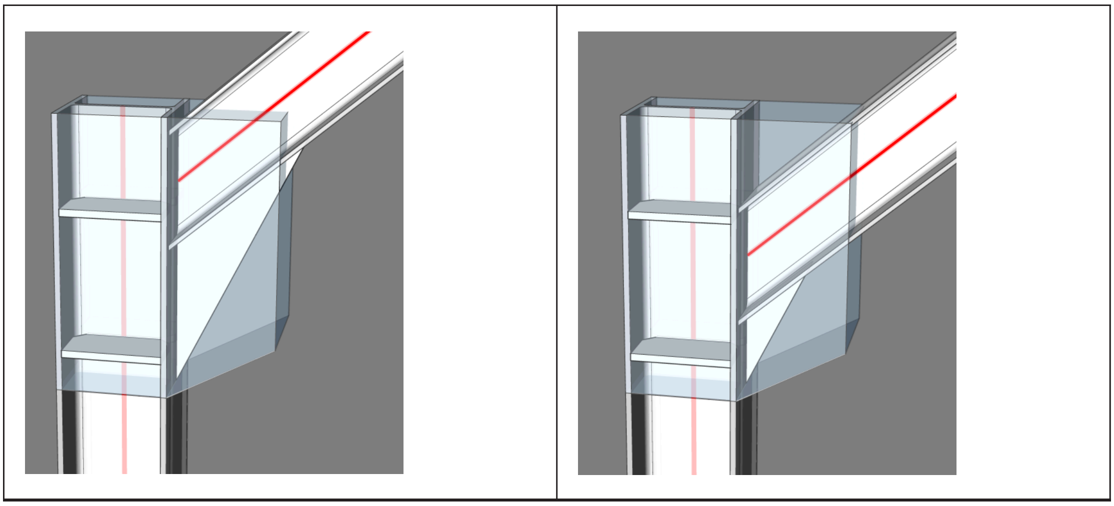
- Change the **Offset1_Beam_2E** from 0 to 200. You see the girder moves 200 mm downwards.

  A warning pops up:



This is because the **OffsetTolerance** parameter was set to 31. The **OffsetTolerance** parameter automatically changed to 200 because you changed the **Offset1_Beam_2E** parameter by 200 mm. Click **OK**.

Before	After



24.8.5 Propagate a detail in a project

There are different methods to propagate a detail:

- With the **Details** Panel: propagate a detail from the project on all similar connections in the project.
- With the BIMPROPAGATEFROMFILE command: propagate a detail from a file.

Fewer suggestions are given by the BIMPROPAGATE command for a detail that is not parametrized. This is because a detail that is not parametrized has fewer possible locations.

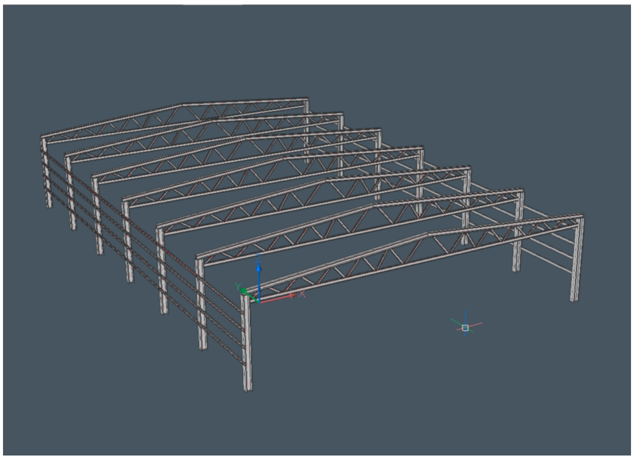
For example: if a non-parametrized detail of a wall-slab connection is propagated, BricsCAD BIM won't be able to propagate this detail to a wall-slab connection when the composition is different or a particular ply in the wall differs 10 mm from the corresponding ply in the wall of the original detail. To solve this problem parametrize the detail.

The BIMPROPAGATE command can adapt a detail with parameters to situations that differ from the original (for example: another corner, a smaller profile, etc.). You can decide how much the connections can differ from the original with the tolerance parameters.

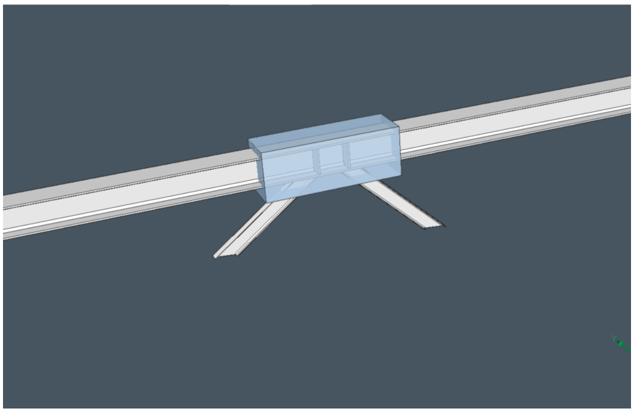
Different tolerance parameters:

- **Angular**: Sets the tolerance on angle parameters of the detail in degrees. Angles in the target situation cannot deviate more than this tolerance value from the original angles in the detail.
- **Dimension**: Sets the tolerance for the profile dimensions of linear reference solids in the detail. Profile widths and heights in the target situation cannot deviate more than this tolerance from the original widths and heights in the detail.
- **Offset**: Sets the tolerance for offset parameters of the detail. Offsets in the target situation cannot deviate more than this tolerance from the original offsets in the detail.
- **Thickness**: Sets the tolerance for the total thickness of planar reference solids in the detail. Thicknesses in the target situation cannot deviate more than this tolerance from the original thicknesses in the detail.

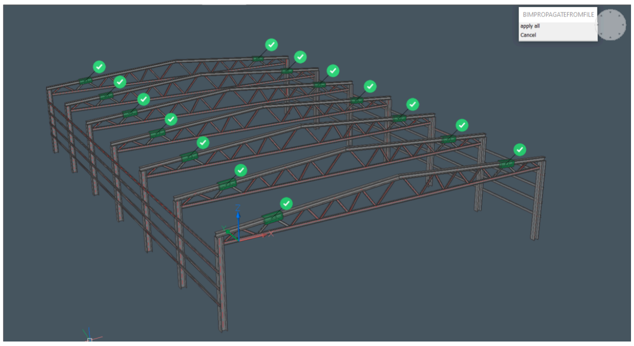
In the following project, there are a lot of similar connections present. However, the angles of the connections vary and the lower beam in the truss and has a different profile size than the upper beam.



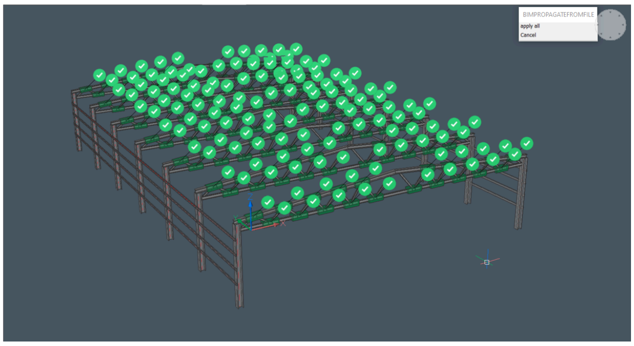
Propagate a detail in the project:

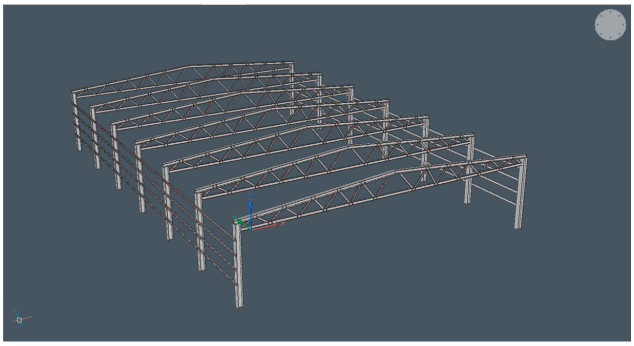


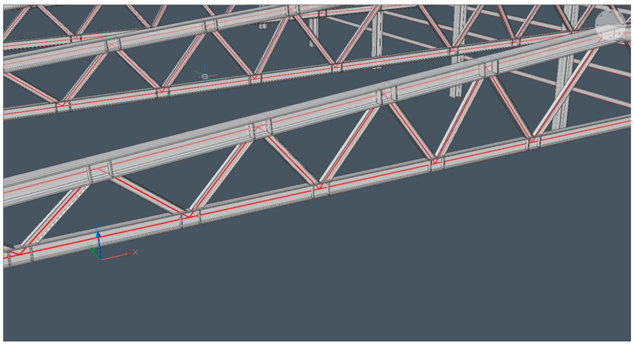
The **Mechanical Browser** panel displays the parameters. The **AngularTolerance** parameter is set to 0 degrees, the detail will only propagate to connections with exactly the same angle.



Change the **AngularTolerance** parameter to 45 degrees. The detail will propagate to all the similar connections. This is because none of the connections exceed the tolerance value.





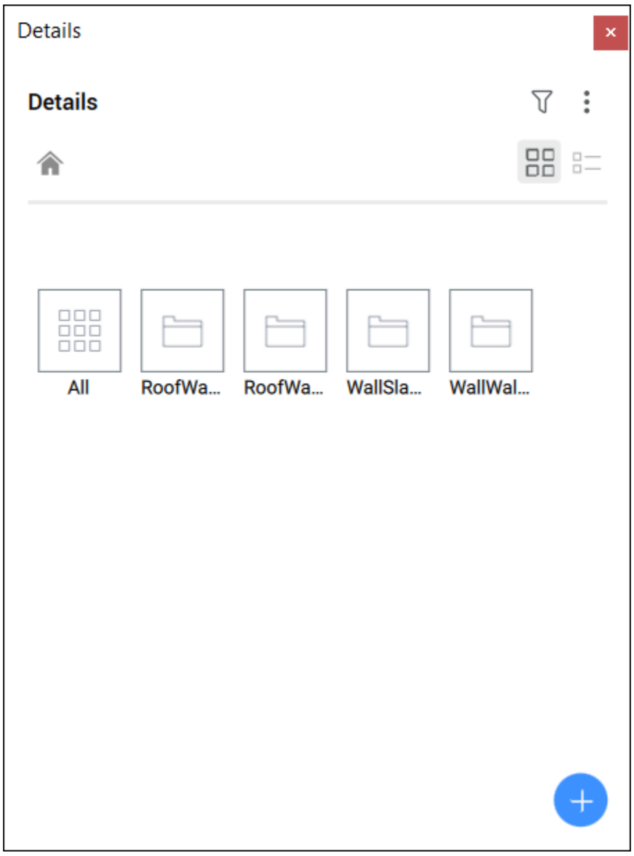


24.9 Using the Details panel

24.9.1 Opening the Details panel

If the **Details** panel doesn't display in the panel, right-click in the panel and select **Panels** > **Details** or execute the DETAILSPANELOPEN command.

The **Details** panel displays and the **Details** button ( ) is added to the collapsible panelset at the right-hand side of the BricsCAD application window.



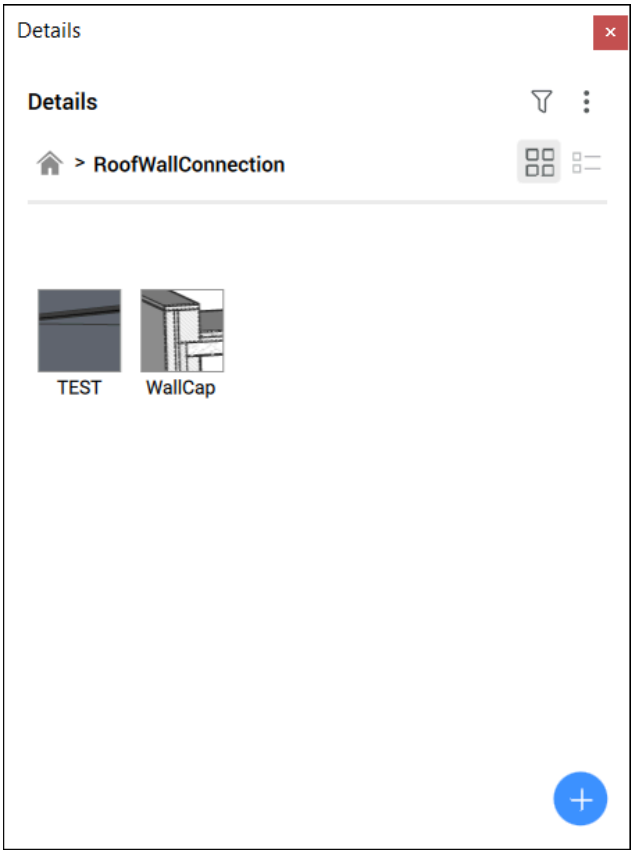
Click on the **Details** button () to expand/collapse the **Details** panel.

24.9.2 Saving details to the Details panel

After you created a new detail, it's possible to save it to the **Details** panel. This way, you can use the same detail in other files. Use the command BIMCREATEDETAIL or click **Create Detail** in the **Details** panel.

24.9.3 Viewing a detail

1 Click the category of the detail you want to view. The details assigned to that category display.

1 

2 Click the detail you want to view.

3 The **View detail** dialog box displays.

Managing tags

1 In the **Details** panel, click on the detail you want to adjust.

The **View Detail** dialog box displays.

2 Go to the **Solid Reference Tags** category to manage the solid reference tags.

You can only delete these tags, you cannot add new tags.

3 Click in the add tags button and type the name for the new tag, then press ENTER to add it into **General Tags**.

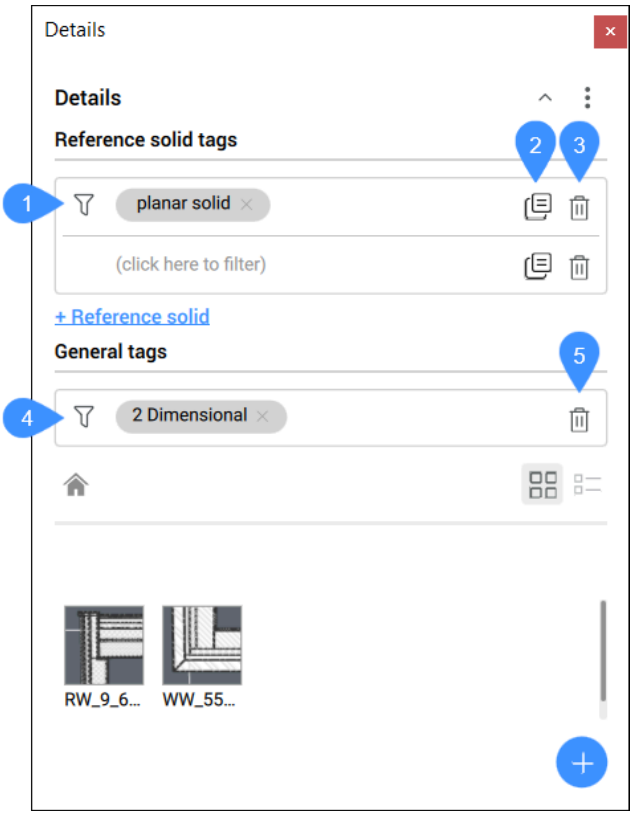
The new tag is added to the **General Tags** category in an orange box.

Repeat this procedure to add more tags.

4 Click **Save** to save the changes and close the **View Detail** dialog box.

Searching details with tags

Click the filter icon ( ) to open the filter options. Here you can search the library for specific details using tags. You can add one or more tags at once to filter your result and select all matching details.



1 Add Reference solid tag

2 Copy Reference solid

3 Delete the Reference solid

4 Add General tag

5 Delete General tag

Add Reference solid tag

Click in the box to open the drop-down list with Reference solid tags and select a tag from the list. You can also type a name of a tag to add the tag.

Note: You can add several tags per reference solid. to add a reference solid, click on **+ Reference solid**.

Copy Reference solid

Copies the selected Reference solid with its tags.

Delete the Reference solid

Deletes the selected reference solid.

Note: you can click the cross next to the tag name to delete only one tag from a Reference solid

Add General tag

Click in the box to open the drop-down list with General tags and select a tag from the list. You can also type a name of a tag to add the tag.

Note: You can add multiple tags.

Delete General tag
Deletes all the General tags.

Note: you can click the cross next to the tag name to delete only one tag.

24.9.4 Editing a detail

1 Click the detail you want to edit to open the **View Detail** dialog box.

2 Click the **Open** button.

The .dwg file of the detail opens in a separate drawing tab.

3 Make the necessary adjustments to the detail.

4 Save the detail .dwg file.

5 The detail is updated in the **Details** panel.

6 The existing details won't be updated automatically. To replace the existing details by the new one, follow the steps below to propagate a detail.

24.9.5 Propagating a detail

You can propagate a detail to all of the similar connections in your project.

1 Click a detail in the library.

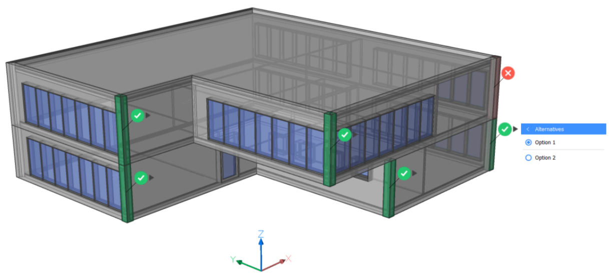
1 The **View Detail** dialog box displays.

2 Click **Propagate**.

To apply all suggestions, press Enter.

To reject a suggestion click on the check mark. A red cross icon replaces the check mark. Click the icon again to accept the suggestion. If there are several alternatives, the alternatives display when you hover over the check mark and you can select the option you want.

Press Enter to accept all accepted suggestions.



You can also propagate a detail in your project using the BIMPROPAGATEFROMFILE command. This command allows you to propagate detail files that are not in the library.

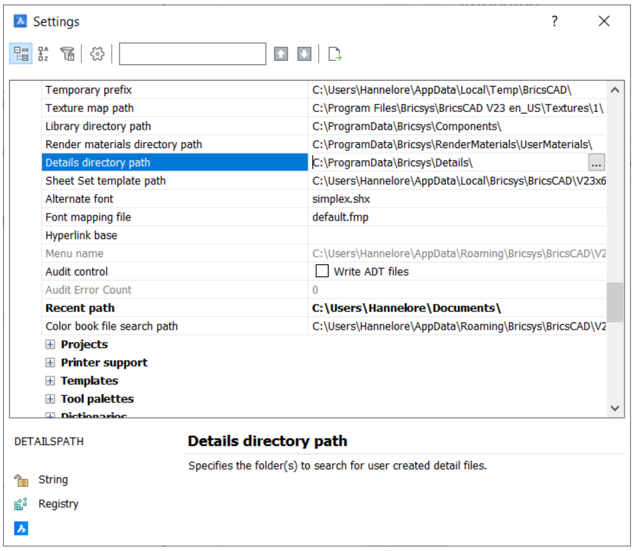
24.9.6 Sharing details

You can save details as a .dwg file and share the details folder/library. Save the detail drawings at the correct location to display it in the **Details** panel.

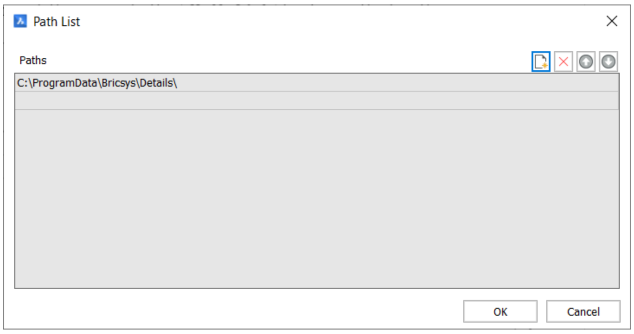
1 If you want to make a new detail to share, you can create a new detail first, as is explained above.

1 The following steps apply both to new details as well as existing details.

2 All the details you have created are automatically saved in the folder, and its subfolders, which is set through the DETAILSPATH system variable. By default, this is *C:\ProgramData\Bricsys\Details*.



Then click on the three dots next to the existing name in the **Path List** to open the **Path List** dialog box. Here you can change the location.

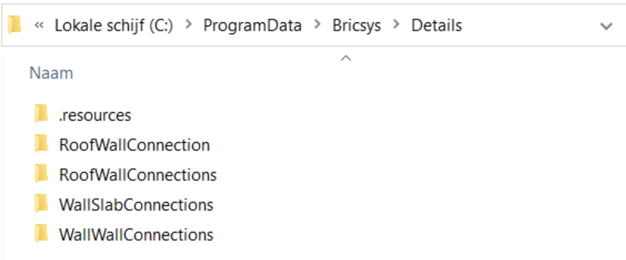


You can also create additional paths.

Click the New icon ( ), then click on the three dots of the new row to add a path for the **Details** library.

3 If you want to share a folder, go to the file explorer on your computer and follow the path to the details folder. If you haven't created any details yet, this folder only contains the folder **.resources**.

If you have created some details, you can find the same folders here as in the **Details** panel.



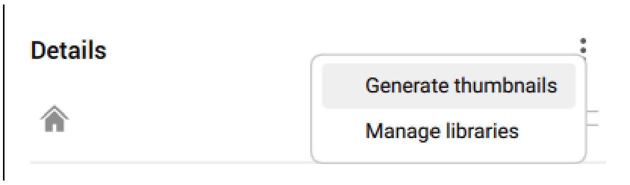
4 Copy the folder that you want to share and turn it into a zip file. You can share this zip file with others.

5 If you receive a zip file you should unpack it and copy the folders containing the details to the folder *C:\ProgramData\Bricsys\Details*.

Note: Make sure the details are saved in a folder that you place in the **Details** folder. If you just place the dwg files in the **Details** folder, they will not appear in the **Details** panel.

6 To load the new details in your library panel, go to the **Details** panel, open the **Menu** and click **Generate thumbnails**.

The new details folders will appear in the **Details** panel.



24.10 Rhino/Grasshopper integration

24.10.1 About

Rhinoceros3D is a 3D modeler developed by Robert McNeel & Associates of Seattle, Washington USA. Rhino is used to create, edit, analyze, document, render, animate and translate NURBS (curves, surfaces, solids, point clouds and polygon meshes).

Grasshopper is a visual programming language environment that runs within the Rhinoceros 3D computer-aided design application. It is tightly integrated with Rhino's 3D modeling tools.

24.10.2 Installing Rhino/Grasshopper connection

1 Download Rhino from here.

Note: Users can use the Rhino/Grasshopper Connection within BricsCAD with trial license keys of both current version of BricsCAD BIM and Rhino 7 or Rhino 8. Since the release of BricsCAD V21, keys for older versions of Rhino will not work for this integration.

2 Download the Grasshopper-BricsCAD Connection from the BricsCAD Application Catalog. The installer copies all required files to the *Program Files\Bricsys\Grasshopper-BricsCAD Connection* folder.

Note: The Grasshopper-BricsCAD Connection currently available from the Application Store will work in the current BricsCAD Pro, BIM, Mechanical or Ultimate versions.

Note: The Grasshopper-BricsCAD connection works with Pro, Mechanical, BIM and Ultimate for creating geometry. However, the BIM specific functionality (adding classification, spatial locations, profiles, properties, IFC export) will only work with a BricsCAD BIM or a Ultimate license.

3 Restart BricsCAD.

4 To launch Grasshopper or Rhino, open and save a new drawing. Then use the buttons in the new tab that is displayed in the ribbon or launch with the Quad.

To launch a Grasshopper file from BricsCAD

1 Open an empty BricsCAD file.

2 Save it.

3 Open Grasshopper with the button in the new **Grasshopper** tab inside BricsCAD.

4 You can close the Rhino pop-up window. Rhino will continue to work in the background.

5 In the **File** tab of Grasshopper, choose **New document** to open a new and empty Grasshopper script.

6 Choose **Open document...** to open an existing Grasshopper script. You can use any script you made with Grasshopper in the past and open it in BricsCAD, as long as the input geometry from another program has been internalized. (To internalize data right-click the input geometry component and hit **Internalise data**.)

7 The geometry you created with the script should now pop-up in your BricsCAD model space, displayed in red. Make sure you don't close the **Grasshopper** window, otherwise the preview geometry will disappear, as the Grasshopper document is closed.

8 If your Grasshopper script is open and you still don't see the preview geometry, it probably means your Grasshopper file is linked to the wrong BricsCAD document. To link it to the correct one, see **To link a Grasshopper file with a BricsCAD file**.

To link a Grasshopper file with a BricsCAD file

By default, the BricsCAD file that the Grasshopper document will link to the one that was active when you launched Grasshopper from BricsCAD. If you want to link it to another BricsCAD file, you need to:

1 Make the target BricsCAD drawing the current drawing (i.e. the one you see in the model space window of your current session).

2 Link the Grasshopper script to this open drawing. You do that by clicking the Link icon in the bar just above the Grasshopper canvas that contains the script that you are trying to link.

To launch a sample file

The sample files are provided with the Grasshopper-BricsCAD Connection. They can show you what is possible with the Connection. They are stored in the *Program Files\Bricsys\Grasshopper-BricsCAD Connection* folder.

1 Open up a sample .dwg file by opening a .dwg from the *Program Files\Bricsys\Grasshopper-BricsCAD Connection* folder.

2 You will get a warning that the file is read-only. This happens because the document is located in a secured folder. Click **Yes** to open as read-only.

Optional: If you don't want this warning, or if you want to edit the file, copy-paste the sample files into one of your own folders and open them from there instead.

3 From inside Grasshopper, choose **File** > **Open Document**. Go to the *Program Files\Bricsys \Grasshopper-BricsCAD Connection* folder again.

4 Choose the .ghx-file with the same name as the .dwg file.

5 The sample file now opens and you should see a red preview of the geometry in your BricsCAD model space.

Make sure you don't close the **Grasshopper** window, otherwise the preview geometry will disappear, as the Grasshopper document is closed. If your Grasshopper script is open and you still don't see the preview geometry, it probably means your Grasshopper file is linked to the wrong BricsCAD document. To link it to the correct one, see **To link a Grasshopper file with a BricsCAD file**.

6 You can play around with the input sliders in the Grasshopper script.

7 If you are happy with your model and would like to convert the preview geometry into real BricsCAD geometry, you can 'bake' the geometry into BricsCAD. You do that by selecting all the **Bake Building Element** components (Shift-clicking them all) and pressing the **Bake** button.

8 A pop-up window appears, just click **OK** to accept.

9 The geometry is now converted into real BricsCAD geometry, even with BIM data attached to it.

10 You won't be able to save your changes to your files, as they were read-only. So, discard your changes or save the files as new ones in a different location.

24.10.3 Customizing the Rhino/Grasshopper connection

The code for the Rhino/Grasshopper Connection is 100% open-source and is available here. This allows you to fully customize your Connector by coding to your heart's content (assuming, of course, that you know how to code).

24.10.4 Uninstall the Rhino/Grasshopper connection

1 Launch **Add or Remove Programs** in Windows.

2 Search for **Grasshopper-BricsCAD connection**.

3 Hit **Uninstall**.

24.10.5 How to draw a box in Rhino and bake into BricsCAD via Grasshopper

1 Open a new BricsCAD file.

2 Save the file.

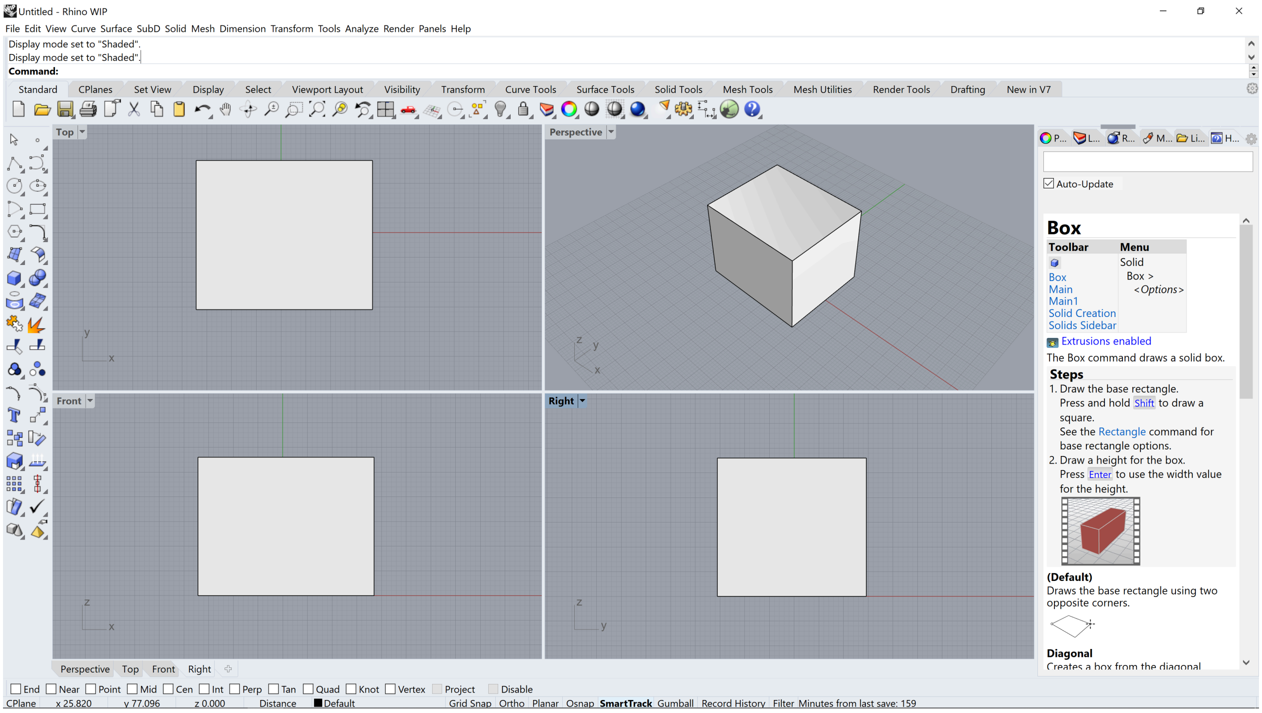
3 In BricsCAD, click on **Grasshopper** > **Rhino**.

4 Draw a Rhino box by typing BOX in the Rhino Command line. (This works in the same way as the BricsCAD BOX command).

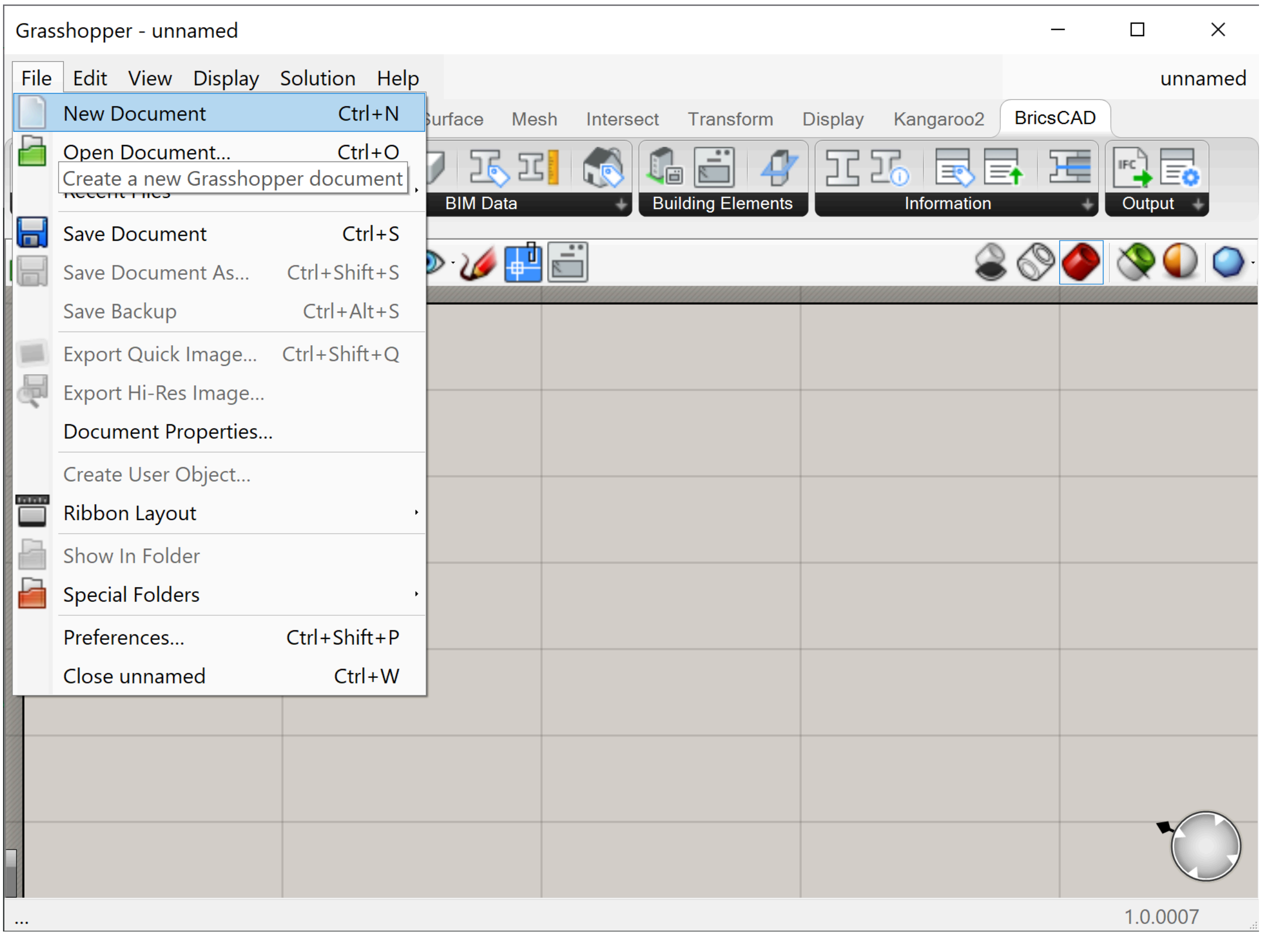
5 Click for the first corner of the base.

6 Click for the other corner of the base.

7 Click to define the height of the box.



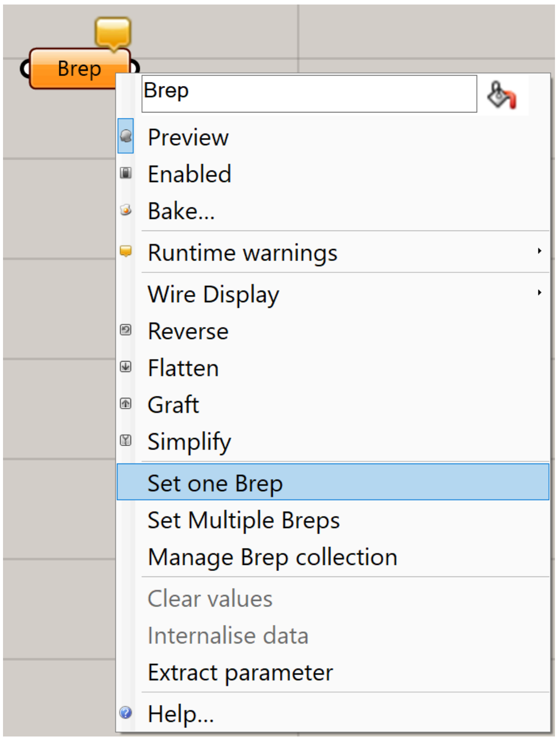
8 To open a new Grasshopper script, inside the Grasshopper environment, click on **Grasshopper** > **Grasshopper** then click on **File** > **New Document**.



9 Save the file.

10 Drag and drop the **Params** > **Geometry** > **Brep component** onto the Grasshopper canvas.

11 Right-click on the component and select **Set one Brep**.

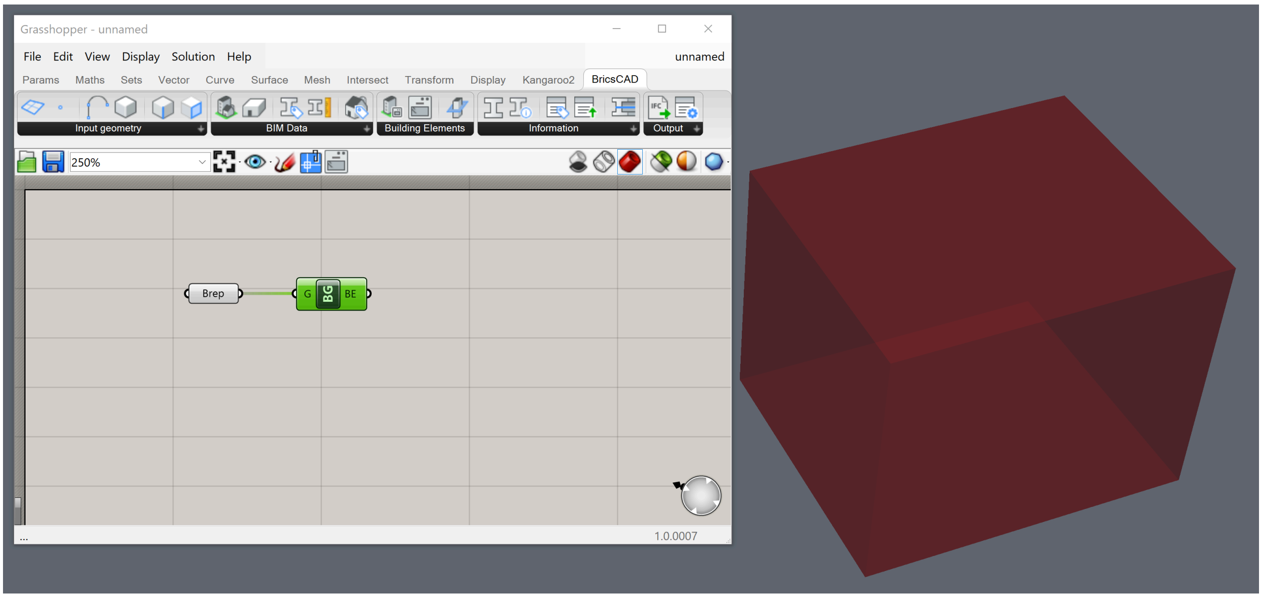


12 Go to the Rhino canvas and select your cube.

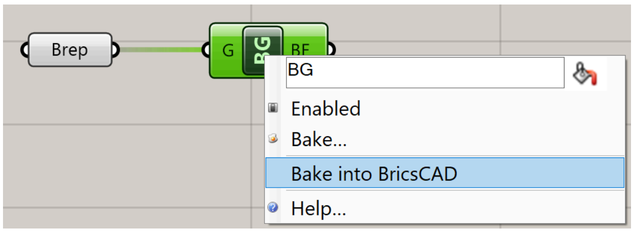
13 The Brep component will change color from orange to grey. If you close the Rhino window and return to the **Grasshopper** window, a preview of the cube will display in the BricsCAD model space.

14 Drag and drop the **BricsCAD** > **Building Element** > **Bake Geometry component** onto the Grasshopper canvas.

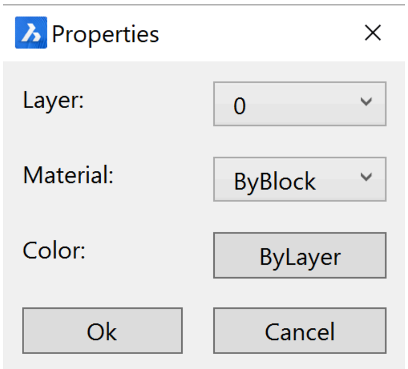
15 To link the two components, click and drag from the Brep component's right dot to the BricsCAD Bake component's left dot.



16 Right-click onto the **Bake Geometry component** and choose **Bake into BricsCAD**.



The following dialog displays. Here you can choose the destination **Layer**, **Material** and **Color**:

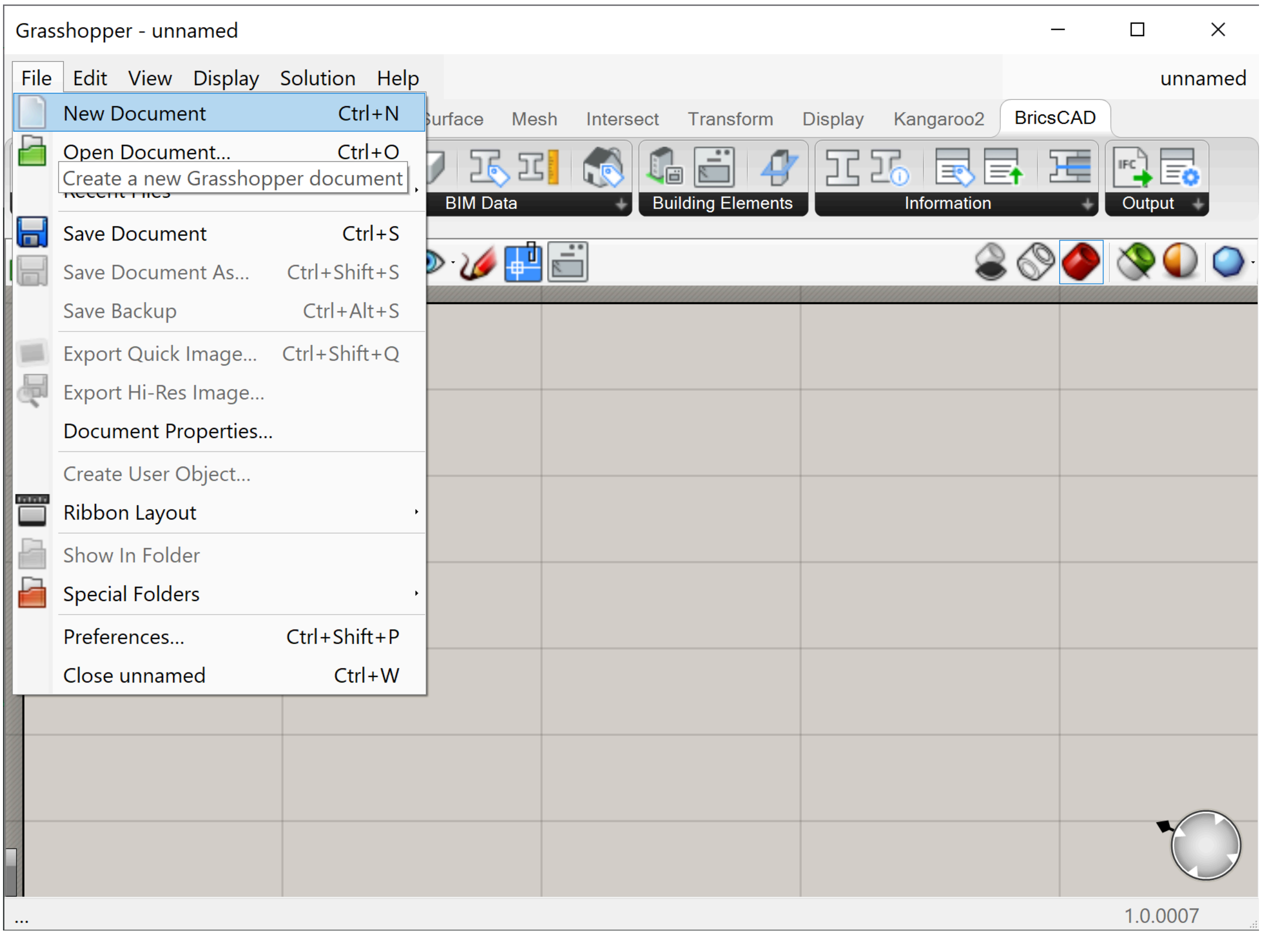


17 Click **OK**. The box is baked as a solid into BricsCAD.

24.10.6 How to draw a box in Grasshopper from a BricsCAD rectangle

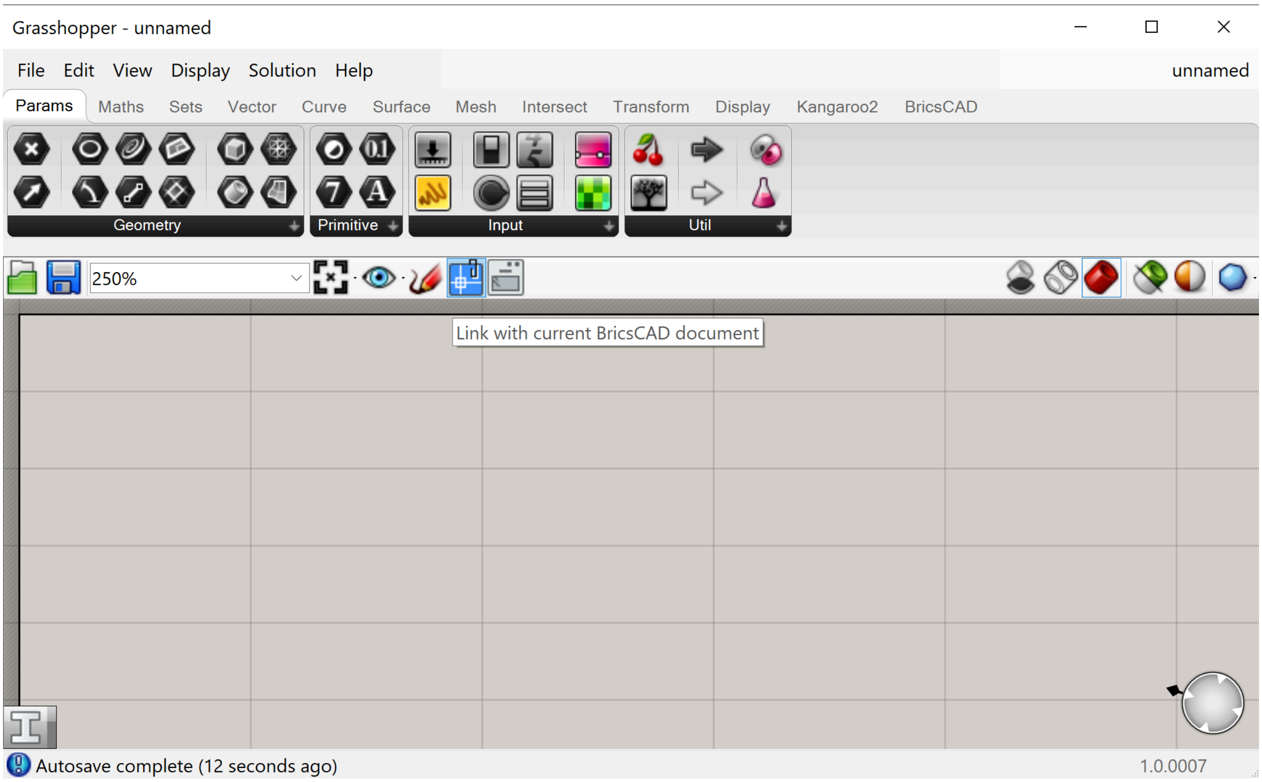
1 To open a new Grasshopper script, click on **Grasshopper** > **Grasshopper** and clicking on **File** > **New**

Document inside the Grasshopper environment. Save the file.



2 Open a new BricsCAD file in a **meter template** and save it.

3 To link the new script to the open .dwg file hit the **Link** button in the **Canvas** Toolbar.



4 Go to the BricsCAD model space and draw a rectangle.

Note: Use a polyline to create irregular geometry.

5 Drag and drop the **BricsCAD** > **Input Geometry** > **Curve component** onto the Grasshopper canvas.

6 Right-click onto the **Curve component** and choose **Set One BricsCAD Curve**.

7 Go to the BricsCAD model space and select the rectangle.

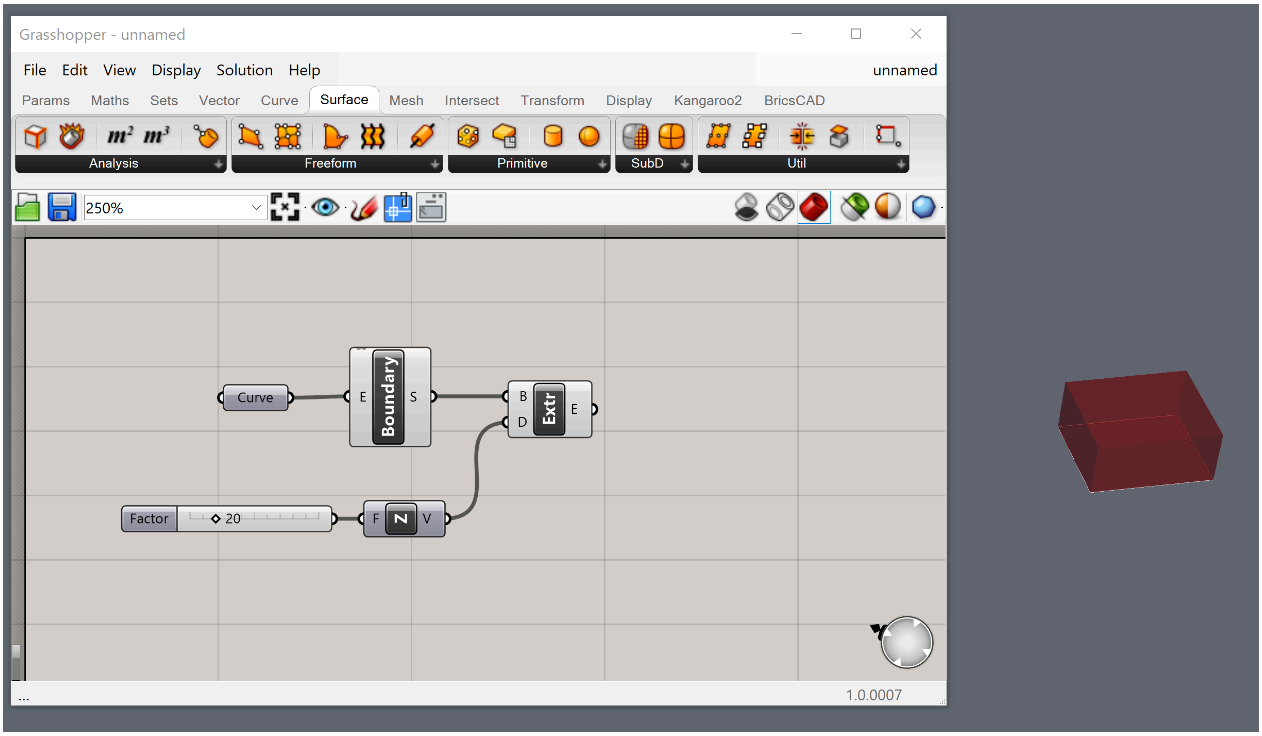
Note: Save time with the Quad: select the curve in BricsCAD and then select **ToGrasshopper** under the **Grasshopper** tab of the Quad. A Grasshopper component appears in the upper left corner of your Grasshopper canvas.

8 Link the Curve component to a **Surface** > **Freeform** > **Boundary Surfaces component**.

9 Link this to a **Surface** > **Freeform** > **Extrude component**.

10 In the Direction input of the Extrude component link a **Vector** > **Vector** > **Unit Z component** with **Params** > **Input** > **Number Slider component**. Set this to the height of your box, 17 for example.

Note: You can edit the range and preciseness of the slider by right-clicking the component and selecting **Edit...**. A dialog box pops up where you can set the digits and min and max value of the slider.



11 You now have drawn a box in Grasshopper, using BricsCAD geometry as an input.

24.10.7 How to split up a box into different floors using Grasshopper

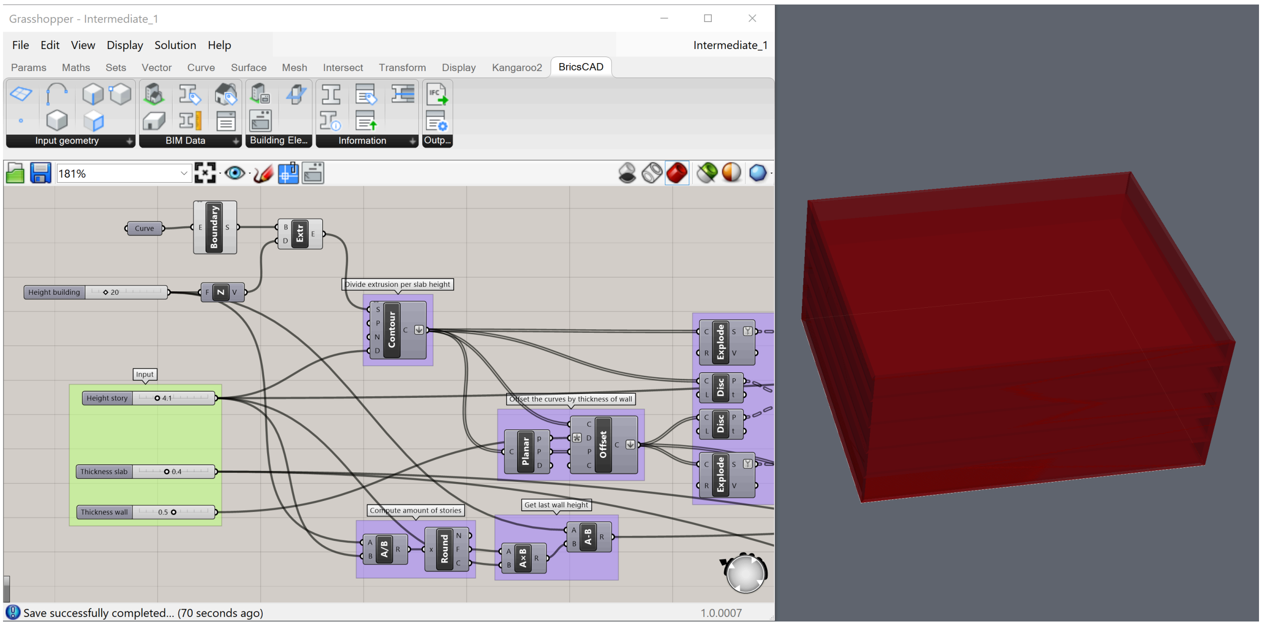
1 Add the **Start_1.ghx**-code to your previous Grasshopper script.

- You do that by downloading the zip file at the bottom of this page and extracting its content.
- Then open the **Start_1.ghx** file by going to **File** > **Open Document...** in your **Grasshopper** window.
- You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.
- Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current document (Start_1).
- You then get a drop-down of all the active Grasshopper scripts, so open the one you saved in **How to draw a box in Grasshopper from a BricsCAD rectangle**.
- Now click on the canvas and hit **Ctrl+V** to paste the **Start_1.ghx**-code.

2 Right-click to rename the number slider from the previous procedure to **Height building**.

3 Link this number slider to input A from the Division component in the **Compute amount of stories** group and to input A from the Subtraction component in the **Get last wall height** group.

4 Link your extrusion to the Shape input of the Contour component.



5 Do some cleaning up of the file:
   - Drag the green **Input** group to the front of your canvas.
   - Add the **Height building** slider to the group by selecting the component and then right-clicking on the group and choosing **Add to group**.
   - Turn off the preview for your Boundary and Extrusion components: select the components (Shift-click them) and right-click on an empty space of your canvas and choose **Preview Off**.

6 **(Optional)** For intermediate files that already have these changes, use **Intermediate_1.ghx** and **Intermediate_1.dwg**. (Located in the zip file at the bottom of this page.)

7 The script you just made allows you to:
   - Divide the extrusion per slab height, using the Contour component.
   - **Offset** the curves by the thickness of the walls.
   - Make the bottom surfaces of the walls (the surfaces are split into lower floor walls and roof walls).
   - Compute the last wall height.
   - **Extrude** the surfaces into roof, slabs, and walls.
   - Adapt the height of the building, the height of the stories, and the thickness of the slabs and walls, by using the sliders in the green **Input** group.

8 Add 3 **BricsCAD** > **Building Element** > **Bake Building Element components** to the end of your Grasshopper script.

9 Attach the extruded elements (the Flip components) to the Geometry input nodes of the **Bake Building Element** components:
   - Connect the extruded lower floor walls and extruded roof walls to one **Bake Building Element** component. (Hold Shift while connecting multiple nodes to one input node).

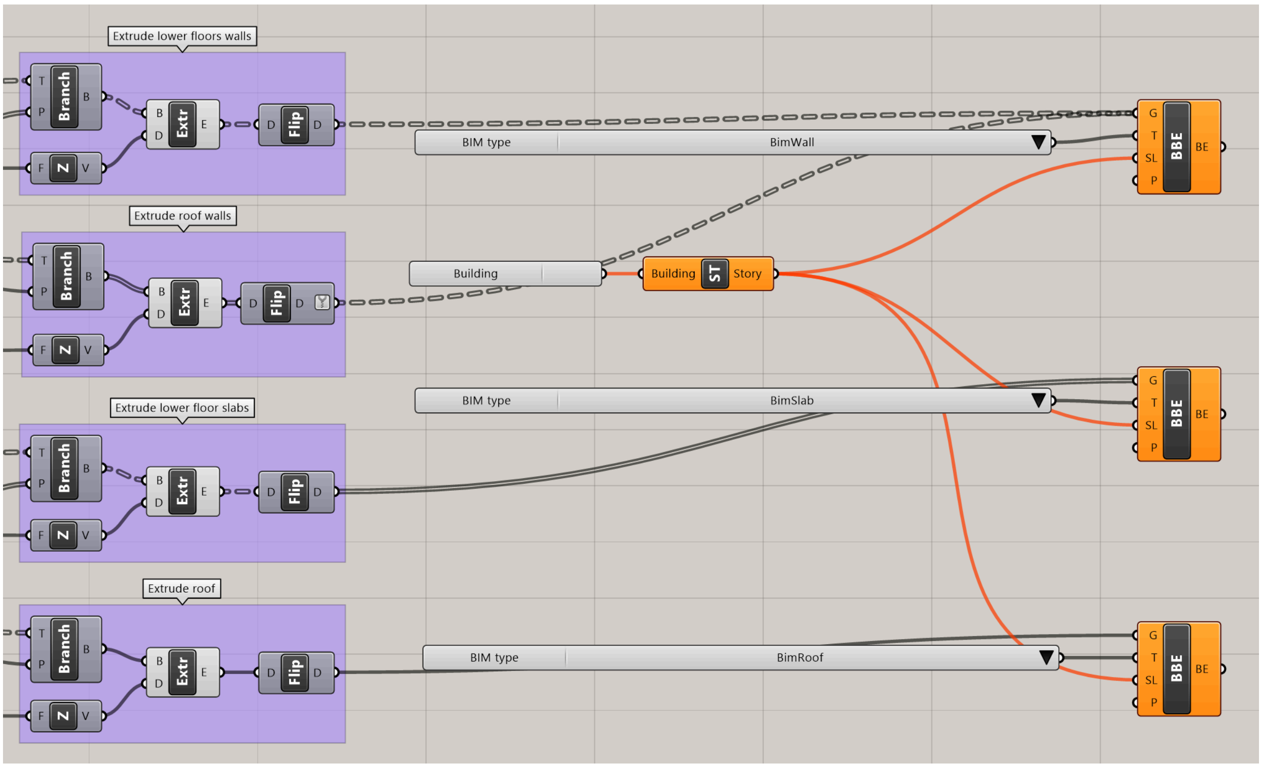
- Connect the extruded lower floor slabs to another **Bake Building Element** component.
- Connect the extruded roof to the remaining **Bake Building Element** component.

10 Add the following components to your canvas from the **BricsCAD** > **BIM Data** category: 1 **Buildings**, 1 **Stories**, and 3 **BIM Types components**.

11 Attach Buildings to the Building input of the Stories component and attach this to the Spatial Location input of all the **Bake Building Element** components.

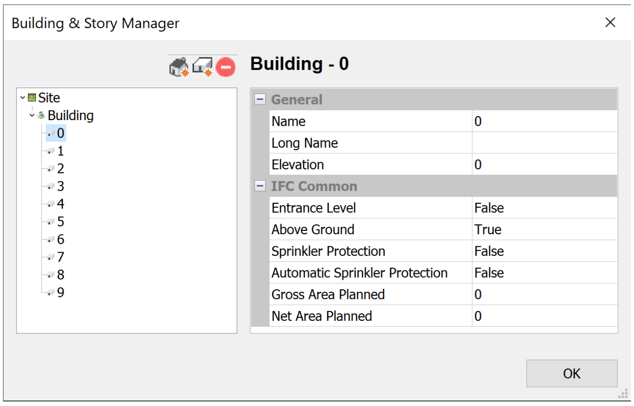
12 Attach BIM Types (set respectively to **BimWall**, **BimSlab**, and **BimRoof** in the selection menu) to the Element Type input of the **Bake Building Element** components of the extruded walls, the slabs, and the roof respectively.

13 The following warning will display: **Input parameter Building failed to collect data**. This is because, when no buildings are defined in the BricsCAD drawing, Grasshopper fails to collect input.

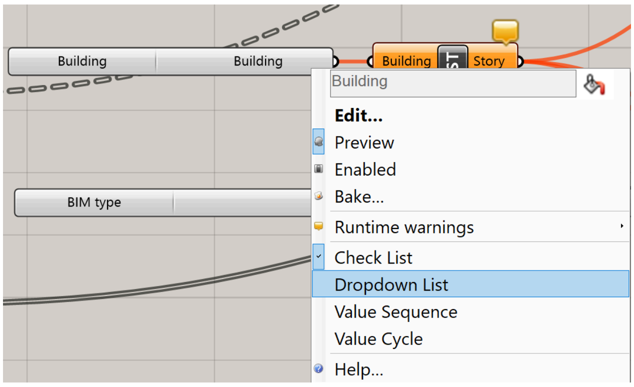


14 To make spatial locations for the component to detect, go to BricsCAD and click the **Spatial Locations** tool in the Ribbon.

15 Add a building and a few stories (more stories than you want to divide your building into).



16 Go to Grasshopper and right-click the Buildings component and put it onto **Drop-down List** instead of **Check List**.

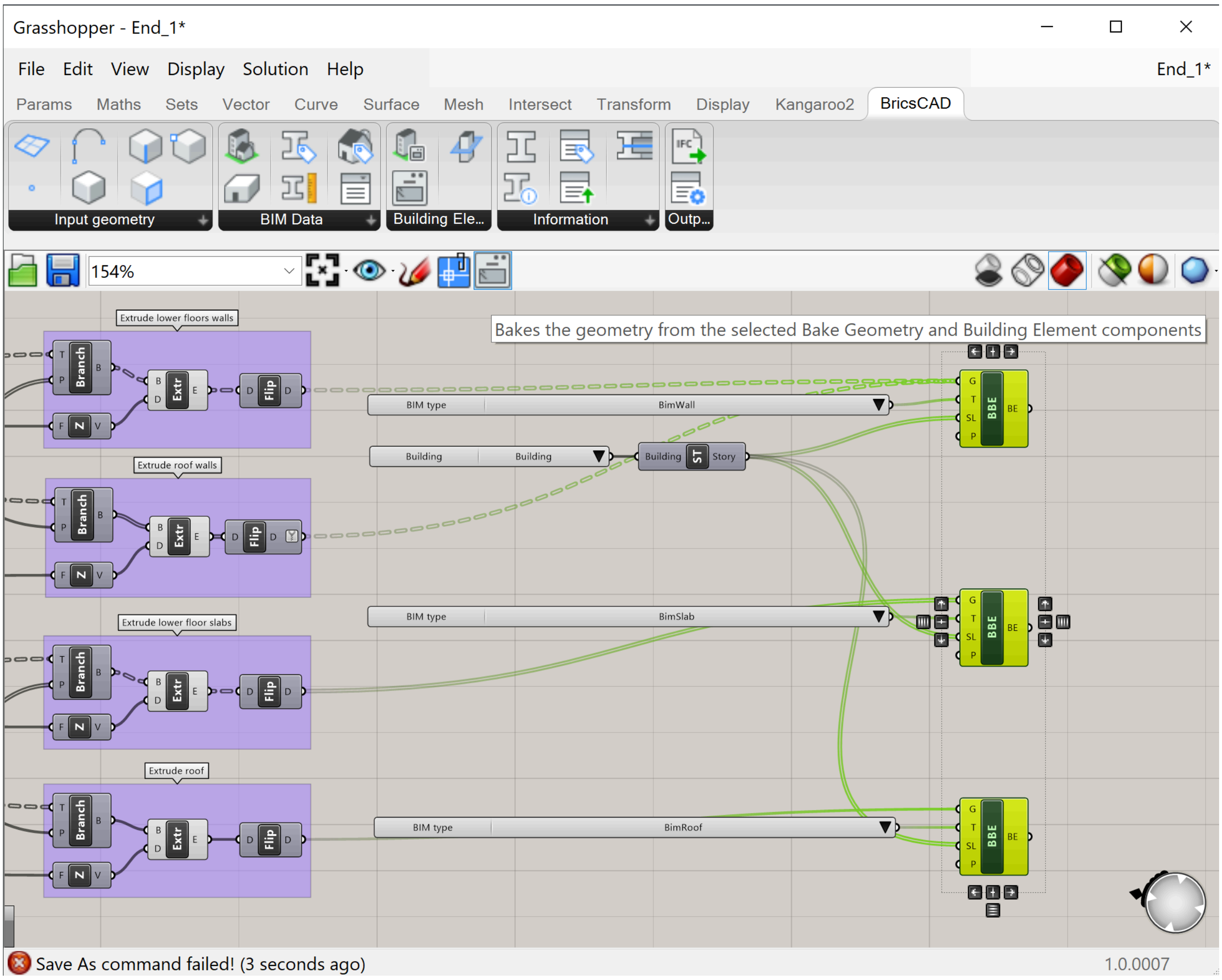


17 Select the newly made building in the drop-down list.

18 Shift-select all the **Bake Building Element** components and choose the **Bake** option from the **Canvas** Toolbar. Click **OK** to accept the **Layer**, **Material**, and **Color**.

19 The drawing will now have different floors.

Note: Check the properties to see that the geometry is correctly classified as **Wall**, **Slab** or **Roof**. These elements will also have a spatial location assigned to them.

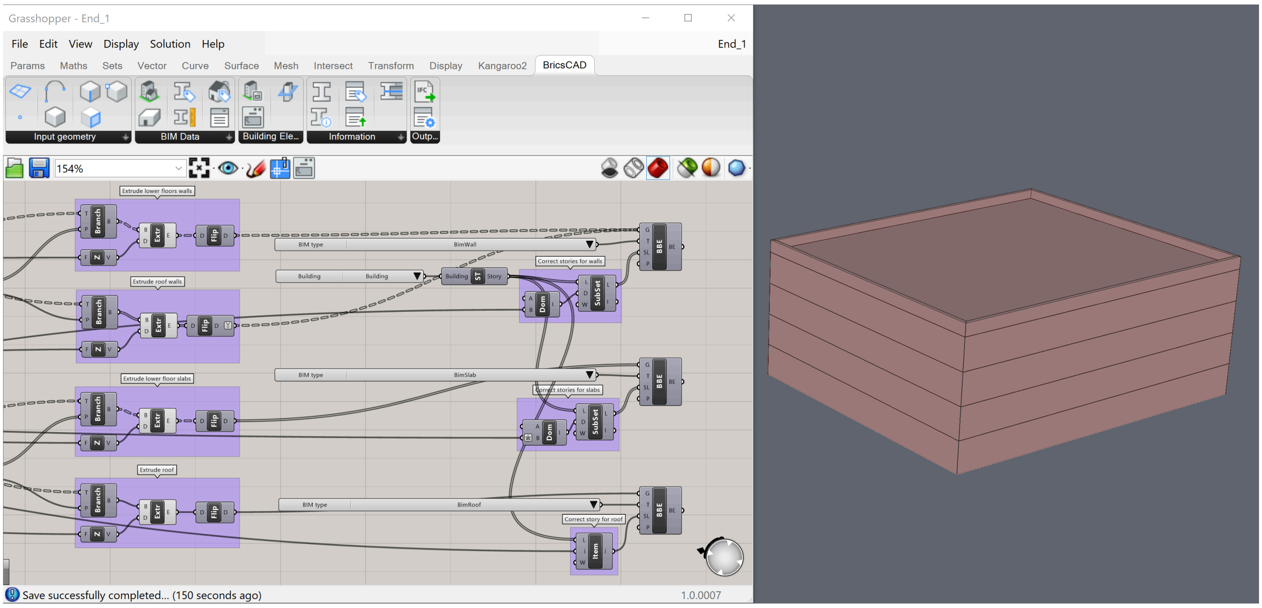


20 The last input geometry will be baked onto all remaining stories as there are now more stories than geometry. To correct this, select a correct subset of the stories to assign. Bake the geometry in BricsCAD and the stories will be correct. No duplicate geometry will be baked.

You can see how it's done by opening the **End_1.dwg** and then opening the Grasshopper file **End_1.ghx** that is linked to the **End_1.dwg**.

21 You have now successfully created a building.

22 If you want to see the result of the baked building, check the file **End_1_Baked.dwg** or bake it yourself.

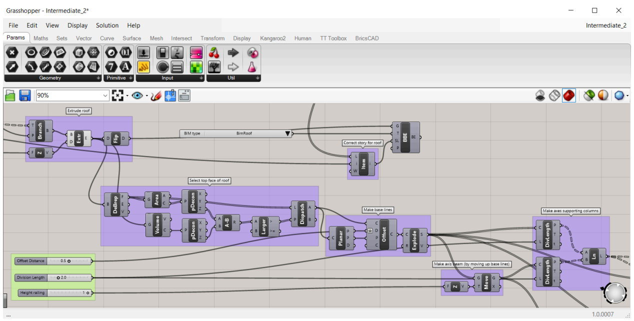


24.10.8 How to add a railing to the top of the building

We will continue with the building made in the previous procedure. So open the **End_1.dwg** and then open the Grasshopper file **End_1.ghx** that is linked to the **End_1.dwg**.

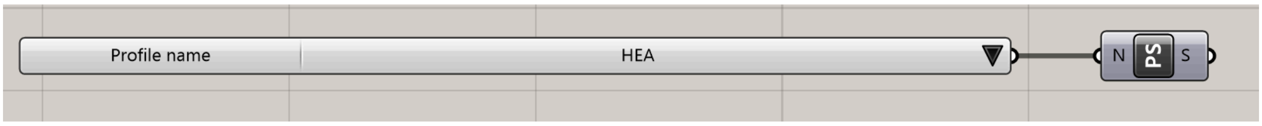
1 Add the **Start_2.ghx**-code to the Grasshopper canvas.
   - You do that by downloading the zip file at the bottom of this page and extracting its content.
   - Then open the **Start_2.ghx** file by going to **File** > **Open Document...** in your **Grasshopper** window.
   - You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.
   - Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current document (Start_2).
   - You then get a drop-down of all the active Grasshopper scripts, so open **End_1.ghx**.
   - Now click on the canvas and hit **Ctrl+V** to paste the **Start_2.ghx**-code.

2 Link the **Extrude Roof** group's end component to the Brep input of the Deconstruct Brep component of the **Select top face of roof** group.



3 The script does the following:
   - The **Select top face of roof** group will select the top face of the roof.
   - The remaining script will take that face and use it to make the baselines for the railing.
   - It will then make the axes of the supporting columns and beams.
   - It will also calculate half of the height of the beam profile.
   - There is still a profile required, however, to complete this step and the following one. We will, therefore, create a profile from step 6 onward.
   - With that profile and the axes, the beams and supporting columns are made as extrusions.
4 The intermediate result is found in the **Intermediate_2.ghx** and **Intermediate_2.dwg** files. (Located in the zip file at the bottom of this page.)
5 **(Optional)** Skip the **Select top face of roof** group and replace it with a **BricsCAD** > **Input Geometry** > **Face component**. Set it to the top face of the roof slab in your BricsCAD baked building. (Do this by right-clicking on the component, click on **Set one BricsCAD face** and select the face in the BricsCAD drawing). Connect this Face component with a **Params** > **Geometry** > **Surface component** and connect this surface to the correct inputs.
6 Go to **BricsCAD** > **BIM Data**.

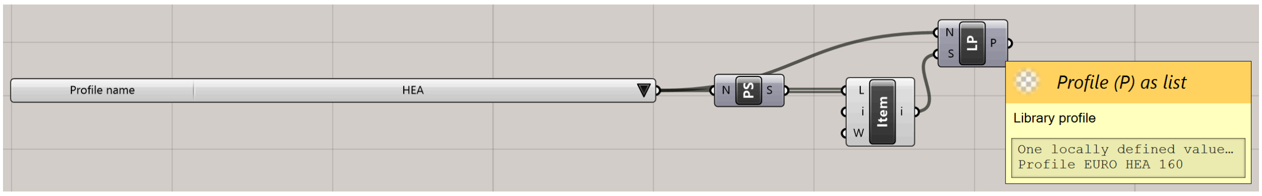
   Drag the **Profile Names** and the **Profile Sizes** components onto the canvas.
7 Select **HEA** from the drop-down list in **Profile Names**. This drop-down list displays all the names of the profiles in the **Profile Library** of BricsCAD.
8 Link the component to the input **ProfileName** of the **Profile Sizes** component.



9 Link the **Profile Sizes** component to the input node **List** in a **Sets** > **List** > **List Item component**, where i is set to **4** (right-click on the **i** input and **Set integer** to **4**). This **List Item** lets you pick one size from the list with all available profile sizes for the **Profile Name** linked to it.

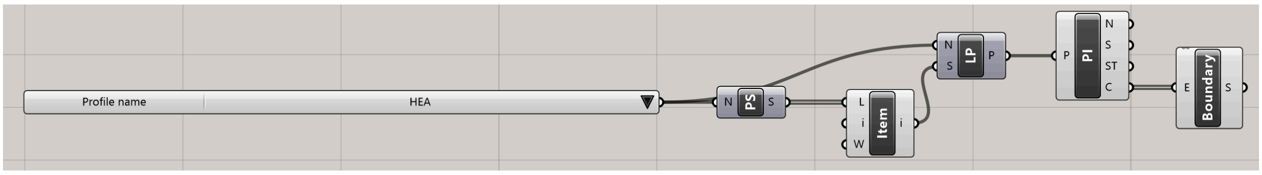
   Note: If you link a **Params** > **Input** > **Panel component** to the Profile Sizes output, you will see that the index 4 stands for size 160.

Link the **Profile Names** and **List Item** components to the **BricsCAD** > **Information** > **Library Profile component**, to get an HEA 160 profile as an output.



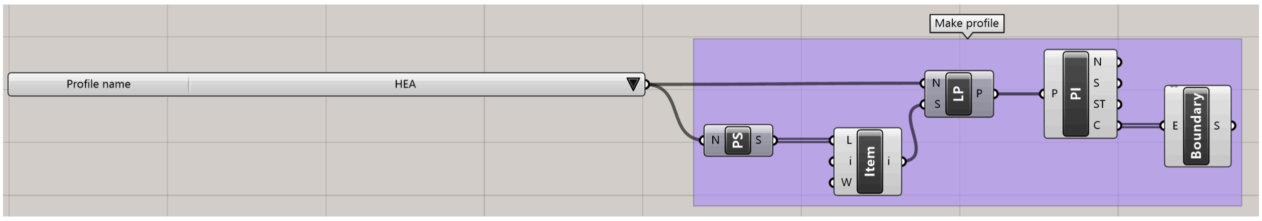
10 To get the curves that make up this profile, attach a **BricsCAD** > **Information** > **Profile Info component** to the **Library Profile**. One of the outputs created is the **Profile Curves** as a tree.

11 Attach a **Surface** > **Freeform** > **Boundary Surfaces component** to the **Profile Curves** output node. This is to later extrude the surface as a solid, instead of the lines as a surface.

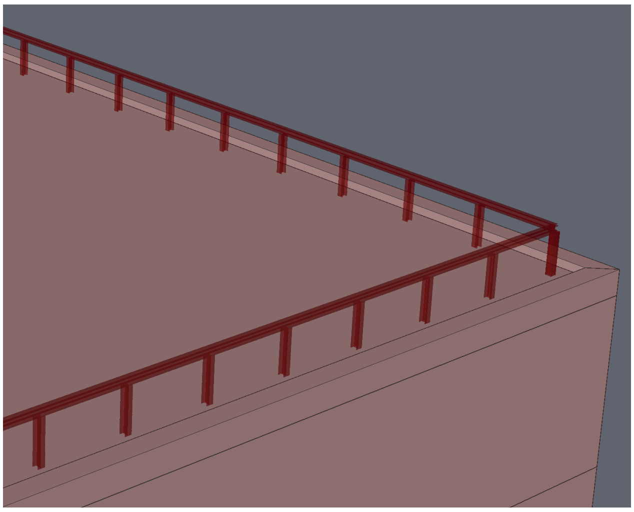


12 To group together the components just created, select them (with a selection box) and right-click on **Group**.

13 **(Optional)** To rename the group, right-click on the purple rectangle. Rename the group in the upper text box, e.g. to **Make profile**.



14 The Boundary Surfaces component should now be linked to the Content input of the Bounding Box component of the **Calculate half of height of beam** group. The Boundary Surfaces component should also be linked to the two Profile inputs of the Extrude components in the groups **Make supporting columns** and **Make beams**. The extrusions should now show you a railing with HEA 160 beams and columns.



15 To bake those beams and columns into BricsCAD attach two **BricsCAD** > **Building Elements** > **Bake Building Element components** to the Grasshopper canvas.

16 Do the following:

- Set the **Geometry** input to the extrusions.
- Set the **Element Type** input to a **BricsCAD** > **BIM Data** > **BIM Types component**, which is set to **BimColumn** for the column extrusions and to **BimBeam** for the beam extrusions.
- Set the **Spatial Location** input to the same as the roof.
- Set the **Profiles** input to the **Library Profiles** component from the **Make profile** group.

17 You now successfully created a railing on top of your building. (The full script to do this is in the **End_2.ghx** file at the bottom of this page, which is linked to the **End_2.dwg**.)

18 Bake into BricsCAD. The profiles are now set to the one in the Grasshopper script. You can still modify the connections of the beams in the corners by using L-Connect in BricsCAD. (To see the finished building with the railing on top of it, check out the **End_2_baked.dwg** file at the bottom of this page.)

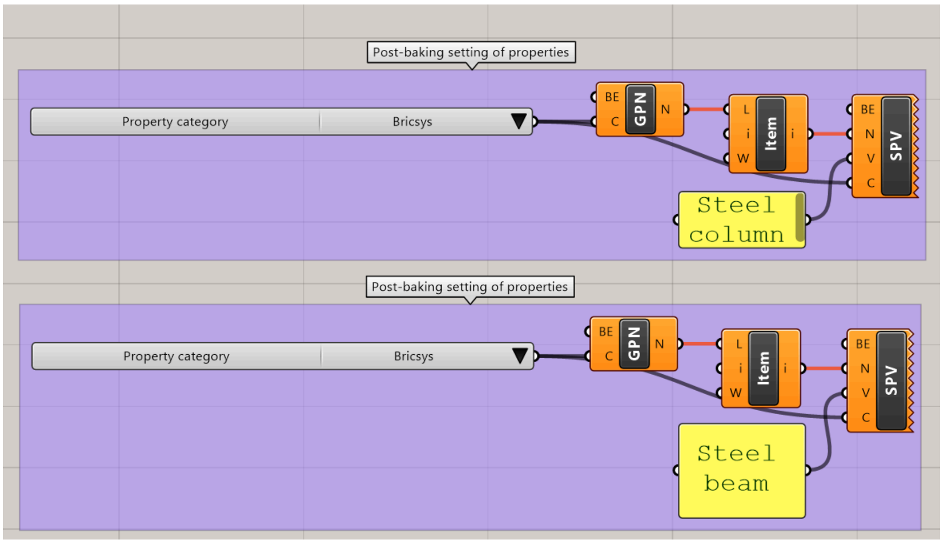
24.10.9 Procedure: add information to a baked model

We will continue with the building made in the previous procedure. So, open the **End_2.dwg** and then open the Grasshopper file **End_2.ghx** that is linked to the **End_2.dwg**.

1 Add the **Start_3.ghx** code to the canvas.

- You do that by downloading the zip file at the bottom of this page and extracting its content.
- Then open the **Start_3.ghx** file by going to **File** > **Open Document...** in your **Grasshopper** window.
- You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.
- Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current document (Start_3).

- You then get a drop-down of all the active Grasshopper scripts, so open **End_2.ghx**.
- Now click on the canvas and hit **Ctrl+V** to paste the **Start_3.ghx**-code.



2 This code has some post-baking settings of classified and baked beams and columns:
   - The script uses the **Get Properties Names component** to display the BIM properties in the Property category **Bricsys**.
   - In those properties, it selects item 1 for the columns and item 0 for the beams, **ColumnType** and **BeamType** respectively.
   - To conclude, it sets the properties to the value specified in the **Panel components**, using the **Set Property Value components**.

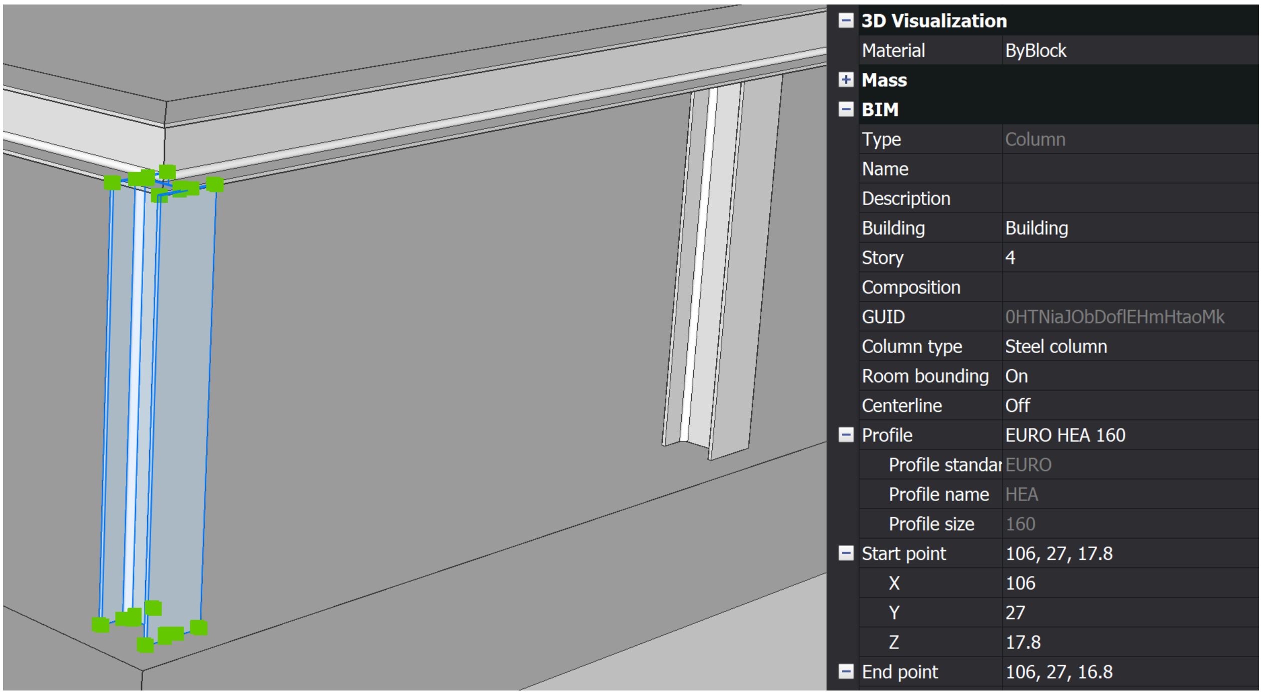
3 To make the script work, bake the two **Bake Building Element components** that make up the railing.

4 Now link the Building Element output of the **Bake Building Element components** to the respective Building Element input of the **Get Properties Names components**.

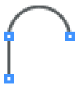
5 Also, link the Building Element output of the **Bake Building Element components** to the respective Building Element input of the **Set Property Value components**.

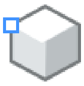
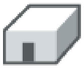
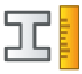
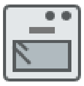
6 If you now select a column in BricsCAD and open its Properties, the **ColumnType** will be set to **Steel column**. When you change the value of the **Panel component**, the Properties in BricsCAD change in real-time.

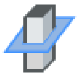
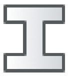
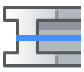
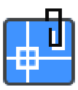
7 The resulting script: 'End_3.ghx' can be found in the zip file, located at the bottom of this page. It works with the 'End_3.dwg' file in which nothing is yet baked. Bake all the Building Elements, to get the post-baking settings.

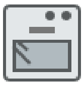


24.10.10 The BricsCAD toolset in Grasshopper

Icon	Name	Description
	Plane	Represents a plane in BricsCAD.
	Point	Represents a point in BricsCAD.
	Curve	Represents a BricsCAD curve.
	Entity	Represents a BricsCAD entity.
	Edge	Represents a BricsCAD edge.
	Face	Represents a BricsCAD face.

Icon	Name	Description
	Vertex	Represents a BricsCAD vertex.
	Buildings	Provides a name picker for all the buildings present in **Spatial Locations** in BricsCAD.
	Stories	Returns all the stories attached to the input building.
	Profile Names	Provides a name picker for all the profiles present in **Profiles** in BricsCAD.
	Profile Sizes	Returns all the sizes attached to the input profile.
	BIM Types	Provides a type picker for all the **BIM Types** available in BricsCAD.
	Property Categories	Provides a category picker for all the property categories available in BricsCAD.
	Bake Geometry	Bake the Grasshopper geometry into the current BricsCAD drawing, while disregarding the BIM data attached to it. The output of Bake Geometry is a reference to the baked building element without BIM data.
	Bake Building Element	Bake the Grasshopper geometry into the current BricsCAD drawing, while adding BIM data to it. The output of Bake Building Element is a reference to the baked building element with BIM data.

Icon	Name	Description
	Elements on Spatial Location	By default, returns all the building elements present in BricsCAD. When using input parameters, returns the building elements filtered by element type and/or spatial location.
	Library Profile	Returns a profile from the library, according to the given name and size.
	Profile Info	Returns the information (name, size, standard and curves) of the specified profile.
	Property Names	Returns the property names, attached to a building element, in the specified property category.
	Property Value	Returns the property value, attached to a building element, for the specified property name and category.
	Linear Solid Info	Returns information (axis, extrusion path and profile curves) about a linear solid present in the BricsCAD drawing.
	Set Property	Sets the property value of the building element according to the specified name, category and value.
	IFC Export	Exports the specified building elements to IFC.
	Link	Links the Grasshopper script to the open BricsCAD .dwg file.

Icon	Name	Description
	Bake	Bakes the geometry from the selected Bake Geometry and Bake Building Element components.

Starter files.zip

24.11 Interference checking

24.11.1 About interference checking

The **Interference Checking** tool is used during the modeling phase to find conflicts in the model by comparing 3D solid models of building elements. Interference occurs when two 3D entities overlap in 3D space. In BricsCAD BIM, the Interfere tool helps you to find intersections between the entities that you have selected. You can also do the interference checking for all elements in your model at a time. Optionally, the tool creates interference solids after running the command. The newly created solids are placed on their own layer, as specified by the INTERFERELAYER system variable. Running an interference check on existing models allows you to prevent the conflicts in your entire model.

For more information about this command, visit the Command Reference article INTERFERE.

24.11.2 Procedure: performing an interference check

1 Select the desired 3D solids to be checked for interference, then launch the INTERFERE command.

1 ***Note:*** Multiple solids can be selected using the selection box or in the **Structure** panel, depending on whether the selected entities appear when selected in the tree.

The first set of entities is defined.

The Command line displays how many entities in total are set for interference checking e.g. Entities in set: 20.

Note that the Nested Selection option allows you to select 3D solid models that are inside blocks and external references (XRef) and once they are selected you can compare them against other entities in the selection set.

2 Select the 3D solids in the model to add them to the second set of entities.

3 The entities in the first set are compared with the entities in the second set.

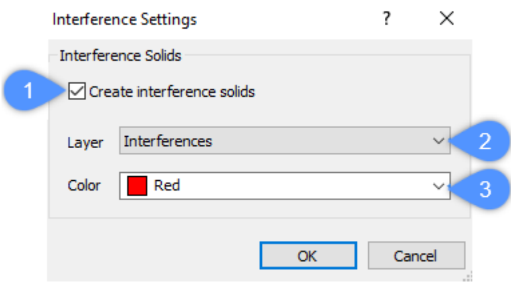
Note: If you press the **Enter** key without selecting any solids for defining the second set of entities, the entities in the first set will be compared to each other.

4 Press **Enter** to run the interference check.

5 The interference volume is created for each collision and it is shown in a solid model.

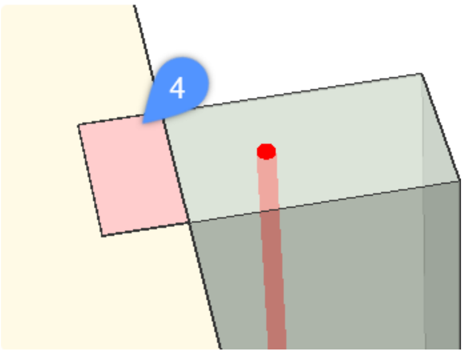
6 ***Note:*** The solids of the interference volume are displayed in red since they are placed on layer **Interferences**. The default settings on the Interference Checking tool makes it easy to find the intersecting solids in the model. This setting can be changed using the prompt menu or the Command line. To open it, click the **Settings** option in the prompt menu or type "**S**" while the interfere command is still active. The **Interference Settings** dialog box displays. The dialog box allows you to configure interference settings, which are stored in variables.

The following illustration shows the **Interference Settings** dialog box. The table below presents each component of the dialog box.



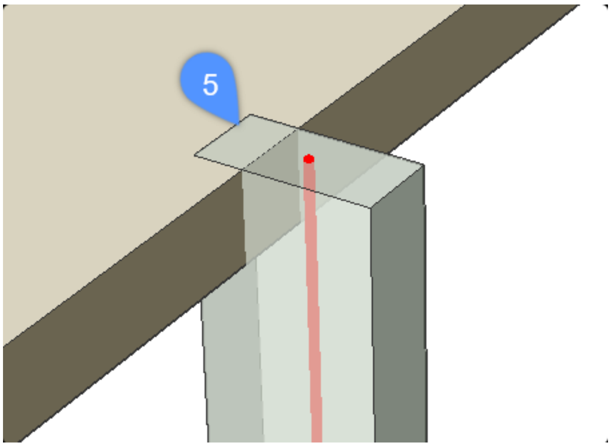
Create interference solids box (1)	Determines whether a solid or region of the interference is created. Check to create a solid of the interference volume or region of the interference area.
Layer (2)	Shows the current layer for interferences.
Color (3)	Shows the current layer color.

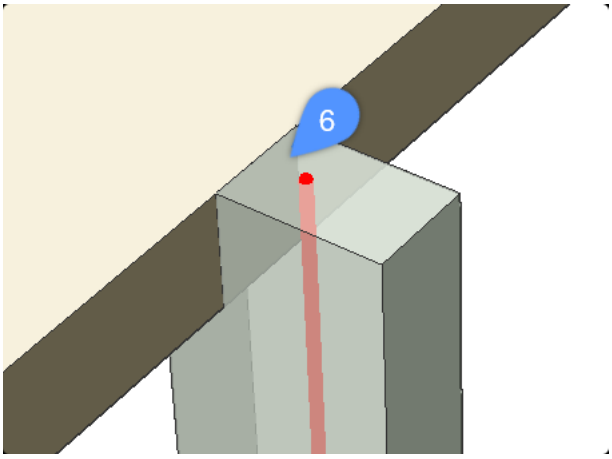
The following illustration shows the detected collision (4) between the floor slab and column.



7 To remove the undesired intersections, turn off the **Interferences** layer, and use one of the 3D solid modifying tools e.g. **BIM Drag** or **Subtract** on the interfering solid. This process will let you make the necessary corrections in your model.

In the following illustration, the created solid of the interference volume is shown (5). In the second image, the intersection is fixed by subtracting the slab geometry from the column.





8 Run the INTERFERE command a second time to check whether all interferences are fixed.

9 If there are no collisions left, the Command line reports that "**Solids do not interfere**".

24.12 The Story Bar

The **Story Bar** displays the current active building and its stories.

24.12.1 About Story Bar

Offers an overview of all stories in the currently active building.

Makes it easy to identify floor levels as you scroll through them.

Shows which story is active, if any.

Allows you to activate or deactivate a story.

You can handle multiple buildings in one project, even if their stories are completely different.

- Click the building icon to open the **Spatial Locations Manager** dialog box to create and edit sites, buildings and stories.
- Click the building label (name) at the top of the drawing area to open a menu that lets you switch between the available buildings.

Right-click a story disk to open the context menu:

- **Activate story**: makes the story active.
- **Exit active story**: deactivates the story.
- **Update story height**: lets you update the height of the story via the **Update Story Height** command context panel that opens with the story selected in the **Spatial location** tab. See the **BIMUPDATESTORYHEIGHT command** article.
- **Select**: selects the entities that belong to the story.
- **Open Story Plan**: opens the drawing file related to the BIM section entity. See also the BIMSECTIONOPEN command.

Note: The **Story Bar** becomes visible when the STORYBAR system variable value is set to 1 or 2.

24.12.2 The 'No Building' mode

When no spatial location is available, the story bar automatically enters the **No Building** mode and shows one story disk named **No story**. Left-click this story disk to enter the **Top View Mode** (TVM) without having to create a spatial location. The story elevation used in this case is UCSZ==0.

Note: If at least one building exists, then one extra **No Building** item is shown in the list of buildings at the top of the drawing area. Select this item to enter the **No Building** mode.

24.12.3 Activate a story

Click a story disk to activate the story.

Note: When activating a story through this option:

- The **Top View Mode** (TVM) is enabled.
- The perspective is turned Off, if it was previously active.
- Two temporary section planes cut the view above and below the elevation. Sliders on the **Story Bar** allow you to change the position of each section plane inside their story slot.

  Note: The bottom active story section can be dragged above the story elevation.

  Note: The active story top and bottom section elevation are remembered between BricsCAD sessions.
- Elements below the elevation are rendered grayed out, not highlighted and not selectable. They can be snapped to and magnetically stuck when editing or creating entities above the elevation.
- The UCS is set to the story elevation.
- The OSNAPZ system variable is switched On.
- You can click another disk to move to that story without changing the zoom level.
- When you select a wall, a window, or a door, the editable dimensions which position the entity are displayed. Also, the reference point of these editable dimensions can be moved to another location.

Activate a story through the story disk context menu.

Note: When activating a story through this option:

- The view is not changed.
- Two temporary section planes cut the view above and below the elevation. Sliders on the **Story Bar** allow you to change the position of each section plane inside their story slot.
- Elements below the elevation are rendered grayed out, not highlighted and not selectable. They can be snapped to and magnetically stuck when editing or creating entities above the elevation.
- The UCS is set to the story elevation.
- The OSNAPZ system variable is switched ON.
- You can click another disk to move to that story.

You can also use the ACTIVATESTORY command to activate a story.

24.12.4 Deactivate a story

Click the **Exit** button at the top of the drawing area to deactivate an active story, without any view modification. The UCS, OSNAPZ and perspective (if from **Top View Mode**) are reset back to original values.

You can also use the DEACTIVATESTORY command or **Exit Active Story** from the story disk context menu.

25. Project collaboration

25.1 BIM collaboration format (BCF) panel

25.1.1 About BCF

The BIM Collaboration Format (BCF) is an open file format used to exchange textual comments, images, and more on top of an IFC model. This allows better communication between different parties in the BIM process.

BCF makes it possible to link information such as a comment, a screenshot, a list of involved objects, and a camera viewpoint to an issue. An issue can be anything ranging from a small change in the design to clashes between different disciplines in the model. These issues can then either be saved as BCF files or they can be managed on the cloud, using any sort of BCF cloud management system.

The advantage of BCF is that it allows communication over different software packages (for example, a BCF issue created in Solibri can be opened in BricsCAD BIM), and that it enables easy navigation through a model based on issues.

25.1.2 BCF files

Issues can be saved in standalone files. This is generally a more cumbersome workflow than working with a cloud system but it can be useful in case you are working offline. A BCF file can then be imported in any BCF manager (for example, BricsCAD BIM) to read the issues and see where in the model it applies.

Note: A BCF file cannot be created in BricsCAD BIM. For this, you will need access to another service (for example, Solibri Model Checker or BIMcollab ZOOM). This software allows you to create a BCF file based on one or more IFC files. You can then create issues containing:

- Comments
- Screenshots or other images
- A list of involved objects, based on GUID (Global Unique IDs) to distinguish them unambiguously
- A camera position

Note: It is not possible to create or modify issues within BricsCAD BIM if a BCF file was imported. Issues can only be created when using cloud services.

25.1.3 Using the cloud

It is generally easier to use cloud services to manage BCF issues. Currently, BricsCAD BIM supports automatic connection to several of such services: **Bricsys 247**, **Catenda**, **BIMcollab**, and **BIM Track**. However, other services can be manually entered. If you wish to connect to these services from within BricsCAD BIM you will need an account of these services first. These allow you to create issues and manage them in real-time from anywhere.

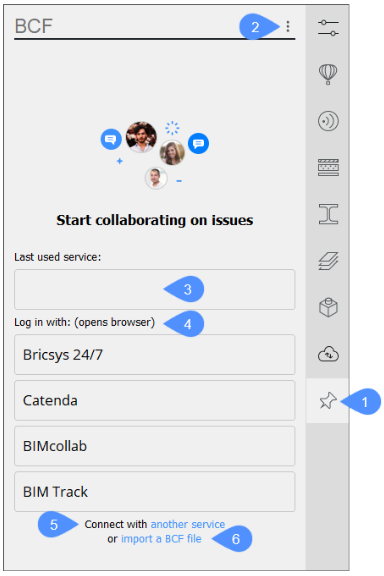
When using cloud services, it is possible to change statuses of existing issues within BricsCAD BIM, as well as creating new issues. The **BIM BCF** panel allows you to connect to these services from within

BricsCAD BIM, log into your account and add issues, comments, screenshots and viewpoints, which will guide your team members to the exact location of the issues.

25.1.4 The BCF panel

The **BCF** panel can be opened by clicking the board pin icon on the left of your screen (1). If the icon is not there, right-click an empty space in the ribbon and under **Panels**, enable the **BIM BCF** panel.

The panel displays the three most popular collaboration cloud services (4) as well as the last one used (3). You can directly connect to the desired one by just clicking on the corresponding button. The options to connect with **another service** (5) or to **import a BCF file** (6) are also available.



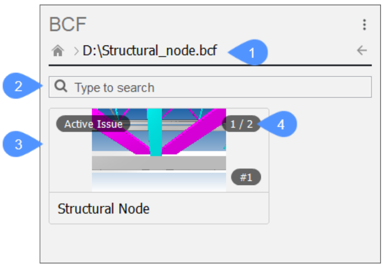
The menu at the top (2) gives you the same options, as well as refreshing the panel and clearing the sources. Refreshing the panel makes sure you see the latest updates in case you are working on the cloud. Clearing the sources disconnects you with the cloud service in case you are online, or closes the BCF file in case you are working on an imported BCF file.

25.1.5 Procedure: working with the BCF panel

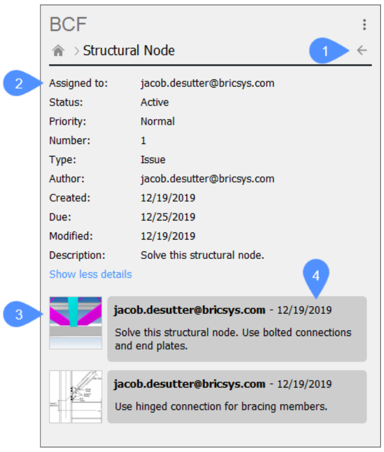
The following example will be shown with the files attached at the end of this article. If you want to follow along, open **West_str.dwg** and import **Structural_node.bcf**.

When importing a BCF file:

You will see the path of the currently opened file (1), a search bar that can be used to filter on issues (2) and a list of issues (3). In the case of this imported BCF file, there is only one issue with two associated views (4).



Click on the issue to open it. When opening an issue, you see a button to return to the issue overview (1), some more detailed information about the issue such as status, creation date etc. (2) and different comments on this issue (4). If this comment has an associated camera position, clicking the thumbnail image (3) brings the camera in the current drawing to this position. If there are entities attached to the viewpoint, they are highlighted or isolated. This is useful if you want to know where exactly the issue is and you don't want to manually search through the model.



When using a cloud service:

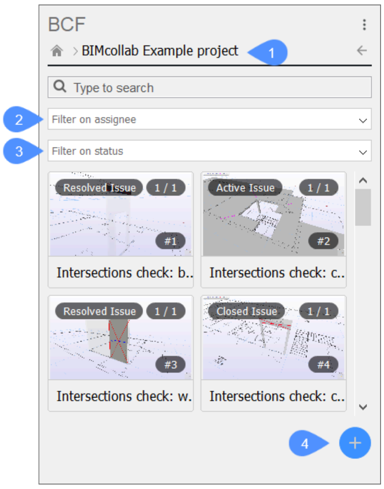
You are presented with some more options, such as setting statuses and adding comments and screenshots. You can also add new issues.

To connect to a cloud service, you can click on one of the collaboration services buttons listed in the first panel tab or connect with another service via the **Custom log in** dialog box. To connect to a **BIMcollab** service, you are prompted to enter a corresponding name space. To connect to a **Bricsys 247**, **Catenda** or a **BIM Track** service, you are automatically redirected to their login page in your browser. To connect to another service, you have to manually enter the service URL.

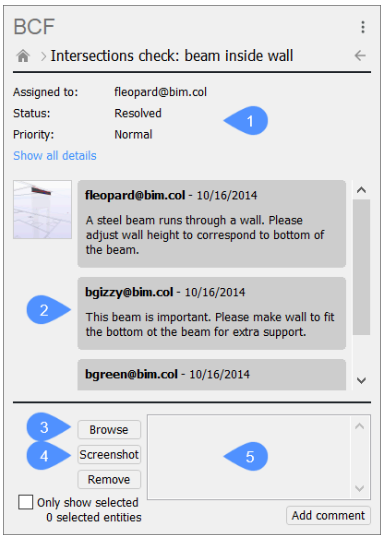
The following example is shown on an internal cloud space so it is not possible to follow these steps exactly. It is however possible to create a free account on for example **BIMcollab**. Clicking **Log In** takes you to a login page of the chosen service.

After logging in, the source URL should be shown (1), together with a list of projects that you are involved in (2). In this example, only one project is shown with 22 issues.

Clicking the project opens it and shows a list of the issues, similar to what happened when importing the BCF file. At the top of the panel, the current project is displayed (1). It is possible to filter the issues on assignee (2) and status (3). This facilitates finding relevant issues. It is also possible to add a new issue (4).



Clicking an issue opens it and you see similar options as before: detailed information about the issue (1) and comments on this issue (2). However, note that at the bottom of the panel it is also possible to add bitmap images (3) or a screenshot from model space (4). You can add some textual comments as well (5).



Structural_node.bcf

West_Str.dwg

25.2 Using the Bricsys 24/7 panel

25.2.1 About Bricsys 24/7

The **Bricsys 24/7** panel helps team members to find and share their drawings.

Bricsys 24/7 enables project management, a collaboration between project teams, cloud computing security and task automation. The big advantage of this cloud-based service is that BricsCAD is directly linked to it by the **Bricsys 24/7** panel.

25.2.2 Use the Bricsys 24/7 panel in BricsCAD BIM

1 Open the **Bricsys 24/7** panel by clicking on the Bricsys 24/7 icon ( ) in the **Tool** panel on the right side of the workspace.

Note: If you do not find this icon in the **Tool** panel, right-click the **Tool** panel and check **Bricsys 24/7** in the drop-down menu, as is described in the **Panels** article.

2 Log in to your 24/7 account if necessary (see **Bricsys 24/7 panel** help article).

25.2.3 Add a file to the project

1 Open a folder where you want to add the file.

2 Press the plus icon that opens the **Add new document** and choose a file to add.

3 Choose a status from the drop down list, for example Draft.

4 Enter a description.

5 Press the **Upload** button.

The file is displayed in the folder list files and it is **Available for download**. The folder path for files downloaded through the **Bricsys 24/7** panel is defined with the CLOUDDOWNLOADPATH system variable.

The file is opened by double-clicking it.

After editing and saving it, the status of the file is changed to **Locally modified**.

By right-clicking the file, you can choose to lock it for editing by another person.

You can upload the modified file with a new status and description. The file will be marked as **Up-to-date**.

25.3 IFC import and export in BricsCAD

25.3.1 About IFC

The Industry Foundation Classes (IFC) data model is designed to describe building and construction industry information. IFC is a platform-neutral, open-format file type. IFC provides interoperability between different BIM software platforms.

BricsCAD BIM supports IFC2x3 and IFC4 for import and export. The IFC2x3 and IFC4 property namespaces are merged into IFC namespace. Differences between the IFC schemas will be dealt with in the background during IFC Import or Export. The IFC standard includes both parametric and non-parametric shapes, material definitions, metadata in the BIM file, and property sets (PSets).

When you export an IFC file from BricsCAD, the properties will be exported as IfcPropertySet. These property sets contain common IFC properties. User-defined property sets can also be exported.

Note:

- BricsCAD BIM created IFC files can be used in different coordination workflows with many tools (for example, Solibri).
- Some software allow for editing only IFC files created with specific Authoring Tools, while actively blocking files created with other tools. Other uses of BricsCAD BIM created IFC files on these software are possible (for example, viewing).
- When IFCEXPORTIDSPROPERTIESONLY system variable is ON, only the IFC and custom properties of the active namespace (selected in the drop-down menu) specified in the imported ISD file are exported.
- BricsCAD BIM also supports exporting files in RVT format (only for viewing).

Mechanical system layouts, structural models, and architectural models that are exported from different IFC compliant software applications can be imported into BricsCAD BIM.

IFC2x3 vs IFC4

IFC4 is an advanced schema of IFC2x3. It is extended and overcomes certain limitations of IFC2x3. There are geometric and data improvements as well as new and changed Model View Definitions. Geometric improvements include more parametric support, more efficient handling of meshes and an improved way to add textures, lighting settings, etc. An increased number of property sets make the IFC data model more complete and more granular. For IFC4, the coordination view has been replaced by two official Model View Definitions: the **IFC4 Reference View** and the **IFC4 Design Transfer View**. Differences between the IFC schemas will be dealt with in the background during IFC Import or Export and the IFC2x3 and IFC4 property namespaces are merged into IFC namespace.

25.3.2 IFC import and export settings

The IFC import and export settings are set in the **Settings** dialog box:

Variable	Description
IfcImportSettingsConfig	Specifies the name of the IFC import settings configuration file.

Variable	Description
IfcExportBaseQuantities	Enables/disables exports derived from base quantities of BIM elements. If this setting is enabled, calculated quantities (e.g. length, thickness, height) for each BIM element will be exported to IFC. ***Note:*** Ply quantities are exported when both IFCEXPORTMULTI-PLYELEMENTSASAGGREGATED and IFCEXPORTBASEQUANTITIES system variables are set to ON. For **IFC4 Reference View file**, next to the quantities for the wall as a whole, the ply quantities are exported as a sub set for each ply (as IfcPhysicalComplexQuantity).
IfcExportElementsOnOffAndFrozenLayers	Enables/disables the export of BIM elements that are in Off and Frozen layers. By default, this setting is enabled.
IfcExplodeExternalReferences	Enables/disables exploded external references in IFC spatial structures. If this setting is enabled, each external reference will be divided into the elements (e.g. IfcSite or IfcBuilding) based on the project spatial structure. ***Important:*** Use the BIMCLASSIFY command to Classify Xrefs as BIM XReferences prior to doing an export with this setting enabled. Otherwise, the result will be a block per Xref.
IfcTesselateBSplineCurvesAndSurfaces	Controls whether BSpline curves and surfaces are tessellated.
IfcExportSweptSolidsAsBRep	Allows to export all extrusions, revolutions, swept 3D solids with clippings and subtractions using a boundary representation (BRep).
IfcExportProfileCenterOfGravity	Only for IFC2x3. Exporting the center of gravity may cause a wrong positioning of linear solids in certain IFC viewers. The center of gravity is never exported to IFC4 or higher.

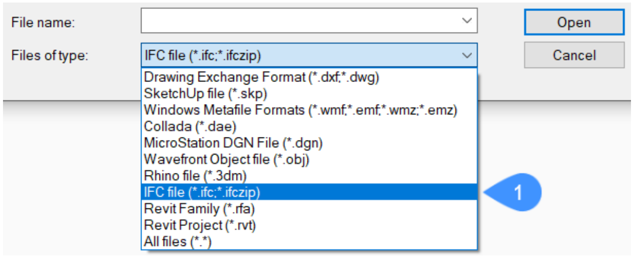
Variable	Description
IfcExportValidateModel	Checks that an IFC model complies with schema rules during IFC export. Problems are reported in an export log next to IFC file. ***Warning:*** Validation takes extra time and can slow down the export of big IFC files. Off (0): Does not apply IFC model validation On (1): Applies IFC model validation
IfcExportMappingPath	Allows to specify an export mapping file path. See **Import/Export mapping of IFC files**.
IfcExportSubtractOpenings	Subtracts the openings from the host geometry before export. Use this to increase the reliability of the geometry when opening in another software (it avoids relying on the boolean operations of the target software). It will make editing the model in the target software more difficult. ***Note:*** This behavior is the default for **IFC4 Reference View** export.
IfcExportMultiPlyElementsAsAggregated	Enables/disables the export of building elements with multi-ply compositions as aggregated entities. If this setting is enabled, each ply material belonging to a building element (e.g. wall or column) is calculated and exported individually. This setting can be used to export quantities and value definitions for Ply-elements. ***Note:*** Ply quantities are exported when both IFCEXPORTMULTI-PLYELEMENTSASAGGREGATED and IFCEXPORTBASEQUAN-TITIES system variables are set to ON. For **IFC4 Reference View file**, next to the quantities for the wall as a whole, the ply quanti-ties are exported as a sub set for each ply (as IfcPhysicalComplex-Quantity).
IfcExportTessalation	Controls the level of tessellation of the exported geometry. If the '**Current**' faceting option is chosen, the faceting as set by FacetRes or Modeler properties is used. Choosing the **Low**, **Medium** or **High** options of regeneration of the facets. This makes the process longer.

Variable	Description
IfcCreateUniqueGuid	Defines whether a unique GUID for nested elements should be generated upon export. 1: unique guids for inside classified blocks 2: inside classified Xrefs 3: to enable both inside classified blocks and Xrefs
IfcExportAuthor	Enables/Disables the export of the author name in the IFC file header.
IfcExportOrganization	Enables/Disables the export of the organization name in the IFC file header.
IfcExportAuthorization	Enables/Disables the export of the authorization information in the IFC file header.
IfcExportIDSPropertiesOnly	When an IDS-XML file has been imported, this setting controls whether only the properties required by the IDS should be exported in the IFC file or all properties should be exported. Off (0): Export all properties On (1): Export only the properties required by the IDS

25.3.3 Import an IFC file in BricsCAD

1 To import an IFC file into your project, type **Import** in the Command line. The import file dialog box displays.

2 Select the IFC file format (1) from the **Files Of Type** drop-down list and choose the IFC model you want to import.



3 Click **Open** to import the IFC model.

Note:

- All IFC2x3, IFC4, IFC4x1 and, IFC4x3 files can be imported, but all are merged into a single IFC namespace. Differences between these schemas will be dealt with in the background during IFC Import. When the IFC file is imported the data from the imported file integrates with the BricsCAD data structure which means you can work with them right away. If the imported data contains custom properties, these property sets will be made available.

- To import **IFC4x3 file** (IFC 4.3.2.0) format, the **Experimental mode** needs to be enabled.

You can create custom property sets and properties directly in BricsCAD BIM and associate them with imported entities.

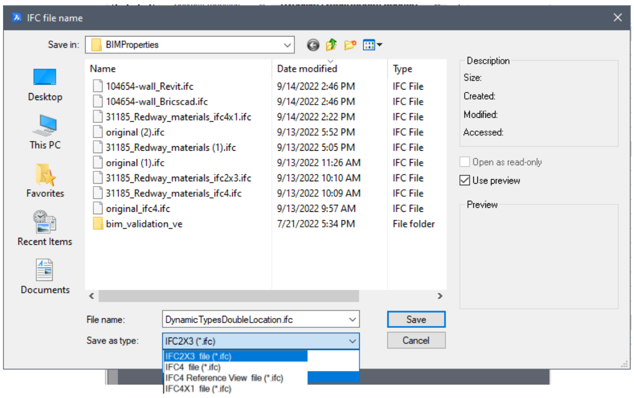
4 **(Optional)** Change the IFC import settings using the **BIM** section in the **Settings** dialog box. See **Change IFC import and export settings**.

25.3.4 Export an IFC file in BricsCAD

1 To export an IFC file in your project, type IFCEXPORT in the Command line. Select the **Entire model** option. The **IFC file name** dialog box displays.

2 Select the desired IFC file format in the **Save as type** list and save the IFC model. The options are: IFC2x3, IFC4, IFC4x1, and IFC4x3.

Note: The **IFC4x3 file** export format is available only when the **Experimental mode** is enabled.



3 Click **Save** to export the IFC model.

The exported BIM model is saved as an IFC2X3 file, IFC4 Design Transfer View file, IFC4 Reference View file, IFC4x1, or IFC4x3 file and is now ready to use in any IFC certified application.

4 **(Optional)** Change the IFC export settings using the BIM section in the **Settings** dialog box.

25.3.5 Export an IFC file only for selected entities

1 To export IFC model partially in your project, type IFCEXPORT in the command line or press the **Export to IFC** icon in the **Home Toolbar**.

You are prompted: Select entities to export [Entire model/selection options (?)] <Entire model>:

2 Select the entities in your model space to export as IFC and press Enter.

3 The **IFC file name** dialog box displays.

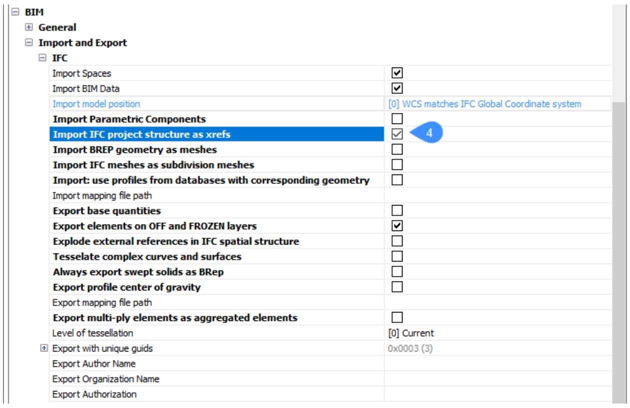
4 Enter a name for the IFC file, choose the IFC format for export (**IFC2X3 file**, **IFC4 Design Transfer View file**, **IFC4 Reference View file**, **IFC4x1 file (deprecated)**, or **IFC4x3 file**) and click the **Save** button. The selected entities are exported to IFC.

25.3.6 Import an IFC Project as Xrefs

1 Go to **Settings** (type SETTINGS in the Command line or go to Settings in the ribbon > **Settings**).

2 Navigate to **BIM** > **Import and Export** > **IFC**.

3 Check **Import IFC project structure as xrefs** box (4).



4 Type IMPORT in the Command line. Specify which IFC file you want to open and click **Open**.

The drawing will now open with Xrefs made for each floor in each building. This allows you to easily split large files into smaller BricsCAD files.

25.3.7 Export a BricsCAD BIM Project with Xrefs exploded

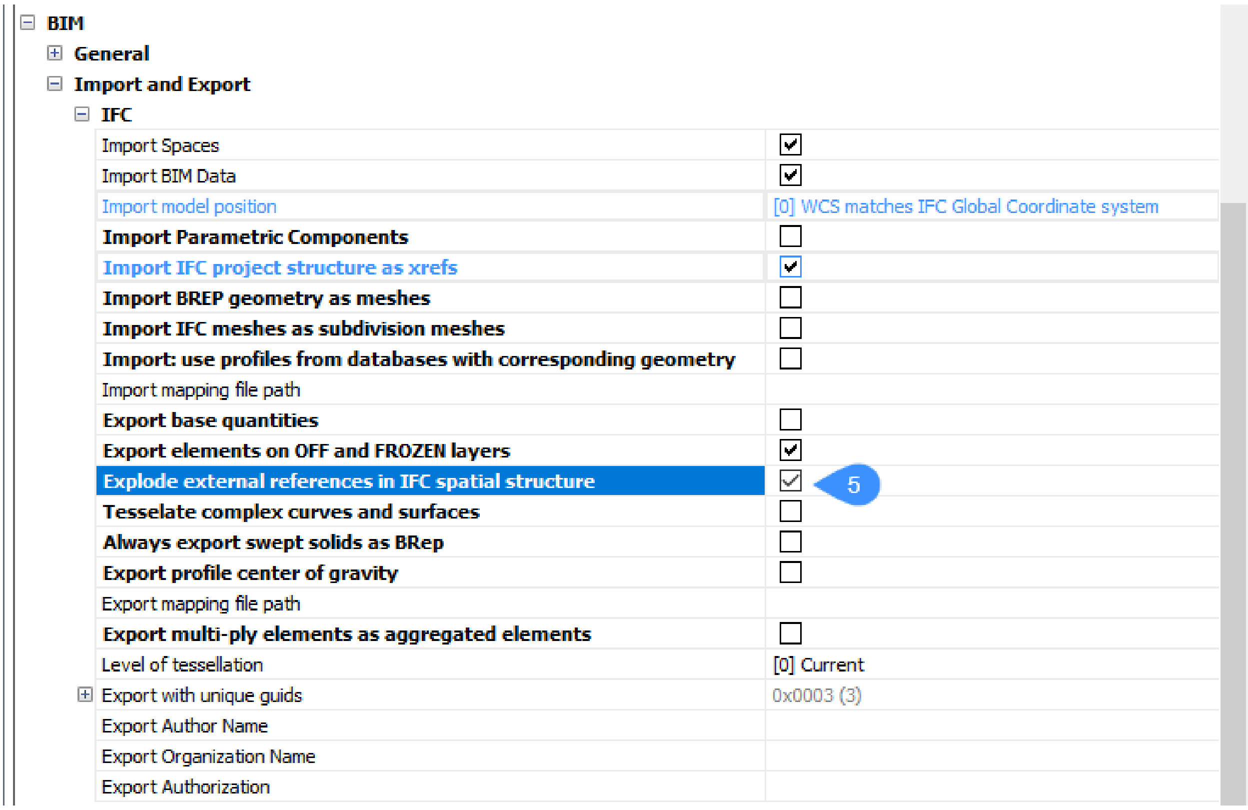
1 Use the BIMCLASSIFY command to Classify Xrefs as BIM XReferences.

1 ***Important:*** If you don't go through this step, your Xrefs will be exported as blocks, instead of separate entities that are part of the Xref.

2 Launch the SETTINGS command to open the **Settings** dialog box.

3 Go to **BIM** > **Import and Export** > **IFC**.

4 Check **Explode external references in IFC spatial structure** (5).

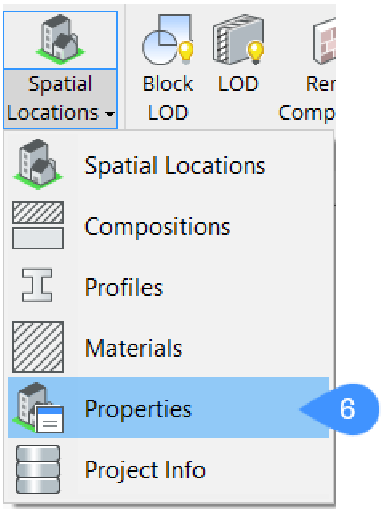


5 Click the BricsCAD icon and then select **Export**. Choose IFC2x3 or IFC4, specify the name of the exported drawing and click **Save**. The exported drawing will now have all Xref elements merged onto one **Site** and one **Building**.

25.3.8 Work with IFC properties

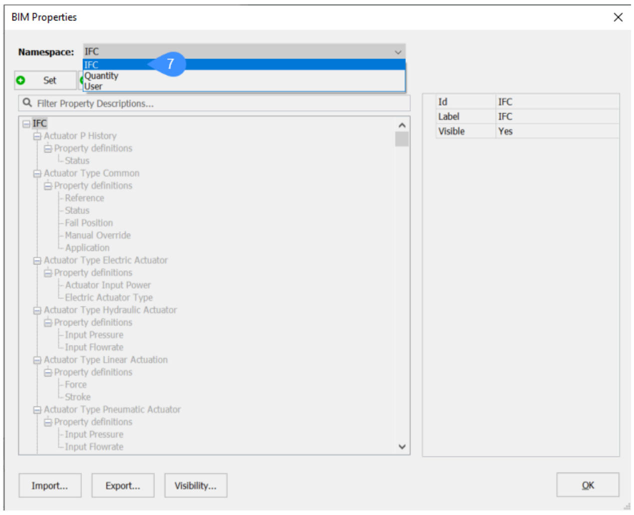
1 Open the **BIM Properties** dialog box using one of these methods:

- In the ribbon go to **Home** > **Classify** > **Spatial Locations** > **Properties** (6). The **BIM Properties** displays.



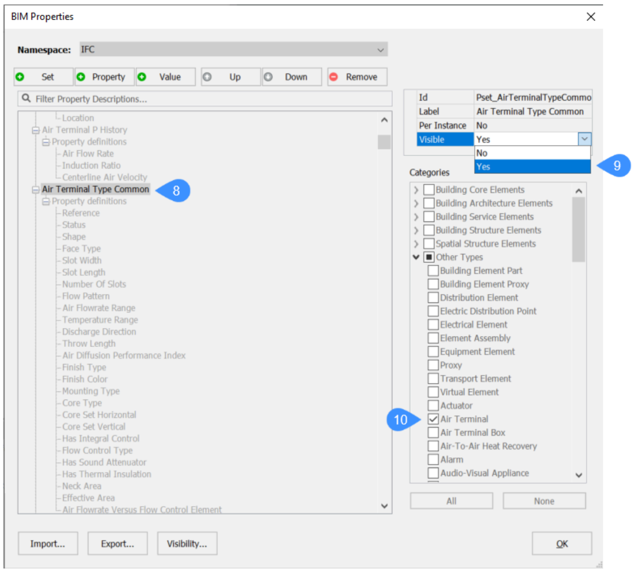
- In the Command line, type BIMPROPERTIES.

2 Select IFC in the **Namespace** drop-down menu (7).



3 Search for the desired property in the properties list (8).

4



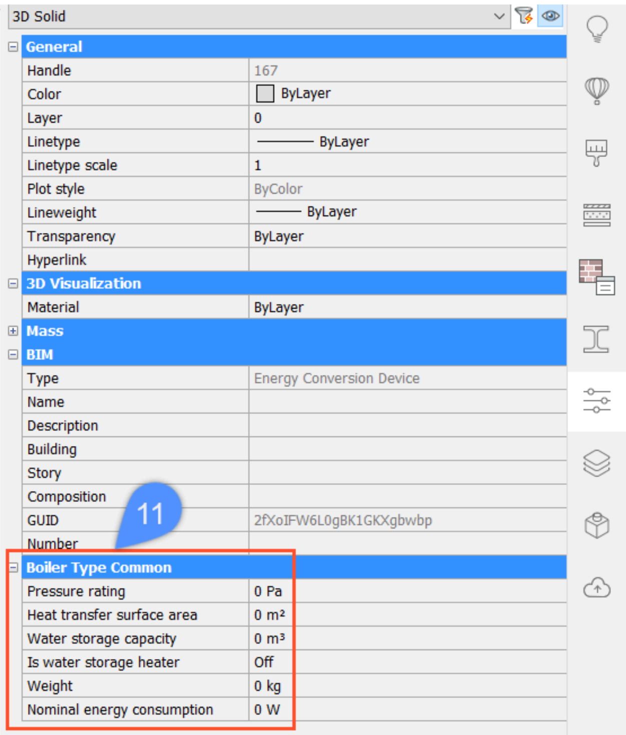
5 Set **Visible** (9) to **Yes** for the section to be visible in the **Properties** panel. Set to **No** to hide the section.

6 In **Categories** select the IFC Classes you wish to display (10).

7 Once all properties have been set, click OK.

8 Select an object of the type/category that you just enabled the Property Set for (in this example an Energy Conversion Device). Go to the **Properties** panel. Scroll down to the section of properties you just enabled.

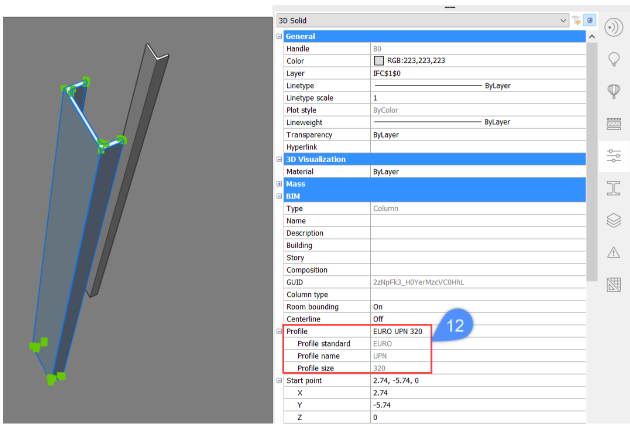
9 Fill in the data as required (11).

10 

Note: IFC properties for spatial elements can also be added through the **Properties** panel.

25.3.9 Profiles in IFC import

When importing profiles from an IFC file to BricsCAD, you can either import the profile properties in the IFC file, or let BricsCAD look for corresponding profiles in the Project and Central Database, based on the geometry. The profiles will then be assigned with proper **Profile Standard**, **Profile Name** and **Profile Size** in the **Properties** panel (12).



25.3.10 Supported IFC Classes

Following IFC Classes are supported by BricsCAD BIM. These are the same categories as the classify options in the BIMCLASSIFY command.

IFC Classes that are backward compatible with older BricsCAD versions:

IfcWall

IfcBeam

IfcColumn

IfcSlab

IfcStair

IfcStairFlight

IfcRamp

IfcRampFlight

IfcRailing

IfcCurtainWall

IfcRoof

IfcBuildingElement

IfcSite

IfcBIMXreference

IfcGridAxis

IfcGrid

IfcAnnotation

IfcWindow

IfcDoor

IfcCovering

IfcFurnishingElement

IfcOpening

IfcDistributionControlElement

IfcDistributionPort

IfcDistributionFlowElement

IfcDistributionChamberElement

IfcEnergyConversionDevice

IfcFlowFitting

IfcFlowController

IfcFlowMovingDevice

IfcFlowSegment

IfcFlowStorageDevice

IfcFlowTerminal

IfcFlowTreatmentDevice

IfcFlowConnectionPoint

IfcMember

IfcFooting

IfcPile

IfcReinforcingBar

IfcReinforcingMesh

IfcTendon

IfcTendonAnchor

IfcPlate

IfcDiscreteAccessory

IfcFastener

IfcMechanicalFastener

IfcSpace

IfcBuilding

IfcBuildingStorey

IfcDate

IFC Classes that are not backward compatible with older BricsCAD versions:

IfcBuildingElementPart

IfcBuildingElementProxy

IfcDistributionElement

IfcElectricDistributionPoint

IfcElectricalElement

IfcElementAssembly

IfcEquipmentElement

IfcProxy

IfcTransportElement

IfcVirtualElement

IfcActuator

IfcAirTerminal

IfcAir TerminalBox

IfcAir-To-AirHeatRecovery

IfcAlarm

IfcAudio-VisualAppliance

IfcBoiler

IfcBurner

IfcCableCarrierFitting

IfcCableCarrierSegment

IfcCableFitting

IfcCableSegment

IfcChiller

IfcChimney

IfcCivil Element

IfcCoil

IfcCommunicationsAppliance

IfcCompressor

IfcCondenser

IfcController

IfcCooledBeam

IfcCoolingTower

IfcDamper

IfcDuctFitting

IfcDuctSegment

IfcDuctSilencer

IfcElectricAppliance

IfcElectricDistributionBoard

IfcElectricFlowStorageDevice

IfcElectricGenerator

IfcElectricMotor

IfcElectricTimeControl

IfcEngine

IfcEvaporativeCooler

IfcEvaporator

IfcFan

IfcFilter

IfcFireSuppressionTerminal

IfcFlowInstrument

IfcFlowMeter

IfcFurniture

IfcGeographicElement

IfcHeatExchanger

IfcHumidifier

IfcInterceptor

IfcJunctionBox

IfcLamp

IfcLightFixture

IfcMedicalDevice

IfcMotorConnection

IfcOutlet

IfcPipeFitting

IfcPipeSegment

IfcProtectiveDevice

IfcProtectiveDeviceTripping Unit

IfcPump

IfcSanitaryerminal

IfcSensor

IfcShadingDevice

IfcSlabElemented Case

IfcSolarDevice

IfcSpaceHeater

IfcStackTerminal

IfcSwitchingDevice

IfcSystemFurnitureElement

IfcTank

IfcTransformer

IfcTubeBundle

IfcUnitaryControlElement

IfcUnitaryEquipment

IfcValve

IfcVibrationIsolator

IfcWallElementedCase

IfcWasteTerminal

IfcAlignment

IfcReferent

25.3.11 Export a structural analysis model

It is possible to export a line model of a structure using the BIMANALYTICALMODEL command. It can either be exported to CIS/2 or to IFC. Choose the **Export and exit** Command line option to open the **Choose a filename and format for export** dialog box, which allows you to choose the file type to export.

For more information and examples, visit the procedure article **Analytical Model**.

25.3.12 Import/Export Mapping of IFC files

When you Import/Export an IFC file in BricsCAD, all entities are imported/exported according to the default mapping. It is possible to import/export elements by special mapping and to exclude entities from import/export.

There are different txt configurations possible.

- **No text mapping**

  All entities will be imported/exported according to the default mapping.

- **Import/Export an entity by special mapping**

  In case of import, you first write the IFC class, followed by the BricsCAD class in which the entities should be imported. For example:
  IfcWall BimDbColumn

  In case of Export, you first write the BricsCAD class, followed by the IFC class in which the entities should be exported. For example:
  BimDbWall IfcColumn

  Note: When an IFC or BimDB entity isn't listed in the Import or Export mapping file, the default mapping will be used.

- **Exclude an entity from the Import/Export**

  Write the IFC or BimDb entity followed by 'skip'.

  For example for Import:
  IfcWall skip

  For example for Export:
  BimDbWall skip

- **Import/Export only the listed entities**

  List the entities that should be imported/exported followed by '* skip' on the next line. The asterisk refers to all not listed entities.
  * skip

- **Import/Export all not listed entities by the same special mapping**

  Write the asterisk followed by the name of the class in which the entities should be imported.

  For example for Import:
  * BimDbWall

  For example for Export:
  * IfcWall

- **Import/Export entities by default mapping when using the asterisk**

  If you used one of the two previous options to skip all entities except the listed ones or to Import/Export by special mapping, you can still Import/Export entities by default mapping. Use the same text as you use to Import/Export an entity by special mapping. Instead of writing a special mapping next to the entity, write the default class or write 'default'.

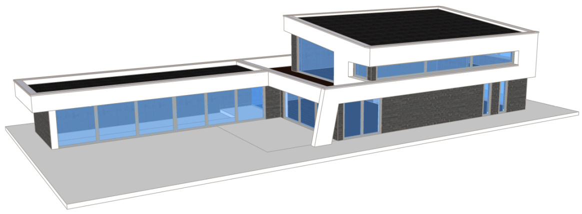
  For example for Import:
  IfcWall BimDbWall or IfcWall default

For example for Export:
BimDbBeam IfcBeam or BimDbBeam default

Note: Use the rules explained in **Export Mapping of IFC Files** to create an Import/Export Mapping file.

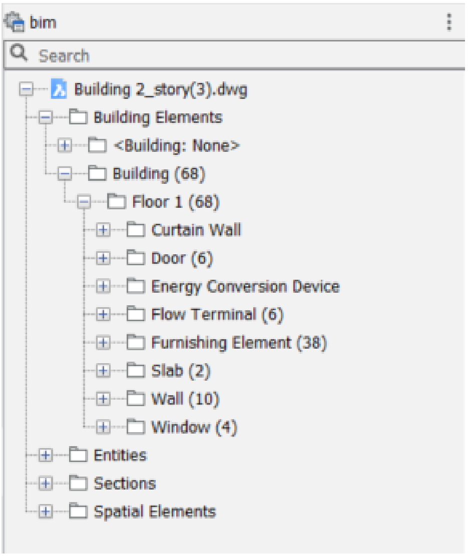
25.3.13 Export Mapping of IFC Files

1 Start from a similar model in BricsCAD with all the building elements classified.



Note: If the elements are not classified, run the BIMIFY command to classify the model.

2 The building consists of several building elements: walls, slabs, doors, windows etc.

3 

A custom Export Mapping file will be created to only export the geometry of the building without windows, doors or spaces.

Note: If you export the file without a mapping file, all the elements will be exported in the default IFC class.

4 Open a new TXT file to create the export mapping file. Use an ASCII text editor, e.g. Notepad.

5 Follow these rules to add the text in the TXT file according to **Import/Export Mapping of IFC files**:

- For export mapping, write the BimDb entity followed by the IFC entity. Separate the two words by a single Tab. Do not use a space or multiple Tabs to align the IFC classes. The lines with a space or multiple Tabs will not be used for Export Mapping.
- Only write one command per line.
- If a composition is attached to a wall in a BricsCAD file, the wall will be exported as

IfcWallStandardCase instead of IfcWall.

Note: When a building entity is imported/exported by special mapping, the properties of the element will change according to the default properties of the new category.

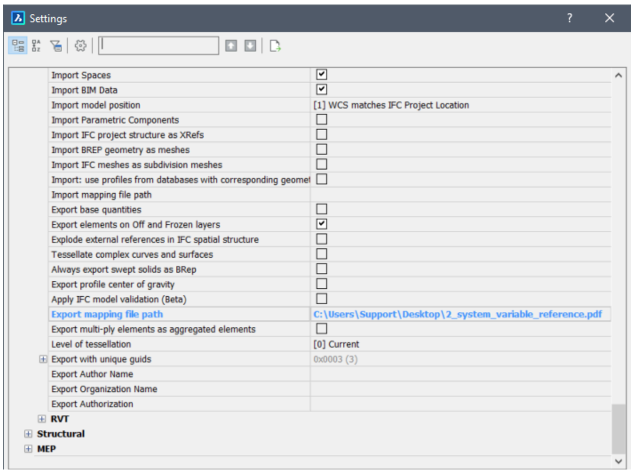
6 Two categories need to be added in the TXT file to create the Export Mapping File for this model.
  - Exclude windows, doors and spaces.
  - The BricsCAD file contains normal windows and curtain walls, exclude both.
    BimDbWindow skip BimDbCurtainWall skip BimDbDoor skip BimDbSpace skip
  - Export beams by special mapping.
  - Some walls are classified as beams in the current BricsCAD file. These have to be exported as walls.
    BimDbBeam IfcWall
  - The resulting file looks like this:
    BimDbWindow skip BimDbCurtainWall skip BimDbDoor skip BimDbSpace skip BimDbBeam IfcWall

Note: You can also create a file where you include the entities you want to export, followed by '* skip'. This will give the same result.

7 Save the file to your computer.

8 Go to the BricsCAD file you want to Export.

9 Go through one of the following steps.
  - Type SETTINGS in the Command line and press **Enter**. Go to **BIM** > **Import and Export** > **IFC** > **Export mapping file path**. Click on the three dots and select the mapping file on your computer.

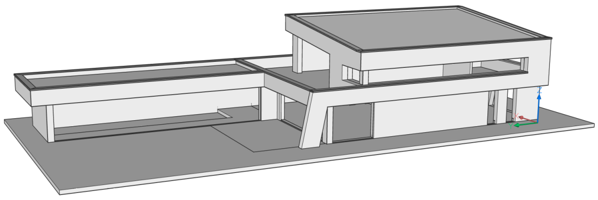


  - Type IFCEXPORTMAPPINGPATH in the Command line.
  - You are prompted: New value for IFCEXPORTMAPPINGPATH <"">:
    Copy the path of the destination file in the Command line and press Enter.

10 Type **Export** in the Command line and press Enter or click the BricsCAD icon in the ribbon and select Export to export the file.

11 Select the desired IFC file format in the **Save as type** list and save the IFC model to your computer.

12 To check if the IFC file is correct you can open the file in an IFC viewer or you can import the file in BricsCAD without an Import Mapping file. Type IMPORT in the Command line, Press enter, select the file on your computer and click **Open**.



Note: The same workflow can be used to use Import Mapping of IFC Files. To use the import mapping, the model should contain BIM information. Start from a model that is divided into different IFC elements.

25.3.14 Export Geographic Location to IFC

When geographic location data exist in a BIM model, proper export of said data is supported in the IFC export.

For IFC2x3:

In the **Spatial Locations Manager** dialog box (BIMSPATIALLOCATIONS command), with the **Use Geographic Location from GEO dialog** setting (under **Site** > **Geographic reference point**), you can choose whether the geographic location information from the **Geographic Location** dialog box (GEOGRAPHICLOCATION command) should be used, or the information provided in the **Spatial Locations Manager** dialog box.

- If this setting is False: Longitude, Latitude, and Elevation from the **Spatial Locations Manager** dialog box is exported (as IfcSite). No CRS (Coordinate Reference System) is exported.
- If this setting is True: Longitude, Latitude, and Elevation from the **Geographic Location** dialog box + CRS (Coordinate Reference System) is exported (as IfcSite and IfcProjectedCRS).

For IFC4:

- If CRS is defined in the **Geographic Location** dialog box, the settings in the **Spatial Locations Manager** dialog box are ignored.

  Longitude, Latitude, Elevation, and True North direction + CRS (Coordinate Reference System) is exported from the **Geographic Location** dialog box (as IfcMapConversion and IfcProjectedCRS).
- If no CRS is defined in the **Geographic Location** dialog box, no IfcMapConversion and IfcProjectedCRS information is exported.

  Longitude, Latitude, and Elevation is exported (as IfcSite) according to the **Use Geographic Location from GEO dialog** setting in the **Spatial Locations Manager** dialog box.

25.3.15 Model origin

Defines where to position the model with reference to the World Coordinate System (WCS) origin. Placing the IFC Global Coordinate System in WCS origin can result in a model positioned far from the origin.

The options are:

- **Place IFC Global Coordinate system in WCS origin**.
- **Place IFC Project Location in WCS origin**.
- **Place IFC Site Location in WCS origin**.

25.3.16 Guidelines for exporting colors and transparency

During IFC export, BricsCAD BIM includes colors and transparency settings in such a way that the model's appearance in IFC viewers should resemble the appearance in BricsCAD BIM as much as possible. Due to limitations of the IFC format, in some specific cases, colors and/or transparency settings might not be exported correctly. Try avoiding the following scenario: within one block, set the color to **By Block** for some solids, and assign a specific color to some other solids. In the IFC export, the **By Block** solids will not have the block reference color assigned.

25.4 Model federation and geo-referencing

The current system of handling reference points (Global Origin/Survey Location, Project Location, Site Location) differs from the one used before BricsCAD V25.

Prior to BricsCAD V25, reference points could only be set by importing an IFC file and were stored in the BSYSLIB file (displayed in the **Spatial Locations Manager** dialog box under **Site** > **IFC project points (relative to WCS)**) and used for all DWG files of the BIM Project.

With the current system, you can also manually set reference points using the SETSURVEYLOCATION, SETPROJECTLOCATION, and SETSITELOCATION commands (add points from scratch or edit existing ones). The X, Y, Z coordinates and angle of the reference points are now stored in the DWG file. This allows you to set different points for separate files.

25.4.1 Adding reference points from scratch

The default value of the reference points' coordinates and angle in a new drawing is 0.

Set a Survey Location point in a drawing using the SETSURVEYLOCATION command. Toggle On/Off the visibility of the Survey Location marker by modifying the GEOMARKERVISIBILITY system variable's value.

Note:

- The Survey Location point corresponds to the Geographic Location point. You can find its coordinates in the **Position** section of the **Geographic Location** dialog box (see the **GEOGRAPHICLOCATION command** article).
- The angle value set for the Survey Location point corresponds to the NORTHDIRECTION system variable's value and represents the angle between the True North and the Y-axis, measured clockwise. Modifying the NORTHDIRECTION system variable's value changes the Survey Location angle value only if a **GIS Coordinate System** is set in the **Geographic Location** dialog box.

Set a Project Location point in a drawing using the SETPROJECTLOCATION command. Toggle On/Off the visibility of the Project Location marker by modifying the PROJECTLOCATIONVISIBILITY system variable's value.

Set a Site Location point in a drawing using the SETSITELOCATION command. Toggle On/Off the visibility of the Site Location marker by modifying the SITELOCATIONVISIBILITY system variable's value.

Note: Both the Project Location and the Site Location will show the True North angle of the drawing. Its value is taken from the True North angle set via the SETSURVEYLOCATION or GEOGRAPHICLOCATION commands.

25.4.2 Importing reference points

Reference points can be imported in an existing drawing by importing an IFC file where the points information is stored.

1 Open the drawing.

2 Run the IMPORT command and select the IFC file.

3 In the **IFC Import Settings** dialog box set the model origin by choosing one of the available options:
   - **Place IFC Global Coordinate system in WCS origin**.
   - **Place IFC Project Location in WCS origin**.
   - **Place IFC Site Location in WCS origin**.

4 Click **Import**.

5 Save the drawing.

Existing reference points coordinates and rotation angles are overwritten by the information stored in the imported IFC file.

25.4.3 Exporting reference points

In case a Site, Project or Survey Location have been added in a drawing, these reference points will be used when exporting to IFC (via the EXPORT or IFCEXPORT commands).

Note: The Site Location, Project Location, and Survey Location, as well as geographical coordinates, can be exported to Datasmith (*.udatasmith via the EXPORT or DATASMITHEXPORT commands). See the related article **Using TwinMotion**.

When exporting from BricsCAD an IFC based project created with BricsCAD versions older than V25, which contains reference points stored in the old system, these will still be exported with the same results as before. In case reference points are added with the new commands, these will overwrite the old information during IFC export.

Note: Exporting to IFC, based on the old reference point information is not preferable, since not all information from the initial IFC import is stored in the project BSYSLIB file (for example, angle information for the Site Location). It is therefore advised to manually migrate the reference points.

25.4.4 Manually migrating reference points

There are two methods to manually migrate reference points information from BIM Projects based on IFC files saved with BricsCAD versions older than V25:

Re-import the original IFC file into a new drawing with the current BricsCAD version

1 Open a new drawing in the current BricsCAD version.

2 Run the IMPORT command and select the original IFC file.

3 Save the drawing.

The Global origin/Survey Location, Project Location, Site location, and rotation angles are derived from the information stored in the imported IFC file and saved in the new DWG file.

Manually add the reference points with the current BricsCAD version

1 Open the drawing in the current BricsCAD version.

2 Add the Survey Location, Project Location and Site Location with the new SETSURVEYLOCATION, SETPROJECTLOCATION and SETSITELOCATION commands based on the information in the **Spatial Locations Manager** dialog box or in the original IFC file you imported into the drawing.

3 Save the drawing.

25.5 Import Autodesk Revit® files in BricsCAD

25.5.1 Disclaimer

Unless otherwise indicated, the word **Revit** used in this article refers to the Revit® software and its peripherals by Autodesk®.

25.5.2 About Revit® import

The Revit® import function allows you to import RVT files into BricsCAD as 3D solids. Most project data and model metadata, i.e. BIM classification, spatial information, composition data of Walls, Slab, Roof, Beam, Columns, materials, etc., from the Revit® model are imported.

You can choose to import selected geometries by 3DViews, by Family Categories, by Levels or by View Worksets, depending on how the Revit® project file has been set out.

BricsCAD *V25* currently supports the import of **Revit 2015-2024** files only. The previous BricsCAD versions support to import Revit® files as shown in the table below. If you would like to import a model created in a later version of Revit®, please consider importing the model into BricsCAD as an IFC file. For more information, please visit the IFC Import and Export in the BricsCAD article.

BricsCAD BIM version	Supported Revit® version
BricsCAD BIM V22	Revit 2015-2022
BricsCAD BIM V23	Revit 2015-2023
BricsCAD BIM V24.1	Revit 2015-2023
BricsCAD BIM V24.2	Revit 2015-2024
BricsCAD BIM V25	Revit 2015-2024

25.5.3 Procedure: importing a Revit® file as a solid in BricsCAD

1 Open a new BricsCAD file.

2 Set the **RVTRFALEVELOFDETAIL** system variable value which determines the elements' level of detail during RVT import. The default value is **Fine**.

Set the **RVTVALIDATEBREP** system variable value which enables/disables BREP geometry validation. The default value is **Enabled**

Important: Only experienced users are encouraged to use other settings than the default ones. They have to be aware of how the incoming geometries have been set out in terms of Detail Levels within the Family Elements Visibility Settings in Revit®.

3 Type **Import** in the Command line and press Enter or click the BricsCAD icon in the ribbon and select **Import** to launch the IMPORT command.

4 The **Import file** dialog box pops up. Select **Revit Project (*.rvt)** in **Files of type**, select the file you want to import from your folder and click **Open**.

5 **You are prompted:** Choose elements [**A**ll/by **V**iews/by **C**ategories/by **L**evels/by **W**ork**S**ets]:

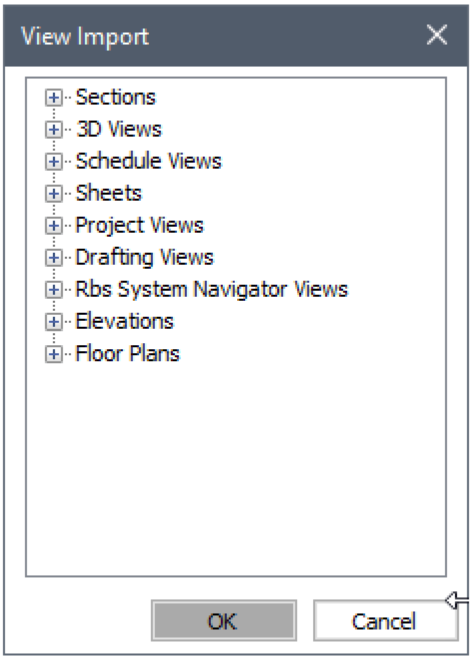
There are different filters to import a file. Select the filter you want to use by clicking your desired option in the Command line or by typing their respective letters in bold.

- **All**

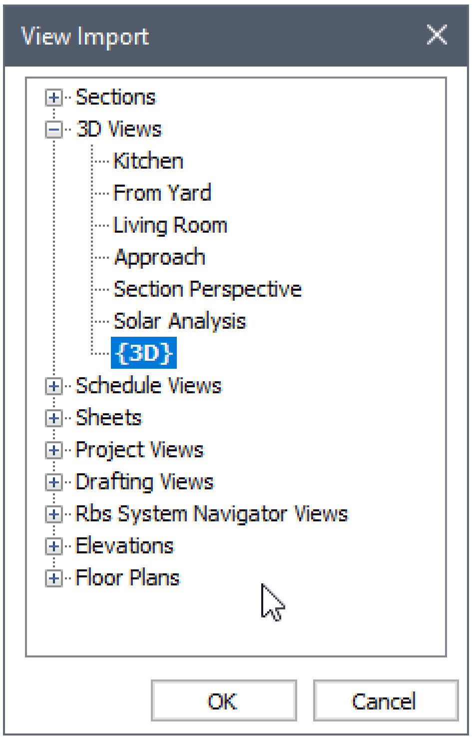
  All elements present within the RVT file will be imported.

- **by Views**

  Select a view from the **View Import** dialog box and the elements will be imported based on the view's Visibility/Graphics Overrides settings in Revit®. The views are structured based on their View Types.



You can expand any relevant or applicable View Type category for the specific views. If the **3D Views** category is expanded, it looks like this:



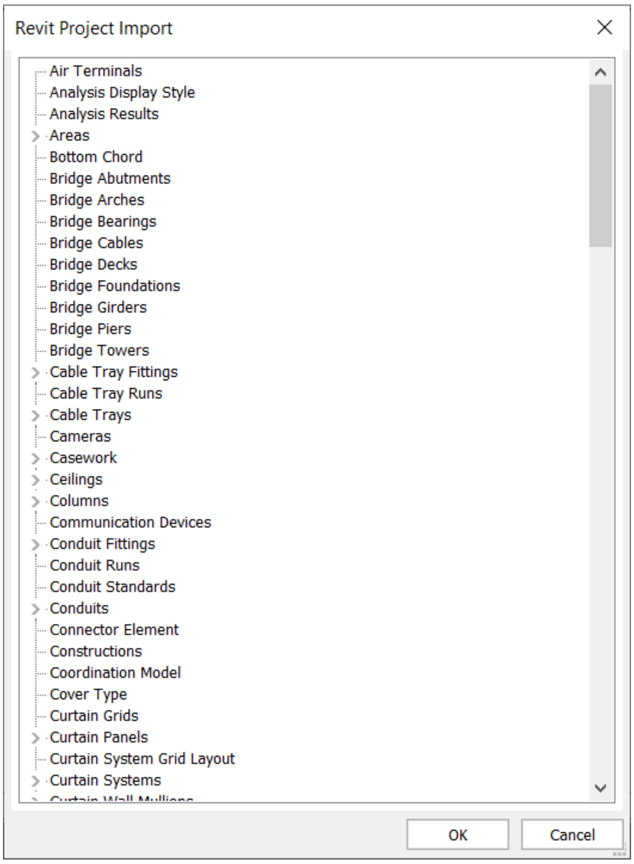
Note: The Revit® 3D views are imported as BricsCAD Named Views. A Revit® Project has several 3D Views defined which have several attributes like camera position, view type (perspective / orthogonal), Scale (Zoom Level) etc.

Note: Layer states are created to control visibility of elements in a generated section. Sections imported, both plan and elevation, are now in volume state.

Note: Importing elements via a 2D view, i.e. Floor Plans etc., only brings elements that are made visible in the Visibility/Graphic Overrides in the Revit® model settings for that particular view. It does not necessarily bring elements hosted onto its corresponding level, if they are made hidden on the plan view.

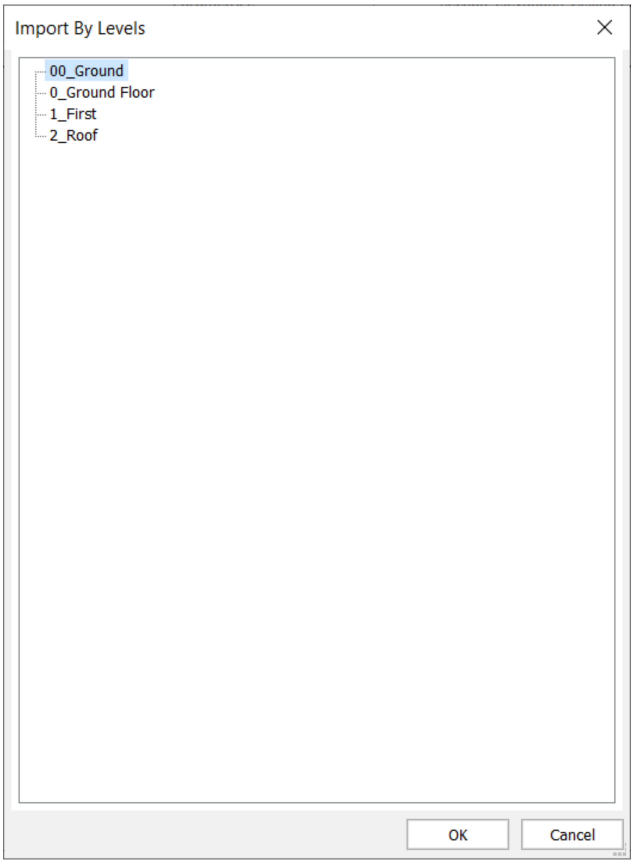
- **by Categories**

  You can select one or multiple built-in Categories and Element Types from the **Revit Project Import** dialog box. Elements of those chosen categories and element types will be imported.



- **by Levels**

  You can select one or multiple levels from the **Import By Levels** dialog box. Elements hosted to the chosen levels will be imported.

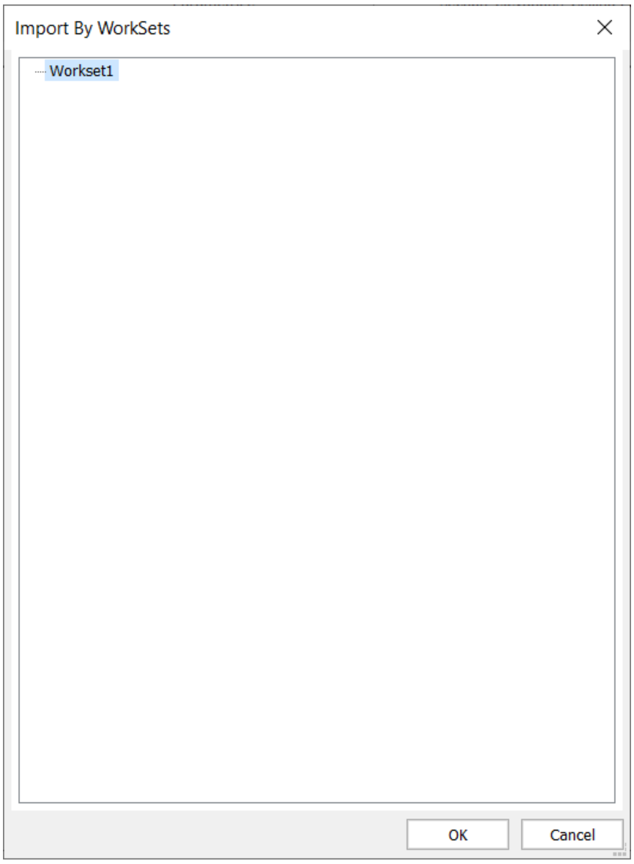


- **by WorkSets**

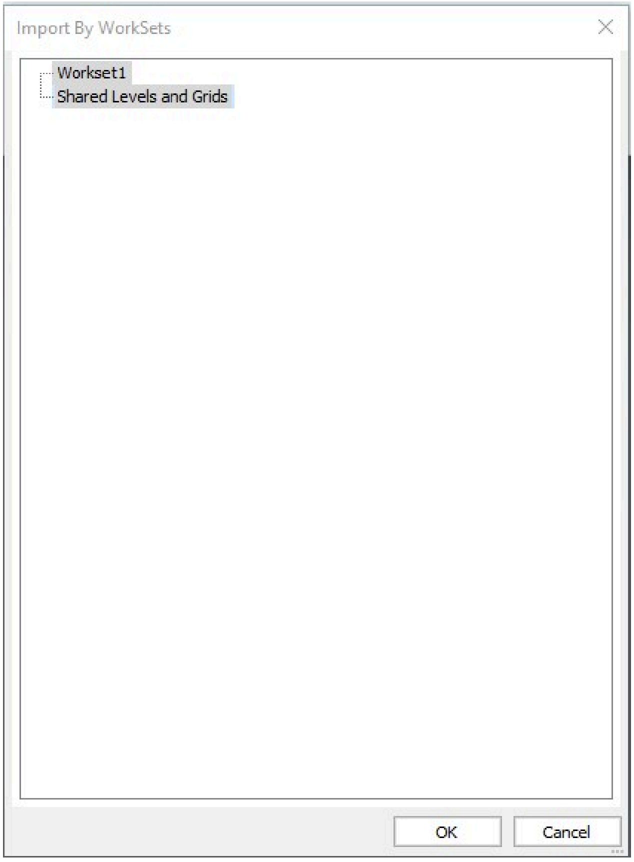
  You can select one or multiple worksets from the **Import By Worksets** dialog box.

  This method of importing works best on a workshared Revit® model. It is recommended that you import a Central Model to ensure you are importing the latest information in the worksets. Detaching the Central Model is optional.

  If you import a non-workshared Revit® model, only **Workset1** is available.



If you import a workshared Revit® model, either Central or Local, the User-Created worksets within the model will be available in the dialog. The elements which belong to this workset will be imported.



Note: The **Shared Levels** and Grids and **Workset1** worksets are created in Revit® by default when Worksharing is enabled and can be managed by the Revit® user.

6 If you have chosen to import by view, the Detail Level of the selected view set in Revit® will be used.

7 For the rest of the other options, the import is controlled by the values of the RVTRFALEVELOFDETAIL and RVTVALIDATEBREP system variables.

8 Select the View, Categories, Levels or WorkSets you want to import and click **OK**. The RVT file will be imported as 3D solid geometries with their BIM classification, materials, compositions, etc.

Composition data imported:

- Wall, Slab, Roof.
- Family Instances with structure material parameter (from tested RVTs, these can be Beams and Columns).
- Plies - including Physical Material, Function, Thickness
- For Physical Materials, appearances (Custom Hatches) are extracted into individual .PAT files
• .PAT files are added under "RvtPatterns" folder in Support folder
- Setting the reference face for Walls (before composition is assigned)

Note: Some of Revit®'s system families, such as walls and floors etc., are imported into BricsCAD as 3D solids, where you can further manipulate their geometries directly. This also applies to Model-

In-Place components and Element Types, such as Fascias, that can be defined geometrically by a Profile.

However, other system families designed with in-built constraints, i.e. stairs and curtain walls etc. are imported in BricsCAD as separate Block References. For example, Curtain Panel and Curtain Wall Mullion Types, which make up a Curtain Wall Type in Revit®, will exist as separate Block References. This also applies to all other loadable families, like furniture, placed within the Revit® model. For more information, please visit the article on Working with Blocks.

25.5.4 Importing a Revit® family in BricsCAD

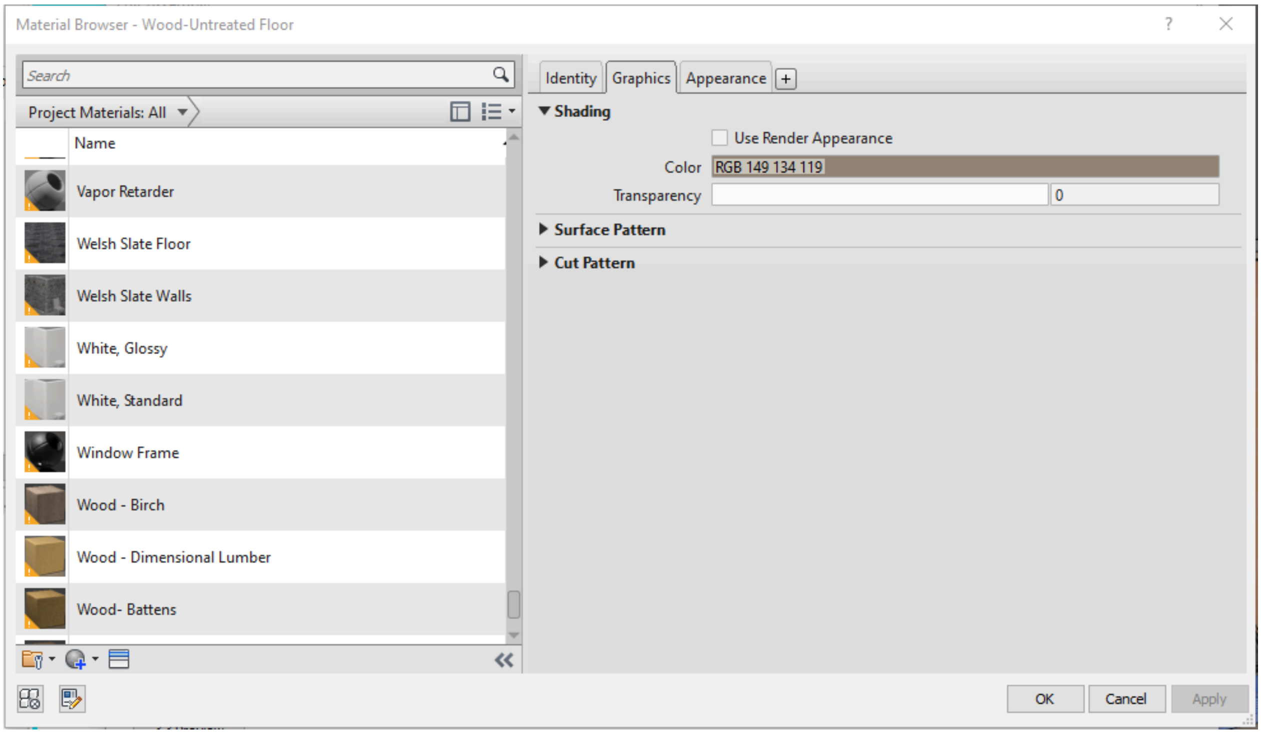
Please refer to the article **Importing Autodesk Revit® families in BricsCAD** for more information.

25.5.5 Texture and graphics of the imported geometries

Although project information and element metadata in a Revit® model are brought into BricsCAD, there are some settings to make the imported 3D solids appearances correspond to the original graphics set in the Revit®.

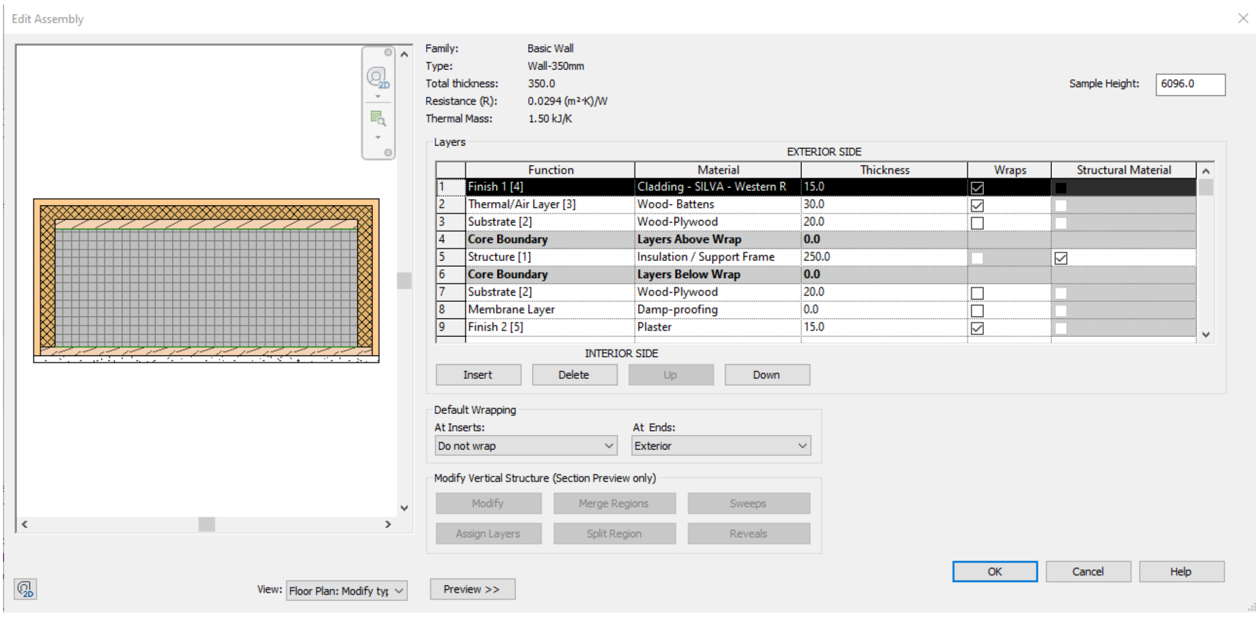
To retain the consistency of the graphical information during the import process, ensure your RGB color and transparency settings set in Revit®'s Material Browser are set to your desired values.

In the example below, the values taken will be **RGB 149 134 119** and **0** (%) for the colors and transparency values respectively.



If the imported element is a Revit® System Family with the ability to define its structure assembly, i.e. Wall Type, and is made up of various layers of materials or functions, BricsCAD applies the color template values from the outermost materials (on both the Exterior and Interior sides) accordingly to the corresponding geometry's faces.

Let's take an example from the illustration below, the values will be derived from the materials assigned on Layers 1 and 9, which are **Cladding...** and **Plaster** respectively.



Otherwise, BricsCAD applies the color value and the transparency settings, if applicable, to the 3D solid, replacing the respective parts of the Revit® Family. This also applies to those Loadable Families and Components, having loaded into the imported Revit® project.

25.6 Importing Autodesk Revit® families in BricsCAD

25.6.1 Commands

IMPORT, BMINSERT

25.6.2 Disclaimer

Unless otherwise indicated, the word **Revit** used in this article refers to the Revit® software and its peripherals by Autodesk®.

25.6.3 Important notification before getting started

RFA import is a work-in-progress. See the BMINSERT command.

BricsCAD *V25* currently supports the import of **Revit 2015-2024** files only. The previous BricsCAD versions support to import Revit® files as shown in the table below. If you would like to import a model created in a later version of Revit®, please consider importing the model into BricsCAD as an IFC file. For more information, please visit the **IFC import and export in BricsCAD** topic.

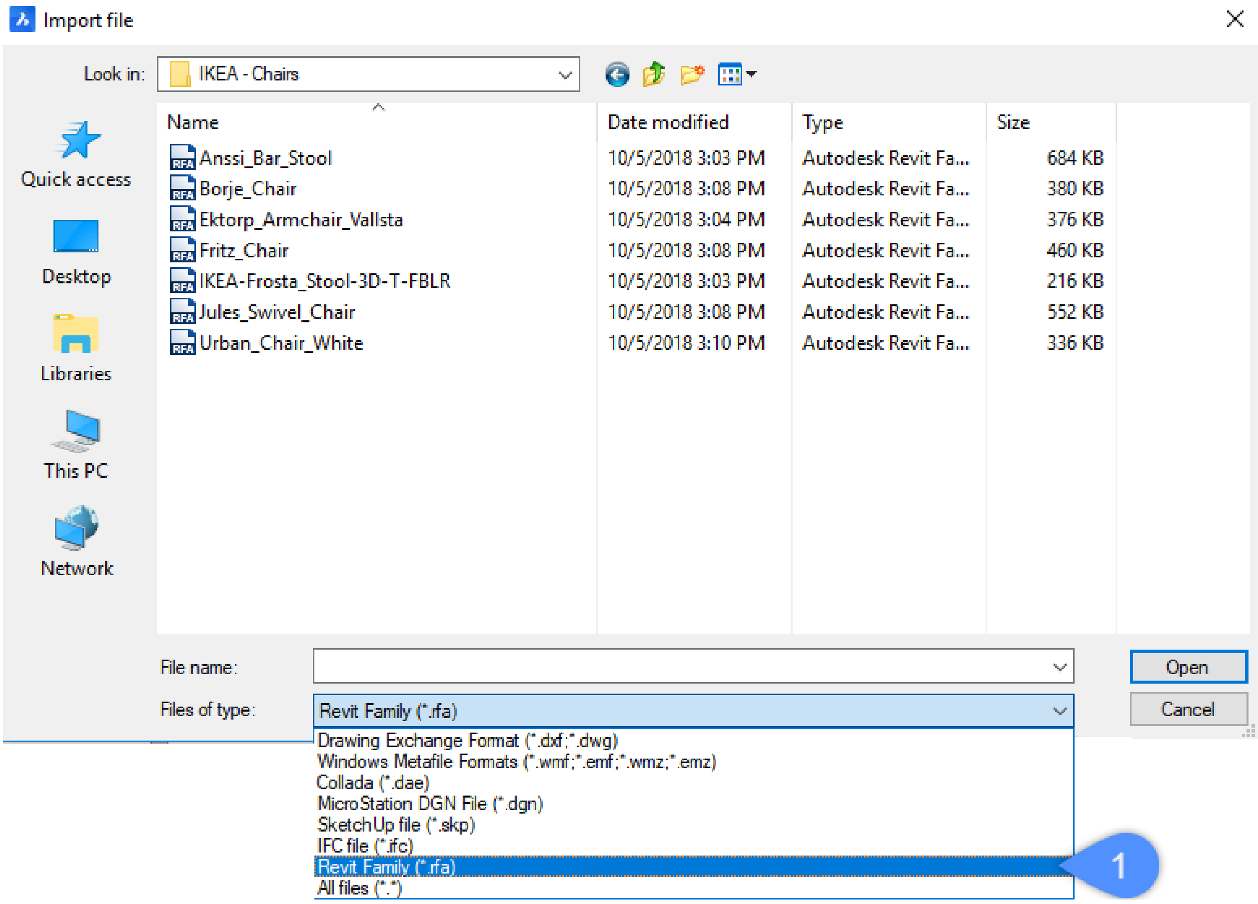
BricsCAD BIM version	Supported Revit® version
BricsCAD BIM V22	Revit 2015-2022
BricsCAD BIM V23	Revit 2015-2023
BricsCAD BIM V24.1	Revit 2015-2023
BricsCAD BIM V24.2	Revit 2015-2024
BricsCAD BIM V25	Revit 2015-2024

25.6.4 Importing RFA family files

1 Launch the IMPORT command.

1 The **Import file** dialog box displays.

2 Select **Revit Family (*.rfa file)** (1) in the **Files of type** list.



3 Select the .rfa file you want to import.

4 Click **Open** to import the Revit® Family.

When the RFA family file is imported the geometry from this file is included in BricsCAD as separate 3D solids.

25.6.5 Procedure: inserting RFA family files

Note:

- When DUCS = ON (Dynamic UCS) the RFA entity aligns with the 3D solid face under the cursor.
- When DYN = ON (Dynamic Dimensions) the distances from the nearest parallel edge on the face of the 3D solid display.

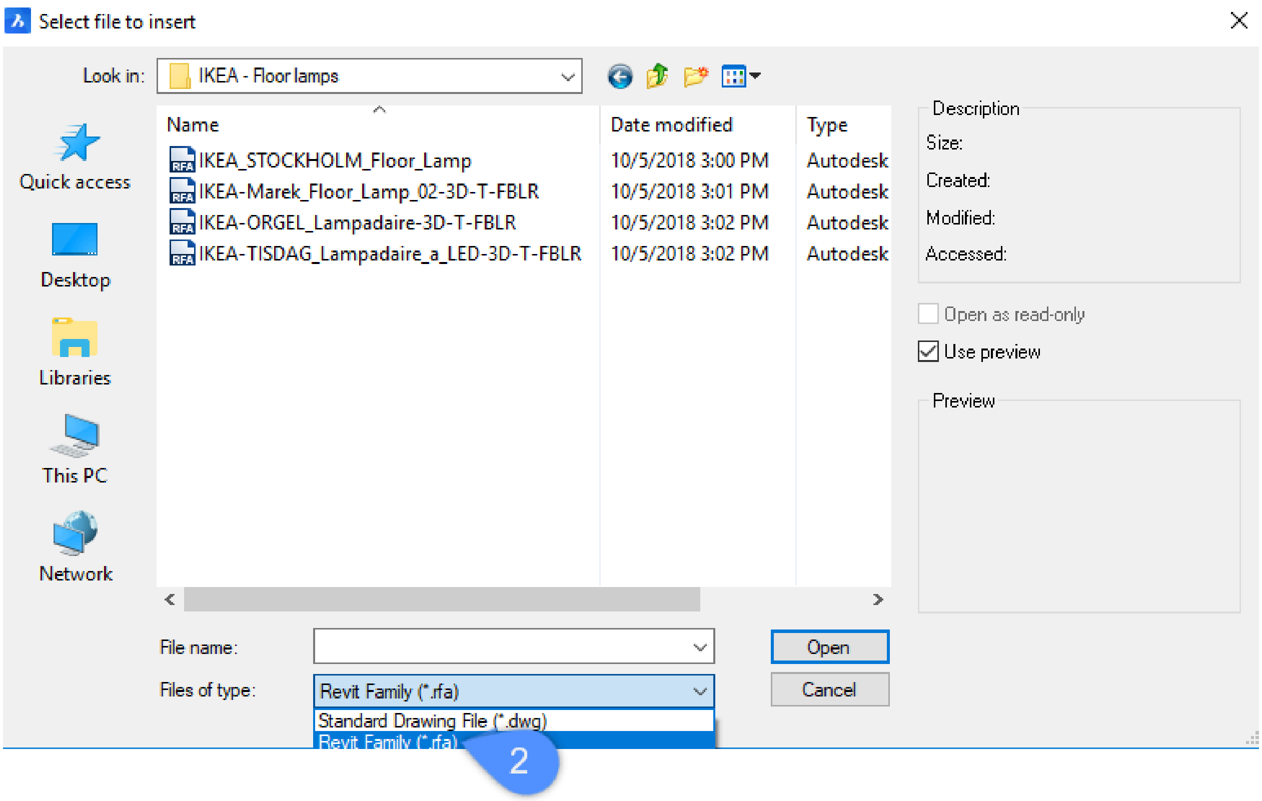
Using BmInsert

1 Launch the BMINSERT command.

1 A **File** dialog box appears.

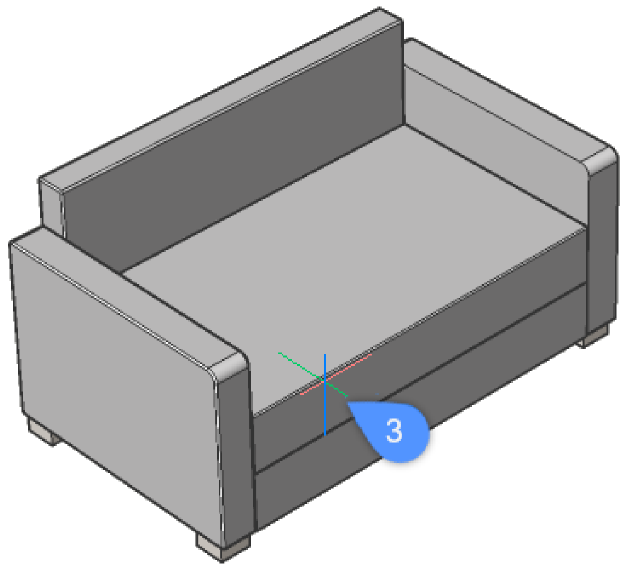
2 Browse to the folder where your .rfa family files are saved.

3 Select **Revit Family (*.rfa file)** (2) in the **Files of type** list.

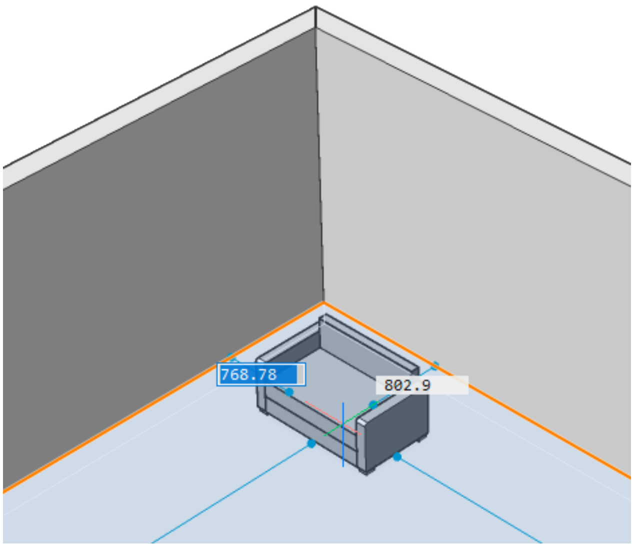


4 Select the .rfa file you want to import.

5 The selected file is attached to the cursor (3).



The following illustration shows inserting a Revit® family sofa when both **DYN** and **DUCS** are On.

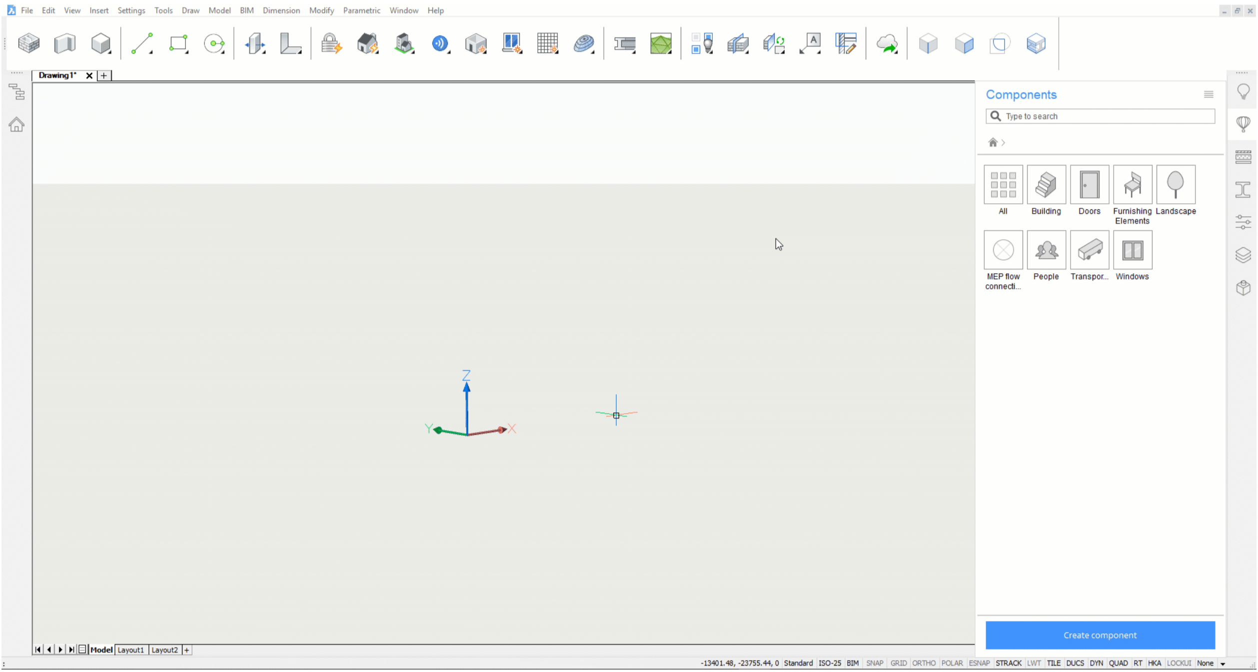


6 Do the following to place the selected RFA family file using the dynamic dimension fields:
   - Hit the Tab key to select one of the dynamic dimension fields, then type a value and hit the Tab key to lock the dimension. The locked dimension will turn red.
   - Repeat the previous step to lock another dynamic dimension field.
   - Optionally hit the Tab key to adjust the dynamic dimensions.
   - Press Enter to accept the current position.

Using the Library panel

1 To insert a RFA family folder in the **Library** panel, add the RFA root folder path to the Library directory path in the **Settings** dialog box. The content of the root folder and all subfolders will be available in the **Library** panel.

The following animated gif shows how to add the RFA root folder path to the components directory path.



The RFA family files are ready to insert in the model space.

If necessary, choose to **Generate Thumbnails** in the hamburger menu on the **Library** panel.

2 Drag and drop the RFA file from the **Library** panel into your model space.

25.7 Using Xrefs

25.7.1 Commands

XREF, XATTACH, XOPEN, XCLIP

25.7.2 About

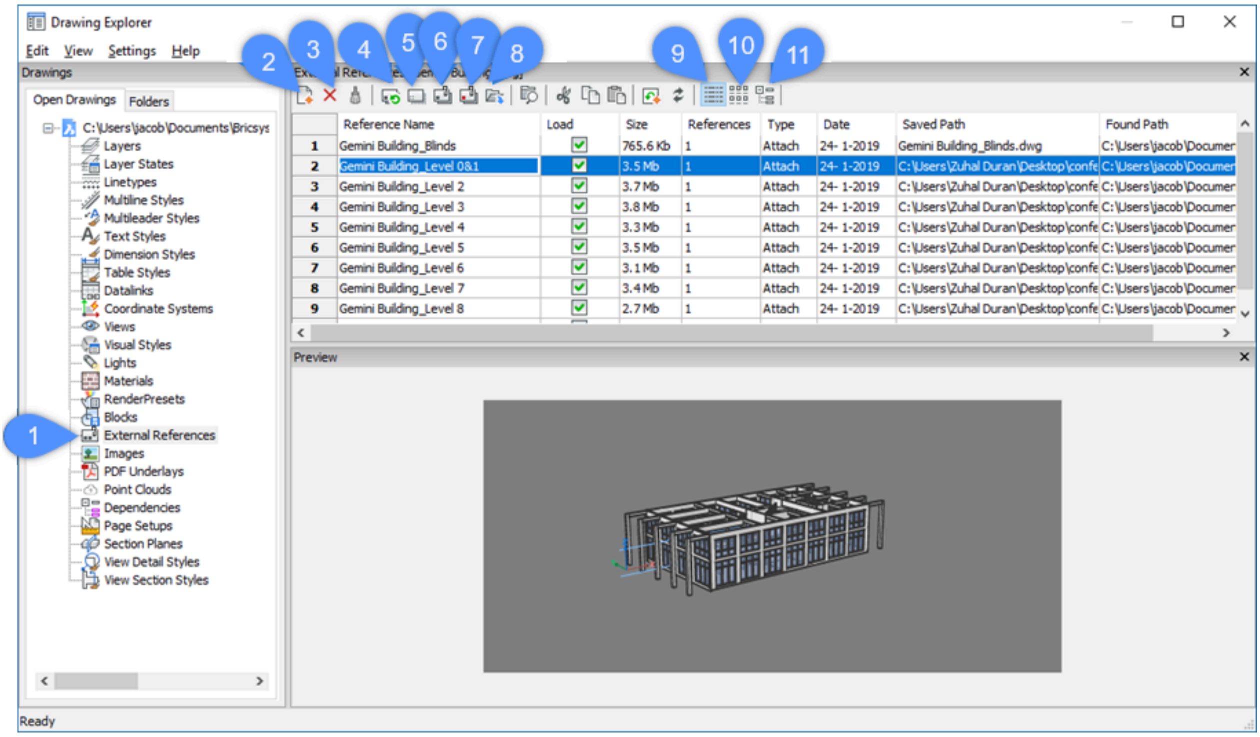
Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. Further advantages of using the Xrefs are that the source models can be edited simultaneously by different members of the design team, and the master model can always be updated to show the latest version of each of the source models.

Note: You can distinguish between XRef entities and entities in the current drawing using the XDWGFADECTL system variable. This system variable allows to fade the display of XRefs. You can give the values between 0 (no fading) and 90 to change the default value of the fading.

25.7.3 Procedure: managing XRefs in the current drawing

1 Open the **Drawing Explorer** dialog box and click the **External References** section.

The XRef preview pane appears, allowing the user to attach a model or manage the existing XRefs on the list. The following illustration shows the XRef preview pane.



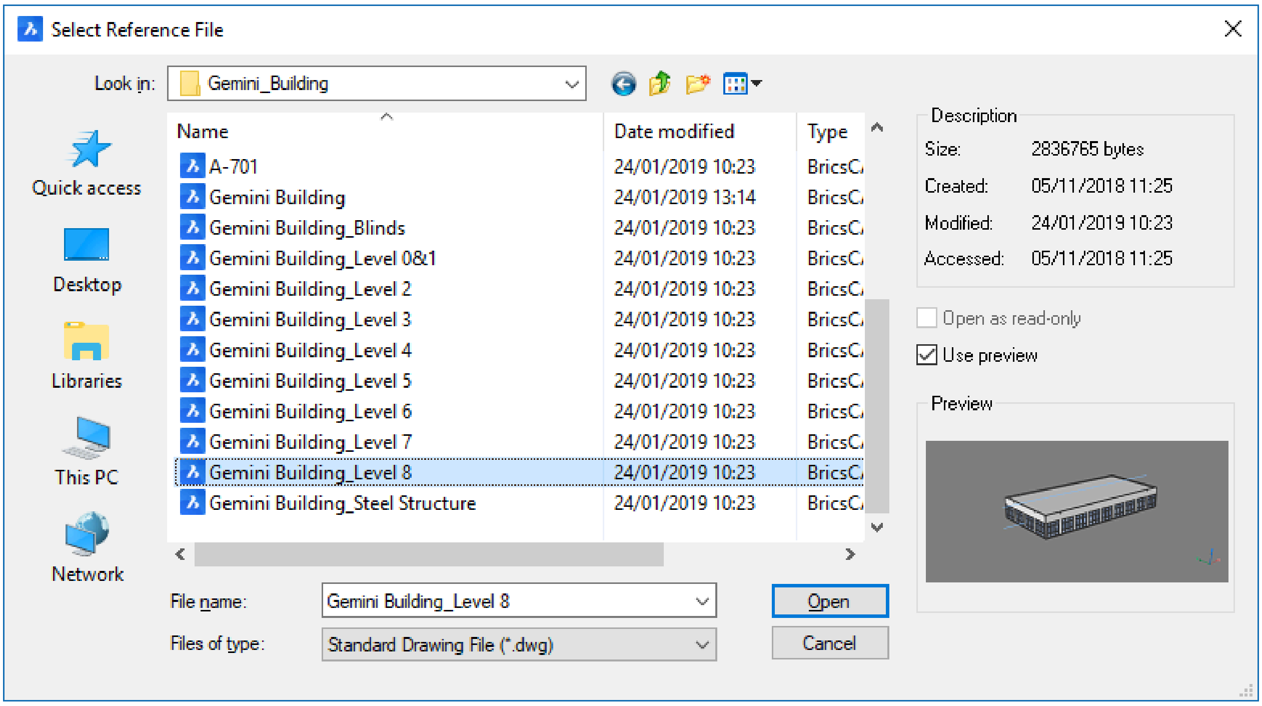
External References(1)	Shows the External References section.
Attach XRef (2)	Attaches a model as an external reference.
Detach XRef (3)	Removes the selected XRef completely from the current model.
Reload XRef (4)	Reloads the selected XRef. This is useful when the source models have been changed, and you want to see the updated versions in the master model.
Unload XRef (5)	Unloads the selected XRef. This hides the XRef from the view.
Bind XRef (6)	Merges the source models permanently to the current drawing. Merging the XRef to the current drawing makes the XRef part of the drawing. References to the XRef disappear from the Drawing Explorer.
Insert XRef (7)	Converts XRefs into blocks.

Open XRef (8)	Opens the referenced drawing.
Detail view (9)	Displays the details of XRefs.
Icon view (10)	Displays icons of the XRefs.
Tree view (11)	Displays the relationship between XRefs in a tree-like view.

2 To attach externally referenced models, click the **Attach XRef** tool button on the **Drawing Explorer** dialog box.

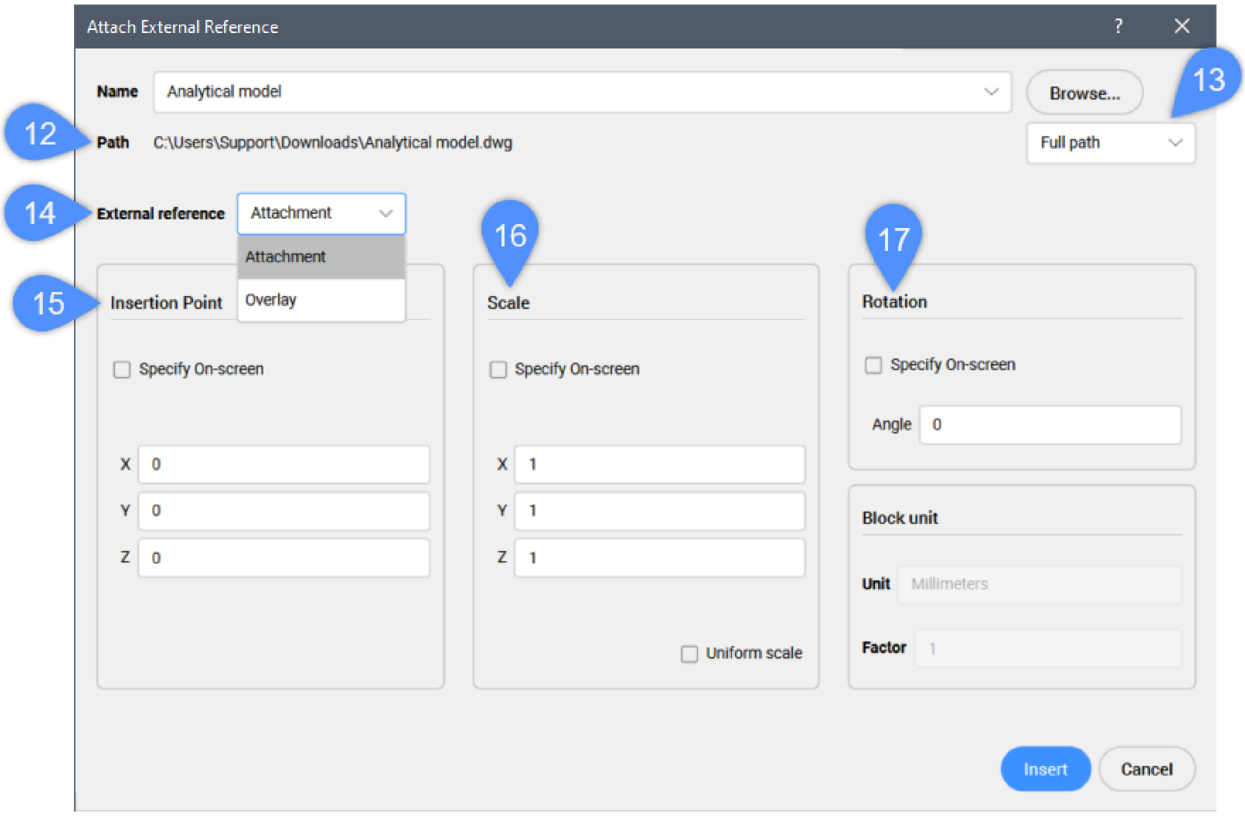
The **Select Reference File** dialog box displays.

The following illustration shows the **Select Reference File** dialog box. From this dialog box, select the DWG file you wish to attach in your main model, and then click **Open**.



Next, the **Attach External Reference** dialog box displays.

The following illustration shows the **Attach External Reference** dialog box. The table below presents each option of the **Attach External Reference** dialog box. From this dialog box, select the options you want to apply, and then click **OK**.

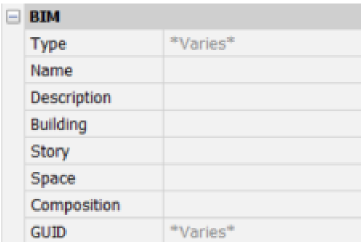


Path field (12)	Displays the full path of the selected drawing.
Path type (13)	Includes **Full path**, **Relative path**, and **No path** options. **Full path**: When you move your drawing containing an XRef to another location, BricsCAD will still search for the XRef in the previous location and it will not find this XRef. **Relative path**: When you move your drawing containing an XRef to another location, the new file path will be created in the new location. The file folder will set up correctly using the same folder structure as the first one.

External Reference type (14)	Includes **Attachment** and **Overlay** reference types. **Attachment**: Attaches all XRefs, including nested XRefs. For example, you have the first level of XRef “X” and this contains nested XRefs “Y” and “Z”. When the attachment option is selected the first level of XRef and its nested XRefs Y and Z will be attached to the drawing. **Overlay**: Attaches only the first level of xref (X), nested xrefs (Y, Z) will be not loaded. This can also be used to prevent circular references.
Insertion Point (15)	Allows the user to specify the Insertion point on the screen or in the X-, Y-, Z- fields.
Scale (16)	Allows the user to specify the Scale on the screen or in the X-, Y-, Z- fields. Check the **Uniform Scale** option to specify the scale in the X- field.
Rotation (17)	Allows the user to specify the Rotation angle on the screen or in the **Angle** field.

Note that the **BIM XReference** option from BIM Classify allows you to classify the model as an external reference. The identification of information will not explicitly be represented in the current model or in the project database. Such information appears after attaching the model to another project. To do so, follow the above step.

The following illustration shows the BIM category in the **properties** panel after attaching the model that has been classified as an XRef.



3 To detach an existing XRef, select the XRef you wish to detach in your main model, and then click the **Detach XRef** tool button on the **Drawing Explorer** dialog box.

The selected XRefs are detached.

Note: Press and hold the CTRL key to select multiple XRefs or to unselect a selected XRef.

4 Do one of the following to unload and reload an existing XRef.

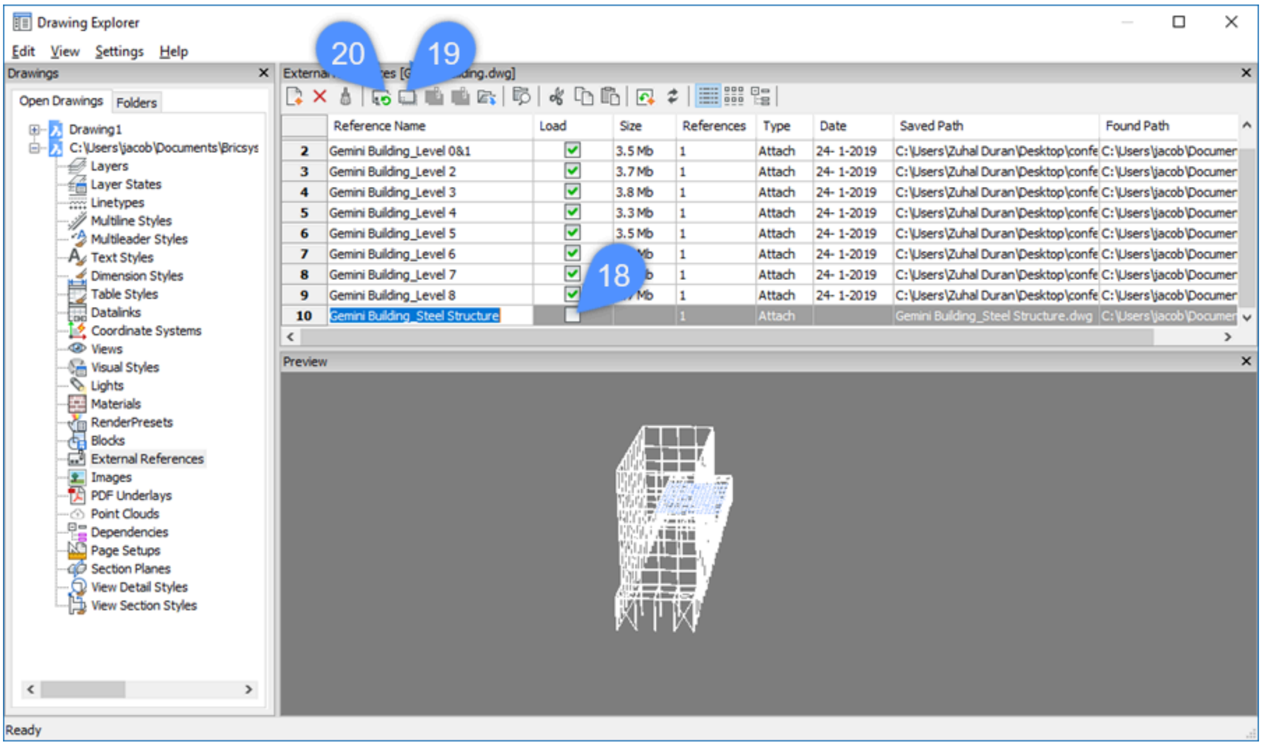
- Click the **Load** (18) column of a loaded XRef.

  This XRef is unloaded in your main model.

To reload the XRef check the load box of this XRef.

- Select the XRef, then click the Unload XRef (19) tool button on the Details toolbar.
- Select the unloaded XRef, then click Reload XRef (20) tool button.

The following illustration shows the **Load** column (18), Unload XRef (19) and Reload XRef (20) tool buttons on the **Drawing Explorer**.



5 To modify the attachment type, click the **Type** column of an existing XRef and switch from **Attach** to **Overlay** or vice versa. The same process can be followed for modifying the **Saved path** of an XRef and **Found path** of an overlay.

6 Click **X** to close the dialog box. Notice that the changes that have been made appear after you close the **Drawing Explorer**.

26. Design documentation

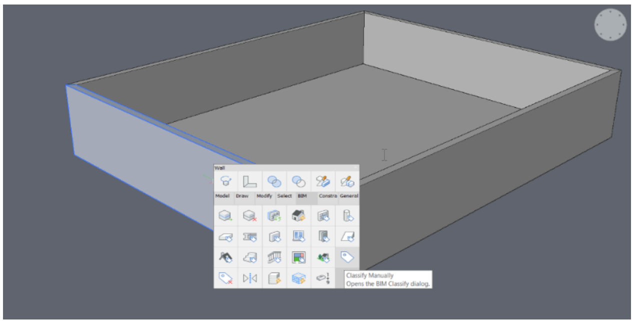
26.1 Annotations in 3D

26.1.1 Commands

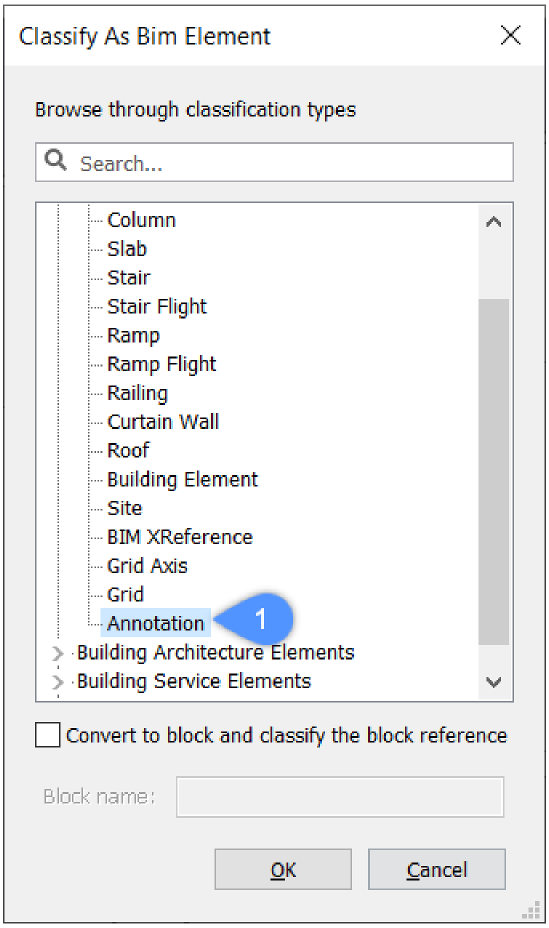
BIMTAG, BIMCLASSIFY

26.1.2 BIM classify as an annotation

1 Select a solid and select **Classify Manually** in the Quad or classify as **Other** after launch the BIMCLASSIFY in the Command line:

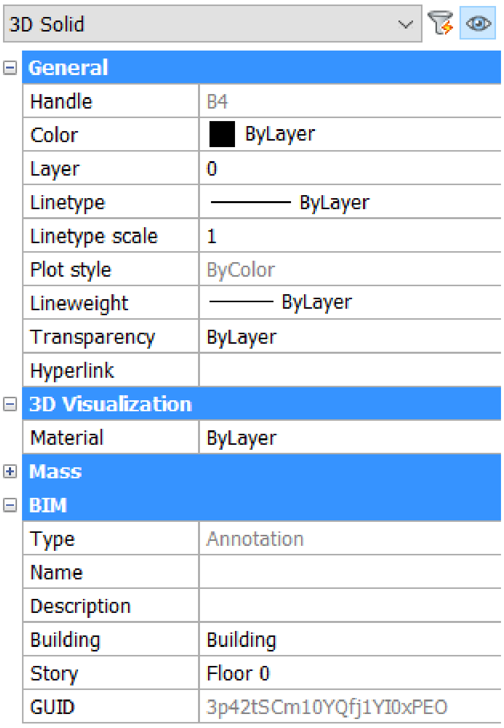


2 Choose to classify as **Annotation (1)** in the folder of **Building Core Elements** and click **OK**:



3 When you open the **Properties** panel of the selected solid you can change the **Name**, **Description**, **Building** and **Story**.

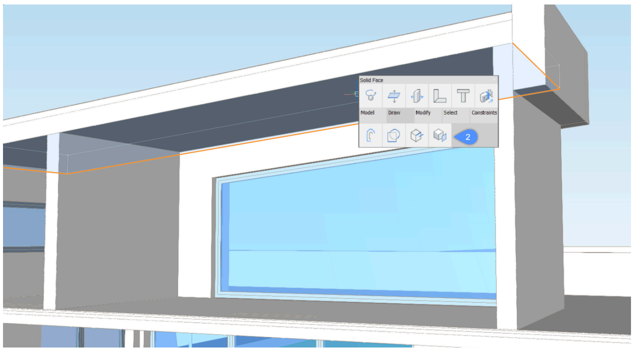
Note: The property **GUID** cannot be changed because it has a unique, automatically generated string.



26.1.3 Show added information of a finishing material

In this procedure, you will add design information to a concrete roof structure.

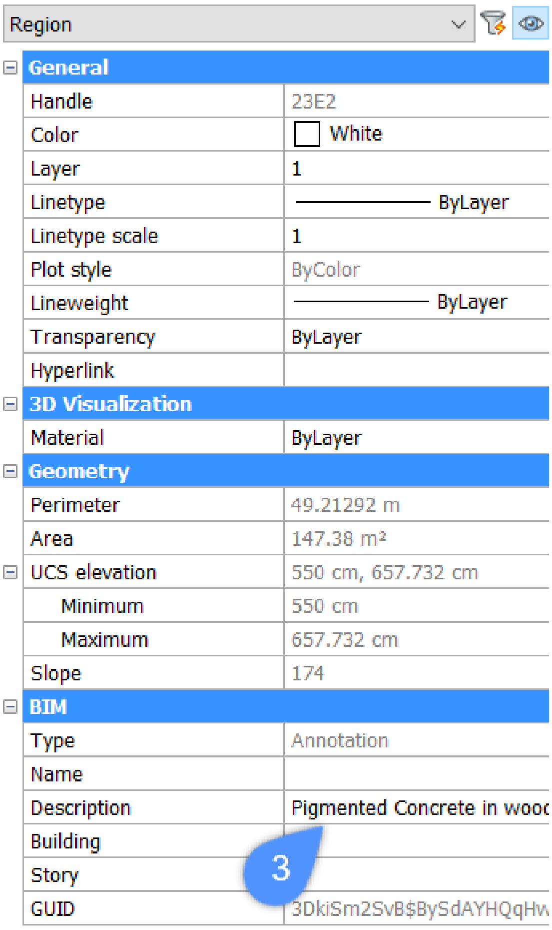
1 Use the XFACES (2) command under **Draw** in the quad, to extract the face of the **Roof** entity. A region is now created.



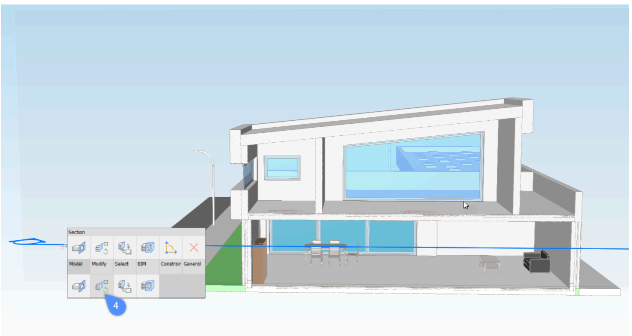
2 Classify the region as an **Annotation** as seen above.

3 Enter the desired text in the **Description** field (3) (found in the **Properties** panel):

Note: By default, the BIM Tag references the **Description** property.



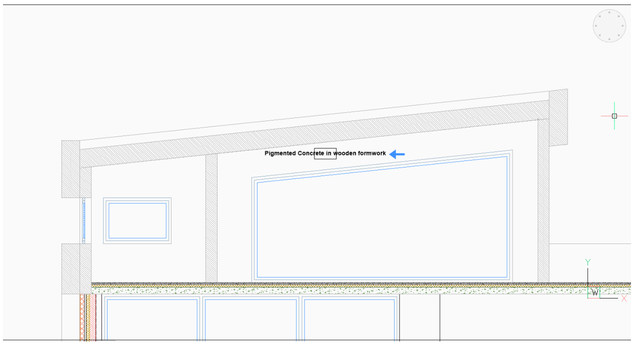
4 Make a section through the region (with BIMSECTION command) and select **Generate Section**(4) ( ) from the Quad:



5 Open the generated section drawing.

6 Type BIMTAG in the Command line OR select **Tag Automatically** in the **Annotate** tab of the Ribbon.

The text that you added is now in the 2D drawing:



26.1.4 Show the surface area of an extracted surface

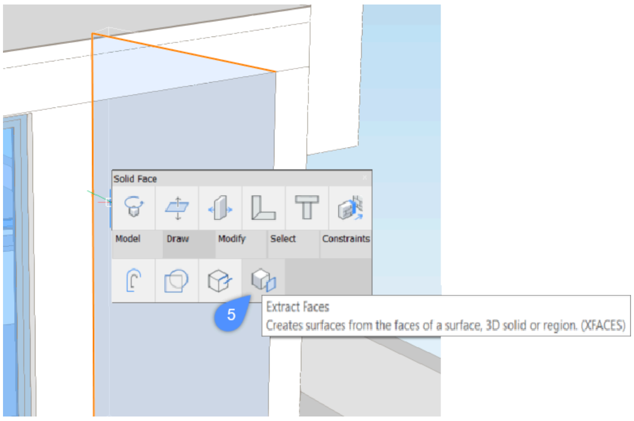
This procedure will teach you how to display other properties of an entity classified as an **Annotation** object.

Note: The annotation tag pulls information from the **Description** property by default.

For more information about tagging, see the article **Using BIM Tags**.

1 Select the face you want to extract the information from.

2 Select **Extract Faces (5)** from the Quad. A region is now created.



3 Classify the region as an **Annotation** as above.

4 Make a section through the solid (with BIMSECTION command) and select **Generate Section** ( ) from the Quad.

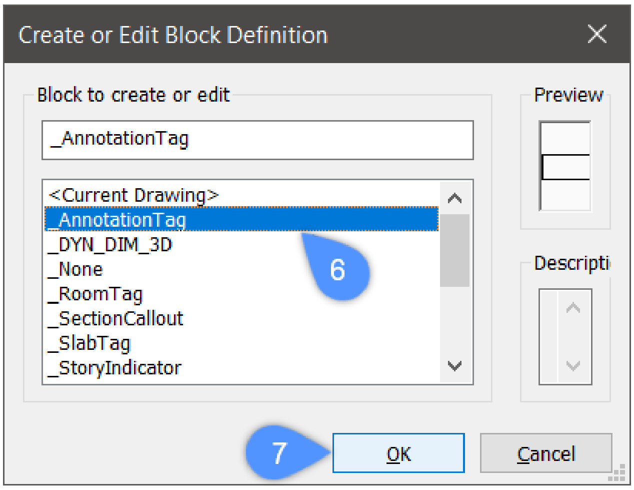
5 Open the generated section drawing.

6 Type BIMTAG in the Command line OR select **Tag Automatically** in the **Annotate** tab of the Ribbon.

Note: The tag appears empty because it refers to the **Description** property by default.

7 Launch the BEDIT command.

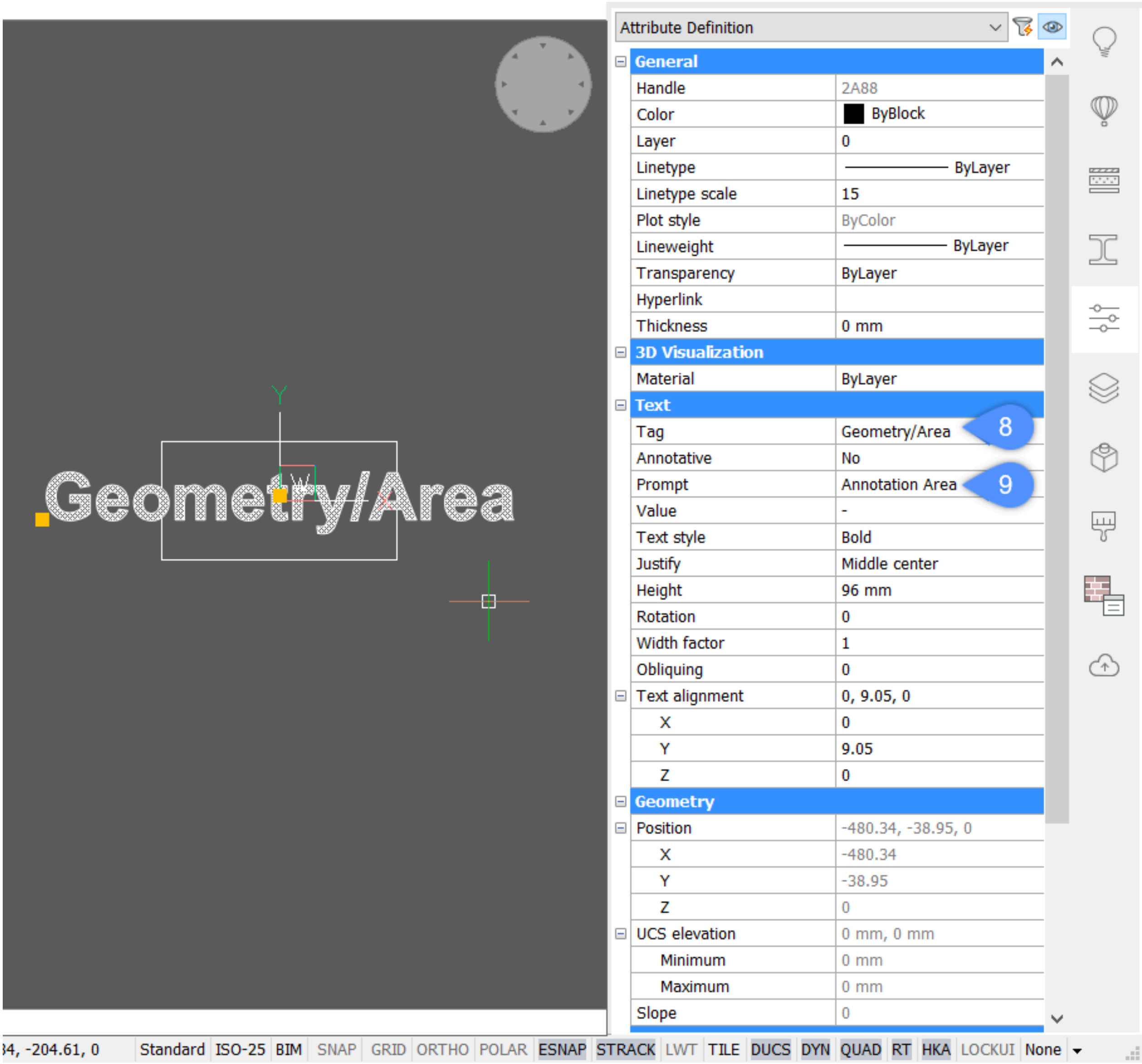
8 Select **_AnnotationTag (6)** and click **OK (7)** .



9 Select the attribute definition, in the **Properties** panel.

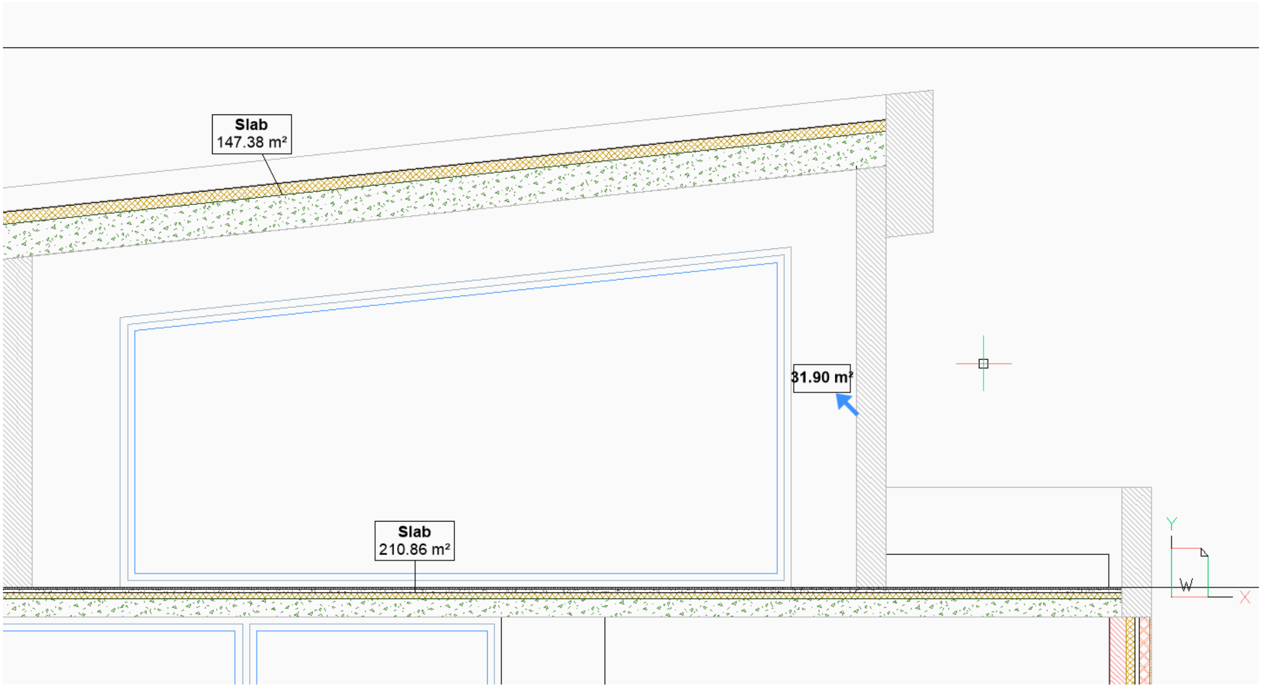
10 Edit the **Tag** property, change it to **Geometry/Area (8)**.

11 **(Optional)** You can also change the **Prompt** property (9).



12 Enter BCLOSE in the Command line and click **Save**.

13 Select the viewport and pick **Generate Section** ( ) from the Quad. The area of the region is now displayed in the tag.



26.2 BIM tags

26.2.1 Source files

The following source files are used to create BIM Tags in a generated BIM section drawing:

- **_SectionTag.dwg**: contains the tag blocks and multileader styles.
- **_TagTypeToStyle.xml**: links a BIM entity type to a multileader style.

Both files can be found in the folder:

C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\Bim\Sections

Note: When assigning tags, BricsCAD needs to access the 3D model. For this reason, it may take several seconds to place the first tag, if the 3D model is not currently opened, especially in large models. If you want to avoid this waiting time, consider having the 3D model open while placing tags.

26.2.2 Assign BIM tags manually

The **Manual** option of the BIMTAG command allows you to select the BIM element that you wish to create a BIM tag for.

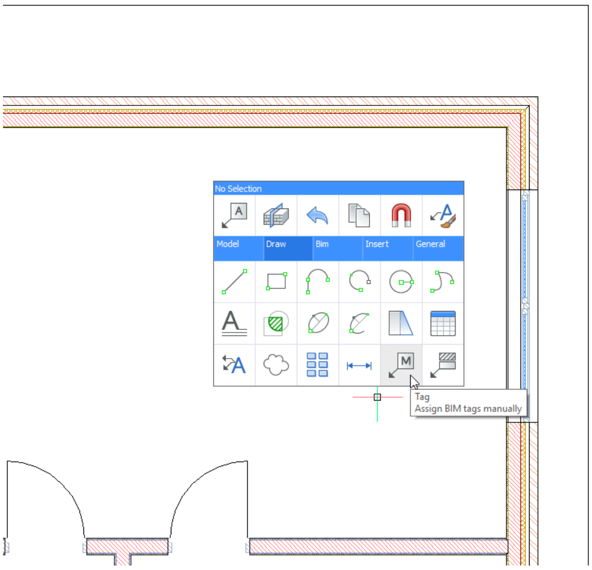
For more information about this option, visit the Command reference article **BIMTAG command**.

Manually create a BIM tag

1 Open the section drawing for which you want to create BIM tags.

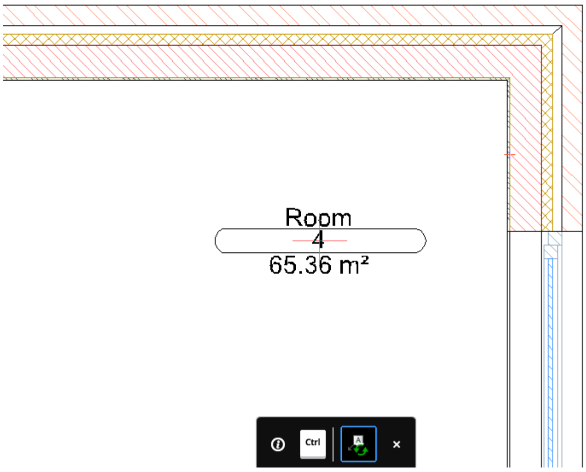
1 ***Note:*** For more information about generating section drawings, please read the BIM article **Generating Drawings** first.

2 With no selection, select **Tag Assign BIM tags manually** from the Quad or enter BIMTAG followed by **M** into the Command line to choose the manual option.



You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

3 Select a point on a line of a sectioned BIM element or click inside its boundary to generate a tag for it. The tag will appear. If the selected point is shared between multiple BIM entities, you can use the **Ctrl** key to cycle through the different possible tags.



You are prompted: Place the tag [Next element]:

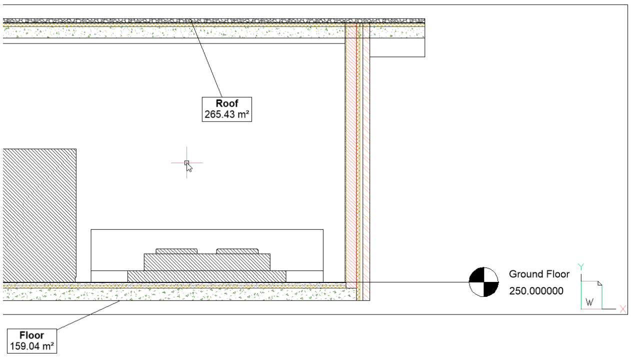
4 Move the cursor to adjust the position of the tag and left-click to place it.

5 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

6 Repeat steps 3 and 4 for all of the entities you wish to tag. If you do not require more tags, press the **Esc** or **Enter** key to complete the command. If you want all elements to be tagged, you can use the

BIMTAG Select all entities command.

The same steps above can be used to tag a floor slab or a roof.



26.2.3 Assign BIM tags automatically

If you want all BIM elements in a section drawing to be tagged, it is better to use the **Auto** option of the BIMTAG command. This command option automatically assigns BIM tags to all BIM entities inside a viewport. If the model is edited or new elements are added, the BIM tags can be updated and new tags added.

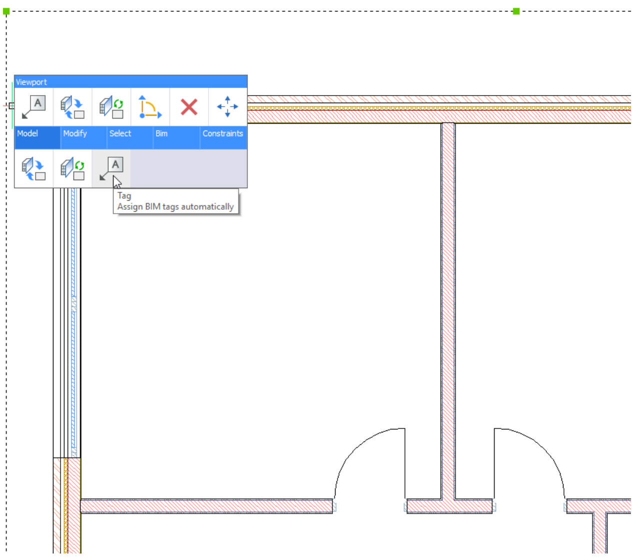
For more information about this option, visit the Command reference article **BIMTAG command**.

Automatically create BIM tags

1 Open the section drawing for which you want to create BIM tags.

2 Select a viewport containing the section you want to add BIM tags to.

3

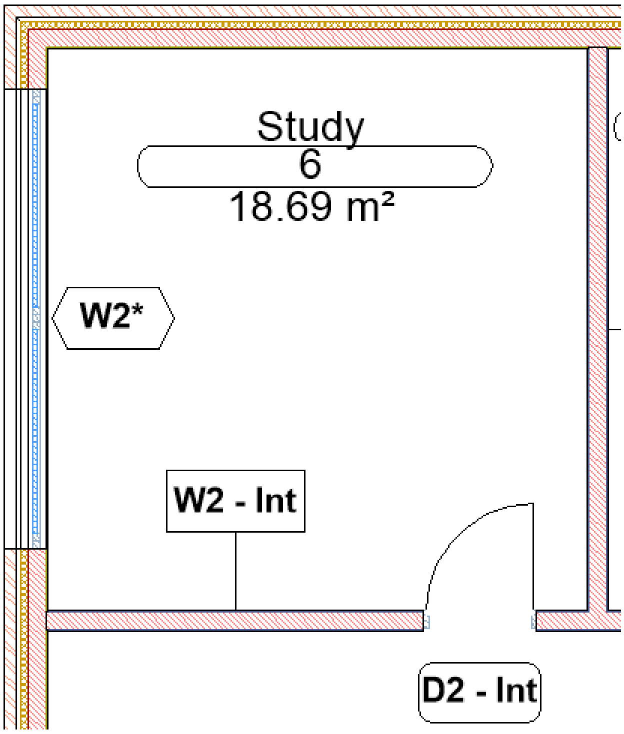


4 Select **Tag Assign BIM tags automatically** from the Quad or enter BIMTAG in the Command line.

As section indicators, story indicators and grid callouts are generated automatically within section update, it means tags already have been assigned to the section, you will be prompted:

Tags already have been assigned to the section. Choose an update option [Update existing/update existing and add New/Regenerate all] <Regenerate all>:

With the **update existing and add New** option, the BIM tags have automatically been generated for each of the BIM entities in the viewport.



You are prompted: Select BIM section viewport [Manual/Box selection/selection options (?)]:

5 If you wish to automatically create BIM tags in another viewport on the sheet, select the viewport. If you do not require any further BIM tags, press **Esc** to complete the command.

26.2.4 Creating tags with fields

1 Open the section drawing for which you want to create BIM tags.

2 Use the MLEADER command to create a tag.

3 ***Note:*** If there are components overlapping one another, some errors may appear. To avoid that, you have to select the edges of the object that needs to be tagged. Otherwise, the command prioritizes the component underneath.

4 Select the **Field** option in the **Text formatting** window.

5 In the **Field** dialog box, navigate to **Extensions** > **Object2dMLeader**.

6 Under the **Property Category** field, select the **BIM** option. Select the **Name** option in the **Property** field.

7 The name of the selected element is displayed in the **Name** field.

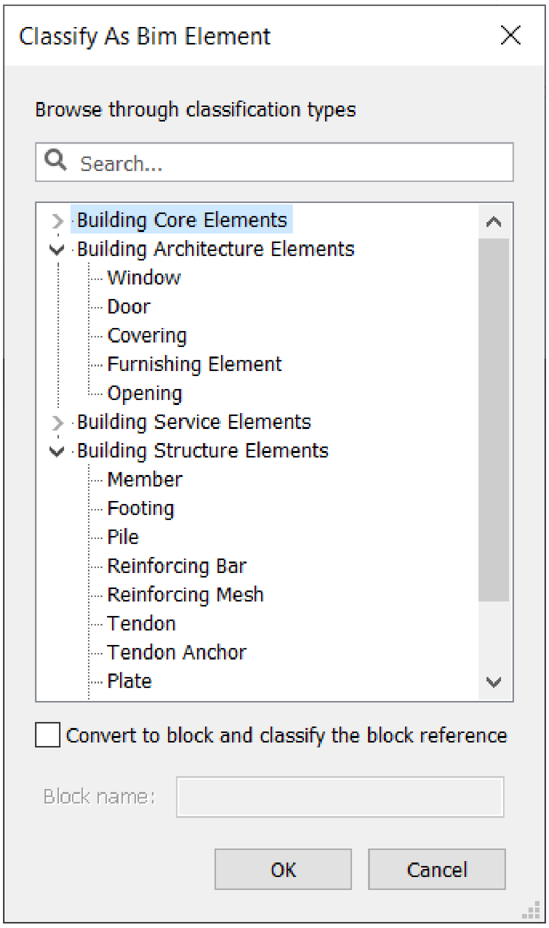
Note: If you want to change the name displayed, you need to rename the component.

8 Press **OK** to exit the **Field** dialog box and the **Text formatting** window.

26.2.5 Create the source data for a BIM entity type

By default the source data for the following BIM entity types already exist; **Annotation**, **Beam**, **Column**, **Curtain Wall**, **Door**, **Plies**, **Railing**, **Ramp**, **Roof**, **Room**, **Slab**, **Stair**, **Wall**, **Window**, and **Detail references**.

Available BIM entity types are listed in the **Classify As Bim Element** dialog box. To open this dialog box, choose the **Other** option of the BIMCLASSIFY command.



Create the tag block and multileader style

To create the source data for a BIM Member entity type:

1 Type SUPPORTFOLDER in the Command line and press **Enter**.

2 Once the folder has opened, navigate to the **BIM** > **Sections** folder and open the _SectionTag.dwg file.

3 Launch the EXPBLOCKS command.

4 The **Drawing Explorer / Blocks** dialog box displays.

5 Select the **_AnnotationTag** block (or any other existing block that you want to use as a template).

6 Right-click and choose **Copy** from the context menu.

7 Right-click and choose **Paste** from the context menu.

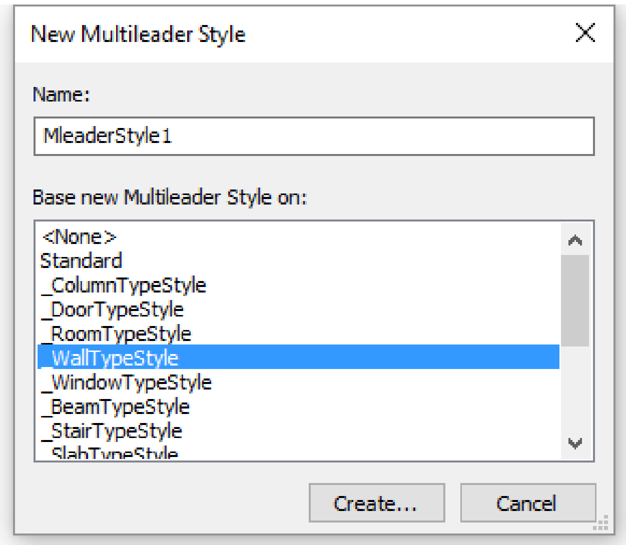
8 Click **Copy, but keep both** on the **Copy/Paste** dialog box.

9 Rename the copied block to **_MemberTag**.

10 Select **Multileader Styles** in the **Drawing Explorer** dialog box.

11 Click the **New** ( ) tool button.

The **New Multileader Style** dialog box displays:



12 In the **Name** field, type **_MemberTypeStyle**.

13 Select a source style in the **Base new Multileader Style on** list.

14 Click the **Create...** button.

15 **(Optional)** Edit the **LeaderProperties**.

16 Click the **Content** tab.

17 Under **Block options**, click the **Source Block list** button and select **_MemberTag** from the list.

18 Close the **Drawing Explorer** dialog box.

19 Save _SectionTag.dwg.

Link the entity type to its multileader style

1 Type SUPPORTFOLDER in the Command line, then go to the **BIM** > **Section** folder and open _TagTypeToStyle.xml.

Use an ASCII text editor, for example Notepad.

2 Add the following right before </TagTypeToStyle>:

```
<tag type="BIM_MEMBER"> <style name="_MemberTypeStyle" autoPlacement="true"
 autoRotation="true" offset="1" /> </tag>
```

3 Save the file.

4 Note that this style has three options:
  - **autoPlacement = "true"** will make sure tags are automatically generated when using the **Auto** BIMTAG option. If this option is set to **false**, this tag type will not be generated automatically (you can still generate these tags manually).
  - **autoRotation = "true"** will rotate the tag along the main axis of the tagged entity. If the option is set to **false**, the tag will be placed horizontally.
  - **offset** will offset the tags perpendicular to the main axis of the tagged entity.

26.2.6 Customize tag blocks

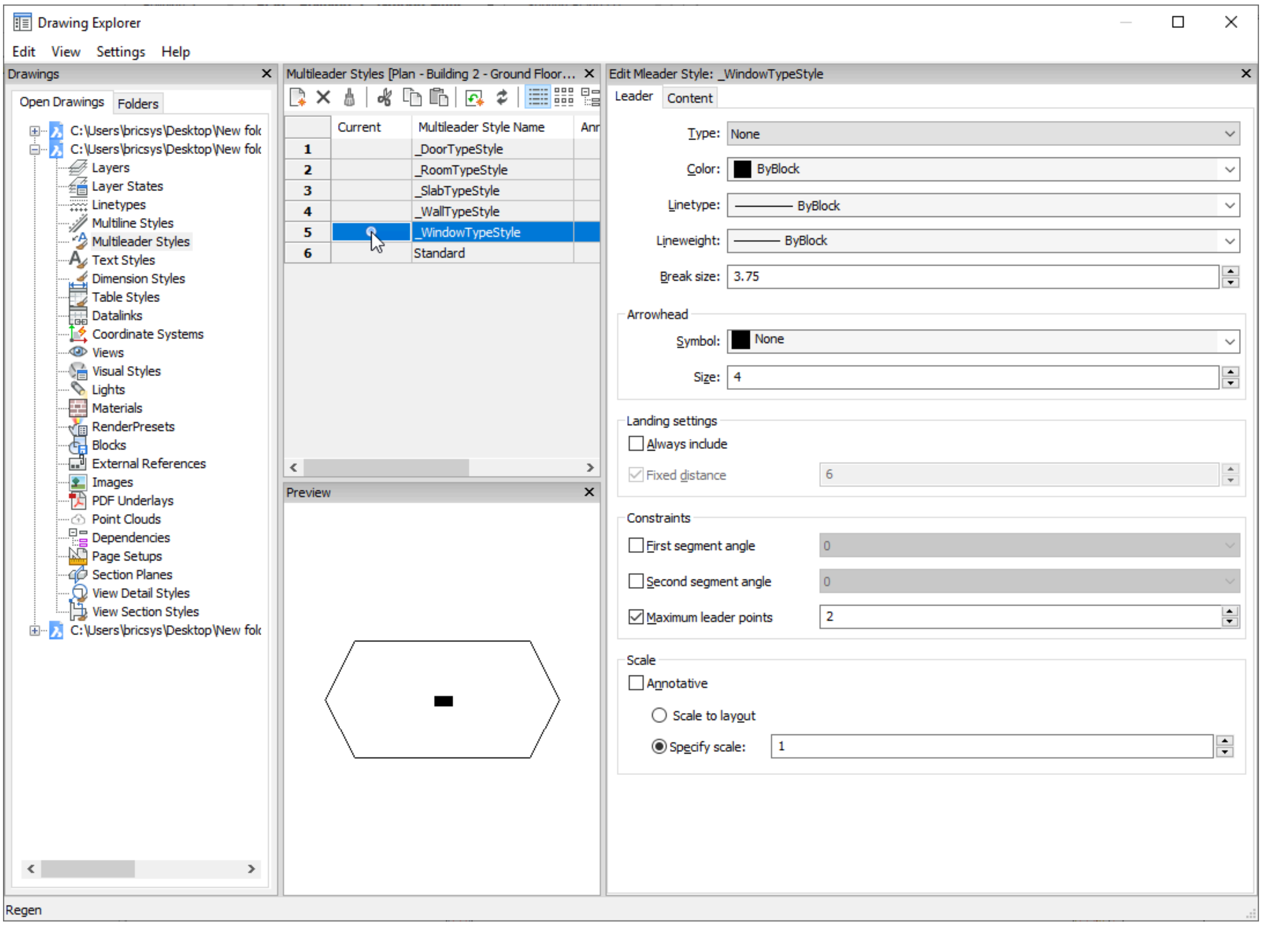
The content of a BIM Tag block consists of:

- Layout entities Linework, fixed texts, etc.
- Attributes BIM entity properties.

Note: When you execute the BIMTAG command, BricsCAD checks whether the source data (tag blocks and their multileader styles) already exist in the BIM section drawing. If not, the source data is copied from the _SectionTag.dwg file. Otherwise, the tag block definitions and multileader styles in the section drawing are used. Therefore, editing tag blocks in the _SectionTag.dwg file only affects the creation of new BIM tags the first time the BIMTAG command is executed in a section drawing. If you edit tag blocks in a section drawing, it affects BIM tags in that drawing only.

26.2.7 Customize tag blocks for the current drawing only

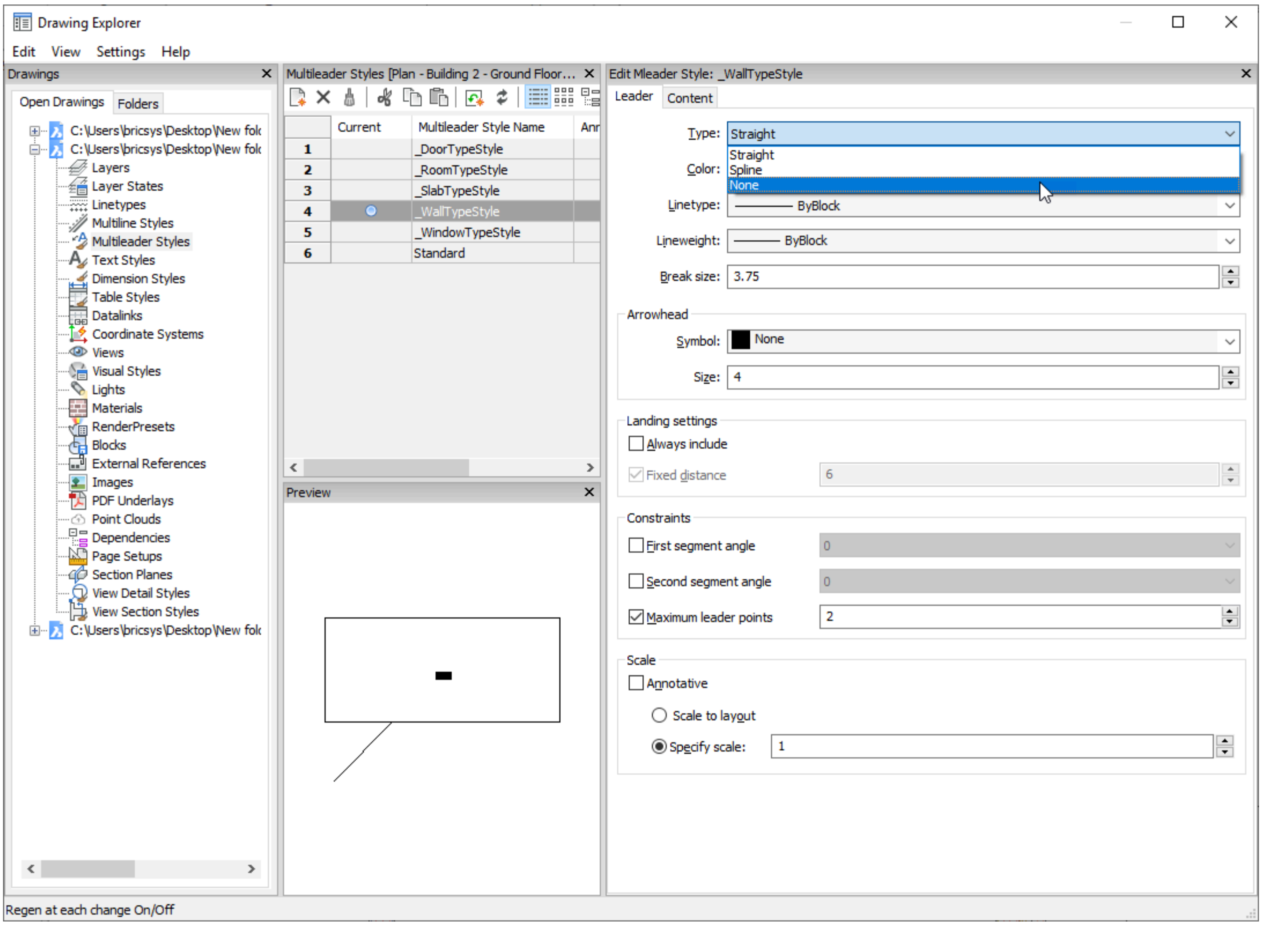
1 In the section drawing for which you wish to customize the BIM tags, type MLEADERSTYLE in the Command line. The **Drawing Explorer** will open in the **Multileader Styles** tab.



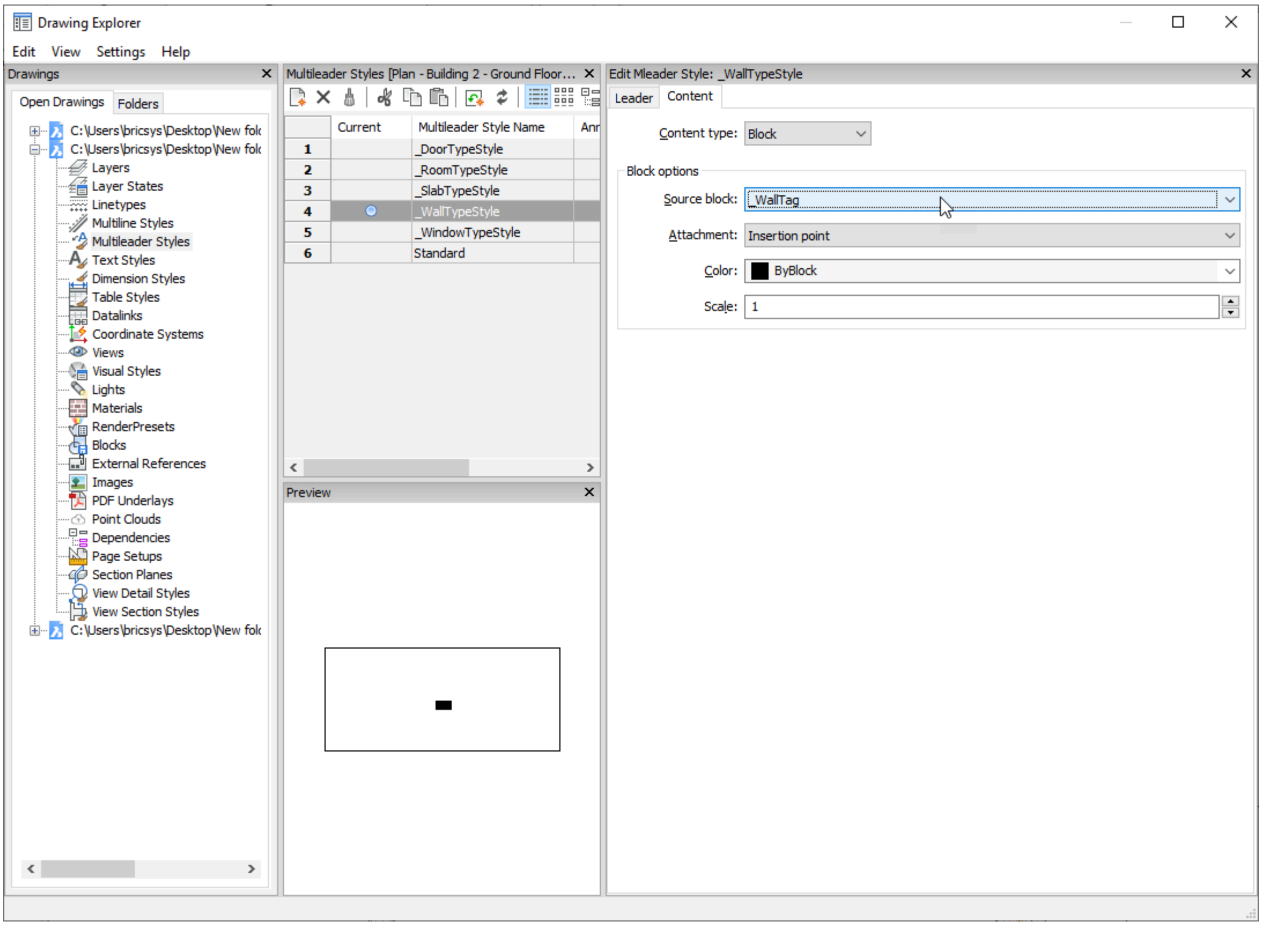
Each BIM tag is a multileader. The multileader style depends on the type of BIM element being tagged. All the multileader styles in the drawing are listed in the **Drawing Explorer**.

2 Select **_WallTypeStyle** from the list to see a preview of the tag. You can change the tag style from here.

3 Change the **Leader Type** from **Straight** to **None**. Now there will be no line connecting the tag to the wall element.



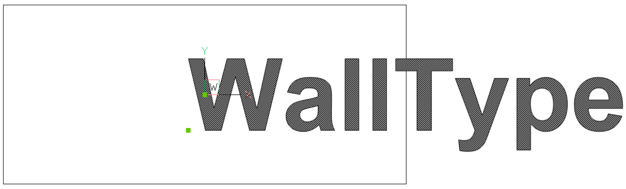
4 In the **Content** tab, the **Source block** is called **_WallTag**. Remember this and close the **Drawing Explorer**.

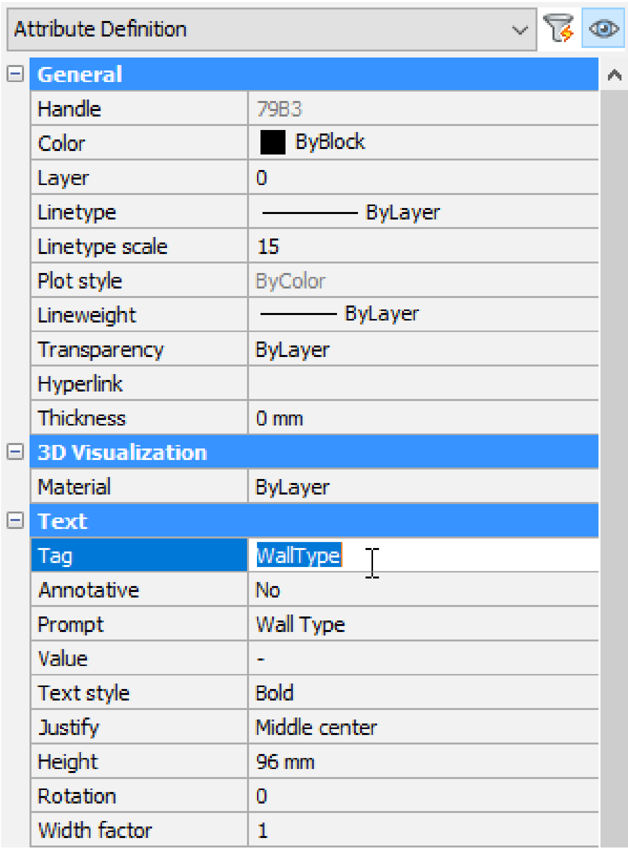


5 In the Command line, type BEDIT and select **_WallTag** from the list. Click **OK**.

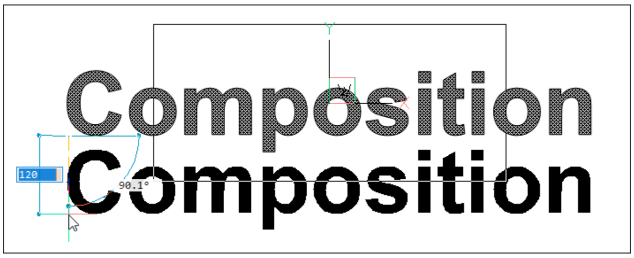
6 Select the **WallType** text and **Tag** value of the **Properties** panel.

7 Replace **WallType** with **Composition** and press **ENTER** (see the next procedure for the syntax of other property types). The attribute text will display **Composition**.





8 Hover over this attribute and select **Copy** from the Quad. Select a base point to paste the copied attribute below the current one.



9 Select the copied attribute and replace **Composition** with **Quantity/Thickness** in the **Tag** field from the **Properties** panel.

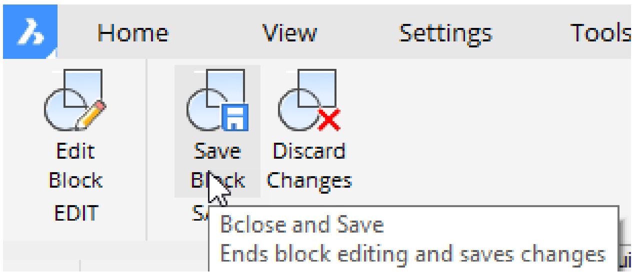
10 Delete the rectangle.

11 The two attributes will be displayed as in the following illustration:

12 Select the **Save Block** icon from the **Block Editor** tab of the ribbon, or type BCLOSE in the Command

line and select the **Save** option.

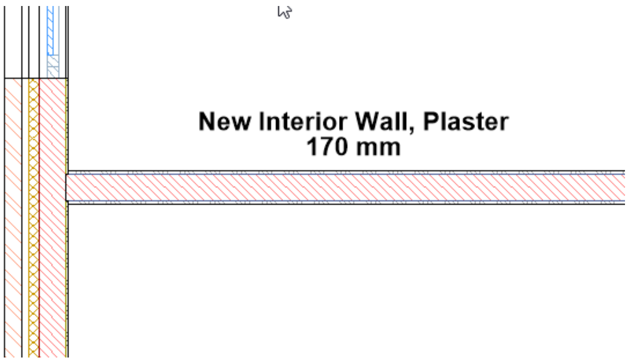
Your section viewport displays.



13 Highlight the viewport and select **Tag Assign BIM tags automatically** from the Quad.

You are prompted: Tags already have been assigned to the section. Choose an update option [Update existing/update existing and add New/Regenerate all] <Regenerate all>:

14 Press the **Enter** key to accept the default option of **Regenerate all**. The wall tags will be regenerated with the new style.



26.2.8 Edit the source data to customize BIM tags for all section drawings

1 With any drawing open, type SUPPORTFOLDER in the Command line. Press **Enter**.

2 Once open navigate to the **BIM** > **Sections** and open the _SectionTag.dwg file.

3 Launch the BEDIT command.

4 Select the tag block you want to edit, then click the **OK** button.

5 Add attributes using the following syntax, in the **Tag** field, in the **Properties** panel:

Property type	Syntax	Examples
BIM	<property name>	name room bounding
Parameters	parameters/ <parametername>	parameters/w parameters/framethickness

Property type	Syntax	Examples
General	general/<property name>	general/layer general/linetype scale
Mass	mass/<property name> mass/<property name>:<child>	mass/surface area mass/ucs elevation:minimum
Geometry	geometry/<property name> geometry/<property name>:<child>	geometry/position geometry/ucs elevation:maximum
Quantity	quantity/<property name>	quantity/length quantity/net footprint area
IFC	<ifc property set>/<ifc property>	wall common/load bearing door window glazing type/ glass layers
User Defined	<property set label>/<property label>	exterior walls/price
Classification system	<classification system label>/ <classification code label>	omniclass/omniclass classification code

Note: To know which properties are available for a specific BIM entity type, select the entity in the model and open the **Properties** panel.

In the image below, the properties of a BIM Wall entity are listed.

Click the **+** button to expand a property type node.

3D Solid

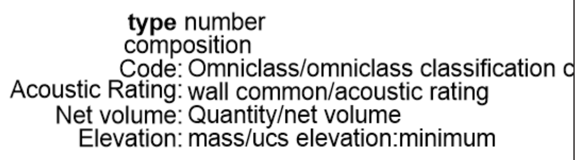
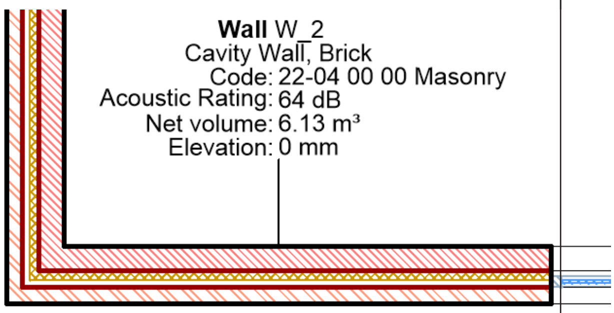
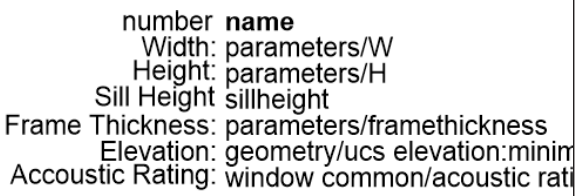
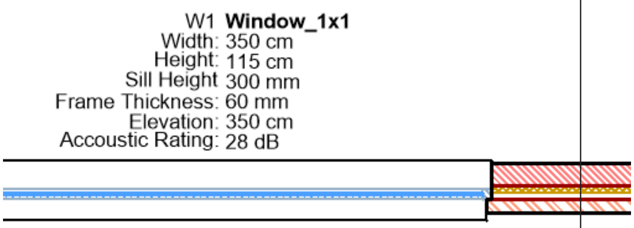
General	
3D Visualization	
Material	ByLayer
Mass	
BIM	
Type	Wall
Name	E1
Description	Ext. Wall type 1
Building	Building
Story	Floor 1
Composition	Cavity Wall, Brick
GUID	1MvYF3FSfA5gFoL$dkVfY5
Wall type	Cavity
Room bounding	On
Centerline	Off
Number	W12
Quantity	
Length	810 cm
Thickness	320 mm
Height	375 cm
Gross volume	9.34 m³
Net volume	9.34 m³
Gross side area	29.18 m²
Net side area	29.18 m²
Gross footprint area	2.49 m²
Net footprint area	2.49 m²
Plies	4
Ply	1
Wall Common	
Reference	
Acoustic rating	45dB
Fire rating	2h
Combustible	Off
Surface spread of flame	
Thermal transmittance	0 W/m²·K
Is external	On
Extend to structure	Off
Load bearing	On
Compartmentation	On
OmniClass	
OmniClass classification code	22-04 27 23 Cavity Wall Unit Masonry

6 For each new attribute do the following:

7
- Copy an existing attribute or create a new attribute (ATTDEF command).
- Select the new attribute.
- ***Note:*** Do not double click the attribute. The **Edit Attribute Definition** dialog box does not accept spaces in the **Tag** field.
- In the **Properties** panel, type the property reference in the **Tag** field (see the syntax in the above table).
- **(Optional)** Define the **Text** properties (**Text Style**, **Justify**, **Height**, …) in the **Properties** panel.

- **(Optional)** Add line work and fixed text.

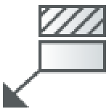
8 On the **Block Edit** toolbar, click the **Save** icon to conclude the BEDIT command, or type BCLOSE in the Command line, **Save** option.

<table>
<tr><th>Definitions in the tag block</th><th>Resulting tag</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>_WallTag block</td><td>Wall tag in a plan section</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>WindowTag block</td><td>Window tag in a plan section</td></tr>
</table>

26.2.9 Assign composition tags

Composition tags allow you to tag a wall or a slab element in order to specify its composition. These BIM tags are a list of the ply materials which make up the element.

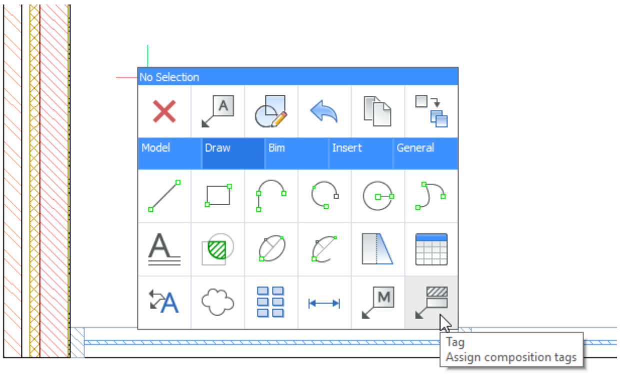
For more information about this option, visit the Command reference article **BIMTAG command**.



Create composition tags

1 Open the section drawing for which you wish to create composition tags.

2 Select **Tag Assign composition tags** from the Quad.

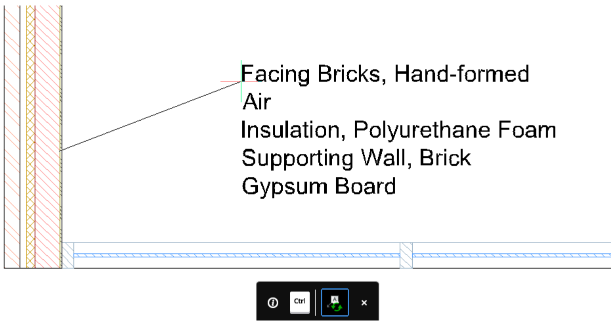


Or type BIMTAG in the Command line, then do one of the following:

- Type **M+Enter** for the **Manual** option; then **T + Enter** for the **change Tag type**, followed by **O+Enter** for the **cOmposition** option.
- Click the options in the Command line.
- Click the options in the **Prompt** Menu.

3 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

4 Select a point on a BIM element to generate a tag for it. The tag will appear. If the selected point is shared between multiple BIM entities, you can use the **Ctrl** key to cycle through the different possible composition tags.



You are prompted: Place the tag [Next element]:

5 To adjust the position of the tag, move the cursor and left click to place it.

6 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

7 Repeat steps 3 and 4 for all of the entities you wish to be tagged. If you do not require any more tags, press the **Esc** or **Enter** to complete the command.

26.3 BIM sections

26.3.1 Source files

The following source file is used to create BIM sections:

- **_SectionSettings.dwg**: contains all types of section callouts.

The file is located in this folder:

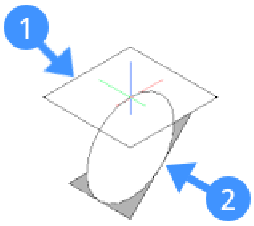
C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\Bim\Sections

26.3.2 Defining a section

In BricsCAD BIM, the **Define Section** tool allows you to create BIM section entities. The BIM section allows you to section a BIM model by a plane and view the interior details.

For more information about this command, see the article **BIMSECTION command**.

When defining a BIM section, the section tracker is attached to the cursor.



The section plane (1) lies in the XY-plane of the current coordinate system (WCS or UCS). The view direction (2) is the negative direction of the current coordinate system.

Note:

- If Dynamic UCS (DUCS) is **On**, the section plane aligns to the face of the 3D solid under the cursor. Press the **TAB** key to select an obscured face.
- Press the **Shift** key to lock a highlighted plane. This allows you to specify the definition point for the BIM section entity outside the face of the solid.
- The section line is parallel to the X-axis of the coordinate system (WCS, UCS or DUCS).

General procedure to create a BIM section

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Do one of the following:

- Click a point anywhere outside the model to create a plan section.
- Hover the cursor over the face of a 3D solid, parallel to the section plane you want to create.
- The section tracker aligns to the face under the cursor, on the condition that **DUCS** is active. Left-click.
- **(Optional)** Press the **Shift** key to lock the highlighted plane. This allows you to start from a

point outside the selected 3D solid face. Next, left-click.

3 The initial section plane displays dynamically and the 3D model is clipped accordingly.

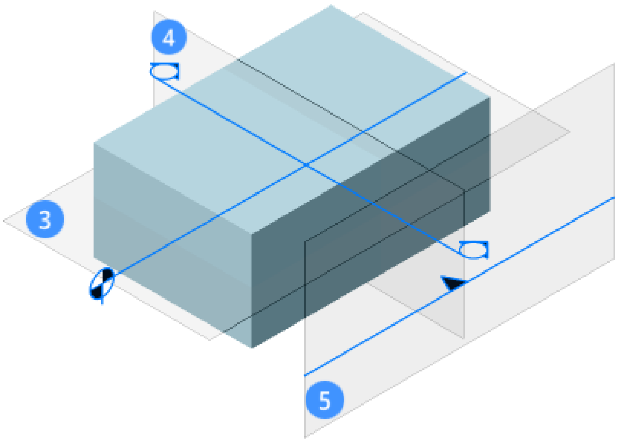
4 You are prompted: Specify distance or [set clipping oN/set clipping Off]:

Do one of the following:

- Enter a value in the dynamic dimension field to offset the section from the initial position.
- Left-click when the section plane is at the location you want it to be.

The BIM Section entity is defined.

The Section Type depends on the direction of the section plane (horizontal or vertical) and whether it intersects with the model.



Plan (3)	Displays horizontal section plane.
Section (4)	Displays a vertical section that cuts through the model.
Elevation (5)	Displays the exterior elevation view.

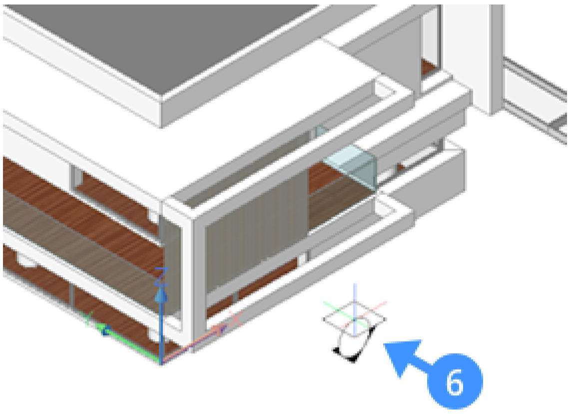
Creating a plan section

Note: A plan section is best defined in the WCS (World Coordinate System).

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

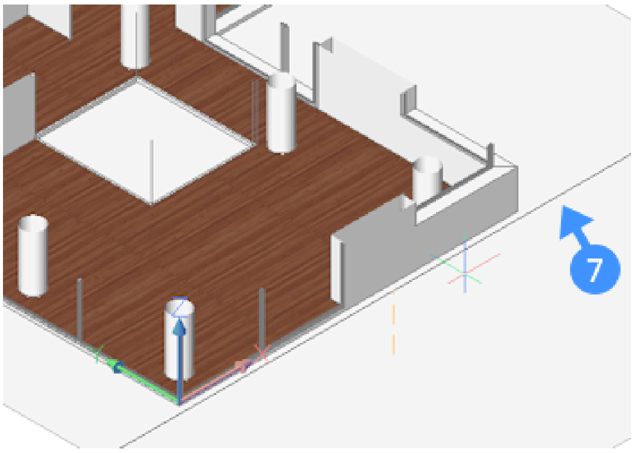
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Place the section tracker near the 3D model and click (6).

3 

The initial section plane lies in the XY-plane (Z = 0) of the current coordinate system (e.g. the WCS).

The section plane displays dynamically and the 3D model is clipped accordingly (7).



You are prompted: Specify distance or [set clipping oN/set clipping Off]:

4 If Dynamic Dimension is **On**, the current distance displays in a dynamic dimension field.

Do one of the following:

- Move the cursor and click when the section plane is at the desired height.
- Use an Entity Snap to specify a point.
- Type a value to define the offset from the initial position of the section plane.

The plan section is defined.

Creating a cross section

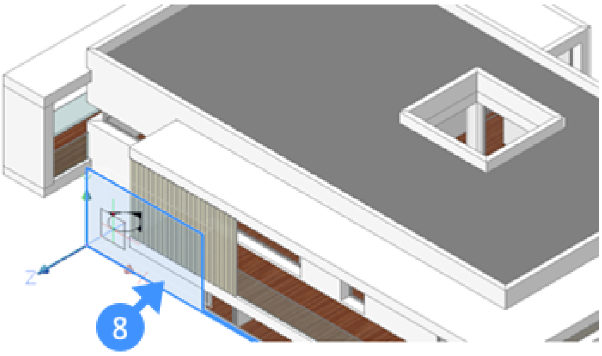
Make sure Dynamic UCS (**DUCS**) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

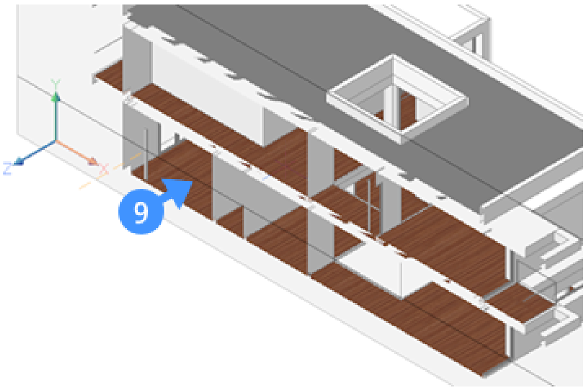
2 Place the section tracker over a 3D solid face (8) parallel to the cross-section to be defined.

3



4 Click when the face highlights and move the section tracker inside the 3D model.

5 The section plane displays dynamically and the 3D model is clipped accordingly (9).



You are prompted: Specify distance or [set clipping oN/set clipping Off]:

If Dynamic Dimension is **On**, the current distance displays in a dynamic dimension field.

6 Click when the section plane is at the desired location or specify a distance.

7 The cross-section is defined.

8 **(Optional)** Click the section entity and hold down the left mouse key a little longer to display the **Manipulator**. Drag an axis of the manipulator to move the section.

9 **(Optional)** Click the Arrow grip to flip the view direction.

Creating an elevation

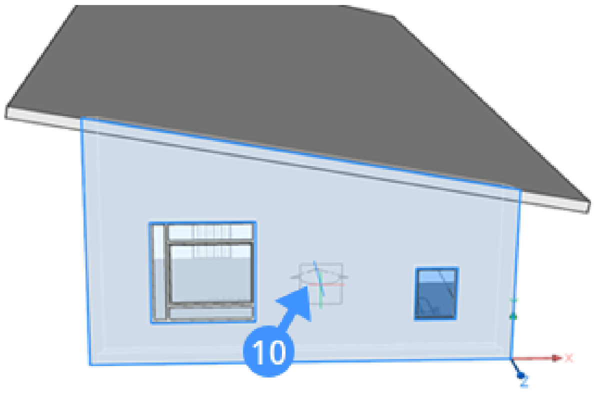
Make sure Dynamic UCS (**DUCS**) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

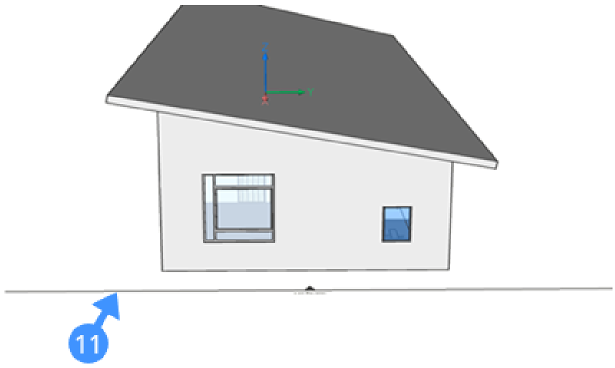
2 Place the section tracker over a 3D solid face (10) which is parallel to the elevation to be defined and click when face highlights.

3



4 You are prompted: Specify distance or [set clipping oN/set clipping Off]:

5 Move the section tracker outside the 3D model and click to define the elevation (11).



Vertical sections outside the model (no geometry is sectioned) get the Elevation section type automatically.

26.3.3 Editing the default properties of a BIM section

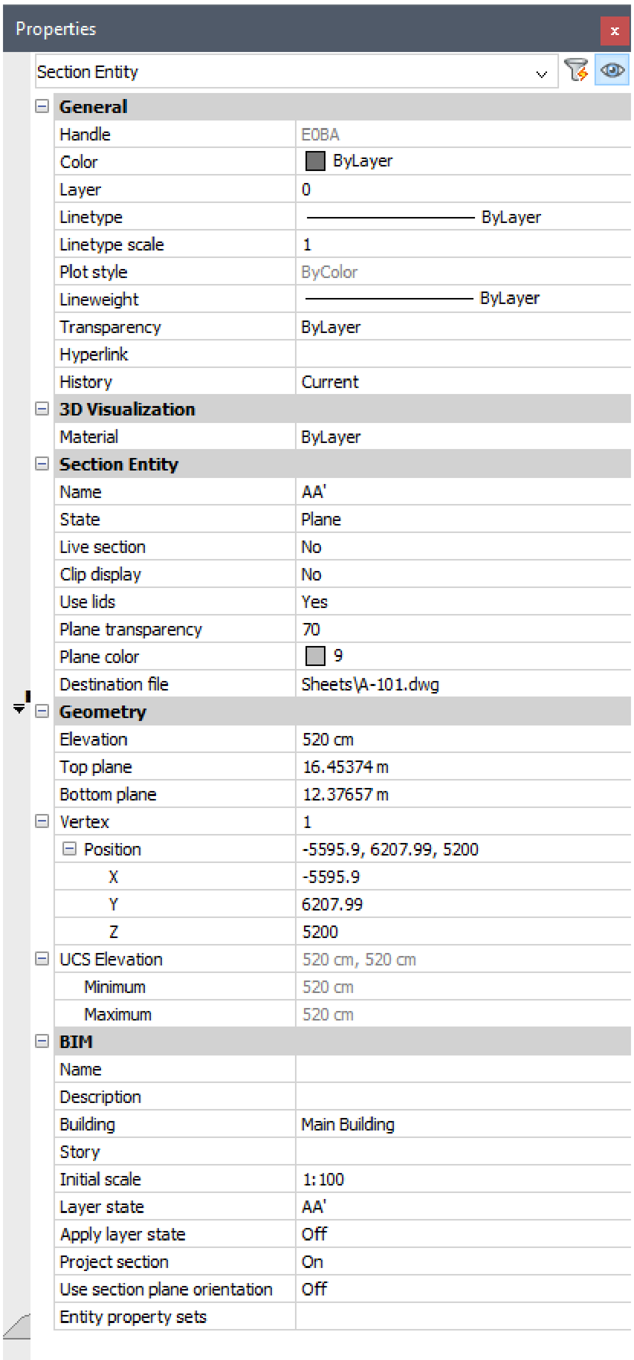
The default properties of a BIM section entity are copied from the section entities in _SectionSettings.dwg in the **BIM/Sections** subfolder of the **Support** folder.

1 Select the BIM section in the drawing or the **Structure** panel.

The section properties display in the **Properties** panel.

2 Define the properties of the BIM section entity.

3



Section Entity:

Name	Type a name.
State	Click the field and select a state. The selected state determines how the section result will be clipped. The options are: **Plane** (default), **Boundary** and **Volume**.

Live section	Sets the live section property of the section plane. Only one section entity can have the **Live section** property **On**. It is recommended to use the **Clip display** property instead. Use this mode for visual checking only, not for 3D modeling.
Clip display	Clips the 3D model. The **Clip display** status can be set for multiple section entities simultaneously. As different from **Live section**, entity snaps are available in the section plane.
Use lids	Closes the top faces of the clipped geometry.
Plane transparency	Sets the transparency of the section plane in rendered Visual Styles.
Plane color	Sets the fill color of the section plane in rendered Visual Styles.
Destination file	Sets the name and path of the drawing file created by the BIMSECTIONUPDATE command. When the path is not specified, the section drawing is created in the folder of the 3D model.

Geometry:

Elevation	Defines the Z-value of the start point of the section plane.
Top plane	Defines the top plane of the section volume.
Bottom plane	Defines the bottom plane of the section volume.
Vertex	Defines to coordinates of the control points of the section plane. The number of vertices depends on the current section plane state. Plane: 2 vertices Boundary and Volume: 4 vertices. Select the field, then click the arrow buttons to select a vertex. An X indicates the selected vertex in the model.
UCS Elevation	Defines the minimum and maximum UCS elevations.

BIM:

Name	Type a BIM name.
Description	Type a description.

Building	Sets a Building name.
Story	Sets a story name.
Initial scale	Sets the initial scale of the paperspace viewport in the generated drawing.
Layer state	Sets the layer state to be used to calculate the section.
Apply layer state	Determines whether the layer state will be used to calculate the section or not. Applying a layer state allows to control which layers are **On/Off** when calculating the section result.
Project section	Controls the display of the section callouts and section line in section drawings which are perpendicular to this section.
Use section plane orientation	Controls whether section plane orientation is used or not.
Entity property sets	Controls whether 'per instance' entity properties can be used to generate tags. Click the **Browse** ( ) button, then select the property sets in the **Enable per-instance property set** dialog box.

26.3.4 How to toggle the clipped display property of a BIM section entity

The **Clip display** property of the **BIM** section temporarily clips 3D models. The **Clip display** can be used in all modeling and editing operations on all drawing content.

The following illustration shows the 3D model of a villa with a plan section (16) for the ground floor plan:

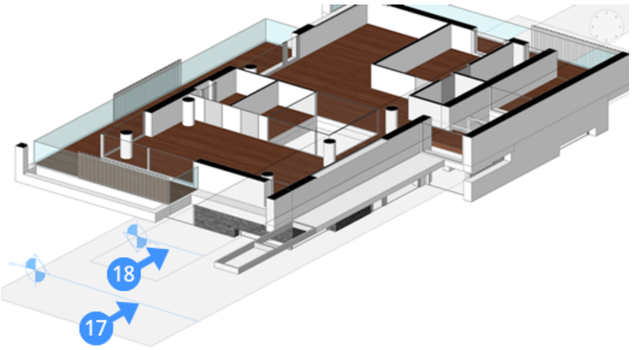
To clip the 3D model, do one of the following:

- Double click the section entity.
- Select **Clip display** from the Quad.

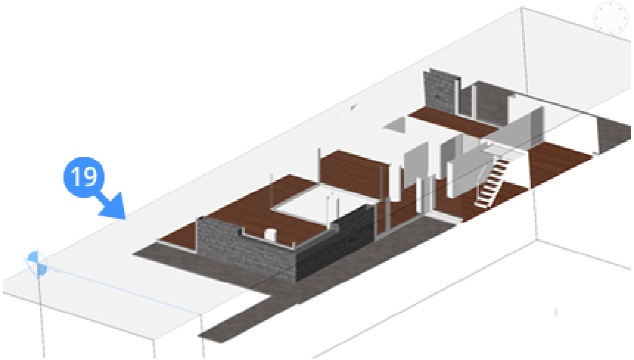
- Select the section plane and set the **Clip display** property to **Yes** in the **Properties** panel.
- When **RT** (Rollover Tips) is selected in the Status Bar, place the cursor over a section entity to set the **Clip display** property.

Note: **Clip display** can be on for multiple section entities simultaneously.

In the following illustration, **Clip display** is turned **On** for two BIM section entities (17,18). These section entities are in opposite view directions: section plane 18 is a plan section, section plane 17 is a reflected ceiling section.



Option: Set the **State** property for the current section to **Volume** in the **Properties** panel.



A clipped section shown in the **Volume** state (19).

26.3.5 Converting a section entity to a BIM section

The BIMIFY command converts a section entity to a BIM section.

1 Launch the BIMIFY command.

1 You are prompted: Apply or [change Selection]:

2 The **Bimify** command panel opens

3 Choose **Select entities in drawing**, then click the **Select** button ( ).

4 Select the section entity, then press **Enter**.

5 Click the **Apply** button in the **Bimify** command panel.

26.4 Define detailed section

26.4.1 Command

BIMSECTION

26.4.2 Define a detailed section from scratch

1 Select **Detail Section** from the ribbon or use the Command line by entering BIMSECTION followed by **D** to choose the detail option.

You are prompted: Select the first corner of the section box or [Based on an existing section/Scale]:

2 Select the first corner of the area you wish to be contained within the detailed section.

3 **You are prompted**: Select the opposite corner of section box:

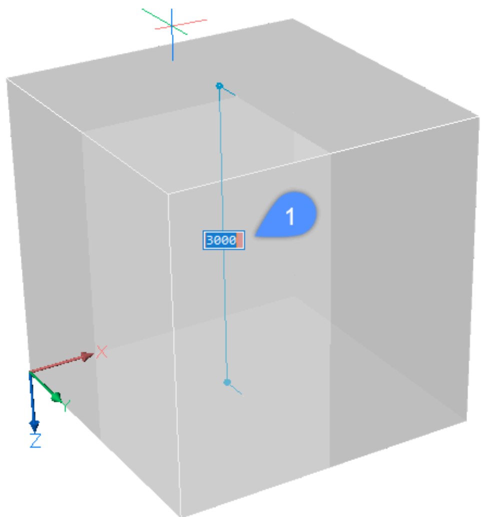
4 Select the opposite corner snap to a point or enter the distances. Use the TAB key to switch between dynamic dimension fields.

5 **You are prompted**: Select the height of volume box <4000>:

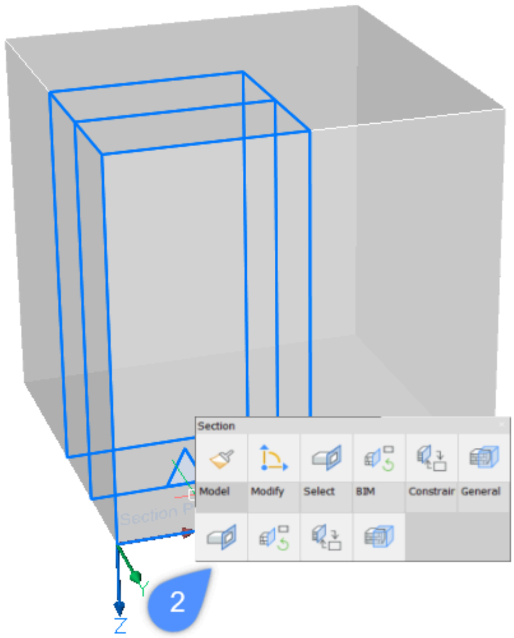
In this case, the default height is 4000mm.

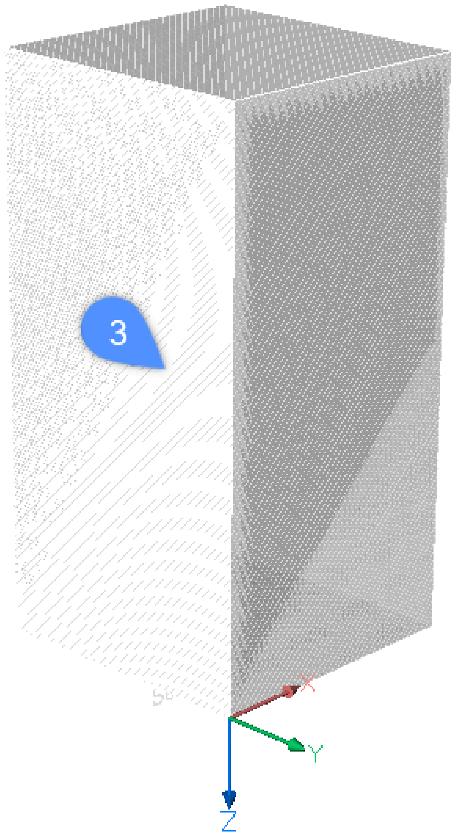
6 To complete the volume, do one of the following:

7
- Press Enter to accept the default height.
- Specify a point.
- Type a value in the dynamic dimension field (1).
-



To view the detailed section which has just been defined, hover over the section and select **Clip display** (2) from the Quad or double-click the selected section. Now only the detailed section will be visible (3).

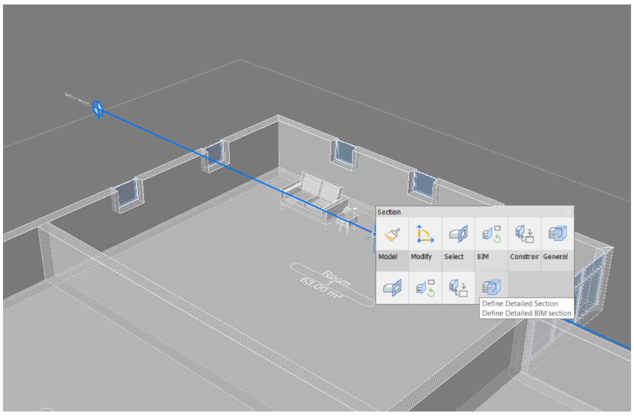

Section
Model
Modify
Select
BIM
Constrair
General
Y
Z
2


3
X
Y
Z

26.4.3 Define a detailed section based on an existing section

Here you will be shown how to position the base of your detailed section on an already existing section.

1 Highlight the existing section that you want the detailed section base to be positioned on and select **Detail Section** from the Quad.

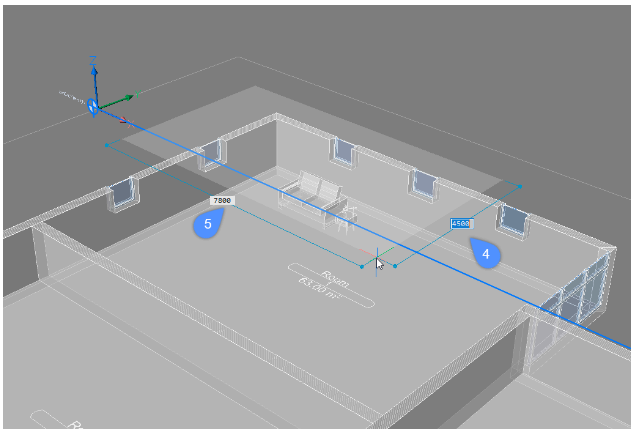


You are prompted: Select first corner of volume box base rectangle:

2 Draw a rectangle around the desired area by typing the values in the dynamic dimension fields.

3 Press the TAB key to go back and forth between the dynamic dimension fields (4, 5). Your selection will be aligned with the plane of the existing section.

4 Press Enter to confirm the values. The **volume section** box displays dynamically.



You are prompted: Select the height of the volume section box.

5 Press Enter to accept the default height.

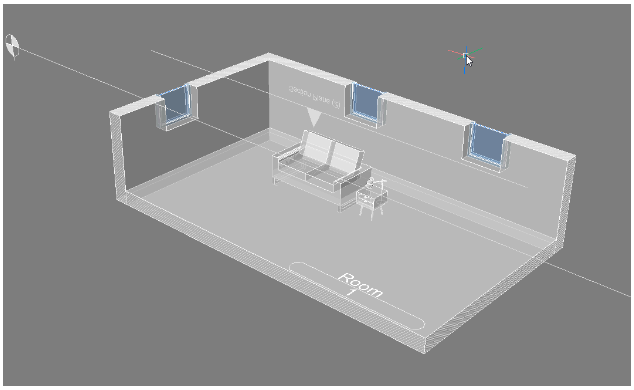
6 Specify a point.

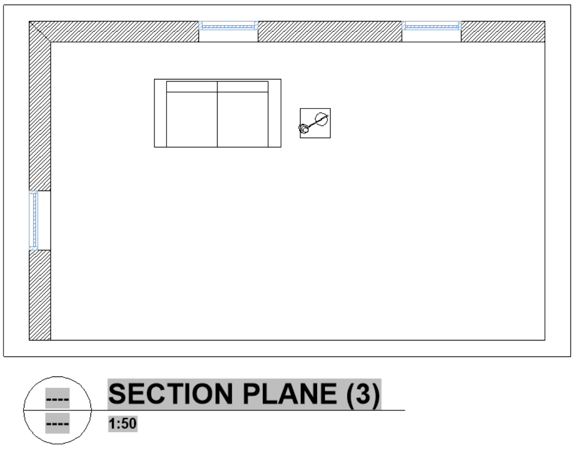
7 Type a value in the dynamic dimension field.

8 The detailed section has now been defined and can be found in the **Structure** panel. Hover over the section and select **Clip display** from the Quad or double-click the selected section to view it.

9 To generate the detailed section on a sheet, select the detailed section and select **Generate Section** from the Quad.

10 You can open the section results by clicking **Open Model** when the section plane(s) are selected. The sheet is saved in the same folder containing the model drawing.





If you need more help with how to generate drawings, visit the **Generating drawings** page.

If you want to annotate your drawings, see how to add tags and linear dimensions.

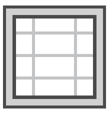
26.5 Define reflected ceiling plan

26.5.1 Command

BIMSECTION

26.5.2 About

Reflected ceiling plans show a mirror image of the ceiling. These are useful to show the layout of lights and other installations in the ceiling. To produce the reflected ceiling plan a section is created. This section displays the part of the building above the section boundary.



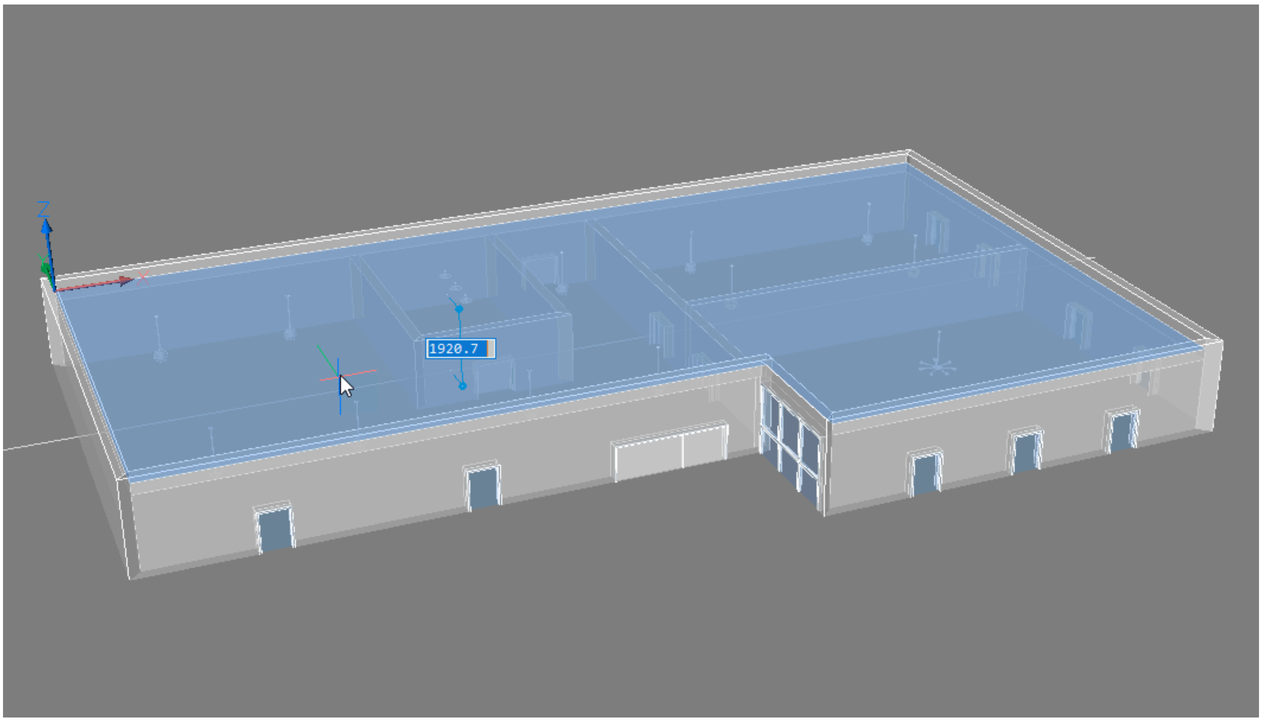
For more information about this command, visit the Command Reference article BIMSECTION.

26.5.3 Create a reflected ceiling plan

1 Select **Reflected Ceiling Plan** from the ribbon **or** enter BIMSECTION into the Command line followed by **R** to choose the **Reflected ceiling** option.

You are prompted: Select a point to place section:

2 Select the ceiling or a face with a plane parallel to that of the ceiling.

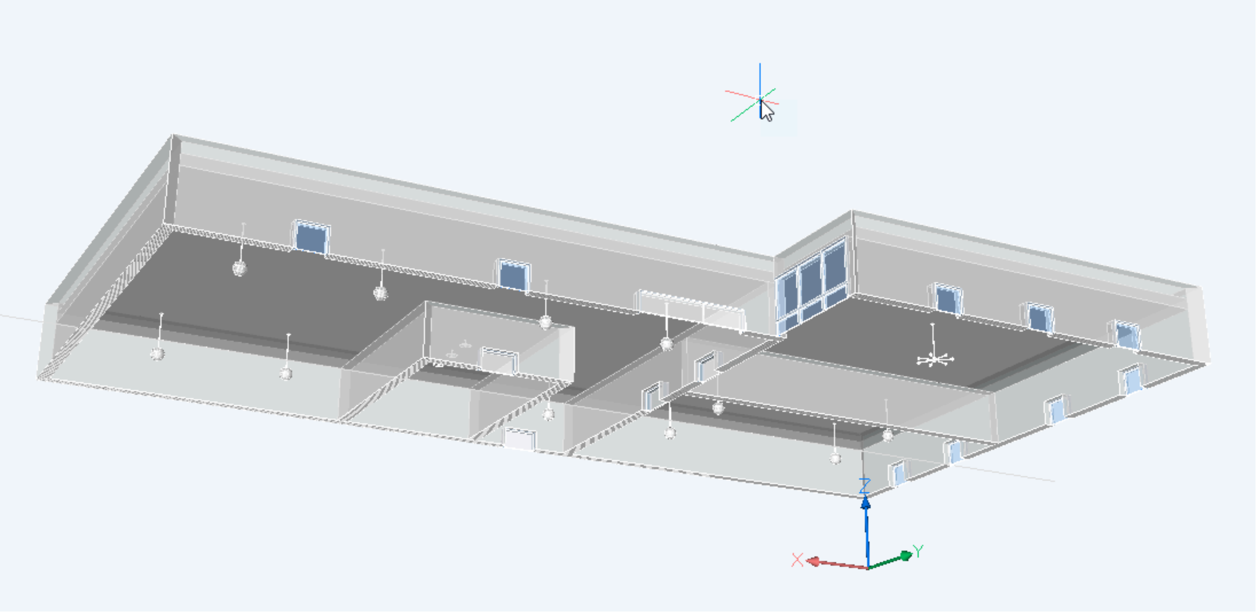


You are prompted: Specify distance:

3 Do one of the following:

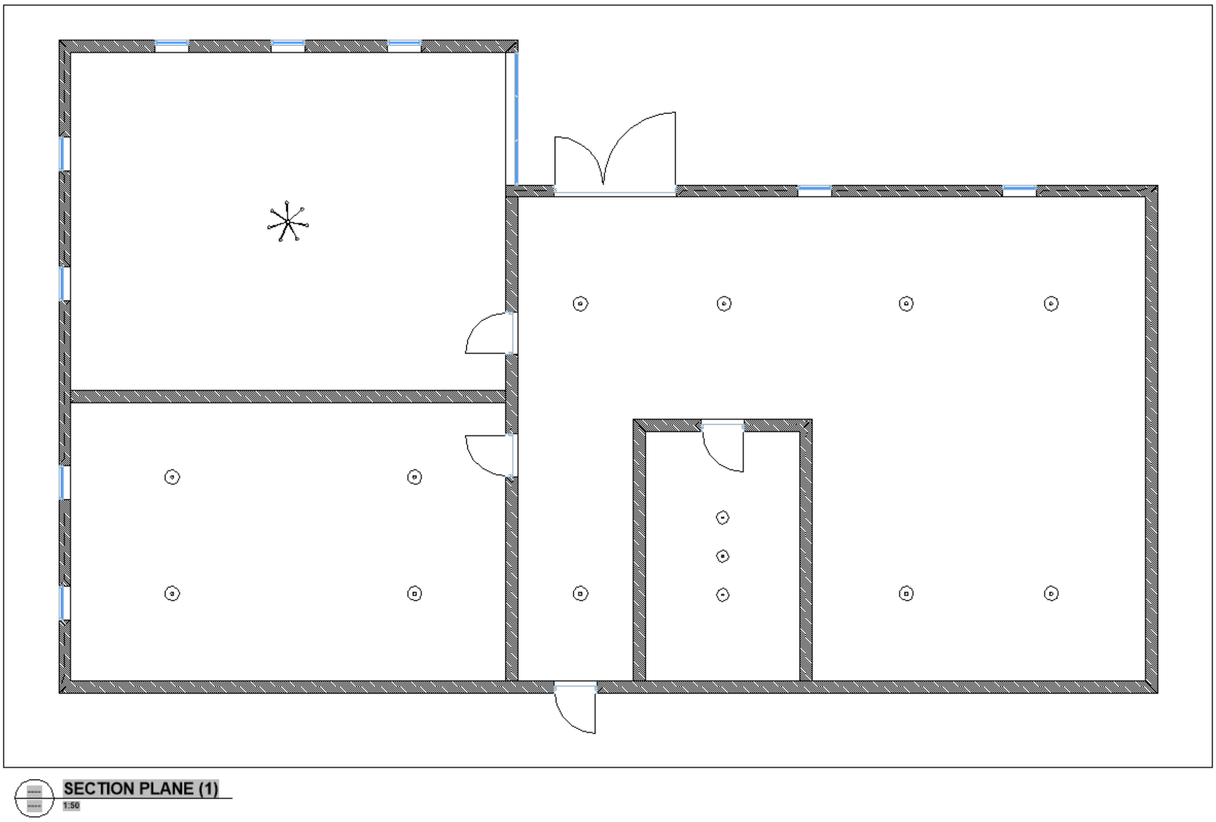
- Type a value in the dynamic dimension field to offset the section from the initial position.
- Left-click when the section plane is at the location you want.

4 The reflected ceiling section has now been defined. To view hover over the section and select **Clip display** from the Quad.



5 To generate the reflected ceiling plan on a sheet, hover over the section in the model and select **Generate Section** from the Quad.

6 To open the section result drawing click **Open Model**. The sheet is saved to the same folder that contains the model drawing.



If you need more help with how to generate drawings, visit the Generating Drawings page.
If you want to annotate your drawings, see how to add tags and linear dimensions.

26.6 Elevation hatch

26.6.1 About elevation hatch

Generate elevation fill is a property of a section that can display the texture or solid hatch of a composition when a 2D elevation is generated. Hatches can also be generated for other types of sections when activated in the Properties panel.

The hatch is generated for every visible boundary on the 2D elevation including boundaries that are not perpendicular to the section. Hatches are generated without perspective corrections. The union of elevation lines can be changed independently from the **Union section** settings in the **Physical Materials** dialog box.

26.6.2 Make elevations

Note: **Elevation** here refers to a Section Plane in a 3D model with the **Section type** property set to **Elevation**. This can then be used to generate 2D drawings that represent the elevations of your model.

1 Create BIM elevations using **Bimify**.

2 Select the BIM elevation and choose **Generate BIM Section** in the **Model** command tab in the Quad.

3 Select the BIM elevation and choose **Open BIM Section Model** in the **Model** command tab in the Quad.

The section result displays in the layout viewport of the destination drawing.

4 Select the paper space viewport.

The viewport properties display in the Properties panel.

5 Under BIM, set the **Generate Elevation Fill** property to **On**.

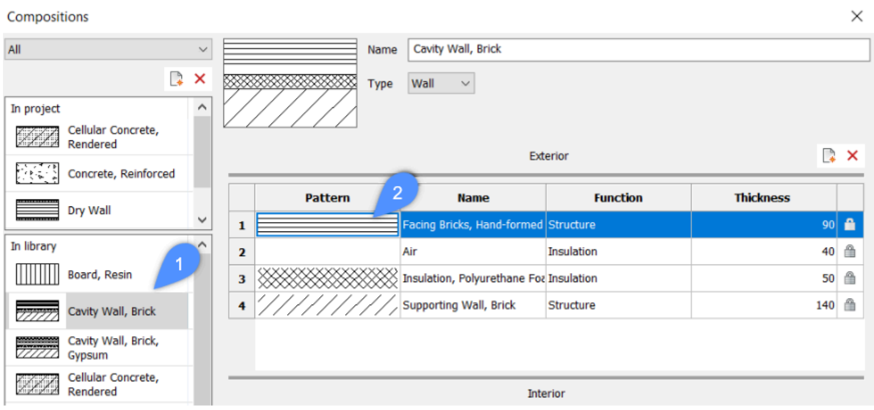
6 Select the viewport and choose **Update BIM Section** in the **Model** command tab in the Quad.

7 **(Optional)** Copy the viewport, then set the **Generate Elevation Fill** property to **Off** to create two versions of the section result, one with and one without elevation hatch.

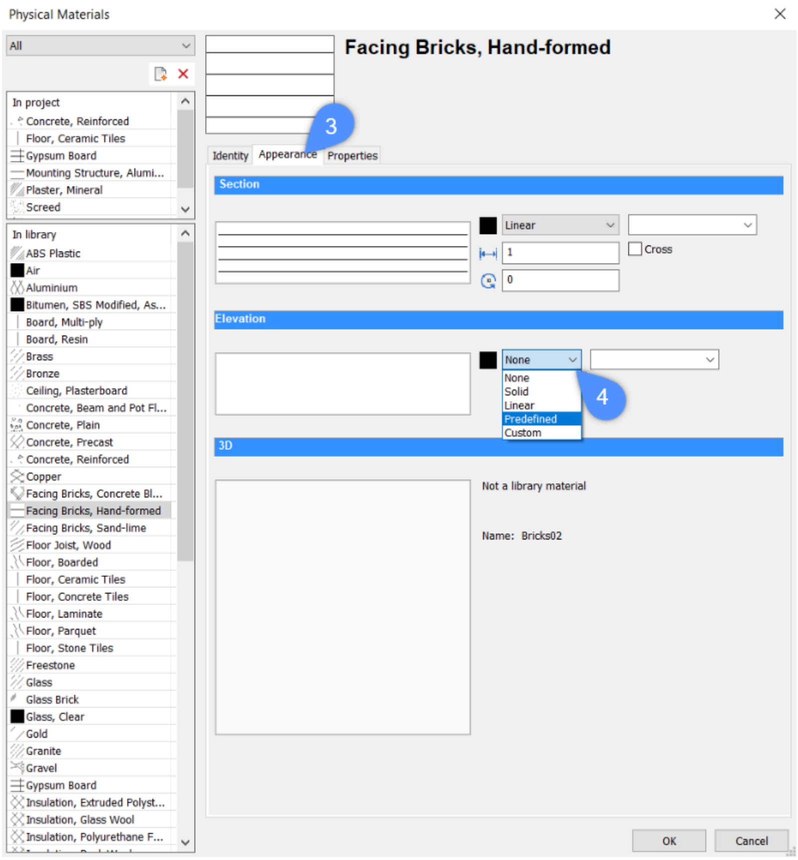
26.6.3 Configuring the hatch

1 Go to the **Physical Materials** dialog box by typing BLMATERIALS or BLCOMPOSITIONS (1) in the Command line.

2 Click on the composition and double click on the pattern of the composition to open the **Physical Materials** dialog (2).

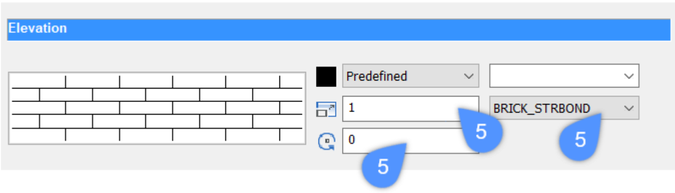


3 To configure the texture or solid fill, go to the **Appearance** tab page (3). The elevation hatch is set by default to **None**. Click the elevation hatch button and select the of hatch: **None**, **Solid**, **Linear**, **Predefined** or **Custom** (4).



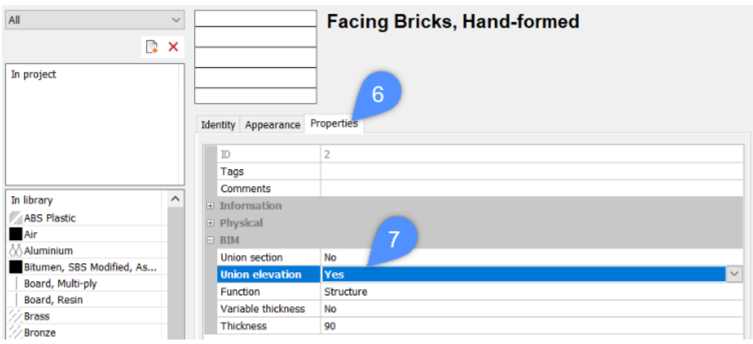
4 Configure the hatch. Define the scale, angle, color and/or layer of the pattern. (5)

Note: A name in the layer field or click the arrow button and select a layer in the list.

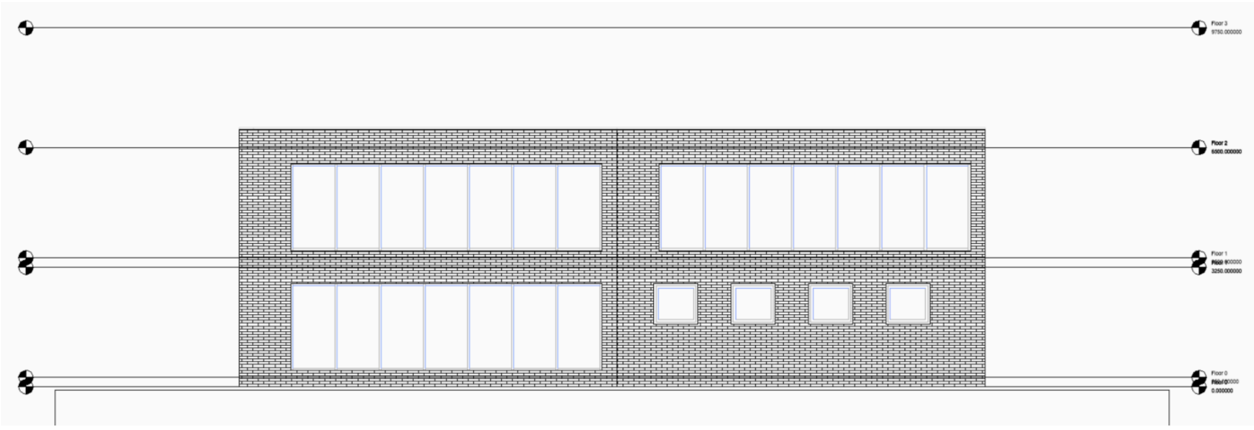


5 To remove connection lines in the elevation between different solids of the material, go to the **Properties** tab (6) and set **Union elevation** to **Yes** (7).

Note: **Union elevation** works independently from the **Union section** setting.



6 After regeneration, the hatch will appear on the elevation.



26.7 Templates for generated drawings

26.7.1 About

BricsCAD BIM contains a series of templates to customize your generated drawings. These templates are stored in your **Support** folder in the following path:

C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\Bim\Sections

The **AppData** folder is a hidden folder by default, you may have to unhide folders first. This path can also be accessed with the SUPPORTFOLDER command.

To understand how it works, you must know how BricsCAD creates layers, hatches, tags, and call-out blocks.

26.7.2 Understanding the created layers

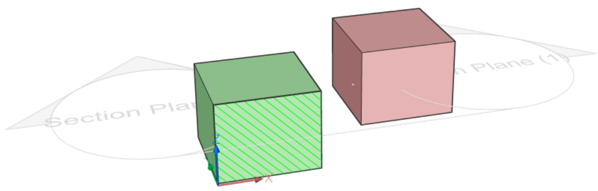
1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

2 Draw two boxes of 500x500x400, one a bit behind the other.

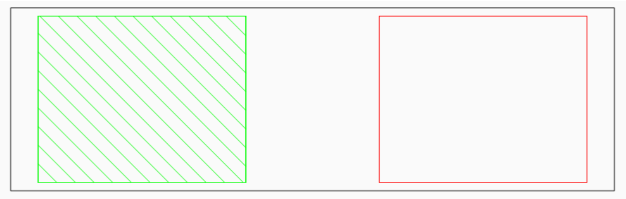
3 Put the first box on the layer **Layer1** (with layer color green) and the second box on the layer **Layer2** (with layer color red).

4 Create a BIM section, aligned with the front face of the first, green box and cutting through it in the middle, but not cutting through the second, red box.

5



6 Generate and open the section drawing.

7 

8 Take a look at the **Layers** panel: three groups of layers have been created: Background, Boundary, and Fill.

	Current	Name	D..	On/Off	Freeze	Locked	Color
1		0					White
2		Background_Layer1					Green
3		Background_Layer2					Red
4		BIM_SECTIONS_PREVIOUS					Magenta
5		BIM_VIEWPORTS					White
6		Boundary_Layer1					Green
7		Defpoints					White
8		Fill_Layer1					Green
9		Title Block Frame					175
10		Title Block Labels					157
11		Title Block Text					251

9 The generated layers for the green solid are:
- **Boundary_Layer1**: the boundary of the section through the box.
- **Fill_Layer1**: the hatching of the section.
- **Background_Layer1**: shows subentities (e.g., vertices of solids) that are not sectioned.

10 The layers all inherit the color of the **Layer1** layer.

11 Since the red solid is not sectioned, only the **Background_Layer2** layer is created. It inherits the color of the **Layer2** layer.

12 Drag the **Cavity Wall, Brick** composition from the **Compositions** panel onto the green box.

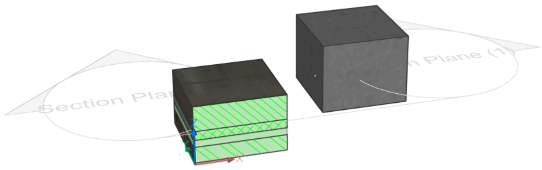
Note: Set the LEVELOFDETAIL system variable to 1 or 2, to display the ply materials.

Note: The composition always gets priority when it is assigned to a solid that is placed on a colored layer. This means that the colors set by default for that composition's layers have priority, not the color assigned to the solid's layer.

13 Double-click the **Cavity Wall, Brick** composition in the **Compositions** panel.

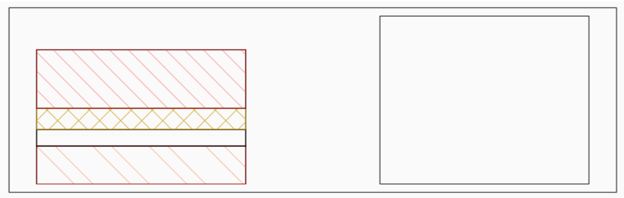
This composition consists of 4 plies, each with its own material. A physical material is associated with a hatch pattern, used for floor plans and sections. Another pattern is specified for elevations.

14 Assign the **Concrete, Reinforced** composition to the red box.



15 Update and open the section.

16



17 Go to your layers and execute the PURGE command to remove the unused layers.

18

	Current	Name	D..	On/Off	Freeze	Locked	Color
1		0					White
2		Background_Air					White
3		Background_Concrete_ Reinforced					White
4		Background_Facing Bricks_ Hand-forn					White
5		Background_Insulation_ Polyurethane					White
6		Background_Layer1					Green
7		Background_Layer2					Red
8		Background_Supporting Wall_ Brick					White
9		BIM_SECTIONS_PREVIOUS					Magenta
10		BIM_VIEWPORTS					White
11		Boundary_Air					White
12		Boundary_Cavity Wall_ Brick					White
13		Boundary_Facing Bricks_ Hand-formed					14
14		Boundary_Insulation_ Polyurethane Fc					42
15		Boundary_Layer1					Green
16		Boundary_Supporting Wall_ Brick					14
17		Fill_Facing Bricks_ Hand-formed					21
18		Fill_Insulation_ Polyurethane Foam					42
19		Fill_Layer1					Green
20		Fill_Supporting Wall_ Brick					11
21		Title Block Frame					175
22		Title Block Labels					157
23		Title Block Text					251

You see the three categories (Background, Boundary, and Fill) again. The second part of the layer name is the name of the physical material used in the solid, instead of the layer name of the box:

- **Background_Facing Bricks_Hand-formed**, **Background_Insulation_Polyurethane Foam**, and **Background_Supporting Wall_Brick**.
- **Boundary_Facing Bricks_Hand-formed**, **Boundary_Insulation_Polyurethane Foam**, and **Boundary_Supporting Wall_Brick**.
- **Fill_Facing Bricks_Hand-formed**, **Fill_Insulation_Polyurethane Foam**, and **Fill_Supporting**

Wall_Brick.

19 Set the **Generate elevation fill** property of the section **ON** to add an elevation hatch.

20 Update the section.

21 The outline of the second box is now hatched and the layer **Elevation_Concrete_Reinforced** is added.

26.7.3 Understanding the BRX_2D layers

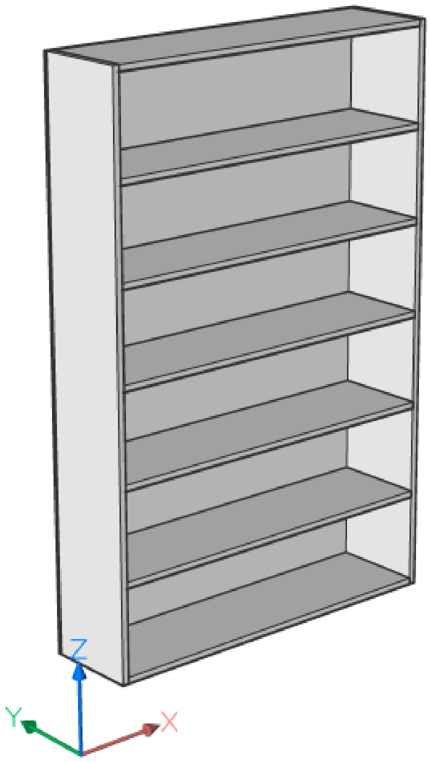
The 2D symbols will either replace the actual 3D Solid section of the component or be added to the 3D Solid section, depending on their layer:

- Symbols on a layer with prefix **BRX_2D_** or **BRX_2D+_** will be added to the section block:
  - Without being processed by analytical hidden line removal (HLR) when the section plane intersects the components.
  - It is not possible to have this geometry occluded by some other geometry. E.g. you have a table and chairs. Both have a 2D symbol, but the table is below the section plane and the chairs are cut by it (back rest is higher than table). Then it is not possible to have the table top (**BRX_2D_** of table) hide parts of the chairs underneath.
  - After going through analytical hidden line removal (HLR) procedure when the section plane does not intersect the components.
- Symbols on a layer with prefix **BIM_2D_SECT_** or **BIM_2D_SECT+_** will be added only when the section plane intersects the components. All geometry goes through analytical hidden line removal (HLR) procedure. Can be used to have a different representation of the component when it is sliced, e.g. for stairs.
- Symbols on a layer with prefix **BIM_2D_BACK_** or **BIM_2D_BACK+_** always will be added and all geometry goes through analytical hidden line removal (HLR) procedure. Makes it possible to solve the table-chair problem mentioned above.
- Symbols layers name
  - with +: the 2D symbol will be added to the 3D solid section of the component.
  - without +: the 2D symbols will replace the actual 3D Solid section of the component.

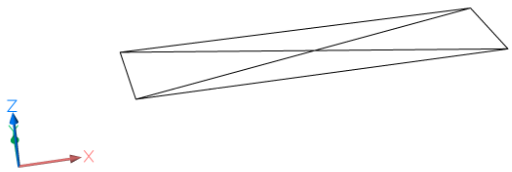
Adding 2D entities

1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

2 Drag the component **Furniture Bookshelf** into the drawing. You can find the component in the **Furnishing Elements** category in the **Library** panel (LIBRARYPANELOPEN command).

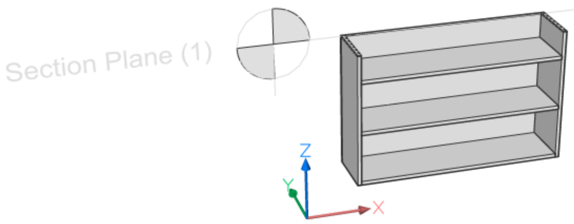


3 If you now show and isolate the **BRX_2D_Symbols** layer, you will see a symbol drawn at the insertion point of the bookshelf.

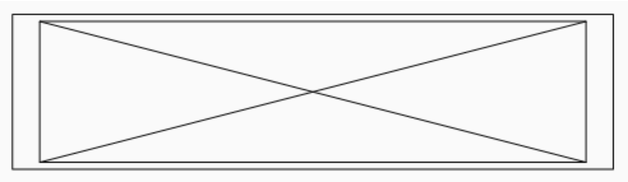


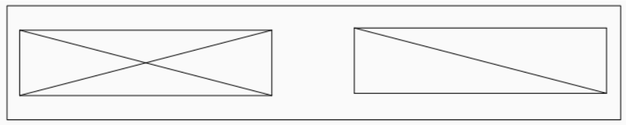
4 Show all layers again (LAYUNISO command) and make a section through the bookshelf.

5

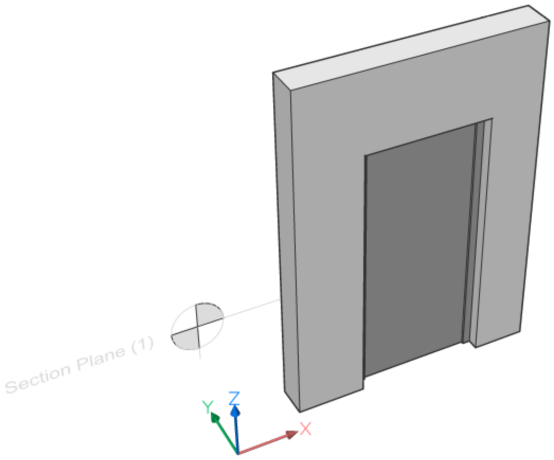


6 Generate and open the **Plans** drawing.



7 You will see (if the **Boundary_BRX_2D_Symbols** layer is on) that only the symbol is drawn onto the plan, and not the real section. It happens because this component had a 2D symbol on the **BRX_2D** layer attached to it.
8 Now return to your 3D model and open a copy of the bookshelf.
9 Hide the bookshelf itself and make sure the **BRX_2D_Symbols** layer is on.
10 Delete one of the lines (of the cross).
11 **(Optional)** Draw extra 2D lines on the **BRX_2D_Symbols** layer.
12 Save the drawing.
13 Go back to the original 3D model and type in BMINSERT in the Command line and go to the file path where you just saved the new bookshelf. Click **OK** and choose a point to insert the bookshelf.
14 Regenerate the section and open it. Widen the viewport if necessary, by dragging the existing viewport.
15 

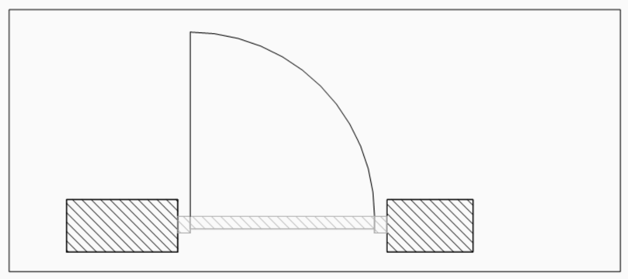
16 The second bookshelf has the adapted symbol displayed.
17 Draw a wall in your 3D model and insert the door **Door Ext Plain** into this wall.



18 Repeat steps 3 to 5 but now use the **BRX_2D+_Swing** layer instead of the **BRX_2D_Symbols** layer.
19 The generated plan displays the symbol of the door swing as well as the door section itself. The **BRX_2D+** layer of a component makes sure you can show the real section together with a symbol.
   - Symbols on a layer with prefix **BRX_2D_** will be used to replace the geometry resulting from the

actual section.

- Symbols on a layer with prefix **BRX_2D+_** will be added to the section result. Only those 2D entities in a plane parallel to the section plane are considered.



26.7.4 Customizing the colors and line weights of hatch patterns and lines

1 In the above procedure **Understanding the created layers**, you saw that for objects without a composition, the colors and line weights of the **Fill_**, **Background_**, and **Boundary_** layers are copied from the layer properties in the 3D model.

2 However, for objects with a composition, the default colors are set by a template found by typing in SUPPORTFOLDER command and browsing to *Bim/Sections/_SectionSettings.dwg*.

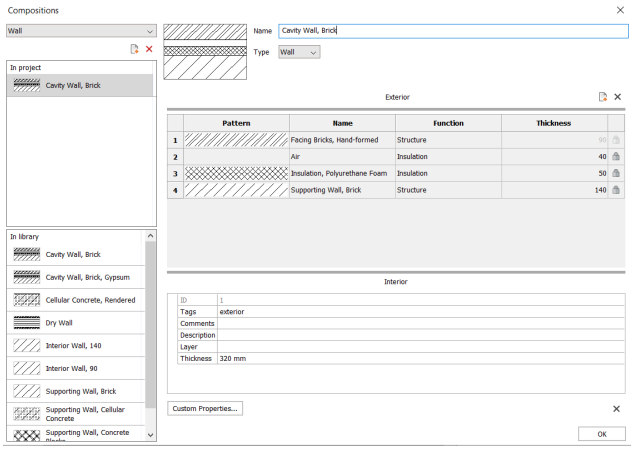
3 Open this drawing.

4 Go to the **Layers** panel, you will find a list of all the possible layers that can be created for every material present in the material library. They all have a color and line weight. Changing the color or line weight and saving the file will change the default color/line weight for this layer in your future generated drawings. You can fully customize how your drawing will look for every type of line (**Fill**, **Background**, **Boundary**, or **Elevation**) per material.

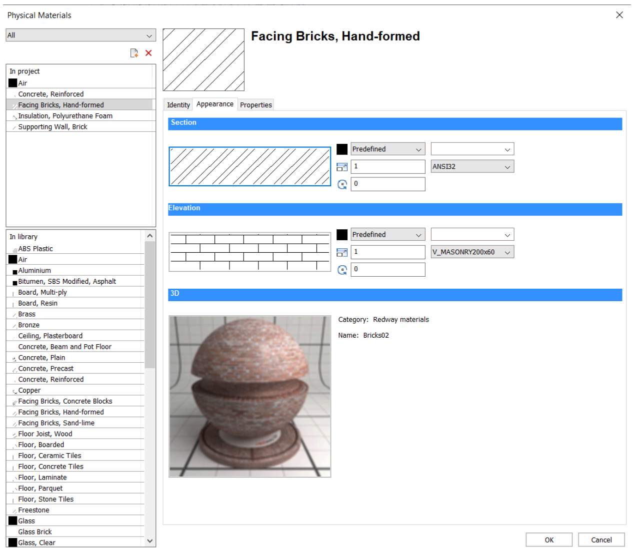
195		Fill_Ceiling_ Plasterboard					253
196		Fill_Concrete_ Beam and Pot Floor					85
197		Fill_Concrete_ Plain					85
198		Fill_Concrete_ Precast					85
199		Fill_Concrete_ Reinforced					85
200		Fill_Copper					151
201		Fill_Facing Bricks_ Concrete Blocks					21
202		Fill_Facing Bricks_ Hand-formed					21
203		Fill_Facing Bricks_ Sand-lime					21
204		Fill_Floor Joist_ Wood					33
205		Fill_Floor_ Boarded					9
206		Fill_Floor_ Ceramic Tiles					251
207		Fill_Floor_ Concrete Tiles					251
208		Fill_Floor_ Laminate					251
209		Fill_Floor_ Parquet					251
210		Fill_Floor_ Stone Tiles					43
211		Fill_Freestone					43
212		Fill_Glass					141
213		Fill_Glass Brick					141
214		Fill_Gold					151
215		Fill_Granite					43
216		Fill_Gravel					251
217		Fill_Gypsum Board					251
218		Fill_Insulation_ Extruded Polystyrene					42
219		Fill_Insulation_ Glass Wool					42
220		Fill_Insulation_ Polyurethane Foam					42
221		Fill_Insulation_ Rock Wool					42
222		Fill_Iron					251
223		Fill_Lead					151
224		Fill_Marble					43

26.7.5 Customizing the hatch patterns

1. Open the drawing made in the above procedure 'Understanding the created layers'.
2. Open the **Compositions** panel.
3. Double-click the composition **Cavity Wall, Brick**.
4. In the procedure **Understanding the created layers**, it was stated that you can change the hatch patterns of the materials here. How you need to do this will follow in the next steps.



5 Double-click on the row of **Facing Bricks, Hand-formed**. The **Physical Materials** dialog will pop-up on the material **Facing Bricks, Hand-formed**. You could have entered this dialog box by typing in the command BLMATERIALS, after which you could have selected the **Facing Bricks, Hand-formed** material.

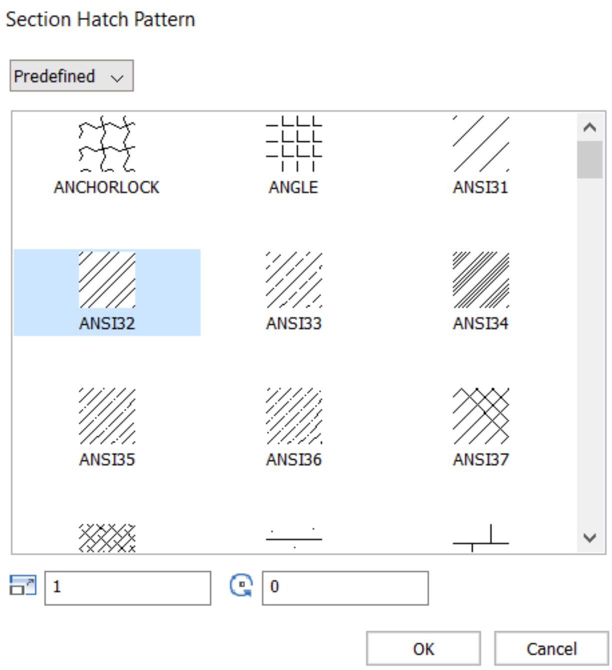


6 Go to the tab **Appearance**.

7 Under the title **Section**, double-click on the hatch pattern.

8 Choose your new standard section hatch representation for this particular material. In every composition with this material, the same hatch representation will be used.

9



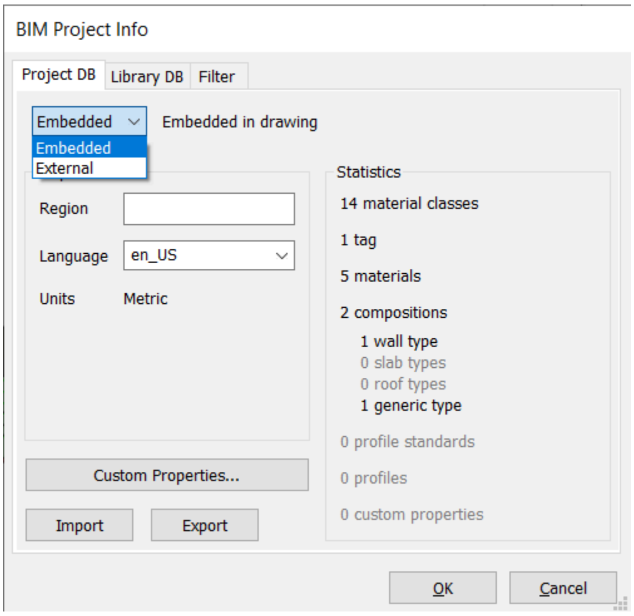
10 If you want to change the appearance of an elevation hatch, you can do so by double-clicking the hatch pattern under the **Elevation** title. The outer most visible material of a composition will be used to determine the elevation hatch.

11 Note that the representation of these hatches will not be copied into new drawings as the **Physical Materials** dialog is not a template, but rather a way to define the representation in a specific drawing.

If you would like to store this information, it is recommended to create a library of compositions. The best way to do so is to make a drawing where you attach the compositions you would like to store to little cubes. Make sure each composition has the hatch appearance you want by going through the process described above.

12 Once this is finished, type in BIMPROJECTINFO in the Command line. You will enter the **BIM Project Info** dialog box.

13 In the **Project DB** tab, switch the value for the database from **Embedded** to **External**. You will be asked to save the database on your computer and give it a name, for example, **CompositionsTemplate.bsyslib**.



14 To use this database (Project DB) in a new drawing, switch the value for the database from **Embedded** to **External**, then select the BSYSLIB file you created.

26.7.6 Customizing tags per entity type for all section drawings

For this procedure, please visit the following procedures from the **BIM Tags** guide:

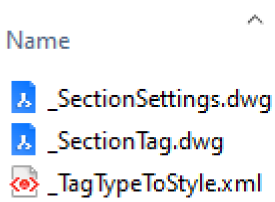
- Create the source data for a BIM entity type.
- Customize tag blocks.
- Edit the source data to customize BIM Tags for all section drawings.

To change the tags in the current drawing, visit the procedure **Customize tag blocks for the current drawing only**.

26.7.7 Customizing call-out blocks

1 Source files of the various call-out blocks and the section tracker exist in the ..*\Support\Bim\Sections* subfolder of the **Roamable Root** folder.

By default this is: *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\Bim\Sections*. You can access this file by typing SUPPORTFOLDER in the Command line and browsing to **Bim** > **Sections**.



2 You can edit **_SectionSettings.dwg** file to customize the layout of the call-outs and the section tracker blocks.

3 When a drawing is generated, the needed call-out blocks are created from the call-out blocks source file. When a BIM section is updated, the existing call-out blocks in the target drawing are used (type BEDIT to choose one to edit).

26.7.8 Customizing the sheet set template

1 The sheet set templates are stored in the *C:\Users\%username%\AppData\Local\Bricsys\BricsCAD\V25x64\en_US\Templates* folder.

2 If you want to edit an existing template, you have two options. Either you edit it directly, or you make a copy.

- To edit it directly, you can go to **Open** under the BricsCAD icon, choose **.dwt** in the **Files of type** drop-down list and choose the file you want to edit. You can then make the modifications and save the file.
- To make a copy, double-click on the DWT file you want to copy. A new DWG file with this template will open. Make your modifications and save your file as DWT. Save it in the templates folder to easily use the copied template in a new drawing.

3 If you want to start from scratch, using a wizard, follow the procedure **Creating a drawing using a wizard**. After the creation and modification of the DWG file, save it as a DWT file in your sheet set templates folder.

4 If you have saved the template DWT to the sheet set templates folder, it appears when you use the **New Wizard** to open a new drawing. You can select it as the template for creating generated drawings in the sheet set set-up of the **Project Browser**.

5 What can you store in a template?

6 You can make a title block for your template, with sheet set properties, by following the procedure **Using sheet set properties in a title block**. Do this for every layout in your template. You can also store the following things by using them in your DWT file:

- Layers
- Colors
- Line types
- Hatch patterns
- Blocks
- Text styles
- Dimension styles
- Page set-ups

26.8 Generating drawings

26.8.1 About BIMSECTIONUPDATE

The BIMSECTIONUPDATE command generates section drawings from BIM section entities. The command can be used either by selecting a BIM Section entity in the 3D model or by selecting an existing section in a generated drawing. The result is placed in a new drawing, saved to the same folder as the model drawing unless a different folder is specified in the **Destination File** property of the BIM Section entity.

For more information about this command, visit the article **BIMSECTIONUPDATE command**.

26.8.2 Procedure: generating BIM sections

BIM Sections can be generated in 2 ways:

- In the model space.
- Using the BIM **Project Browser**.

Model space

1. Select one or more BIM section entities.
2. **(Optional)** Edit the section properties (e.g. **Scale**, **Name**, **Destination File**).
3. Choose **Generate Section** in the **Model** command tab of the Quad.

BricsCAD reports the number of generated sections and their name(s) in the Command line e.g.:

8 section(s) were generated:

D:\Bricsys Reports\V25\Villa\Elevations.dwg

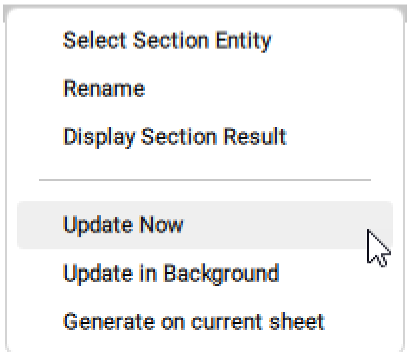
D:\Bricsys Reports\V25\Villa\Plans.dwg

D:\Bricsys Reports\V25\Villa\Sections.dwg

In the example above 2 plans, 4 elevations, and 2 cross-sections were generated and exported to 3 drawings.

Project browser

1. In the **Sections and views** tab of the **Project Browser** panel, right-click the section.
2. **(Optional)** Choose **Select Section Entity** to change its properties in the **Properties** panel.
3. Choose **Update Now** or **Update in Background**.



4. Choose **Refresh project** in the **Project Browser** panel.

   A new section is generated in the **Sheets** segment of the **Project Browser**.

   Note:

   - **BIM_SECTIONS_PREVIOUS** layer: the result of the previous section generation is stored on the dedicated **BIM_SECTIONS_PREVIOUS** layer. This is frozen by default.

- **BIM_VIEWPORTS** layer: generated viewports are placed on the dedicated **BIM_VIEWPORTS** layer if desired.
- The **SheetSet manager** automatically and silently adjusts the file paths after the project folder has been relocated if the same folder is used for all files.

26.8.3 Open BIM sections

BIMSECTIONOPEN navigates between the model drawing containing a BIM Section entity and the corresponding generated drawing. The command works bi-directionally.

For more information about this command, visit the article **BIMSECTIONOPEN command**.

BIM Sections can be opened in different ways:

- From the 3D Model.
- Using the BIM **Project Browser**.

From the 3D model

To open a 2D document from the 3D model:

1 Select one or more BIM sections.

2 Choose **Open Model** in the **Model** command group of the Quad.

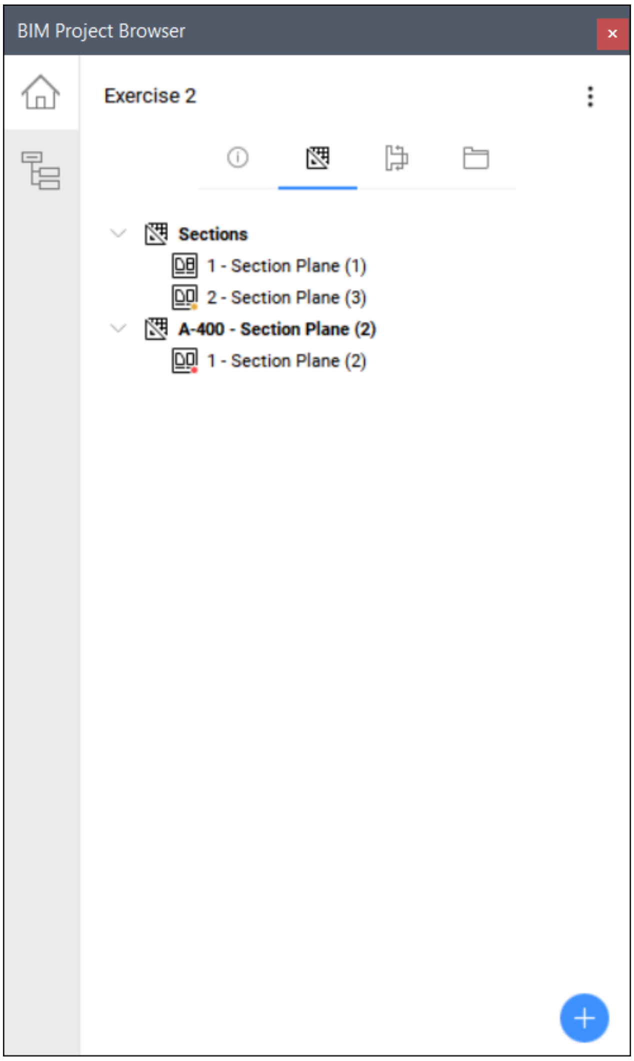
3 The 2D section drawing(s) open(s).

Project browser

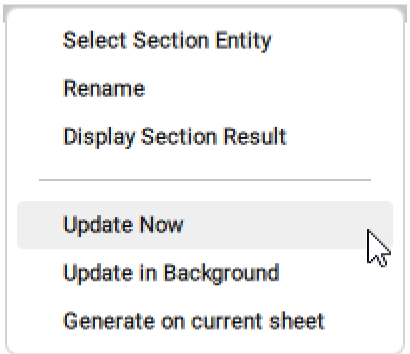
To open the 2D section drawings from the **Project Browser**, do one of the following:

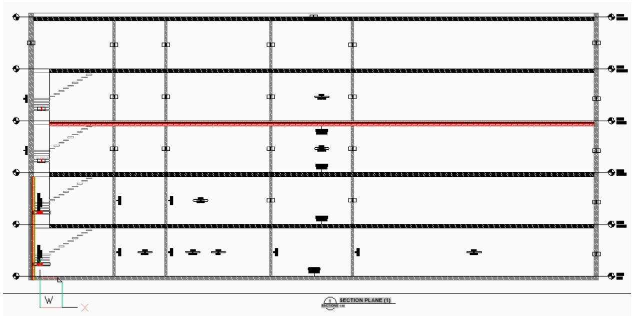
- Double click the sheet view in the **Sheets** tab of the **Project Browser** panel.

  The 2D section drawing opens with the selected sheet view.



- In the **Sections and views** tab of the **Project Browser** panel, right-click the section and choose **Display Section Result** from the context menu.





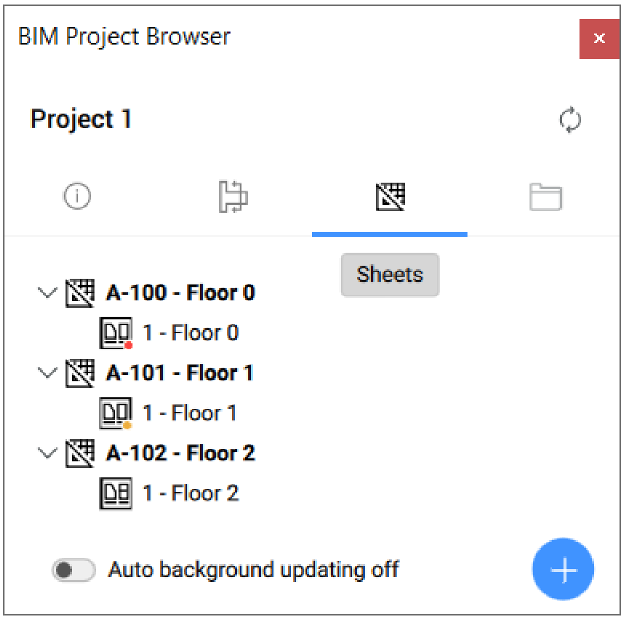
Note: When a viewport linked to a section result is deleted, the linked view label, the sheet view and the result block (only when no other viewport views it) are also erased.

26.8.4 Update BIM sections

BIM Sections can be updated in different ways:

- In the 3D model.
- In the generated 2D documentation file.
- In the **BIM Project Browser** panel.

When the 2D documentation is no longer in sync with the 3D model, the corresponding sheet views display a red mark in the **Sheets** tab of the **Project Browser** panel.



The sheet views display an orange mark in the **Sheets** tab of the **Project Browser** panel if they are not yet updated.

Update BIM sections in your 3D model

To update BIM Sections via your 3D model using the same process as used to generate views.

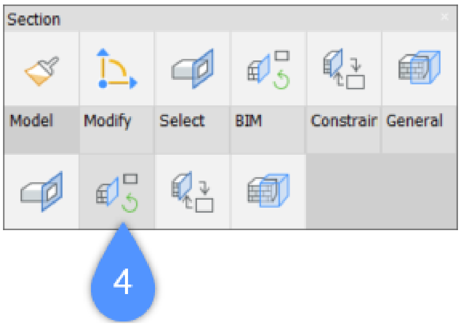
Update BIM sections in the 2D documentation file

A red mark of the section result indicates that the BIM Section generated block is no longer in sync with the 3D model and needs to be updated. This can be done in **Model** space or in the **Layout**.

Update a BIM section in model space

1 Place the cursor over the section block to update.

2 Choose **Update Section** in the **Model** command group in the Quad (4).

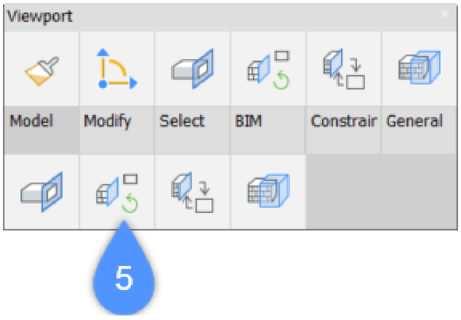


Update a BIM section in a layout

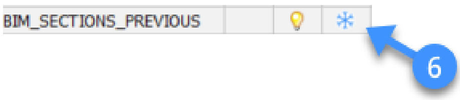
1 Click the **Layout** tab.

2 Place the cursor over the viewport of the BIM Section to update.

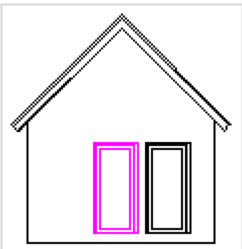
3 Choose **Update Section** in the **Model** command group of the Quad (5).



The result of the previous section generation is stored on the dedicated **BIM_SECTIONS_PREVIOUS** layer. Thaw the layer (6) to check the modifications.



The content of the **BIM_SECTIONS_PREVIOUS** layer displays in magenta.



Customize the BIM Section result

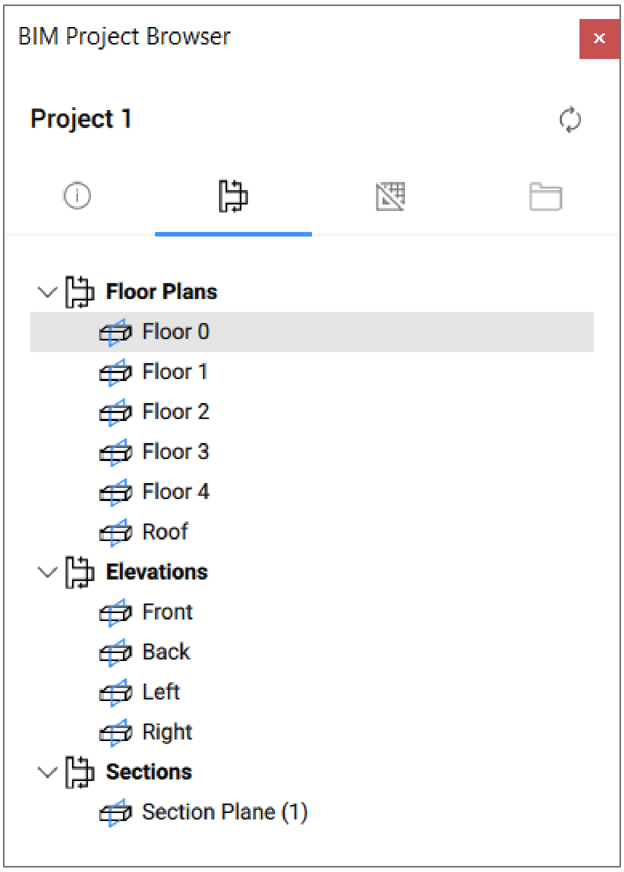
You can customize a BIM section result by assigning it a typed plan with the desired settings. To do this, select the section block, go to the **BIM** properties category in the **Properties** panel and choose one option from the **Typed plan** property's drop-down menu.

Through typed plans, the graphics of model entities can be customized independent of their assigned layers or layer states. You can also add a background image, to generate shadows on elevations or to enrich the graphical quality of the section. The background image is created in the active visual style of the model.

For more information, see the articles **BIM Typed Plan Editor dialog box** and **Working with Typed Plans**.

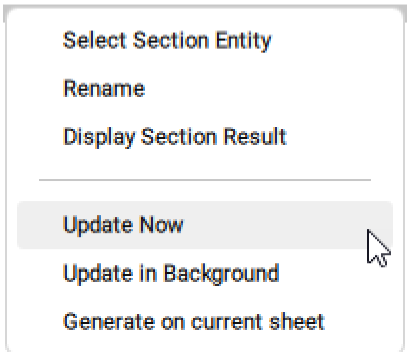
Note: The RENDERCOMPOSITIONMATERIAL system variable must be set to **On**.

Update BIM sections in the project browser



1 Right-click a section in the **BIM Project Browser** panel.

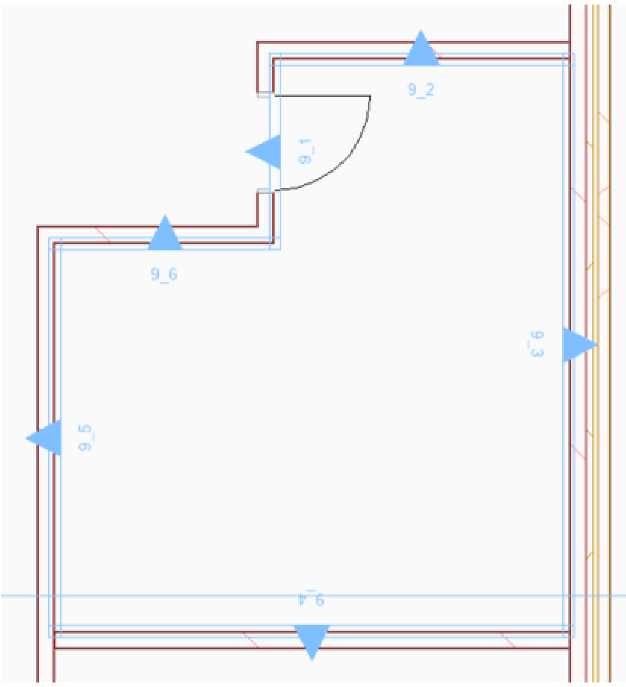
2 Select **Update Now** or **Update in Background** from the context menu.



26.8.5 Section indicators, story indicators and grid callouts

Section indicators are provided for any section which intersects with the source one, if the source section is a plan. They are stored on the dedicated **BIM_SECTIONS** layer. Turn the layer **On** or **Off** to manage the indicators visibility.

Note: Section indicators, story indicators and grid callouts are BIM tag entities that can be edited and customized. See the **BIM Tags** help article.



Story indicators are provided for any non-horizontal section if the model drawing has stories.

Grid callouts are provided if the model drawing has a grid and it's parallel or perpendicular to the source section plane.

26.9 Editing generated drawings

When a BIM Section is generated, a block containing the drawing generation result is added to the model space of the target drawing, as well as a paper space viewport looking into this block. The block and the viewport initially are placed so they do not overlap with the existing geometry contained in the drawing. You can move this block or viewport to a new location, which will be preserved on subsequent updates.

In the default template drawing which is used to create the generated drawings **Viewlabel** and **NumberBubble** blocks exist. You can place them associated with the selected sheet view. The **Title** block in this drawing contains attributes that will be filled in automatically when the associated **Sheet Set** properties are defined.

A sheet list table can be placed on any sheet of the sheet set. In the current implementation, the table content is not associative yet.

26.9.1 Inserting the title block

1 Open the **Drawing Explorer - Blocks** dialog box.

2 Double-click the **Title** block.

The **Drawing Explorer** dialog box closes and the block is attached to the cursor.

In a metric units project the size of the **Title** block fits the A1 paper size (594 x 841 mm).

In an imperial units project the size of the **Title** block fits the Arch D paper size (24 x 36 inches).

3 Click at the lower left corner of the paper sheet in the layout.

4 Open the **Sheet Sets** panel (SHEETSET command) and define the **Project** and **Custom Properties** available in the context menu.

5 Select the **Title** block, then click the **Update Fields...** tool button (

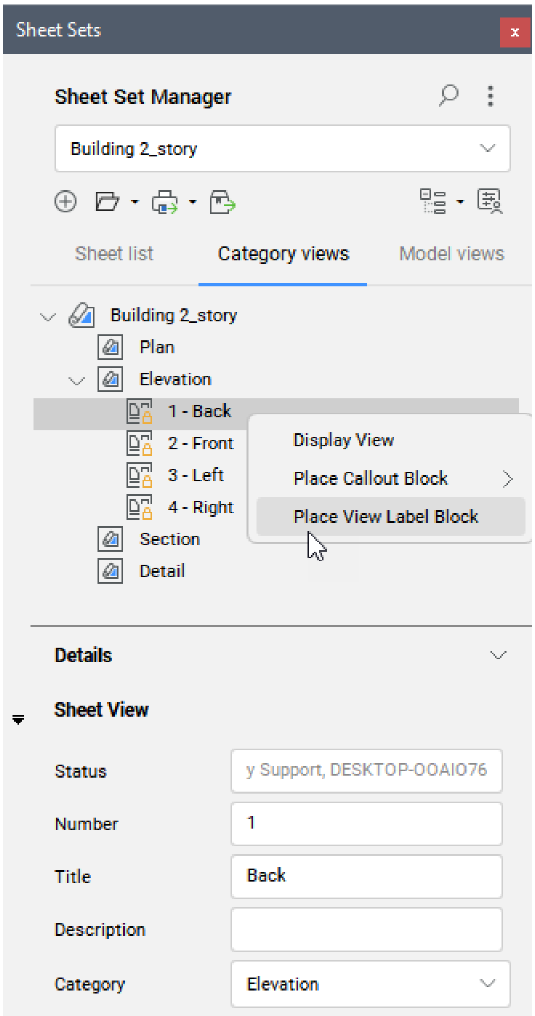
) in the **Redraw/Regen** toolbar.

26.9.2 Inserting a view label for a sheet view

1 Select the sheet view in the **Sheet Sets** panel.

2 Right-click the sheet view for which you want to insert the view label block.

3



4 Choose **Place View Label Block** in the context menu.

You are prompted: _VIEWLABEL

Units: Millimeters Conversion: 1

Insertion point for block:

5 Specify the insertion point.

6 **You are prompted**: Scale factor for block <1.0>:

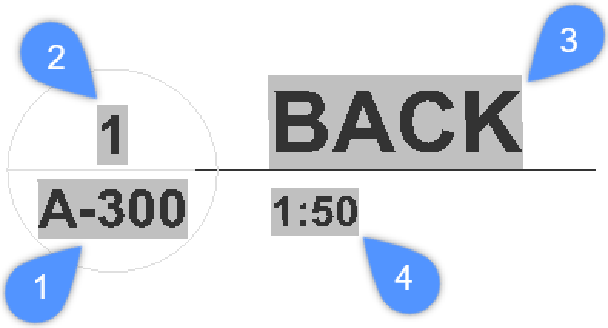
7 Specify the scale factor or right-click to accept the current scale.

8 **You are prompted**: Rotation angle for block <0>:

9 Specify the rotation angle or right-click to accept the current angle.

10 The View Label block is inserted.

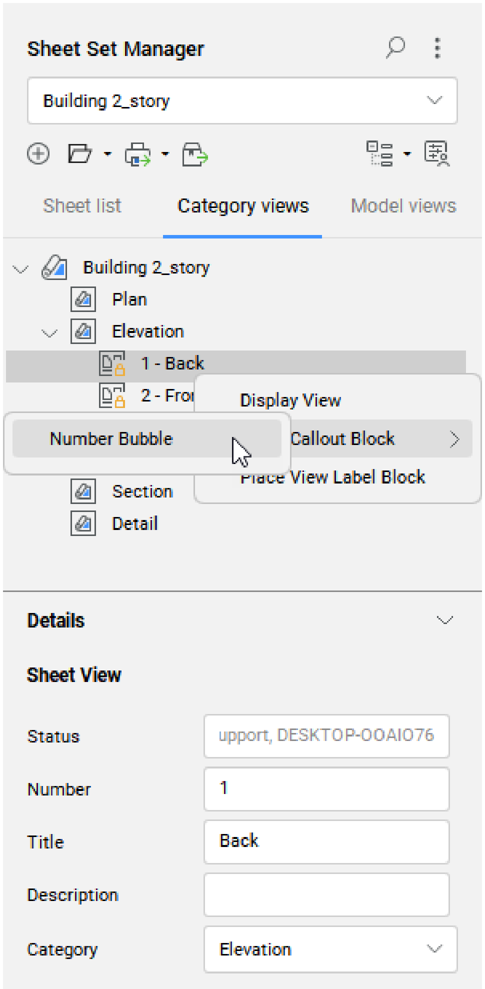
The Sheet Number (1), Sheet View Number (2), Sheet View Title (3) and Viewport Scale (4) of the selected sheet view are filled in automatically.



26.9.3 Inserting a number bubble block for a sheet view

1 Select the sheet view in the **Sheet Sets** panel.

2 Right-click the sheet view for which you want to insert the **Number Bubble** block.



3 Choose **Place Callout Block** > **Number Bubble** in the context menu.

You are prompted: _CALLOUT

Units: Millimeters Conversion: 1

Insertion point for block:

4 Scale factor for block <1.0>:

5 Rotation angle for block <0>:

6 Specify the insertion point.

7 **You are prompted**: Scale factor for block <1.0>:

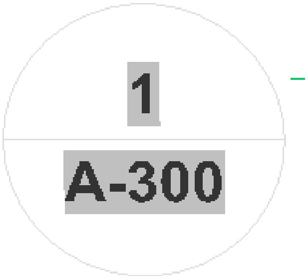
8 Specify the scale factor or right-click to accept the current scale.

9 **You are prompted**: Rotation angle for block <0>:

10 Specify the rotation angle or right-click to accept the current angle.

11 The **Number Bubble** block is inserted.

The Sheet Number (5) and Sheet View Number (6) of the selected sheet view are filled in automatically.



26.9.4 Inserting a sheet list table

A **Sheet List** table can be inserted on any sheet of the sheetset. In the current implementation, the table content is not yet associative and therefore will not be updated automatically when sheets are added, removed or renamed.

1 Right-click the sheetset name in the **Sheet Sets** panel and choose **Insert Sheet List Table** in the context menu.

You are prompted: Select insertion point.

2 Specify a point in the drawing.

26.10 Interior elevations

26.10.1 Command

BIMSECTION

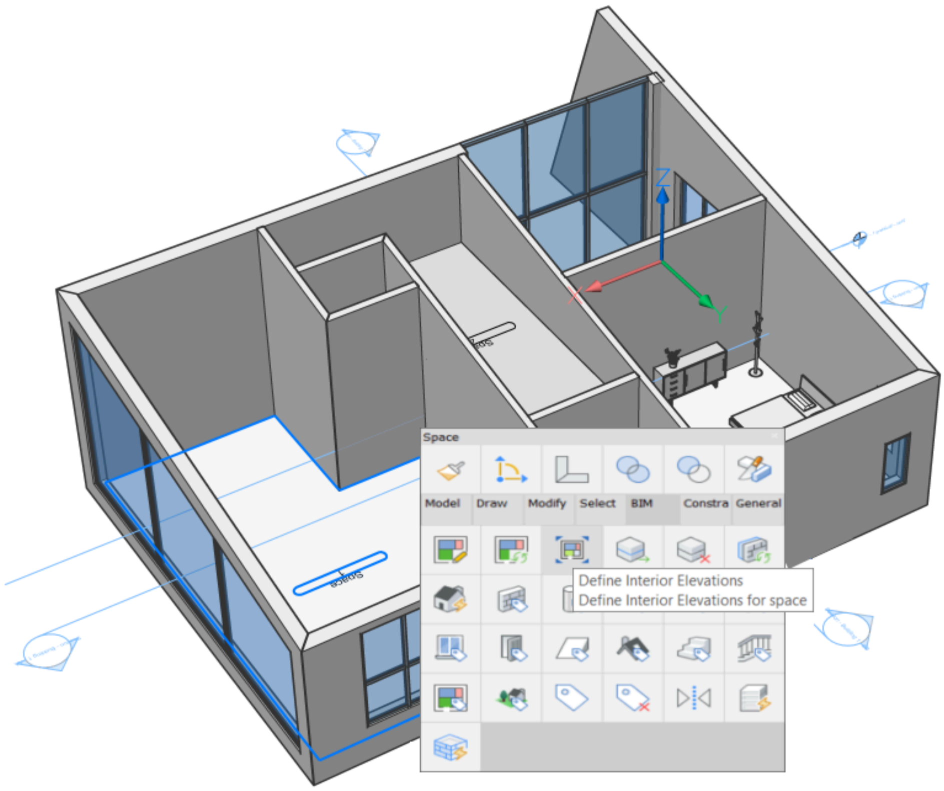
26.10.2 About

Interior elevations show wall elevations and floor plans for individual Spaces. They provide the details and dimensions of building elements. This can be particularly useful for interior designers or to make a layout of power sockets and light switches. In BricsCAD they can be automatically defined by a BIM Space.

For more information about this command, please read BIMSECTION.

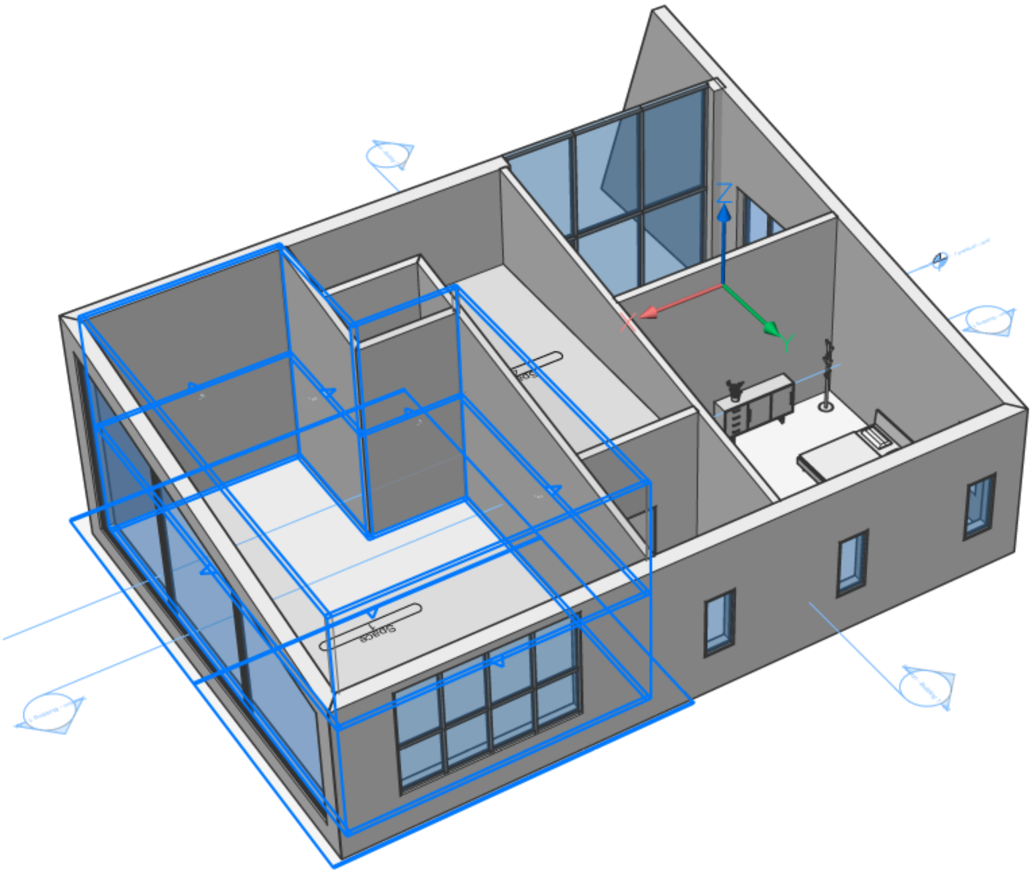
26.10.3 Create interior elevations from a BIM space

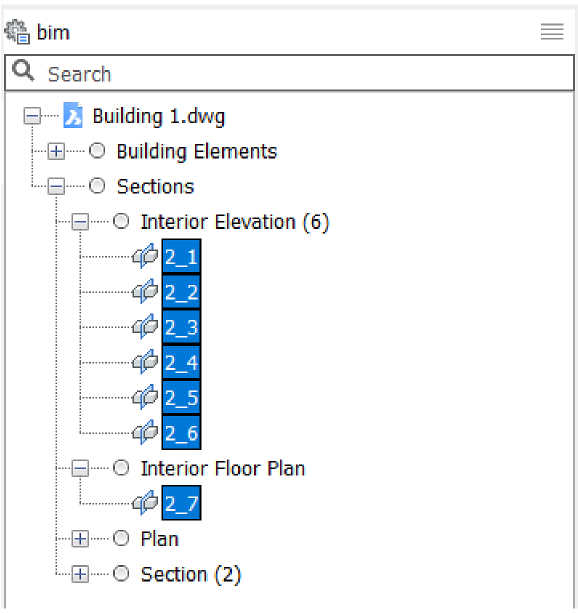
1 Hover the cursor over a Space Tag and select **Define Interior Elevations** from the **BIM** Tab in the Quad
Or type BIMSECTION in the Command line, followed by i and Enter to choose the interior option.
You are prompted: Select BIM Spaces to generate Interior Elevations or [Attach section]:



2 (Skip if you used the Quad) Select the BIM Space for which you want to create the interior elevations and hit Enter.

3 The interior elevations for each wall and the interior floor plan are now defined. They can be found in the **Structure** panel.

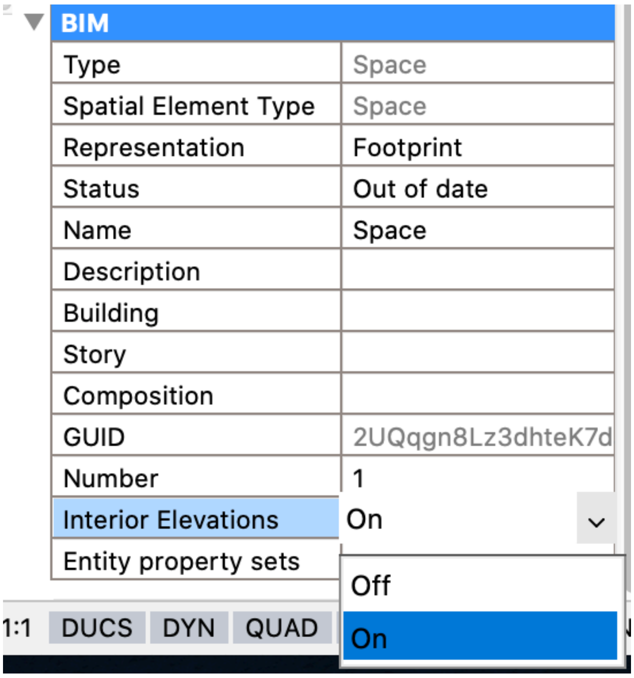




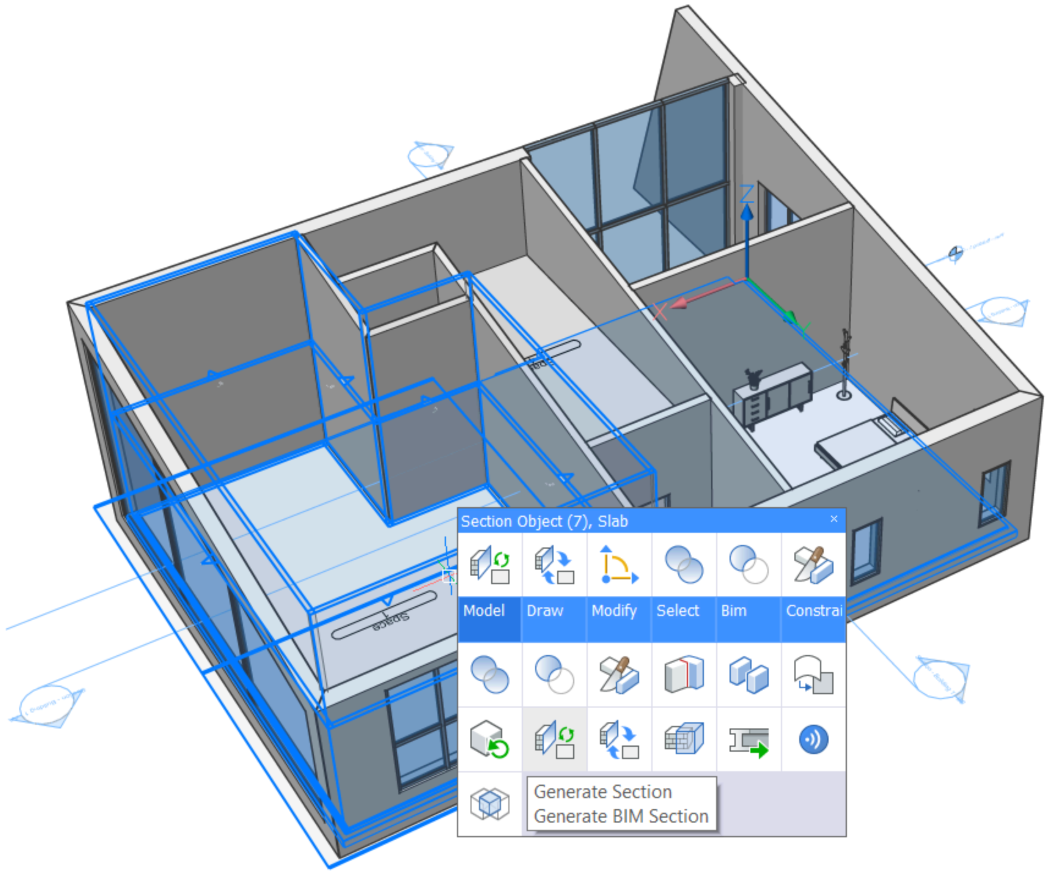
Six interior elevations and one interior floor plan have been created and will appear in the ***Structure*** *panel.*

Note: If there already are interior sections for a Space, running BIMSECTION again on that Space will have no effect. To regenerate the interior elevations, you must delete the existing interior elevations in that Space and run BIMSECTION again.

4 To show or hide the interior elevations you can use the HIDE/ISOLATE/UNISOLATE objects commands **or** you can go to the properties of the Space and switch the interior elevations on and off in the **BIM** tab.

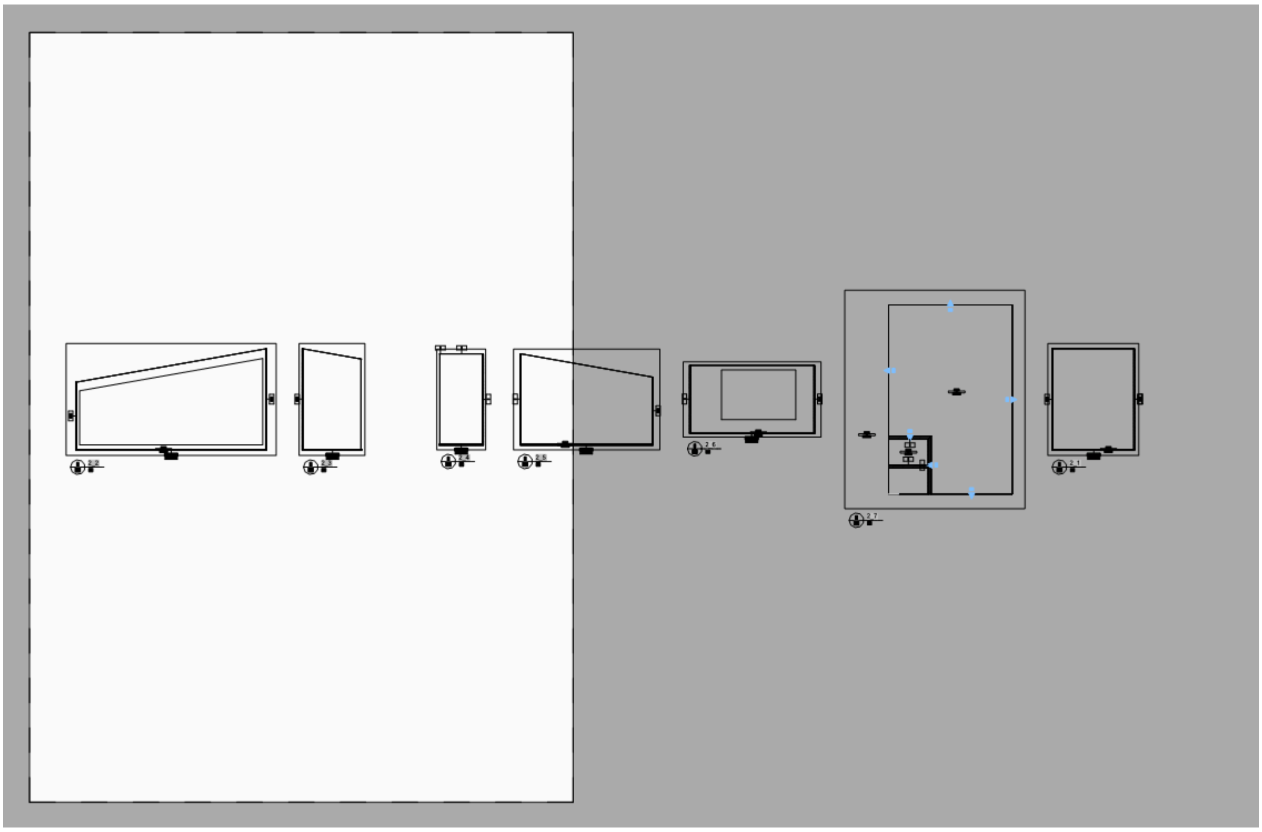


5 Select all elevations, as illustrated above and select **Generate Section** from the Quad to generate the interior elevations on a sheet.



6 To open the section results click **Open Model** whilst the section planes are selected.

The sheet is automatically saved in the same folder as the model drawing. The following elevations have been produced and can be edited for the selected Space in the model above:



Note: The section drawings are stored in the file **InteriorElevations** by default. If you generate sections of a second Space, the sections will be added to the same file as the previous ones.

If you need more information on how to generate drawings, visit the **Generating drawings** page.

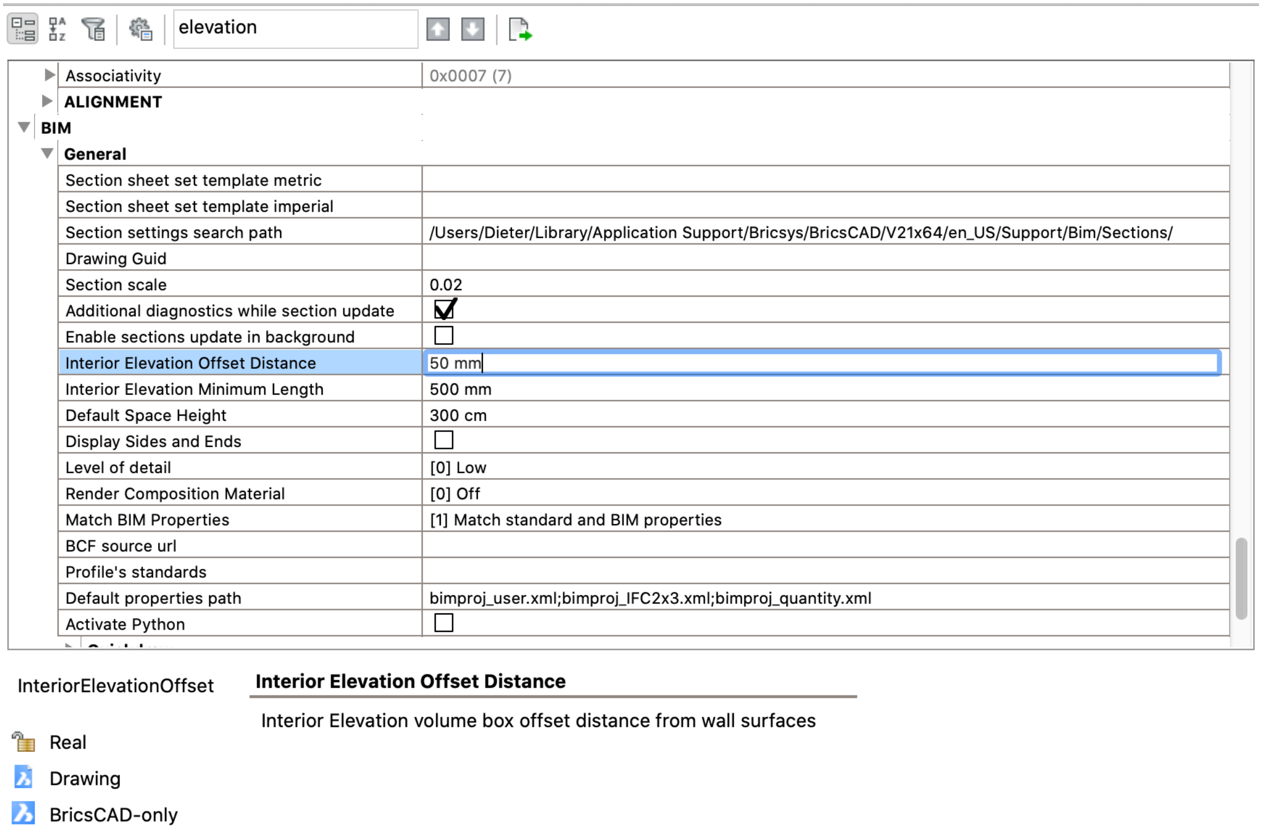
If you want to annotate your drawings, see **Annotate** > **Creating linear dimensions**.

26.10.4 Edit the size of the interior elevations

It's possible to edit the size of these sections to include furniture for example. If you create interior elevations for a BIM Space, the offset value is 50mm by default. To include furniture in a section, you have to change this value.

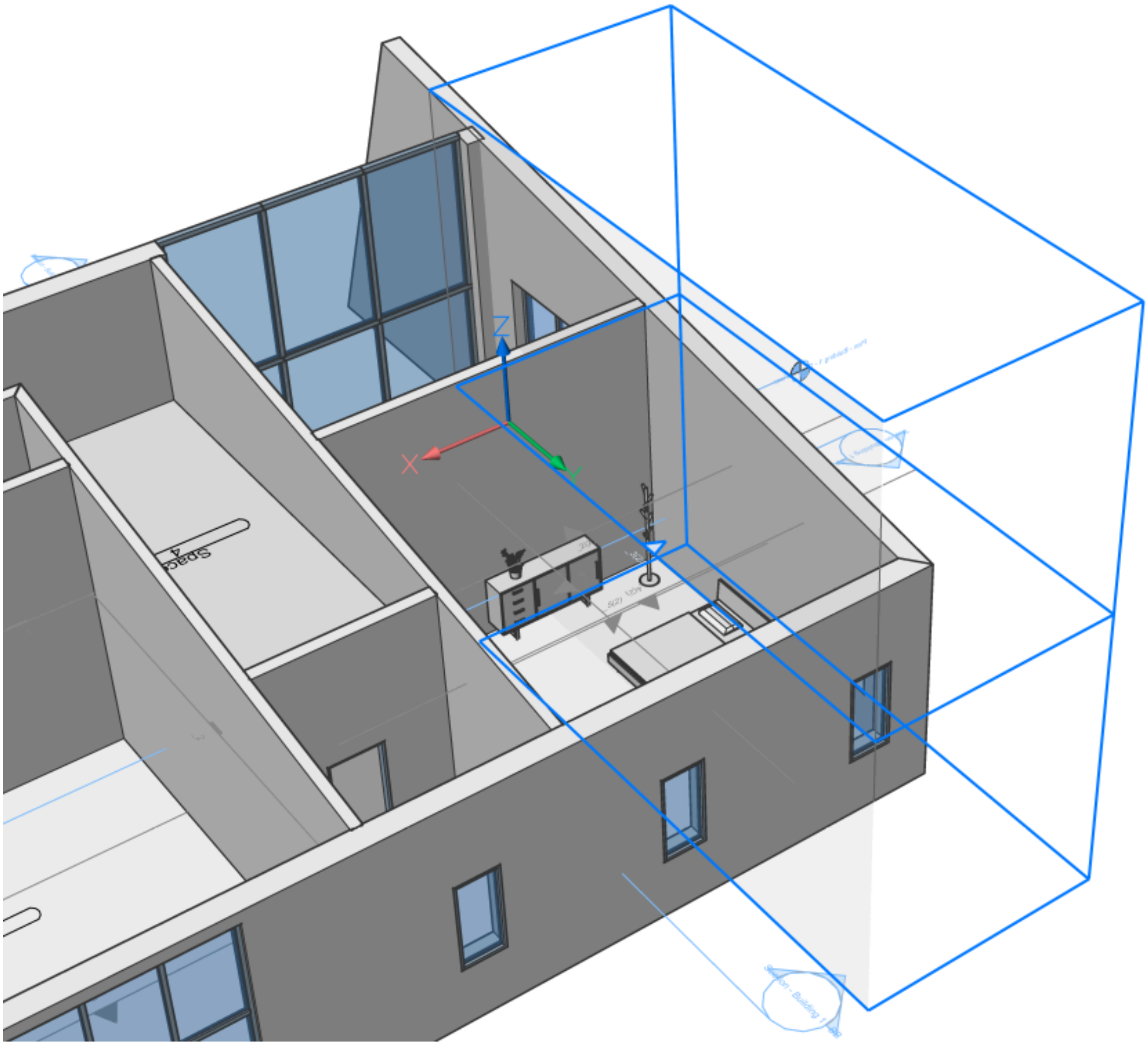
1 To change this setting type INTERIORELEVATIONOFFSET in the Command line and hit Enter. Type a new value and press Enter **or** click two points in your drawing and the distance between them will be selected.

   You can also change the offset by typing Settings in the Command line, search for elevation and adapt the **Interior Elevation Offset Distance** value.



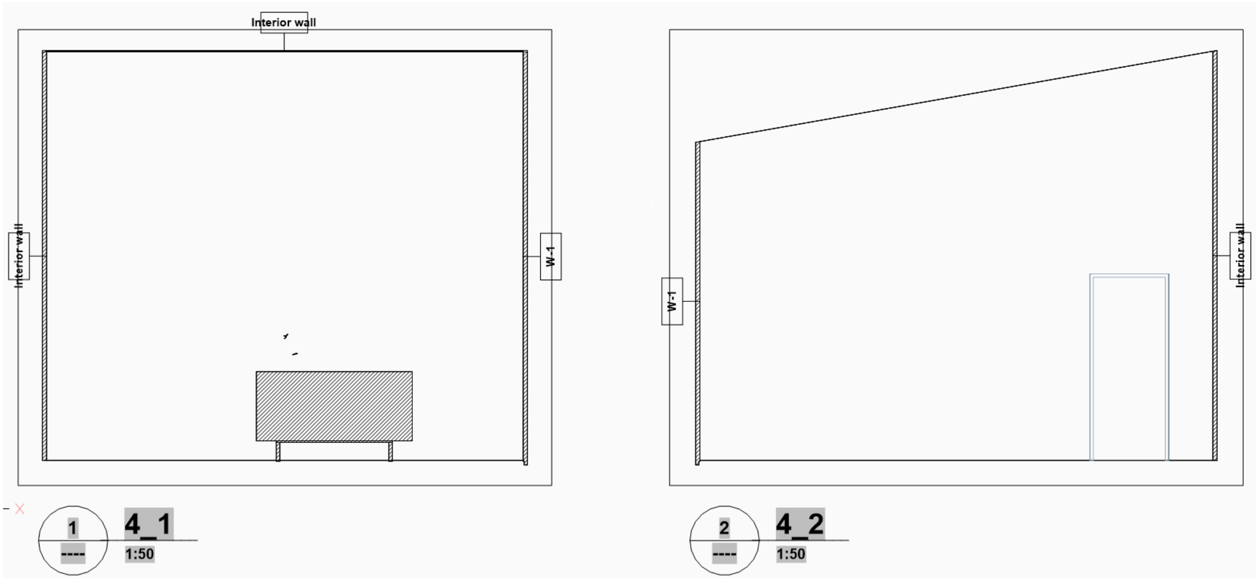
2 If you make new Interior Elevations, the offset distance will be the chosen value. This is also visible when you select the interior elevations in your drawing:

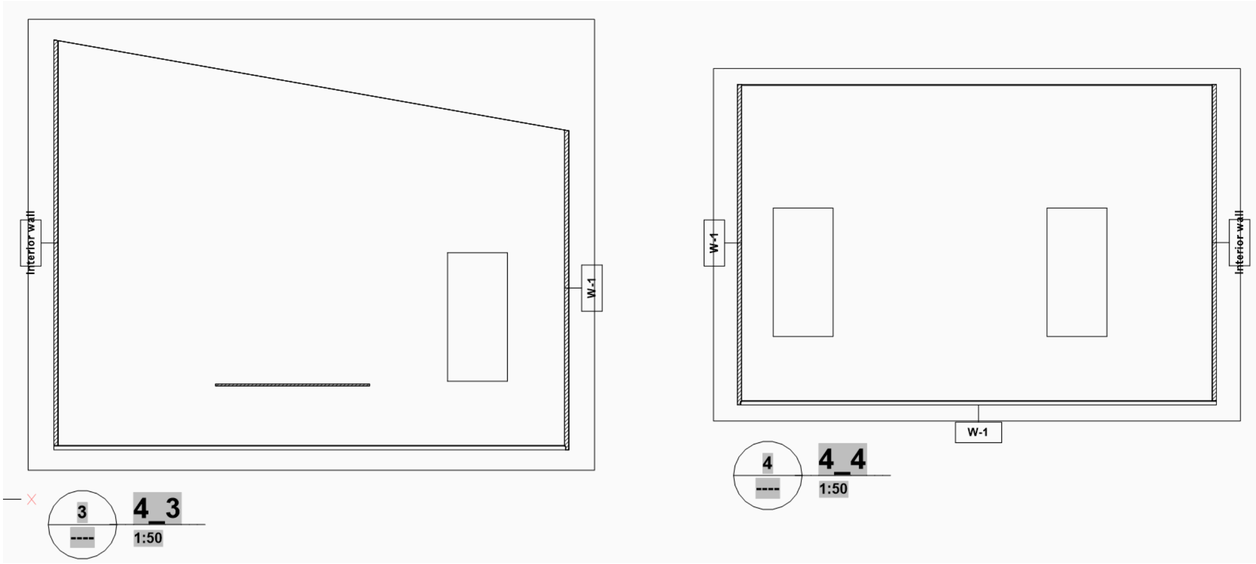
3



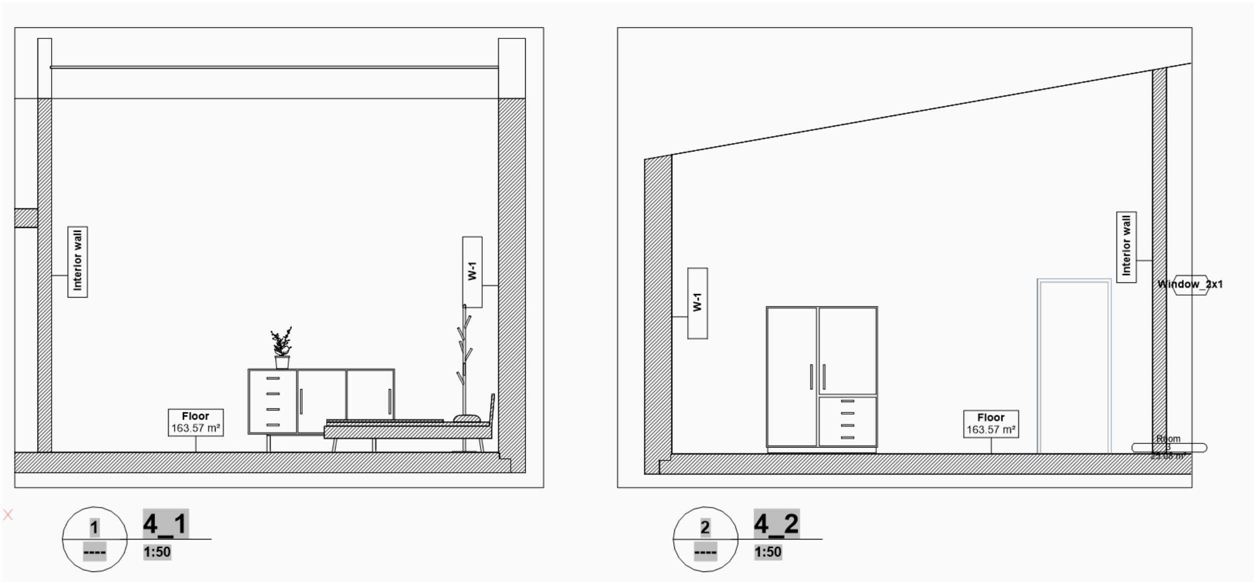
4 This results in more detailed section sheets with the furniture included.

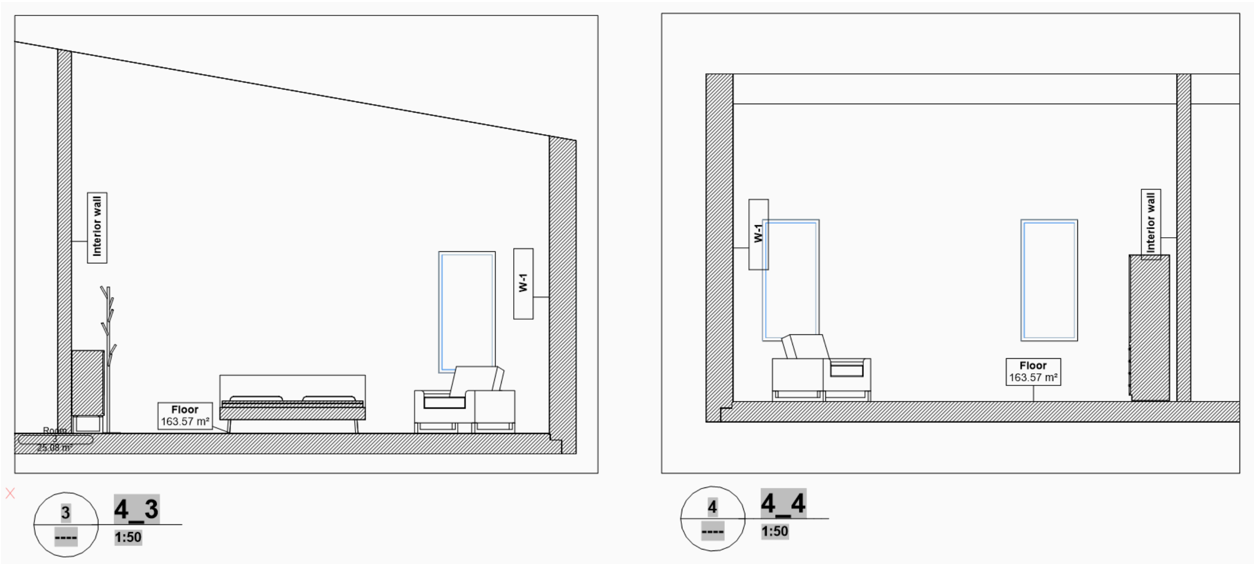
5





Without specified offset distance (50 mm)





With specified offset distance (2100 mm)

26.11 Spot elevation level

26.11.1 Command

BIMTAG

26.11.2 About

The Spot Elevation tag displays the absolute elevation of a selected point of a BIM object displayed in a section view, based on the **World Coordinate System (WCS)**.

As the Spot Elevation tag is an associative tag type, it can only be used in the **paper space** of a drawing sheet.

For more information on how to set up Sheets and create Projects, please visit the **Project Browser** article.

26.11.3 Procedure: how to place a spot elevation tag on a section sheet

1 Open a section sheet of the model. Right-click on the section in the **Project Browser** and select **Display Section Result**.

2 Type BIMTAG in the Command line **or** click on **tag** in the **Draw** menu of the Quad.

3 **You are prompted**: [Manual/Box selection/selection options (?)]

Type **M**t o choose **Manual** and press Enter.

4 **You are prompted**: [Auto/Current mleader style/change Tag type]

Type **T** to change the **Tag** type and press Enter.

5 **You are prompted**: [Tag/cOmposition/Load-bearing directions/Spot-elevation]

Type **S**to choose **Spot-elevation** and press Enter.

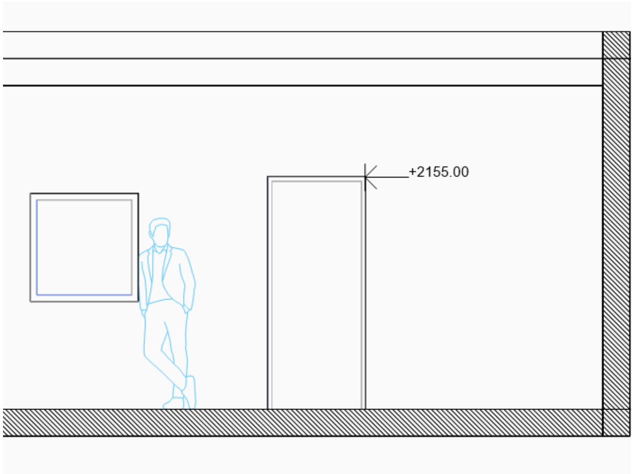
6 Click on a point on the edge of a BIM object to display the elevation height. The displayed value is the

elevation level measured relative to level 0 of the model.

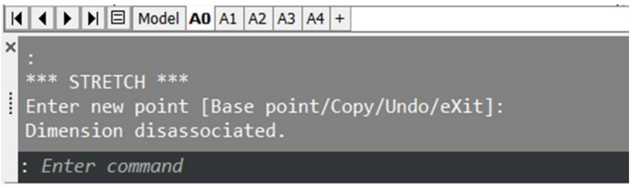
Note: You can select any point of the edge of a BIM object. Turn off ESNAP if you don't only want to select snapping points.

7 When there are several tags possible on the location where you place the tag, hit CTRL to scroll through the different options while the Hotkey Assistant (HKA) appears at the bottom of the screen.

8 Move your cursor in order to adjust the position of the tag and left-click to place the tag.

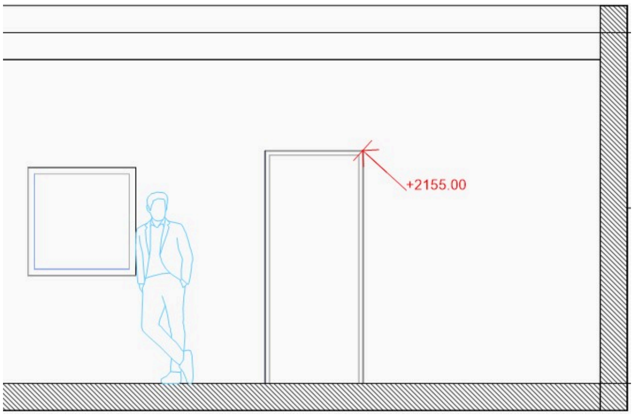


Note: The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value becomes dissociated in the Command line.

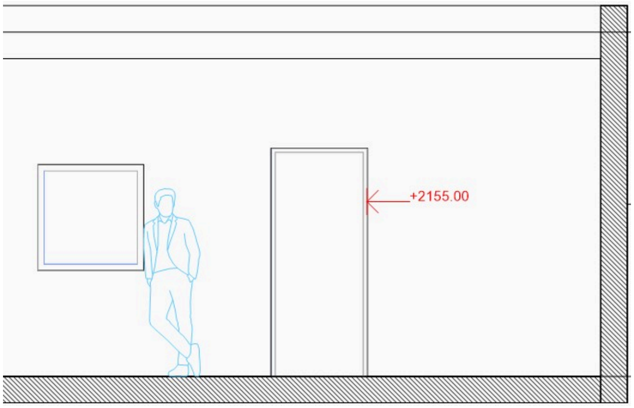


9 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

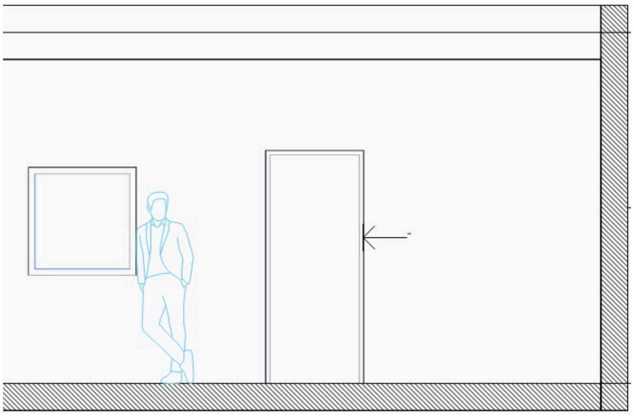
Note:If the tag has been displaced, it may behave irrationally or haphazardly by either turning red or omitting its value, as illustrated below:



Tag turned red after displacing the arrowhead of the leader and updating the section.



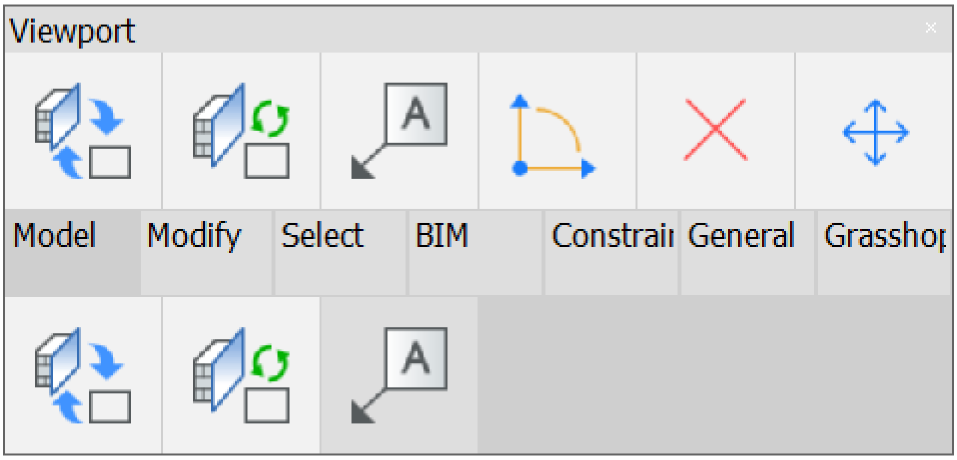
Tag turned red after displacing the entire tag and updating the section.



Tag value disappeared after displacing the tag and updating the section.

26.11.4 Procedure: how to place a spot elevation tag on an elevation sheet

1. Open an elevation sheet of the model. Right-click on the elevation sheet in the **Project Browser** and select **Display View**.
2. Type BIMTAG in the Command line, or click on **Tag** in the **Model** menu of the Quad.



3 **You will be prompted**: [Manual/selection options (?)]

Type **M** to choose **Manual** and press Enter.

4 **Later, you will be prompted**: [Auto/Current mleader style/change Tag type]

Type **T** to change the **Tag** type and press Enter.

5 **Finally, you will be prompted**: [Tag/cOmposition/Load-bearing directions/Spot-elevation]

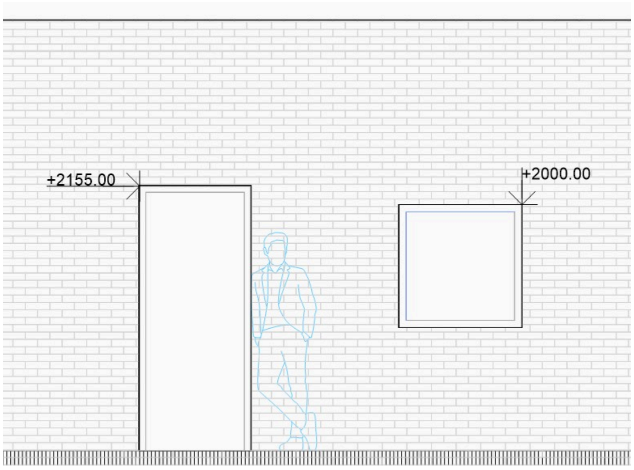
Type **S** to choose **Spot-elevation** and press Enter.

6 Click on a point on the edge of a BIM object to display the elevation height. The displayed value is the absolute elevation level measured relative to Floor 0 of the model.

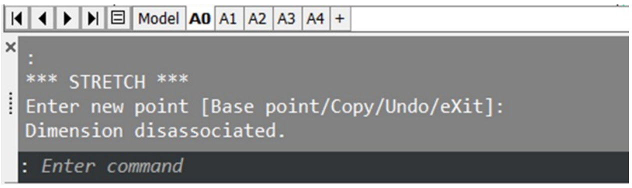
Note: You can select any point of the edge of a BIM object. Turn off ESNAP if you don't only want to select snapping points.

7 When there are several tags possible on the location where you place the tag, hit CTRL to scroll through the different options while the HotKey Assistant (HKA) appears at the bottom of the screen.

8 Move your cursor in order to adjust the position of the tag and left-click to place the tag.

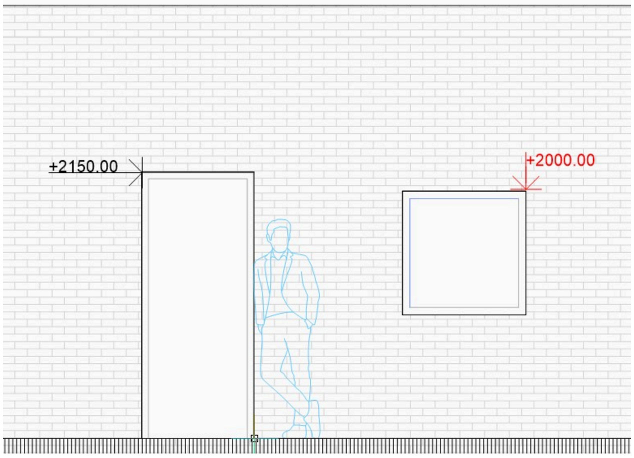


Note: The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value is dissociated in the Command line.

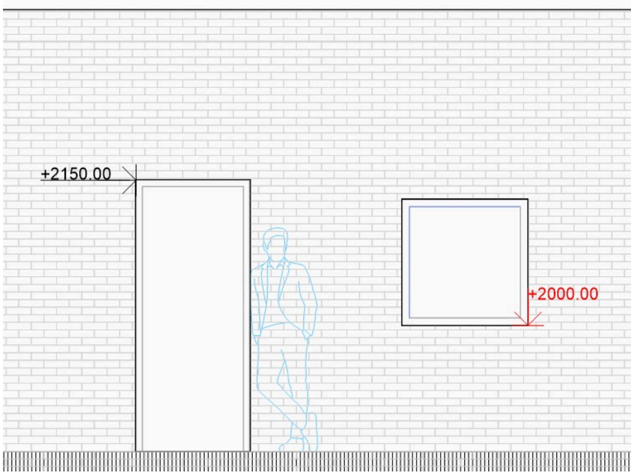


9 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

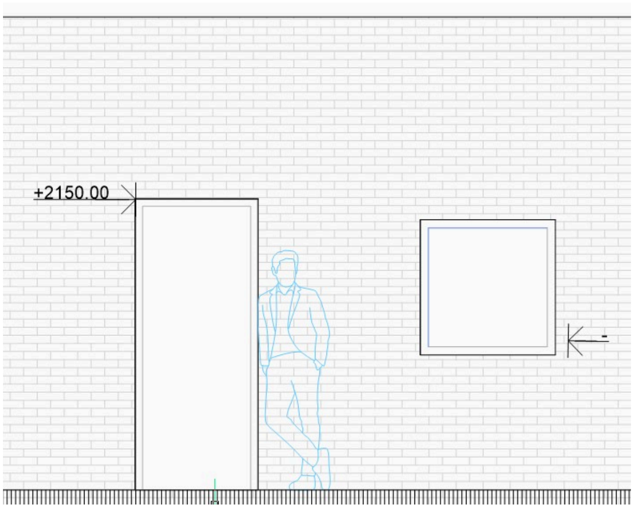
The tag should not behave irrationally or haphazardly by either turning red or omitting its value when it's displaced.



Tag turned red after displacing the arrowhead of the leader and updating the section.



Tag turned red after displacing the entire tag and updating the section.

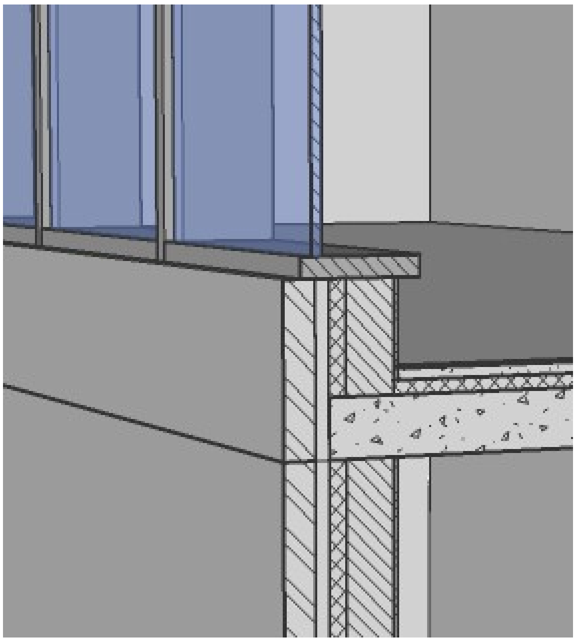


Tag value disappeared after displacing the tag and updating the section.

26.11.5 Procedure: placing spot elevation level tags on floor plans or reflected ceiling plans

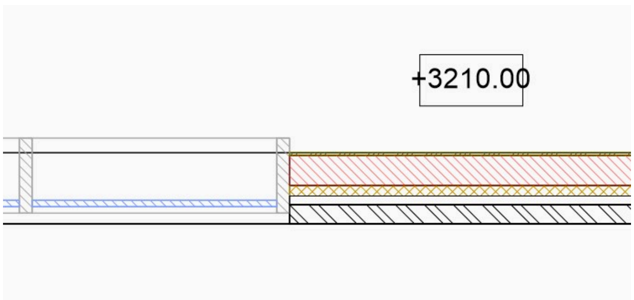
1. Open a floor plan or a reflected ceiling plan from the **Project Browser**.
2. Go through step 2-7 of 'How to place spot elevation level tag in a section sheet'.
3. In a floor or ceiling plan, there are 2 different tags possible.
   - **With MLeader**

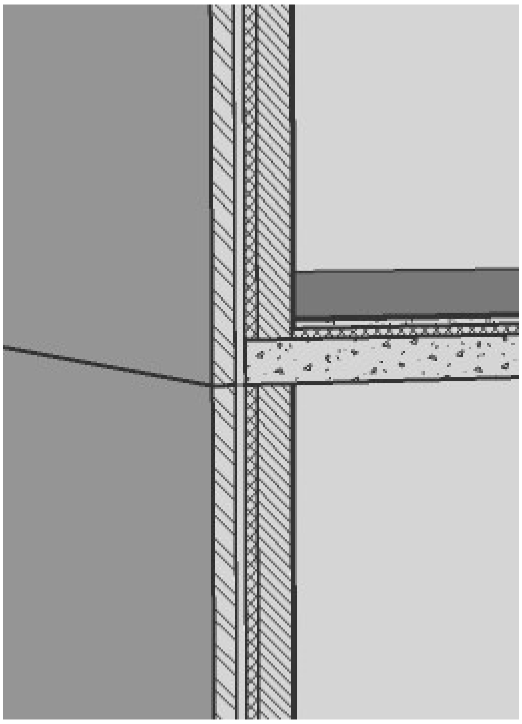
     This annotation tag type, which illustrates with a leader line, is also normally used in the section or elevation sheets. You can use this tag on entities that do not reach or aligned to the floor/ceiling level, such as a window sill in the picture below.

     Spot elevation level tag on window sill level which is not on floor level.
   - **Without MLeader**

     This tag has a different layout to the tags found in the elevation or section sheets. This tag appears when you place a spot elevation tag on entities that reaches the floor/ceiling plane, thereby displaying the elevation level of the floor plane.

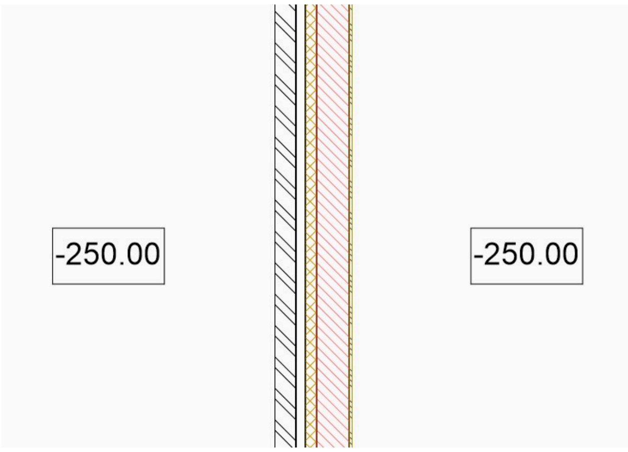


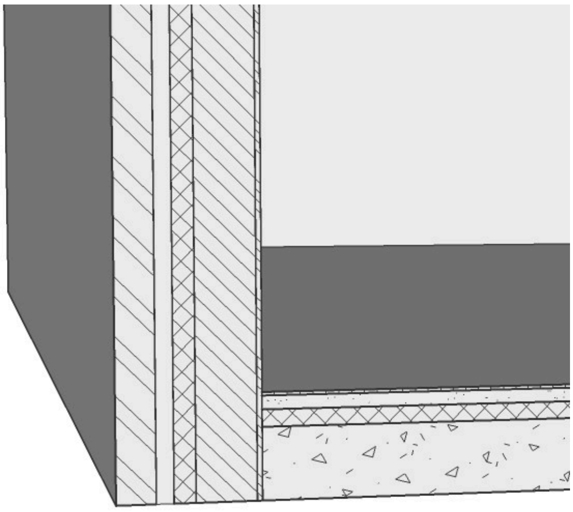
Spot elevation level tag on wall that reaches till the floor slab.

4 When placing a tag on a wall in plan view, you may encounter different results for different plies of the wall. This is due to how elevation levels are measured In BricsCAD. When placing a spot elevation tag on an entity in plan view, it refers to the elevation value of the lower level of the entity's ply, if applicable. Similarly, when you place a tag on a ceiling plan, the upper level of the ply is measured. Since plies of a wall may differ in their lengths based on how they are designed at detail junctions, the spot elevation tag value may differ according to the ply you have selected.

- **Construction detail with unedited plies**

  The interior and exterior plies have the same height. All the spot elevation tags you place on this wall in plan view will therefore display the same value.

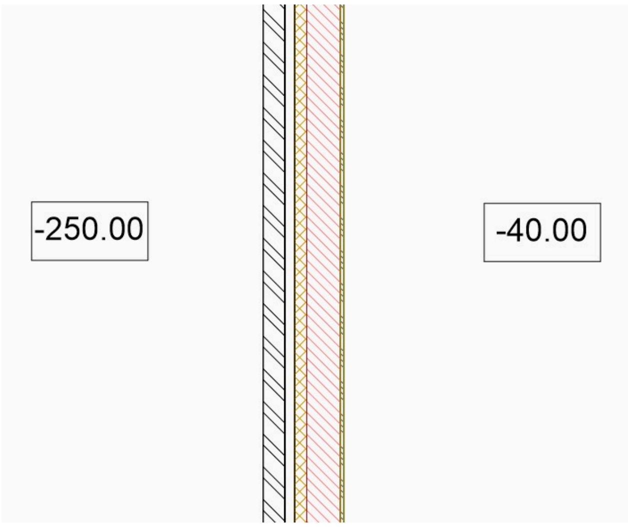


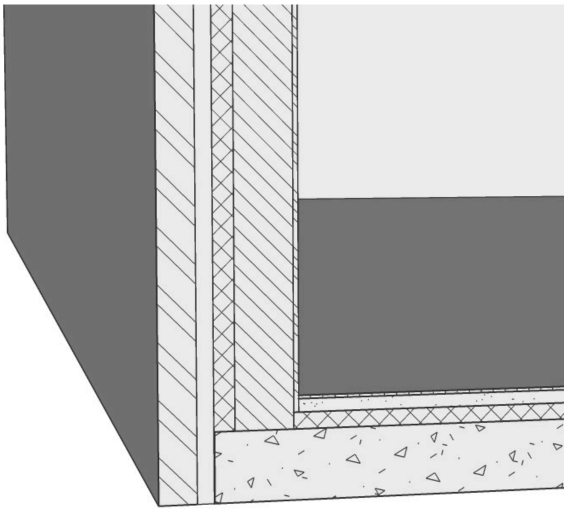


Note: The value is -250 instead of 0 because Floor 0 is defined as the level of the upper side of the ground floor in this model.

- **Edited construction detail**

  If you have adjusted how the wall would connect to a floor slab based how it would be constructed, the plies of the wall will therefore have different heights. Depending on which ply you select to place the spot elevation level tag, a different value will be displayed.

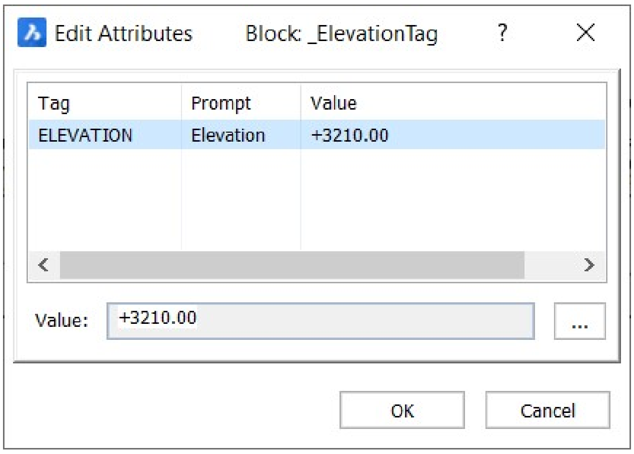




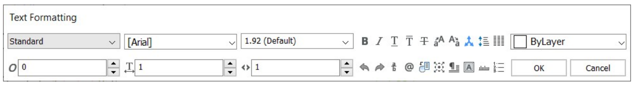
Elevation level of interior ply is higher because ply does not reach till the base level of the floor.

26.11.6 Procedure: adjusting the layout of the tag

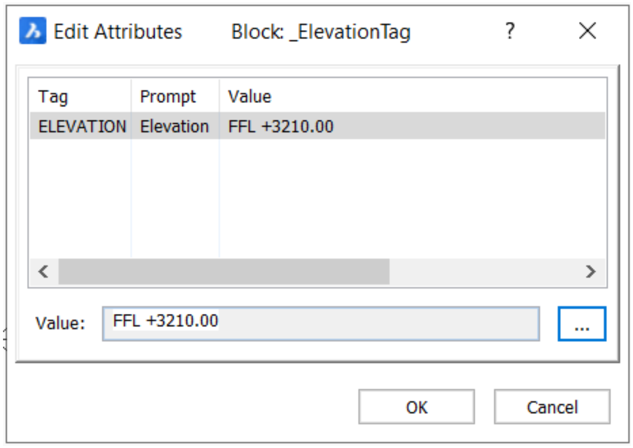
1 Double-click on the value of the spot elevation tag to adjust the layout. A dialog box will pop-up where you can edit the attributes.

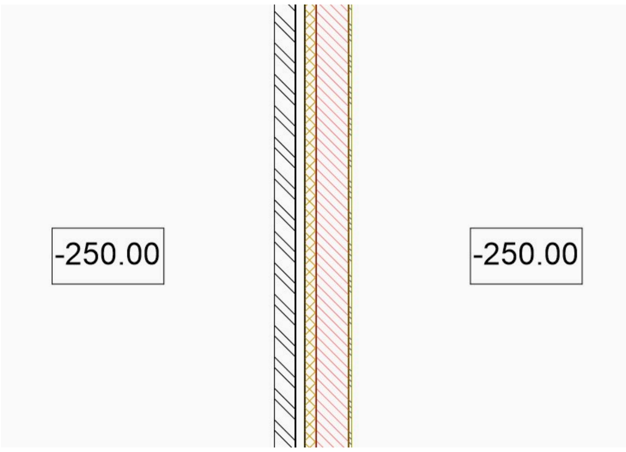


2 Click on the three dots to open the **Text Formatting** dialog box.



3 Here you can edit the attributes of the displayed value of the spot elevation level tag. You can put prefixes and/or suffixes to the values, e.g. FFL, SSL, to better illustrate the levels you are referring to.





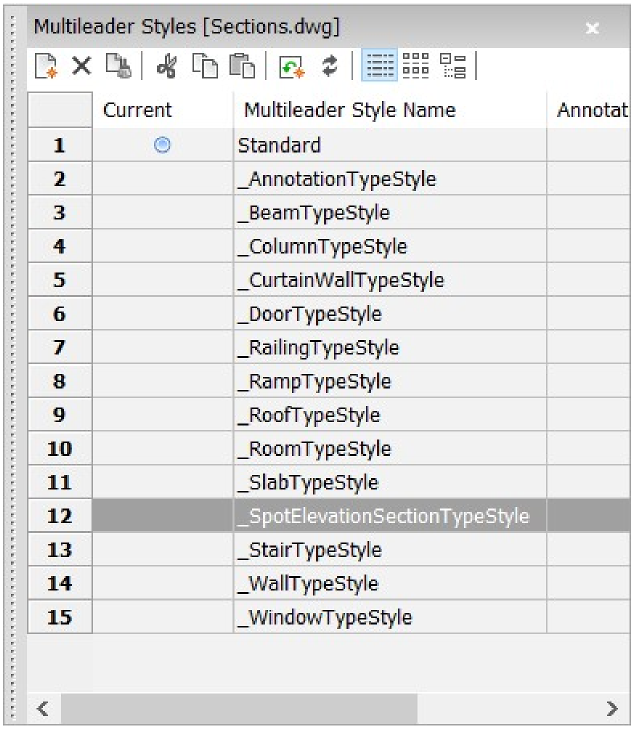
4 If the prefixes and/or suffixes are commonly used, you can create or edit the tag blocks so you do not have to repeat Step 3. For more info, visit the command article BEDIT.

5 For a detailed explanation about attributes, visit the command article MText.

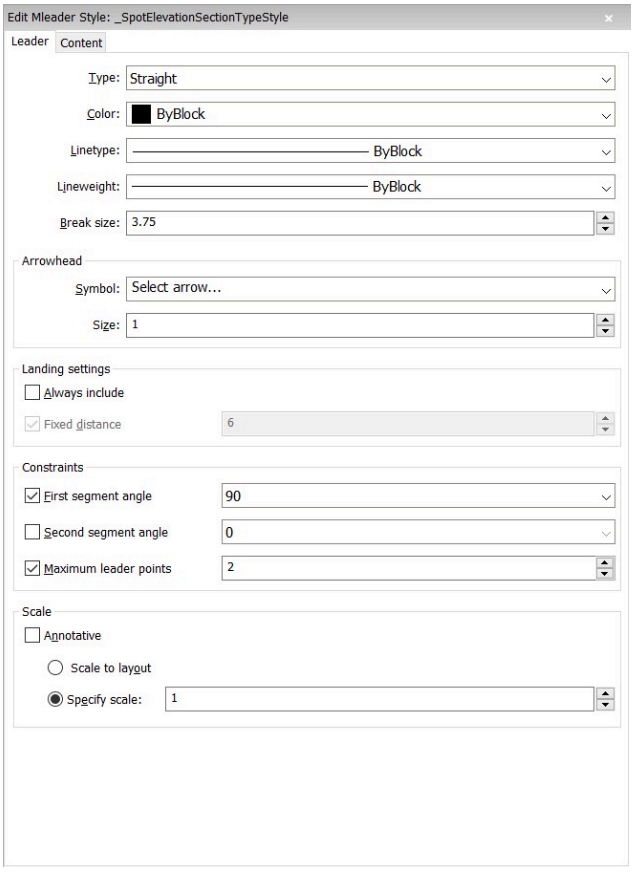
26.11.7 Procedure: adjusting the layout of the leader

1 Type MLEADERSTYLE in the Command line and press Enter. The **Drawing Explorer** dialog box will pop-up where you can adjust the layout of the leader.

2 Click on **SpotElevationSectionTypeStyle** in the **Multileader Styles** dialog box.

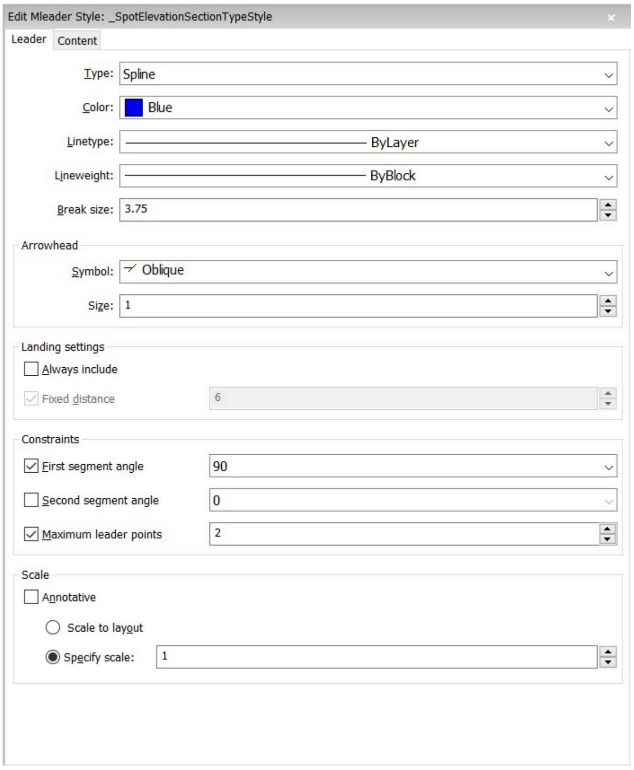


3 You can edit the settings of the leader in the **Edit MLeader Style** dialog box. For more info about the MLeader style settings, visit the article MLeaderStyle.

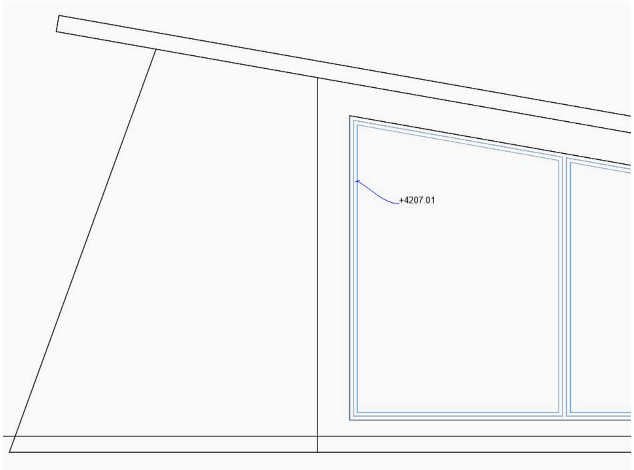


4 After adjusting the settings, click on regen, close the **Drawing Explorer** and update the section to load the new settings.

Note: If the new settings don't load, type BIMTAG in the Command line, select the viewport, click **R** to select **Regenerate All** and press Enter to load the new settings.



New Settings



Resulting BimTag

26.12 Adding BIM schedules using BOM templates

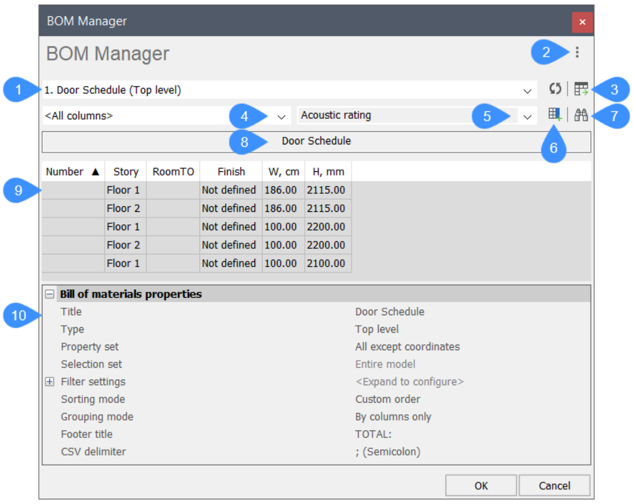
Shape Lite Pro Mechanical BIM

26.12.1 BOM Manager for BricsCAD BIM overview

The **BOM Manager** panel (Bill of Materials Manager panel) for BricsCAD BIM allows you to view and edit Bills of Materials (BOMs) in the current BIM Project.

An architectural BOM table is a list of unique BIM elements used in a BIM model. In BricsCAD, the list is represented as a table entity and can be placed anywhere in the drawing. Each element in the table has its own serial number (starting with 1), name, and number of occurrences.

The properties of the selected table/column are displayed in the property grid at the bottom of the **BOM Manager** panel, under the **Bill of materials properties** node.



1 Name of loaded BOM file and its type

2 Main menu

3 Place table

4 Category selector

5 Column selector

6 Add column (selected in the column selector)

7 Search in table

8 Table title

9 Table content

10 Property grid

With the **BOM Manager** panel for BricsCAD BIM, you can:

- Edit bills of materials:
  - Add or remove columns (includes formula columns).
  - Move selected columns to new positions.
  - Rename the columns.
  - Configure how the BOM table is sorted.
  - Configure the footer value for each individual column.
  - Configure the units and how they are displayed for each column.
  - Configure the width of each column with column properties.
  - Configure the visibility of each individual column.

- Filter the components with a filter expression.
- Group similar rows.
- Change the title of both the title row and the footer row.

- Search in the table.
- Highlight and select entities in the drawing.
- Save the BOM configuration as a template file.
- Load a BOM table configuration from a template file.
- Place one or more BOM table instances in a drawing.

Note: The BricsCAD Mechanical and BricsCAD Ultimate license levels offer the full functionality of the **BOM Manager** panel. See also the **Bill of Materials Manager** and **BOM Manager panel** articles.

26.12.2 Creating a door BOM table from scratch/Setting a filter

Note: Creating a BOM template in a large BIM drawing can significantly slow the process. It is best to create a BOM template in a small BIM drawing and then use it with large files.

To create a new door BOM table, follow the next steps:

1. Open the **BIM Project Browser** panel.
2. Click the **Files** tab.
3. Click the **Add** button (⊕) and select **New BOM**.

   The generic **Save file** dialog box opens.
4. Name the new BOM file and save it in the desired location.
5. The **BOM Manager** panel opens with the new BOM file loaded.
6. Click the **Main menu** button (⋮) at the upper right-hand corner of the **BOM Manager** panel.
7. Select the **Set filter** option ( ).

   The **Filter Configurator** dialog box opens. See the **Filter Configurator dialog box** article for more information.
8. From the **Category selector** drop-down menu, select **BIM**.

   The **Column selector** drop-down menu is populated with all the BIM properties present in the model.
9. From the **Column selector** drop-down menu, select the **Type** property.

   The **Type** property is added to the **Conditions** table of the dialog box.
10. For the **Type** row in the table, click the **Conditions** cell and select **Equal to** from the drop-down menu.

    The properties of the **Type** category are now available for selection in the **Value** cell.
11. Click the **Value** cell and select **Door** from the drop-down menu.
12. Click **OK** to save the created table filter.
13. (Optional) Repeat steps 5-11 to add another filter (for example, to list only the first-floor doors, select

the **Story** property from the **BIM** category, then for the **Equal to** condition select **Floor 1** as value).

14 (Optional) Change the title of the table (for example, **New door schedule**)

15 Add the needed columns (see the **Adding new columns** section of the article).

Some frequently used properties added as columns for a door schedule are **Story**, **Space**, and **Construction type** (all belonging to the **BIM** category).

16 Click **OK** to save the table.

See also the **Operations on a BOM table**, **Adding new columns**, **Adding a thumbnails column**, **Changing the unit of a numeric column**, **Combining columns**, and **Adding a formula column to calculate weight** sections of the article.

26.12.3 Importing a BOM template

1 Open the **BIM Project Browser** panel.

2 Click the **Files** tab.

3 Click the **Add** button () and select **Import BOM**.

The **Select BOMs to import** dialog box opens.

4 Select the file/s and click **Open**.

The files are copied to the BIM Project's folder and listed in the **Schedules** folder of the **Files** tab.

26.12.4 Creating a table from a predefined BOM table

1 Open the **BIM Project Browser** panel.

2 Click the **Files** tab.

3 Expand the **Schedules** folder.

4 Right-click a predefined BOM template and select the **Duplicate** option.

A new BOM file with the **- copy** suffix is created and saved as part of the BIM Project's files in the **Schedules** folder.

5 (Optional) To rename the Project's file, right-click the new BOM file and use the **Rename file** option.

6 Edit the table as needed (some operations that can be performed on BOM tables are described in the next sections of the article).

7 Click **OK** to save the table.

26.12.5 Placing a BIM schedule on a sheet

There are three table templates in the **Schedule** folder of a BIM Project: **Door Schedule**, **Material List**, and **Window Schedule**. Each table displays a minimum number of relevant columns. To add more columns, follow the **Adding new columns** section steps.

To place a BIM schedule on a sheet, follow the next steps:

1 Open the **BIM Project Browser** panel.

2 From the **Sheets** tab, open the sheet on which to place the BIM schedule.

3 In the **Files** tab, expand the **Schedules** folder.

4 (Optional) Select a table (BOM file) from the list and drag and drop it on the sheet at the desired position.

5 Double click one of the predefined BOM files.

6 The **BOM Manager** opens with the selected template loaded and the _BMBOMTEMPLATEEDIT command is launched.

7 Click the **Place table** button and select the desired position/s on the document.

8 Right-click or press **Enter**/**Esc** to end the command. The **BOM Manager** closes as well.

26.12.6 Operations on the BOM table

Renaming a column

1 Double click the BOM table in the **Schedule** folder of the **BIM Project Browser** panel to open it in the **BOM Manager** panel.

2 Click the column's header to select the column.

3 Go to the property grid at the bottom part of the **BOM Manager**.

4 Type a new name in the **Title** field of the **Column properties** node.

5 Click **OK** to save the table.

Sorting a table

1 Double click the BOM table in the **Schedule** folder of the **BIM Project Browser** panel to open it in the **BOM Manager** panel.

2 Right-click the name of a column to display a context menu.

3 Hover the cursor over the **Sorting** option, then select the desired sorting method from the menu.

4 Click **OK** to save the table.

Moving a column

1 Double click the BOM table in the **Schedule** folder of the **BIM Project Browser** panel to open it in the **BOM Manager** panel.

2 Click the header of a column and drag and drop it to its new position in the table.

3 Click **OK** to save the table.

Highlighting table items in the drawing

Click items (any row cell) in the table to be highlighted in the drawing.

Right-click an item in the BOM manager table to display a context menu:

Select

Selects the item in the drawing.

Zoom to

Zooms in the current view to the item in the drawing.

See also the **Adding new columns**, **Adding a thumbnails column**, **Changing the unit of a numeric column**, **Combining columns**, and **Adding a formula column to calculate weight** sections of the article.

26.12.7 Adding new columns

Element properties can be added as columns in a BOM table.

Every property of an entity is listed under a property category in the **Properties** panel. To know which category a property belongs to, select in the drawing an entity having that property and locate it in the **Properties** panel.

For example, the **FrameDepth** property of a selected door can be found in the **Properties** panel under the **Parameters** category.

In the **BOM Manager** panel, the **Category selector** drop-down menu lists all the property categories available for the model. The **Column selector** drop-down menu lists all the properties belonging to the category selected in the **Category selector**.

To add the **FrameDepth** column to the table, follow the next steps:

1 From the **Category selector** drop-down menu, select the **Parameters** category.

2 From the **Column selector** drop-down menu, select the **FrameDepth** property.

3 (Optional) Select the column after which to insert the new column (if no column is selected, the new column is added after the last column).

4 Click the **Insert column after** button .

5 Click **OK** to save the table.

26.12.8 Adding a thumbnails column

1 Click the **Main menu** button (⋮) at the upper right-hand corner of the **BOM Manager** panel.

2 Select the **Insert thumbnails column** option.

The **Thumbnails** column is added with the **View type** property set to **Top Back Right view**.

3 (Optional) Change the **View type** property from the **Bill of materials properties** node in the property grid at the bottom of the **BOM Manager**.

4 Click **OK** to save the table.

26.12.9 Changing the unit of a numeric column

To change the unit of a column:

1 Click the header of the column to select it (for example, the **W** column in **Door Schedule**).

2 Go to the **Column properties** node in the property grid at the bottom of the **BOM Manager** panel.

3 (Optional) For the **Unit placement** property, select **In additional column**.

A separate **Units** column is added to the table next to the selected column.

4 For the **Units mode** property, select the **User defined** option.

A new **Units** property is displayed for the column under the **Column properties** node.

5 Repeat step 1.
6 For the **Units** property, select the desired unit from the drop-down menu.
7 Click **OK** to save the table.

26.12.10 Combining columns

1 Select the columns in the table by clicking on the column headers. Use the **Ctrl**/**Shift** key for multiple column selection.
2 Right-click a column header to display the context menu.
3 Select the **Add template column from selected** option.

   The **Template column** dialog box opens (see **Template column dialog box** article for more information).
4 In the **Title** field, type the name of the column.
5 (Optional) Delete the old columns.
6 Click **OK** to save the table.

26.12.11 Adding a formula column to calculate weight

You can use a formula column to calculate the weight of used materials by multiplying the **Ply Net Volume** property from the **Formula** category by the **Density** property from the **Physical Material** category.

1 From the **Category selector** drop-down menu, select **<Evaluated columns>** category.
2 From the **Column selector** drop-down menu, select **<Formula column>**.
3 Click the **Insert column after** button.

   The **Formula column** dialog box opens (see **Formula column dialog box** article for more information).
4 In the **Title** field, type the name of the new column (for example, **Weight**).
5 For the **Unit** field, select **Mass**.

   The second field auto-completes with **kg**. Change the unit as needed.
6 From the **Category selector** drop-down menu of the **Formula column** dialog box, select the **Formula** category.
7 From the **Column selector** drop-down menu of the **Formula column** dialog box, select the **Ply Net Volume** property.

   The formula field auto-completes with the string '<Formula:Ply Net Volume>'.
8 In the formula field type the multiply symbol ('*') next to the existing string.
9 Go back to the **Category selector** drop-down menu of the **Formula column** dialog box and select the **Physical Material** category.
10 From the **Column selector** drop-down menu of the **Formula column** dialog box, select the **Density** property.

   The formula field auto-completes with the string '<Density>' next to the existing string. The formula is now complete: '<Formula:Ply Net Volume>*<Density>'

11 Click **OK** to close the **Formula column** dialog box and to add the new column to the table.

12 Click **OK** to save the table.

26.13 Editing data extraction definition (.dxd) files

Note: Since V22, the syntax of DXD files uses version 2.0:

- The Paths section has been added.
- Logical operations use infix notation: (EntityType=="3D Solid") and (Type=="BIM_COLUMN") instead of prefix notation: AND (EntityType=="3D Solid") (Type=="BIM_COLUMN")

DXD (Data Extraction Definition) files are plain text files. You can edit DXD files created by the DATAEXTRACTION command using a text editor such as MS Notepad and use them in the Creating Schedules procedure in the **Project Browser**.

Lines starting with semicolon (;) are comments and ignored on parsing. Empty lines are ignored as well.

The file can contain 4 sections, each section name is written between square brackets, section names are English case-insensitive (ECI) strings. The sections are:

- **Paths**
- **Settings**
- **Entity Types**
- **Properties**
- **Filter**

26.13.1 The Paths section

Lists the paths to be searched for drawing files.

$CurDwg: current drawing

$CurProj: current project

26.13.2 The [Settings] section

The section contains values of the following parameters which control the behavior:

- **ExtractFromBlocks** - whether to take entities from blocks into account, considering all nesting levels.
- **ExtractFromXrefs** - whether to take entities from external reference blocks into account.
- **CountXrefs** - whether xrefs are taken into account at all.
- **CombineEqualRows** - whether to group equal rows in the output table; if yes, then the **Count** column is added, which indicates how many equal data rows are included.
- **IncludeNameColumn** - whether the output includes the **Name** column. The **Name** column indicates the source entity type: 3D Solid, 3D Solid Ply, ...

Parameter names are ECI strings. The format is:

ParameterName=value,

The value is either 1 or 0. No spaces should be written around the equal (=) character.

26.13.3 The [Entity Types] section

This section contains list of entity types which will be processed. The format of each line can be one of the following types:

EntityType | **DisplayEntityType**

- **EntityType** is the ECI entity type name, such as **Circle** or **AttributeDefinition**. This is the mandatory part of the line.
- **DisplayEntityType** is optional and can be any character string including spaces in any language. If present, this string appears in the **DATAEXTRACTION** dialog and in the output table.

Block | **BlockName** | **DisplayBlockName**

- **Block** is the predefined ECI word, and **BlockName** is case-insensitive name of the block to be processed, including **xRefs** and **MInsert**(s).
- **DisplayBlockName** is optional and has the same meaning and format as **DisplayEntityType**.

Block | *

This is a variant of the previous format meaning that all blocks will be taken into account; if present, then all other lines regarding blocks in this section are ignored. You can use the Filter section to filter the content to be processed.

*

All entity types and blocks will be taken into account, all other lines in this section are ignored then. You can use the **Filter** section to filter the content to be processed.

26.13.4 The [Properties] section

This section contains a list of all properties to be included in the data extraction output. The format of each line can be one of the following types: **Entity Property**, **Attribute**, **Drawing**, **Formula** or **Block View**.

EntityPropertyName | **DisplayEntityPropertyName** | **ValueOutputFormat**

- **EntityPropertyName** is the ECI name of entity property, e.g. **Lineweight**, **Color**, **Center Y** and so on. Note, all names do not contain spaces except those which represent coordinates of 2D or 3D points or vectors. In the last case, coordinate letters X, Y and Z should be separated by single space from the general property name.

  EntityPropertyName is the mandatory part of the line, the remaining part is optional.
- **DisplayEntityPropertyName** is user-defined and can be in any string including spaces in any language. By default, it is equal to **EntityPropertyName**.
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

Attribute | **AttributeTagName** | **DisplayAttributeTagName** | **ValueOutputFormat**

- **Attribute** is the predefined ECI word.
- **AttributeTagName** is the ECI attribute tag name as it is stored in the drawing.
- **DisplayAttributeTagName** has the same meaning as for **EntityPropertyName** (see above).

- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

Drawing | **DrawingPropertyName** | **DisplayPropertyName** | **ValueOutputFormat**

- **Drawing** is a predefined ECI word.
- **DrawingPropertyName** is an ECI name, which can be one of the following:
  - **Comments**
  - **DrawingRevisionNumber**
  - **FileAccessed**
  - **FileCreated**
  - **FileLastSavedBy**
  - **FileLocation**
  - **FileModified**
  - **FileName**
  - **FileSize**
  - **HyperlinkBase**
  - **Keywords**
  - **Subject**
  - **Title**
  - **TotalEditingTime**
- **DisplayPropertyName** has the same meaning for **EntityPropertyName** (see above).
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

Formula | **EntityPropertyName** | **DisplayEntityPropertyName=FormulaExpression** | **ValueOutputFormat**

- **Formula** is a predefined ECI word.
- **EntityPropertyName** is a unique name for this formula item.
- **DisplayEntityPropertyName** is an optional output name of the property, if it is omitted, then **EntityPropertyName** is used as the table column title.
- = is a mandatory sign, which must exist even if **DisplayEntityPropertyName** is omitted.
- **FormulaExpression** is the expression in format as in \AcExpr formula fields, which can contain constants such as PI or functions such as SIN (see Using Expressions). An operand in an expression can be a **DisplayPropertyName** enclosed in arrow brackets (<>).

  For instance: =2 * PI * <Radius> calculates the circumference of a circle.
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

BlockView | **BlockViewPropertyName** | **DisplayBlockViewPropertyName** | **BlockViewOptions**

- **BlockView** is a predefined ECI word
- **BlockViewPropertyName** is a unique name for this block view item

- **DisplayEntityPropertyName** is the output name of the property; cannot be omitted
- **BlockViewOptions** is a string in the following format:

  ViewType;DwgPath;SectionName

  - **ViewType** is one of the following: **Front**, **Top**, **Left** or **Right**; cannot be omitted.
  - **DwgPath** is path to dwg file, may be relative with respect to the dxf file; can be omitted.
  - **SectionName** is the name of the section to extract the section settings from for the block view construction; can be omitted.

If **DwgPath** or **SectionName** are omitted or not found, default settings will be applied.

An example:

BlockView|BlockViewProperty1|Elevation symbol|Front;../Sections/_SectionSettings.dwg;_BlockViewSettingsSection

*

If present all entity properties are taken into account; all other lines in this section, except Drawing and Formula lines, are ignored.

Note: Empty fields search is " " (space required)

26.13.5 The [Filter] section

Syntax description:

Relational operators:

Operator	Meaning
==	Equal to
!=	Not equal to
<	Less than
>	Greater than
<=	Less than or equal to
>=	Greater than or equal to

- Relational expressions are enclosed in '()' brackets.
- Logical operators AND and OR are case-insensitive.
- Logical expressions use an infix notation.
- Tokens can be separated by zero or any number of spaces, except logical expression begin,
- example:

(Layer == "MyLayer") and (Thickness<=3)

- Expressions can be split into any number of lines; tokens are not broken.
- Use the **EntityPropertyName** to refer to properties (not the **DisplayEntityPropertyName**).
- Lines can be commented with ';'.
- String values are enclosed in double quotation marks "".

(Type == BIM_WINDOW) or (Type == BIM_DOOR)

The result of the filter in the above example is that only BIM entities classified as windows or doors are included in the data extraction. See the BIMCLASSIFY command to learn more about BIM entity types.

The filter expression is either a single relational one or several ones grouped by logical operations.

Example:

((0) and ((1) or (2))) or ((3) and (4) and (5)) or (6)

where (0),...,(6) are relational expressions such as (Radius >= 3).

```
              OR
              /|\
       -------  |  ------
      /         |        \
   AND         AND       (6)
   / \        / | \
 (0)  OR   (3) (4) (5)
      / \
    (1) (2)
```

26.14 Working with Point Clouds in BIM

26.14.1 About point cloud scan to BIM workflow

The point cloud Scan to BIM semi-automatic workflow consists of four steps:

1. Detecting floors
Use the POINTCLOUDDETECTFLOORS command to detect the floors. A floor number is assigned to each selected point in the point cloud.

2. Detecting rooms
Use the POINTCLOUDDETECTROOMS command to detect rooms. A room number is assigned to each selected point in the point cloud.

3. Creating room solids
Use the POINTCLOUDFITROOMS command to create room solids based on the previously found rooms.

4. Creating BIM components
Use the BIMINVERTSPACES command to create walls in between the selected solids, wall openings, outer walls and slabs.

For more information about the workflow, see the **Point cloud Scan to BIM workflow** article.

26.14.2 About POINTCLOUDFITPLANAR

POINTCLOUDFITPLANAR is used to semi-automatically create 3D geometry based on a point cloud scan. It will create a planar surface or solid after selecting one point in a point cloud. The points that seem to be in a plane are never exactly in one plane, therefore a threshold value is set as a property of the point cloud entity. This also works in bubble view.

Select planes in the point cloud to generate 3D solids based on the given geometries. These 3D solids can be further used in the process of creating a full BIM model.

For more information about this command, visit the Command reference article POINTCLOUDFITPLANAR.

Creating 3D solids from the bubble view

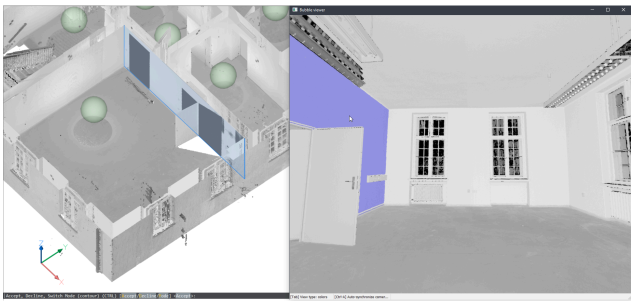
If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click, a preview in both bubble view and model view is displayed. You can switch between 2 shape representations using the CTRL-key.

1 Double-click a green point cloud bubble to open the Bubble viewer.

2 Type POINTCLOUDFITPLANAR in the Command line.

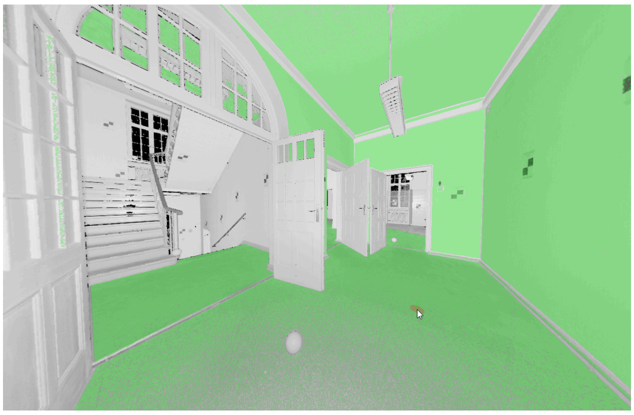
3 Select the desired wall.

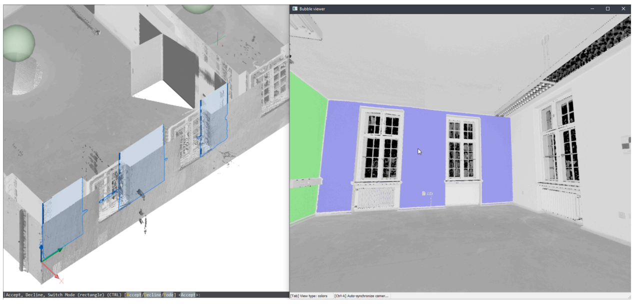
4



5 Press A to accept the selection.

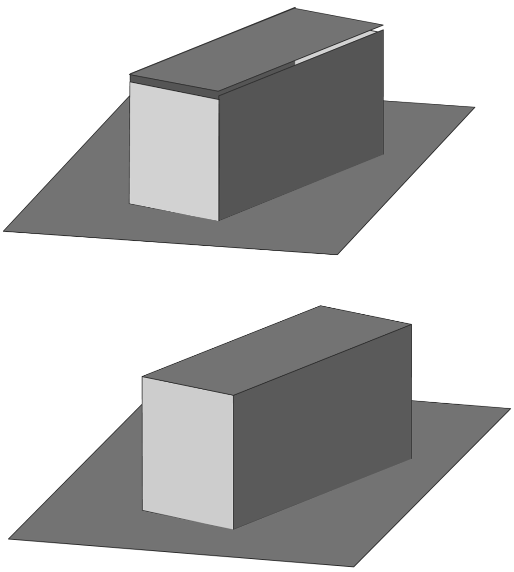
6 (Optional) Select more walls.





7 Close the Bubble viewer.

8 The surfaces are created and you can edit them to closed spaces with LCONNECT or other connecting tools.



Creating 3D entities from the model space

You can also use this command in the model space when the Bubble viewer is not open. BricsCAD will ask you to select a point of the point cloud in model space. Depending on the size of the cropped point cloud, it takes longer but it has 2 advantages by searching multiple scan positions:

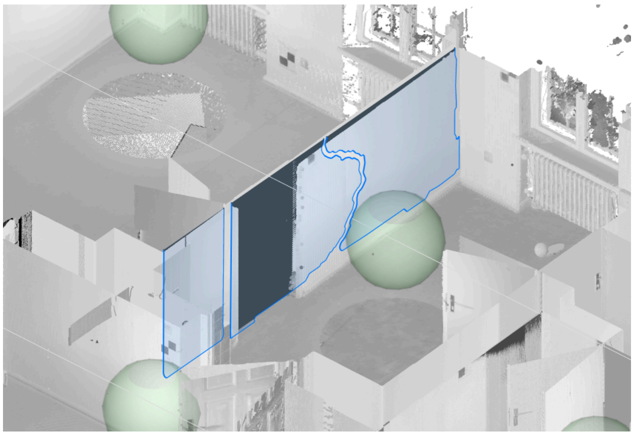
- It can create larger surfaces where only parts are visible in each scan position.
- It can detect wall and slab thickness since it can take the opposite surface into account.

1 Type POINTCLOUDFITPLANAR in the Command line.

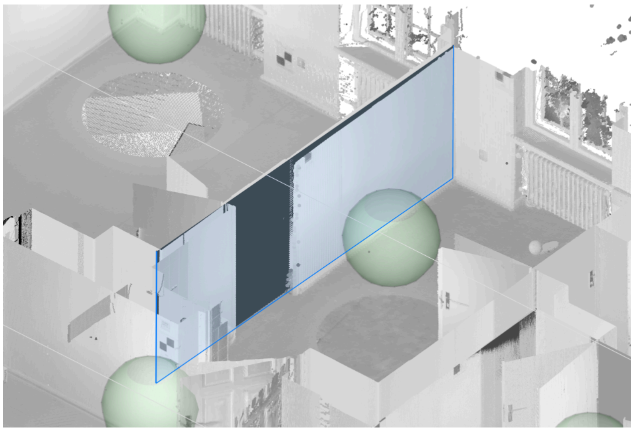
2 Select a wall where you want to create a plane or solid.

3 You are prompted: Accept, Decline, Switch Mode (contour) (CTRL) [Accept/Decline/Mode]:
   - Press A to create the wall.
   - Press D to decline.
   - Press M or CTRL repeatedly to switch between different modes.

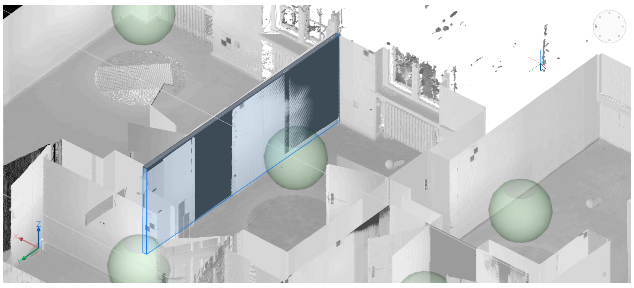
   Contour mode will create the best fitting plane:



   Rectangular mode will create a rectangular plane around region of planar points:



   Solid mode will create a solid from two parallel point cloud planes:



Note: If the point cloud consists of multiple rooms, you can use the solid mode to automatically create solid walls.

4 Press A to create the wall.

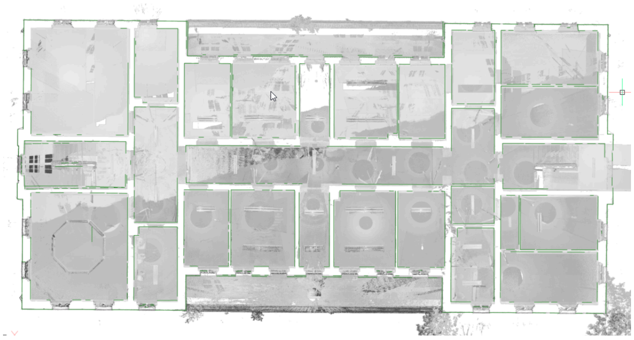
These solids can be the base for a more complex BIM Model. Use modeling commands to complete the building and add BIM data with **Bimify**, **Automatch** and **Propagate** to quickly transform your point cloud into a complete BIM Building.

It may be helpful to turn on the **Pointcloud Nearest Point ESNAP**setting (PCNEAREST command/3DOSMODE system variable). This snap setting significantly improves your ability to select relevant point cloud points.

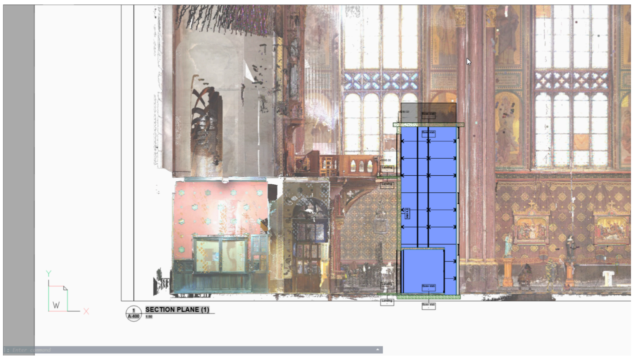
26.14.3 About POINTCLOUDPROJECTSECTION

The POINTCLOUDPROJECTSECTION command automatically generates a raster image with optional contour lines out of a defined section box. These lines can be easily manipulated into a detailed 2D drawing.

The POINTCLOUDPROJECTSECTION command enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections with the BIMSECTION command. You can use these sections to generate 2D lines to create a 2D floor-plan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way, it is possible to run this command in full resolution on all sections.



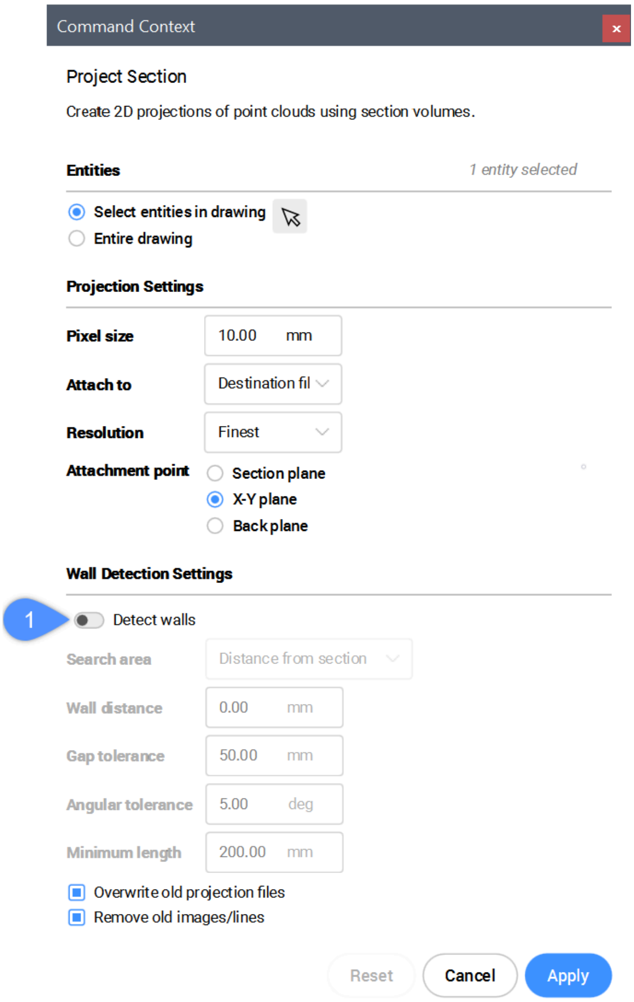
At the same time, a raster image is generated to give the user some context. In some cases, it is not necessary to recreate the existing building. Background images can give much more context to the design documents. These can be used to verify the created 2D geometry but in high quality scans, these images can also be used as graphical material. For example, as a background image for a BIM model in renovation projects where modern intervention are made in historical buildings.



For more information about this command, visit the Command reference article POINTCLOUDPROJECTSECTION.

26.14.4 Creating a raster image from a point cloud

1. Type POINTCLOUDPROJECTSECTION in the Command line.
2. The Command Context panel opens.
3. Edit the options to customize your projected section. Turn the **Detect walls** (1) option off.



4 Click **Apply** to generate the raster image.

Point Cloud scan by Realistic Visual styles.

26.14.5 Detecting walls from a point cloud and generate in 2D

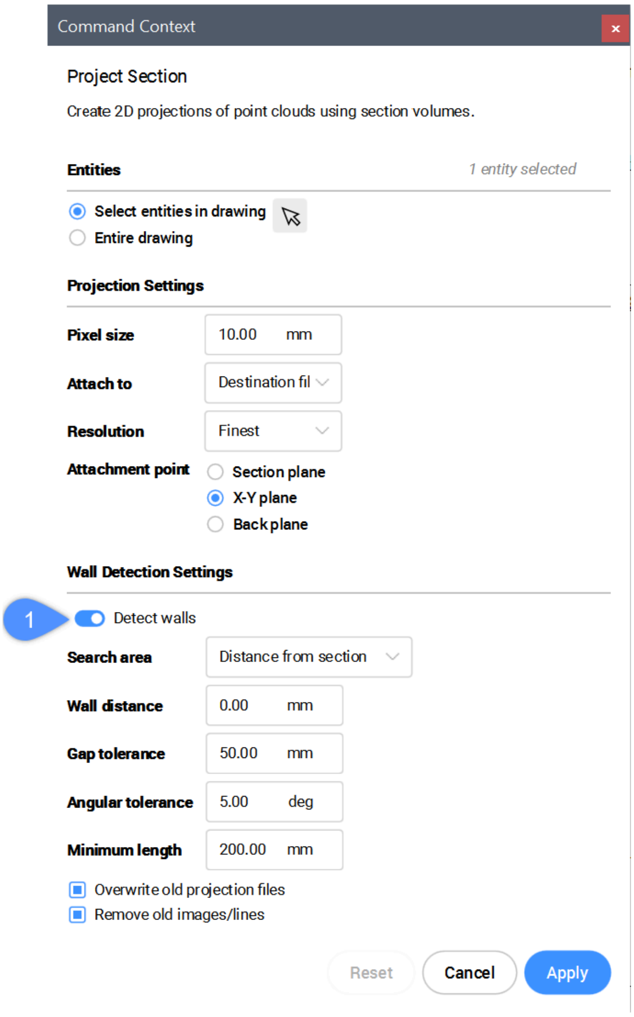
Making horizontal sections from point clouds

1 Type POINTCLOUDPROJECTSECTION in the Command line.

2 The Command Context panel opens.

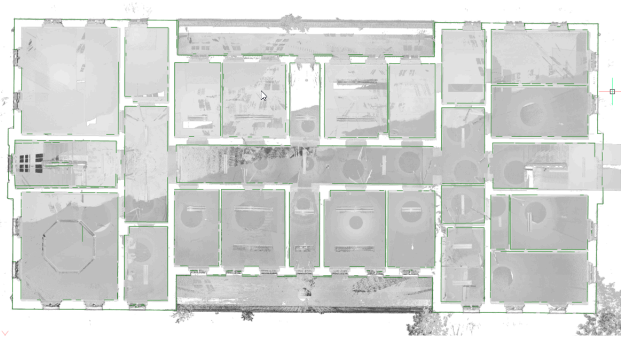
3 Edit the options to customize your projected section.

Note: Turn on the **Detect walls** option (1) to generate the green lines as seen in the image below.



4 Click **Apply** to generate the raster image.

Note: If there are some missing lines, you can draw them yourself by tracing the background image.



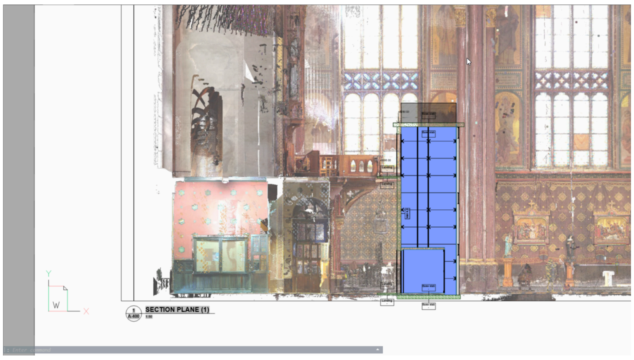
Making vertical sections from point clouds

Make a vertical section box on the location where you want the vertical projection.

Make sure that the blue round arrow aims to the side you want your projection.

26.14.6 Generating BIM sections with point clouds

In some cases it is not necessary to recreate the existing building. Background images can provide more context to the design documents. You can use these images to verify the created 2D geometry and in high quality scans, you can also use them as graphical material. For example, as a background image for a BIM model in renovation projects where modern interventions are made in historical buildings.



Point Cloud scan by Realistic Visual styles.

26.15 Working with Typed Plans

Shape Lite Pro Mechanical BIM

26.15.1 About Typed Plans

BricsCAD BIM uses drawing templates to customize your generated drawings. For a general understanding of how these templates are created and work, please visit the **Templates for generated drawings** article.

Typed Plans allow you to create customized templates for drawing types based on a set filter parameter rule against an entity type's value or property within the 3D BIM model. This is especially helpful for customizing graphics of model entities independent of their assigned layers or layer states, therefore transferable and usable on other projects with similar entity properties. This makes it easier to ensure adherence to office drawing standards, local government requirements and to achieve consistency across 2D drawings, especially on detailed design and building permit packages.

Typed Plans can be stored in two ways:

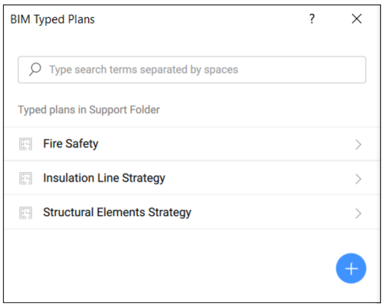
- At the level of the BIM project.
- In the support folder.

1 Typed Plans in a project can be managed in the UI of the **BIM Project Browser**. It is possible to create a new Typed Plan from the **BIM Project Browser**, to rename or remove a Typed Plan, or to edit an existing Typed Plan.

2 Typed Plans in the support folder cannot be managed from the **BIM Project Browser** as they don't belong to a project. Use the TYPEDPLANS command to manage them in the **BIM Typed Plans** dialog box.

26.15.2 BIM Typed Plans dialog box

1 To open the **BIM Typed Plans** dialog box, enter the TYPEDPLANS command in the Command line or click the Typed Plans icon in the Ribbon panel.



2 Click to create a new template. By default, the template is named **New Typed Plan** with the text highlighted.

3 Rename the template while the text is being highlighted. When finished, press Enter to save and apply the name.

Note: You can duplicate, delete or rename your typed plans. To do that, hover the cursor over the desired Typed Plan and click the context menu button to expand the options.

4 Edit a Type Plan listed in the dialog box by clicking its name to open the **BIM Typed Plan Editor** dialog box.

26.15.3 Typed Plans at the level of the Project browser

From the **BIM Project Browser** panel you can create new **Typed Plans** or import **Typed Plans**. To do that, go to the **Files** tab of the panel and click at the bottom right of the panel to reveal its options and choose one of the following:

New Typed Plan
Creates a new Typed Plan through the **BIM Typed Plan Editor** dialog box.

Import Typed Plan
Allows you to import an existing Typed Plan through the **Select a Typed Plan to import**dialog box. Typed Plans are stored as .tp folders that contain two JSON scripts (one to customize the tags and one to customize the generated drawings) and a Settings.dwg file (stores all the customizations settings referred to in the JSON scripts). You can browse to the location containing the desired .tp folder to be added.

Once created/imported, the typed plan is stored in the BIM project folder, in the subfolder named TypedPlans, and thus accessible through the **BIM Project Browser** panel's **Files** tab. Here several options are available by right-clicking a listed Typed Plan to open the context menu.

Open Folder
Opens the folder in the File Explorer of your computer.

Rename
Allows you to rename the selected Typed Plan.

Duplicate
Duplicates the selected Typed Plan using the default name 'Original Name - Copy'.

Delete
Deletes the selected Typed Plan from the project.

To edit one of the Typed Plan in the list, double-click the desired Typed Plan. This opens the **BIM Typed Plan Editor** dialog box.

Note: Typed Plans in the support folder can not be managed from the **BIM Project Browser** as they don't belong to a project.

26.15.4 Editing Typed Plans

The **BIM Typed Plan Editor** dialog box allows you to edit the customizations of the selected Typed Plan.

The first step in the template brings you to the definition of the overrides, where you can create overrides to change entity properties to customize the typed plan and add filters. A new override element is added to the list by clicking the **Create new override** button if no override is already created or **Add new override** button at the top of the overrides list.

The overrides list functions as a top down filter list. Starting at number one and working down to numbers 2, 3, etc., the list is checked and BricsCAD BIM determines whether an entity fits any criteria. If the entity falls into one filter, the other filters are no longer verified. Thus, the order in which you place the overrides in the list is very important. It is best to start with the most specific filter lines and then add more broader filter lines. For example you can leave your last filter line so generic to encompass all other elements, by not specifying any filter expression in it.

Note: User defined properties can be used as filter in Typed Plans only when the BIM Project uses a Project database stored as an external BSYSLIB file. For more information about the Project and Central databases, see the articles **BIM data structure** and **BIM project info**.

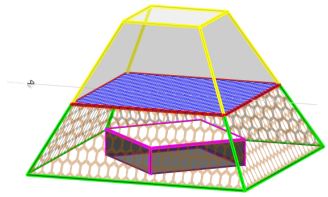
Drawing overrides refer to the visibility settings of how you would like to have your elements in your model drawing to look like.

There are 4 different types of customizations, which can be specified for each filtered group of elements separately. You can choose how the element should look like as a section result, how a centerline type representation should appear, how an external symbol representation should replace the original entity and you can modify and customize a specific ply layer of your desired entity composition, but only if appropriate, for example a **Wall**, **Slab** or **Roof** entity.

Section result customization
Click the context menu button to open the menu options.

Add customization
Click the arrow next to Add customization to open the drop-down list with customization types. Click a type to add it to the Section result customization so you can customize the lines for a more specific control of the section result graphics.

The diagram below illustrates how you could control the visibility and customization settings of an entity type with the **Typed Plans**.

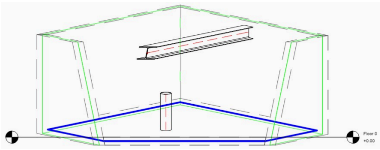


- The section plane is aligned with the blue square with red outline in this diagram.
- The green outline represents **Background**, which is essentially the elevation outline of the cut object.
- The red outline represents **Intersection Boundary**, which traces over the cut area.
- The blue hatch represents **Intersection Fill**, as it shows the area of being cut by the section plane.
- The orange hexagons hatch represents **Elevation Fill**, which refers to the area bound by the elevation outline (see **Background**).
- The pink outline represents **Hidden Lines**, as it is below the blue hatch.
- The yellow outline represents **Cut away geometry**, as it is above the section plane.

Center line customization

Centre Customization determines the appearance of the center of planar or linear entities. The resulting center lines, for example from cutting a wall's center plane in a floorplan, can be customized.

Center customization is particularly useful in scenarios where you would like to display the center elements of various planar and/or linear entities. For example, if you were to apply the center customization settings against a planar entity, for example a wall, you would be referring to the display behavior of its center plane. However, if you were to apply the same settings against a linear entity, for example a beam, you would be adjusting how the beam's center line appears on the sheet.



The diagram above illustrates an example of how the center elements differ between planar (wall center plane highlighted in green; slab center plane highlighted in blue) and linear entities (both column and beam center lines in dashed red lines).

There is no default setting to adjust the visibility of these center elements, and they are not normally displayed in section results.

Symbol customization

Click the context menu button to open the menu options.

In this instance, simply add a new **symbol layer customization** value containing part of the desired layer name. If they are part of the default **BRX_2D** layers, adding "BRX" as a name value will suffice. **Typed Plans** will now search within the component entity that the customization template is applied to. This method, however, will not allow any exchanging or sharing of 2D symbols across entities, for example replacing a Column with a symbol derived from a BRX layer stored within a (table) **Furnishing Element**.

For more information on how BRX layers operate, please visit the chapter **Procedure: Understanding the BRX_2D layers** in the **Templates for generated drawings article**.

Ply Customization
Can be used to show only a certain type of plies in a plan view. An easy example is the visibility of only structural plies in a structural plan. Click the context menu button to open the menu options.

Add ply customization.
Allows to filter entity plies by adding one or more parameter filters from the drop down list. A main categorisation for the customisation of plies is predefined as Structure function and No structure function. Filter rows can be added to filter by multiple parameter combinations.
For each Filter combination, ply customizations can be defined.

26.15.5 Drawing customizations styles

Once a customization is added using the steps mentioned before, the styles of selected entity types can be created and customized on the fly. Several characteristics can be specified.

Visibility
Sets the visibility of the entity type on or off.

Color
Specifies the color of the entity type.

Linetype
Specifies the linetype of the entity type.

Lineweight
Specifies the lineweight of the entity type.

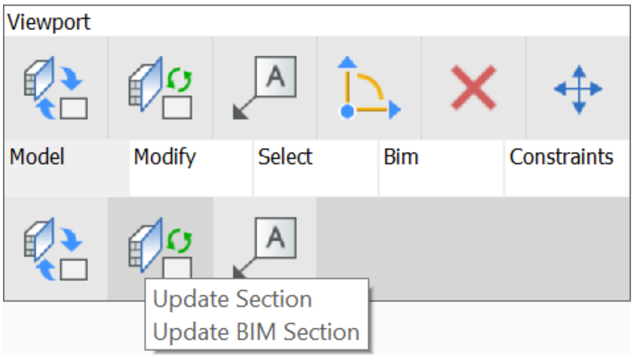
Linetype Scale
Specifies the linetype scale of the entity type.

26.15.6 Applying templates to viewports on sheets

With your **Drawing overrides**, **Tag overridesFilters** and **Styles** all set, your **Typed Plan** template is now ready.

1. To apply the template, navigate to your sheet drawing with your desired viewport.
2. Open the Properties panel and select the viewport to view its properties.

   Note: This can be done by using the PROPERTIES command or by clicking the Properties panel icon.
3. Navigate to the **Typed Plan** property under the BIM properties section and click the drop-down selection to the right to select the desired typed plan.
4. Once done, hover over your selected viewport to access the Quad. Click **Update Section** in the Quad under **Model** tab.



5 You should see a refreshed Viewport with the graphical changes as per your Typed Plans settings.

26.15.7 Accessing customization templates

By default settings, these templates are stored in the **Customizations** folder within your **Support** folder in the following path, under the **Bim** folder and then **Sections** subfolder:

C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\Bim\Sections \Customizations

The **AppData** folder is hidden by default, so you may have to unhide it first. This path can also be accessed by entering SECTIONSETTINGSSEARCHPATH in the Command line.

You can specify a different path if your Customization files are stored elsewhere in an accessible folder. Simply redefine the path in the **Section settings search path** in the **Settings** dialog box, or in the Command line.

New customizations created with BIM Typed Plans will be stored in a **Customizations** subfolder in the specified path. Each customization template is saved as an individual folder which contains in its basic form **Settings.dwg**, a **tag-customization.json** and a new **drawing-customization.json** file. The folder also contains any external symbol sources you created using the **Create new external file...** button.

Settings.dwg contains the values created in the Customization sections in the **Drawing Override** tab in the form of layers and if you have created any custom properties that you wish to use as a **Filter** parameter, you have to ensure they are also reflected in the **Settings.dwg.**

Note: As these **Customization** folders are stored locally in the machine you have created it with, you need to ensure the folders and their contents within are copied and pasted into the same location on other machines, should you wish to share it with other users on other machines.

27. Getting Started

27.1 Mechanical workspace

27.1.1 Start a mechanical design project

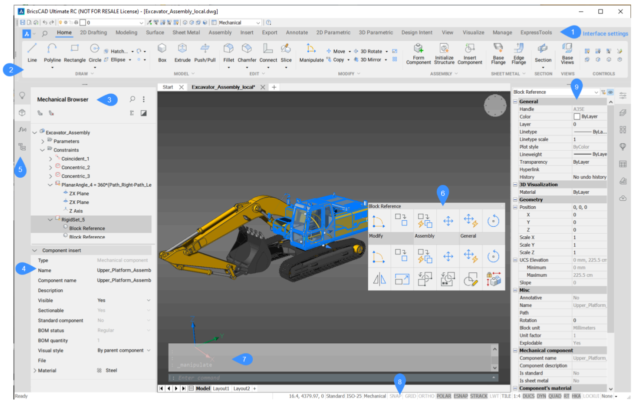
You can start a mechanical design project:

1 Launch BricsCAD.

The *BricsCAD Launcher* window appears.

2 Click **Start** in the **Mechanical Design** section.

27.1.2 BricsCAD mechanical application window



1 **Ribbon tabs**: click to switch to a particular tab.

2 **Ribbon panel**: every tab contains several panels, which combine similar tools.

3 **Mechanical Browser**: browse through a tree-like representation of your model, with the possibility to highlight a particular node in the model area and edit its mechanical properties.

4 **Mechanical Properties** of a particular node: some are read-only. Check the properties of the root node in the tree.

5 **Dockable tool panels**: **Mechanical Browser** and **Library**.

6 **Quad cursor menu**: apply a particular tool when you hover over an affected entity.

7 **Command line**: type in the command.

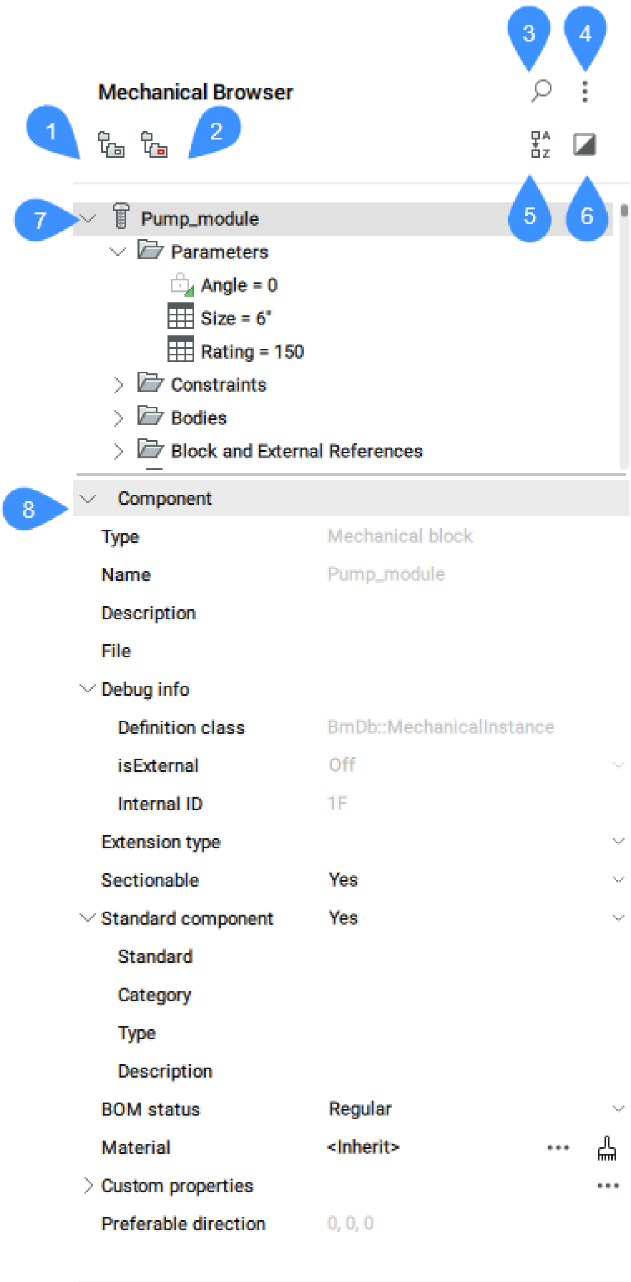
8 **Status bar**: status and switch hot options.

9 **Properties panel**: adjust non-mechanical properties of selected entities, such as color and layer.

27.2 Mechanical browser panel

To open the **Mechanical Browser** panel, run the MECHANICALBROWSEROPEN command or right-click a ribbon panel/toolbar and choose **Mechanical Browser** from the **Panels** list.

The **Mechanical Browser** panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.



1 Group by entity
2 Group by type
3 Show search
4 Settings
5 Sort Alphabetically
6 Show selected
7 Mechanical entities list

27.2.1 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

27.2.2 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

27.2.3 Show search

Opens the search field and the associated tools.

Search field
Searches the mechanical browser tree for the character string you enter. The search acts as a filter, so that the browser shows all components whose names or component names (or part of the names or component names) match the character string. Press **X** on the right of the search field to close the field.

27.2.4 Settings

Expressions of constraints
Controls whether the numeric value or the assigned parameter name displays.

Components parameters
Controls the visibility of sub-component parameters.

Expressions of components parameters
Controls the visual representation of sub-component parameters: numeric value or assigned parameter name.

Sub-components of standard parts
Controls the visibility of sub-components of standard parts which are mechanical assemblies.

Arrays of entities
Toggles the display of Array nodes in the browser tree, including Components arrays.

Block and External references
Toggles the display of Blocks and External References nodes in the browser tree. Also, toggles node building in the **Bodies** folder for Block entities.

Always synchronize selection
Toggles the synchronization of the selection nodes in the **Mechanical Browser** with the selection of the corresponding objects in the model space. When an object is removed from a selection, only nodes corresponding to the objects remaining in the selection remain selected. If no objects are selected, the root node will be selected (similarly to when a document is open).

Keep values list order
Toggles the sorting of the values list for parameters inside the drop-down list.

Highlight selections
Toggles the highlighting of the selected entities in the drawing.

Asynchronous properties load
Controls if the **Mechanical Browser** collects properties asynchronously if more than one node is selected in the Mechanical Browser tree.

Expose all solids
Toggles the display of solids for mechanical blocks and solids not related to constraints under the **Bodies** folder.

Note: **Mechanical Browser** settings are controlled also by the MECHANICALBROWSERSETTINGS system variable.

27.2.5 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

27.2.6 Show selected

When enabled, only nodes corresponding to selected objects or their parents are displayed.

When enabled and no objects are selected, or selection is cleared, the full model is shown in the browser.

The state of the **Show selected** mode is remembered between sessions, and is the same for all opened documents.

27.2.7 Mechanical entities list

Displays a hierarchical tree list of mechanical entities, such as user parameters, constraints, parts, arrays and features. Top-level nodes can have sub-nodes. For example, constraint node have sub-node(s) for its argument(s). An array node has sub-nodes for its arguments, etc.

Double click a node to expand or collapse it, double click (click-pause-click) a node to edit its label if applicable. Not all node names can be edited.

Each node in the list has associated context menu. Right-click a node to invoke the menu. The content of the menu can be different for each kind of node. You can also select a group of nodes and invoke a context menu for them. The menu will contain only those elements which are applicable to each selected node.

Context menu for the root component contains the following items:

Update
Updates the hierarchy of mechanical components for the current drawing in case the referenced drawing files of sub-components have been modified.

Visual style > All by Viewport
Applies the current visual style to all components in the assembly.

Switch all to local
Converts all external components in the model to local components.

Switch all to external…
Switches all internal components to external components.

Add new parameter
Creates a new parameter in the assembly.

Custom properties
Opens the **Mechanical Properties** dialog box to define custom properties for the drawing.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same name and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlight all components with the same name and the same parameter values.

Create sequence
Creates a block with a representation of the current assembly.

Renumerate child nodes
Renumerates all child nodes according to their types.

Renumerate child nodes continuously
Renumerates all child nodes with continuous numeration when applicable.

Collapse all
Collapses the main component and all components and subcomponents.

Expand all
Expands the main component and all components and subcomponents.

Context menu for child (non-root) components, internal or external, also offers:

Open
Opens the referenced drawing (see the BMOPEN command).

Open a copy
Opens a copy of a component insert as a new drawing (see the BMOPENCOPY command).

Update
Reloads all referenced components from external files and updates BOM tables (see the BMUPDATE command).

Replace…
Replaces a component insert (see the BMREPLACE command).

Note: Replacing a local insert turns it into an external insert.

Replace all inserts...
Replaces all inserts that refer to the same source (see the BMREPLACE command).

Switch to local
Switches an external component to an internal component (see the BMLOCALIZE command).

Switch to external
Switches an internal component to an external component (see the BMEXTERNALIZE command).

Dissolve
The selected feature is removed from the part, but it will keep its geometry. However, the design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

Custom properties
Opens the **Mechanical Properties** dialog box to define custom properties for the drawing.

Reset instance custom properties
Resets instance custom properties to defaults inherited from a corresponding block.

Hide / Show
Hides / Shows the component.

Visual Style
Sets a visual style for the component.

Zoom to
Zooms to the selection set.

Select
Selects the component.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same definition and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlights all components with the same definition and the same parameter values.

Delete
Analogue to the SMDELETE command. In this case, the feature is removed from the browser and the geometry is changed depending on the type of the feature.

Set material to component
Opens the **Physical Materials** dialog box, which allows you to assign a physical material to the component.

Remove material from component...
Removes the physical material definition from a local component.

Make component standard / Make component non-standard
Changes the type of the component.

Include all inserts to section
Sets the **Sectionable** property of all similar inserts to **YES**. Defines whether an insert is affected by the VIEWSECTION command.

Exclude all inserts to section
Sets the **Sectionable** property of all similar inserts to **NO**. Defines whether an insert is affected by the VIEWSECTION command.

Renumerate similar nodes
Renumerates all nodes of the same type and the same level.

Component's BOM status
Controls the appearance of the component in BOM tables.

Context menu for Parameters contains additional tools:

Geometry-driven
If ticked, it makes the parameter geometry-driven.

Note: If the parameter is assigned to a geometry-driven 3D constraint, its boundaries are taken into account while moving the constrained geometry.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates models by means of parameters.

Context menu for Component parameters contains additional tools:

Link to parameter
Links subcomponent parameter to the main level parameter.

Context menu for Constraints contains additional tools:

Enabled
Toggles the enabling of the constraint.

Flip side
Flips the side of the constraint (not available for fix constraints).

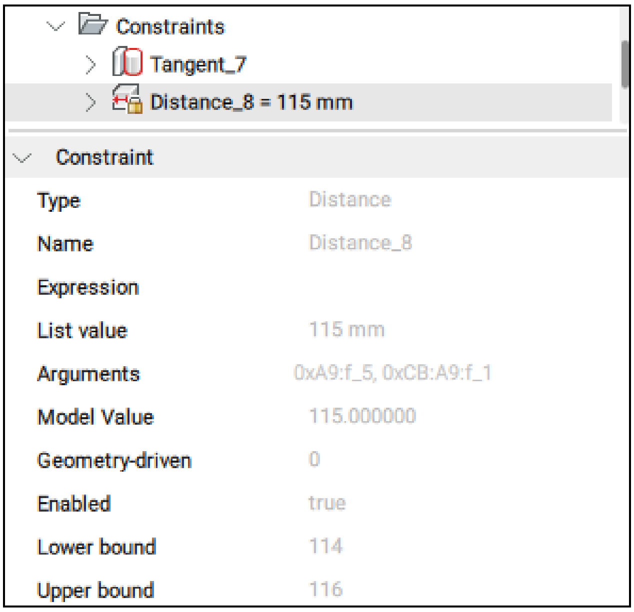
Select geometry
Selects the entities involved in the constraint.

Context menu for dimensional Constraints contains additional tools:

Geometry-driven
If ticked, it makes the constraint geometry-driven.

Note: If the parameter is assigned to a geometry-driven 3D constraint, its boundaries are taken into account while moving the constrained geometry.

Note: You can set upper and lower bounds in the **Mechanical Browser** panel, which are taken into account in dynamic operations, like DMMOVE and ROTATE3D.



Create design table
Creates a design table to drive parametric block parameters (see the **Parameters Manager** panel).

Context menu for constraint arguments contains additional tools:

Flip
Reverses direction.

Replace...
Replaces the current argument with a another one (see the DMCONSTRAINT3D command).

Exclude
Excludes the current argument.

27.2.8 Properties

Displays the properties of the selected item(s). If several nodes are selected in the tree, the Properties section displays the properties of all the selected nodes. Changing a property modifies all the selected nodes.

Change a property by clicking a numeric field to edit its value or by selecting another option from the drop-down list, which is opened when you press the down arrow from the right side of the field.

The most important properties of a mechanical block are:

- **Extension type**: defines the extension type for the mechanical block.
- **Sectionable**: defines if the mechanical block is sectionable.
- **Standard component**: defines if the mechanical block is a standard component.
- **BOM status**: defines the BOM status.
- **Material**: defines the material.

  Note: Click the browse button (…) to open the **Physical Materials** dialog box.

- **Custom properties**: displays the custom properties.

Note:

- Click the browse button (...) to open the **Mechanical Properties** dialog box.
- Click the **Clear material** button to remove the material.

- **Preferable direction**: indicates the direction for exploded views, which is set with the BMEXPLODECONFIG command.

Note: Select a Sheet Metal Body to display the **Measurements** properties in the **Mechanical Browser**.

Measurements	
Bend length	239.91 mm
Cut length	153.64 mm
Junction length	153.64 mm
Miter length	0.00 mm

27.3 3D comparison

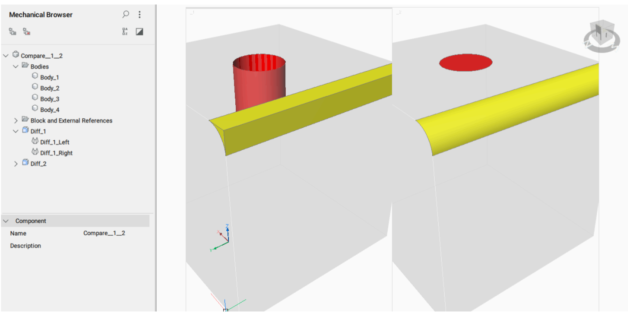
Command: 3DCOMPARE.

You can compare two 3D models, for example 3D solids and surfaces stored in 2 DWG files, including models imported in BricsCAD from other file formats. The tool can process large models and reports the differences in the **Mechanical Browser**. Multiple visualization modes are supported.

27.3.1 Definitions

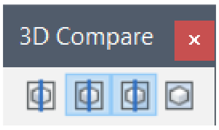
A 3D model is a DWG file with one or more 3D solids or surfaces. When comparing two 3D models, BricsCAD first tries to automatically align them in 3D space to obtain the maximal similarity. Next, all faces of these 3D solids and surfaces are compared geometrically with a given tolerance. If for every face in the first file an equal face in the second file exists and vice versa, such models are considered as equal. However, if for a particular face in one file there is no equal face found in another file, or if only part of this face is equal to a part of a face in the other file, a difference is detected. Therefore, a difference is a full face or a part of a face of a 3D solid or a surface, for which there are no equal parts of full faces found in the other file. Two or more differences are grouped together if they are geometrically adjacent to each other.

A difference node is created in the **Mechanical Browser** panel for every group of adjacent differences. If you expand this node, you can see the particular differences it consists of. When you select the difference in the **Mechanical Browser** panel, the corresponding feature is highlighted in the drawing.



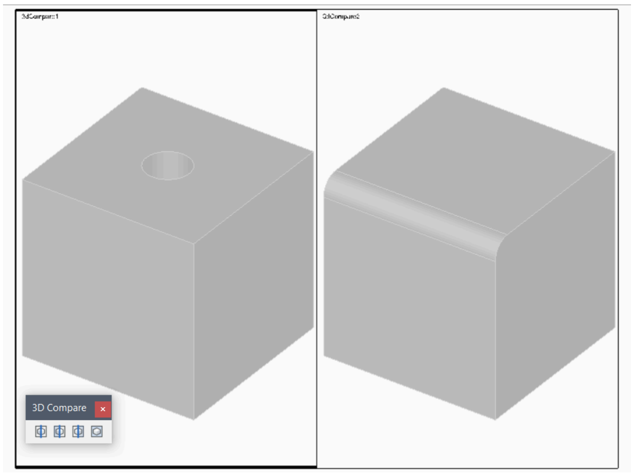
27.3.2 Toolbars

You can access the 3D Comparison procedure by using the tools from the **3D Compare** toolbar.



27.3.3 Visualization

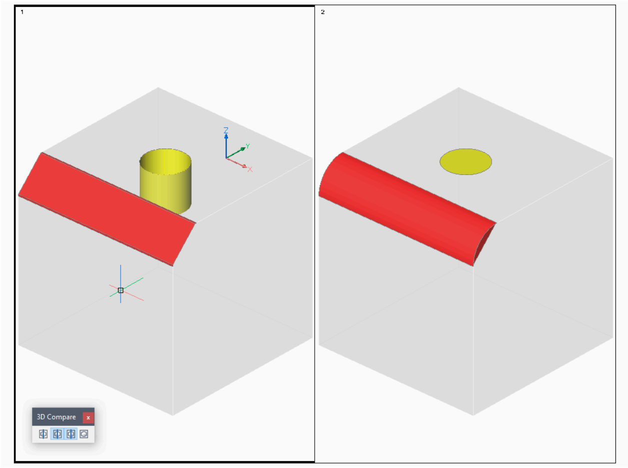
Use the 3DCOMPARE command to create a new DWG document named **Compare_<File_1>_<File_2>.dwg**. The compared models are attached as external references to this drawing. A special layout named **Comparison** is created automatically with 2 viewports, one for each drawing, illustrating the differences between the drawings by using colors.



The display is controlled via the 3DCOMPAREMODE system variable. If the value of 3DCOMPAREMODE is different from 0 (zero), the corresponding model becomes semi-transparent, while the differences are displayed using different colors. Each difference is painted in its individual color. You can zoom in on a particular difference using the **Mechanical Browser** context menu.

To display the differences:

1 Click the **Left Viewport** tool button () and/or the **Right Viewport** tool button ().



2 Double-click in one of the viewports.

3 Use the mouse/keyboard shortcuts to manipulate the display.

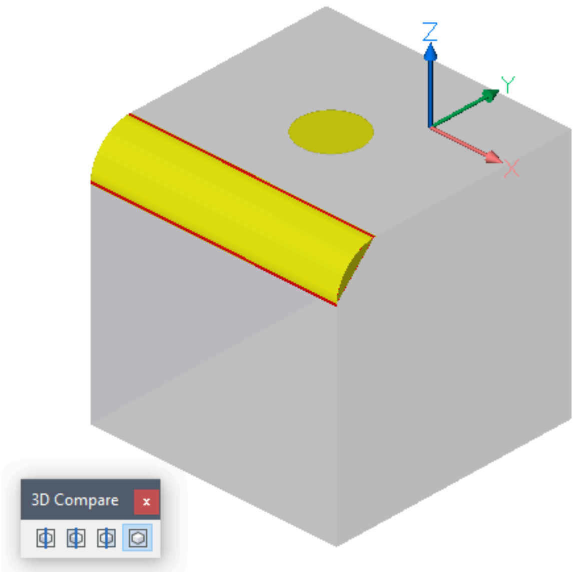
4 Example: Zoom in/out or rotate.

The display in the other viewport is synchronized automatically. This allows you to compare the same part in both models in detail.

As an alternative, you can:

Click the **Single Viewport** tool button ( ).

The differences in model space are displayed.

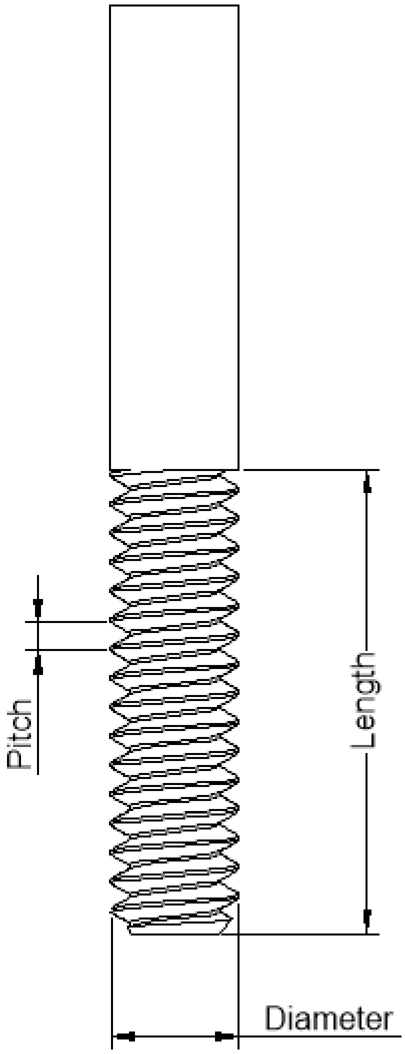


28. Part Design

28.1 Thread feature

Use the DMTHREAD command to create a thread feature.

The following picture shows the thread structure:



A thread feature has the following parameters:

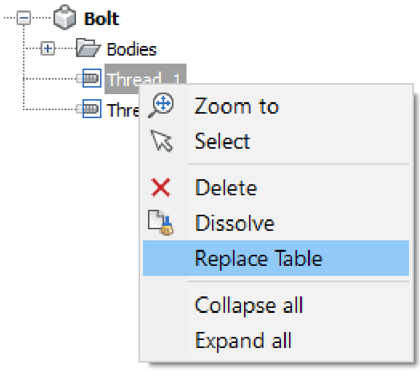
- **Thread standard**

  It is a set of standard threads. This list can be extended with a custom thread. When you change the standard, the geometry will be rebuilt automatically.
- **Pitch** of the thread.
- **Length** of the thread.
- **Diameter** of the thread.

- **Type** of the thread: male or female. If the type is set as auto, the type will be determined automatically.

28.1.1 Adding a thread configuration

1 Right-click on the feature in the **Mechanical Browser** panel.



2 Select **Replace Table**.

3 Select a CSV file with the following configuration:

```
Size,Diameter,Pitch Name1, Diameter_value1, Pitch_value1 Name2,
 Diameter_value2, Pitch_value2 Name3, Diameter_value3, Pitch_value3
```

28.2 Part Design Methods

BricsCAD Mechanical allows you to create and modify 3D models.

Ways to create a 3D model:

- Create a 3D solid primitive, such as a box and a cylinder.
- Sketch a 2D profile and extrude or revolve it to make a 3D solid.
- Model a 3D surface and thicken it to make a 3D solid, or stitch several surfaces together to create a watertight 3D solid.
- Import existing 3D solid and surface geometry from other CAD programs.

Ways to edit and add more details to your 3D models:

- By pushing/pulling, moving and rotating faces.
- By creating Fillets or Chamfers on the model edges.
- By applying Holes.
- By adding protrusion and depressions, and by using Boolean operations, Direct Editing and Deformable Modeling tools.

No parent-child dependencies = history-free approach

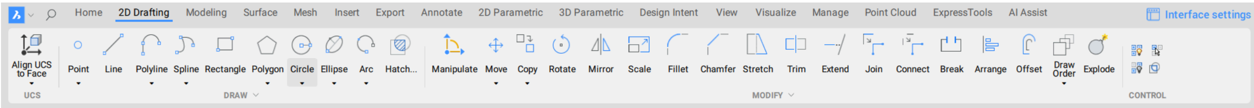
When CREATESKETCHFEATURE system variable is OFF, BricsCAD Mechanical does not create parent-child dependencies between geometric primitives and features. All changes made to your part are local; they do not force geometry regeneration. This history-free approach allows you to focus on your design instead of thinking about how it can be changed in the future.

3D Constraints = Parametric

When CREATESKETCHFEATURE system variable is ON, BricsCAD Mechanical provides a powerful toolset to make your part parametric by adding 3D constraints to its faces and edges and link them with global parameters using expressions and design tables.

28.3 Sketching

The sketching toolset allows you to draw 2D profiles, which can be extruded or revolved to create 3D solids and surfaces, or to add/subtract volume to/from existing entities. Depending on the CREATESKETCHFEATURE system variable value, the created 3D solids will be parametrized. The sketching tools are available on the **2D Drafting** tab of the **Mechanical** workspace ribbon:



Sketches are created in the XY plane of the current User Coordinate System (UCS). Since the dynamic UCS is enabled by default, you can create 2D entities on the 3D solid face under the cursor. Use the **Align UCS to Face** tool to position the UCS according to your needs.

The drawing tools include familiar 2D primitives, such as rectangle/polygon, circle/arc/ellipse, polyline, or spline.

Use the **Move**, **Copy**, **Rotate**, **Mirror** and **2D Array** tools to manipulate the sketches.

Use commands FILLET, CHAMFER, STRETCH, TRIM, JOIN, CONNECT, BREAK, OFFSET, and EXTEND to edit the sketches.

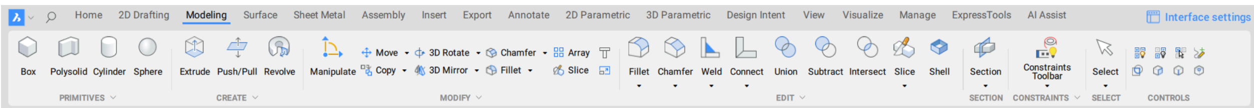
Use 2D Constraints to make the sketches parametric.

Note: The value of the DELOBJ (Delete Entity) system variable controls whether sketching entities are deleted when used by DMEXTRUDE or DMREVOLVE commands to create 3D solids or surfaces.

28.4 Solid Modeling

BricsCAD Mechanical provides a rich toolset for solid modeling, including traditional 3D modeling tools like **Extrude** or **Subtract** and intuitive Direct Editing tools like **Push/Pull**, **Copy Faces** and powerful 3D Constraints, which allow you to adjust the shape and size of your 3D part. Also, when the CREATESKETCHFEATURE system variable is ON, the created 3D solids are parametrized, and sketch features are visible in the **Mechanical Browser** panel.

3D solids tools are available on the **Modeling** tab of the **Mechanical** workspace ribbon:



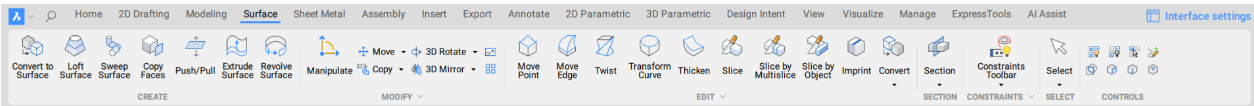
Tool/toolset	Description
Extrude	Create 3D solids from sketches.
Revolve	
Loft	
Sweep	
Thicken	
Primitives toolset: **Box**, **Polysolid**, **Cylinder**, **Sphere**, **Wedge**, **Pyramid**, **Cone**, and **Torus**	Create 3D solids directly, without underlying sketches.
Push/Pull	Edit 3D solids (applied to faces and/or edges).
Move	
3D Rotate	
Copy Faces	
Fillet	
Chamfer	
Shell	Edit 3D solids.
Slice	
Boolean tools:**Union**, **Subtract**, **Intersect**, and **Separate**	
Scale	
3D Mirror	
Array	

Tool/toolset	Description
Erase	Can be applied to the whole 3D solid, and to a set of its faces forming a feature (for example a fillet or a hole).
Audit 3D	Find and fix errors in imported geometry.
Simplify 3D	Automatically recognize canonical faces (planar, cylindrical, conical, spherical, toroidal) from spline faces of your 3D solid. This is often needed before applying 3D constraints.
Stitch	Stitch touching surfaces and regions, that form a watertight boundary, into a single 3D solid.
Twist, **Move Point**, **Move Edge**, and **Transform Curve**	Deform your 3D solid uniformly.
Select toolset	Select faces and edges of your 3D solids based on particular geometric and topological criteria, such as length, area and smoothness.
Constraints	Add 3D constraints to faces and edges of 3D solids, to align and resize them. Make your 3D part parametric. Automatically apply geometric 3D constraints, such as parallel, perpendicular, coaxial, and tangent.

28.5 Surface Modeling

BricsCAD Mechanical provides a powerful toolset for surface modeling and conversion of 3D solids to surfaces and vice versa.

Surface Modeling tools are available on the **Surface** tab of the **Mechanical** workspace ribbon:



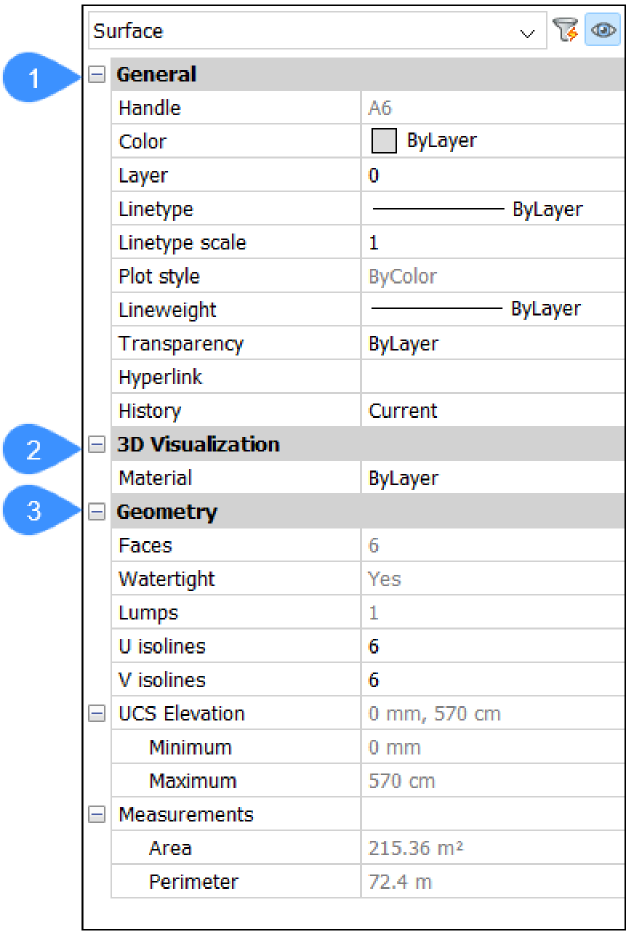
Surface Modeling and Solid Modeling are similar tools concerning editing and applying 3D constraints. The only difference is that **Extrude Surface**, **Revolve Surface**, **Loft Surface**, **Sweep Surface** tools can create surface entities from closed 2D profiles and also accept non-closed profiles.

The Region tool can create a region entity from a closed 2D profile and regions can be stitched together to create a larger surface.

Other methods to create a surface are to extract one or more adjacent faces of a 3D solid/surface, and to explode a 3D solid into a set of surfaces and regions.

28.5.1 Surface properties

Surface properties can be accessed or modified through the **Properties** panel. In case a surface is selected, several main tabs are available.



1 General

2 3D Visualization

3 Geometry

General

Describes properties that are applied to the selected entities and are valid for almost any type of entity.

Note: For more information about the General properties, see the **Properties** panel article.

3D Visualization

Describes the materials assigned to the selected entities.

Material

Describes the current material of the selected entities. You can specify a new material from the drop-list.

Geometry

Describes the geometric characteristics of the selected entities. In case of a surface, several options are available.

Faces
Describes the amount of faces the selected surface contains.

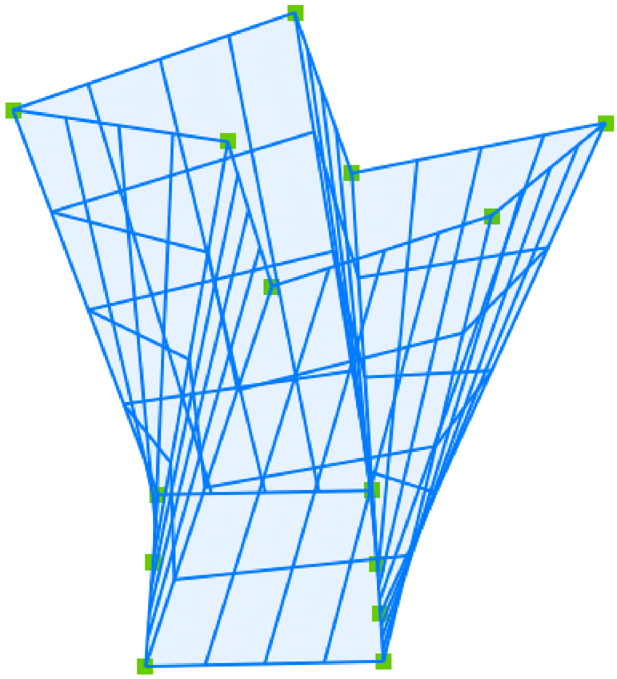
Watertight
Defines whether the surface is watertight or not.

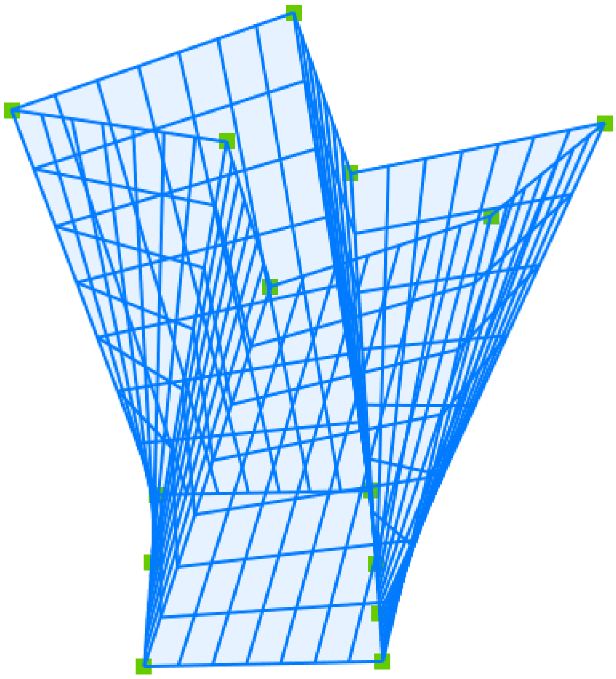
Lumps
Defines the amount of lumps display.

U Isolines
Defines the amount of horizontal isolines to display.

V Isolines
Defines the amount of vertical isolines to display.

Note: The isolines properties are editable in the **Properties** panel, allowing to modify the number of contour lines that are displayed on the curved surfaces. As an example, both isoline properties were set to 3 in the left figure, while 6 was entered in the right figure.





Note: Isolines are only visible in the visual styles **Wireframe**, **2D Wireframe** and **X-Ray**.

UCS Elevation
Defines the elevation of the upper and lower extend of the surface compared to the User Coordinate System.

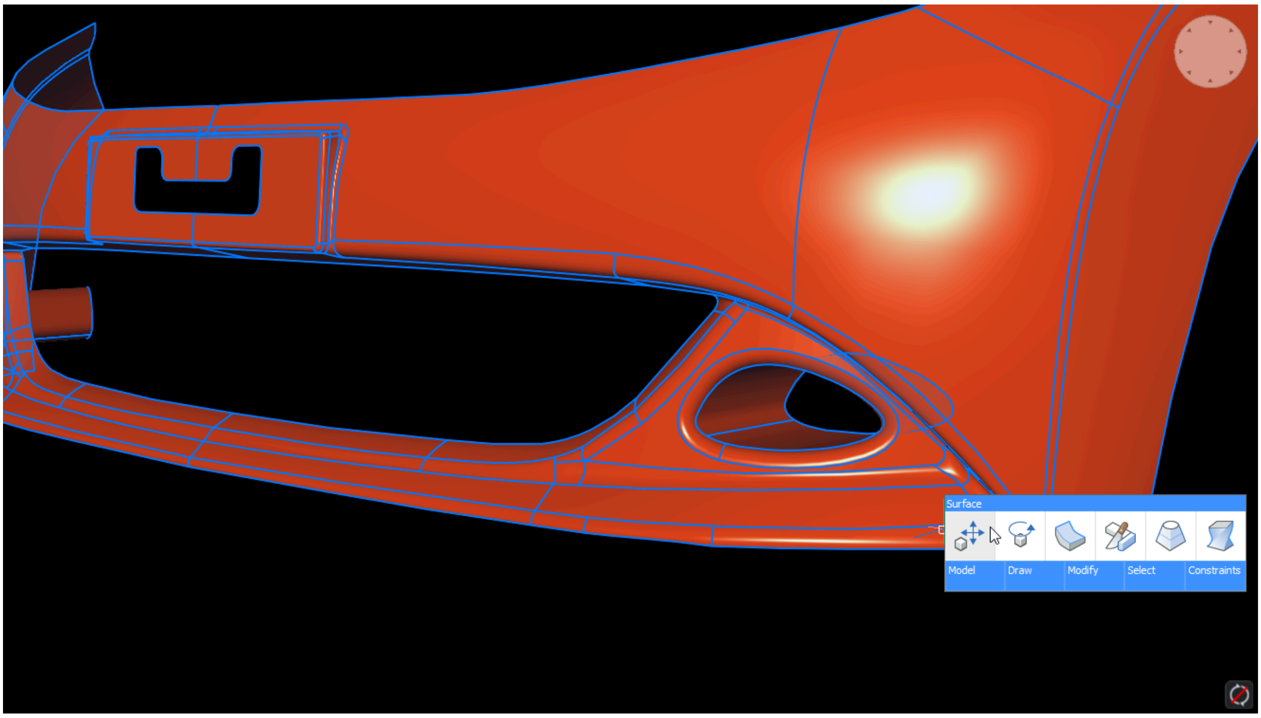
Measurements
Specifies measurements of the selected surface such as area and perimeter.

28.6 Freeform Surface Modeling

28.6.1 Overview

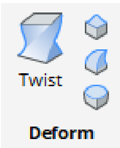
Add complex distortions to your solid or surface models by using BricsCAD Mechanical deform tools. You can create smooth, or organic shapes. Deform your solids and surfaces by moving, rotating and twisting their edges to change the geometry independently with no design history.

History-free approach allows you to focus on your design, instead of thinking about how it can be changed in the future.



28.6.2 Deform tools

You can use the following deform tools:



Tool	Command	Description
Move Edge	*DMDEFORMMOVE*	Deform faces by moving and rotating their edges.
Move Point	*DMDEFORMPOINT*	Deform the selected set of faces by moving a point.

Tool	Command	Description
Transform Curve	*DMDEFORMCURVE*	Deform faces by changing the geometry of their edges.
Twist	*DMTWIST*	Deform 3D object by twisting a portion around a given axis.

29. Assembly Design

29.1 Bottom-up and Top-down Assembly Design

29.1.1 Overview

Assembly modeling represents the structure of a designed product and consists of using components to group geometric entities in hierarchies. Assembly modeling is commonly used in Mechanical Computer-Aided Design (MCAD) applications, it simplifies the design of complex products containing thousands or millions of geometric entities.

The following terms are used in this guide:

- An assembly is a component that contains inserts of other components
- A subassembly is a separate assembly that is inserted into another assembly
- A part is a component that does not contain inserts of another component.

An assembly with inserted components (which in turn can be assembled with other inserted components) forms a tree that is called a mechanical structure of the 3D model. This tree is displayed in the **Mechanical Browser** panel.

There are 2 main ways to design mechanical products: **top-down** and **bottom-up** approaches. You can also add mechanical structure to existing geometry, designed in BricsCAD or imported from another source.

Use the BMNEW command to create a new document for a new mechanical component.

Use the BMMECH command to convert existing blocks and external references to components. Use the BMUNMECH command to do the opposite.

29.1.2 Bottom-Up Design

With bottom-up approach, you can start to design a product by detailing its low-level component. These components are then grouped into higher-level components. The process is continued until the very top component is corresponding to the product itself.

The main command for bottom-up design is BMINSERT. Use this command to insert an existing component into the current drawing.

29.1.3 Top-Down Design

With top-down approach, you can start to design a product by defining, but not detailing, its first-level components. Each component is then refined in more detail by defining its subcomponents. The process is continued till all low-level components are defined and detailed.

Use the BMFORM command to create a new subcomponent for an existing component.

29.1.4 Adding Mechanical Structure to Existing Geometry

You can add component hierarchy to any existing drawing, created in BricsCAD, or imported from a different CAD format.

Use the BMFORM command to convert a selected set of geometric entities to a subcomponent of the current drawing.

29.1.5 Changing Mechanical Structure

During the design process, you can realize that a particular component must be moved up or down in the component hierarchy.

Use the BMFORM command to form a new subcomponent for the current drawing that will contain a selected set of a component.

Use the BMDISSOLVE command to delete a component and move all its subcomponents to the level of the dissolved component.

29.1.6 Using Standard Components

A few standard components are:

- screws
- nuts
- washers
- bearings
- holes

BricsCAD provides a huge library of standard components, more than 30,000 details in a variety of sizes. You can significantly increase design performance by using standard parts.

Use the **Library** panel to insert standard components into your parts and assemblies.

29.1.7 Assembly Constraints

Assembly constraints are ordinary 3D Constraints, that link faces and edges of bodies to represent the components of an assembly. You can link 3D Constraints at the top-level and at the subcomponents level.

29.1.8 Component Visibility

A component within a complex assembly may seem invisible due to other components surrounding it.

Example: a piston is invisible inside a cylinder block. To make such a component visible, you may need to hide the other components surrounding it.

You can hide any subcomponent inserted into the current drawing or any subcomponents, on any level of nesting.

You can also make a component transparent and/or visualize it as a wireframe model.

Use the BMHIDE command to hide a particular subcomponent.

Use the BMSHOW command to make a hidden subcomponent visible.

29.2 Assembly Design Commands and Tools

29.2.1 Commands

Icon	Command	Description
	BMNEW	Create a new mechanical component as a new drawing file.
	BMMECH	Convert the current drawing into a mechanical main component.
	BMINSERT	Insert an existing mechanical component into the current drawing.
	BMINSERT	Insert an existing mechanical component into the current drawing with automatic resizing of the new part and connecting it to the parent part.
	BMHARDWARE	Open the **Library** panel.
	BMFORM	Create a new mechanical component and insert it into the current drawing.
	BMCONNECT	Connect 2 Piping standard parts by creation of 3D constraints between their connection entities.
	BMREPLACE	Replace a component insert.
	BMLOCALIZE	Switch external components to local components.
	BMEXTERNALIZE	Switch local components to external components.
	BMOPEN	Open the source drawing of an external mechanical component.
	BMOPENCOPY	Open a copy of a component insert as a new drawing.
	BMDISSOLVE	Dissolve a mechanical component inserted into the current drawing.
	BMLINK	Change target 3D solids of component-based features.
	BMUNLINK	Break the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening.

Icon	Command	Description
	BMHIDE	Hide a mechanical component or subcomponent in the current drawing.
	BMSHOW	Show a previously hidden mechanical component or subcomponent in the current drawing.
	BMVSTYLE	Apply a visual style to a mechanical component.
	BMBALLOON	Create associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout.
	BMBALLOON	Allow automatic placement of associative balloons on all components in a particular drawing view in a Paper Space layout.
	BMBOM	Insert the Bill-of-Materials (BOM) table in the current drawing.
	BMMASSPROP	Compute mass properties for the current model using densities assigned to its components.
	BMUPDATE	Reload all referenced components from external files and update BOM tables.
	BMUPDATE	Reload all selected referenced components from external files and update BOM tables.
	BMEXPLODE	Create an exploded representation for an assembly by moving components to make them all visible, store it in a new block, and insert the block in Model Space.
	BMEXPLODEMOVE	Move selected parts to form an exploded representation in a given direction, taking into account possible physical collisions between components (similar to the Linear option of the BMEXPLODE command).
	BMTRAILINGLINES	Create all necessary trailing lines for the selected parts.
	MECHANICALBROWSEROPEN	Display the **Mechanical Browser** panel.

Icon	Command	Description
	PARAMETERSPANELOPEN	Display the **Parameters Manager** panel.
	BMDEPENDENCIES	List all files in the command window, containing component definitions inserted in the assembly.
	BMRECOVER	Recover a broken mechanical structure.
	BMUNMECH	Convert the current mechanical component into a plain drawing.
	BMXCONVERT	Convert X-Hardware solids in the current drawing into mechanical components.

29.2.2 Tools

You can use the following tools to work with mechanical components:

1 On the **Assembly** toolbars:

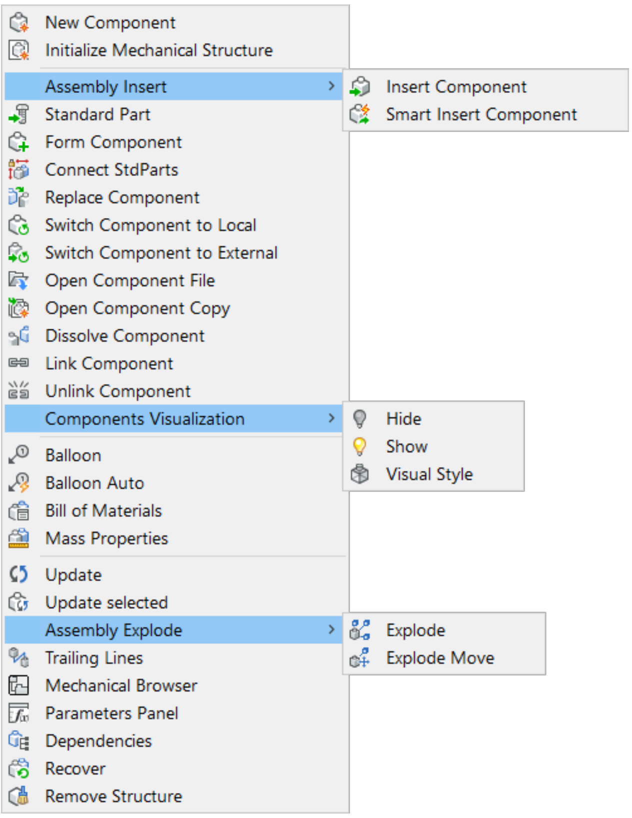
- **Assembly**

- **Assembly Insert**

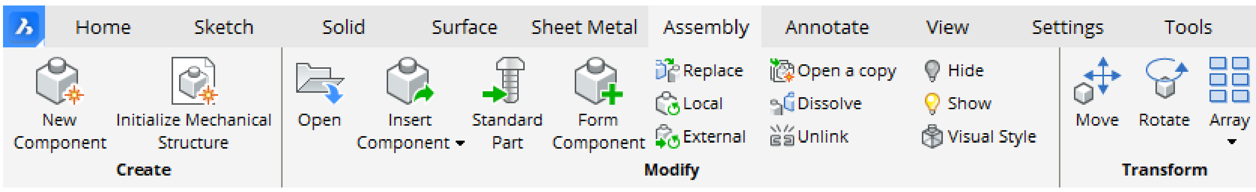
- **Assembly Explode**

- **Assembly Visualization**

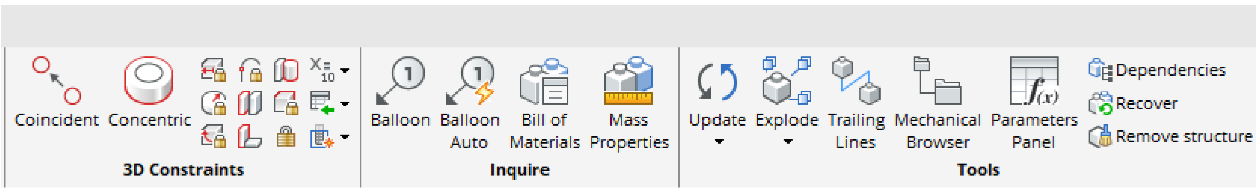
2 In the **Assembly** menu (Mechanical Workspace only):



3 On the **Assembly** tab of the Mechanical Workspace Ribbon:



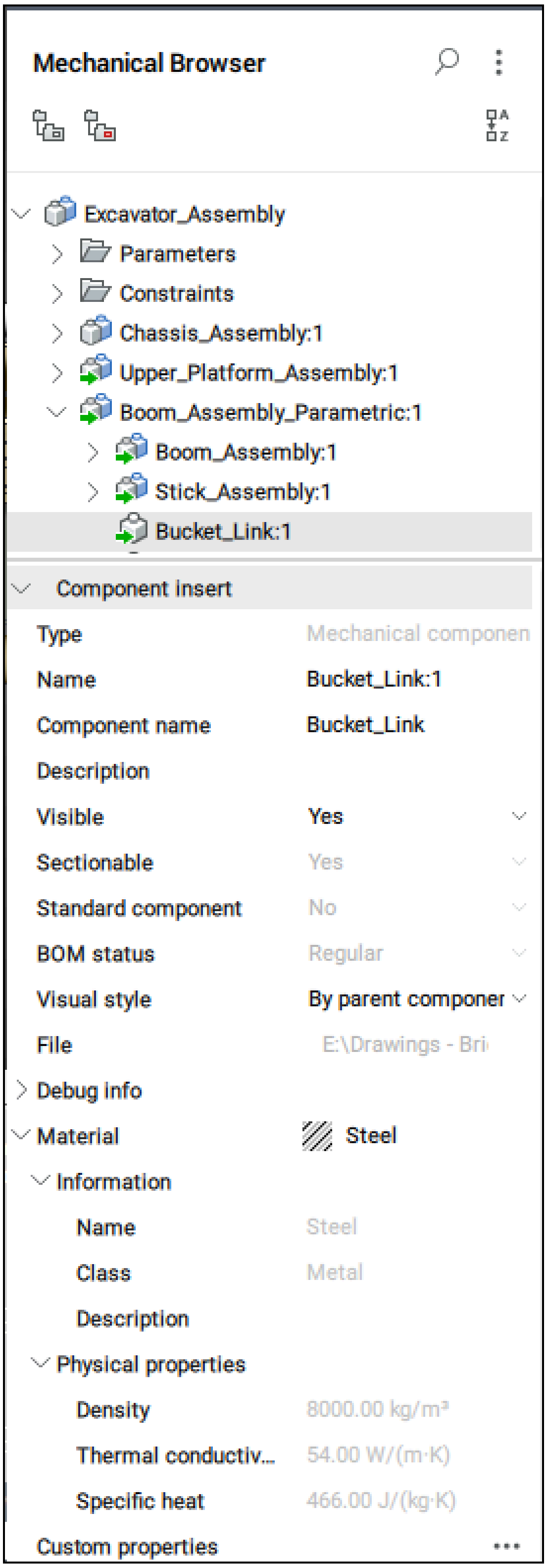
Tools



29.3 Component Properties

Component properties and context menus are located in the **Mechanical Browser** panel.

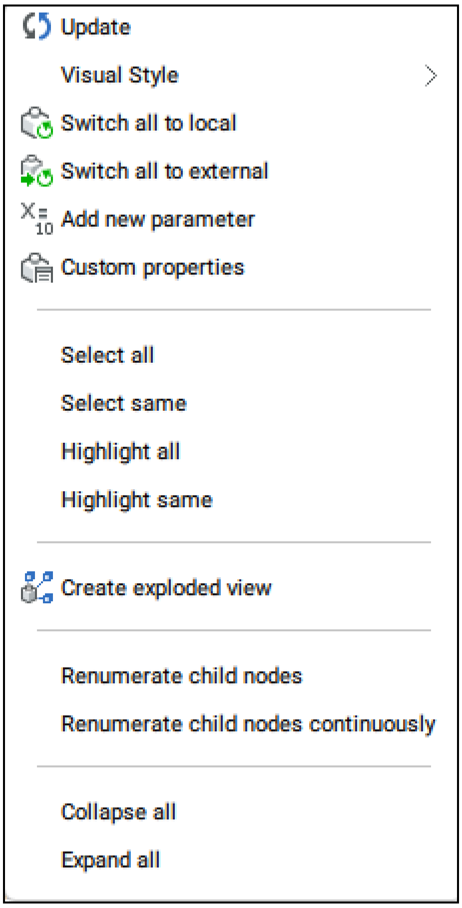
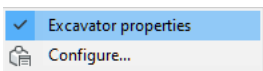
Component context menus display when you right-click the component. Component properties display automatically when selecting one or more components in the **Mechanical Browser** panel.



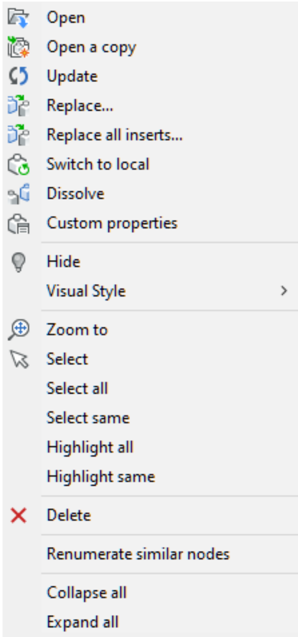
Components can be: **Main**, **Insert** and **Subcomponent**. If **Mechanical blocks and entities** are enabled, they also will be listed as components in the browser.

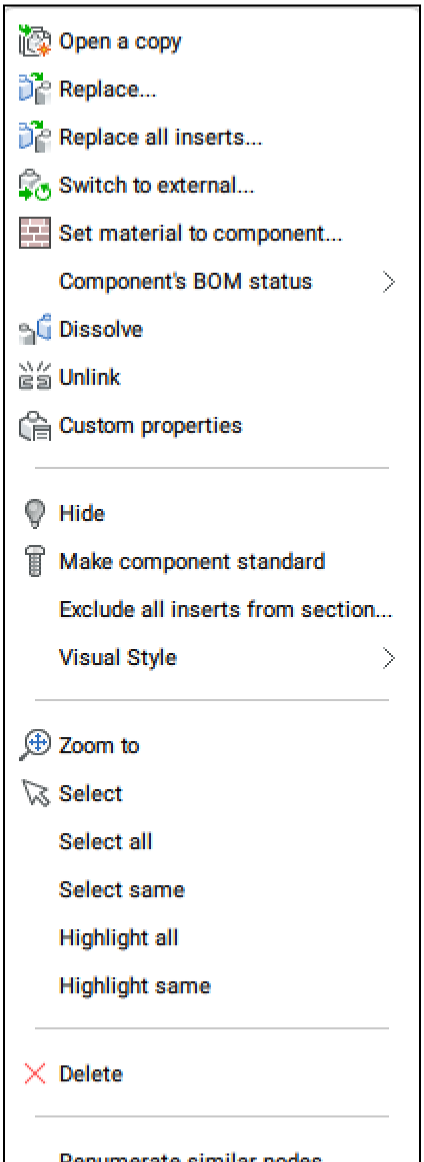
Type of component	Rendered visual style	Wireframe visual style	Hidden
Component without subcomponents	*	*	*
Component with subcomponents	*	*	*
Standard part component			
Mechanical block			
Mechanical external reference			
Mechanical entity			

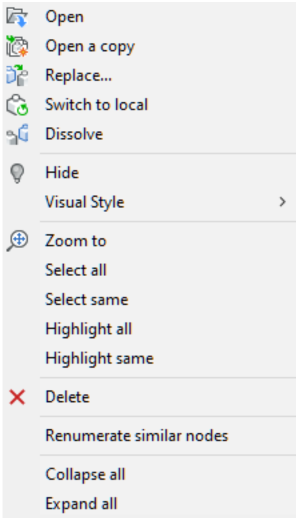
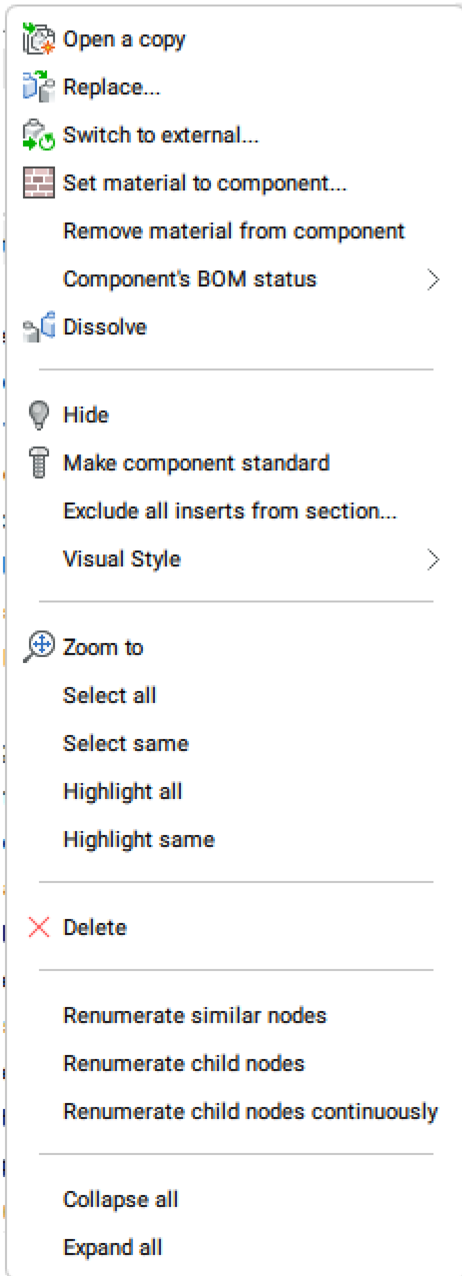
* External components have a green arrow: or .

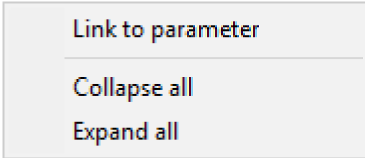
Context menus	
Main Component context menu	Right-click the main component name:  • **Update**: updates the hierarchy of mechanical components for the current drawing in case referenced drawing files of sub-components have been modified. • **Visual Style**: **All by Viewport**: applies the current **Visual Style** to all components in the assembly (see the SHADEMODE command). • **Switch all to local**: switches all components to internal components (see the BMLOCALIZE command). • **Switch all to external**: switches all components to external components (see the BMEXTERNALIZE command). • **Add new parameter**: creates a new parameter in the assembly. • **Custom properties**: allows you to define custom properties for the drawing. If there are per-instance property sets, a submenu opens:  Toggle property sets that should be applied to the root component only, or select Configure to open a dialog to configure custom properties. If no per-instance property sets are present, the **Mechanical Properties** dialog box opens. • **Select all**: select all components with the same definition.

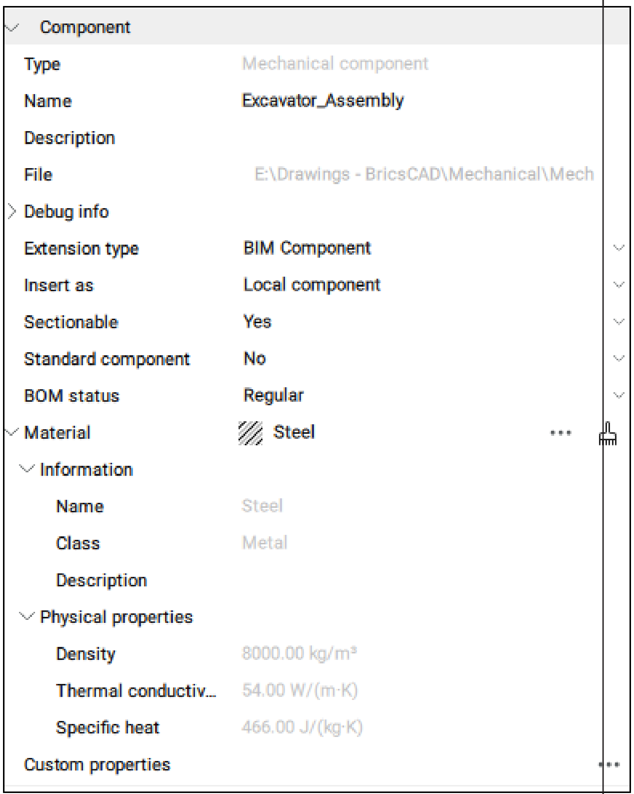
Context menus	
Component context menu	Right-click a component name. External components context menu (left), local components context menu (right):



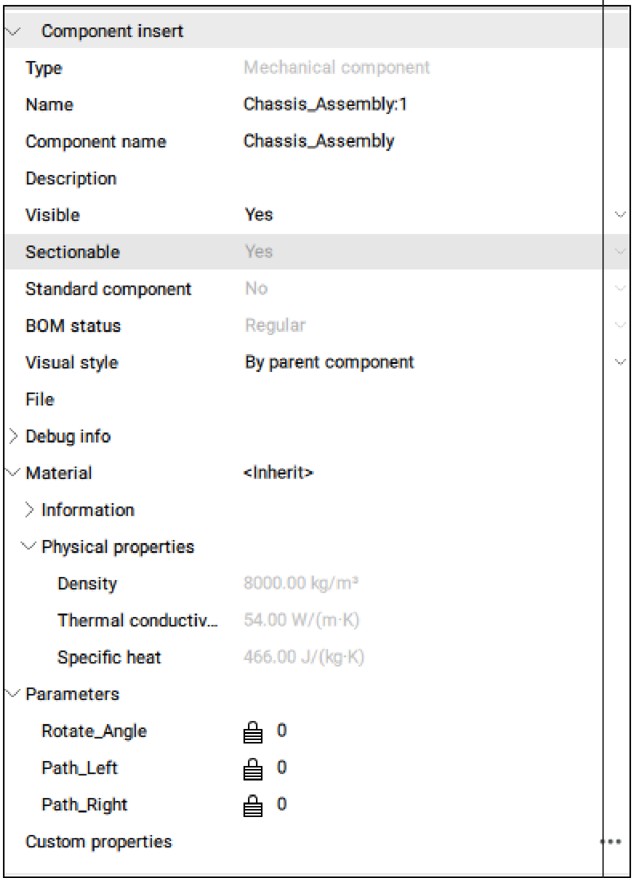


Context menus	
Subcomponent context menu	External subcomponents context menu (left), local subcomponents context menu (right):   • **Open**: opens the referenced drawing (see the

Context menus	
Component parameter context menu	Right-click a parameter:  • **Link to parameter**: create a link between one or more component parameters and assembly parameters. • **Collapse all**: collapse the main component and all components and subcomponents. • **Expand all**: expand the main component and all components and subcomponents.
Properties	

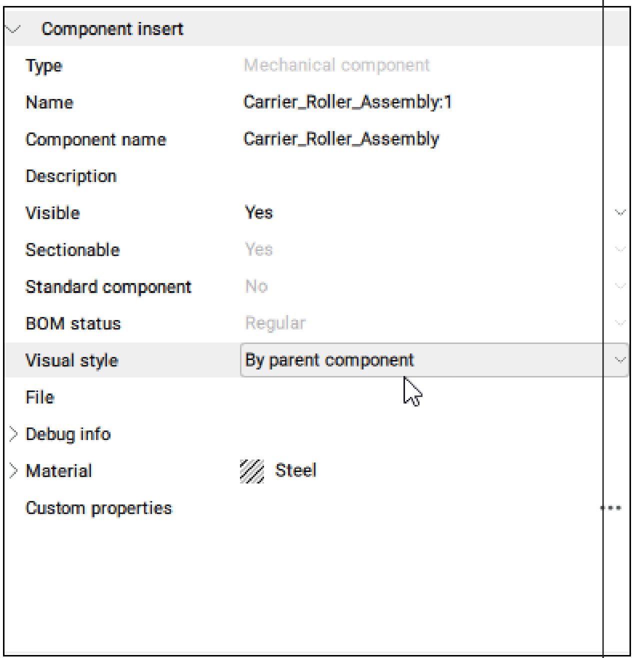
Context menus	
Main component properties	Click the main component name:  • **Name**: type a name for the main component. By default, the name of the main component equals the file name. • **Description**: optional description of the main component. • **File**: shows the path and name of the drawing. • **Extension type**: allows you to identify the component as a Face Feature, a BIM Component or a Sheet Metal Feature. • **Insert as**: select **Internal Component** or **External Component**. • **Sectionable**: defines whether the component is affected by the VIEWSECTION command. • **Standard component**: defines whether this component is a standard component or not. • **BOM status**: defines the behavior of this component in a BOM table. • **Material**: material of the main component. The **Name**, **Density**, and the optional **Description** are defined in the **Physical Materials** dialog box. Select the **Material** node and perform one of the following: - Click the **Browse** button ( ) to open the **Physical Materials** dialog box and choose a material in the Project or Central database list. The Density property of the selected material is used by the BMMASSPROP command.

<table>
<tr><th>Context menus</th><th></th></tr>
<tr><td>Component insert properties</td><td>Click the component name:



Type: type of the component. Shows one of the following types: Mechanical component, Mechanical block or Mechanical Entity.
Name: type a name for the component. By default, the name of the component equals the source file name.
Component name: name of the component as defined in the source file.
Description: description of the component as defined in the source file.
Visible: allows you to show or hide the component in the assembly.
Choose between:
- Click the Visible field, click the down arrow button and choose Yes or No.
Sectionable: defines whether the component is affected by the VIEWSECTION command.
Standard component: defines whether this component is standard or not.
BOM status: defines the behavior of this component in a BOM table.
Visual style: allows you to control the visual style for the component. By default, the visual style is inherited from the viewport.
Click the Visual style field, click the down arrow button, and choose a visual style in the list.
File: shows the path and name of the referenced drawing
</td></tr>
</table>

<table>
<tr><th>Context menus</th><th></th></tr>
<tr><td>Subcomponent properties</td><td>
Click the subcomponent name:



Name: name of the component as defined in the source file of the parent component.
Component name: name of the component as defined in the source file.
Description: description of the component as defined in the source file.
Visible: allows you to show or hide the component in the assembly. Click the Visible field, click the down arrow button and choose Yes or No.
Sectionable: defines whether the component is affected by the VIEWSECTION command.
Standard component: defines whether this component is standard or not.
BOM status: defines the behavior of this component in a BOM table.
Visual style: allows you to control the visual style for the component. By default, the visual style is inherited from the viewport.

Click the Visual style field, click the down arrow button, and choose a visual style in the list.
File: show the path and name of the referenced drawing.
Material: show the material of the subcomponent.

If set to <Inherit>, the material is copied from the main component.

If the material is defined explicitly in the source file, this material is used.
Custom properties: shows the insert's custom properties. By default, properties are inherited from a corresponding component. For top-level inserts, any of those properties

</td></tr>
</table>

Context menus	
Selection methods	
Select component(s) in the model	The component(s) highlights in the Mechanical Browser.
Click a component in the **Mechanical Browser** panel	The component highlights in the drawing. Press the space bar to select the component. If the **Always synchronize selection** option is enabled in the **Mechanical browser options** setting, the component will be selected in the drawing automatically.

29.4 Custom properties of mechanical components

29.4.1 About

Custom properties for components are based on BIM custom properties, but are stored in a different location and not in a BIM project.

Custom properties are grouped in property sets, which could be applied either to all objects of selected types in the document, or to only selected components and instances. The latter could be achieved with "Per-instance" property sets.

To configure custom properties, *BMPROPERTIES* command should be used. It can be called from the ribbon, Command line, or from mechanical browser.

To assign properties for the root component, the **Mechanical Browser** panel should be used. You need to select the root node in the browser, then set value of custom properties as usual.

It is also possible to assign properties to component instances. By default, component instances inherit properties of corresponding component. However, you may override some properties for selected instances, by setting new values to those properties. Instance properties could be edited in either the **Mechanical Browser** or in the **Properties** panel.

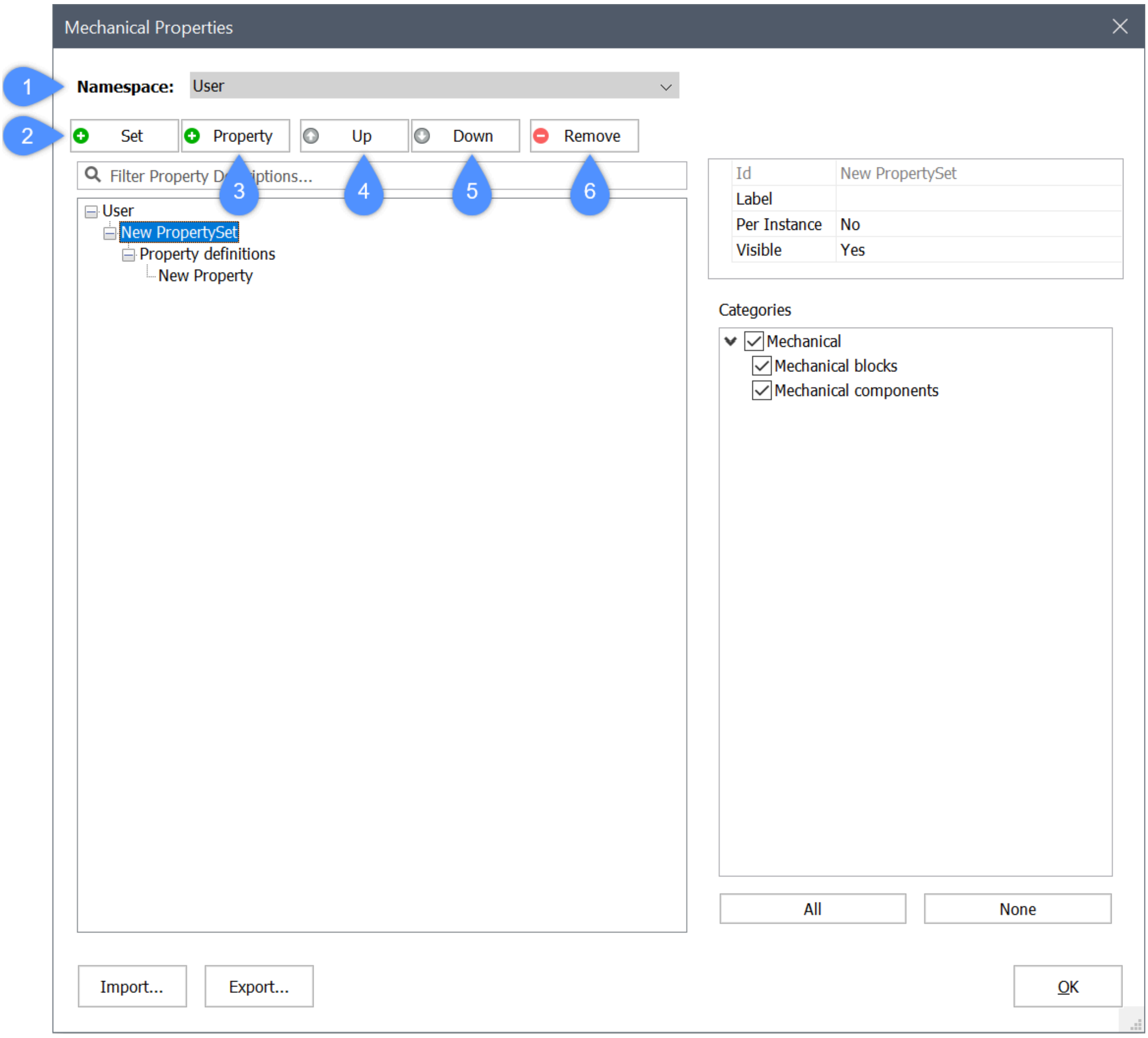
When a component with custom properties is inserted into an assembly, its property definitions are merged to the target assembly. In case of normal (not per-instance) property sets, it means that other components will also get those custom properties, albeit with their default values. In case of per-instance property sets, the property sets will be merged to the assembly, but those properties will be shown only for components those sets have been enabled for.

Custom properties are supported in BOM. They also could be theoretically extracted by data extractor.

29.4.2 Using the Mechanical Properties dialog box

1 Open the **Mechanical Properties** dialog box by typing BMPROPERTIES in the command line. The

following illustration shows the dialog box:



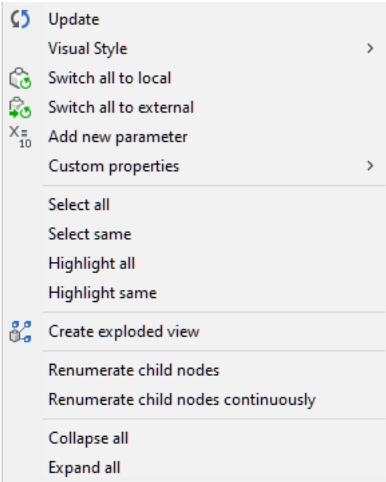
(1) Namespace	Groups property sets in namespaces.
(2) Add property set button	Creates a new property set.
(3) Add property button	Allows adding a property to the currently selected property set.
(4) Move up button	Moves the selected property or value up the list.
(5) Move down button	Moves the selected property or value down the list.
(6) Remove button	Removes the selected property set, property definition or value definition.

2 To create a custom property set, click the **Add Set** button. To edit the property set do the following:

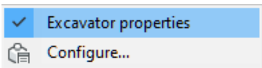
- Type a name in the **Id** (7) field.
- Type a name in the **Label** (8) field.

  Note: The **Label** name appears in the properties tree. If the **Label** name is not defined, the **Id** name displays. The **Id** must be unique.
- **Per Instance** (9) allows to enable/disable user-defined properties, on a per-entity basis.

  To assign a per-instance property set to a root component, do the following:

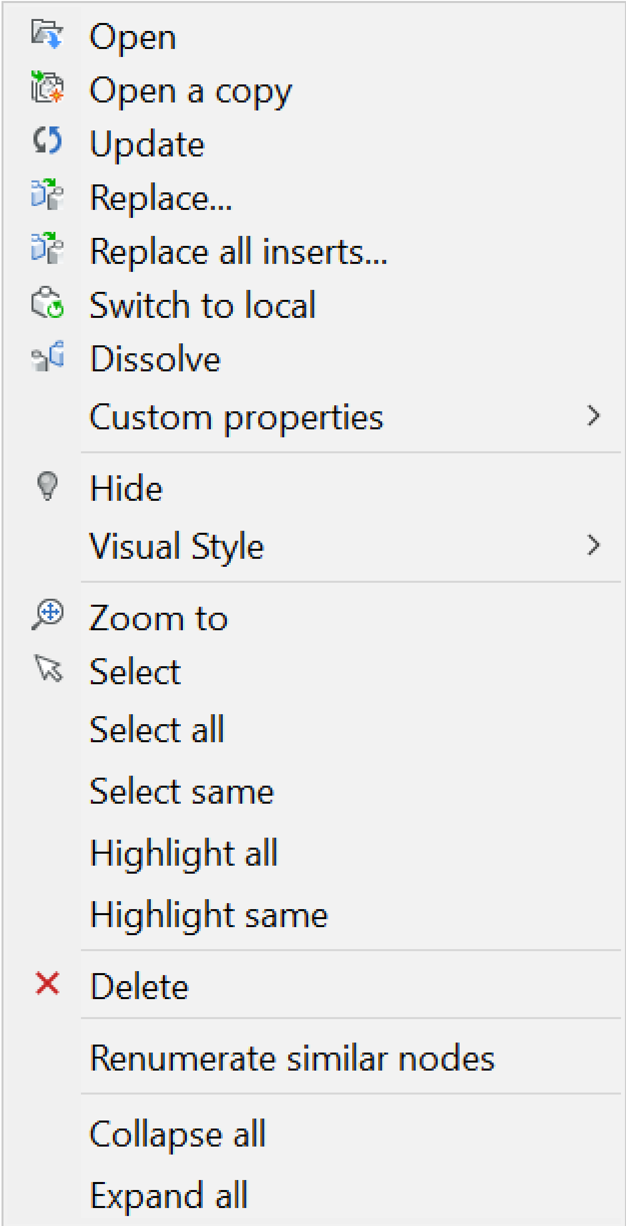
- Open the **Mechanical Browser** panel.
- Select the main (root) component and right-click to open the context menu:
- 

- Select **Custom properties** submenu. Click on property sets that you want to enable for the root component.

  To assign a per-instance property set for top-level component instances or mechanical solids:

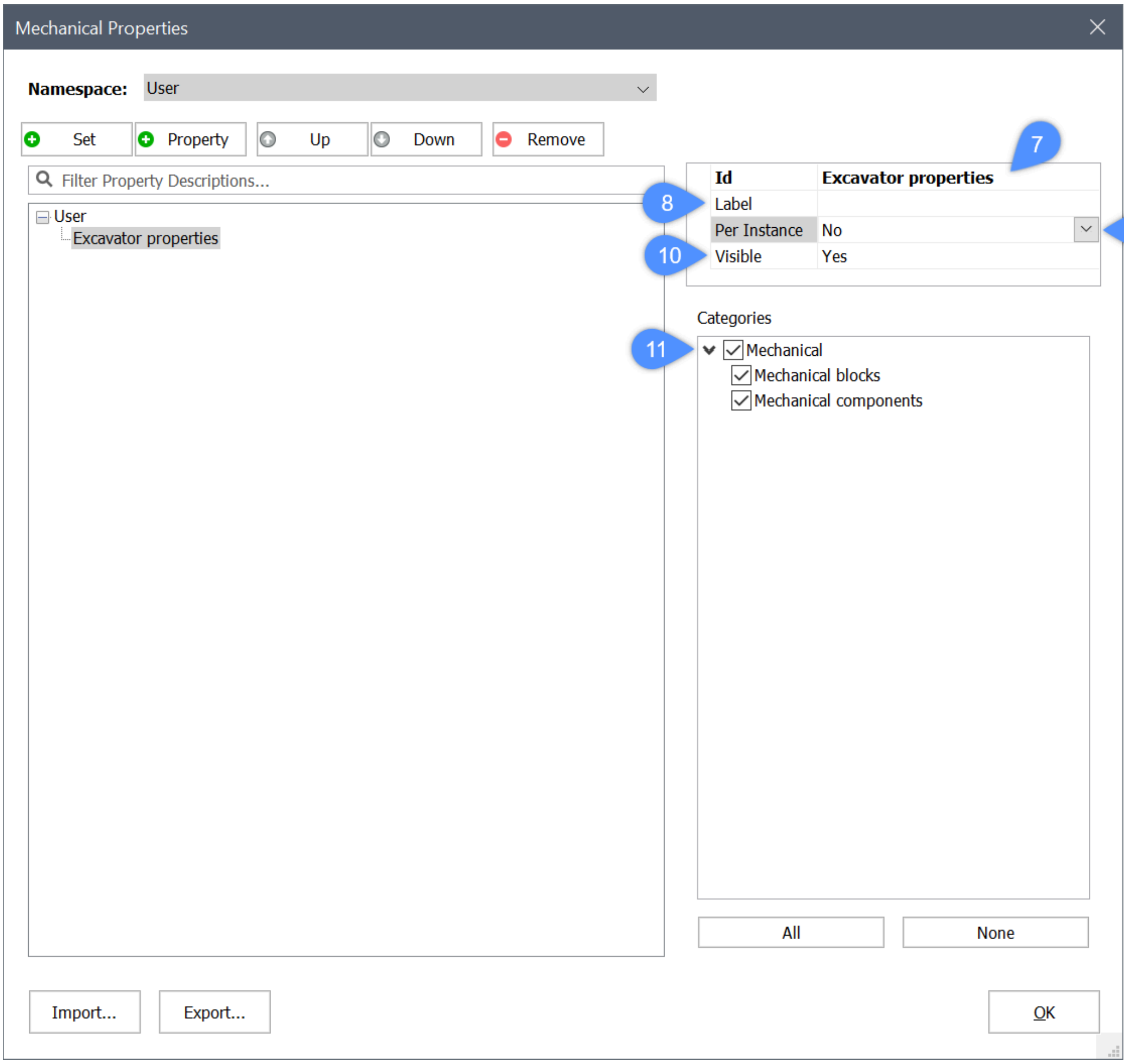
- Open the **Mechanical Browser** panel.
- Select an inserted component and right-click to open the context menu:

- 

- Select **Custom properties** submenu. Click on property sets that you want to enable for the inserted component.
- Define whether the properties in the property set are visible in the **Properties** panel by changing the **Visible** (10) property.

Select the component categories the property set applies to. Currently, for mechanical components, there are two categories: Mechanical components and Mechanical blocks (11).

- Click a category checkbox to select all component types in that category
- Click the **All** button to select all component types in all categories.
- Click the **None** button to uncheck all component types in all categories.



3 To add a property to the currently selected property set, first, select a property set, then click the Add Property button to add a new property.

4 To edit the property do the following:

- Type a name in the **Id** (12).
- Type a name in the **Label** (13).
- Optionally, type a description in the **Description** (14).
- Set the property type (15). The options are
- **Boolean**: an ON or OFF value.

  Integer: a whole number between -2147483648 and 2147483647.

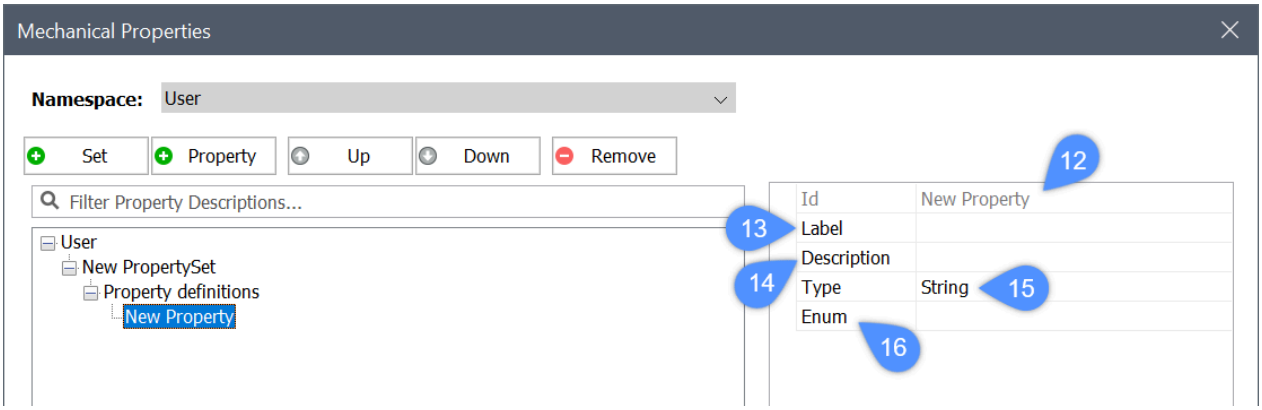
  Real: an approximation of a real number.

  String: a sequence of characters, can be both numbers and letters.
- You can define enumerated values (16). Allows you to define a fixed number of possible values

for custom properties. These values can be selected in a drop-down list in the **Properties** panel or Component properties in the Mechanical browser.

Optionally, check the Mapping option to add a description for each value.

The following illustration shows the **Mechanical Properties** dialog box after inserting a new property.



5 To remove the property set/property, select a property set/property in the tree, then click the **Remove** button to remove.

6 Press the **OK** button to accept it.

29.5 How to create a custom standard part

29.5.1 About

The -BMHARDWARE command can be used to create custom standard parts of several categories, for example: Pipe, Cross, Tee, etc. It has the option **create Custom** that allows to select TXT file(s). These files should contain the following parts:

1 Mandatory creation parameters

2 Optional creation parameters

3 Configuration table

Let's describe these parts one by one, but first, some general rules:

- Lines which start from semicolon will be ignored.
- All lines with creation parameters should have the following signature:
- ```
  PARAMETER_NAME: "parameter_value"
  ```

29.5.2 Mandatory creation parameters

Mandatory creation parameters are parameters without which part just will not be created. Currently, there are the following mandatory parameters:

- **NAME**: Name of the created part.
- **CATEGORY**: Category of the created part (supported categories will be listed below, english names must be used).
- **PATH**: The path to the created part in the **Library** panel.

  Note: Folders can be separated by semicolon. For example, the value "Piping;Elbows" means that part will be created in the folder Standard Parts ? Piping ? Elbows ? the value of the STANDARD creation parameter.
- **STANDARD**: The standard of the created part, for example:ASME, ANSI, DIN, ISO, etc. (families of standards), and not ASME B16.5, DIN EN 1092-1, etc. (particular standards).
- **DESCRIPTION**: Description of the created part.

29.5.3 Optional creation parameters

The following creation parameters can be omitted:

- **EXPOSED_PARAMETERS**: The parameters in the configuration table that will be accessible to users of the given parts.

  Note: The parameters should be separated by spaces.
- **INITIAL_VALUE**: The key value from the table that will be used when the part is dragged from the library.

  Note: If this parameter is omitted, then the first line of the table will be used as the initial value.
- **MATERIAL**: The material of the created part.
  ```

Note: Only materials from the material library can be used.

- **CUSTOM_PARAMETERS**: The parameters that can have any value and cannot be determined by the table (for example length of pipe).
- **KEY_PARAMETER**: The parameter that will be used as key of the parameter table.

  Note: If this parameter is omitted, then the first column will be used as the key parameter.
- **LENGTH_MEASUREMENT_TYPE_PARAMETERS**, **AREA_MEASUREMENT_TYPE_PARAMETERS**, **VOLUME_MEASUREMENT_TYPE_PARAMETERS**, **UNKNOWN_MEASUREMENT_TYPE_PARAMETERS**: The parameters that will have the appropriate type. For example, it will be changed in length/area/volume when a part is inserted into documents with other units. Length type is by default.

29.5.4 Configuration table

The configuration table also has mandatory and optional parameters. The mandatory configuration parameters are the following:

- **Geometrical parameters**: These parameters determine the geometry of the part and differ for different categories of parts. The full list of such parameters for each category will be described below.
- **Key parameter**: This parameter should unequivocally determine the part configuration.
- **Unit parameter**: Determines the units in which the part will be created. It can have "IN" – inch and "MM" – millimeters values.

Also, there can be any number of other parameters.

Note: Some general rules:

- String parameters should be inside quotation marks: "value".
- If quotation marks are part of the parameter value symbol, the symbol **** should be placed before symbol **"**. For example:

  ```
  "1/2\" X 1/2\" SCH 5"
  ```

  will be recognized as 1/2" X 1/2" SCH 5.

29.5.5 How to create a file to create a standard part

The ordinary way to create a file to create a standard part is the following:

1. Create the table using the values from standards, which will contain key values and geometrical parameters. Add column with units and custom parameters columns if needed.
2. Convert the table to text format (for example to CSV). Space should be used as separator, and text values should be placed inside double quotation mark (").
3. Copy and paste the content of this file into a TXT file, or even change extension of this CSV file to TXT.
4. Add all mandatory creation parameters and, if needed, optional parameters to the beginning of this TXT file.
5. Your file is ready to use.

29.5.6 Creation of custom standard part from existing TXT file

After creating your TXT file, you need to do following:

1. In any drawing, run the -BMHARDWARE command and choose the option **create Custom**.
2. In the **Select configuration table file** dialog box, select your created TXT file (or files).
3. The command will create a new standard part. For example:
   - It will open a new drawing with proper units.
   - It will create components with properties according to the TXT file.
   - It will save the standard part in proper place.
   - It will create thumbnails for the library.

Example of a possible .TXT file to create a sample elbow:

```
; Mandatory creation parameters

NAME: "Sample elbow"
CATEGORY: "Elbow"
PATH: "Piping;Elbows"
STANDARD: "Sample standard"
DESCRIPTION: "Sample description"

; Optional creation parameters

EXPOSED_PARAMETERS: "PartNumber" "Size" "OuterDiameter" "Thickness" "Schedule"
INITIAL_VALUE: "1/2\" X 0.065\" (SCH 5S)"
LENGTH_MEASUREMENT_TYPE_PARAMETERS: "OuterDiameter" "Thickness"
MATERIAL: "Steel"

; other possible optional parameters
; CUSTOM_PARAMETERS:
; KEY_PARAMETER:
; UNKNOWN_MEASUREMENT_TYPE_PARAMETERS:
; AREA_MEASUREMENT_TYPE_PARAMETERS:
; VOLUME_MEASUREMENT_TYPE_PARAMETERS:

; table

"PartNumber"                   "Size"      "UNIT"     "OuterDiameter"     "Thick-
ness"      "Schedule"      "B"      "DEG"
"1/2\" X 0.065\" (SCH 5)"      "1/2\""     "IN"       0.84                0.065
        "SCH 5"        0.62      45.0
"1/2\" X 0.065\" (SCH 5S)"     "1/2\""     "IN"       0.84                0.065
        "SCH 5S"       0.62      45.0
"1/2\" X 0.083\" (SCH 10)"     "1/2\""     "IN"       0.84                0.083
        "SCH 10"       0.62      45.0
```

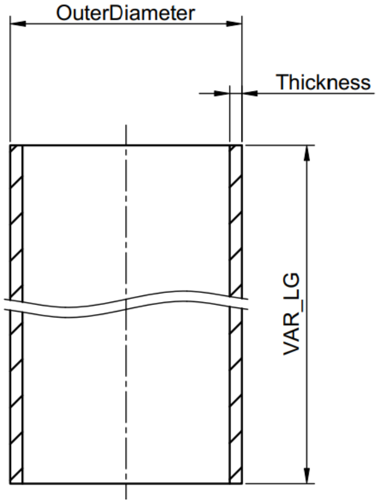
29.5.7 Transferring of models with custom standard parts

Currently, data for the custom standard parts is stored separately from the drawing in which they are inserted. In order for another user to be able to work with models with custom standard parts, he should also have the appropriate TXT file from which they were created. This user should perform all the steps from **Creation of custom standard part from existing TXT file**.

29.5.8 Parts categories and geometrical parameters

Pipe

```
Category: "Pipe"
```



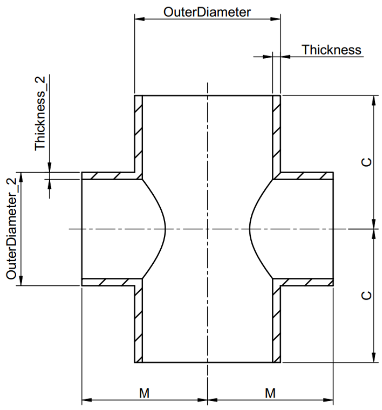
All parameters are mandatory:

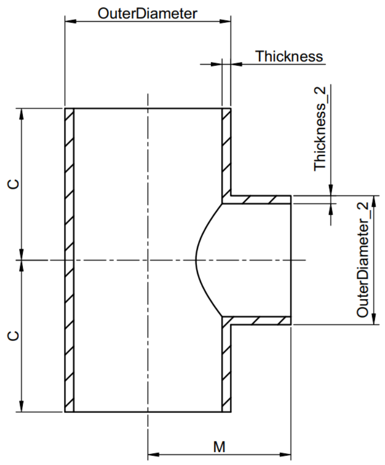
- **OuterDiameter**: Outer diameter.
- **Thickness**: Thickness.
- **VAR_LG**: Initial length.

Cross and tee

```
CATEGORY: "Cross"
```

```
CATEGORY: "Tee"
```





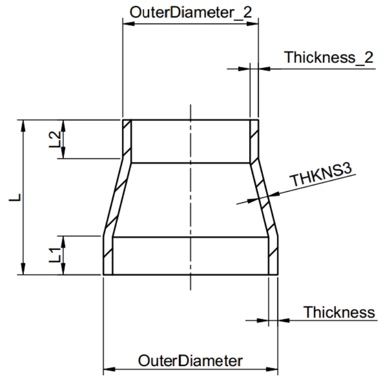
All parameters are mandatory:

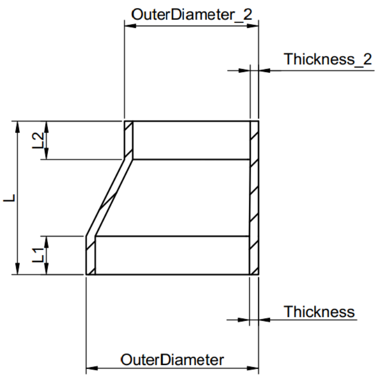
- **OuterDiameter**: Run outer diameter.
- **OuterDiameter_2**: Outlet outer diameter.
- **C**: Run center to end distance.
- **M**: Outlet center to end distance.
- **Thickness**: Run thickness.
- **Thickness_2**: Outlet thickness.

Reducer and eccentric reducer

```
CATEGORY: "Reducer"
```

```
CATEGORY: "Eccentric reducer"
```





Mandatory parameters:

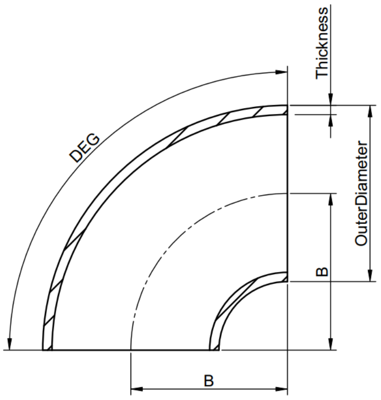
- **OuterDiameter**: Outer diameter of the 1st end.
- **OuterDiameter_2**: Outer diameter of the 2nd end.
- **Thickness**: Wall thickness of the 1st end.
- **Thickness_2**: Wall thickness of the 2nd end.
- **L**: Overall length of the reducer.

Optional parameters:

- **THKNS3**: Wall thickness of the conical part, not for eccentric reducer. The default value is maximum of Thickness and Thickness_2.
- **L1**: Length of the 1st end's cylindrical part. The default value is 0.25 × L.
- **L2**: Length of the 2nd end's cylindrical part. The default value is 0.25 × L.

Elbow

```
CATEGORY: "Elbow"
```

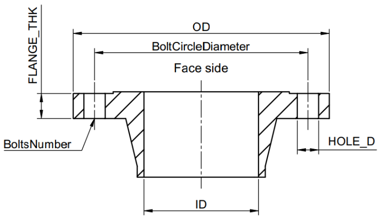


All parameters are mandatory:

- **OuterDiameter**: Outer diameter.
- **Thickness**: Thickness.
- **B**: Center to end distance.
- **DEG**: Angle in degrees.

Flange

```
CATEGORY: "Flange"
```



Mandatory parameters:

- **OD**: Flange outer diameter.
- **ID**: Flange inner (bore) diameter.

  Note: If the flange type is BLIND, it must be omitted, or its value must be 0.
- **FLANGE_THK**: Flange thickness. It does not include face height.
- **BoltCircleDiameter**: Holes' centers circle diameter.
- **BoltsNumber**:Number of holes.
- **HOLE_D**: Hole diameter.
- **BoltDiameter**: Bolt diameter.
- **OuterDiameter**: Pipe outer diameter.
- **GasketThickness**: Gasket thickness.

  Note: If the face type is ASME_RING_JOINT, it must be omitted.
- **DistanceBetweenFlanges**: Distance between flanges.

  Note: If the face type is not ASME_RING_JOINT, it must be omitted.
- **FLANGE_TYPE – flange type**
  The value must be the same for all rows. The following types are supported.

  BLIND – no bore
  No hub's raised part. No pipe connection.

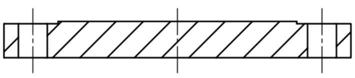
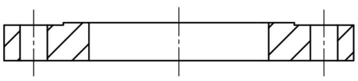
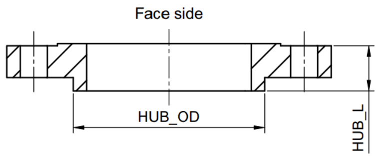
  

  PLATE – no hub's raised part
  The pipe is inserted into the bore.

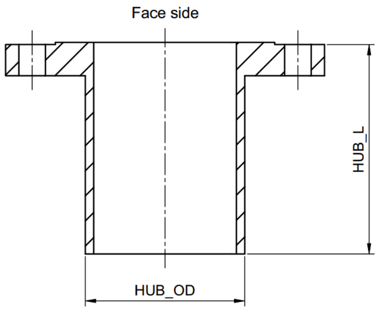
  

  CYL_RAISE_HUB_PIPE_INS_INTO_BORE – the hub has a cylindrical raised part
  The pipe is inserted into the bore.



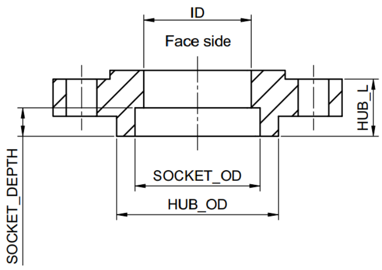
Mandatory parameters: **HUB_OD**, **HUB_L**.

CYL_RAISE_HUB_PIPE_CONN_TO_HUB – the hub has a cylindrical raised part
The pipe is connected to the hub.



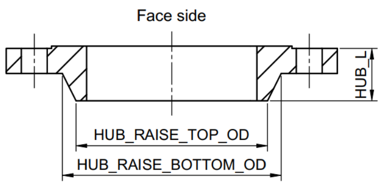
Mandatory parameters: **HUB_OD**, **HUB_L**.

CYL_RAISE_HUB_WITH_SOCKET – the hub has a cylindrical raised part with a socket
The pipe is inserted into the socket.



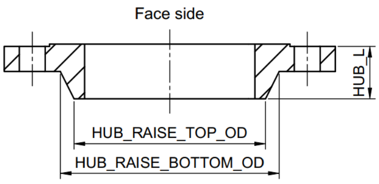
Mandatory parameters: **HUB_OD**, **HUB_L**, **SOCKET_OD**, **SOCKET_DEPTH**.

CON_RAISE_HUB_PIPE_INS_INTO_BORE – the hub has a conical raised part
The pipe is inserted into the bore.



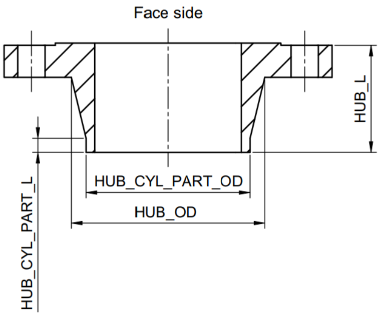
Mandatory parameters: **HUB_RAISE_BOTTOM_OD**, **HUB_RAISE_TOP_OD**, **HUB_L**.

CON_RAISE_HUB_PIPE_CONN_TO_HUB – the hub has a conical raised part
The pipe is connected to the hub.



Mandatory parameters: **HUB_RAISE_BOTTOM_OD**, **HUB_RAISE_TOP_OD**, **HUB_L**.

CON_CYL_RAISE_HUB_PIPE_CONN_TO_HUB – the hub has a raised part
It consists of the conical part at the bottom and the cylindrical part at the top. The pipe is connected to the hub.

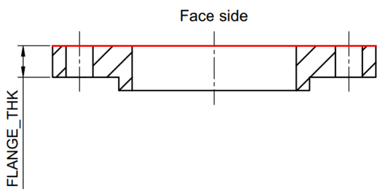


Mandatory parameters: **HUB_OD**, **HUB_L**, **HUB_CYL_PART_OD**, **HUB_CYL_PART_L**.

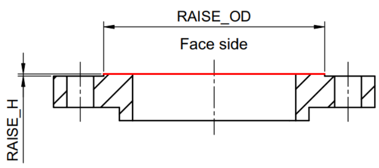
- **FACE_TYPE – face type**
  The value must be the same for all rows. The following types are supported.

  Note: The red line shows the face from which GasketThickness (or DistanceBetweenFlanges for ASME_RING_JOINT) is measured along Z-axis.

  FLAT – flat face

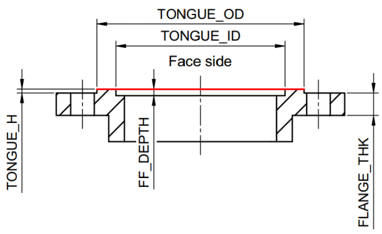


RAISED



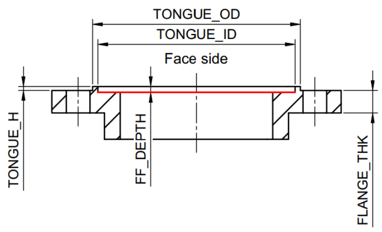
Mandatory parameters: **RAISE_OD, RAISE_H**.

TONGUE



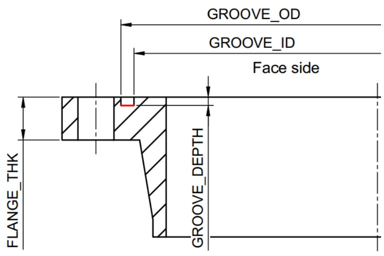
Mandatory parameters: **TONGUE_ID, TONGUE_OD**, **TONGUE_H**, **FF_DEPTH**.

FEMALE



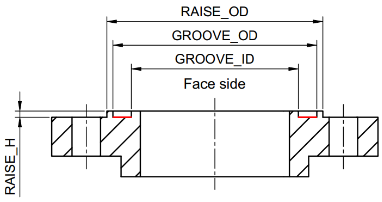
Mandatory parameters: **TONGUE_ID**, **TONGUE_OD**, **TONGUE_H**, **FF_DEPTH**.

FLAT_WITH_GROOVE



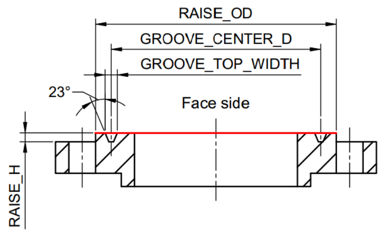
Mandatory parameters: **GROOVE_ID**, **GROOVE_OD**, **GROOVE_DEPTH**.

GROOVE



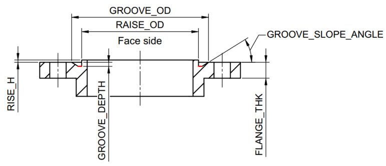
Mandatory parameters: **RAISE_OD**, **RAISE_H**, **GROOVE_ID**, **GROOVE_OD**.

ASME_RING_JOINT



Mandatory parameters: **RAISE_OD**, **RAISE_H**, **GROOVE_CENTER_D**, **GROOVE_TOP_WIDTH**.

DIN_O-RING_GROOVE



Mandatory parameters: **RAISE_OD**, **RAISE_H**, **GROOVE_OD**, **GROOVE_DEPTH**, **GROOVE_SLOPE_ANGLE**.

Optional parameters:

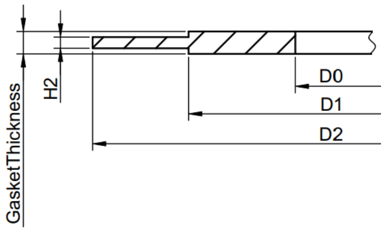
- **Thickness**: Pipe thickness.
- **Gap**:
  - If the pipe is inserted into the bore, it is the distance along Z-axis from the highest point of the bore to the pipe connection point.
  - If the pipe is inserted into the socket, it is the distance along Z-axis from the bottom of the socket to the pipe connection point.
  - ***Note:*** The default value is 0. For other pipe connection types, it must be omitted.
- **IS_CUSTOM_PIPE_THK_ALLOWED**:
  - If the value is not 0, custom **Thickness** values are supported.
  - If the value is 0, custom **Thickness** values are not supported.

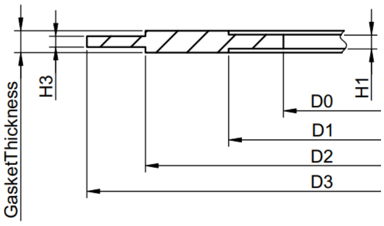
    Note: At this moment, it works only for the flange type **CON_RAISE_HUB_PIPE_CONN_TO_HUB** if the parameter **HUB_RAISE_TOP_OD** corresponds to **OuterDiameter**. The default value is 0.

Gasket

```
CATEGORY: "Gasket"
```

MULTIRING



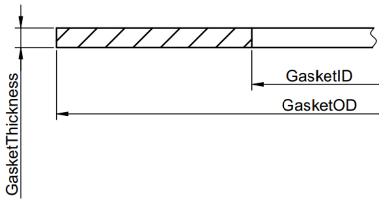


This is a type of gasket consisting of one or more subsequent rings. Rings have rectangular profile. Ring thickness H (i), bounding diameters D(i-1) and D(i).

If the **GasketThickness** parameter is set, it will be used for all rings that do not have the H parameter set.

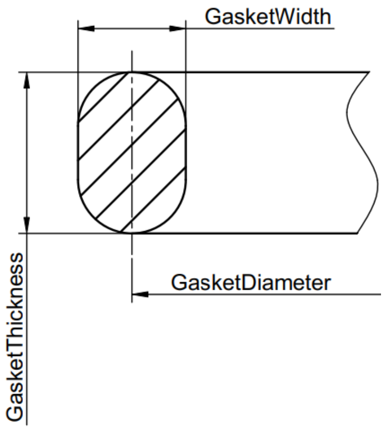
Note: H cannot be greater than **GasketThickness**.

FLATFACE



This is a gasket with a rectangular profile.
GasketID: Inside diameter of gasket.
GasketOD: Outside diameter of gasket.
GasketThickness: Thickness of gasket.

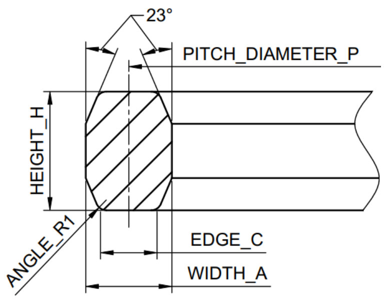
OVAL



This is a gasket with an oval or round profile.
GasketDiameter: Gasket pitch diameter.
GasketWidth: Width of gasket.
GasketThickness: Thickness of gasket.

Note: **GasketWidth** can be greater than **GasketThickness** as in ASME B16. 20 Jacketed, or less than as in ASME B16.20 Oval, or equal to as in AS 2129 O-Ring.

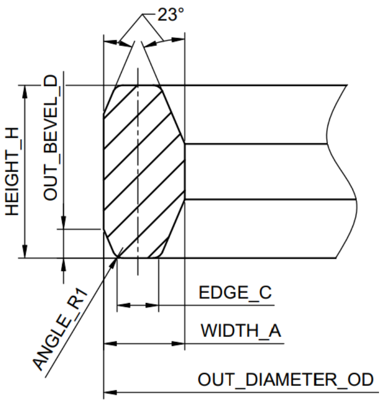
OCTAGONAL_R



Specific shape gasket for ASME B16.20 Type R standard.
WIDTH_A: Width of gasket.

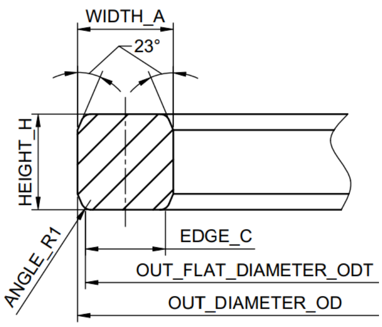
HEIGHT_H: Height of gasket.
EDGE_C: Width of flat.
PITCH_DIAMETER_P: Average pitch diameter of gasket.
ANGLE_R1: Radius of ring.

OCTAGONAL_RX



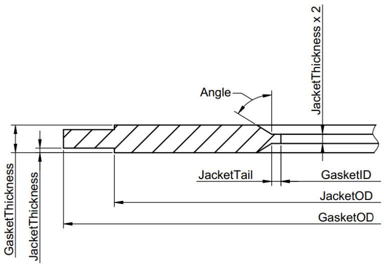
Specific shape gasket for ASME B16.20 Type RX standard.
WIDTH_A: Width of gasket.
EDGE_C: Width of flat.
OUT_BEVEL_D: Height of outside bevel.
HEIGHT_H: Height of gasket.
OUT_DIAMETER_OD: Outside diameter of gasket.
ANGLE_R1: Radius of ring.

OCTAGONAL_BX



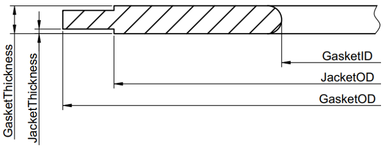
Specific shape gasket for ASME B16.20 Type BX standard.
WIDTH_A: Width of gasket.
EDGE_C: Width of flat.
HEIGHT_H: Height of gasket.
OUT_DIAMETER_OD: Outside diameter of gasket.
OUT_FLAT_DIAMETER_ODT: Outside diameter of flat.
ANGLE_R1: Radius of ring.

JACKET_A



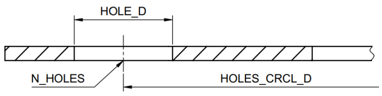
Specific shape gasket for DIN EN 1514-3 Type A standard.
GasketID: Inside diameter of gasket.
JacketOD: Outside diameter of jacket.
GasketOD: Outside diameter of gasket.
JacketThickness: Thickness of jacked.
GasketThickness: Thickness of whole gasket (core and 2x jacked).
JacketTail: Length of jacket tail.
Angle: Jacket inclination angle in front of the tail.

JACKET_C



Specific shape gasket for DIN EN 1514-3 Type C standard.
GasketID: Inside diameter of gasket.
JacketOD: Outside diameter of jacket.
GasketOD: Outside diameter of gasket.
JacketThickness: Thickness of jacked.
GasketThickness: Thickness of whole gasket (core and 2x jacked).

Note: The gasket may contain bolt holes. In order to add them, you need to specify three parameters. You can see the sample in ASME B 16.21 FullFace gasket.



N_HOLES: Number of holes.

HOLE_D: Hole diameter.

HOLES_CRCL_D: The circle diameter on which the holes are located.

29.6 Assembly mass properties

29.6.1 Overview

Use the BMMASSPROP command.

An assembly can consist of multiple parts made from different materials. To correctly compute the mass, center of gravity (centroid), and inertia of such assemblies, BricsCAD supports individual densities for components.

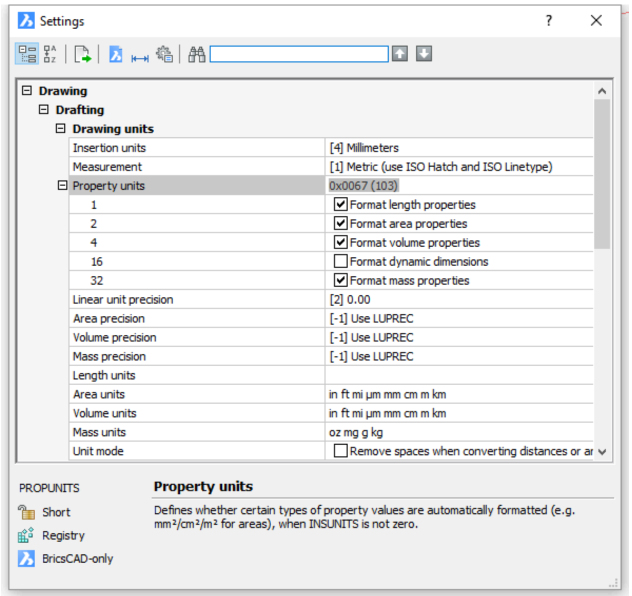
A physical material for a mechanical component is defined in the Material property in the **Mechanical Browser**.

By default, a component inherits the density of the assembly, into which it was inserted, or formed from.

29.6.2 Set the mass units

1 Open the *Settings* dialog box.

2 Expand the **Property units** settings under **Drawing / Drafting / Drawing units**.

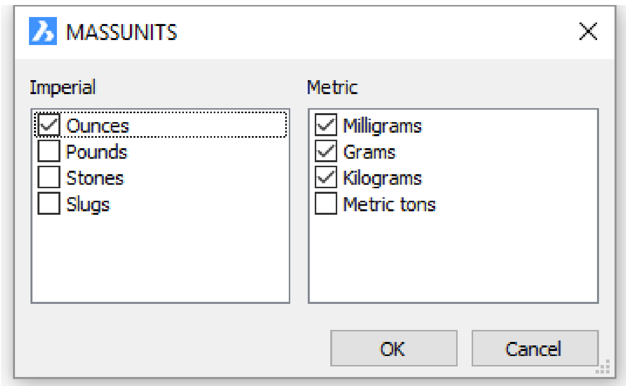


3 Tick the **Format mass properties** checkbox to enable formatting of mass values with appropriate units.

4 Select the **Mass units** field.

5 Click the **Browse** button .

6 Specify the desired mass units in the *MASSUNITS* dialog box.



Note: The **Mass units** setting affects the mass values only. Other mass properties such as density or moments of inertia will be formatted in SI units for the metric system and in imperial units for the imperial system, regardless of the value of the **Mass units** setting.

7 Specify the desired precision in the **Mass precision** field.

8 Close the *Settings* dialog box.

29.6.3 Compute the mass properties

Use the BMMASSPROP command.

You can compute mass properties for the entire model or for a selection of 3D solids and/or subcomponents. The values of mass, volume, centroid, moments of inertia, products of inertia, radii of gyration, principal moments, and X-Y-Z directions about centroid are reported in the Command line.

Optionally, you can align the User Coordinate System with the inertia axes. As a result, the center of mass or centroid of the component lies at the origin of UCS, which can be useful to investigate the behavior of a product under gravity force.

29.7 Standard Mechanical Components

29.7.1 Overview

Use the LIBRARYPANELOPEN command.

BricsCAD comes with a library of more than 1,000 standard mechanical parts available in more than 170,000 different sizes. They are available in the **Library** panel, and can be drag-and-dropped from the library into your assembly. You can adjust the parameters in the **Properties** panel, which temporarily lists parameters of the component being inserted. After insertion, parameters can be edited in **Mechanical Browser** and the **Properties** panel.

In the **Mechanical** workspace, the **Library** panel contains all categories, including:

- Standard Parts
- Holes
- Sheet Metal
- Tools

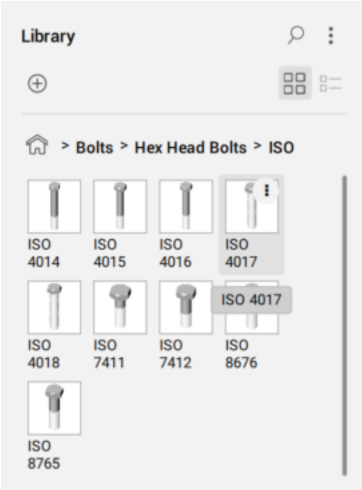
29.7.2 Standard Parts

The **Standard Parts** category contains mechanical blocks grouped into 4 subcategories:

- **Fasteners**, such as bolts, nuts, and washers.
- **Machine Parts**, such as bearings, and sprockets.
- **Piping**, such as pipes, elbows, and flanges.
- **Structural Shapes**, such as beams, channels, and bars.

The content of each subcategory is structured by type, subtype, and standards.

Example: Standard Parts / Fasteners / Bolts / Hex Head Bolts / ISO / ISO 4017.



Note: There are 2 specific settings you can configure for Standard Parts in the **Settings** dialog box:

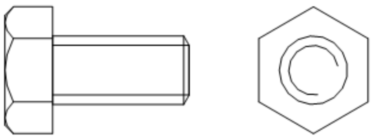
- **Thread representation (THREADDISPLAY)** is used to represent (or not to represent) a 3D thread for bolts and nuts. This setting allows to decrease the memory consumption and increase the performance for large assemblies
- **Maximum number of sprocket teeth (GEARTEETHNUMBER)** is used to limit the number of teeth for sprocket wheels (available under **Standard Parts / SPROCKET)**. This setting allows to decrease memory consumption and increase the performance.

By default, THREADDISPLAY is set to Off and GEARTEETHNUMBER is set to 1, in order to optimize the performance of the model. When changed, the settings apply to all new documents.

Standard representation of thread in drawing views

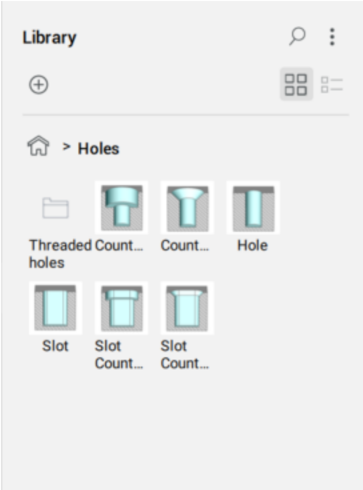
Drawing views of your assemblies, generated with the VIEWBASE command, contain thread annotations for standard parts as thin lines for side views, and as a 270-degree thin arc for top/bottom views.

Example:



29.7.3 Holes

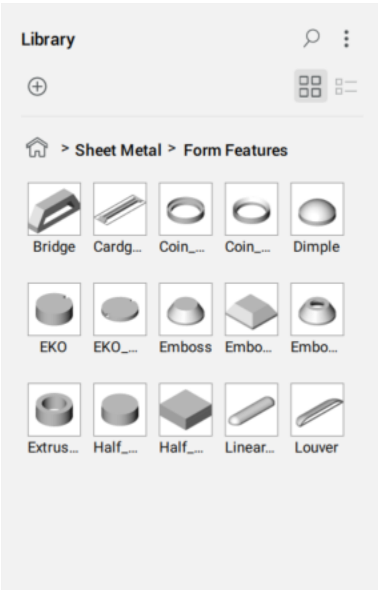
The **Holes** category contains a parametric library of standard holes, such as slots, counterbores, and countersinks.



There is also a subcategory of **Threaded holes**, represented in drawing views.

29.7.4 Sheet Metal Form Features

The **Sheet Metal** category contains a parametric library of sheet metal form features, such as bridges and embosses.

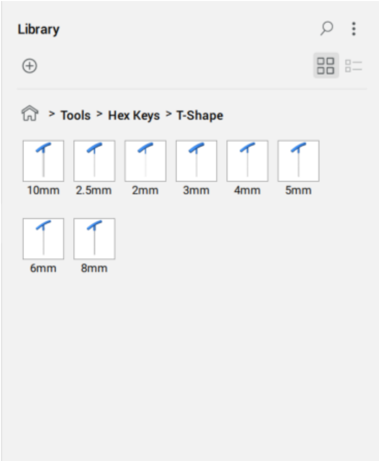


29.7.5 Tools

The **Tools** category contains tools grouped into 4 subcategories:

- **Hex Key**, such as L-Shape and T-Shape hexagonal key
- **Torque Wrenches**
- **Wrench Keys**, such as Combination and Ratcheting Wrenches
- **ScrewDriver**

Example: Tools / Hex Keys / T-Shape



29.7.6 Place a standard component into the current drawing

1 Choose a category.

2 Choose a standard.

3 Select a type.

4 Click the desired standard component.

5 Edit the parameters in the **Properties** panel.

6 Click in the drawing area to insert the selected component.

As most standard parts are mechanical blocks, you can edit them using the BEDIT command. Additionally, it is possible to assign standard, category, type and description for custom standard part, and to override those properties for parts supplied with BricsCAD.

Old standard parts or standard parts inserted as mechanical components when mechanical blocks are disabled (MECHANICALBLOCKS=0), can be also handled.

If mechanical blocks are disabled (MECHANICALBLOCKS = 0), but a file with a mechanical root block is inserted with BMINSERT, a component definition is created using the internal name stored in the mechanical root, and corresponding properties are copied to that definition. If a definition with the same internal name already exists, this will be used.

In BOM tables, standard parts with the same name and parameters will be treated as equal even if one is an old component and another one is a mechanical block.

29.7.7 Create your own standard component

Use the CREATELIBRARYBLOCK command to create your own standard parts and add them to the **Library** panel.

29.7.8 Control the display of threads

1 Open the **Settings** dialog box.

2 Go to the **Thread representation** variable (THREADDISPLAY).

3 Enable the setting.

4 If applicable, update the document to make threads appear on existing parts:

5
- Temporarily set **Assembly components updating mode (BMUPDATEMODE)** to **[1] Update all components**.
- Use the BMUPDATE command to update the document.
- If applicable, set the **Assembly components updating mode (BMUPDATEMODE)** to the previous value.

29.7.9 Control the number of sprocket teeth

1 Open the **Settings** dialog box.

2 Go to the **Maximum number of sprocket teeth** variable (GEARTEETHNUMBER).

3 Set the value to be sufficient for used sprockets (500 is recommended).

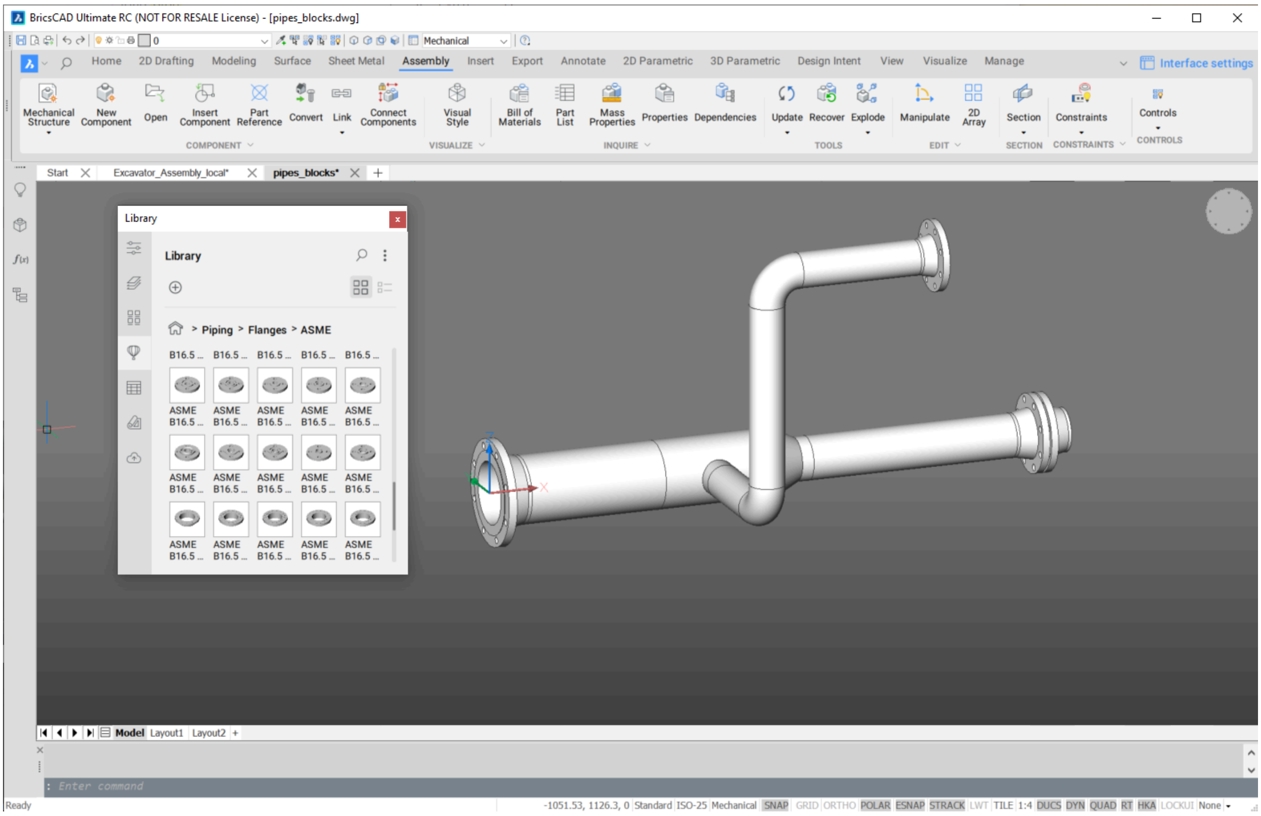
4 ***Note:*** Larger values slow down the update of standard sprockets. If the value is smaller than the required number, only that number of sprocket teeth displays.

5 If applicable, update the document to make sprocket teeth appear on existing parts:

6
- Temporarily set the **Assembly components updating mode (BMUPDATEMODE)** to **[1] Update all components**.
- Use the BMUPDATE command to update the document.
- If applicable, set the **Assembly components updating mode (BMUPDATEMODE)** to the previous value.

29.8 Piping in Assembly Design

29.8.1 Overview



Seamless and welded pipes, elbows, tees, crosses, reducers and flanges are widely used in machinery, automotive, and aviation industries. They are also found in oil-and-gas, chemical industry, and water supply. BricsCAD has a library of parametric piping parts of different standards, such as ASME, DIN, and ISO. These standard parts are parametric. You can easily adjust their parameters, link them to different global parameters, and use 3D constraints to properly connect the parts. BricsCAD allows you to streamline these routine operations using the **Smart mode** of the BMINSERT command and a dedicated BMCONNECT command.

29.8.2 Insert piping parts in your assembly

Drag and drop any piping standard part from the **Library** panel into your assembly.

By default, all parts from the Piping category are inserted in Smart mode, which means that if you hover over a previously inserted piping part, BricsCAD:

- Copies parameter expressions of that part to the corresponding parameters of the part being inserted.
- Adds a set of 3D constraints to connect the parts.

Note: Press the **Ctrl** key to switch between Smart mode and Regular mode, in which 3D constraints with other parts are not generated.

29.8.3 Create 3D constraints between piping standard parts manually

Piping standard parts have a set of wireframe entities (points and lines) on the STD_PARTS_PIPING_CONNECTION layer, which is switched off by default.

Constraining a fitting

To constrain a fitting (elbow, reducer, flange, ...) with another fitting or a pipe, create 2 Coincident 3D constraints between:

- 2 points of the fittings, which should coincide
- 2 lines of the fittings, which should be collinear.

Constraining a pipe

Constraining pipes is more complex because a pipe can have any length. If a pipe is already constrained with a fitting at one side using 2 Coincident 3D constraints, then constraining it from the other side must be different.

You must create:

- A Coincident 3D constraint between the line, which represents the axis of the pipe, and the corresponding line of the other fitting
- A geometry-driven Distance 3D constraint between the planar face of the pipe on the opposite (already constrained) side and the corresponding point of the fitting, which is being connected.

You must then type the name of the distance constraint in the expression field for the Length parameter of the pipe. Because geometry-driven constraints allow controlling of a particular dimension, the pipe will always have the correct length, and such a parametrization will not limit any further editing of your assembly using direct manipulation of additional non-geometry-driven dimensional constraints.

29.8.4 Automatically connect piping parts

The set of constraints is created automatically using the Smart mode of BMINSERT command or the dedicated BMCONNECT command to connect 2 standard parts.

29.8.5 Create a Bill of Materials for a piping assembly

In a BOM you may want all pipes of the same type and size (PartNumber), but with different lengths, listed as the same part

Use the Aggregate function of the BOM Manager to combine all such parts in a single row with their total length.

29.9 Arrays of Components

29.9.1 Overview

Use the ARRAY, ARRAYRECT, ARRAYPOLAR, ARRAYPATH and BMINSERT commands.

BricsCAD allows you to insert the same component multiple times in a regular way. All inserts can be distributed uniformly (at the same distance) along a line, a circle, or an arbitrary curve. Higher-order patterns (for example: rectangular or cylindrical) are also supported. This is achieved by using the technology of associative arrays: rectangular, polar, and path.

Associative arrays are persistent entities, which can be edited (manually or automatically) at any moment after creation to change the number of elements or distance between them. All arrays in your model are listed in the **Mechanical Browser** panel, where they can be edited.

All components can be arrayed - including parts, subassemblies, standard parts, and holes.

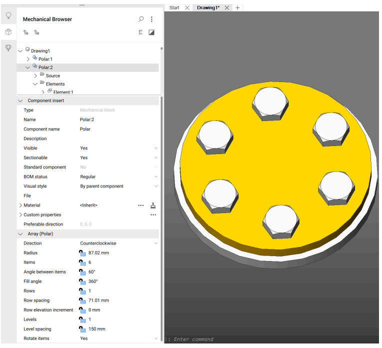
29.9.2 Create an array of components

Use one of the array tools from **Assembly** > **Edit** in the Ribbon, in the **Mechanical** workspace:

- **Rectangular Array**
- **Polar Array**
- **Path Array**

Another way to create an array of component is to use the **Multiple** > **Array** option of the **Insert Component** tool .

29.9.3 Arrays of components in the Mechanical Browser



Arrays of components are represented as special nodes in the **Mechanical Browser** panel. They are named after the array type (**Rectangular**, **Polar**, or **Path**) followed with semicolon and the index of the array (for example, Rectangular:1, Polar:2, Path:3). These nodes have the same properties as regular associative arrays. The properties can be used to change the number of elements in the array, distance/angle between them, and other array parameters. You can also assign a persistent expression to any property. It will be used to automatically recompute the value of the property upon changing its driving parameters.

Each array node has 2 child nodes, **Source** and **Elements**:

- Under **Source** node you can find the inserted component and its parameters.
- **Elements** node can be expanded into a list of array elements, with the possibility to hide or change the visual style of a particular element inside the array.

29.9.4 Arrays of components and Bill of Materials

Arrays of components are transparent for product structure. Components inside arrays are considered as inserted directly into the same assembly. So, in top-level BOM, components inserted into top-level arrays are listed as top-level components.

The **Quantity** column in BOM is filled with the total number of inserts of the same component including the components inside arrays.

29.9.5 Arrays of parametric components, parametric blocks, external parametric references, standard parts, and holes

To provide availability of blocks' parameters inside an array, parameters must be linked (using the **Link to parameter** context menu command) before the array creation. After the array creation, parameters will be available for editing in the **Parameters Manager** panel.

The link to the parameter is removed, but it can be restored again for array's blocks parameters using the same **Link to parameter** context menu item. Technically, the parameters are associated with the source element of an array. When this source element is edited parametrically, all other items of the array are changed after it.

For the former implementation of parametric components (MECHANICALBLOCKS system variable is **Off** and BMINSERT is used) parameters of the components nested inside array are available simply after the array creation.

Standard parts and holes are parametric components and you can edit their parameters inside the arrays. You can easily create a part with many holes and you can control the hole number, positions, and size parametrically plus insert a parametric bolt. This technology allows you to create parametric flange assembly with multiple configurations.

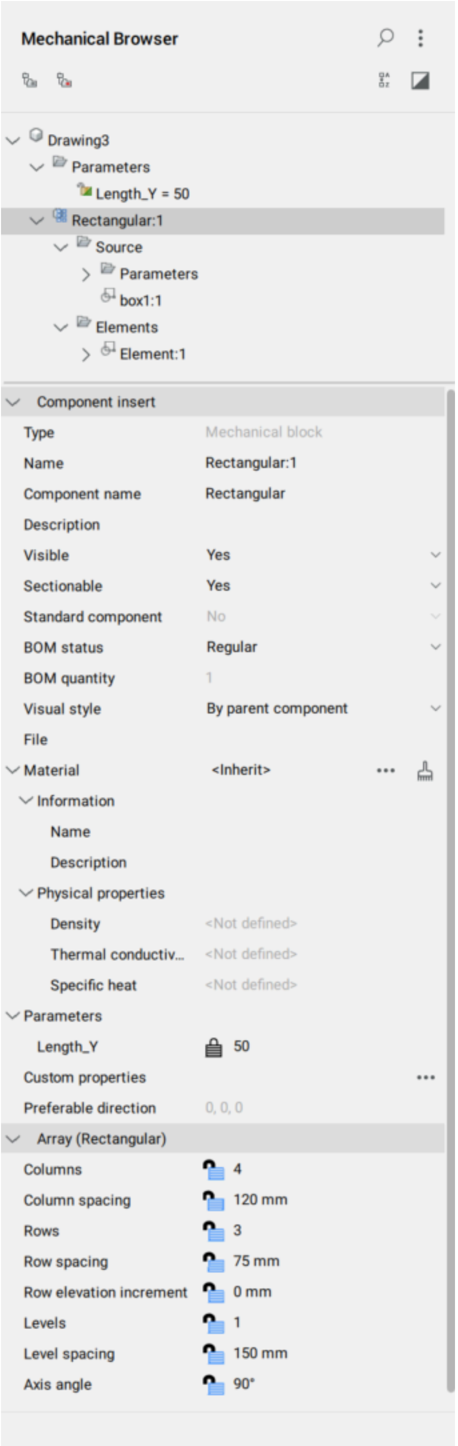
29.10 Array Properties

29.10.1 Overview

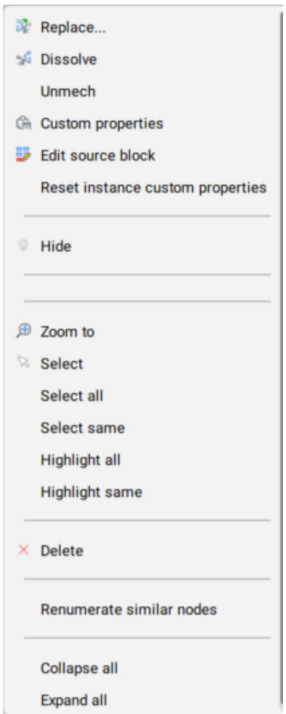
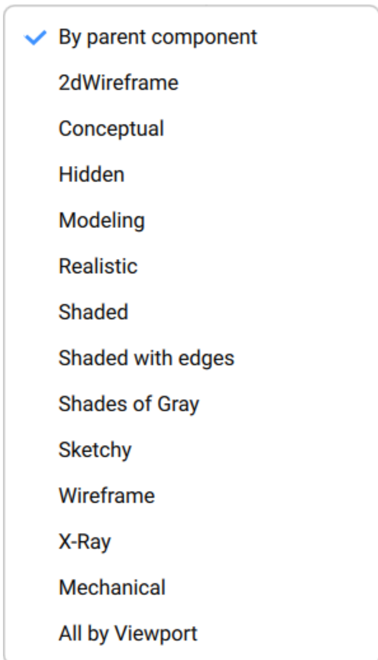
Component array is an associative array made of a parametric component. Component arrays have a special representation in the **Mechanical Browser** panel.

Component array have 2 main subnodes:

- **Source**: node that contains source object and its parameters. Source object can be presented as a component or subassembly with specific properties.
- **Elements**: node that contains all instances of source object.



Component array context menu is displayed by a right click on the array node, while array properties are displayed automatically when selecting an array in the **Mechanical Browser**.

Context menu	Description
Component array context menu	Right-click an array:  • **Replace...**: replaces a component insert (see the BMREPLACE command). • **Dissolve**: dissolves an array created in the current drawing (see the BMDISSOLVE command). • **Unmech**: removes all mechanical structure (see the BMUNMECH command). • **Custom properties**: opens the **Mechanical Properties** dialog box. • **Edit source block**: opens for editing the source entity. • **Reset instance custom properties**: resets instance custom properties to defaults inherited from a corresponding block. • **Hide** / **Show** : hides or shows the selected array. • **Visual Style**: displays the **Visual Style** menu. Visual Styles that are saved in the current drawing are available.  - **By parent component**: renders the selected array according to the shademode of the parent component (see the SHADEMODE command). - **All by Viewport**: renders all arrays according to the current viewport

29.11 Representations

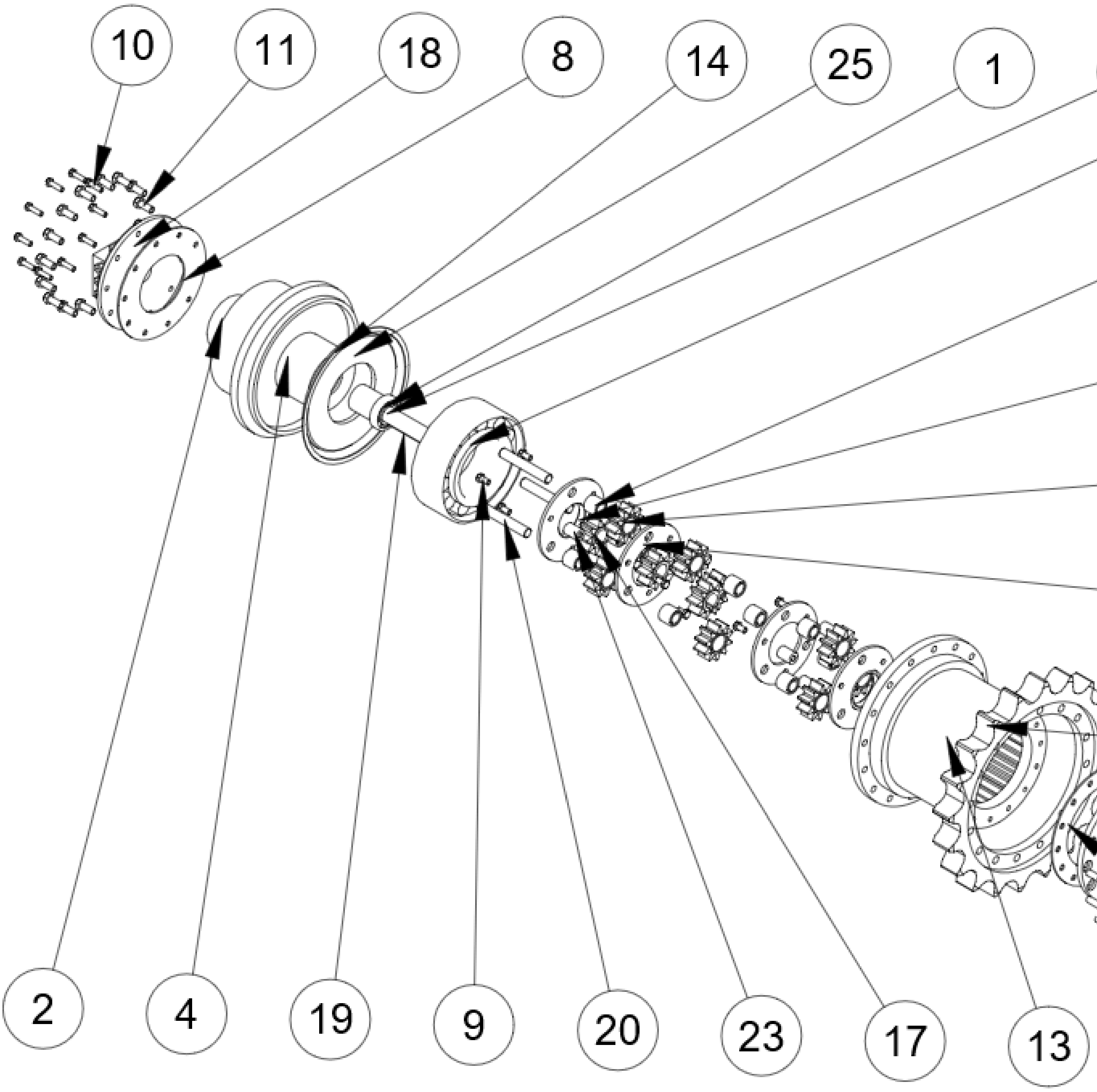
29.11.1 Overview

Use the BMEXPLODE, BMEXPLODEMOVE, BMTRAILINGLINES, and VIEWBASE commands.

The exploded (created with BMEXPLODE command) and sequence (created with BMSEQUENCE command) representations functionality create associative representations of assemblies and do not modify the assembly itself. Representations are stored in dedicated blocks, which can be edited and inserted according to your needs. You can also generate drawing views from the representations.

Representations are associative. Each part in a representation is linked with the corresponding part in the assembly. Use the BMBALLOON command to update exploded representations to the current state of the assembly, as well as place balloon annotations on the corresponding drawing view(s).

Example of Exploded Representation



29.11.2 Levels of the exploded representation

Representations contain references to the components.

2 levels of Representations are supported: Top and Bottom.

- **Top level**:

  The representation contains references to the top-level components only. Top-level components are components inserted directly into the main assembly. A top-level component in a representation is not exploded further - all its parts and subassemblies remain assembled. A top-level representation allows you to see the basic composition of your assembly.
- **Bottom level**:

  The representation contains references to the bottom-level components only. Bottom-level components are terminal parts, which do not contain other parts or subassemblies. Bottom-level representations allow you to view all parts in isolation.

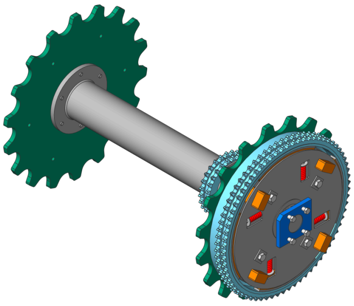
Note: If **Bottom level** is selected and there are some middle-level components containing not only inserts of components, but also other entities, these entities will be lost in the representation because it is not possible to reference them correctly. Those entities must be converted into corresponding bottom-level components first.

29.11.3 Algorithms

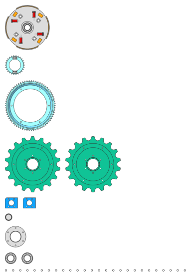
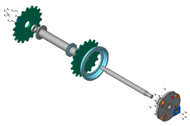
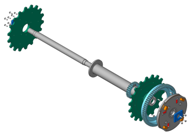
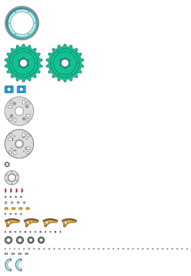
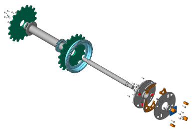
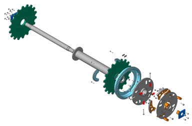
BricsCAD provides 3 algorithms to automatically create representations and a manual algorithm.

Algorithm	Description
Table by Type	Creates a table-like explosion where components of the same type are grouped in rows.
Linear	Finds the disassembly sequence of components in a given direction and order the components. ***Note:*** This considers possible physical collisions between components. The components can be moved by the algorithm only if there are no other components (not yet moved), that block them.
Automatic	Finds the disassembly sequence of components with respect to the assembly hierarchy. ***Note:*** This considers all possible physical collisions. For each part or subassembly, the direction of movement is identified automatically.
Manual	Creates an exact copy of the assembly ready for custom edit. In this mode, you can create a custom representation without changing the main assembly.

Example: The main drive shaft assembly



Example: main drive shaft assembly

	Table by Type	Linear	Automatic
Top level			
Bottom level			

29.11.4 Steps and animations

Representations may contain one or several steps, which describe a particular assembly or disassembly sequence. Each step corresponds to a set of components that must be moved at this particular stage in order to assemble or disassemble the main assembly. All representation algorithms automatically create all the required steps. You can also delete, merge, split, move, or add extra steps.

Each step has a unique name that can be used to describe the step.

It is possible to animate a particular step, as well as the entire sequence of steps, both in direct and reverse directions (to get disassembly and assembly animations). In order for the animation to work properly, an initial step that defines the initial state of the assembly must be present in the representation. Use the **Auto hide** property of the representation to automatically hide components that are not important for a particular step during the animation.

29.11.5 Edit representations

Each representation is stored as a block. This allows you to edit the representation with the block editor.

Important: Do not use the REFEDIT command to edit representations. This command is incompatible with exploded representation steps.

To open the representation for edit:

1 Right-click the required representation in the **Mechanical Browser**.

2 Select **Edit** from the context menu.

The block editor will open automatically.

Note: You can open the representation for edit by double-clicking it.

Tip: As an alternative, you can open the representation for edit at the end of the BMEXPLODE command.

When the representation is open for edit, you can change the state of a step to the current step by double-clicking it.

Note: If you change the current step, you will discard all unsaved modifications, as well as any operation on steps. Make sure that all modifications are saved before you change the current step or perform any other operation.

Use the DMMOVE, MOVE, DMROTATE, and BMEXPLODEMOVE commands to move components, inside the representations, to the required positions. The BMEXPLODEMOVE command automatically adds all necessary steps after the current step. For other operations, save the current step after the edit is complete. Only the difference between the current step and the previous one is stored.

It is also possible to change the representation by performing operations on steps.

Example: if you remove a step, all components moved at this step will be returned to their respective positions before that step.

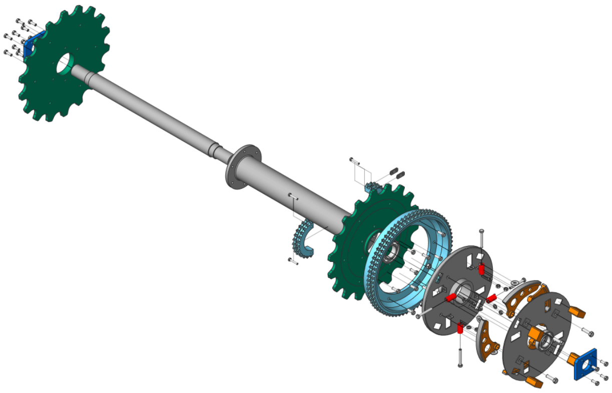
It is also possible to associate text annotations (MTEXT command) and any entities other than assembly parts with a selected step. Those annotations and entities will be visible only at the selected steps and will be hidden when other steps are active; however, their positions are not stored in steps.

Save the current step (**Save current state** option) once you have finished the edit. If not, changes will be discarded.

29.11.6 Trailing lines

Trailing lines help explain the relationship between parts. It is a 3D polyline that represents a trajectory and displays the movement of a part during the disassembly process.

Example of trailing lines



Use the BMTRAILINGLINES command to add trailing lines to a representation that is open for edit. This tool creates all trailing lines. The information from the exploded view steps is used to do this.

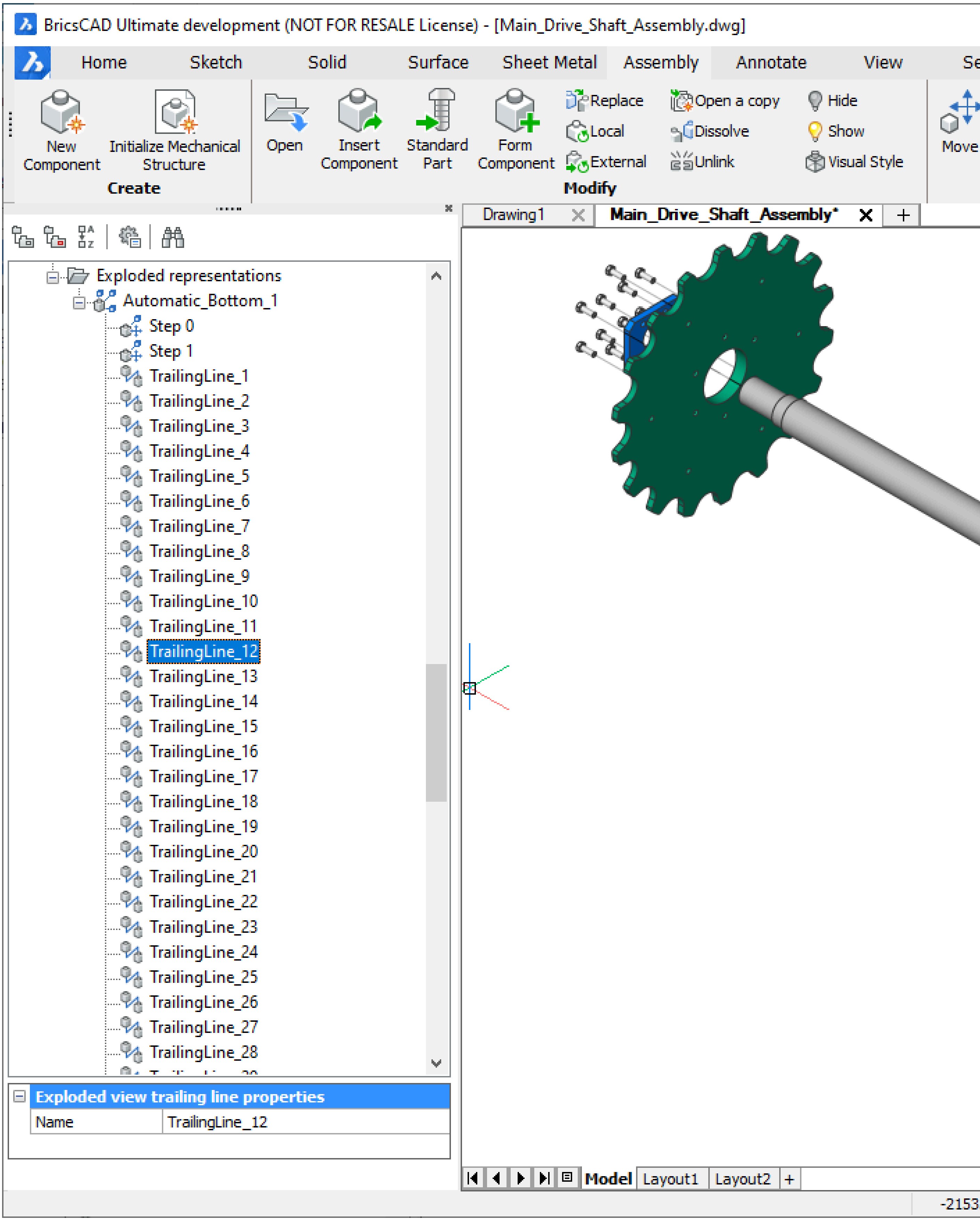
You can generate trailing lines for all parts, for a subset of parts, or for each step. For a subset, trailing lines will be built for parts from that subset only, although only necessary trailing lines will be added. Some parts may not have trailing lines at all. You can also manually select 2 parts to add a trailing line between. A trailing line for a step will be shown only for this step.

When a trailing line is computed, the algorithm automatically determines the location of the part and the location from where the part was removed to compute the trajectory. It considers the movements of both parts.

Each trailing line is displayed in the **Mechanical Browser** panel. With the **Mechanical Browser** panel, you can see all trailing lines in the model; highlight, select and zoom to parts connected with the selected line; rename or remove the trailing line.

You can edit the trailing line(s) with the standard 3D polyline edit tools.

Exploded view trailing line properties


BricsCAD Ultimate development (NOT FOR RESALE License) - [Main_Drive_Shaft_Assembly.dwg]
Home
Sketch
Solid
Surface
Sheet Metal
Assembly
Annotate
View
New Component
Initialize Mechanical Structure
Create
Open
Insert Component
Standard Part
Form Component
Replace
Local
External
Open a copy
Dissolve
Unlink
Hide
Show
Visual Style
Modify
Move
Drawing1
Main_Drive_Shaft_Assembly*
Exploded representations
Automatic_Bottom_1
Step 0
Step 1
TrailingLine_1
TrailingLine_2
TrailingLine_3
TrailingLine_4
TrailingLine_5
TrailingLine_6
TrailingLine_7
TrailingLine_8
TrailingLine_9
TrailingLine_10
TrailingLine_11
TrailingLine_12
TrailingLine_13
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TrailingLine_18
TrailingLine_19
TrailingLine_20
TrailingLine_21
TrailingLine_22
TrailingLine_23
TrailingLine_24
TrailingLine_25
TrailingLine_26
TrailingLine_27
TrailingLine_28
Exploded view trailing line properties
Name
TrailingLine_12
Model
Layout1
Layout2
-2153

29.11.7 Create a representation

Use the BMEXPLODE command to create a representation of the current assembly.

Note: Local mechanical components based on blocks and solids, and mechanical external references are fully supported in exploded views.

1 Choose one of the following actions to launch the BMEXPLODE command:

1
- Click the **Explode** tool button on the **Tools** panel of the **Assembly** tab.
- Click the **Explode** tool button on the **Assembly** toolbar.
- Select **Explode** in the **Assembly** menu.
- Type BMEXPLODE in the Command line.

You are prompted: Select explosion algorithm [Table by types/Linear/Automatic/Manual/Update/Settings]:

2 If applicable, set the **Level** and the **set Name** of the exploded representation using the **Settings** option.

3 Select the algorithm.

4 You are prompted: Select exploded view behavior [Edit/Generate drawing views/Finish]:

5 Perform one of the following actions:

6
- Select **Edit** to edit the exploded representation.
- Select **Generate drawing views** to generate drawing views of the exploded representation.
- Select **Finish** to finish the command.

29.11.8 Generate drawing views of the representation

1 Launch the VIEWBASE command.

1 You are prompted: Select objects or [Entire model/preseTs/Special views] <Entire model>:

2 Select the **Special views** option.

You are prompted: Select view [Exploded view/Unfolded view/Back]:

3 Select the **Exploded view** option.

A dialog box to select the representation and the step appears.

4 Select the representation.

5 You are prompted: Enter new or existing layout name to make current <Layout1>:

6 If applicable, enter the *Layout name* of the layout for the representation.

7 Position the generated drawing views of the representation in the paper space layout.

Note: To use balloons (see the BMBALLOON command) with drawing views of **Representations**, the GENERATEASSOCVIEWS system variable must be ON.

29.11.9 Update a representation

Run the BMUPDATE command or the Update option of the BMEXPLODE command to synchronize representations with the current state of the assembly.

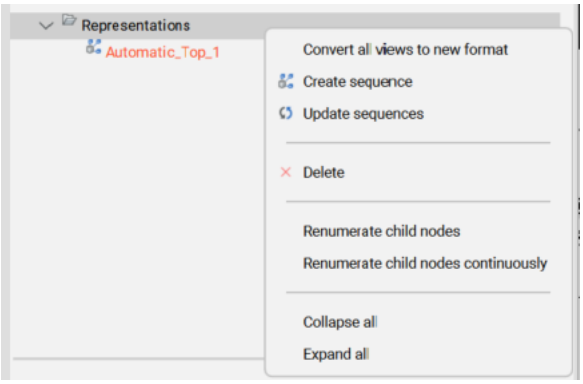
If applicable, adjust the representation. Such adjustments may be required because of the addition and/or deletion of some parts.

Note: The position of parts in **Representations** created in **Manual** and **Linear** modes are not updated automatically.

29.11.10 Manage representations

Manage representations

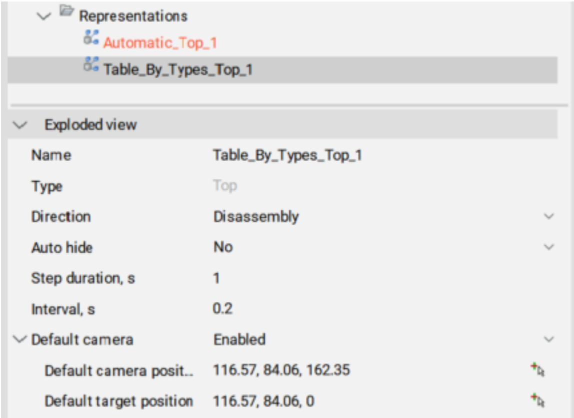
Open the **Mechanical Browser** to see all representations in the document. All representations and sequences will be listed under the **Representations** group. Open the context menu for the selected representations to see all supported operations in this context:



Note: The **Convert all views to new format** option is available only when there are old format representations in the drawing.

Properties of exploded views

Each exploded view has a set of properties. You can edit some of the properties.

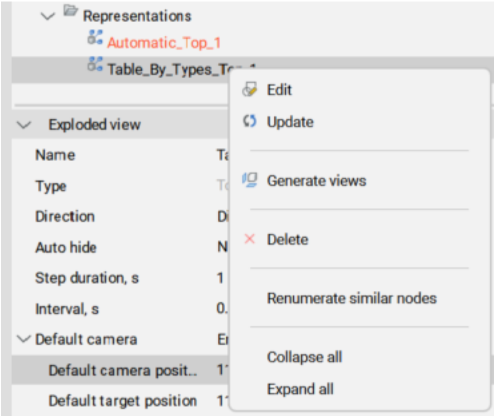


Property	Description
Name	Name of the representation. This must be unique for all representations, as well as for all the blocks in the model.
Type	Type of the representation: *Top* or *Bottom*.
Direction	The direction of animation. Choose: *Disassembly* (from start to end) or *Assembly* (from end to start).
Auto hide	If set to *Yes*, all unimportant parts for a particular step will be hidden during the animation.
Step duration, s	Default duration of each step during the animation (in milliseconds). The default value is 1000, which is equal to 1s duration.
Interval, s	Default interval between steps in milliseconds.
Default camera	Sets the default camera action. If **Enabled**, uses the default camera position and target for all steps which have the **Camera** set as default. If **Disabled**, uses the camera position and target of the previous step for a step which has the **Camera** set as default.

Operations on representations

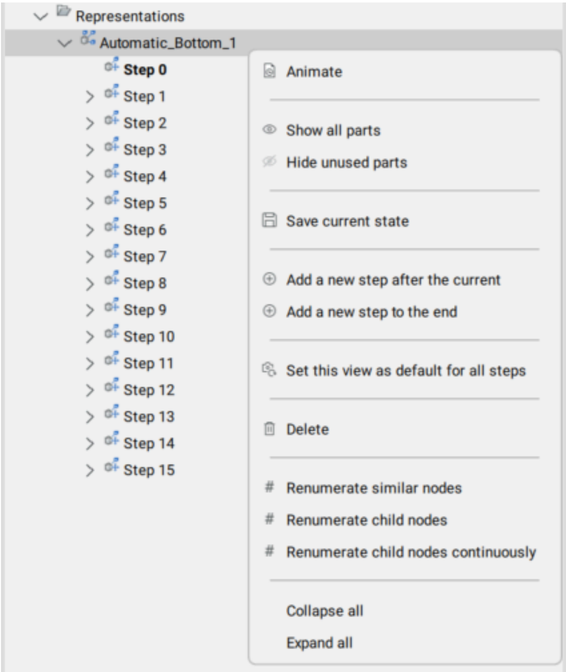
The supported operations depend on whether or not the representation is opened for edit.

If the representation is **not opened for edit**, the following operations are possible:



Operation	Description
Edit	Opens the representation for edit. ***Note:*** If configured, the custom camera of the current step is used. Otherwise, if enabled, the default camera of the exploded view is used, and if not, the current camera is used. ***Tip:*** You can also edit the representation with the BEDIT command, or by double-clicking it.
Update	Synchronizes the representations with the current state of the assembly.
Convert to new format	Converts old format representation to the new one. ***Note:*** This option is available only when there are old format representations in the drawing.
Generate views	Generates views for the representation.
Delete	Removes the representation and the associated block from the document.
Renumerate similar nodes	Renumerates representation nodes. Each group of nodes with identical names will have their own numeration.

If the representation is **opened for edit**, the following operations are possible:



Operation	Description
Animate	Animates the entire sequence of steps.
Show all parts	Shows all parts in the step.
Hide unused parts	Hides unused parts from the step.
Save current state	Saves the current positions of parts in the current step.
Add a new step after the current	Adds a new step after the current one. ***Note:*** The new step will be automatically set as *current*. All unsaved modifications of the exploded representation will be lost.
Add a new step to the end	Adds a new step after the last one. ***Note:*** The new step will be automatically set as *current*. All unsaved modifications of the exploded representation will be lost.
Set this view as default for all steps	Sets the default camera position, which is associated with the current exploded view itself and not with the initial step. This camera position will be used as default orientations for all steps that use default camera position. It is possible to have a custom camera for initial step and default camera that are different.
Delete	Removes the representation and the associated block from the document.
Renumerate similar nodes	Renumerates representation nodes. Each group of nodes with identical names will have their own numeration.
Renumerate child nodes	Renumerates similar step nodes of current representation.
Renumerate child nodes continuously	Renumerates all step nodes of current representation in spite of names.

In addition, all steps will be listed in the **Mechanical Browser** for this representation.

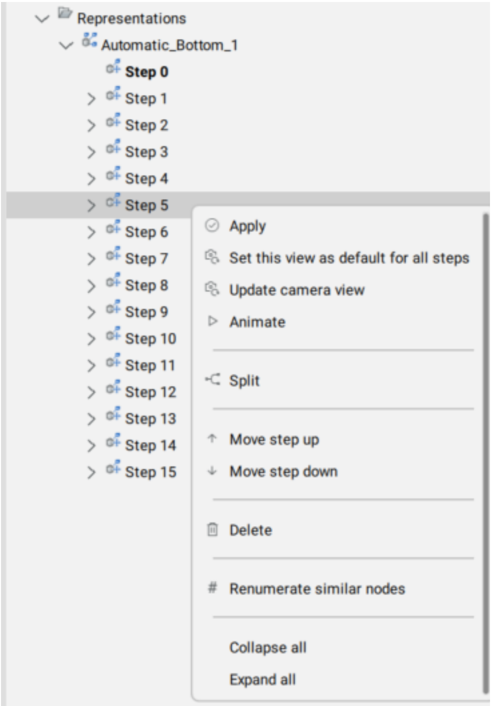
29.11.11 Manage representation steps

Open the **Mechanical Browser** and open the required representation for edit. All steps will be listed under the corresponding node in the tree.

Properties of steps

Property	Description
Name	The name of the step. Must be unique for this representation.
Duration, s	The duration of the step. By default, this is equal to the Step duration set for the representation.
Duration type	Specifies if the duration must be taken from the representation, or if it is specific for this particular step.
Duration value, s (only for Absolute duration type)	The duration for this step in seconds.
Camera	The camera position: **Default**: • uses the **Default camera** position, if enabled. • uses the previous step camera position if **Default camera** position is disabled.
Camera position (only for custom camera)	Displays the camera position. By pressing the icon from the right side, user can change the camera position.
Target position (only for custom camera)	Displays the camera target position. By pressing the icon from the right side, user can change the target position.

Operations on steps



Operation	Description
Apply	Sets the step as the current and update the parts in the representation according to this step. ***Note:*** Double-clicking the step has the same result.
Set this view as default for all steps	Sets the camera position for the current step. This camera position will be used as default orientations for all steps that use default camera position. If no specific camera position is associated with a step, it will inherit the camera position from the initial step. For newly created exploded views, the initial step will get the same camera position as the model.
Update camera view	Updates the camera position only for the current step.
Animate	Animates the step.
Merge	Merges two or more consecutive steps.
Split	Splits the step to a sequence of steps, each of them corresponding to exactly one part.
Move step up	Moves the step one position up in the tree.

Operation	Description
Move step down	Moves the step one position down in the tree.
Delete	Completely removes the step as well as the associated movements of parts. After that, the positions of parts are updated in the exploded representation.
Renumerate similar nodes	Renumerates step nodes. Each group of nodes with identical names will have their own numeration.

29.11.12 Edit representations

1 Select the representation in the **Mechanical Browser**.

2 Right-click the exploded representation and select **Edit** from the context menu.

   Note: As an alternative, you can run the BEDIT command and select the block with the exploded representation.

3 Perform the necessary operations to get the desired representation.

4 Save the current step.

5 Save changes using the BCLOSE command with the option **Save**.

Note: Do not use the **Discard** option of the BCLOSE command, as it can lead to an inconsistent state of the block and exploded representation steps. Use **Undo** instead.

29.11.13 Add trailing lines

1 Open the representation for editing.

2 Run the BMTRAILINGLINES command.

3 Select the subset of parts to create trailing lines for, or select the entire model.

4 Select the points of the parts to be used in order to create the trajectory: origin or center.

5 If applicable, edit the resulted trailing lines.

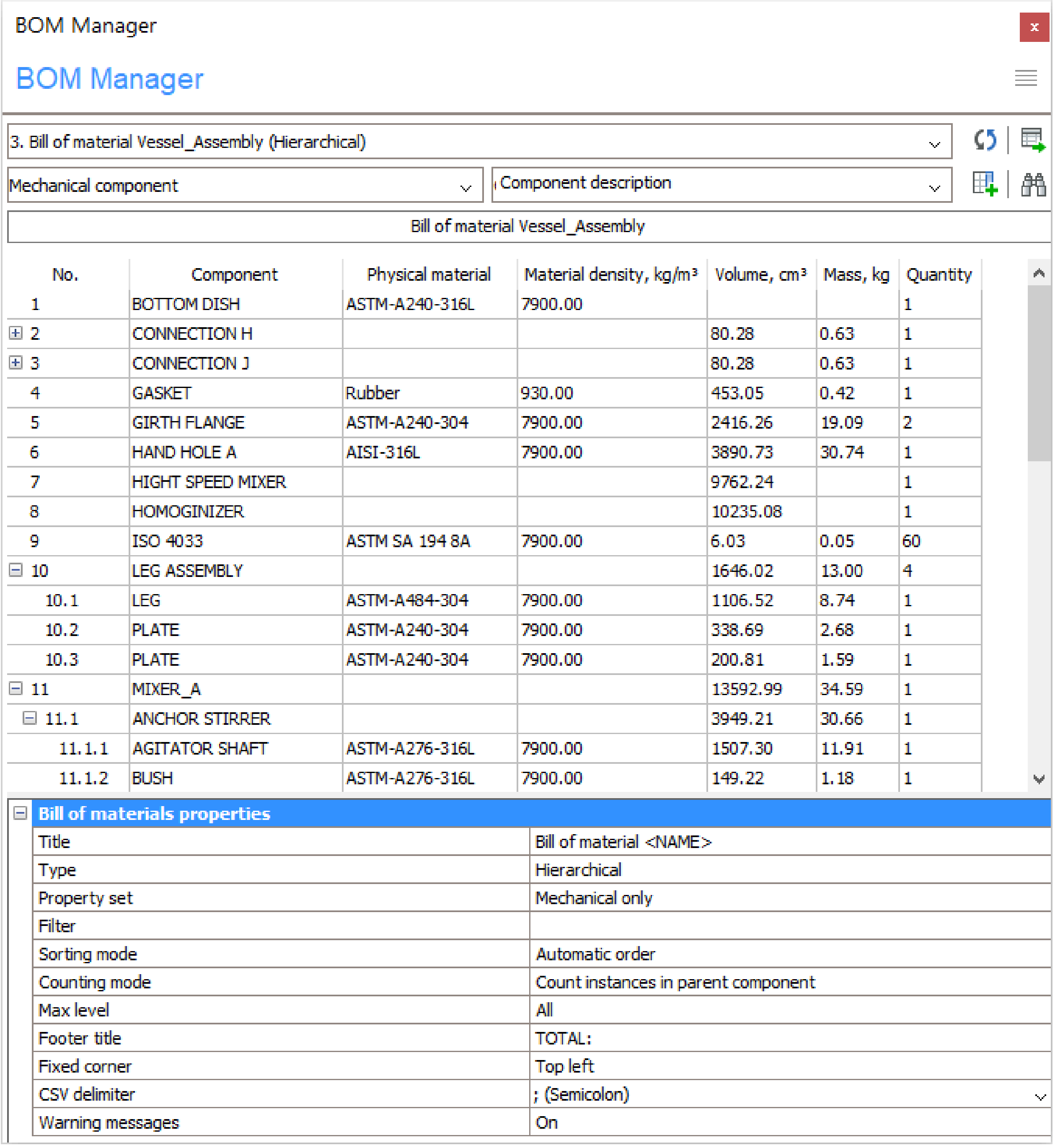
6 Save changes.

29.12 Bill of Materials Manager

29.12.1 Overview

Use the BMBOMPANELOPEN, BMBOMPANELCLOSE, BMBOM and BMBOMEDIT commands.

The **BOM Manager** panel (Bill of Materials Manager panel) allows you to browse, create and edit bills of materials (BOMs) in the current drawing. Use the manager to interactively configure, both new and existing, bills of materials. You can save the configuration of a bill of materials as a template file and reuse it later in other drawings. You can also add a bill of materials that is not linked to a table in the drawing.



With the **BOM Manager** panel, you can:

- Browse existing bills of materials in the drawing.
- Create a new bill of materials using either a template file stored in BOMTEMPLATE system variable, a standard type (such as top-level, bottom-level, hierarchical) or an existing BOM template. If BOMTEMPLATE system variable contains a valid path to a BOM template file, this file will be used with priority to create a new BOM table.

- Edit bills of materials:
  - Add or remove columns (includes formula columns).
  - Move selected columns to new positions.
  - Rename the columns.
  - Configure how the BOM table is sorted.
  - Configure the footer value for each individual column.
  - Configure the units and how they are displayed for each column.
  - Configure the width of each column with column properties.
  - Configure the visibility of each individual column.
  - Filter the components with a filter expression.
  - Change the title of both the title row and the footer row.
- Search for components in the table.
- Highlight and select component instances.
- Save the BOM configuration as a template file or as a default template file (stored in BOMTEMPLATE system variable).
- Load a BOM table configuration from a template file.
- Place one or more BOM table instances in a drawing.
- Delete BOM tables.

Note: A Mechanical or Ultimate license is required to modify or create bills of materials. However, the manager is available with a Pro license.

29.12.2 Open the Bill of Materials Manager

Do one of the following:

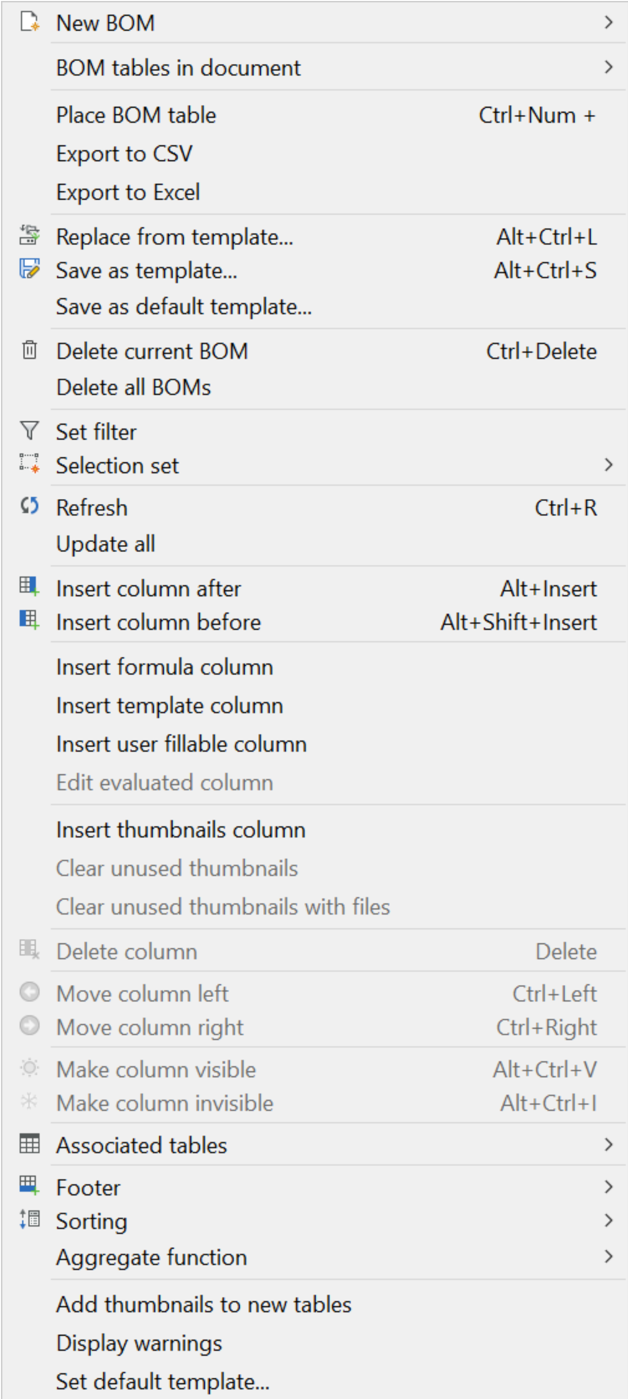
- Place the cursor on a toolbar, right-click, choose **Panels**, and choose **BOM Manager** from the context menu.
- Launch the BMBOMPANELOPEN command.

29.12.3 Tools

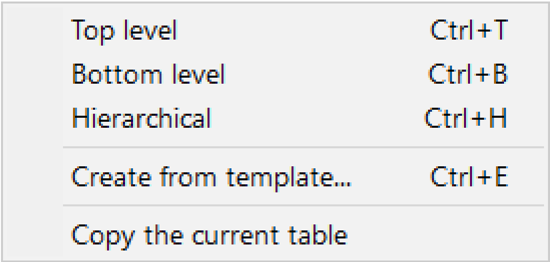
Most of the configuration operations in the manager are available in the main menu. A context menu is available for selected columns or rows. You can also change the properties of the selected columns and the BOM itself in the properties section of the manager. To work with a particular column, select it in the BOM grid. Multiple columns can be selected simultaneously.

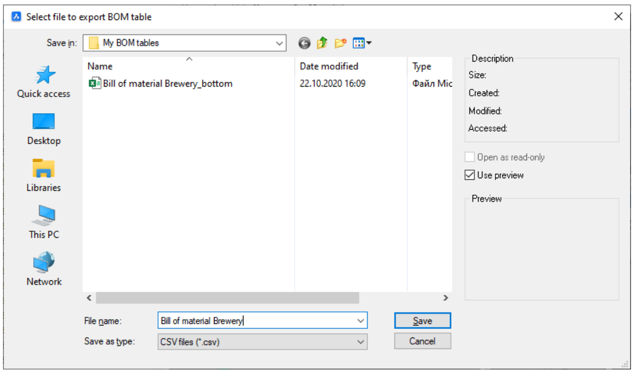
Main menu

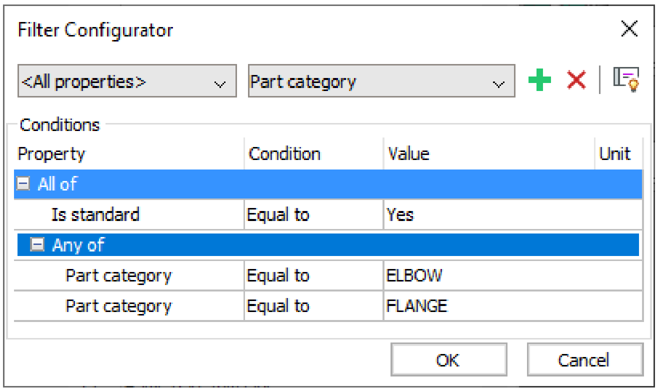
Click **Menu** (☰) at the top right corner of the **BOM Manager** panel.

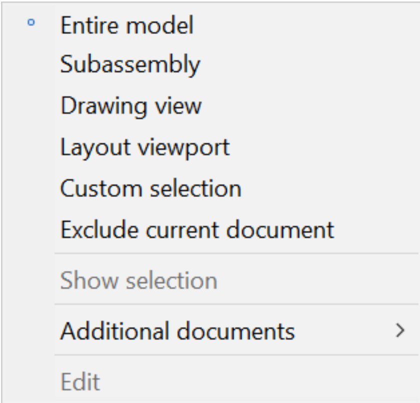

New BOM
BOM tables in document
Place BOM table Ctrl+Num +
Export to CSV
Export to Excel
Replace from template... Alt+Ctrl+L
Save as template... Alt+Ctrl+S
Save as default template...
Delete current BOM Ctrl+Delete
Delete all BOMs
Set filter
Selection set
Refresh Ctrl+R
Update all
Insert column after Alt+Insert
Insert column before Alt+Shift+Insert
Insert formula column
Insert template column
Insert user fillable column
Edit evaluated column
Insert thumbnails column
Clear unused thumbnails
Clear unused thumbnails with files
Delete column Delete
Move column left Ctrl+Left
Move column right Ctrl+Right
Make column visible Alt+Ctrl+V
Make column invisible Alt+Ctrl+I
Associated tables
Footer
Sorting
Aggregate function
Add thumbnails to new tables
Display warnings
Set default template...

Main menu

Icon	Tool	Description
	**New BOM**	Opens a submenu:  ***Note:*** The BOMTEMPLATE system variable defines the path to the file with default BOM template which will be used to create new BOM tables. ***Note:*** If the BOMTEMPLATE system variable is empty or the path is not valid, the below default BOM configuration will be used. • **Top level**: creates a new, top-level components only, BOM. • **Bottom level**: creates a new, bottom-level components only, BOM. • **Hierarchical**: creates a new hierarchical BOM, which contains all components with their subcomponents listed directly below. • **Create from template**: opens a dialog box to select a BOM configuration template file. • **Copy the current table**: creates a new BOM based on the current table.

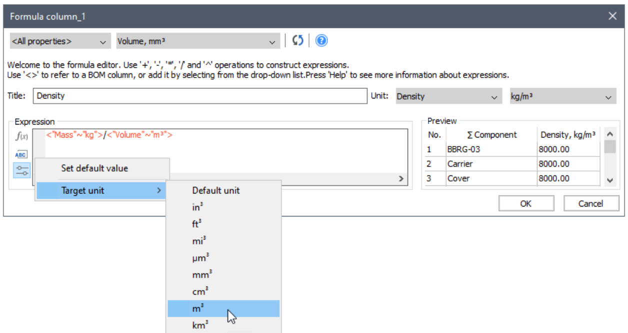
Icon	Tool	Description
	BOM tables in the document	Opens a submenu:  • **Go to first table**: switches to the first BOM table in the document. • **Go to previous table**: switches to the previous BOM table in the document. • **Go to next table**: switches to the next BOM table in the document. • **Go to last table**: switches to the last BOM table in the document. • **<Name of the BOM>**: switches to the corresponding BOM table in the document.
	Place BOM table	Starts the interactive placement of the current BOM table in the drawing. ***Note:*** Multiple instances of a BOM table can be inserted into the document.
	Export to CSV and **Export to Excel**	Saves the content of the current BOM table to a CSV or a XLSX file. Opens a dialog box to select the location for the CSV or the XLSX file:  ***Note:*** You can adjust the separator for CSV values using the table properties.

Icon	Tool	Description
	Replace from template	Replaces the current BOM table configuration with the selected template. Opens a dialog box to select the template file.
	Save as template	Saves the configuration of the current BOM table as a template. Opens a dialog box to select the location for the new template file.
	Save as default template	Saves the configuration of the current BOM table as a default template. Opens a dialog box to select the location for the new template file. The BOM file path will be stored in BOMTEMPLATE system variable and will be used as default template for newly created BOM tables.
	Delete current BOM	Deletes the current BOM table from the document.
	Delete all BOMs	Deletes all available BOMs.
	Set filter	Opens a dialog to configure the **main filter**: 

Icon	Tool	Description
	Selection set	Opens a submenu:  • **Entire model**: uses entire model as a selection set. • **Subassembly**: allows you to select a subassembly to generate BOM table for. • **Drawing view**: synchronizes BOM with a selected drawing view. • **Layout viewport**: only parts visible in the selected layout viewports will be included in BOM. • **Custom selection**: allows to select top-level components to be included in the BOM table. • **Exclude current document**: the BOM table will not include objects from the currents document. • **Show selection**: highlights the current selection set. • **Additional documents**: the BOM table will include parts from other documents. - **Add drawing or sheet set**: the BOM table will include objects from the selected drawing or sheet set. - **Add folder**: the BOM table will include objects from drawings or sheets set which are present in a folder. ***Note:*** You may include only the documents that match specified wildcards, entire BIM project or entire sheet set. • **Edit:** allows you to change objects included in the current selection set.
	Refresh	Updates the current BOM, and list all BOM tables in the drawing.
	Update all	Updates all BOM tables in the drawing.

<table>
<tr><th>Icon</th><th>Tool</th><th>Description</th></tr>
<tr><td></td><td>Insert column after</td><td>If a column is selected: inserts a column from the Column selector after the selected column in the table.
If no column is selected: adds a new column after the last column in the table.</td></tr>
<tr><td></td><td>Insert column before</td><td>If a column is selected: inserts a column from the Column selector before the selected column in the table.
If no column is selected: adds a new column as the first column in the table.</td></tr>
<tr><td></td><td>Insert formula column</td><td>

Opens the **Formula column** dialog box to configure a formula column:



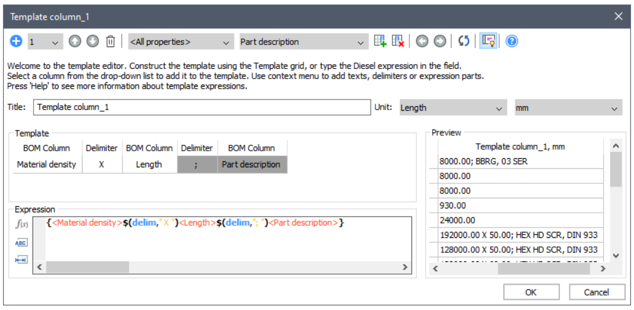
Click **OK** to add a new formula column after the selected one. If no column is selected, a new formula column is added after the last column in the table.

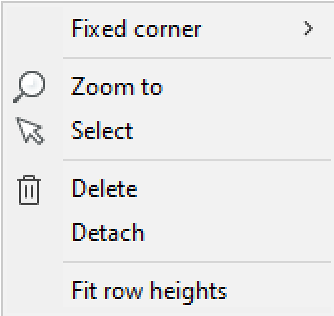
In formula and template expressions, their syntax was extended to allow to set target unit for variables (if the variable supports a unit conversion). It allows to write unit-dependent formula (such as density computation) without bothering about the initial unit of the variable.

If a formula unit is known to BricsCAD, then unit conversions will be supported for formula values.

The default unit mode for formula columns is set to be user units.

</td></tr>
</table>

Icon	Tool	Description
	Insert template column	Opens the **Template Column** dialog box to configure a template column:  Click **OK** to add a new template column after the selected one. If no column is selected, a new template column is added after the last column in the table. ***Note:*** Template column support the same units mode as a normal column, with options to select best for each or best for all units.
	Insert user fillable column	Inserts a user fillable column.
	Edit evaluated column	Opens a dialog to edit the selected formula or template column.
	Insert thumbnails column	**If a column is selected**: inserts a thumbnail column with default parameters after the selected column in the table. **If no column is selected**: adds a new thumbnail column after the last column in the table.
	Clear unused thumbnails	Removes blocks and attachments with thumbnail images that are no longer used.
	Clear unused thumbnails with files	Removes blocks, attachments and files with thumbnail images that are no longer used.
	**Delete column**	Removes the selected column from the table. You can also select a column in the **BOM grid** and hit the **Delete** key to remove a column.
	**Move column left**	Moves the selected column(s) one column to the left.
	**Move column right**	Moves the selected column(s) one column to the right.

Icon	Tool	Description
	Make column visible	Shows the selected column in the associated table in the drawing.
	Make column invisible	Removes the selected column from the associated table. Invisible columns remain visible in the BOM manager but are marked with a diagonal hatch pattern in the background.
	Associated table	Opens a submenu:  • **Fixed corner**: sets a fixed corner table to Top left, Top right, Bottom right or Bottom left. • **Zoom to**: zooms to the **table**. If necessary, set the table layout as the current one. • **Select**: selects the **table**. If necessary, set the table layout as the current one. • **Delete**: deletes the **table** from the document. • **Detach**: detaches the **table**. The table in the document is no longer updated when the BOM table is modified. • **Fit row height**: sets the row height that best fit the text or thumbnails height.

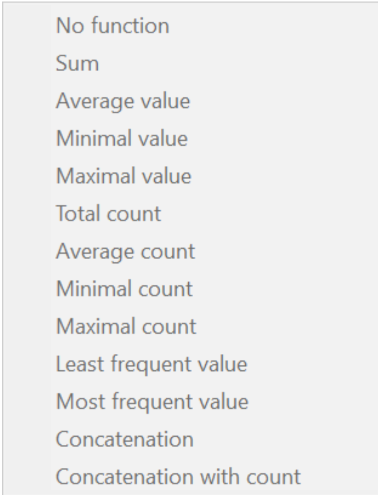
<table>
<tr><th>Icon</th><th>Tool</th><th>Description</th></tr>
<tr><td></td><td>Footer</td><td>Opens a submenu:

No footer Alt+Ctrl+N
Sum Alt+Ctrl+U
Average value Alt+Ctrl+A
Minimal value Alt+Ctrl+M
Maximal value Alt+Ctrl+X

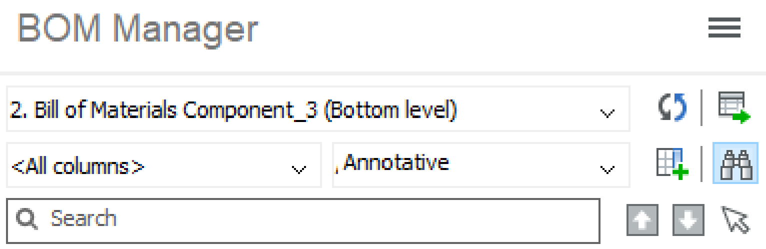
For each of the selected columns:
No footer: removes the footer.
Sum: displays the sum in the footer.
Average value: displays the average in the footer.
Minimal value: displays the minimum value in the footer.
Maximal value: displays the maximum value in the footer.
Note: Footers for template columns are supported, if they have mostly numeric values.

For more details, see the Footer article.</td></tr>
<tr><td></td><td>Sorting</td><td>Opens a submenu:

Automatic order Ctrl+Num *
Natural order Ctrl+Num /
Fixed order
Ascending order Ctrl+Up
Descending order Ctrl+Down
Automatic order: sorts the table automatically.
Natural order: sorts the table by the natural order of component instances in the document.
Fixed order: the positions of parts in the BOM are remembered, and their numbers stay the same after update, regardless of added or deleted parts.
Ascending order: sorts the selected columns in ascending order.
Descending order: sorts the selected columns in descending order.
For more details, see the Sort a BOM table article.</td></tr>
</table>

Icon	Tool	Description
	Aggregate function	Opens a submenu:  • **No function**: removes the aggregate function. • **Sum**: displays the sum of all values in the group. • **Average value**: displays the average of all values in the group. • **Minimal value**: displays the minimum of all values in the group. • **Maximal value**: displays the maximum of all values in the group. • **Concatenation**: displays a concatenation of all values in the group in ascending order, separated by a specified delimiter. • **Concatenation with count**: displays a concatenation of all values in the group with their corresponding number of encounters in ascending order. Value and count pairs are separated by a specified delimiter.
	Add thumbnails to new tables	Changes the setting to add thumbnail column to any new BOM table.
	Display warnings	Controls the display of warning messages during the creation or update of BOM tables.
	Set default template	Opens the **Select file with BOM table template** dialog box to choose a BOM table template (*.bom) file.

Toolbars

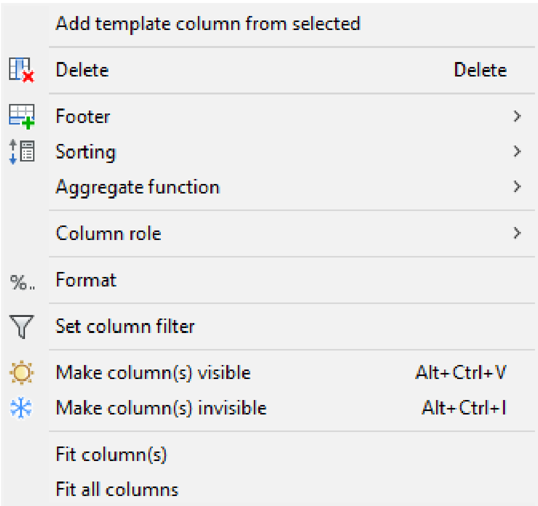


Icon	Tool	Description
⋮	**Main menu**	Opens the main menu.
2. Bill of Materials Componen	**BOM Selector**	Selects the current BOM table. The selection is stored in the drawing. When the drawing is closed and reopened, the selected BOM is restored in the manager. When selected, the table is updated to the current state of the drawing.
	Refresh	Updates both the current BOM and the list of all BOM tables in the drawing.
	Place table	Starts the interactive placement of a table in a drawing. ***Note:*** It is possible to insert multiple instances of a BOM table into the drawing.
<All properties>	**Category selector**	Selects a properties category.
Annotative	**Column selector**	Selects the column to be added to the current BOM. You can also use this to enter a formula for a formula field. To add a formula, type an expression starting with "=", such as "=<Mass>*<Quantity>". The formula may refer to other columns. Use their display names enclosed in <>.
	Insert column after	**If a column is selected**: inserts the column from the **Column selector** after the selected column in the table. **If no columns are selected**: adds the new column after the last column in the table.

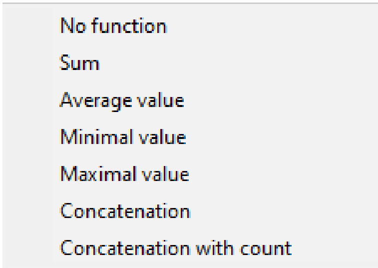
Icon	Tool	Description
	Find	Toggles the search field On and Off. You can also use **Ctrl+F**.
Search	**Search field**	Enters a search string; wildcard characters are supported.
	Find previous	Selects the previous match in the BOM table. You can also use **Ctrl+F3**.
	Find next	Selects the next match in the BOM table. You can also use **F3**.
	Select all found	Selects all matches in the BOM table.

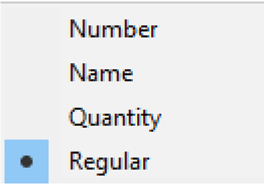
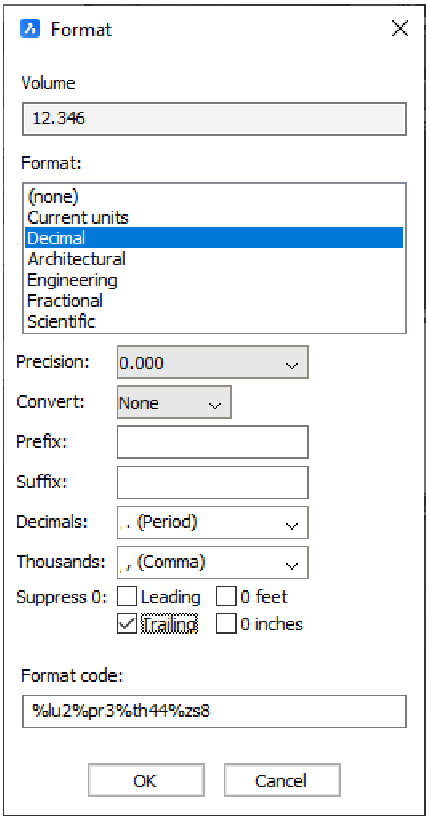
Column context menu

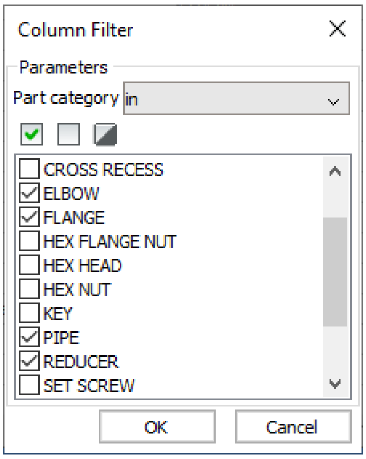
Right-click a column label to open the context menu. The column is added to the current selection.



Icon	Tool	Description
	Add template column from selected	Opens a dialog to configure a new template column. The selected columns will be automatically added to the template expression.
	Delete column	Removes the selected column(s) from the table. You can also select a column in the **BOM grid** and press the **Delete** key.

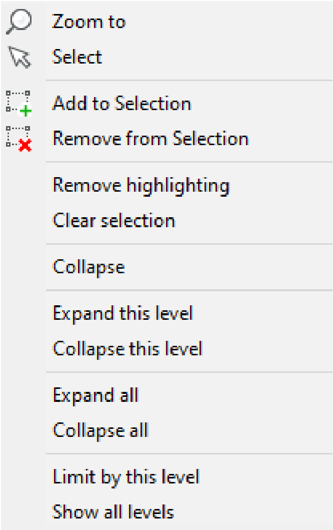
Icon	Tool	Description
	**Footer**	Opens a submenu for the selected column(s): • **No footer**: removes the footer. • **Sum**: displays the sum in the footer. • **Average value**: displays the average values in the footer. • **Minimal value**: displays the minimum value in the footer. • **Maximal value**: displays the maximum value in the footer.
	**Sorting**	Opens a submenu: • **Automatic order**: sorts the table automatically. • **Natural order**: sorts the table by the natural order of component instances in the document. • **Ascending order**: sorts the selected columns by ascending order. • **Descending order**: sorts the selected columns by descending order.
	Aggregate function	Opens a submenu for the selected column(s):  • **No function**: removes the aggregate function. • **Sum**: displays the sum of all values in the group. • **Average value**: displays the average of all values in the group. • **Minimal value**: displays the minimum of all values in the group. • **Maximal value**: displays the maximum of all values in the group. • **Concatenation**: displays a concatenation of all values in the group in ascending order separated by a specified delimiter ('; ' by default). • **Concatenation with count**: displays a concatenation of all values in the group with their corresponding number of encounters in ascending order. Value and count pairs are separated by a specified delimiter ('; ' by default).

Icon	Tool	Description
	Column role	Opens a submenu for the selected column:  • **Number**: the column is used as a source of numbers by the BMBALLOON command. • **Name**: the column is used as a source of names by the BMBALLOON command. • **Quantity**: the column is used as a source of quantities by the BMBALLOON command. • **Regular**: the column has no specific role.
%..	**Format**	Sets the specified format for selected column(s). Opens a dialog box: 

Icon	Tool	Description
	Set column filter	Opens a dialog box to configure the column filter: 
	Make column visible	Displays the selected columns in the associated table in the drawing.
	Make column invisible	Removes the selected column from the associated table. Invisible columns remain visible in the BOM manager but are marked with a diagonal hatch pattern in the background.
	Fit column(s)	Fits the width of the selected column(s) to their content. The column width automatically fits the content unless the width has been changed manually.
	Fit all columns	Fits the width of all columns to their content. The column width automatically fits the content unless the width has been changed manually.

Row context menu

Right-click any cell to open the context menu.



Icon	Tool	Description
	Zoom to	Zooms to the select part(s) in the model.
	Select	Selects the part(s). Replace an existing selection.
	Add to Selection	Adds part(s) to the selection.
	Remove from Selection	Removes the parts from the selection.
	Remove highlighting	Removes highlighting.
	Clear selection	Clears a selection.
	Collapse	Collapses the subassembly group in the BOM grid.
	Expand	Expands the subassembly group in the BOM grid.
	Expand this level	Expands all subassembly groups at the same level of the assembly tree as the current one.
	Collapse this level	Collapses all subassembly groups at the same level of the assembly tree as the current one.
	Expand all	Expands all subassembly groups in the current BOM.
	Collapse all	Collapses all subassembly groups in the current BOM.

Icon	Tool	Description
	Limit by this level	Sets the current level as the maximal level for hierarchical BOM tables.
	Show all levels	Removes the maximal level limit for hierarchical BOM tables.

BOM Title

Bill of Materials Vessel_Assembly

Displays the title of the current BOM, as displayed in the drawing. Click this control to edit the drawing. In this mode, the **BOM title** control shows the **title template** instead. The **title template** is a string that may contain a *<NAME>* placeholder, which will be replaced by the assembly name.

Grid

No.	Component ▲2	Physical material	Material density	Is standard	Quantity ▼1
1	ISO 4033			Yes	60
2	MM-THD			Yes	30
3	LEG			No	4
4	FLANGE			No	2
5	CONNECTION H	ASTM-A276-316L	7900.00	No	1
6	CONNECTION J	ASTM-A276-316L	7900.00	No	1
7	GASKET			No	1
8	HAND HOLE			No	1
9	HIGHT SPEED MIXER			No	1
10	HOMOGINIZER			No	1
11	MIXER_A			No	1
12	NOZZLE B			No	1
13	NOZZLE D			No	1
14	NOZZLE F	AISI-316L	7900.00	No	1
15	NOZZLE L			No	1

The grid displays the content of the selected BOM table. Each row is linked with its corresponding instances. When a row is selected, the corresponding parts are highlighted in the drawing. Use the context menu, to select corresponding parts, add or remove them from the current selection, and to zoom to those parts. Double-click any cell in the row to zoom to the corresponding parts.

Note: To remove the title or header row, the BOM title or all column headers must be empty. Also, titles entered from a command line will be trimmed to remove all leading or trailing spaces.

Moving a column

To move a column in the BOM table drag and drop it to its new position.

Visible and invisible columns

No.	Component	Physical material	Material density, kg/m³	Volume, cm³	Mass, kg	Quantity
1	CONNECTION H	ASTM-A276-316L	7900.00	80.28	0.63	1
1.1	ACTUATOR VALVE	ASTM-A276-316L	7900.00	32.76	0.26	1
1.2	TRANSITION PIECE	ASTM-A276-316L	7900.00	47.52	0.38	1
2	CONNECTION J	ASTM-A276-316L	7900.00	80.28	0.63	1
2.1	ACTUATOR VALVE	ASTM-A276-316L	7900.00	32.76	0.26	1
2.2	TRANSITION PIECE	ASTM-A276-316L	7900.00	47.52	0.38	1
3	FLANGE			2416.26		2
4	GASKET			453.05		1
5	HAND HOLE			4523.51		1
6	HIGHT SPEED MIXER			9762.24		1
7	HOMOGINIZER			10235.08		1
8	ISO 4033			6.03		60
9	LEG			1646.02		4
9.1	EURO-CHAN 24-62			1106.52		1
10	MIXER_A			13592.99	34.59	1
10.1	ANCHOR STIRRER			3949.21	30.66	1
10.1.1	AGITATOR SHAFT	ASTM-A276-316L	7900.00	1507.30	11.91	1
10.1.2	BUSH	ASTM-A276-316L	7900.00	149.22	1.18	1
10.1.3	BUSH	ASTM-A276-316L	7900.00	109.95	0.87	2
10.1.4	CLOSURE PLATE	ASTM-A276-316L	7900.00	8.48	0.07	2
10.1.5	FLAT BAR	ASTM-A240-316L	7900.00	45.09	0.36	2
10.1.6	FLAT BAR	ASTM-A240-316L	7900.00	139.65	1.10	2
10.1.7	FLAT BAR	ASTM-A240-316L	7900.00	34.22	0.27	4
10.1.8	PIPE	ASTM-A312-TP-316	7900.00	415.23	3.28	2

You can make any column in the BOM table invisible, so it does not show in the resulting table. Invisible columns remain visible in the BOM grid with a hatched background. You can change the visibility of columns in the main menu, the column context menu, or by editing the **Visible** property of a column in the **Properties grid**.

Width adjustment

You can adjust the width of any column in the BOM grid, which is independent from the width in the associated table. If the width of the column is less than needed to display the text of the cell, the height of the corresponding row increases to fit the text.

When the column width is adjusted manually, the width remains the same regardless of the content of the table. The width is also stored in the BOM configuration. All adjusted column widths remain the same even when the drawing is closed and reopened.

To fit the column width to the content automatically, do one of the following:

- Select the column(s), right-click the context menu and select **Fit column(s)**.
- Double-click the separator in the column header.

Sorting a BOM

Use the grid to control how the table of content should be sorted. Double-click a column label to change the order between ascending, descending, and unordered. If a column is used to sort the table, its label has a symbol added (▲ for ascending and ▼ for descending). Double-click the label to revert the sorting order.

You can use multiple columns to sort. The sort symbols of these columns have a subscript to denote the sort order. For example, if you sort by quantity in descending order and then by component name in ascending order, the Quantity column has a ▼? symbol and the Component column has a ▲? symbol.

Expand/collapse groups

No.	Component	Physical material	Material density, kg/m³	Volume, cm³	Mass, kg	Quantity
⊞ 9	LEG			1646.02		4
⊟ 10	MIXER_A			13592.99	34.59	1
⊞ 10.1	ANCHOR STIRRER			3949.21	30.66	1
10.2	BOLT	DIN 931 - A2	7900.00	6.15	0.05	1
10.3	BOLT	DIN 931 - A2	7900.00	1.57	0.01	4
10.4	GEARBOX & MOTOR			9130.51		1
10.5	MECHANICAL SEAL	Rubber	930.00	17.73	0.02	1
⊟ 10.6	MOUNTING MIX_A			310.97	2.46	1
10.6.1	AGITATOR MOUNTING FLANGE	ASTM-A240-304	7900.00	79.91	0.63	1
10.6.2	AGITATOR MOUNTING PAD	ASTM-A240-316L	7900.00	159.44	1.26	1
10.6.3	ROUND BAR	ASTM-A479-304	7900.00	17.91	0.14	4
10.7	STD. BOTTOM BEARING	AISI-316L	7900.00	159.87	1.26	1
10.8	WASHER	ASTM-A240-304	7900.00	12.27	0.10	1

In hierarchical BOMs, each subassembly forms a group that can be collapsed or expanded in the BOM grid.
To collapse or expand a group, do one of the following:

- Click the node icon in the first column of the BOM grid.
- Right-click the row, then select **Collapse** or **Expand** from the context menu.

You can also collapse or expand all groups or all groups corresponding to the same assembly level using the context menu.

Note: The subassembly group status (collapsed or expanded) is not stored within the document. When the BOM table is closed and reopened, all subassembly groups are expanded by default.

29.12.4 Properties

Bill of materials properties	
Title	Bill of Materials <NAME>
Type	Bottom level
Property set	All except coordinates
Selection set	Entire model
Target component	<Root component>
Filter settings	<Expand to configure>
Filter expression	... x
Mechanical objects	Include
Other local objects	Exclude
Other external references	Exclude
Solid plies	Exclude
Ignore BOM status	No
Sorting mode	Automatic order
Grouping mode	Auto
Footer title	TOTAL:
CSV delimiter	; (Semicolon)
Warning messages	On
Column properties	
Title	Volume
Category	Mass
Role	Regular column
Format	%lu2%pr2
Visible	Yes
Width	Auto
Filter	No filter
Aggregate function	Sum
Unit placement	In additional column
Units mode	Best for all values
Footer type	Average value
Footer format	%lu2%pr2
Footer units mode	Auto

The properties grid is located at the bottom of the manager's panel. To adjust its size, use the splitter control, located between the grid and the properties grid.

The properties grid contains the properties of the current table; title, type, filter, sorting mode and footer title. If any columns are selected, the grid will also display their common properties. You can change any common property for all the selected columns at once. Column properties change the title, visibility, and width of the column.

For numerical values, you can set the footer type, footer units mode, unit placement, units mode, and the footer unit (if the column has units that can be changed).

The **unit placement** defines where the units are displayed: in the column title, in the same cell as the value itself, or in its own column.

The **units mode** defines how the displayed unit should be obtained; the best unit for all values, the best unit to display a particular value in the cell, or a specified (user-defined) unit. You can choose the **Units mode** for distance, area, volume, or mass properties.

The **footer type** defines what should be displayed in the cell, sum, average value, minimal value or maximal value.

The **footer units mode** defines the units for the footer cell, auto, best or user-defined.

The **footer unit** sets a different unit for footer value.

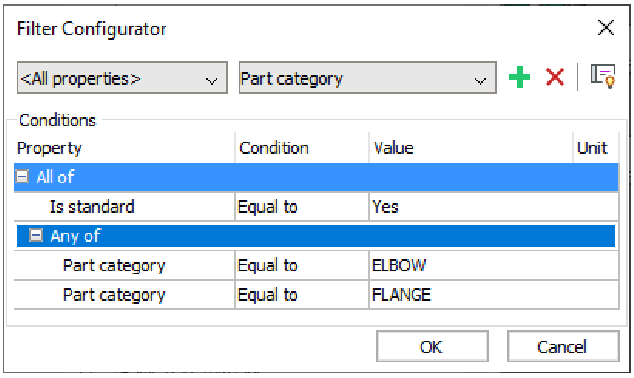
29.12.5 Filter settings

Filter expression

It is possible to add a custom filter to the current BOM. To apply a filter, do one of the following:

- Use the **Filter expression** property in the **Bill of materials properties** section of the **Property grid**.
- Use the **Set filter** command from the **Main menu**.

In both cases, the *Filter Configurator / expression* dialog box is displayed:



Each row in this dialog box represents an elementary condition. It is possible also to add groups to combine several elementary conditions and/or groups. Groups allow to create complex filters.

Adding a new elementary condition

Do the following:

1. If applicable, choose the required category to limit the list of properties to that particular category.
2. Select the required property from the list of properties. When selected, a condition for the property is added automatically.
3. If applicable, the required condition type:
   - Not equal to
   - Less than
   - Greater than
   - Not less than
   - Not greater than
4. Set the target value. For text values, you can select the target value from the drop-down list of all possible values. For numerical values, you can specify the unit of the target value. The value will be automatically converted to the document unit.

Removing one or more conditions

Do the following:

1. Select the conditions to remove. You can use the Ctrl key to select several conditions, and Shift key to select a range of conditions.
2. Click the **Delete** tool (×), or right-click the row you want to delete and select **Remove condition** in the context menu.

Adding a new group of conditions

Do the following:

1. Click the **Add** tool (+).
2. Select **Add "All of" group** to add a new "All of" group, or select **Add "Any of" group** to add a new "Any of" group.
3. Add the necessary conditions.

Grouping selected conditions

Do the following:

1. Select the conditions to remove. You may use the Ctrl key to select several conditions, and Shift key to select a range of conditions.
2. Right-click the row you want to group.
3. Select **Form "All of" group from selected** to form a new "All of" group, or select **Form "Any of" group from selected** to form a new "Any of" group.

To apply the filter, click **OK**.

Mechanical objects

Includes or excludes the mechanical components.

Other local objects

Includes or excludes the non-mechanical local blocks and solids.

Other external objects

Includes or excludes the non-mechanical external references.

Solid plies

Includes or excludes the solid plies.

Ignore BOM status

Decides if the BOM status will be ignored.

29.12.6 Column filters

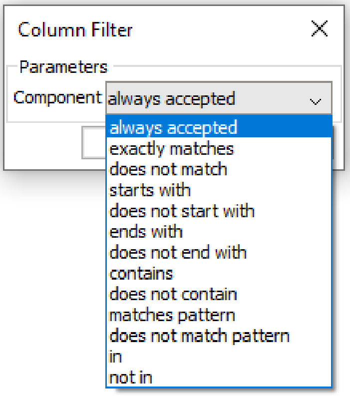
Column filters allow you to set the conditions on which rows to include in the table. Each column filter sets a condition that column values should meet in order to be included. For example, you may set a column filter to include only parts with quantities larger than a certain number, or parts whose names contain **Nozzle**. A column can have no more than one filter. However, it is possible to simultaneously set filters for as many columns as you need. Columns with active filters are marked with a filter icon ( ) in their title.

Example of columns with active filters:

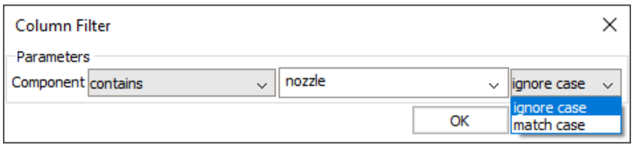
No.	Part category	Component	Quantity
1	FLANGE	ASME B16.5 Flange BL RF	9
2	FLANGE	ASME B16.5 Flange SO RF	177
3	FLANGE	ASME B16.5 Flange SO RF	4

To add a column filter, right-click the column title and select **Set column filter** in the context menu.

A *Column filter* dialog box is displayed.

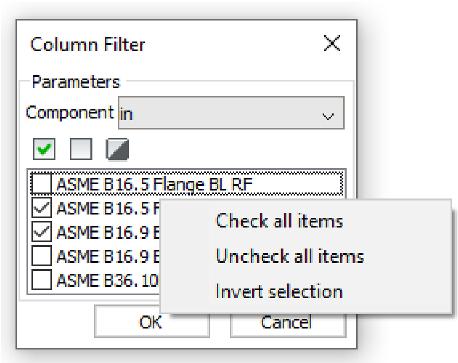


Select the required filter type. Next, you will need to set the target value(s) for the filter, depending on the filter type and the column content.



For columns with numerical content, you can set the unit for the target value. The target value will be automatically converted to the document unit internally.

For text filters, you can choose between **ignore case** and **match case** options. **In** and **Not in** filters show you the list of all possible values for this column (with respect to already applied filters). Check the values that you want to be included or excluded correspondingly. You can also check all values, uncheck all values, or invert the current selection by using the corresponding buttons on toolbars or commands from the context menu.



To apply the column filter, click **OK**.

To remove a column filter, right-click the column title and select **Remove column filter** from the context menu.

For more details, see the **Filter a BOM table** article.

29.12.7 Selection set

It is possible to link a BOM table with a selection set, which will act like a filter for BOM content. Following types of selection sets are supported:

- **Entire model**: the BOM table will be created for the entire model.
- **Subassembly**: the BOM table will be created for a selected instance of a subassembly.
- **Drawing view**: the BOM table will be created for the parts selected for the corresponding drawing view, regardless of their visibility in the viewport.
- **Layout viewport**: the BOM table will be created for the selected layout viewport.
- **Custom selection**: the BOM table will be created only for the selected parts.
- **Exclude current document**: the BOM table will not include objects from the currents document.
- **Show selection**: shows the current selection for the BOM table.
- **Additional documents**: the BOM table will include parts from other documents.
    - **Add drawing or sheet set**: the BOM table will include objects from the selected drawing or sheet set.
    - **Add folder**: the BOM table will include objects from drawings or sheets set which are present in a folder.

      Note: The user may include only the documents that match specified wildcards, entire BIM project or entire sheet set.
- **Edit**: Allows you to edit the selection.

It is possible to see the current selection set using **Show selection** command from the main menu.

You can change type of the selection set at any moment by simply choosing a desired selection set type in the main menu. In addition, you may edit existing selection set by adding or removing objects from that selection set.

29.12.8 Sort

BOM tables support several sort modes (for more details, see the **Sort a BOM table** article).

To switch between these modes, use the **Sorting mode** property in the **Bill of materials properties** section of the **Properties grid**. Use the **Sorting mode** drop-down menu or double-click the column labels in the **Grid** to toggle between ascending, descending or natural order for the respective column.

29.12.9 Column units

It is possible to change the **Unit placement** and **Units mode** for most numerical properties with units. Select the required columns in the **Grid** and use the **Column properties** section of the **Properties grid** to change the **Unit placement** and the **Units mode**.

For more details, see the **Column units** article.

See also the **Changing the units and unit placement modes for a column** section of this article.

29.12.10 Evaluated columns

A BOM supports evaluated columns. There are 2 types of evaluated columns: formula and template columns. Thumbnail columns are also considered evaluated columns, although they don't require any expressions. For more details about formula fields, such as expression syntax, see the **Evaluated columns** article.

See also the **Adding an evaluated column** section of this article.

Formula columns

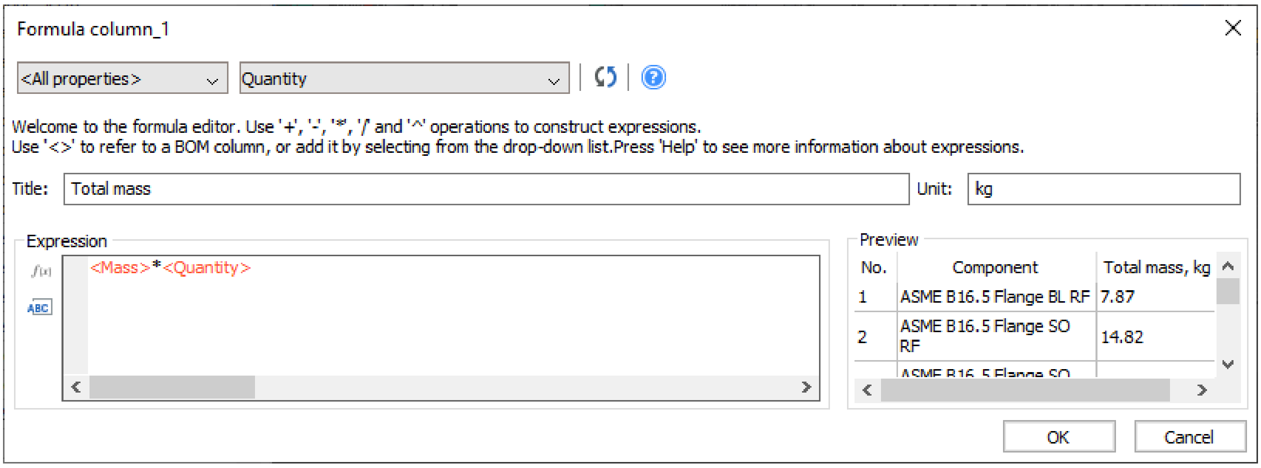
To add a formula column, do the following:

1 Select the **Evaluated column** category using the **Category selector**.

2 Select **Formula column** in the **Column selector**.

3 Use **Insert column after** or **Insert column before** tools.

As an alternative, you can use the **Insert formula column** option from the **Main menu**.
In both cases, an *Expression configurator* dialog box is displayed:



This dialog box allows you to configure the title, unit and expression for the formula column. It also displays a preview of the formula results. The expression field supports autocompletion.
To add a reference to a column or a property to the expression at the cursor position, select the required property from the list of properties. You can also click the **Add column reference** tool ( ) and select the property from the menu, or click the **Add function** tool ( ) to select the required function.
To add a formula column to the BOM table, click **OK**.

Template columns

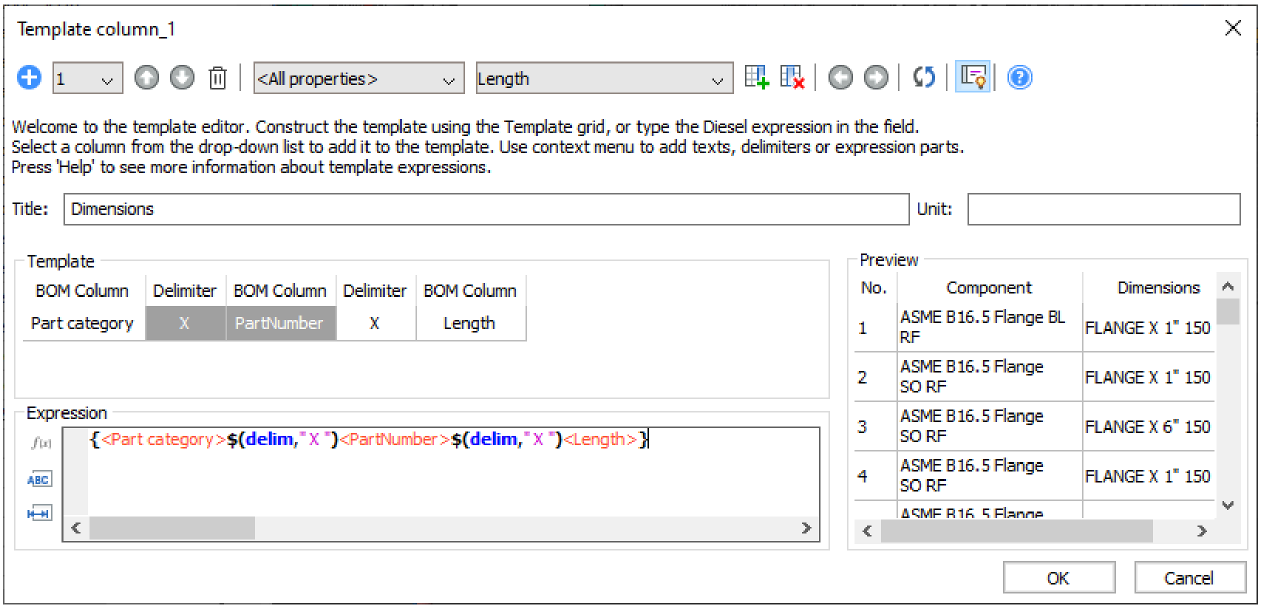
To add a template column, do the following:

1 Select <*Evaluated column*> category using the **Category selector**.

2 Select <*Template column*> in the **Column selector**.

3 Use **Insert column after** or **Insert column before** tools.

As an alternative, you can use the **Insert template column** option from the **Main menu**.
In both cases, a *Template configurator* dialog box is displayed.



This dialog box allows you to configure title, expression and, if necessary, unit for your template column. It also displays a preview of the results. The expression field supports autocompletion.

Template columns may contain more than one template subexpression. In that case, they are evaluated from first to last, and the first subexpression yielding in non-empty result is used. The **Template configurator** allows you to add, delete and switch between template subexpressions using the toolbar. It also allows to change the order of template subexpressions.

Note: In the expression field, all subexpressions are shown at once, where each subexpression is enclosed in {} brackets.

To add a subexpression, click .

To delete the current subexpression, click .

Note: When the last subexpression is removed, it will be replaced with an empty one.

To change the current subexpression, select the number of the required subexpression from the drop-down list.

Use and to change the order of the subexpressions.

The currently selected subexpression can be configured using the **Template** section. Each column in this section represents an item of the template expression, which you can change.

The types of template items are:

- **Column reference**: represents a reference to a BOM column or a property. To add a column reference, select the required column from the list of properties. It will be automatically added after the last selected column, or, if no column is selected, after the last column. If necessary, a **Delimiter** will be added in front of the **Column reference** automatically.
- **Delimiter**: represents a separator between 2 items. The delimiter text will appear in the final result only if the text on both sides of it is not empty. For example, if you have PartNumber and Length column references with the "X" delimiter between them, the "X" will be added to the final text only if both

PartNumber and Length are not empty. In most cases, delimiters are added automatically. However, you can add delimiters by using the button and selecting **Add delimiter** from the drop-down list. Alternatively, you can add a delimiter from the context menu.

- **Text**: represents a custom text. You can add custom text by clicking and selecting **Add text** from the drop-down list. Alternatively, you can add a custom text from the context menu.
- **Evaluated text**: represents an item that must be added to the final expression as is. Evaluated text may contain functions or other special symbols that will modify the result. You can add an evaluated text by using the button and selecting **Add expression** from the drop-down list. Alternatively, you can add an evaluated text from the context menu.

At any moment, you can change the content of any item in the template. You can also delete items by selecting them and using the button, or by choosing **Delete column** in the context menu. To move an item, select a column in the template and then drag and drop it at the desired position. Any modifications of the template will be reflected in the expression field. Similarly, any changes in the expression field will be reflected in the template area.
To add a template column to the BOM table, click **OK**.

Thumbnails column
To add a thumbnails column, do one of the following:

- Select *"<Evaluated column>"* category using the **Category selector**.
- Select *"<Thumbnails column>"* in the **Column selector**.
- Use **Insert column after** or **Insert column before**tools.

or

- Use the **Insert thumbnails column** option from the **Main menu**.

In both cases, a thumbnails column will be inserted into the table.
After insertion, it is possible to adjust parameters of thumbnails using column properties, such as size, background color, view type and visual style.

29.12.11 Footer row

A BOM table can include a footer row. To add a footer row, specify the footer type for one or more columns. By default, the title of the footer row is *TOTAL:*. Change the footer title with the **Footer title** property in the **Bill of materials property** section of the **Properties grid**.

It is possible to add footers to formula or template columns as well.

For more information about a footer row in a BOM, see the **Footer** article.

See also the **Adding a footer to a column** section of this article.

29.12.12 Aggregated columns

In top-level, bottom-level, and hierarchical BOM tables, almost any column can be assigned with an aggregate function. Aggregate functions group values of several rows in a BOM (that differ only in those values) into a single value. For example, aggregate functions can be used to compute the total length of pipes with respect to their size and thickness, or to concatenate all sizes of bolts and nuts used in the assembly in a single cell.

For more details, see the **Column units** article.

To set an aggregate function, do one of the following:

- Right-click the column title and select the required aggregate function from the **Aggregate function** submenu.
- Select the column and change the aggregate function in the corresponding column property.

Aggregated columns are marked with the ∑ symbol in their title.

Example of aggregated columns:

No.	Part ▲	Thickness, mm	∑Length, m
1	PIPE D= 1"	1.65	107.88
2	PIPE D= 2"	1.65	60.61
3	PIPE D= 6"	2.11	1.74
4	PIPE D= 6"	2.77	7.14

29.12.13 Templates

You can save BOM configurations as templates. Use the BOM manager to create a new table from a template and to replace the current table configuration with one from a template. You can save the configuration as a template at any moment. You can change a loaded template configuration with the regular tools and save it as another template.

For more information about BOM templates, see the **Templates** article.

29.12.14 Searching

The BOM manager has a search field that allows you to search for a particular value in the table. You can also use wild cards in the search field. By default, BricsCAD searches the entire table. To limit the search area, select the required columns. If the value is found, the cursor moves to that cell. If no columns are selected, the row is selected. The **Grid** automatically scrolls to make the cell visible. To find another value, use the or buttons. You can also use **F3** or **Ctrl+F3**. To select all found results, use the button.

By default, the text in the search field is searched for. However, you can also use **wild cards**.

The following special symbols are supported:

Name	Character	Description
Back quote	`	Escapes the next character (takes it literally).
Comma	,	Separates patterns.
Asterisk	*	Matches any character sequence, including a null sequence, anywhere.
Question mark	?	Matches a single character.

Name	Character	Description
Period	.	Matches a single non-alphanumeric character.
At sign	@	Matches a single alphabetic character.
Pound sign	#	Matches a single numeric character.
Tilde	~	If the first character in a pattern matches anything but the pattern.
Brackets	[...]	Matches a single enclosed character.
Tilde brackets	[~...]	Matches a single non-enclosed character.
Hyphen	-	In brackets, matches a single character in the specified range.

To open the search field, use the **Search** tool on the toolbar or use **Ctrl+F**.

29.12.15 Associated table

Each BOM table in the manager can have an associated table in the document, which updates when the BOM configuration is changed. A BOM table can have one associated table only. However, you can create BOM tables that do not have an associated table in the document.

To create an associated table, use **Place/Update table**. If the BOM table was created with *BMBOM*, the associated table is created automatically.

You can locate the associated table in the document with **Associated table - Zoom to** or **Associated table - Select**. If necessary, the active layout switches to the associated table layout.

You can delete the associated table with **Associated table - Delete**. As an alternative, you can detach the associated table from the BOM table with **Associated table - Detach**. After it is detached, the associated table loses its link to the BOM table and will not update with the BOM.

29.12.16 Updating the BOM

From BricsCAD V20.2, the BOM manager automatically updates the table content in the panel, when the assembly is modified or updated. For large assemblies, this may be inconvenient. The BOM manager automatically updates the table when active.

To avoid the automatic update, close the BOM manager to make the **BOM Manager** panel inactive.

You can manually refresh the BOM manager content using **Refresh**.

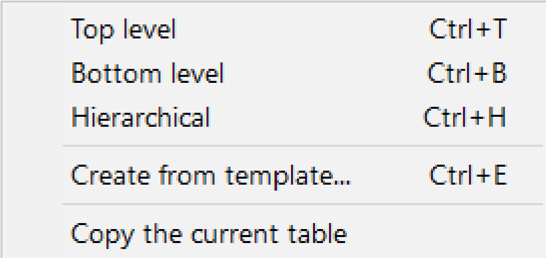
Note: The associated table is not updated automatically when the assembly is changed or modified. Use **Place/Update table** to update the associated table.

29.12.17 Adding a new bill of materials

1 Use the **New BOM** submenu of the **Main menu**.

Note: If BOMTEMPLATE system variable contains a valid path to a .bom template file, this template will be used to create new BOM tables. If it is empty or the path is not valid, the below default BOM configuration will be used.

1



2 Select the BOM type or select **From template** to choose the template file for a BOM.

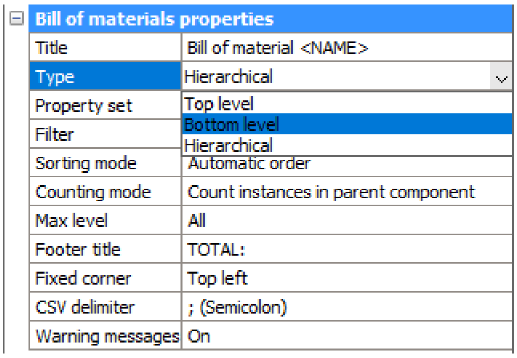
29.12.18 Selecting the current BOM

Use the **BOM selector** to select an existing bill of materials.

Note: The table will automatically update to the current state of the assembly and the associated table in the drawing (if it exists).

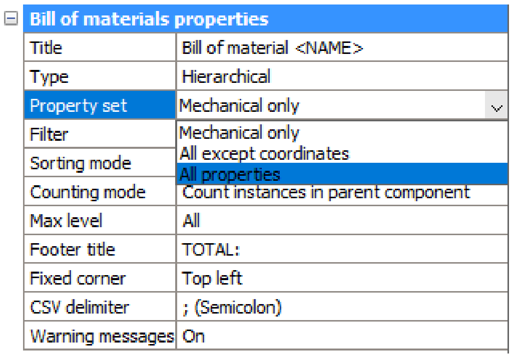
29.12.19 Changing the type of the current BOM

Change the **Type** property in the **Bill of materials properties** section of the **Properties grid**.



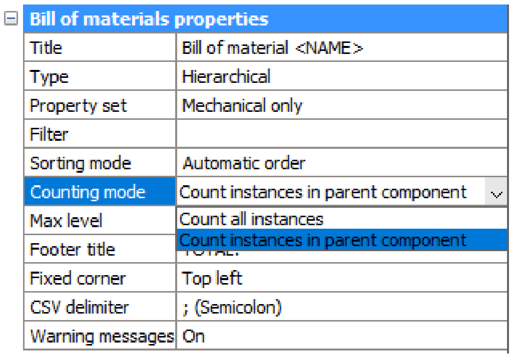
29.12.20 Changing the property set of the current BOM

Change the **Property set** property in the **Bill of materials properties** section of the **Properties grid**.



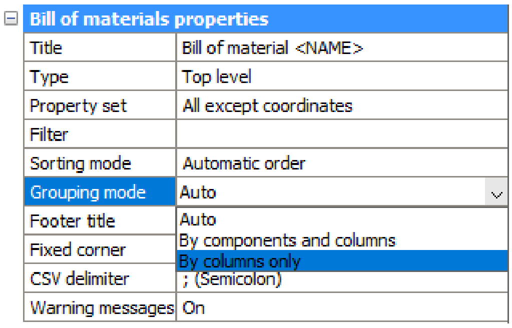
29.12.21 Changing the counting mode of a hierarchical BOM

Change the **Counting mode** property in the **Bill of materials properties** section of the **Properties grid**.



29.12.22 Changing the grouping mode of top- or bottom-level, and hierarchical BOM

Change the **Grouping mode** property in the **Bill of materials properties** section of the **Properties grid**.



29.12.23 Setting a custom filter for the current BOM

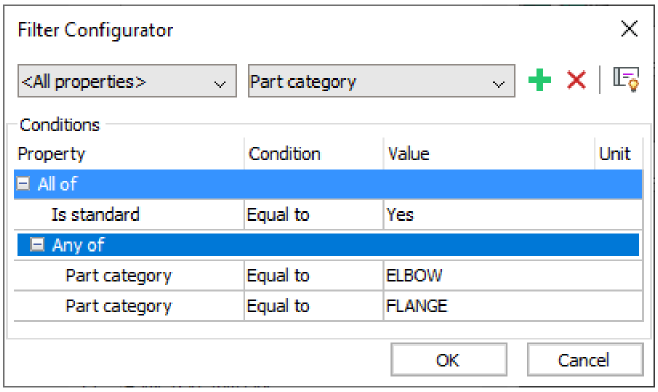
1 Select the **Set filter** option from the **Main menu**.

A *Filter Configurator* dialog box is displayed.

2 Configure the filter using the dialog tool.

See **Filter** for details.

3 Click **OK** to apply the filter.



29.12.24 Adding new columns to the current BOM

1 If applicable, use the **Category** selector to select the category of the properties you want to add.

2 Select the property to add from the list of the available columns. This list contains all columns from the selected category which are not yet added to the BOM. If you start to type the name of the field, BricsCAD automatically suggests names to you. Click to select it from the list.

3 (Optional) Select the column in the BOM you want to place the new column next to.

4 Click **Insert the column after** () to place the new column after the selected one, or **Insert the column before** () to place the new column before the selected one.

If no column is selected, the new column is placed as the last or the first position respectively.

29.12.25 Adding an evaluated column

1 Select the **Evaluated columns** category using the **Category selector**.

2 Select **Formula column** to create a formula column, or select **Template column** to create a template column.

3 Click **Insert the column after** () to place the new column after the selected one, or **Insert the column before** () to place the new column before the selected one.

If no column is selected, the new column is placed as the last or the first position respectively.

A dialog box to configure the evaluated column is displayed.

4 Change the **Title** of the column.

5 Change the **Unit** if necessary.

6 Configure the expression using the dialog tools.

7 Click **OK** to add the column to the BOM table.

29.12.26 Adding a template column from selected columns

1 Select the columns to be combined in a template column. Use the Ctrl key to select multiple columns or the Shift key to select a range of columns.

2 Right-click the title of one of the selected columns.

3 Select **Add template column from selected** from the context menu.

A dialog box to configure the evaluated column is displayed.

4 Change the **Title** of the column.

5 Change the **Unit** if necessary.

6 Configure the expression using the dialog tools.

7 Click **OK** to add the column to the BOM table.

8 If necessary, remove or hide columns that are no longer necessary.

29.12.27 Changing an evaluated column

1 Right-click the column title in the **BOM grid**.

2 Select **Edit column expression** from the context menu.

A dialog box to configure the evaluated column is displayed.

3 Modify the column title, unit and expression as necessary.

4 Click **OK** to apply the changes.

29.12.28 Removing a column from the current BOM

1 Select the column in the grid.

2 Click the **Remove the column** tool ( ), or press **Delete**.

29.12.29 Changing the BOM title

Use **BOM Title** to edit the title, or use the **Title** property in the **Bill of materials properties** section of the **Properties grid**.

You can use the *<NAME>* placeholder to refer to the assembly name.

Note: To remove the title or header row, the BOM title or all column headers must be empty. Also, titles entered from a command line will be trimmed to remove all leading or trailing spaces.

29.12.30 Changing the footer row title

Use the **Footer title** property in the **Bill of materials properties** section of the **Properties grid**.

29.12.31 Adding a prefix or suffix to numbers

1 Select the **Number** column.

2 Select **Number prefix** property in the **Column properties** section.

Column properties	
Title	No.
Category	BOM Standard
Role	Number
Number prefix	A
Number suffix	
Number type	Full
Delimiter	.
Format	
Visible	Yes
Width	Auto
Filter	No filter

3 Enter the new prefix string if necessary.

4 Select **Number suffix** property in the **Column properties** section.

5 Enter the new suffix string if necessary.

29.12.32 Changing the delimiter between numbers in a hierarchical BOM

1 Select the **Number** column.

2 Select **Delimiter** property in the **Column properties** section.

Column properties	
Title	No.
Category	BOM Standard
Role	Number
Number prefix	A
Number suffix	
Number type	Full
Delimiter	.
Format	
Visible	Yes
Width	Auto
Filter	No filter

3 Enter the new delimiter string.

29.12.33 Changing the number type in hierarchical BOM

1 Select the **Number** column.

2 Select **Number type** property in the **Column properties** section.

Column properties	
Title	No.
Category	BOM Standard
Role	Number
Number prefix	A
Number suffix	
Number type	Full Full Short Continuous
Delimiter	
Format	
Visible	Yes
Width	Auto
Filter	No filter

3 Select the required number type.

29.12.34 Change the column title

1 Select the column in the **BOM grid**.

2 Select **Title** property in the **Column properties** section.

Column properties	
Title	Component
Category	BOM Standard
Role	Name
Format	
Visible	Yes
Width	Auto
Filter	No filter
Aggregate function	No function

3 Change the **Title** property value to the new title of the column.

Note: To remove the title or header row, the BOM title or all column headers must be empty. Also, titles entered from a command line will be trimmed to remove all leading or trailing spaces.

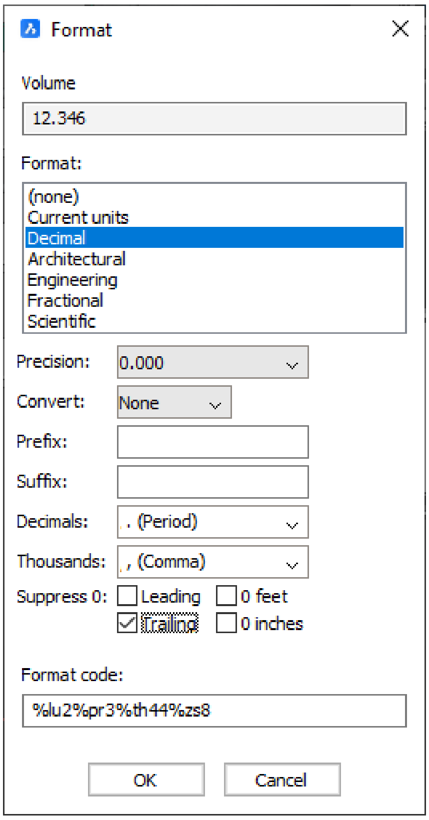
29.12.35 Change the column format

1 Select the column(s).

2 Right-click the title of a selected column.

3 Select **Format** in the context menu.

The *Format* dialog box displays.



4 Define the desired format.

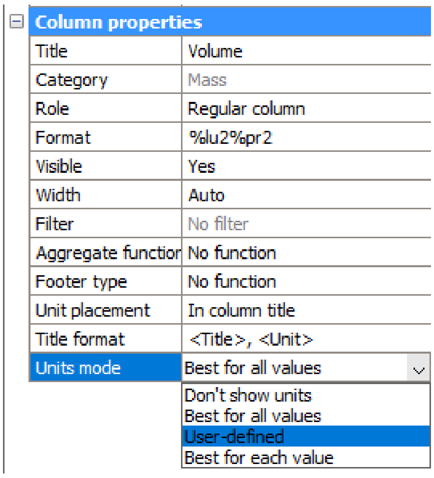
5 Click **OK** to apply the changes.

29.12.36 Changing the units and unit placement modes for a column

1 Select the column(s).

2 (Optional) Change the **Unit placement** property in the **Column properties** section.

3 (Optional) Change the **Units mode** property in the **Column properties** section.



4 (Optional) Change the **Title format**.

- Use *<Title>* to add the column title.
- Use *<Unit>* to add the column unit.
- Use *n* to add a line break.

29.12.37 Adding a footer to a column

1 Select the column(s).

2 Use the **Footer** submenu of the **Main menu** to select the type of footer value, or right-click the column label and then select **Footer** submenu.

The following types are available:

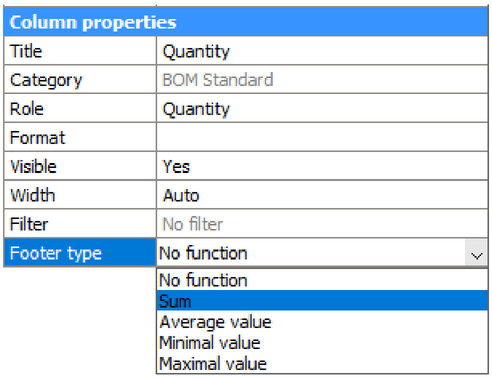
- **No footer** (the column will be removed from the footer. If there is no column with another footer type, the footer row will be removed completely)
- **Sum**
- **Average value**
- **Minimal value**
- **Maximal value**

Note: Footer columns are supported for template file.

Alternatively:

1 Select the column(s).

2 In the **Properties grid**, select the **Footer type** property and choose the footer type.



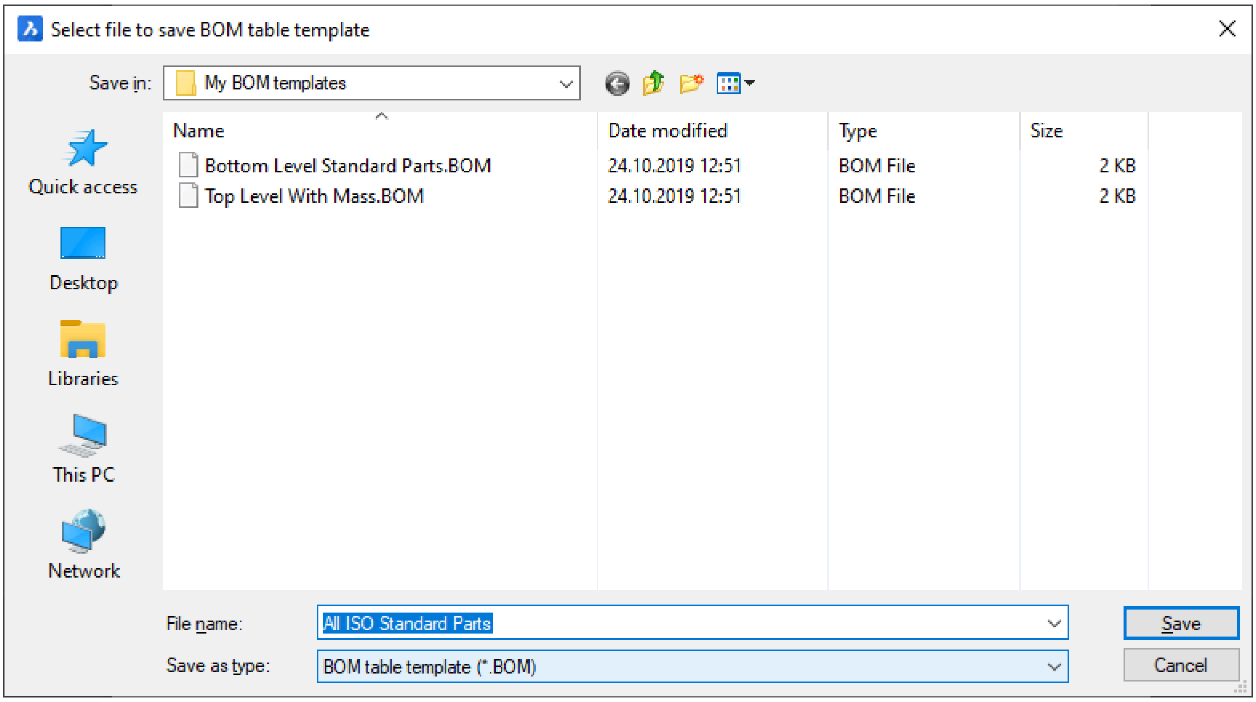
Note: To toggle between **Sum** and **No function**, click the **Footer** button ( ).

Note: It is possible to add footers to formula or template columns as well.

29.12.38 Saving the current configuration as a template

1 Choose **Save as template** ( ) in the main menu ( ).

The dialog box to select the name and location for a template file displays.



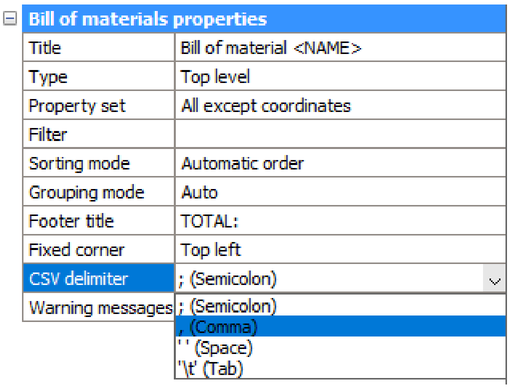
2 Specify the name and location of the template file.

3 Click **Save**.

Note: It is possible to add footers to formula or template columns as well.

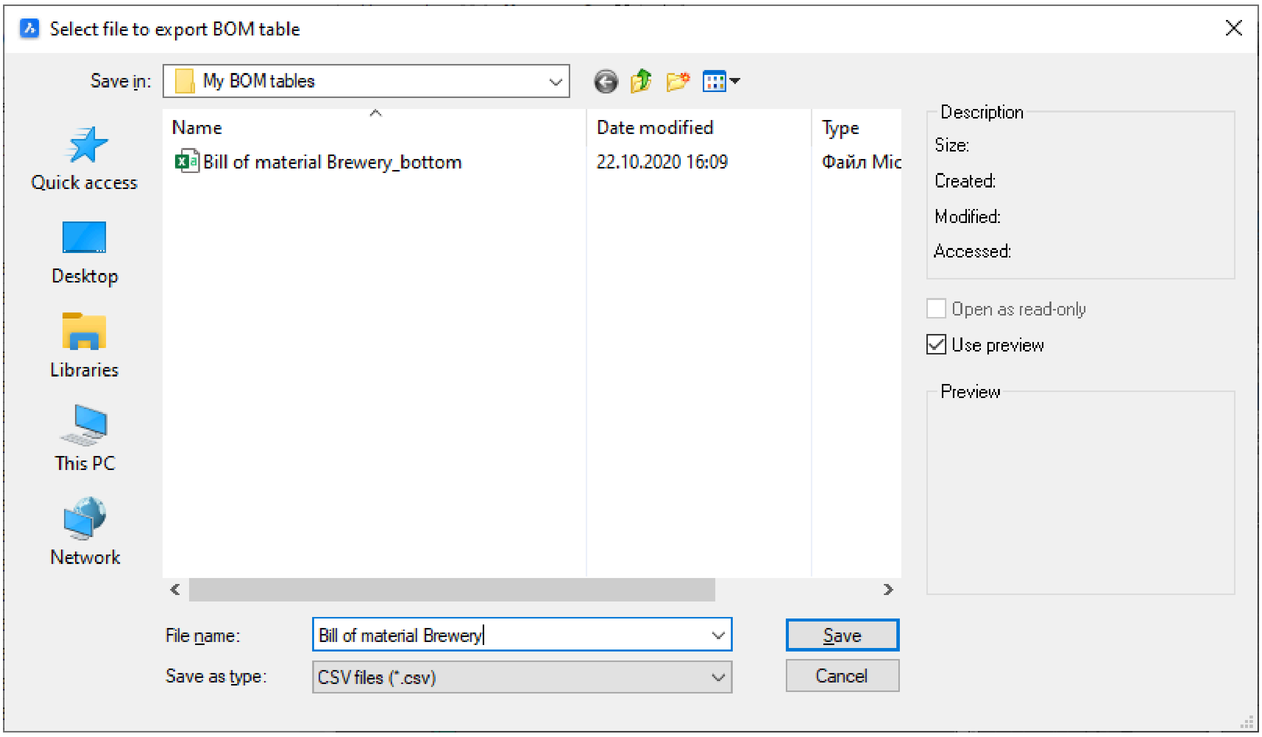
29.12.39 Exporting the current table to CSV

1 (Optional) Adjust the **CSV delimiter** property of the table.



2 Choose **Export to CSV** in the main menu (☰).

The dialog box to select the name and location for a CSV file displays.



3 Specify the name and location of the CSV file.

4 Click **Save**.

29.13 Bill of Materials

You can use the *BMBOM*, *BMBOMEDIT*, *BMBOMPANELOPEN* and *BMBOMPANELCLOSE* commands.

An engineering Bill of Materials (BOM) is a list of unique mechanical components, mechanical blocks and mechanical entities used in an assembly. In BricsCAD, the list is represented as a table entity and can be placed anywhere in the drawing. Each entity has its own serial number (starting with 1), name, and number of occurrences.

You can view and configure Bills of Materials in the **Bills of Materials Manager**.

29.13.1 Bill of Materials types

There are 3 BOM types in BricsCAD:

- **Top level**

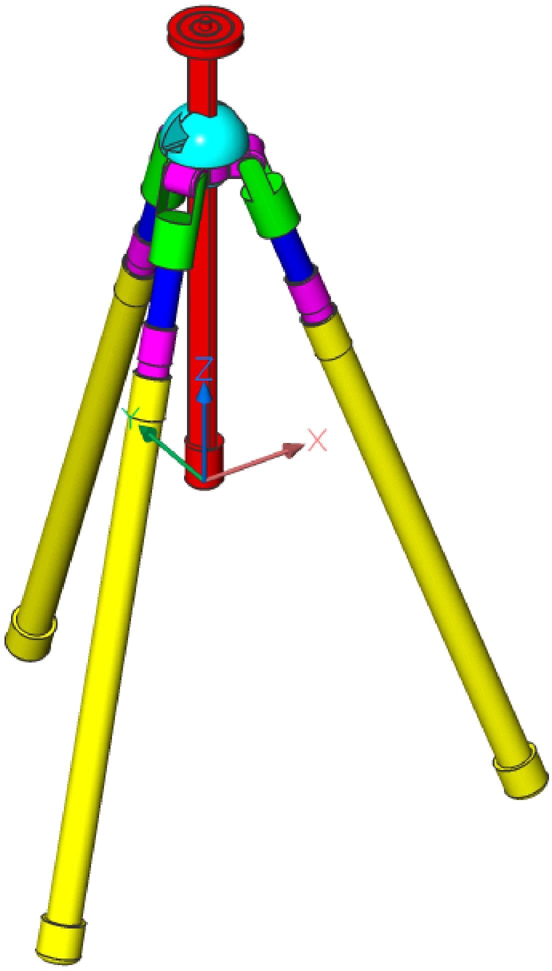
  Contains top-level components only. Top-level components are components inserted directly into the main assembly. This type of BOM lists the main components only, without details of subcomponents.
- **Bottom level**

  Contains bottom-level components (parts) only. Bottom-level components are parts that do not contain other parts or subassemblies. This type of BOM lists all parts in the main assembly, regardless of which component the parts belong to.
- **Hierarchical**

  Contains all components with their subcomponents listed directly below. The serial numbers of subcomponents is a sequence of numbers separated by dots (for example, "1.2.3"). This BOM type shows the hierarchical structure of the assembly.

Note: The **BOM status** property controls which parts are included in a Bill of Material. The level of a component in a BOM table is set with respect to the **BOM status** of all related components. For example, a component with a Terminal BOM status is always considered as bottom-level, even if it contains other parts or subassemblies.

Example: Tripod assembly



<table>
<tr><th>Type</th><th>Bill of Material</th></tr>
<tr><td>Top level</td><td>
<table>
<tr><th colspan="3">Bill of Materials tripod_main_assy</th></tr>
<tr><th>No.</th><th>Component</th><th>Quantity</th></tr>
<tr><td>1</td><td>tripod_axis_assy</td><td>1</td></tr>
<tr><td>2</td><td>tripod_foot_assy</td><td>3</td></tr>
</table>
</td></tr>
</table>

<table>
<tr><th>Type</th><th>Bill of Material</th></tr>
<tr><td>Bottom level</td><td>
<table>
<tr><th colspan="3">Bill of Materials tripod_main_assy</th></tr>
<tr><th>No.</th><th>Component</th><th>Quantity</th></tr>
<tr><td>1</td><td>lower_tube_prt</td><td>3</td></tr>
<tr><td>2</td><td>middle_tube_prt</td><td>3</td></tr>
<tr><td>3</td><td>tripod_axis_prt</td><td>1</td></tr>
<tr><td>4</td><td>tripod_bond_prt</td><td>3</td></tr>
<tr><td>5</td><td>tripod_crosspiece_prt</td><td>1</td></tr>
<tr><td>6</td><td>tripod_sphere_prt</td><td>1</td></tr>
<tr><td>7</td><td>upper_tube_prt</td><td>3</td></tr>
</table>
</td></tr>
<tr><td>Hierarchical</td><td>
<table>
<tr><th colspan="3">Bill of Materials tripod_main_assy</th></tr>
<tr><th>No.</th><th>Component</th><th>Quantity</th></tr>
<tr><td>1</td><td>tripod_axis_assy</td><td>1</td></tr>
<tr><td>1.1</td><td>tripod_axis_prt</td><td>1</td></tr>
<tr><td>1.2</td><td>tripod_crosspiece_prt</td><td>1</td></tr>
<tr><td>1.3</td><td>tripod_sphere_prt</td><td>1</td></tr>
<tr><td>2</td><td>tripod_foot_assy</td><td>3</td></tr>
<tr><td>2.1</td><td>bond_assy</td><td>3</td></tr>
<tr><td>2.1.1</td><td>tripod_bond_prt</td><td>3</td></tr>
<tr><td>2.2</td><td>tubes_assy</td><td>3</td></tr>
<tr><td>2.2.1</td><td>lower_tube_prt</td><td>3</td></tr>
<tr><td>2.2.2</td><td>middle_tube_prt</td><td>3</td></tr>
<tr><td>2.2.3</td><td>upper_tube_prt</td><td>3</td></tr>
</table>
</td></tr>
</table>

29.13.2 Counting mode and maximal level

For hierarchical BOM tables, 2 counting modes are supported:

- **Count all instances**

  The total number of subassemblies and parts in the entire assembly.

- **Count instances in parent component**

  The total number of subassemblies and parts in the parent component. The parent component is considered as the root component.

For example, the main Tripod assembly contains 3 Tripod foot assemblies. Each instance of a Tripod foot assembly contains 1 Bond assembly and 1 Tube assembly.

If the **counting mode** is set to **Count all instances**, the result is:

#	Component	Quantity
1	Tripod assembly	1
1.1	Tripod foot assembly	3
1.1.1	Bond assembly	3
1.1.2	Tubes assembly	3

If the **counting mode** is set to **Count instances in parent component**, the result is:

#	Component	Quantity
1	Tripod assembly	1
1.1	Tripod foot assembly	3
1.1.1	Bond assembly	1
1.1.2	Tubes assembly	1

The default value for the **counting mode** is **Count instances in parent component**. You can change the counting mode in the **BOM Manager**, or with the BMBOM command.

Note: Before BricsCAD V20.2, all hierarchical BOM tables used the **Count all instances** mode by default. When such tables are updated in BricsCAD V20.2 or later, the default mode is set to **Count instances in parent component**. This may result in different quantities for some parts. If necessary, manually set the counting mode in the **BOM Manager**.

By default, hierarchical bills of materials show all levels in the model. It is possible to limit a hierarchical BOM to a certain level using **Max level** property. When the maximal level is set, only parts up to that level will be included. For example, if **Max level** = 2, only top level components and subcomponents directly included in top level components will be listed.

29.13.3 BOM columns

By default, the BOM table contains the Number, Name and Quantity columns, Thumbnails, or the desired configuration, previously defined in a template .bom file stored in BOMTEMPLATE system variable.

Note: Thumbnails will be added automatically if the **Add thumbnails to new tables** option is checked.

Using the Configure option of the *BMBOM* command you can add more columns, such as Description, Density, Volume, Mass, Material, Thickness (for sheet metal parts), and Parameters (for parametric components).

You can add any built-in property of components, instances or block references to the BOM table. Custom evaluated columns are also supported, using the **DataExtraction** technology.

Bill of Materials tripod_main_assy					
No.	Component	Quantity	Density	Volume	Mass
1	tripod_axis_assy	1	8000.00 kg/m³	330.77 cm³	2.65 kg
1.1	tripod_axis_prt	1	8000.00 kg/m³	194.42 cm³	1.56 kg
1.2	tripod_crosspiece_prt	1	8000.00 kg/m³	60.06 cm³	0.48 kg
1.3	tripod_sphere_prt	1	8000.00 kg/m³	76.29 cm³	0.61 kg
2	tripod_foot_assy	3	8000.00 kg/m³	166.24 cm³	1.33 kg
2.1	bond_assy	3	8000.00 kg/m³	21.32 cm³	0.17 kg
2.1.1	tripod_bond_prt	3	8000.00 kg/m³	21.32 cm³	0.17 kg
2.2	tubes_assy	3	8000.00 kg/m³	144.92 cm³	1.16 kg
2.2.1	lower_tube_prt	3	8000.00 kg/m³	62.85 cm³	0.50 kg
2.2.2	middle_tube_prt	3	8000.00 kg/m³	34.83 cm³	0.28 kg
2.2.3	upper_tube_prt	3	8000.00 kg/m³	47.24 cm³	0.38 kg

Grouping modes

BOM tables list unique components only; multiple inserts of the same components are grouped, according to the Grouping mode.

The following grouping modes are supported:

- **Auto** (default mode)

  Multiple inserts of the same components are grouped together with respect to their parameters and added properties. It means that multiple inserts of the same parametric component are always grouped in separate BOM rows with respect to their parameters, regardless if those parameters are added to BOM as columns.

Bill of Materials Bookshelf					
No.	Component	Quantity	L	T	W
1	Frame Board	2	360.0	20.0	200.0
2	Frame Board	2	700.0	20.0	200.0
3	Shelf Board	2	266.7	2.5	200.0
4	Shelf Board	2	466.7	2.5	200.0

- **By components and columns**

  In this mode, multiple inserts of the same components are grouped together with respect to added properties only. It means that multiple inserts of the same parametric component could be grouped in the same row even if their parameters are different, depending on what columns are added to BOM. However, instances of different components are always grouped in different rows. This mode is useful, if some parameters correspond to different states of components but not result in being different parts.
- **By columns only**

  In this mode, inserts of components are grouped with respect to their properties added to the BOM, regardless if they are inserts of the same component. This mode is useful to get a summary table, such as a list of all physical materials or a list of all sizes of pipes.

Visible and invisible columns

You can make any column in the BOM table invisible. Invisible columns do not appear in the table in the document. However, you can still use these columns in formulas. Invisible columns are also displayed in the **BOM Manager** panel.

Aggregated columns

It is possible to set an aggregate function to almost every column in top- or bottom-level BOM tables. Aggregate functions group values of several rows in BOM (that differ only in those values) into one single value. Following aggregate functions are supported:

- **Sum**: Sum the grouped values.
- **Average**: Find the average value of grouped values.
- **Minimum**: Find the minimal value of grouped values.
- **Maximum**: Find the maximal value of grouped values.
- **Concatenate**: Concatenate all unique values in the group in ascending order.
- **Concatenate with counts**: Concatenate all unique values with their number of counts in the group.

For example, take the following BOM table:

Component	Width	Length	Height	Quantity
Beam	16	250	10	1
Beam	16	200	10	2
Beam	16	220	20	1
Beam	20	215	20	3
Beam	20	225	20	1

Apply an aggregate function to the **Length** column. The results are combined in the following table:

Component	Width	Height	Length (sum)	Length (average)	Length (minimal)	Length (maximal)	Length (concatenate)	Length (concatenate with counts)
Beam	16	10	650	216.6667	200	250	200; 250	2x200; 250
Beam	16	20	220	220	220	220	220	220

Component	Width	Height	Length (sum)	Length (average)	Length (minimal)	Length (maximal)	Length (concatenate)	Length (concatenate with counts)
Beam	20	20	870	217.5	215	225	215; 225	3x215; 225

For **Concatenated with counts** columns, you can:

- Adjust the position of the number of counts (in front or behind the value).
- Add the delimiter between values.
- Add the delimiter between values and their number of counts.
- Define the postfix and/or prefix for the number of counts.

If an aggregated column is used in an evaluated column, the final values are used by default. However, it is possible to apply an aggregate function to an evaluated column. In this case, the results for such an evaluated column is first computed for all original values (before aggregation), and then the resulted values are aggregated.

Number column and column roles

A number column is a special column to enumerate rows. It assigns numbers automatically for all rows in the BOM table, without skips and independently of other tables. Rows are numbered starting from 1.

Following settings can be changed for the number column:

- **Prefix**: A string that is added in front of any number. For example, you want all numbers to start with "A".
- **Suffix**: A string that is added after the number. For example, you want to add "-1" to all numbers.

For a hierarchical BOM, the following settings are available:

- **Number type**: Specifies the number format:
    - **Full**: Numbers are formatted as "1.2.3".
    - **Short**: Only numbers for the current level show. It is used together with the Level column, which shows the level of the current part in the assembly.
    - **Continuous**: All components are numbered continuously starting from 1, regardless of their level.
- **Delimiter**: Specifies the symbol to separate numbers for different levels when the number type is set to **Full**.

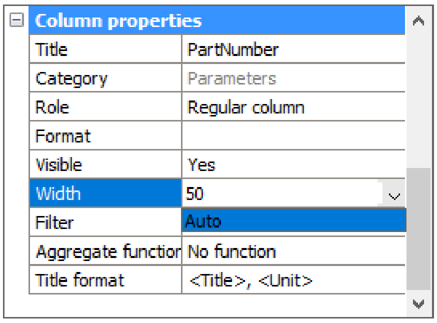
In general, the number column is used as the source to annotate the drawing by the *BMBALLOON* command. However, any column in the table can be used as the source for the annotation balloons, by changing its role. The following column roles are available:

- **Number**: The column contains numbers for annotations.

- **Name**: The column contains part names for annotations.
- **Quantity**: The column contains quantities of parts for annotations.
- **Regular**: The column has no special role.

Only one column can have a **Number**, **Name** or **Quantity** role. If no column has one of these roles assigned, automatic annotations are not possible.

Column width



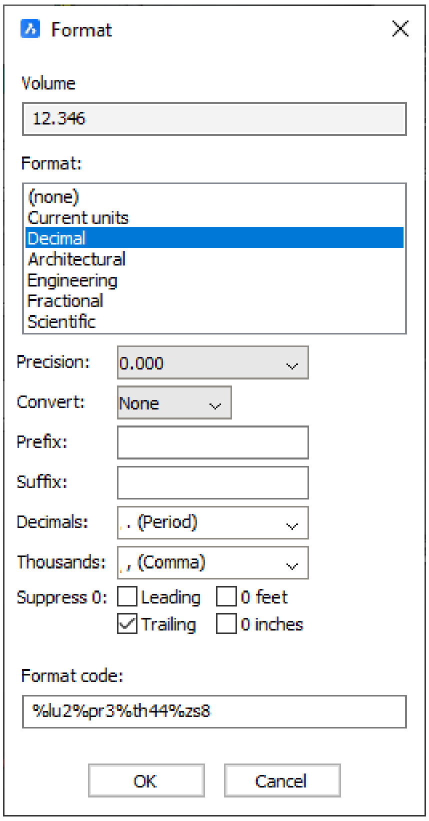
By default, the width of each column is set automatically, so its content fits without additional line breaks. However, it is possible to set a fixed column width.

To set the column width, perform one of the following:

- Select the required column in the **BOM manager** panel, then type a value for the **Width** property in layout units.
- Change the width of the required column in the associated table. BOM tables automatically keep the width of columns that have been changed manually.

To restore the automatic width mode, select the column in the **BOM manager** panel and select **Auto** in the drop-down list for the **Width** property.

Column format



You can format column values:

1 Select the required column in the **BOM manager** panel.
2 Choose the **Format** option in the context menu.
3 Define the desired format in the **Format** dialog box.

Note: The available settings depend on the data type of the selected column. You can select multiple columns of the same type to format them equally.

Property sets

Because BOM tables use DataExtraction technology, you can list properties of mechanical components and entity properties, for example, block references in BOM tables.

Note: These properties are not always necessary. In large assemblies, these properties could negatively affect performance. This is because BricsCAD caches properties during BOM table creation.

Property sets are built-in sets of properties. You can choose the most optimal set of properties for your needs. The following property sets are supported:

- **Mechanical only**(default property set): Only properties of mechanical components are listed.
- **All except coordinates**: Lists all properties of components, except coordinate ones. Coordinate properties are typically used only for component instances.
- **All properties**: Lists all possible properties of components and component instances.

You may change the property set for a BOM table in the **BOM Manager** panel or use *BMBOM* when you add a new table.

29.13.4 Filter a BOM table

You can filter parts that will form the Bill of Material. There are 2 filter types: **main** and **column**.

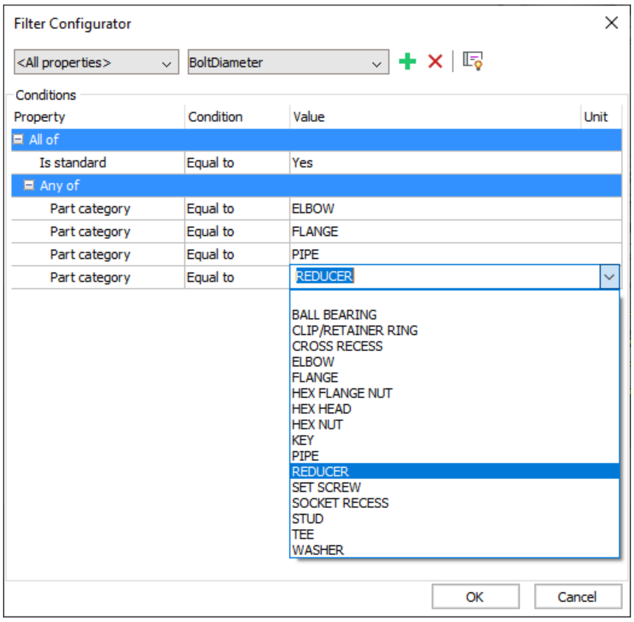
Main filter

A Main filter works in the same way as the DATAEXTRACTION command. First, all parts in the assembly, at the required level (top, bottom or hierarchical), are matched against the filter expression. Then, only the parts that have passed the filter are included in the BOM.

The filter expression may not refer to BOM columns because filtering is done before the BOM is generated. For example, it is possible to create a filter for all standard parts because **Is standard** is a normal property of the component. However, it is not possible to set the main filter to filter out all parts with a quantity greater than 1. You must use Column filters for that.

Filter configurator

The easiest way to configure the main filter is to use the **Filter Configurator** in the **BOM Manager** panel.



The Filter Configurator allows to define a set of conditions. Each condition defines a relation between a selected property and a target value.

Possible relations are:

- Equal to
- Not equal to
- Less than
- Greater than
- Less than or equal to
- Greater than or equal to

Conditions can be added to a group, which defines how those conditions are treated. There are 2 types of groups:

- **All of** group: is "true" if all conditions in that group are "true". This is the same as the "and" operator.
- **Any of** group: is "true" if at least one condition in that group results in "true". This is the same as the "or" operator.

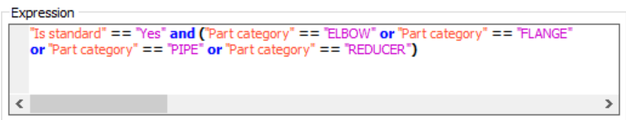
A group can contain an arbitrary number of conditions. It can also contain nested groups, thus making complex filters possible.

To add a condition:

- Select a property from the list of properties.
- Select a required relation operator and specify the target value.
- For text values, a target value can be selected from the list of all possible values also. For numerical values, their target unit can be specified also in order to automatically convert the value with respect to the default unit of the document.

For text values, "Equal to" and "Not equal to" relations support wild cards.

The Filter Configurator automatically converts the set of conditions into a filter expression, which can be made visible by clicking :



This expression is fully editable and two-way associative with the set of conditions.

Syntax description:

Operator	Meaning
==	equal to
!=	not equal to
<	less than
>	greater than
<=	less than or equal to
>=	greater than or equal to

- Property names must be enclosed in quotes.

- Relational operators can be combined using "and" and "or" operators. Operator "and" results in "true" if both sides of the operator are true. Operator "or" results in "false" if either left or right side of the operator is true.
- Logical operators "and" and "or" are case sensitive.
- Use "(" and ")" to combine two operators into one.
- String values are enclosed in double quotation marks "".
- Relational operators for strings are case sensitive.
- "==" and "!=" operators support wild cards.

For example, to get a BOM of all standard parts, use the following expression:

"Is standard"=="Yes"

To get a BOM of all ISO standard parts, use the following expression:

"Is standard"=="Yes" and "Part standard"=="ISO"

To get a BOM of all ISO or DIN standard parts, use the following expression:

"Is standard"=="Yes" and ("Part standard"=="ISO" or "Part standard"=="DIN")

Wild cards

Wild cards allow you to match property values against a specific template. For example, if you want to find all components containing "nozzle" in their names, the following expression could be used: "Component name"=="*nozzle*".

Wild cards can also be used as target values in the **Filter Configurator**.

Only "Equal to" and "Not equal to" relational operators support wild cards.

Note: Wild cards in the main filter are case-sensitive. For Column filters, you can specify if you need case-sensitive or case-insensitive matching.

The following special symbols are supported:

Name	Character	Description
Back quote	`	Escape the next character (takes it literally).
Comma	,	Separate patterns.
Asterisk	*	Match any character sequence, including a null sequence, anywhere.
Question mark	?	Match a single character.
Period	.	Match a single non-alphanumeric character.
At sign	@	Match a single alphabetic character.

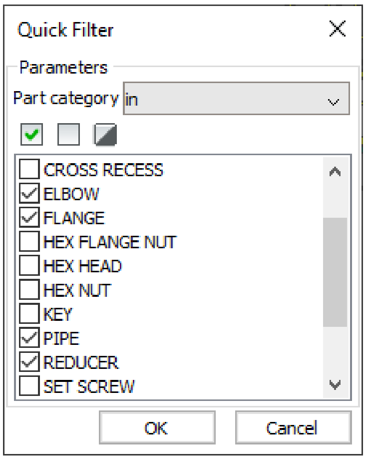
Name	Character	Description
Hash tag	#	Match a single numeric character.
Tilde	~	If the first character in a pattern matches anything but the pattern.
Brackets	[...]	Match a single enclosed character.
Tilde brackets	[~...]	Match a single non-enclosed character.
Hyphen	-	In brackets, match a single character in the specified range.

Column filter

Column filters allows you to quickly set the conditions which rows display, depending on the column content (somewhat similar to filters in some spreadsheet processors). Unlike the main filter, they are applied after the BOM table has been generated. Therefore, it is possible to use column filters for columns with aggregated content, such as quantities.

Each column can have one filter only. However, you can add column filters to as many columns as you need. Only rows that pass all column filters display in the final BOM.

You can configure column filters using the **Column Filter** dialog box in the **BOM manager**:



In this dialog box, you can select the type of filter from the list and configure it accordingly. The list of possible filters depends on the column content type, as shown in the table:

Filter type	Column type	Description
always accepted	any	all column values are accepted

Filter type	Column type	Description
equal to	numerical	value should be equal to the target one
not equal to	numerical	value should be not equal to the target one
less than	numerical	value should be less than the target one
greater than	numerical	value should be greater than the target one
less than or equal to	numerical	value should be less than or equal to the target one
greater than or equal to	numerical	value should be greater than or equal to the target one
between	numerical	value should be in the given range, both ends included
not in range	numerical	value should not be in the given range, both ends excluded
exactly matches	text	value should exactly match the target one
does not match	text	value should be different from the target one
starts with	text	value should start with the given text
does not start with	text	value should not start with the given text
ends with	text	value should end with the given text
does not end with	text	value should not end with the given text
contains	text	value should contain the given text
does not contain	text	value should not contain the given text
matches pattern	text	value should match the given wild card template
does not match pattern	text	value should not match the given wild card template

Filter type	Column type	Description
in	text and numerical	value should be in the given set of values
not in	text and numerical	value should not be in the given set of values

For all text filters, you can also specify whether they should be case-sensitive or case-insensitive.

29.13.5 Selection set

In addition to main and column filters, it is possible to specify a selection set for a bill of materials. When a selection set is specified, the corresponding BOM table will show only parts from that selection set. Following types of selection sets are supported:

- **Entire model** – default selection set, all parts and subassemblies in the current model will be included.
- **Subassembly** – allows to select a subassembly, which will act as a root component for the bill of materials.
- **Drawing view** – allows to link a BOM table with a certain drawing view, so only parts included in that view will be shown in BOM. Note that even if a part is not visible in a particular drawing view, but has been included into this view, then it will be included in the corresponding BOM table.
- **Layout viewport** – allows to link a BOM table with a layout viewport (including a drawing view), so only parts visible in this viewport will be shown in BOM. The difference with the previous mode is that only visible parts will be included; parts, completely hidden or not in the viewport will be not shown in the BOM table.
- **Custom selection** – allows to select a subset of top-level components, which will be shown in a BOM table.

To set a selection set for a table, use **BOM manager** panel or a corresponding option of BMBOM. It is also possible to change a selection set for an existing BOM table using either *Edit selection* command of **BOM manager** panel or BMBOMEDIT command.

29.13.6 Sort a BOM table

There are 3 sorting modes for BOM tables:

- **Automatic order**

  The table content is sorted as follows: component name, physical material name, thickness (if applicable), parameters (in alphabetic order), and quantity.
- **Natural order**

  The table content is not sorted. Instead, all components are in the order of their appearance in the document. In most cases, parts added earlier will precede parts added later.
- **Custom order**

  The table content is sorted according to the columns you select. If no columns are specified, the natural order is used.

BOM tables use so-called "natural sorting" of text. It means that if text contains a mix of letters and numbers, it will be sorted with respect to those numbers. For example, strings "N1", "M2", "N11", "N2", "AA1" will be sorted as "M2", "N1", "N2", "N11", "AA1." Therefore, it is possible to assign custom numbers with various prefixes to components using custom parameters, and then sort the BOM tables by those numbers in the correct order.

29.13.7 Thumbnails

Bill of Materials Rotary platform assembly			
No.	Thumbnails	Component	Quantity
8		Motor	1
9		Gear	1
10		Reducer	1
11		Valve support plate	1
12		Pneumatic silencer N18	1
13		Push-in fitting 1/8-4	2
14		Push-in fitting 1/8-10	1

Thumbnails are preview images of parts included in the BOM table, generated from 3d geometry of corresponding parts. When a BOM table with thumbnails is added to a layout or to a model space, those images are stored as attachments, so they can be manipulated using Attachments panel. Note that unless the document is saved, those images will be in a temporary folder. They will be moved to a specified folder (usually named **bom_thumbnails** in the drawing's folder) only when the document is saved.

To add thumbnails to your BOM, simply add Thumbnails column and adjust thumbnail properties if necessary. Following properties can be adjusted:

- **Path** – folder where images will be stored after the document is saved.
- **View type** – orientation of the camera. Options correspond to Look from widget options.
- **Visual style** – visual style applied to parts. By default, current visual styles in the document are used, but it is possible to select any visual style available in the document. In addition, a special visual style **BM_tangent** is available, which imitates a draft quality black and white drawing views.
- **Background color** – color used for images background. You can select any color as a background, or a transparent background. Note that generation of thumbnails with transparent background usually takes longer time.
- **Size (width and height)** - thumbnail image size in pixels. Default size could be set in **Default thumbnail width** and **Default thumbnail height** global settings.

One BOM table can contain multiple thumbnail columns with different parameters.

In case of large assemblies, thumbnail generation may take a considerable time. By default, thumbnails are updated automatically when the model is changed. However, it is possible to force bill of materials to reuse existing thumbnails with specified parameters, even if they are no longer match the part geometry. To do that, uncheck **Automatically update thumbnails** in **Bill of materials defaults** setting. In addition, you may temporarily disable thumbnail columns, so they will not generate thumbnails. When enabled, missing thumbnails will be generated automatically. To disable a thumbnail column, change its **Enabled** property to **No** in the **BOM manager** panel.

29.13.8 Evaluated columns

Evaluated columns are special columns, of which results are depending on other columns using user-defined expressions. 2 types of evaluated columns are supported: **formula** and **template** columns.

Formula columns

A Bill of Materials supports formula fields. The same syntax is used in the DATAEXTRACTION command. **FormulaExpression** is the expression in the format of **\AcExpr** formula fields. This can contain constants such as PI or functions such as SIN. An operand in an expression can be a **ColumnTitle** enclosed in arrow brackets (<>).

For example, formula *=<Mass>/<Volume>* calculates the average density of the component.

If a row does not contain a value, a default value will be used instead. It is possible to specify the default value for an operand using the following syntax: *<"ColumnTitle",DefaultValue>* where **ColumnTitle** is the title of the required column, and **DefaultValue** is the default value. **ColumnTitle** must be between double quotes ("). If no default value is specified, an empty value is used, which in most cases results in an empty output.

A formula column can contain multiple expressions. Each expression must be enclosed by sum brackets ({}). These expressions are evaluated from first to last, and the first one yielding a non-empty output is used.

A formula can refer to other formula columns, provided that they do not form a loop. In a hierarchical BOM, a formula can also refer to values of the parent row by adding *Parent:* before the column name. It also can refer to properties that are not explicitly added to the table, but all other formula columns referred to in the formula must exist in the table. You can still use such columns displayed as invisible in the BOM table in formula expressions.

For distance, area, volume and mass properties, the values will be converted to the **user-defined** unit before using them in the formula expression. If the **user-defined** unit is not set, the document unit is used. For the mass property, kilogram is used in metric documents and lbs (pound) in imperial documents.

Template columns

Template columns allow you to combine several properties, as well as a custom text, into a single column. A special syntax, based on Diesel expressions, is used to define template columns. Expressions for template columns must start with the **$** sign. An operand in an expression can be a **ColumnTitle** enclosed in arrow brackets (<>).

It is also possible to use special functions:

Function	Description
$(+, arg1, arg2[, ... , arg9])	Add the provided arguments to the first one.
$(-, arg1, arg2[, ... , arg9])	Subtract the provided arguments from the first one.
$(*, arg1 , arg2[, ... , arg9])	Multiply the first argument by the provided arguments.
$(/, arg1, arg2[, ..., arg9])	Divides the first argument by the provided arguments.
$(=, arg1, arg2)	Return 1 if arg1 is equal to arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(<, arg1, arg2)	Return 1 if arg1 is less than arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(>, arg1, arg2)	Return 1 if arg1 is greater than arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(!=, arg1, arg2)	Return 1 if arg1 is not equal to arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(<=, arg1, arg2)	Return 1 if arg1 is less than or equal to arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.

Function	Description
$(>=, arg1, arg2)	Return 1 if arg1 is greater than or equal to arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(and, arg1, arg2[, ... , arg9])	Apply bitwise AND operation to arguments.
$(angtos, angle[, mode, precision])	Convert the angle to string with respect to given mode (0 - degrees, 1 - d/m/s, 2 - grads, 3 - radians) and precision.
$(delim, delimiter_string)	Insert a smart delimiter. The delimiter string is added to the output only if the output before and after this delimiter is not empty.
$(edtime, time, format)	Format the date according to the given format string.
$(eq, arg1, arg2)	Return 1 if arg1 is equal to arg2; otherwise, return 0. Both arg1 and arg2 must be text.
$(eval, expression)	Evaluate the given expression as a Diesel expression.
$(fix, value)	Truncate the real value to the integer one.
$(getvar, variable)	Return the value of the given system variable.
$(if, arg, then[, else])	Return 'then' if 'arg' is not 0; otherwise, return 'else' or empty string if 'else' is not provided.
$(index, position, comma_separated_string)	Return the substring at the given position in the given comma-separated string.
$(nth, position, arg1[, arg2, ... , arg9])	Return the argument at the given position.
$(or, arg1, arg2[, ..., arg9])	Apply bitwise OR operation to arguments.
$(rtos, value[, mode, precision])	Format the given value with respect to the required mode and precision.
$(strlen, string)	Return the length of the given string.
$(substr, string, start[, length])	Return the substring of the 'string' from 'start' of the given 'length'. Numeration starts from 1.
$(upper, string)	Convert 'string' to uppercase.

Function	Description
$(xor, arg1, arg2[, ... , arg9])	Apply bitwise XOR to all arguments.

One template column can contain multiple expressions. Each expression must be enclosed by sum brackets ({}). These expressions are evaluated from first to last, and the first one yielding a non-empty output is used.

A template column can refer to other evaluated columns, provided that they do not form a loop. In a hierarchical BOM, a template column can also refer to values in the parent row by adding *Parent:* before the column name. It can also refer to properties that are not explicitly added to the table, but all other evaluated columns referred to in any of the expressions must be present in the table. You can still use columns displayed as invisible in the BOM table in template expressions.

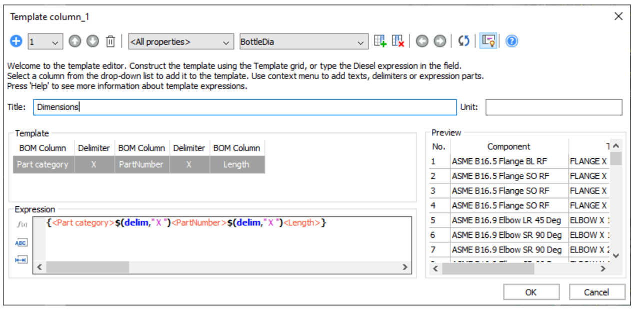
You can use "\n" string to add a line break.

For example, the following expression combines PartNumber, Width, Height and Length properties:

${<PartNumber>$(delim," X ")<Width>$(delim," X ")<Height>$(delim," X ")<Lenght>}

Expression configurator

You can use the *Expression configurator* to configure an evaluated column. This dialog box is available in **Bill of Materials Manager** panel.



This dialog box allows you to set the title and unit for the column, edit the expression using an editor with syntax highlight and autocompletion, and see a preview of the BOM table with the evaluated column. For a template column, the **Template** section allows you to easily insert, move and delete properties, as well as delimiters and custom texts.

29.13.9 Column units

You can adjust the **units placement** mode for columns that have associated units. The following options are available:

- **Don't show units**

  The BOM table does not display units for this column.
- **In column title**

if all values in the table use the same unit or can be converted to a common unit, all values are converted to that common unit. The unit name is added to the column title. You can configure the position of the title and the unit using the **Title format** property. Use *<Title>* to denote the title, and *<Unit>* to denote the unit. For example, the default title format is "*<Title>*, **<Unit>**." You can use "\n" to add a line break.

The common unit is picked in the following order:

a A unit set by you.
b If the property is a distance, an area, a volume or a mass, and the unit mode is **Best for all values**, the best representation for the majority of the values is used.
c If all values have the same unit, this will be used.

- **In cell**

  The unit name is added to the value. Value and unit display in the same cell.
- **In additional column**

  The unit name is placed in a separate column.

You can set the **Unit mode** for distance, area, volume and mass values to define the units used to display a particular value. For example, if there are masses going from 1 gram to several tons, choose **Best for each value** to display each mass converted to the most suitable mass unit. To see all masses in kilograms, set kilograms as the **user-defined** unit for the column. **Best for all values** is the default.

Units mode options:

- **Don't show units**

  Units are not displayed.
- **Best for all values**

  The unit with the best representation for the majority of the values is displayed. Only dimensions, area, volume and mass properties are supported.
- **User-defined**

  You choose the unit to display. In addition to dimensions, area, volume, and mass properties, it is also possible to select a required unit for material properties, such as density, thermal conductivity and specific heat.
- **Best for each value**

  The unit that is most suitable for each value is used. The **In column title** mode of the **Unit placement** automatically switches to the **In additional column** mode. Only dimensions, area, volume, and mass properties are supported.

Note: If a user-defined unit is set for a column and this column is used in a formula field, the values converted to that unit will be substituted in the formula. Otherwise, the document unit will be used.

29.13.10 Footer

The Bill of Materials table supports a footer. To define which columns will form the footer, set the **Footer type** for the required columns:

- **No footer**

  The column ID will not be included in the footer row.
- **Sum**

  The sum of all values for each individual component instance displays in the footer.

  Note: This is not the equivalent of the sum of all values displayed in the table, because the number of each component is used. For formula fields, the sum of values for Quantity=1 is used.
- **Average**

  The average value, of all values, for each individual component, displays in the footer. For formula fields, the sum of values for Quantity=1 is used.
- **Minimal**

  The smallest value for each individual component displays in the footer. For formula fields, the sum of values for Quantity=1 is used.
- **Maximal**

  The largest value for each individual component is displayed in the footer. For formula fields, the sum of values for Quantity=1 is used.

Note: If all columns have the footer type **No footer**, the footer row does not display.

Note: It is possible to add a footer to formula or template columns as well.

29.13.11 Templates

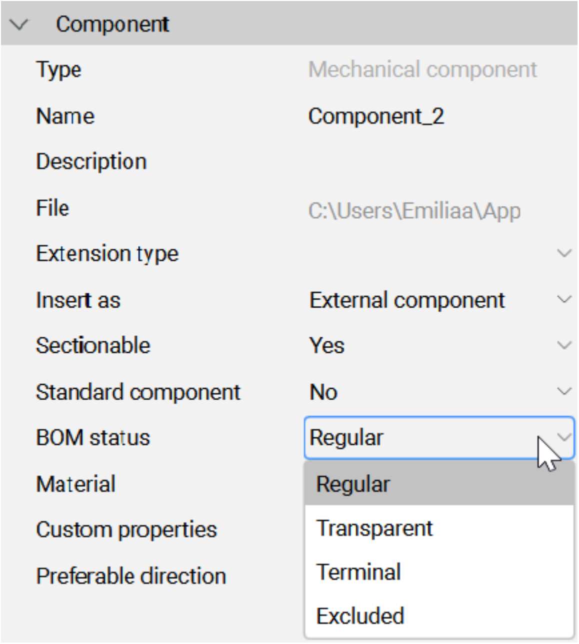
You can save a BOM configuration as a template and use it to create new BOM tables. The default template to create a new BOM table is stored in BOMTEMPLATE system variable. The template file contains all necessary data. It includes table type, title, filter, and set of columns with all the associated properties. You can use <NAME> in the title as a placeholder for the actual name of the assembly. For example, if your assembly name is 'My Assembly' and the title of the BOM is specified as Bill of Materials for <NAME>, the BOM title will be 'Bill of Materials for My Assembly', because 'My Assembly' replaces <NAME>.

By default, BOM template files have a .bom extension.

A BOM template file is an extension of the Data Extraction Definition file (.dxd). However, you cannot use a .dxd file as a BOM template, because some BOM-specific data will be missing. You cannot use a .bom file to define a DataExtraction template.

You can edit template files in any text editor. However, Bricsys recommends to use the BMBOM command or the BOM manager to create and edit template files.

29.13.12 BOM status



Each component has a **BOM status** property. You can use this to control the behavior of the component in **Bill of Materials** tables. This property can have one of the following values:

- **Regular**

  The component appears in a Bill of Materials at the same position as the assembly structure and the default value of the selected mode for a Bill of Materials.
- **Transparent**

  The component acts as a transparent container for its subcomponents. The component does not appear in the Bill of Material, but subcomponents can appear in the BOM, depending on their BOM status, position in the assembly structure, and the mode of the BOM table.
- **Terminal**

  The component is treated as a leaf component (has no subcomponents). This is useful for complex parts or standard components.
- **Excluded**

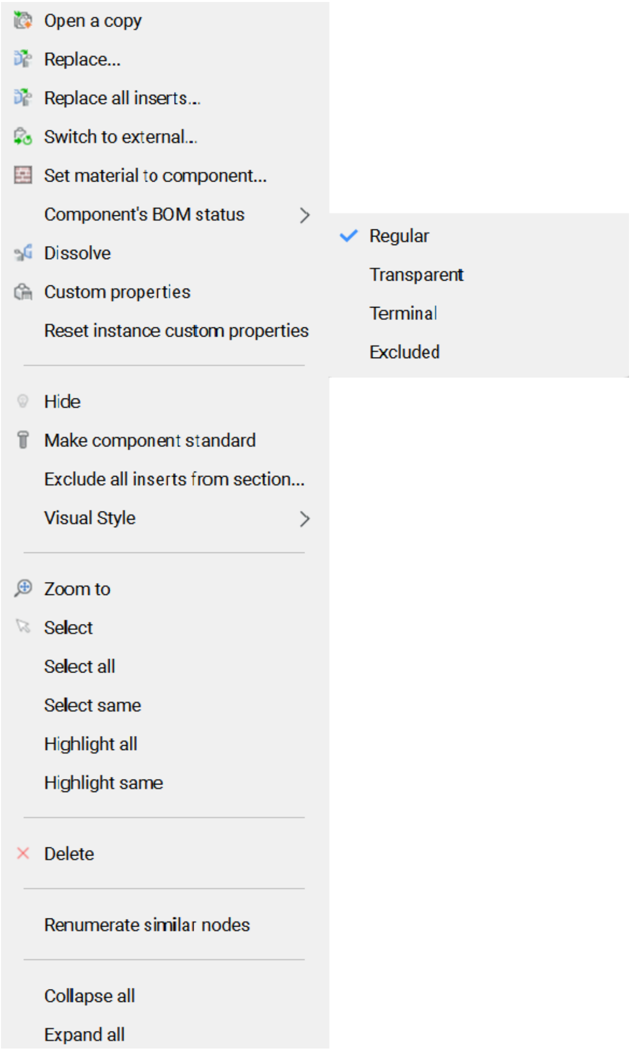
  The component and its subcomponents do not appear in the Bill of Materials.

To set the BOM Status of an external or root component

1. Open the file that contains the component.
2. Select the root component in the Mechanical Browser.
3. Set the **BOM status** property.

To set the BOM Status of a local component

1. Select the component in the **Mechanical Browser** panel.
2. Right-click on the selected node to open the context menu.
3. Select a BOM status from the **BOM status** submenu.



Note: It is not possible to set the BOM status via the context menu for local components that are inserted into an external component. To do this, you must open the component that contains the local component.

29.13.13 Fixed corners

You can set any corner of the table as fixed. The fixed corner will always keep its initial position during BOM update. When the table has been moved manually, the new position of this corner will be kept. Fixed corners allow to align BOM tables with other objects.

You may choose which corner to fix during the table placement by either the BMBOM command or the **BOM Manager** panel; use the Tab key to cycle the corners. You can change the fixed corner of a BOM table in the **BOM manager** panel.

29.14 Multi-parametric 3D animations

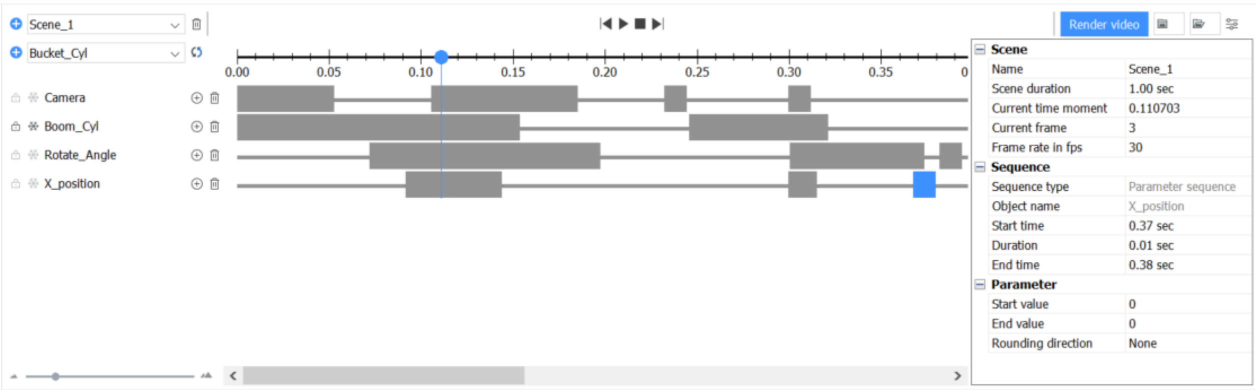
29.14.1 Overview

With the **Animation Editor** panel you can animate several parameters and program camera movements. Once the animation is programmed, you can play it in BricsCAD or render it as a 2D video.

Each parameter animation has its own timeline. You can add as many animation sequences to a parameter as you like, as long as these sequences do not overlap.

There is also a special timeline for camera movements. You can add as many camera sequences as you like. You can set both the point of focus and the position of the camera with either a polyline or a point. This is similar to the ANIPATH command.

Timeline



29.14.2 Scenes, Timelines and Animation Sequences

Scenes

A scene consists of one or more **timelines** and each of these timelines has one or more **animation sequences**. When the editor is launched for the first time, it automatically creates a scene. A scene sets the duration of the animation, the current time moment, and the current frame, which are editable. Together, they define the **animation schedule** used for playback and render. Currently, you can only render the current scene with the **Animation Editor**. However, you can have multiple scenes in the same document and you can switch between them.

Animation sequences

An animation sequence is the basis of an animation. It represents the single, continuous animation of an object. An animation sequence has an associated start time and duration. It also has values specific to the animated object, such as start and end values for parameters, or camera and viewpoint paths for camera sequences:

- **Parameter**

  An animation sequence represents the continuous change of a parameter from a start value to an end value. For example, a distance constraint between two faces: the animation will move one face to a maximum and minimum distance value.

- **Camera**

An animation sequence corresponds to a continuous camera movement on a path. You can adjust various camera movements with the camera timeline.

- **Exploded view**

  An animation sequence represents the exploded view assembling or disassembling process from the first step to the last step.

Timelines

A timeline is a visual representation of an animation sequence for an object. The timeline displays how and when the parameters will animate. Each parameter has its own timeline in a scene. Animation sequences cannot overlap. A timeline can be temporarily disabled to exclude it from an animation, or can be locked in order to disable all the possible changes of it. All timelines are independent. This means that you can animate multiple parameters at the same time and see their combined effects.

29.14.3 Opening the Animation Editor

Do one of the following:

- Place the cursor on a toolbar, right-click, and choose **Animation Editor** from the Panels context menu.
- Click the **Animation Editor** tool ( ) in the Ribbon.
- Enter ANIMATIONEDITOROPEN in the Command line.

29.14.4 The Animation Editor interface

The **Animation Editor** is a panel. It has 3 main areas:

- **Toolbar**: contains tools to select a scene, adjust an animation and render a video.
- **Timelines area**: adds and edits the timelines and animation sequences.
- **Configurator**: configures the parameters of the current scene and the selected animation sequence.

Toolbar

The toolbar contains tools to:

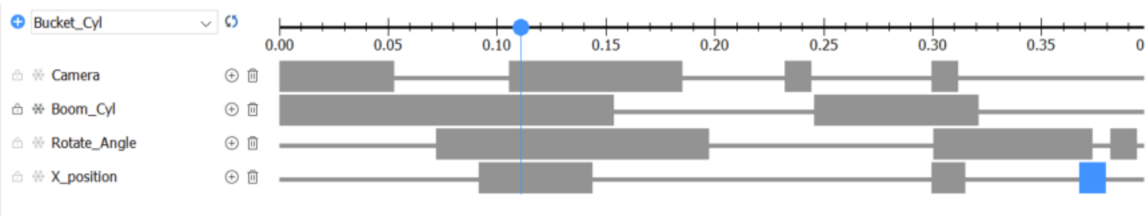
- Add and remove scenes.
- Control the playback of the animation.
- Set the duration of the current scene.
- Add a camera timeline.
- Start a render.
- Record animation graphics and save it to a zip-archive or folder.
- Load the saved graphics and playback it in the special mode.

Tool	Name	Description
	Add Scene	Adds an empty scene with a default duration.
Scene_1	**Scene Selector**	Selects the current scene.
	Remove Scene	Removes the current scene from the document.
	To previous sequence	Jumps to the start of the previous animation sequence.
	Play	Starts the animation. Toggles to the pause button after pressing.
	Pause	Pauses the animation. Toggles to the play button after pressing.
	Stop	Stops the animation and return the cursor to the start position.
	To next sequence	Jumps to the start of the next animation sequence.
Render video	**Render video**	Opens a dialog box to start the render of a 2D video.
	Record animation graphics	Records animation graphics and saves it to a zip-archive.
	Record animation graphics	Records animation graphics and saves it to a folder of your choice.
	Load animation graphics	Loads animation graphics data from zip-archive and opens the special mode for playing this graphics.
	Load animation graphics	Loads animation graphics data from a folder of your choice and opens the special mode for playing this graphics.
	Unload animation graphics	Unloads the animation graphics which was loaded before. The button is available in the special mode for playing animation graphics only.

Tool	Name	Description
	**Toggle configurator**	Toggles the configurator panel.

Timelines Area

Use the **Timelines Area** to add and remove timelines and animation sequences, and to display the animation schedule. Click an animation sequence to select it, or drag the start or end of the bar to adjust the duration. The selected animation sequence displays in blue, other sequences display in gray. Each timeline in the timelines area can be temporarily disabled to exclude it from the animation.

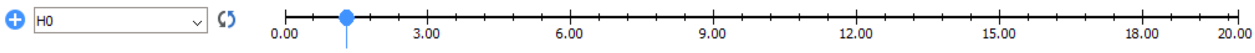


The **Timelines Area** has a cursor that represents the current position in the animation. When the animation is played, the cursor represents the current playback position. You can also manually move the cursor to a new position at any time, or edit the **Current time moment** or **Current frame** parameter. When the cursor is moved, all parameters change to match the current frame. The camera position is also updated.

To make sure that the model state corresponds to the current animation position, click .

Note: You can manually adjust this at any moment, but this will not affect the animation.

Time Scale



Tool	Name	Description
	**Add Timeline**	Adds the selected parameter to the timeline.
	**Object Selector**	Selects the object to add to the timeline.
	**Update model**	Updates the model state with respect to the current cursor position.
	Time Scale	Displays the time marks along the timeline.
	**Cursor**	Displays the current position of the playback.

Timeline

Tool	Name	Description
	Timeline UnLocked	Indicates that the timeline is opened for changes. Click to lock the timeline.
	Timeline Locked	Indicates that the timeline is locked for changes. Click to unlock the timeline.
	Timeline Enabled	Indicates that the timeline is enabled. Click to disable the timeline.
	Timeline Disabled	Indicates that the timeline is disabled. Click to enable the timeline.
Height	**Timeline Object**	Displays the name of the associated object.
	Add Sequence	Adds a new animation sequence, at the cursor position.
	Remove Timeline	Removes a timeline from the animation.
	Animation Sequence	Represents the duration and position of an animation sequence. Left-click to select the sequence.

Tool	Name	Description
	**Selected Animation Sequence**	Represents the duration and position of the selected animation sequence. You can see and change the properties in the **Configurator**. Move the mouse to the end of the sequence, until the cursor changes to a double-sided arrow. Then left-click and drag to change the start time, end time and duration. Left-click at the middle of the sequence and drag to a new position at the timeline, without changing the duration. ***Note:*** Moving and dragging sequences supports three types of snapping that are displayed with an yellow line: • Snapping by scale • Snapping by cursor • Snapping by sequence intervals of other timelines (snapping by exploded view steps is also supported)

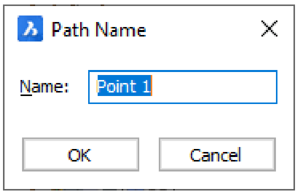
Configurator

Access all the properties of the animation scene and the selected sequence via the **Configurator**. All properties are grouped by object.

- **Scene**: properties of the current scene.
    - **Name**: displays the name of the scene.
    - **Scene duration**: displays the duration of the scene, in seconds.
    - **Current time moment**: displays the current animation position.
    - **Current frame**: displays the current animation frame.
    - **Frame rate in fps**: displays the frame rate of the scene, in frames per second.
- **Sequence**: properties of the selected sequence.
    - **Sequence type** (read only): displays the type of the sequence.
    - **Object name** (read only): displays the name of the object controlled by that sequence.
    - **Start time**: displays the start time of the sequence in seconds. The duration will be adjusted, the end time will remain unchanged.
    - **Duration**: displays the duration of the sequence in seconds. When changed, the end time will be adjusted with respect to the start time and duration.
    - **End time**: displays the end time of the sequence in seconds. When changed, the duration will be adjusted, the start time will remain unchanged
- **Parameter** (for parametric sequences)
    - **Start value**: displays the start value of the parameter.

- **End value**: displays the end value of the parameter.
- **Rounding direction**: defines the rounding direction type.
- **Precision**: sets the tolerance for rounding precision.

- **Camera** (for camera sequences)
  - **Camera position**: camera path entity type. Values:
  - **Fixed**: the camera is fixed in the given point.
  - **Dynamic**: the camera moves along the given path.
  - **Camera point** (when the **Camera position** is set to **Fixed**). The following controls are displayed:

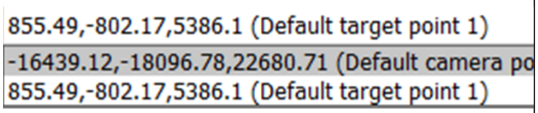
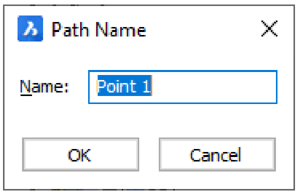
855.49,-802.17,5386.1 (Default target -16439.12,-18096.78,22680.71 (Defau 855.49,-802.17,5386.1 (Default target	List of previously used points. Select the point from the drop-down list to use one.
	Click to select the point in the model. When the point is selected, a dialog box displays:  • **Name**: allows you to enter the point name. • **OK**: accepts the point selection. • **Cancel**: cancels the point selection.
	Removes the point from the list.

  - Camera path (when the **Camera position** is set to **Dynamic**). The following controls are displayed:

Path1 Path1 Path2	List of previously used paths. Select the point from the drop-down list to use one.

	Click to select the path in the model. When the path is selected, a dialog box displays:  • **Name**: allows you to enter the path name. • **OK**: accepts the point selection. • **Cancel**: cancels the point selection.
	Removes the path from the list.

- **Target**: The target defines the point the camera will look at the given moment.
  - Values:
  - **Fixed**: The target is fixed to a given point.
  - **Dynamic**: The target moves along a given path.
  - **Target point** (when the **Target** is set to **Fixed**). The following controls display:

 	List of previously used points. Select the point from the drop-down list to use one.
	Click to select the point in the model. When the point is selected, a dialog box displays:  • **Name**: allows you to enter the point name. • **OK**: accepts the point selection. • **Cancel**: cancels the point selection.
	Removes the point from the list.

  - **Target path** (when the **Target** is set to **Dynamic**). The following controls display:

 	List of previously used paths. Select the point from the drop-down list to use one.

	Click to select the path in the model. When the path is selected, a dialog box displays:  • **Name**: allows you to enter the path name. • **OK**: accepts the point selection. • **Cancel**: cancels the point selection.
	Removes the path from the list.

Note: Similar to ANIPATH, all selected points and paths are stored in the drawing.

29.14.5 Animation Playback

Use the animation playback controls to preview an animation.

- To **start** an animation, click the **Play** button.

  The playback starts from the cursor position. The button will switch to the Pause button.
- To **pause** the animation, click the **Pause** button. The button will switch to the Play button.
- To **stop** the animation and move the cursor to the start of the animation, click the **Stop** button.

You can jump to the next or to the previous animation sequence (in chronological order) using the **Jump to Next Sequence** and **Jump to Previous Sequence** tools respectively.

Note:

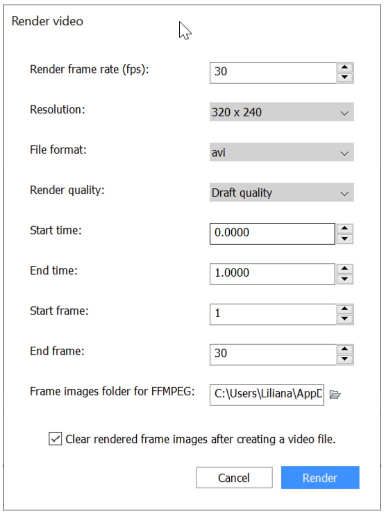
- During playback, the parameters involved in the scene and the camera position are changed directly in the drawing. For complex models, the recalculation of the model may take a significant amount of time. You can use the cursor to check the key moments of the animation instead.
- During playback, the camera will move with any set camera sequences. If no camera sequences are present, the current view is used. The output video is the same as the current viewport.
- After the playback finishes, the model will automatically revert to the state before the animation was started. In some cases, it may cause the model to be in a different state than the cursor position would dictate. To update the model with respect to the current cursor position, click .

29.14.6 Render

1 Click the **Render video** button to render the animation as a 2D video.

  The *Render video* dialog box displays:

Render video



2 Adjust the render parameters:
   - **Render frame rate (fps)**: sets the frame rate.
   - **Resolution**: sets the resolution of the video.
   - **File format**: sets the format of the video (AVI, MPG and WMV formats are supported).
   - **Render quality**: sets the quality of the render (draft and full quality).
   - **Start time**: sets the time position the render will start.
   - **End time**: sets the time position the render will end.
   - **Start frame**: sets the frame position the render will start.
   - **End frame**: sets the frame position the render will end.
   - **Frame images folder for FFMPEG**: displays the folder where the frame images will be stored.
   - **Clear rendered frame images after creating a video file**: toggles the resetting of the frame images after creating a video file.

3 Click the **Render** button to start the render.

   A dialog box displays.

4 Specify the location and the name of the output file.

5 Click **Save** to start the render.

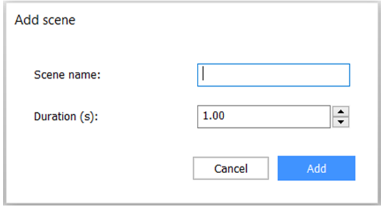
   A preview shows the progress.

29.14.7 Adding a new scene

1 Click the **Add Scene** button.

   The **Add scene** dialog box displays:

Add scene



2 Enter the scene name.

3 Enter the scene duration in seconds.

4 Click **Add** to add the scene.

29.14.8 Removing a scene

1 Select the scene with the **Scene Selector**.

2 Use the **Remove Scene** tool.

29.14.9 Adding an animation of a parameter

1 Add the desired parameter to the timeline:

1 - Select the parameter with the **Parameter Selector**.
  - Use the **Add Timeline** tool to add a new timeline.

2 Place the cursor at the desired start position on the **Time Scale**.

3 Use the **Add Sequence** tool to add a new animation sequence, or double-click at the timeline where you want to add a new sequence.

4 Drag to adjust the desired duration of the sequence, or use the **Configurator** to set the desired start time, end time or duration, in seconds.

For convenience, use the three snappings in case of dragging:

- Snapping by scale
- Snapping by cursor
- Snapping by sequence intervals of other timelines

5 Use the **Configurator** to set the desired start and end values for the parameter.

29.14.10 Adding a camera movement

1 If applicable, add a camera timeline by selecting **Camera** from the **Object selector**.

2 Place the cursor at the desired start position on the **Time Scale**.

3 Use the **Add Sequence** tool on the camera timeline to add a new camera sequence, or double-click at the timeline where you want to add a new camera sequence.

4 Click and drag to adjust the desired duration of the sequence, or use the **Configurator** to set the desired start time, end time or duration, in seconds.

5 Use the **Configurator** to set the camera path:

- Choose the **Camera position**:
- • **Fixed**: the camera position will be fixed.
- • **Dynamic**: the camera will move along the given path.
- Use the Camera point property to select the point or the **Camera path** to select the path.
- Select the property for editing, then click to select the path or the point on the model, or select one of the previous points or paths from the drop-down list.

6 Use the **Configurator** to set the target path. The **target** is the point where the camera will focus on.

Choose the **Target**:

1 - **Fixed**: the target will be fixed.
- **Dynamic**: the target will move along the given path.

Use the **Target point** property to select the point or the **Target path** to select the path:

Select the property for editing, then click to select the path or the point on the model, or select one of the previous points or paths from the drop-down list.

29.14.11 Removing an animation sequence

1 Right-click the sequence.

2 Select **Delete** from the context menu.

As an alternative you can:

1 Left-click the sequence.

2 Press the Delete key.

29.14.12 Disabling or enabling a timeline

1 Check the timeline status icon. If the icon is , the timeline is disabled. If the icon is , the timeline is enabled.

2 Click the timeline status icon to toggle the status.

Animation sequences on disabled timelines are not included in the animation.

29.14.13 Locked or Unlocked a timeline

1 Check the lock timeline icon. If the icon is , the timeline is locked. If the icon is , the timeline is opened for changes.

2 Click the lock timeline icon to toggle the status.

When a timeline is locked, it will not be possible to toggle the timeline effects, to add a new effect to the timeline, to delete this timeline, to add/move/delete effects and the properties related to the timeline animated object will be disabled in the Configurator panel. This can be useful to avoid accidental changes to an already configured timeline.

29.14.14 Animating an exploded view

As the **Animation Editor** is not supported in Block editor or reference editing modes, only exploded views inserted in the model space can be animated.

1. Insert the exploded view in model space.
2. If applicable, suppress the exploded view and the model, if you want to hide or show them during the animation.
3. Select the exploded view object in the **Object selector**.
4. Click ⊕ to add a timeline.
5. Use the **Add Sequence** tool on the exploded view timeline to add a new exploded view sequence, or double-click at the timeline where you want to add a new sequence.
6. Click and drag to adjust the desired duration of the sequence, or use the **Configurator** to set the desired start time, end time or duration in seconds.
7. If applicable, use the **Configurator** to set the start step and the end step of the exploded view, or to enable the auto-hide option.

29.14.15 Rendering a video

See the section **Render**.

29.14.16 Saving animation graphics

Calculating of animation frames for large models with difficult parametrization can take a long time. In this case, you can record the graphics for all the frames to a zip-archive or a folder, in order to load it afterward and already playback calculated graphics again. Such a playback should work significantly faster than a simple recalculation for big models. Unlike 2D video, you can zoom, rotate the model in BricsCAD and change the visual style. Also, you will be able to render videos by the recorded graphics.

To save animation graphics:

1. Click on the toolbar.
2. The drop-down menu with two options will appear:
   - **Export graphics to zip-file**

     In this case, the recorded data will be compressed. But for large models, it can be long. Therefore, this option is convenient if you want to obtain a smaller file, for example, to send it to someone. However, for large models, even compressed files can take up several gigabytes.
     - After clicking this option, a file browser appears. Enter the new name and click **Save** to continue. If you click **Cancel**, the process will be interrupted, and graphics will not be recorded.
   - **Save graphics in folder**

     In this case, you obtain uncompressed data, without wasting time on compressing data (and decompressing this folder while loading). This is useful if you work with the model yourself on the same machine and you need to replay the animation scene for a large model several times.

This option can save time on playback and rendering video compared to recalculating frames in the original Animation Editor mode.

- After clicking this option, a file browser appears.

  Create the new folder and click **Select Folder** button to continue. If you click **Cancel**, the process will be interrupted, and graphics won't be recorded.
  If you select a folder which is not empty, a dialog box appears asking you to confirm to clean and overwrite.

You can choose in which state the graphics will be saved.

3 The progress bar will be displayed while recording.

4 After the end of the process, the animation graphics will be recorded to the selected zip-archive or folder.

29.14.17 Loading the recorded animation graphics and play it

You can load the graphics data recorded by the previous instruction for further playing it in the special mode of **Animation Editor**.

To load animation graphics:

1 Click on the toolbar.

2 The drop-down menu with two options will appear:

- **Load zip-file with graphics**

  Select this option to load the zip-archive with animation graphics, click the first option. After selecting the option, a file browser will open. Select the zip-archive or enter its name and click **Open** to continue. If you click **Cancel**, the process will be interrupted and **Animation Editor** will remain in the original mode.

- **Load graphics from folder**

  Select this option to load the folder with animation graphics. After clicking this option, a file browser will appear. Select the folder or enter its name and click **Select** to continue. If you click **Cancel**, the process will be interrupted and **Animation Editor** will remain in the original mode.

3 If you have chosen a zip archive, wait until the data is prepared. After loading, the special mode of **Animation Editor** is opened. This mode displays options that are identical to the existing options of the original mode.

4 Use the playback buttons to control playback. Click the **Play** button to start playing from the current cursor position. Click the **Pause** button to pause on the current position. Click the **Stop** button to stop playback and move cursor to the start position.

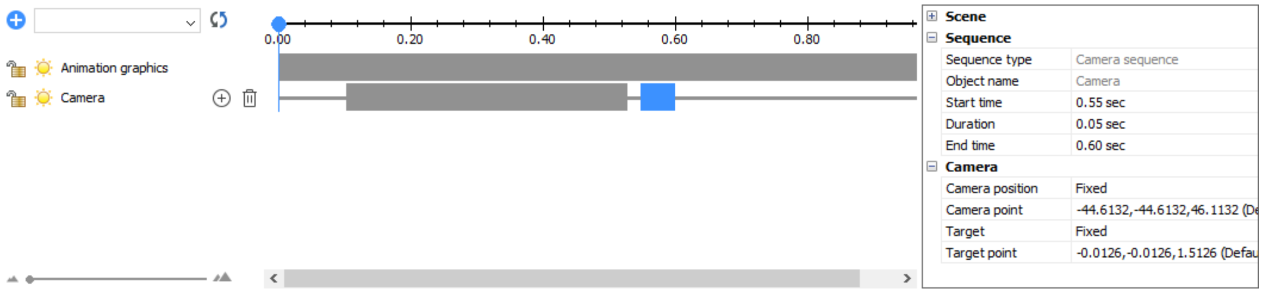
5 You can move the cursor along the scale to jump to the necessary frame. In this case, the nearest recorded frame will be displayed.

6 You can render video by pressing the **Render video** button like in the original mode. (See the **Render** section.)

Note:

- The frame rate in the render dialog is disabled because the graphics were recorded at a fixed frame rate, and you couldn't change it.
- Use the button to reload animation graphics according to the current cursor position if something was wrong.
- You should load animation graphics while the original document (from which the graphics was recorded) is active. Otherwise, the result may differ from the original.

7 Also, there is the possibility to change camera effects for Animation graphics by adding a new Camera timeline with new effects. In this case recorded camera effects will be replaced by the new ones.

8 

9 Press the button to close this mode and clear cache of animation graphics.

29.15 Mechanical blocks, XRefs and 3D solids

Mechanical blocks, XRefs and 3D solids are regular entities with assigned mechanical data (such as component name, physical material, BOM status or custom properties). Such block references, external references and 3D solids will be treated similarly to mechanical components. This means that those objects:

- Are shown in the mechanical browser.
- Can appear in bills of materials.
- Can have a specific physical material assigned to them.
- Are taken into account in the mass calculation.

As those objects are still regular entities, you can use virtually all BricsCAD commands to work with them. For example, you can use block editor to edit local blocks, reference editor to change a reference in place, or use BIM linear solids to model structural shapes while still being able to include them in the bill of materials. You can also make a mechanical block from a drawing and insert it as a block or as an external reference into another document, and still be able to see all internal components in the bill of materials or in the mechanical browser. Parametric blocks and parametric external references are also supported, as well as arrays of such mechanical objects. In short, you can make a mechanical object from any 3D solid or block or external drawing and continue to work with them the same way as with any regular entities using regular tools.

Mechanical blocks, XRefs and 3D solids are enabled by default. Some commands may not support or just partially support those blocks, XRefs or 3D solids. In addition, you can create hybrid assemblies, with both traditional mechanical components and new mechanical blocks, XRefs and 3D solids.

Commands and tools that fully support mechanical blocks, XRefs and 3D solids:

- **Mechanical browser**
- **Bill of materials** (**BMBOM** and **BOM manager**)
- **Mass properties** (**BMMASSPROP**)
- **Drawing views**
- **BMMECH**
- **BMUNMECH**
- **BMOPEN**
- **BMOPENCOPY**
- **BMDISSOLVE**
- **BMBALLOON**
- **BMEXPLODE**
- **BMEXPLODEMOVE**
- **SMUNFOLD** (for mechanical solids)
- **SMEXPORTOSM** (for mechanical solids)
- **BMLISPGET** (except options for component parameters)
- **BRX Components API**
- **BMPROPERTIES**
- **BMNEW**
- **BMFORM** (you can use BLOCK instead)
- **BMINSERT**
- **BMCONNECT**
- **SMASSEMBLYEXPORT**
- **BMREPLACE** (you can also use BLOCKREPLACE instead for blocks)
- **BMLOCALIZE** (you can also use XBIND instead for blocks)
- **BMEXTERNALIZE** (you can also use BLOCKTOXREF instead for blocks)
- **BMTRAILINGLINES**

Commands and tools that do not support mechanical blocks, XRefs and 3D solids:

- **BMCONVERT**
- **BMCREATECOMPONENT**
- **BMDEPENDENCIES**
- **BMHIDE**
- **BMSHOW**

- **BMRECOVER**
- **BMUPDATE**
- **CONVERTCOMPONENTSTOBLOCKS**

29.15.1 Enabling mechanical blocks, XRefs and 3D solids

- Open the **Settings** dialog box. Go to **Mechanical blocks** and tick the option **Enable mechanical blocks**
- In the Command line, set the MECHANICALBLOCKS system variable to 1.

29.15.2 Disabling mechanical blocks, XRefs and 3D solids

- Open the **Settings** dialog box. Go to **Mechanical blocks** and untick the option **Enable mechanical blocks**.
- In the Command line, set the MECHANICALBLOCKS system variable to 0.

After that, existing mechanical blocks will be ignored by all commands and tools in BricsCAD.

29.15.3 To create mechanical blocks, XRefs and 3D solids

You can make a mechanical block from the current drawing and convert all named block references, external references and 3D solids directly placed in the model space. If a local block is opened in block editor, you can make a mechanical block from it, optionally with all named block references, external references and solids.

1 Make sure that Mechanical blocks are enabled.

2 Call the BMMECH command. If it is called in the model space, enter the desired component name. By default, it will match the document name.

3 Select block references, external references and/or solids to be converted. You can also select **Yes** to convert all block references, external references and solids to mechanical objects, or Blocks to convert only block references and external references. If you select **No**, only the document (or the block being edited) will be converted into a mechanical block.

Note: All blocks where those mechanical blocks or entities are inserted will be converted to mechanical as well, and vice versa. This is to ensure that the document will have a valid mechanical structure.

29.15.4 Inserting mechanical blocks

To insert a mechanical block to another document, do the following:

1 Make sure that Mechanical blocks are enabled.

2 Use the BMINSERT or INSERT command to insert the block to the document.

To attach a document with a root mechanical block as an external reference, do the following:

1 Make sure that Mechanical blocks are enabled.

2 Use the XATTACH command to attach the document as an external reference.

Note: If the current document doesn't have a root component or a root mechanical block, a root mechanical block will be automatically created.

29.15.5 Removing mechanical structure

To remove a mechanical structure from a document, do the following:

1. Run the BMUNMECH command.
2. Choose **Yes** to confirm.

To remove a mechanical structure from a local named block, do the following:

1. Open the block using block editor (BEDIT).
2. Run the BMUNMECH command.
3. Choose **Yes** to confirm.

To remove mechanical data from a mechanical entity, do the following:

1. Select the entity in question in the mechanical browser.
2. Right click to call the context menu.
3. Choose **Unmech** from the context menu.

29.15.6 Assign properties to selected instances

Unlike traditional components, mechanical blocks, xrefs and solids allow to set properties for certain instances and also for the block itself. It means that, for example, you can assign individual physical materials to different inserts of the same mechanical block, or to different external references. If an instance doesn't have a certain property, the corresponding property of the block will be used instead.

To assign a property to a top-level instance of a mechanical block, or to a top-level external reference, do the following:

1. Select required instances in the mechanical browser.
2. Go to the **Properties** section.
3. Set the desired value of a corresponding property.

Note: Doing that, you will assign properties only to certain block references, external references or solids and not to their corresponding blocks or external documents (even if all inserts of those blocks are selected.) To set a property of a local block, open the block in block editor, select the root component in the browser and set the desired value of the property. To set a property for all external references of a document, open the document and change the corresponding properties of the root component.

29.15.7 Bill of materials and mechanical solids

Bills of materials in BricsCAD natively support mechanical blocks, XRefs and solids. However, unlike to traditional components, mechanical blocks, XRefs and solids are grouped according to their names and properties only. This means that if you have two different blocks with the same name in one assembly, those blocks will be grouped in the same row in a bill of materials, as long as their parameters and other properties (such as physical material) are identical. It is done intentionally, mainly for performance reasons, but also to provide more flexibility for users. It is especially important when working with mechanical solids. By default, each solid is considered as a unique component with an unique, automatically assigned name. If you copy a solid to the same document, the copy will get a new name and will be treated as a completely different solid in BricsCAD. Therefore, those solids will be in different rows of a bill of materials table. However, if you assign the same component name to them, bill of materials will group those solids together, regardless of their geometry. The same goes for mechanical blocks and external references.

Note: Since BricsCAD V23, it is allowed for mechanical blocks and solids to assign a BOM status for particular instances, which is respected by the bill of materials. In the mechanical browser, setting a BOM status for top-level instances is enabled. The corresponding command is modified to assign the status to the original block.

29.15.8 Copy mechanical solids

When a mechanical solid is copied, a different name is assigned to it by default. Therefore, it will be treated as a completely different solid. However, it is possible to set different instances and component names for mechanical solids. If a component name is set by the user, it will be kept in the copy, and the copy of this solid will be treated as an instance of the same component in the bill of materials.

30. Sheet Metal

30.1 Sheet Metal Design Methods

Sheet metal design allows you to model sheet metal parts and generate their unfolded representations with manufacturing information.

You can create complex sheet metal parts with BricsCAD easily and rapidly because the design process is different from a manufacturing process. Do not think in terms of a planar sheet that should be cut and bent, but model your part directly as you create solid bodies with Direct Modeling tools.

You can create sheet metal parts in BricsCAD in 4 different ways (or in any combination of them):

1 Create a base flange from a closed planar profile, then add flanges:
  - Pull its edges to create additional edge flanges with bends.
  - Create partial flanges with automatic creation of the corresponding bend reliefs.
  - Pull several adjacent edges to create several flanges at once with automatic creation of corner reliefs and junctions between them.
  - Bend a flange to create a new flange with a bend from its portion.

2 Create a combination of flanges and lofted bends connecting two 3D profiles.

  Use the Sheet Metal Loft tool.

3 Create a combination of flanges and bends from a planar profile.

  Use the Sheet Metal Extrude tool.

4 Create a shelled body from a 3D solid.

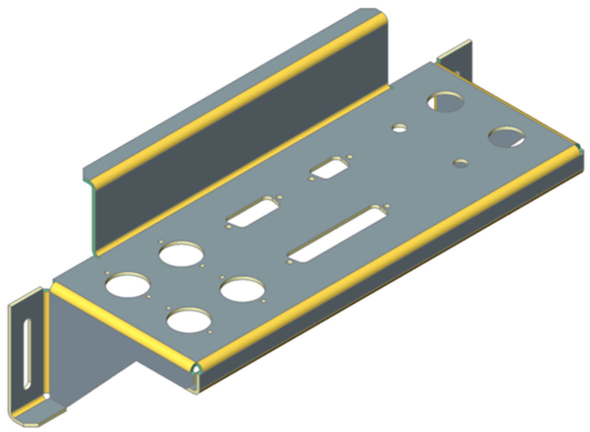
  Use the **Shell** option of the SOLIDEDIT command with the desired thickness, then convert it to a sheet metal part with the SMCONVERT command, and create bends and junctions from its hard edges.
  - Make holes by drawing closed profiles on faces of the flanges and pushing them through the flanges.
  - Adjust material thickness and bend radius using predefined parameters.
  - Apply Direct Modeling operations and 3D constraints to further adjust your design while always maintaining the design intent in terms of sheet metal features.
  - Automatically generate unfolded representation of your part and send it to a CAM system by exporting the corresponding 2D drawing with bend annotations in a .dwg or .dxf file.

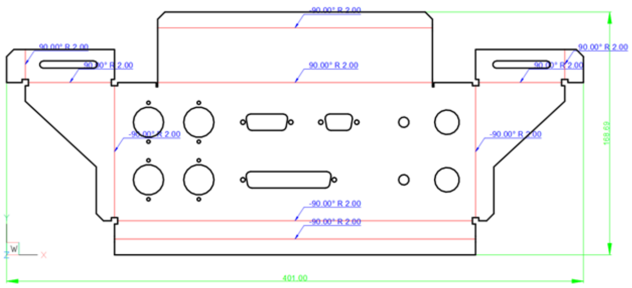
You can further detail your sheet metal part using the following operations:

- Make holes by drawing closed profiles on faces of the flanges and pushing them through the flanges.
- Insert Form Features from a library. A library of parametric form features is installed with BricsCAD Sheet Metal and you can create your own form features.
- Create bead features from a 2D curve lying on the flange by the SMBEAD command.
- Adjust material thickness and bend radius using predefined parameters.

- Apply Direct Modeling operations and 3D constraints to further adjust your design while always maintaining the design intent in terms of sheet metal features.

When your sheet metal part is ready, you can automatically unfold it and send the result to a manufacturing engineer by exporting the corresponding 2D drawing.





30.2 Sheet Metal Commands and Toolbars

Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
			Create a base (initial) flange of a sheet metal part from a closed planar profile.
			Create a sheet metal part by extruding a polyline in the direction of the normal of the curve plane. Thickness is assigned with respect to the external dimensions of the polyline.
			Create sheet metal part with lofted bends and flanges from 2 non-coplanar curves.

Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
			Automatically recognize flanges, bends and lofted bends in a 3D solid.
			Create one or more flanges on a sheet metal part by pulling one or more edges of an existing flange.
			Create a new flange from a closed contour and attach it to an existing sheet metal part.
			Change the reference side for selected flanges with an optional shift of the flange geometry by thickness.
			Rotate a flange.
			Create one or more closed hem features on a sheet metal part by pulling one or more edges of an existing flange.
			Create one or more open hem features on a sheet metal part by pulling one or more edges of an existing flange.
			Create one or more Teardrop hem features on a sheet metal part by pulling one or more edges of an existing flange.
			Create one or more Round hem features on a sheet metal part by pulling one or more edges of an existing flange.
			Insert form features.
			Create form feature from given set of faces.
			Explode form feature in flanges and bend if possible.
			Create an associative bead feature on a flange face from a planar 2D curve.

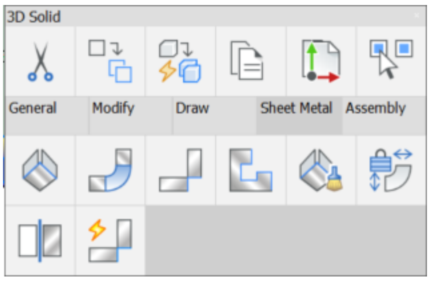
Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
			Replace form features (including recognized ones) in sheet metal parts with form features from a built-in or user library.
			Convert hard edges (sharp edges between flange faces) and junctions into bends.
			Convert bends to lofted bends.
			Bend an existing flange along a line, obeying k-factor for given bend radius.
			Convert hard edges into junctions.
			Change a symmetrical junction feature to one with overlapping faces.
			Close gaps between 2 arbitrarily oriented flanges.
			Create single tab feature to a sheet metal part by switching existing junction into the tab.
			Create array tab feature to a sheet metal part by switching existing junction into the tab.
			Make proper corner and bend reliefs. Corner reliefs are built on corners which have 3 or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.
			Convert a corner relief to a circular relief.
			Convert corner reliefs to a rectangular relief.
			Convert corner reliefs to a VType relief.
			Convert corner reliefs to a Smooth relief.
			Convert corner reliefs to a Round relief.

Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
			Convert corner reliefs to a Rip relief.
			Split a flange along a line drawn on its face.
			Split a flange by a single vertex in the corner.
			Split a flange by a single vertex in the corner and automatically suggest multiple splits of a similar type.
			Split thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.
			Remove sheet metal data from the selected faces or entities.
			Remove a bend or a junction by restoring a hard edge between two flanges; remove a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Remove miter by restoring geometry being cut by the feature.
			Fix problems specific to Sheet Metal parts, both global and local workflows are supported. Optionally heal lofted bend features: merge adjacent bends and provide tangential connection with flanges.
			Fix problems specific to Sheet Metal parts. Only selected flange or wrong bend are repaired.
		Same form features	Select all identical form features in the sheet metal model.
		Hard edges	Select all hard edges on sheet metal parts.
		Non-orthogonal thickness edges	Select all non-orthogonal thickness edges on the sheet metal part.

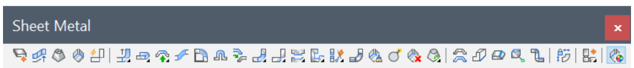
Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
		Flat edges	Select flat edges on the sheet metal part.
		side of the sheet metal part	Select side of the sheet metal part.
			Create a consistent set of 3D constraints for the selected sheet metal part.
			Unfold the sheet metal body.
		Associatively	Open an additional window with model's unfolded representation.
			Export a sheet metal solid to the .osm (Open Sheet Metal) file format.
			Export the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.
			Perform a batch export of assembly components to unfolding, creating a .dxf per each 3D solid recognized as sheet metal part.
			Toggle the *FEATURECOLORS* system variable.

The Sheet Metal tools are available:

- In the **Quad** cursor menu (depending on the feature under the cursor):



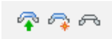
- On the **Sheet Metal** toolbars:
  - **Sheet Metal**



- SM Bend

- SM Flange

- SM Form



- SM Hem

- SM Junction

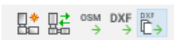
- SM Relief

- SM Repair

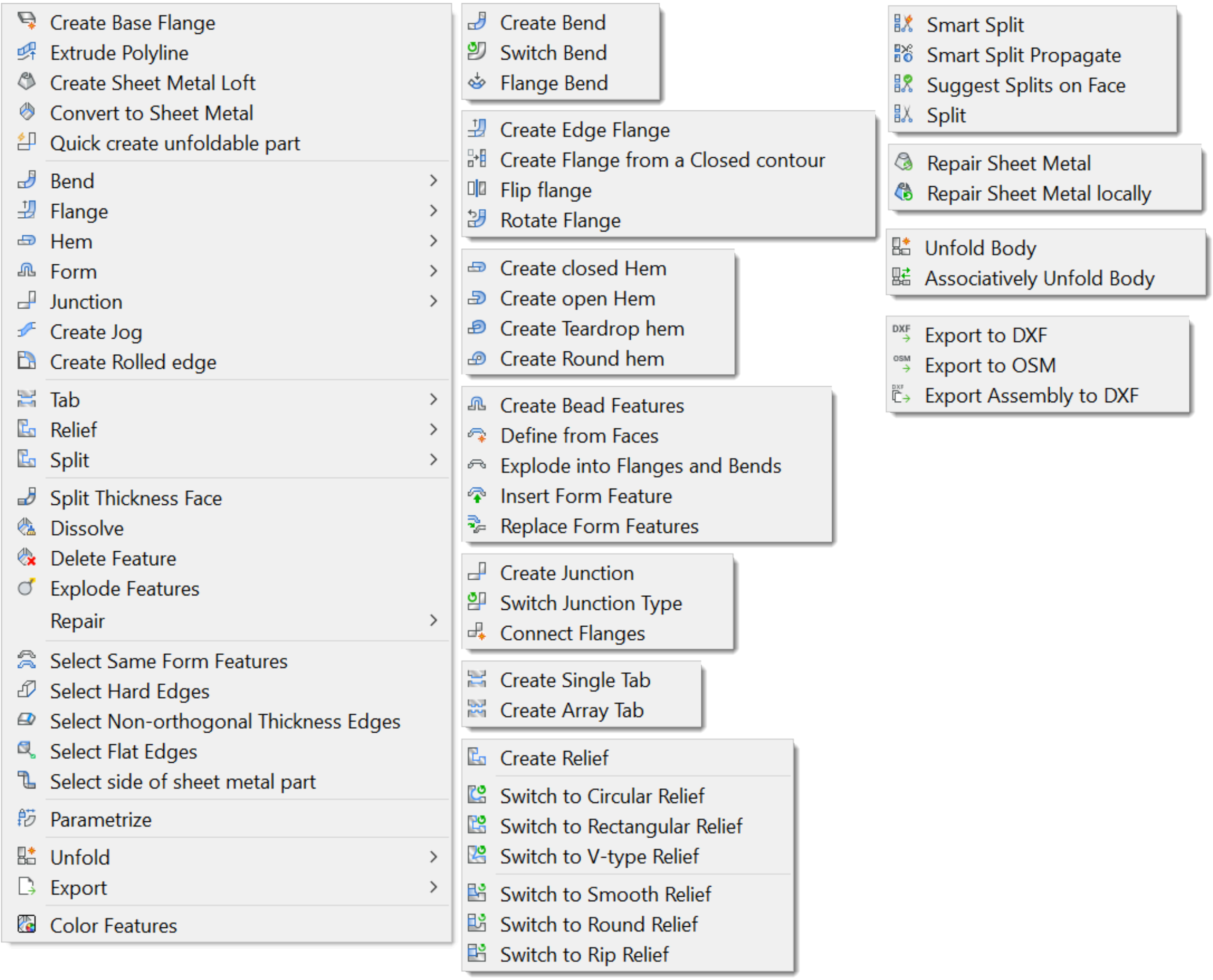
- SM Split

- SM Tab

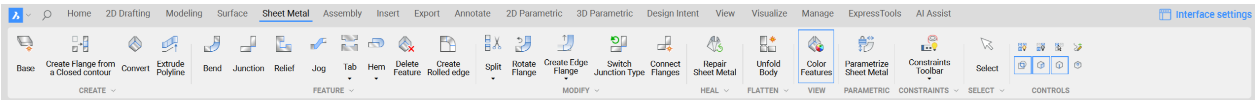
- SM Unfold



- In the **Sheet Metal** menu (**Mechanical** workspace only):



• In the **Sheet Metal** tab of the **Mechanical** workspace ribbon.



See also Direct modeling operations.

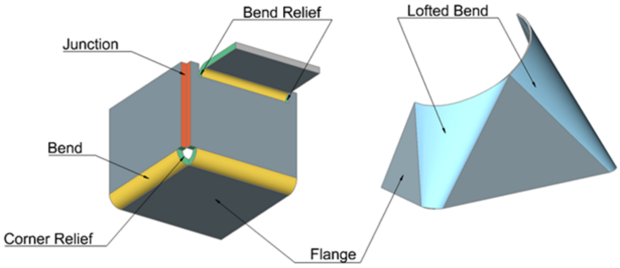
30.3 Sheet Metal Features

30.3.1 Overview

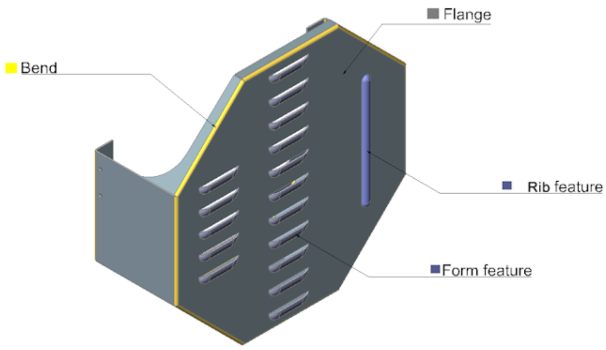
Features are smart groups of faces in a 3D part. Each feature maintains a specific spatial and parametric relationship between its faces and some adjacent faces. The features allow you to embed design intent into your model. The features are created automatically by a particular geometric operation you apply.

Images below show the sheet metal features in BricsCAD:

Example 1 of sheet metal features



Example 2 of sheet metal features



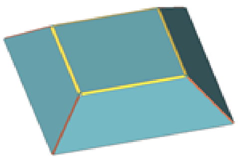
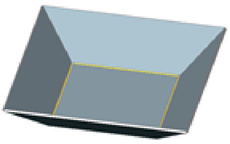
30.3.2 Flange

The main feature of all sheet metal parts is a flange, which consists of 2 parallel planar faces located in front of each other (with some solid volume between them) on the distance, which is significantly smaller than the linear size of the faces. This distance is equal to the material thickness. When you modify the model, this distance relation is always maintained automatically. Other faces, which are adjacent to flange faces and do not belong to bends are called thickness faces. They are always perpendicular to the flange faces.

Flange features are created by the following commands: SMFLANGEBASE, SMEXTRUDE, SMLOFT, SMFLANGEEDGE, SMFLANGEBEND and SMCONVERT.

If the FEATURECOLORS system variable is ON, the display color distinguishes between the reference face of a flange and its opposite side. Parametric modifications of a sheet metal part, such as thickness change, do not move reference faces, if possible. The SMFLIP command swaps sides of a selected flange so that reference faces are on the other geometric side of the flange; optionally the command shifts the flange over the thickness of the sheet metal part.

The images below illustrate the color legend for the flange faces of the same part.

Reference Faces	Other Faces

30.3.3 Bend

Two flanges are connected by a bend. A bend consists of 2 coaxial cylindrical faces, which are always tangent to the adjacent planar faces of the flanges.

Bend features are created by SMFLANGEBASE, SMEXTRUDE, SMFLANGEEDGE, SMFLANGEBEND and SMCONVERT commands.

30.3.4 Zero Bend

This is a design concept of bend, but geometry has a cylindrical face for the external side of the bend with a radius equal to thickness. Instead of an internal face, there is an edge. So effectively geometry represents a bend of zero radii, which is used for design purposes. SMUNFOLD operation supports bends of this kind and generates dimensions as if these bends were regular bends with a radius equal to thickness.

30.3.5 Wrong Bend

Wrong bend features are recognized by the SMCONVERT command and reflect assumptions on the user's design intent. Like regular bends, wrong bends connect flanges, but their faces can be non-coaxial, or they can have non-tangent connections with adjacent flange faces, which prevent using the geometry as regular bend features. Before unfolding the part, all wrong bends must either be repaired by SMREPAIR or removed by SMDELETE and then recreated by SMBENDCREATE.

30.3.6 Lofted Bend

A lofted bend feature consists of 2 ruled surfaces located in front of each other (on the offset equal to the material thickness) with some solid volume between them. A ruled surface can be described as the set of points swept by a moving straight line. Examples are cylinders, cones, helicoids. Lofted bends can be created by the thickening of ruled surfaces. A lofted bend can connect 1 or 2 flanges or be stand-alone, without flanges at all. Some combinations of adjacency between regular bends and lofted bends are also allowed.

Differences between bends and lofted bends:

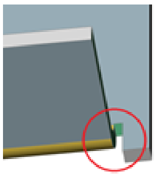
	Bends	Lofted Bends
Surface	Cylindrical	Cylindrical, conical or spline

	Bends	Lofted Bends
Unfolding	Exactly	The approximate unfolding of lofted bends is controlled by an adjustable number of bend operations, which are called the Number of Subdivisions. The more operations are performed, the smoother the real part will be produced and the better it approximates the initial design. Cylindrical and conical lofted bends have an option for exact analytical unfolding.

Lofted bend features are created by the SMLOFT and SMCONVERT commands.

30.3.7 Bend Relief

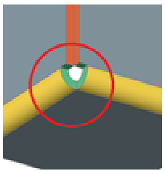
A group of faces representing a technological cut between 2 flanges of different width connected with a bend. The bend relief feature maintains the distance between 2 opposite faces of the cut.



Bend relief features are created by SMFLANGEEDGE, SMBENDCREATE and SMRELIEFCREATE commands.

30.3.8 Corner Relief

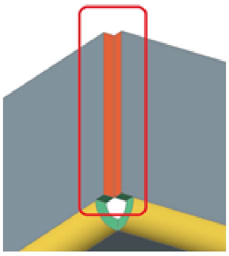
A group of faces representing a cut in the corner where 3 or more flanges meet together. This cut is needed to prevent unwanted material deformation during bending of the sheet metal part. A corner relief feature maintains the form and size of this cut.



Corner relief features are created by SMFLANGEEDGE, SMBENDCREATE and SMRELIEFCREATE commands.

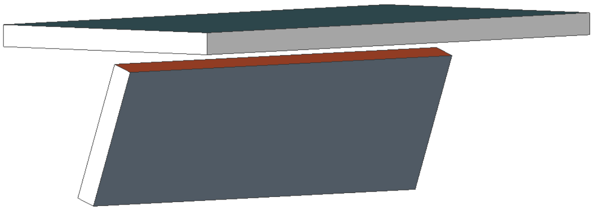
30.3.9 Junction and T-junction

A junction feature consists of 2 thickness faces of adjacent flanges, which are not connected by a bend. A junction can be symmetric or overlapping. Junction features maintain the minimal distance between the flanges: the junction gap.



A T-junction feature consists of a thickness face of a single flange, which is located near a face of another flange,. Usually, a T-junction precedes creation of a T-tab.

For clarity, the junction gap in the image below is intentionally rather large.

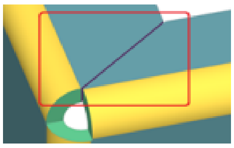


Junction features are created by SMFLANGEEDGE, SMJUNCTIONCREATE and SMFLANGECONNECT commands.

T-junction features are created by SMJUNCTIONCREATE and SMFLANGECONNECT commands.

30.3.10 Miter Feature

A miter feature represents a split between coplanar flanges created by the SMFLANGEEDGE and SMSPLIT commands. A miter feature maintains the miter gap.



30.3.11 Form and BEAD Features

Form and bead features represent a rich variety of shapes which are produced by punching and wheeling.

Two representations of the form features are important: one in 3D and a 2D symbol in the unfolded geometry. In case of imported geometry the symbol is obtained by the projection of the 3D faces of the form feature. In case the form feature was inserted/replaced from the Form Feature Library, the unfolding symbol is taken from the library file.

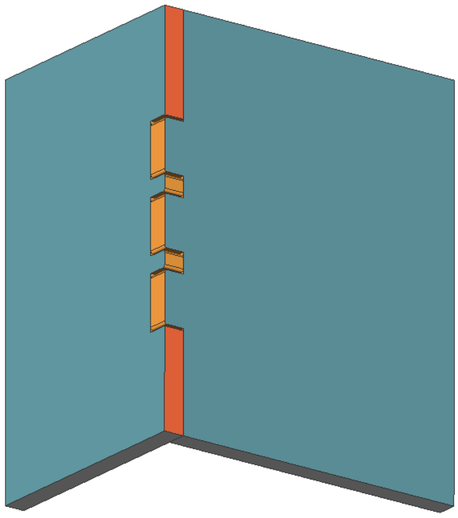
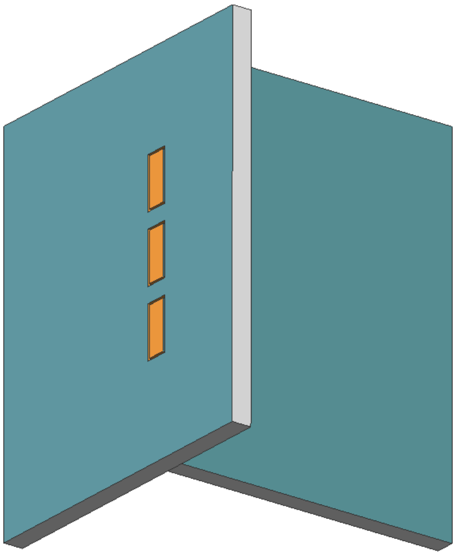
Bead features represent the imprint on a flange made by a wheel following an arbitrary trajectory.

Form and bead features are created by BMINSERT, SMBEAD and SMCONVERT commands.

30.3.12 Tab and T-tab

Both features are used to connect 2 flanges, restricting their mutual displacement.

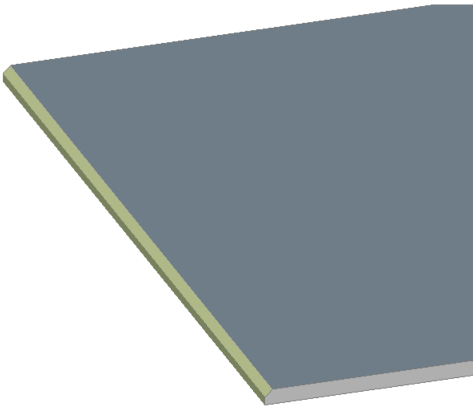
Tab and T-tab features are created by the SMTABCREATE command.

	
Tab feature	T-tab feature

30.3.13  Bevel

Bevel features represent chamfers on thickness faces of Sheet Metal parts. Both sloped and orthogonal faces define a chamfer feature. Bevel features must be shared strictly by a single flange and can not be adjacent to bend features.

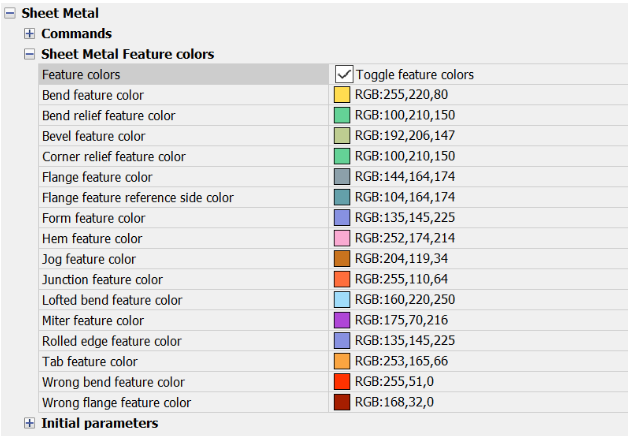
Bevel features can be recognized or erased by the SMCONVERT command, and are supported by the SMUNFOLD command.



30.4 Feature Coloring

For better experience and easy distinguishing between the features, different colors are applied to the various types of features in the geometry. These colors are for visualization only and will never show in generated documents, paper space views, block inserts, etc.

The display of feature colors is controlled through the *FEATURECOLORS* system variable and a series of system variables, one for each feature color. Although the default feature colors are selected carefully to combine with each other, you can set them differently in the **Sheet Metal** settings group on the *Settings* dialog box.



Sheet Metal
Commands
Sheet Metal Feature colors

Feature colors	Toggle feature colors
Bend feature color	RGB:255,220,80
Bend relief feature color	RGB:100,210,150
Bevel feature color	RGB:192,206,147
Corner relief feature color	RGB:100,210,150
Flange feature color	RGB:144,164,174
Flange feature reference side color	RGB:104,164,174
Form feature color	RGB:135,145,225
Hem feature color	RGB:252,174,214
Jog feature color	RGB:204,119,34
Junction feature color	RGB:255,110,64
Lofted bend feature color	RGB:160,220,250
Miter feature color	RGB:175,70,216
Rolled edge feature color	RGB:135,145,225
Tab feature color	RGB:253,165,66
Wrong bend feature color	RGB:255,51,0
Wrong flange feature color	RGB:168,32,0

Initial parameters

To toggle the display of feature colors:

- Click the **Color Features** tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Color Features** tool button ( ) on the **Sheet Metal** ribbon bar.
- Choose **Color Features** in the **Sheet Metal** menu.

30.5 Mechanical Browser for Sheet Metal

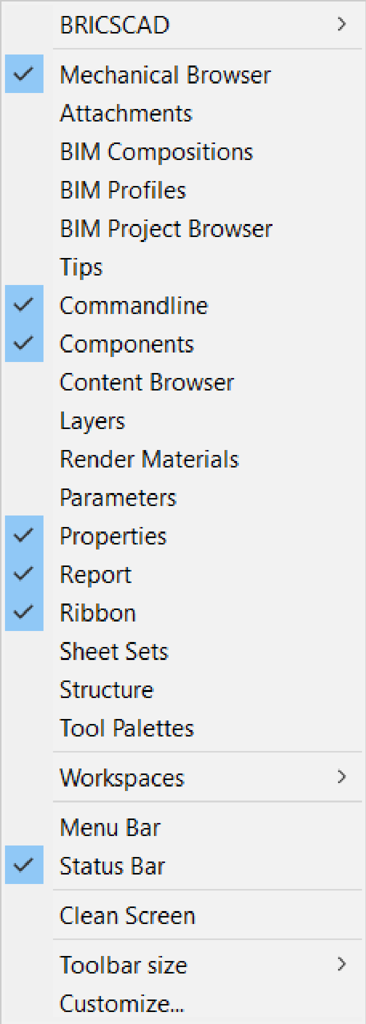
30.5.1 Overview

The **Mechanical Browser** panel opens automatically when switching to the **Mechanical** workspace.

To open the **Mechanical Browser** panel manually, do the following:

1 Move the cursor over a toolbar, then right-click.

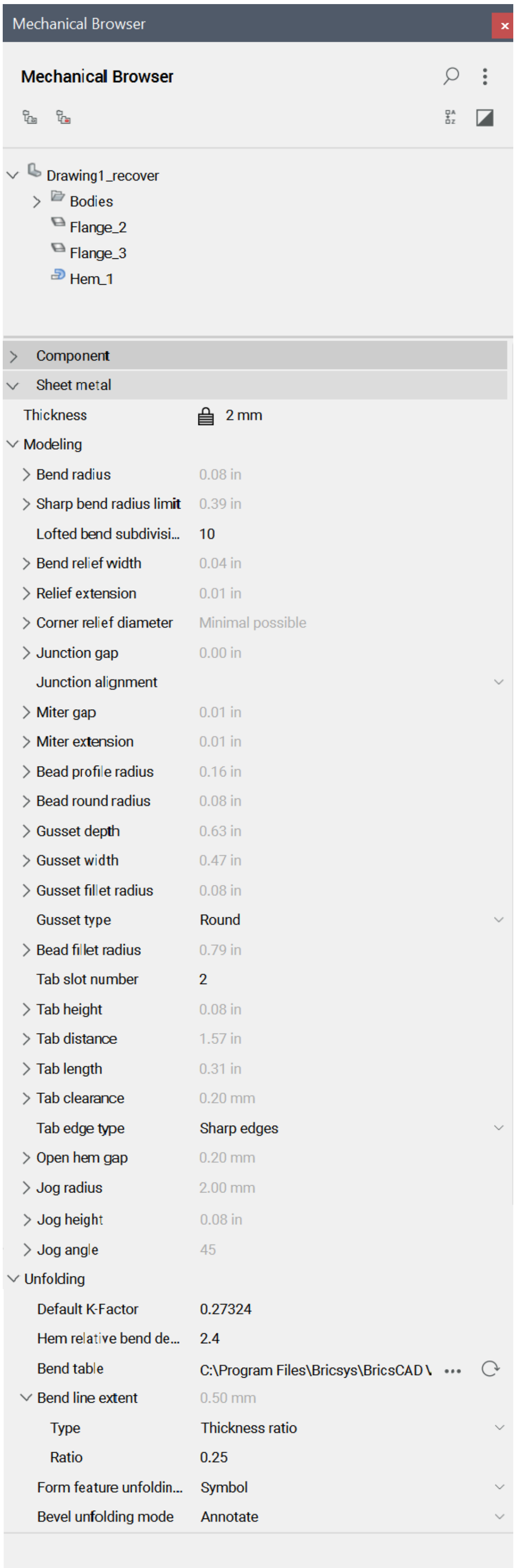
1 A context menu displays:



2 Choose **Mechanical Browser**.

30.5.2 Sheet Metal Context

If you work with a sheet metal part (if you created any sheet metal feature), properties of the root node in **Mechanical Browser** contain **Sheet Metal** section (also called Sheet Metal Context associated with the document).


Mechanical Browser
Mechanical Browser
Drawing1_recover
Bodies
Flange_2
Flange_3
Hem_1
Component
Sheet metal
Thickness 2 mm
Modeling
Bend radius 0.08 in
Sharp bend radius limit 0.39 in
Lofted bend subdivisi... 10
Bend relief width 0.04 in
Relief extension 0.01 in
Corner relief diameter Minimal possible
Junction gap 0.00 in
Junction alignment
Miter gap 0.01 in
Miter extension 0.01 in
Bead profile radius 0.16 in
Bead round radius 0.08 in
Gusset depth 0.63 in
Gusset width 0.47 in
Gusset fillet radius 0.08 in
Gusset type Round
Bead fillet radius 0.79 in
Tab slot number 2
Tab height 0.08 in
Tab distance 1.57 in
Tab length 0.31 in
Tab clearance 0.20 mm
Tab edge type Sharp edges
Open hem gap 0.20 mm
Jog radius 2.00 mm
Jog height 0.08 in
Jog angle 45
Unfolding
Default K-Factor 0.27324
Hem relative bend de... 2.4
Bend table C:\Program Files\Bricsys\BricsCAD\
Bend line extent 0.50 mm
Type Thickness ratio
Ratio 0.25
Form feature unfoldin... Symbol
Bevel unfolding mode Annotate

Sheet Metal properties are represented as grouped fields.

The **Thickness** field controls the material thickness of sheet metal solids.

The Modeling group plays 2 roles:

- It provides default settings for Sheet Metal Commands, such as SMFLANGEBASE, SMEXTRUDE, SMFLANGEEDGE, SMFLANGEBEND, SMLOFT, and SMCONVERT in a newly created model. The initial parameters are saved in the drawing, which allows creating a set of .dwt template drawings for each type of sheet metal models created in a company.
- New features created by the commands usually derive their properties from Sheet Metal Context. For example, the default value of **Radius Type** property for the bend feature is "Use global value". Thus when you change **Bend radius** in Sheet Metal Context, all the bend features with the derived property will change its radius. So the second application of Sheet Metal Context properties is that they allow globally change properties of sheet metal part.

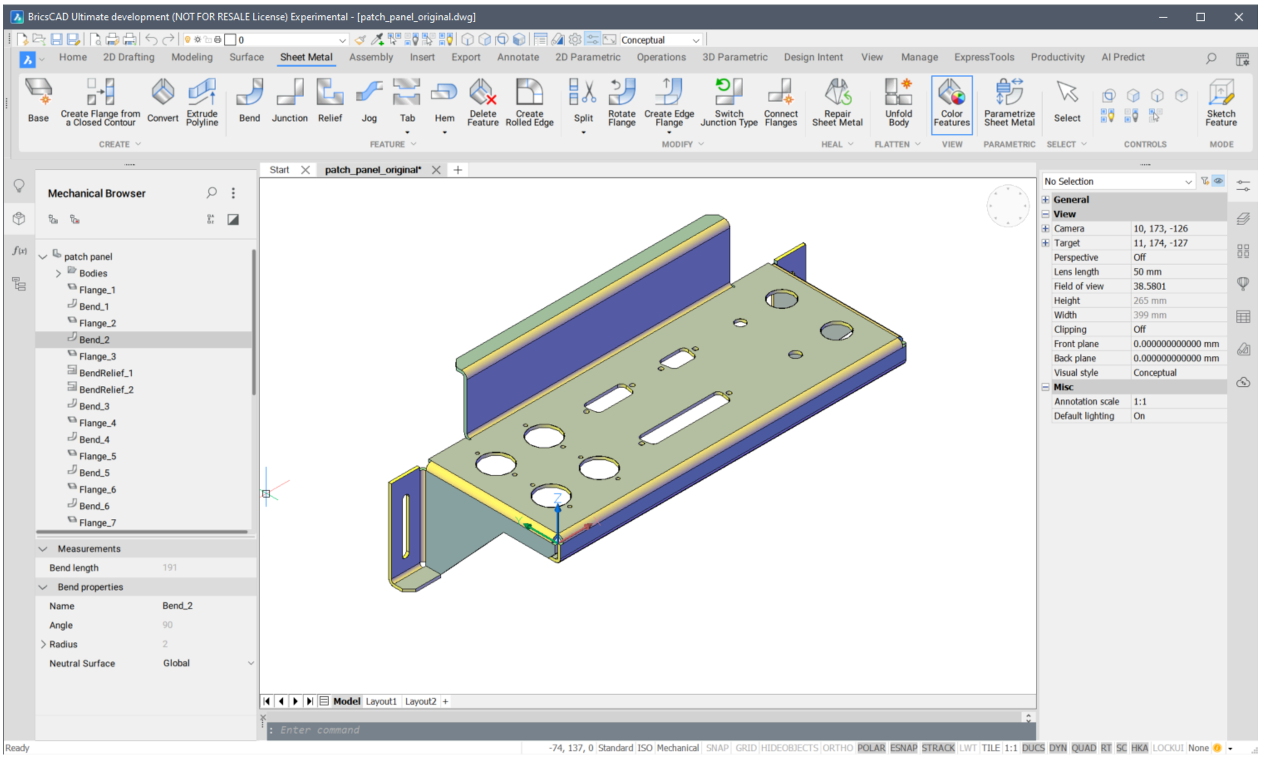
The **Unfolding** group of properties in Sheet Metal Context lists some properties of SMUNFOLD, SMEXPORT2D and SMASSEMBLYEXPORT commands.

Sheet Metal Context is created when first sheet metal part appears in the document and is initialized from settings listed in Initial Parameters. Typical workflows for creating sheet metal part are listed in Sheet Metal Design Methods. Sheet Metal Context is removed when all the features in the document are dissolved by the *SMDISSOLVE* command.

Note: If you just select all sheet metal parts in the document and Erase them, Sheet Metal Context is kept, despite there are no sheet metal solids and no sheet metal features.

30.5.3 Working with features

All features of a sheet metal part are listed in the **Mechanical Browser**. When you select a feature in the **Mechanical Browser**, its faces are highlighted in the model area:



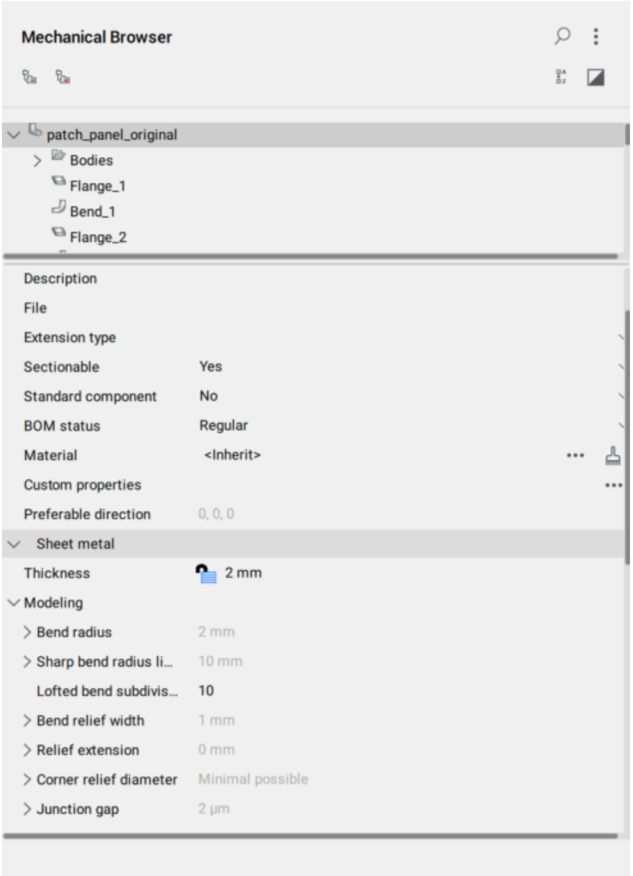
Note: Selecting features in the **Mechanical Browser** allows you to:

- Locate a feature on the part by highlighting its geometry.
- See the properties of given feature. For example, if you select a bend feature, **Bend properties** is displayed in the **Mechanical Browser**.

30.5.4 Changing the thickness

To change the thickness of a sheet metal part:

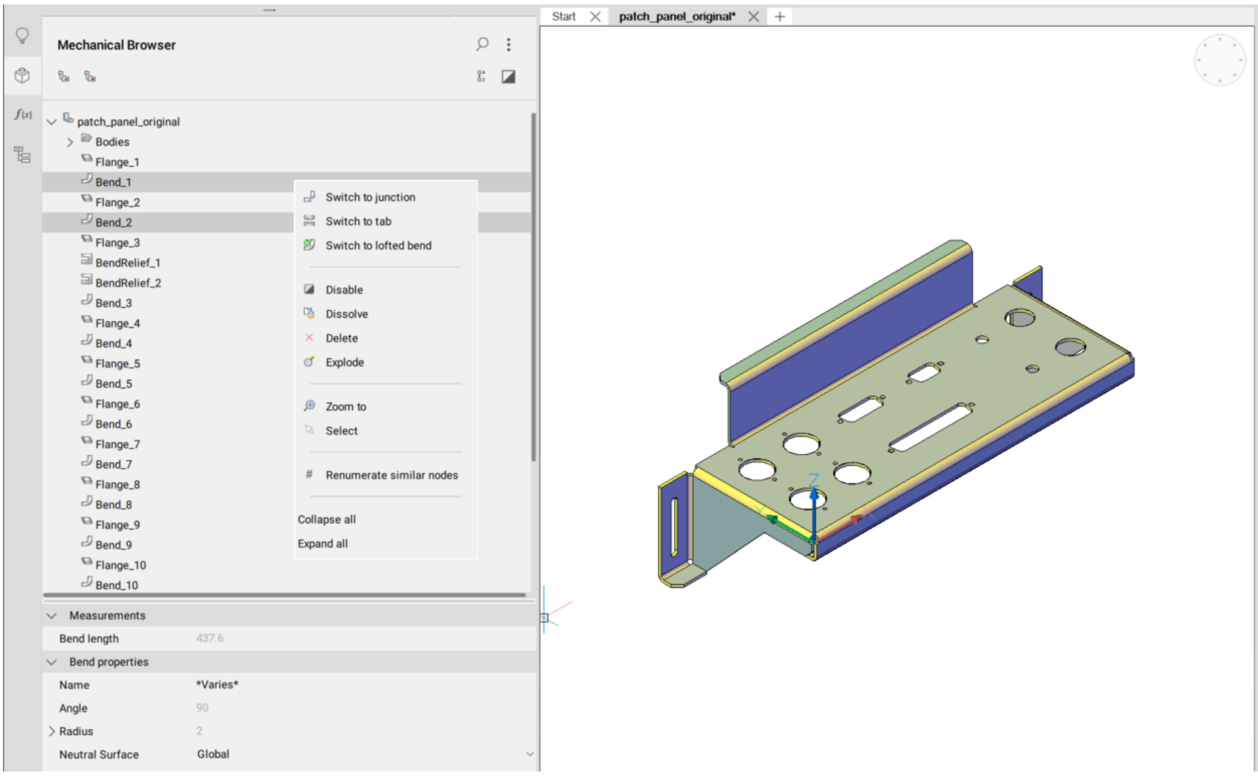
1. Select the root node in the **Mechanical Browser**.
2. Type a value in the **Thickness** field.



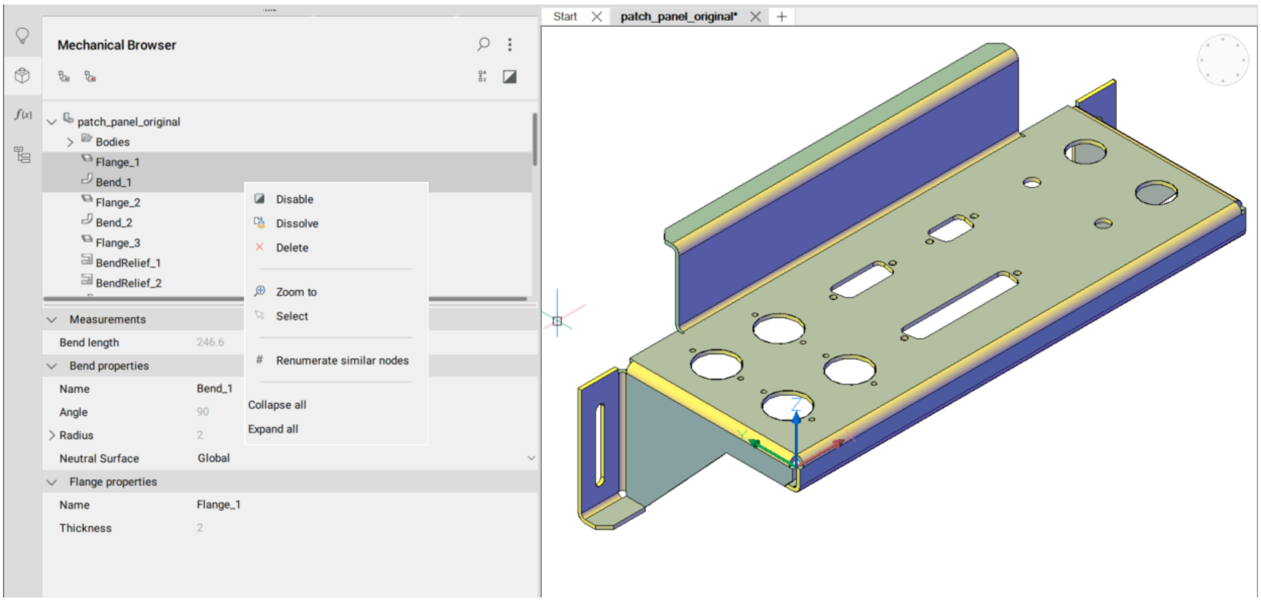
30.5.5 Feature operations

Mechanical Browser panel provides fast access to operations over given sheet metal feature, list depends upon the feature type. List of operations depends upon selected features and contains operations available for the selection.

Selecting 2 bends results in operations for bend feature:



Selecting flange and bend limits list to operations available for both features:



30.5.6 Understanding the difference between Disable, Dissolve and Delete

When you right-click a feature in the **Mechanical Browser**, then select **Dissolve** from the context menu the selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

If your intent is temporary forget about given feature, you can choose **Disable** option. The feature will remain in the **Mechanical Browser**, but the part will behave like the feature was dissolved. It is beneficial in contrast with dissolve, that feature still updates on geometry changes and you can turn it on when you want, avoiding recreating it or recognizing by the SMCONVERT command.

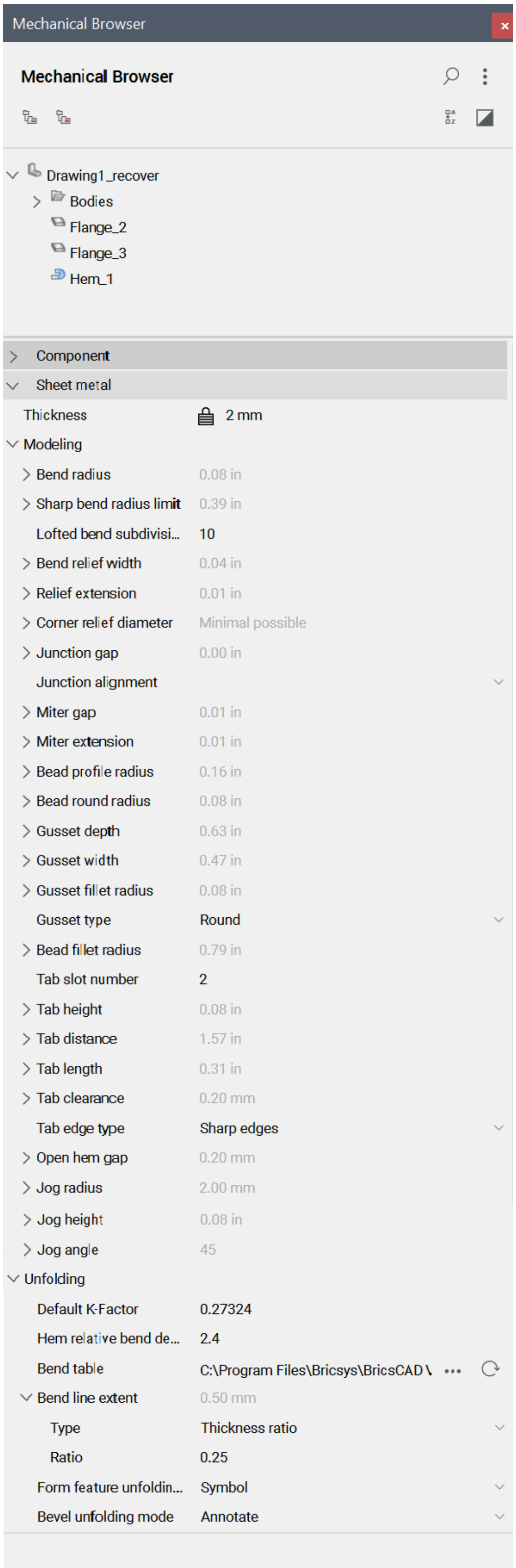
Some of Sheet Metal features support **Delete** operation in the **Mechanical Browser**, which is the analogue of the SMDELETE command. In this case the feature is removed from the browser and geometry is changed depending upon the type of the feature.

30.6 Sheet Metal Global and Feature Properties

30.6.1 Overview

Sheet metal features and context menus are located in the **Mechanical Browser** panel. See **Mechanical Browser for Sheet Metal** to understand the general principles. This article describes the particular properties and context menus for each node type in the **Mechanical Browser** panel.

When working with sheet metal parts, clicking the root node in **Mechanical Browser** panel, displays the properties of entire drawing. Sometimes these properties are referred to as the global Sheet Metal Context associated with the drawing: each Sheet Metal part shares this context by default.


Mechanical Browser
Mechanical Browser
Drawing1_recover
Bodies
Flange_2
Flange_3
Hem_1
Component
Sheet metal
Thickness 2 mm
Modeling
Bend radius 0.08 in
Sharp bend radius limit 0.39 in
Lofted bend subdivisi... 10
Bend relief width 0.04 in
Relief extension 0.01 in
Corner relief diameter Minimal possible
Junction gap 0.00 in
Junction alignment
Miter gap 0.01 in
Miter extension 0.01 in
Bead profile radius 0.16 in
Bead round radius 0.08 in
Gusset depth 0.63 in
Gusset width 0.47 in
Gusset fillet radius 0.08 in
Gusset type Round
Bead fillet radius 0.79 in
Tab slot number 2
Tab height 0.08 in
Tab distance 1.57 in
Tab length 0.31 in
Tab clearance 0.20 mm
Tab edge type Sharp edges
Open hem gap 0.20 mm
Jog radius 2.00 mm
Jog height 0.08 in
Jog angle 45
Unfolding
Default K-Factor 0.27324
Hem relative bend de... 2.4
Bend table C:\Program Files\Bricsys\BricsCAD \
Bend line extent 0.50 mm
Type Thickness ratio
Ratio 0.25
Form feature unfoldin... Symbol
Bevel unfolding mode Annotate

You can assign a local Sheet Metal context for given solid, which allows to define Sheet Metal parts with different design parameters, and in addition, to change the part thickness. Before BricsCAD V21, all the parts in modelspace had to have the same thickness, which forced the user to spread the parts with a different thickness to different blocks or components. Scenarios with multiple imported bodies engaging SMCONVERT and SMASSEMBLYEXPORT also benefit from this new feature. Another advantage is that the local Sheet Metal context can define its own Bend Table.

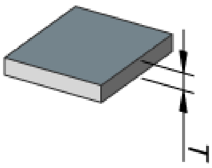
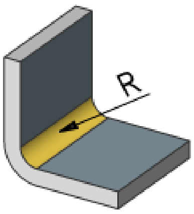
30.6.2 Switching the context of a solid to local

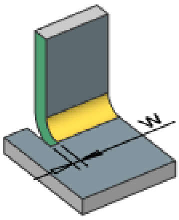
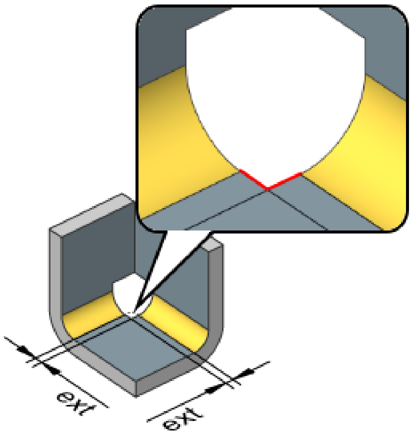
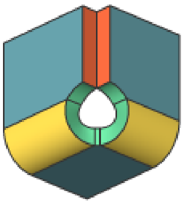
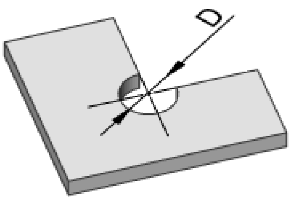
Do the following:

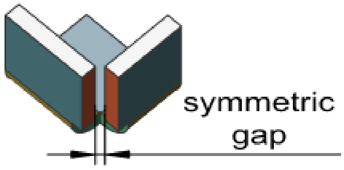
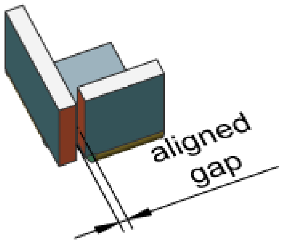
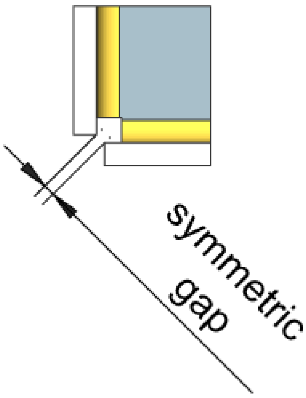
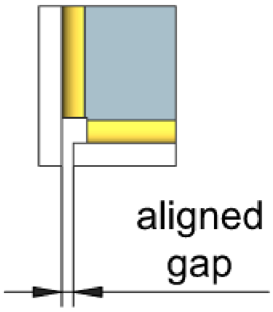
1. Select the body in **Mechanical Browser** panel.
2. In Sheet Metal section, switch **Context Type** from **Global context** to **Context for solid**.

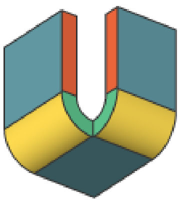
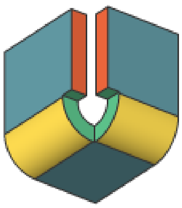
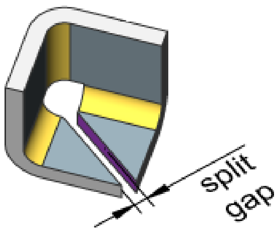
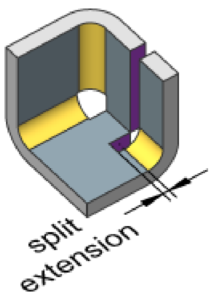
Note: The SMCONVERT command, in case of multiple input with different thicknesses, switches some contexts to local automatically.

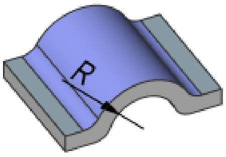
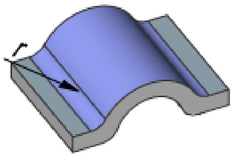
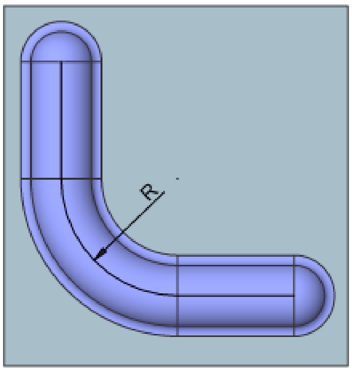
Sheet metal global properties are located in the bottom part of the **Mechanical Browser**.

Global Properties	Description
Thickness	Global sheet metal material thickness. 
Bend radius	Default radius of bend features; it can be changed per bend as well. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Sharp bend radius limit	Maximum bend radius to be considered a sharp (regular) bend. Used in the SMCONVERT and SMRELIEFCREATE commands to distinguish between regular and lofted bends. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value.
Lofted bend subdivisions	Sets the default number of subdivisions for unfolded lofted bends.

Global Properties	Description
Bend relief width	Width of the cut near a single bend. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Relief extension	Size of the bend or corner relief extension. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Corner relief diameter	Default diameter of a circular corner relief. **Type**: minimal possible or absolute value.  

Global Properties	Description
Junction gap	Gap between 2 adjacent flanges. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value.    

Global Properties	Description
Junction alignment	Control the alignment of junction faces with the corresponding corner relief faces. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. • **Junction Alignment**: enabled or disabled **Enabled**  **Disabled** 
Miter gap	Gap between the thickness faces of a flange or the adjacent bend when splitting, see SMSPLIT. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Miter extension	Control the depth cut of an adjacent flange in case of a flange split. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 

Global Properties	Description
Bead profile radius	Radius of a tool, which creates the bead. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Bead round radius	Radius of the fillet between the flange and a bead. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Bead fillet radius	Radius of the bead control curve filleting. See Working with Form and Bead Features. • **Type**: either profile radius ratio or absolute value. • **Value**: numeric value. 
Tab slot number	Defines the tab slot number.
Tab height	Defines the tab height. • **Type**: either thickness ratio or absolute value. • **Value**: numeric value.
Tab distance	Defines the tab distance. • **Type**: either thickness ratio or absolute value. • **Value**: numeric value.

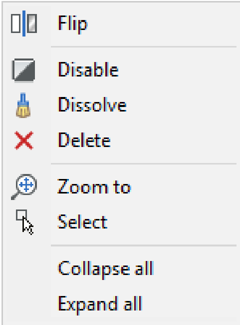
Global Properties	Description
Tab length	Defines the tab length. • **Type**: either thickness ratio or absolute value. • **Value**: numeric value.
Tab clearance	Defines the tab clearance. • **Type**: either thickness ratio or absolute value. • **Value**: numeric value.
Tab edge type	Controls the tab edge type. • **Sharp edges**: tab edges will be sharp. • **Round edges**: tab edges will be round. Have to define the **Tab round radius** type and value. • **Chamfer edges**: tab edges will be chamfered. Have to define the **Tab chamfer distance** type and value.
Open hem gap	Defines the hem gap. • **Type**: either thickness ratio or absolute value. • **Value**: numeric value.
Jog radius	Defines the jog radius. • **Type**: either thickness ratio or absolute value. • **Value**: numeric value.
Jog height	Defines the jog height. • **Type**: either thickness ratio or absolute value. • **Value**: numeric value.
Jog angle	Defines the jog angle value.
Unfolding	Parameters for the unfolded representation.
Default K-Factor	K-Factor for the bend radius equal to the thickness.
Hem relative bend deduction	Defines the bend deduction relative to thickness used in hem features unfolding.
Bend table	Display the Open dialog for bend tables in CSV file format.
Bend line extent	Control how far a bend line extends beyond the unfolding boundary (zero and negative values are allowed). • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value.

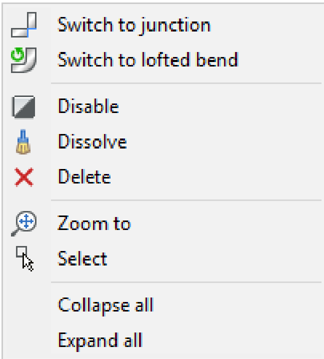
Global Properties	Description
Form feature unfolding mode	Control the appearance of form features in the unfolded representation. • **Keep**: keep the 3D representation of the form feature. • **Remove**: completely remove form features from the unfolded representation. • **Project**: project all edges of form features as 2D entities. • **Contour**: project only the contour of form features as 2D entities. • **Symbol**: place 2D symbols at the insertion points in unfolded representation. • **Project without center mark**: project all edges of form features as 2D entities without center mark. • **Contour without center mark**: project only the contour of form features as 2D entities without center mark.
Bevel unfolding mode	Control the bevel unfolding mode (see **Working with Bevels** article) • **Keep**: keeps all bevel features as they are. • **Remove**: all bevel features are removed. • **Annotate**: each bevel feature is removed and the edge is annotated. This mode is recommended.

Feature properties display automatically when selecting a feature in the **Mechanical Browser** panel.

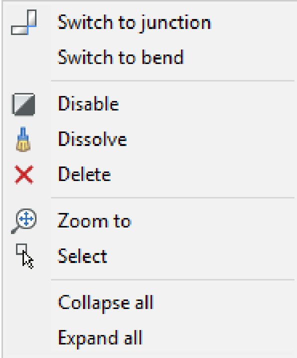
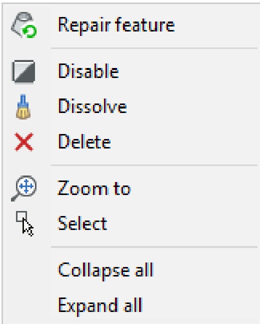
Right-click a feature to display the feature context menu.

Feature Context Menus	Description
	Common tools: • **Disable**: temporarily disable a feature and skip its constraining. • **Dissolve**: remove a feature but keep its geometry. Design intent associated with the geometry of a dissolved feature will be removed. • **Delete**: See the SMDELETE command. Remove features from the Mechanical browser and delete the corresponding geometry in the model. • **Zoom to**: zoom onto the selected feature. • **Collapse all**: collapse the main component including all components and subcomponents. • **Expand all**: expand the main component and all components and subcomponents.

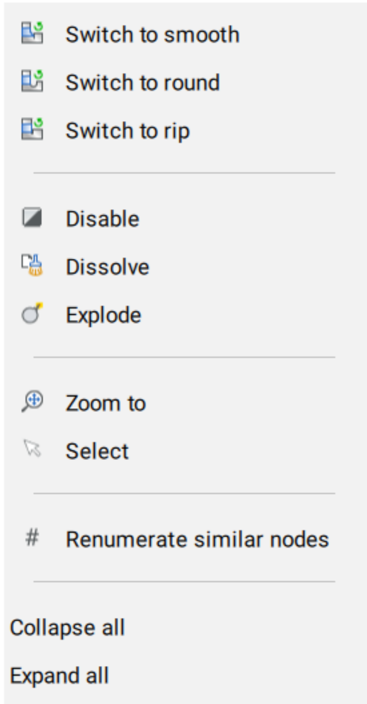
<table>
<tr><th>Feature Context Menus</th><th>Description</th></tr>
<tr><td>Flange feature</td><td>Right-click a flange feature:


Flip: swap the sides of a selected flange so that the reference face is on the opposite geometric side of the flange. This operation shifts the flange by its thickness, to preserve the position of the reference face.</td></tr>
<tr><td>Bend feature</td><td>Right-click on a bend feature:


Switch to junction: switch to a junction feature type with a gap between adjacent flanges.
Switch to tab: switch to tab feature.
Switch to lofted bend: switch to a lofted bend feature.
</td></tr>
</table>

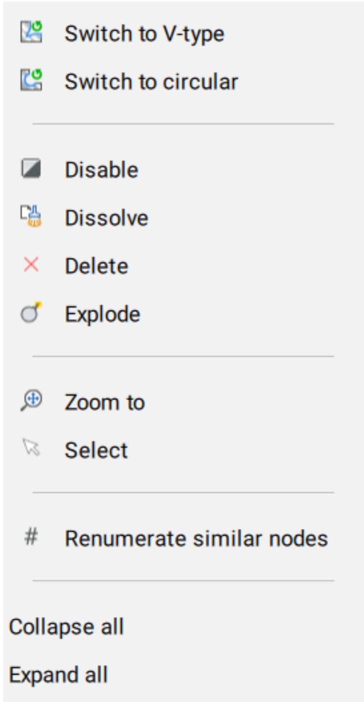
Feature Context Menus	Description
Lofted Bend feature	Right-click on a lofted bend feature:  • **Switch to junction**: switch to a junction feature type with a gap between adjacent flanges. • **Switch to bend**: switch to a bend feature.
Wrong Bend feature	Right-click on a wrong bend feature:  **Repair feature**: repair a wrong bend feature by creating a bend feature with proper geometry, if possible.

<table>
<tr><th>Feature Context Menus</th><th>Description</th></tr>
<tr><td>Bend Relief feature</td><td>Right click on bend relief feature:

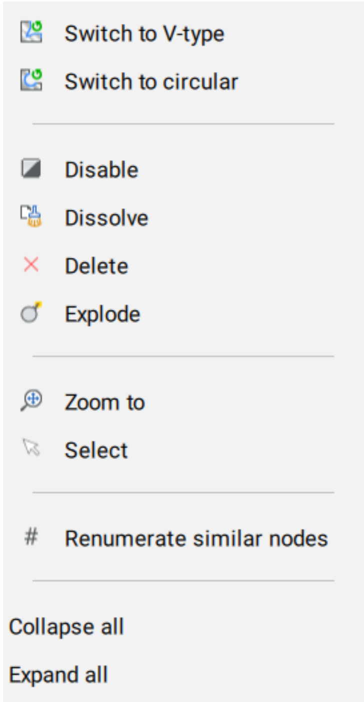
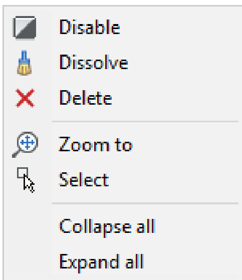


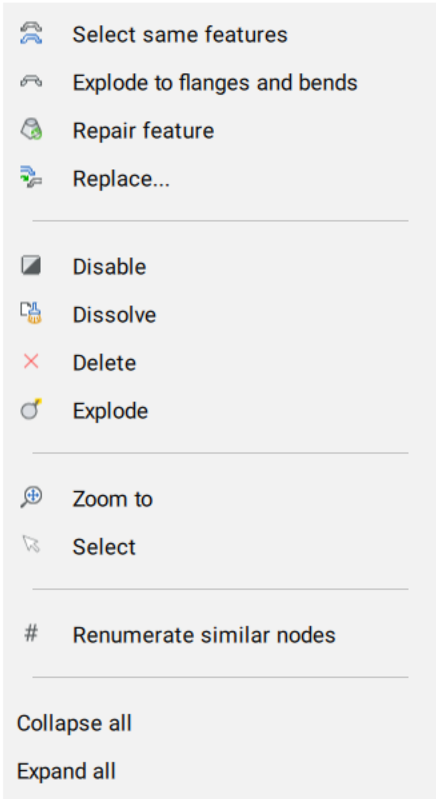
Switch to smooth/round/rip: switch a bend relief to the selected feature type.</td></tr>
</table>

<table>
<tr><th>Feature Context Menus</th><th>Description</th></tr>
<tr><td>Corner relief feature</td><td>Right-click on corner relief feature:

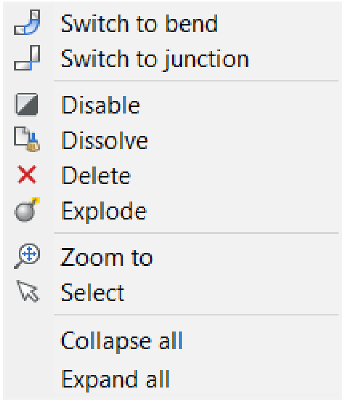


Switch to V-type/circular: switch a corner relief feature to the selected feature type.</td></tr>
</table>

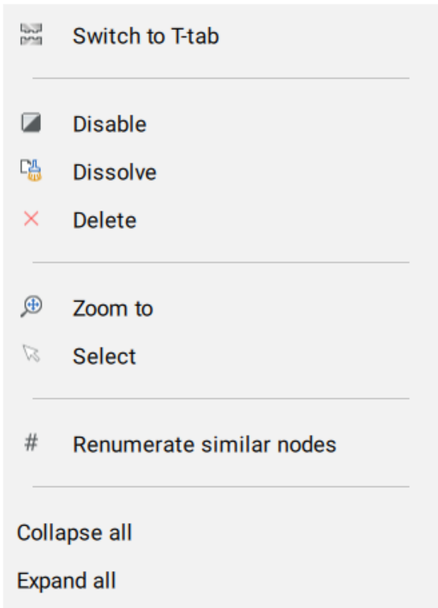
Feature Context Menus	Description
Junction feature	Right-click on junction feature:   • **Switch to bend**: switch to a bend feature. • **Switch to rib**: switch to a rib feature. • **Switch type of the feature**: switch the alignment mode between symmetrical and aligned with one of the flanges.
Miter feature	Right-click on miter feature:  

Feature Context Menus	Description
Form feature	Right-click on form feature:  • **Select same feature**: select all form features of the same type in the model. • **Explode to flanges and bends**: explode a form feature into a set of flanges and bends, if possible. • **Repair feature**: recalculate the insertion point of a form feature. • **Replace...**: display the Open dialog box in order to choose a replacement form feature.

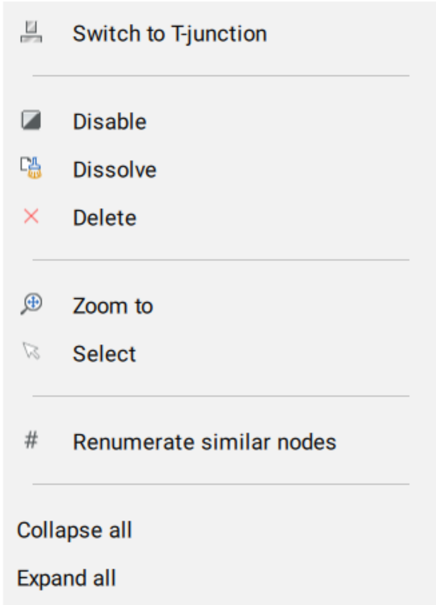
<table>
<tr><th>Feature Context Menus</th><th>Description</th></tr>
<tr><td>Tab feature</td><td>Right-click on tab feature:

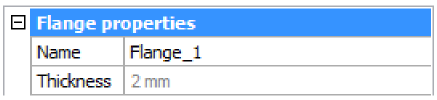


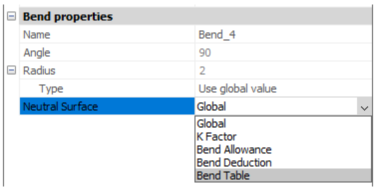
Switch to bend: convert a tab feature to a bend feature
Switch to junction: convert a tab feature to a junction feature</td></tr>
<tr><td>T-junction feature</td><td>Right-click on T-junction feature:



Switch to T-tab: convert a T-junction feature to a T-tab feature.</td></tr>
</table>

Feature Context Menus	Description
T-tab feature	Right-click on T-tab feature:   **Switch to T-Junction**: convert a T-tab feature to a T-junction feature.

Feature Properties	Description
	Common property: **Name**: the name of the selected sheet metal feature. Optionally type a name for the feature. The default name of a feature is the feature type plus an increment, for example, Flange_1, Flange_2, ...
Flange feature	Click a flange feature:  **Thickness**: current thickness. The value is read-only; it equals the sheet metal component thickness property.

<table>
<tr><th>Feature Properties</th><th>Description</th></tr>
<tr><td>Bend feature</td><td>Click a bend feature:


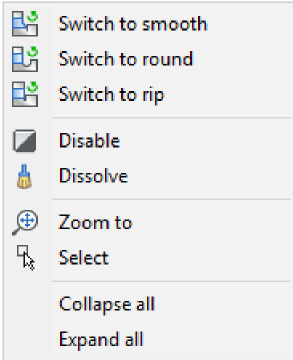
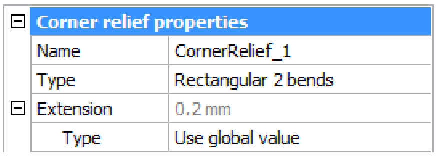
Angle: current angle of the bend feature; can be changed by the SMFLANGEROTATE command.
Radius: current bend radius.
Radius type:

Thickness ratio: ratio of thickness multiplied by a coefficient.
Absolute value: the global value (main sheet metal component properties).
Use global value: effective value of the parameter is taken from Sheet Metal context

Neutral Surface: Sets the neutral surface for given bend:

Global: use global K Factor or Bend Table
K Factor: override neutral surface by defining K Factor
Bend Allowance (BA): override neutral surface by defining Bend Allowance
Bend Deduction (BD): override neutral surface by defining Bend Deduction
Bend Table: override neutral surface by defining Bend Table

</td></tr>
</table>

Feature Properties	Description
Bend relief	Click a bend relief feature:  • **Relief type**: current relief type; either rectangular, smooth, round or rip. • **Width**: width of the cut near the bend; either global, thickness ratio or absolute.
Corner relief	Click a corner relief feature:  • **Relief type**: current relief type; either rectangular 2 bends, rectangular, v-type or circular. • **Width**: width of the cut near corner; either global, thickness ratio or absolute.

<table>
<tr><th>Feature Properties</th><th>Description</th></tr>
<tr><td>Junction feature</td><td>
Click a junction feature:
<table>
<tr><th colspan="2">Junction properties</th></tr>
<tr><td>Name</td><td>Junction_1</td></tr>
<tr><td>Junction gap</td><td>0.002 mm</td></tr>
<tr><td>Type</td><td>Use global value</td></tr>
<tr><td>Connection type</td><td>Symmetric</td></tr>
<tr><td>Junction alignment</td><td>Use global value</td></tr>
</table>

Junction gap: gap between 2 adjacent flanges.
Junction gap type: either global, thickness ratio or absolute.
Connection type: control the relative position of adjacent flanges; either symmetric or adjacent to one of the flanges.
Junction alignment: control the position of the faces of a junction feature:

global
enabled (aligned with the corresponding corner relief feature faces)
disabled.

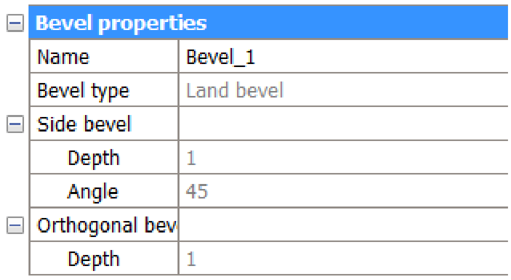
</td></tr>
<tr><td>Miter feature</td><td>
Click a miter feature:
<table>
<tr><th colspan="2">Miter properties</th></tr>
<tr><td>Name</td><td>Miter_1</td></tr>
<tr><td>Miter gap</td><td>0.2 mm</td></tr>
<tr><td>Type</td><td>Use global value</td></tr>
</table>

Miter gap: gap between the side faces of the miter.
Type: either thickness ratio or absolute.

</td></tr>
<tr><td>Form feature</td><td>
Click a form feature:
<table>
<tr><th colspan="2">Form properties</th></tr>
<tr><td>Name</td><td>Bridge_1</td></tr>
<tr><td>Component name</td><td>Bridge</td></tr>
<tr><td>Parameters</td><td></td></tr>
<tr><td>L</td><td>18 mm</td></tr>
<tr><td>W</td><td>4 mm</td></tr>
<tr><td>H</td><td>5 mm</td></tr>
<tr><td>A</td><td>60</td></tr>
</table>

Component name: name of form feature component definition.
Parameters: contains the parameters of a parametric sheet metal form feature.

</td></tr>
</table>

<table>
<tr><th>Feature Properties</th><th>Description</th></tr>
<tr><td>Bevel feature</td><td>Click a bevel feature:


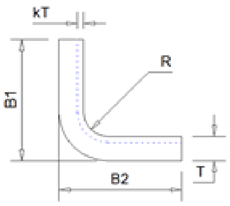
Bevel type: defines a type of the bevel, depending on the geometry. Possible values: one-side, land, X- and K- bevel. Upon a bevel type, parameters may alter.
Side bevel: parameters of chamfer on one side of this bevel.
Orthogonal bevel: parameters of orthogonal side.
</td></tr>
</table>

30.7 K-factor

30.7.1 Overview

When a metal sheet is bent with a press brake, its material is deformed, which is the reason for the difference in overall length of a sheet in folded and unfolded states. BricsCAD automatically considers this deformation when you unfold a sheet metal part. Two approaches are used to compute it: a simplified one (K-factor) and a more advanced one.

The material deformation properties for bending is based on the assumption that there is internal surface for the sheet metal part which is not deformed when the sheet is bent. From the variety of internal surfaces, one surface is selected which is equidistant from one side of the bend. The surface is local for each bend, and for simple cases it can be propagated on entire part, like the one in the picture. We assume that this surface is not stretched during bending.

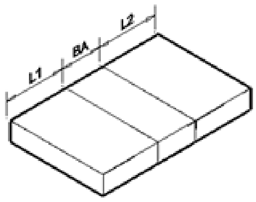


T: thickness of the sheet metal part

R: internal radius of the bend

The K-factor (k) is the ratio of the location of the neutral surface to the material thickness. As a result, the neutral surface lies at a distance kT from the internal surface of the bend. The bend radius of the neutral surface equals R + kT. The K-Factor is a simple geometric calculation of the location of the neutral surface. Forming stresses and other unknown (error) factors are not considered. The K-Factor depends on many factors, such as the kind of material, bending type and tools. The K-Factor typically lies between 0.3 and 0.5. The default K-Factor for a bend radius equal to the material thickness (T) is 0.27324.

Note: The SMDEFAULTKFACTOR system variable sets the default value of the K - Factor.



In the unfolded state of a sheet metal part, the flanges (L1 and L2) are not stretched. The bend part is indicated as BA: **Bend Allowance**. The BA equals the length of the unfolded neutral surface:

Bend Allowance: BA = BendAngle * (R + kT)

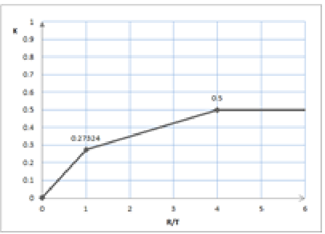
In fact, it is hard to measure the K-factor or the bend allowance. The following formula allows to calculate the tangential **Bend Deduction** (BD):

Bend Deduction: BD = B1 + B2 - Unfolded Length = B1 + B2 - (L1 + L2 + BA)

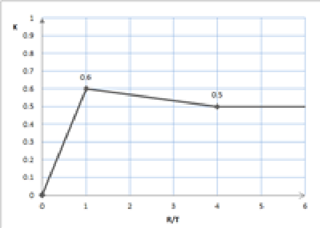
For simple cases, the K-Factor value can be overridden in the Mechanical browser. For maximal precision a bend table containing tangential bend deductions has to be provided. For each sheet metal part, you can either specify a K-factor or use the default one.

30.7.2 Changing the K-factor for a Sheet Metal Part

To change the K-Factor for a sheet metal part, select the root node in the Mechanical Browser and type a value in the **K-Factor** field. The value must lie in the range [0, 1], since the neutral surface is located inside the sheet metal part. In BricsCAD the K-Factor is defined for bend radius equal to the material thickness (R/T = 1). To compute the value of **K-Factor** for an arbitrary bend radius, BricsCAD uses a special interpolation technique which is proved to be valid for industrial sheet metal applications. The image below shows the interpolation strategy: for R/T < 1 the linear interpolation is done between 0 and 0.27324, for 1 < R/T < 4 the K-Factor is linearly interpolated between 0.27324 and 0.5 and if R/T is greater than 4, the K-Factor is constant and equals 0.5. Physically it means that if the bend radius is much larger than the thickness (at least 4 times), the material stretch is neglected.



The only K-factor parameter you can control in the Mechanical browser is for R/T=1. That means if you set the value to 0.6, the interpolation law leads to the following result:

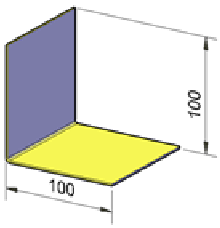


Pic: the varying K-factor in the browser leads to the following effective curve.

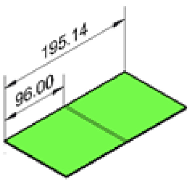
The distinguishing feature of the curve is that varying R in range of (0, T], L1 + L2 + BA gives the same value. This reflects the fact that for radii less than the thickness they relate to are for design convenience rather than for some physical process modeling, since during bending tool radius is usually greater or equal to thickness.

Let us review the example with varying K-factor by control curve setting (the bend radius is kept constant).

T = 2, R = 2, B1 = 100, B2 = 100



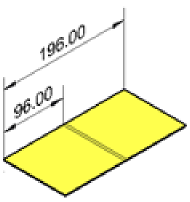
Setting K=0 gives L1=L2=96, Unfolded Length = 195.14, thus BA is 3.14.



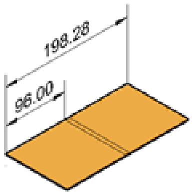
This value fits the formula exactly: a straight angle in radians roughly equals 1.57 (PI/2). The default K-factor 0.27324 gives an Unfolded Length = 196.

Or: L1=L2=96 (flanges are not deformed), the BA = 4 as we can compute from the dimensions (196 - 2*96) or from the Bend Allowance formula:

BA = BendAngle * (R + kT) = 1.57 * (2 + 2*0.27324) = 4



Finally, maximizing K-factor results in an Unfolded Length = 198.28. This is the maximal value one can achieve in this example since the neutral surface is taken from the external side of the bend.



The Bend Deduction (BD) for k = 0.27324:

BD = B1 + B2 - Unfolded Length = 100 + 100 - 196 = 4

30.8 Bend Table

30.8.1 Overview

Bend tables are a more reliable way to express material deformation properties than K-factor. When you physically bend a flat sheet of a particular material, you can measure its length before and after this process. You can repeat this procedure for different bend angles, bend radius, and sheet thickness, and save the measurements in a bend table. These measurements can then be considered to compute the correct unfolded length for all parts made of the same material.

In a bend table, you can add the measurements for as many different bend angles as you want. However, in most cases it suffices to measure the length for a 90 degrees angle. BricsCAD automatically computes the length for other bend angles using a reliable interpolation technique.

Bend tables are stored in a *.CSV (comma separated values) file format. You can create a bend table in an ASCII text editor, such as Notepad. However, a more reliable and easier way is to create a table in a spreadsheet, such as Microsoft Excel or LibreOffice, and then export it to .CSV file format.

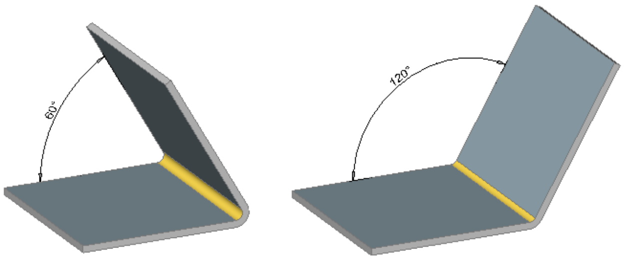
Bend tables in BricsCAD have the following structure:

Example of a bend table

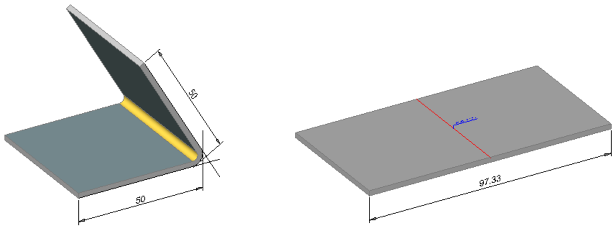
	A	B	C	D	E	F	G
1	BricsCAD						
2	Version	1					
3	AngleType	Internal					
4	LengthType	BendDeductionTangent					
5							
6	Thickness	1					
7	AngleValues		45	60	90	120	135
8	Radius	1	1	1.333	2	0.976	0.657
9	Radius	2	0.466	1.081	2.31	1.004	0.641
10	Radius	3	-0.069	0.828	2.621	1.033	0.624
11							
12	Thickness	1					
13	DieWidth	10					
14	AngleValues		45	60	90	120	135
15	Radius	1	1.225	1.533	2.15	1.076	0.732
16	Radius	2	0.616	1.214	2.41	1.071	0.691
17	Radius	3	0.006	0.895	2.671	1.066	0.649

The following rules apply:

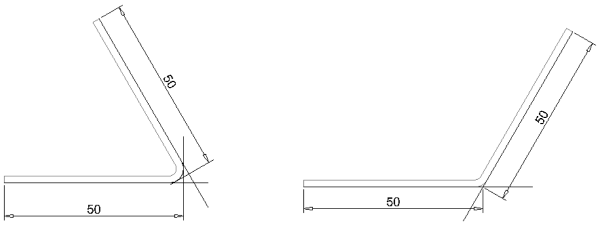
- The content of the first cell reads **BricsCAD**.
- **Version**: currently supported version is 1.
- **AngleType**: currently only internal bend angles are supported:



- **LengthType**: the semantics of the value in the cells of the bend table corresponding to a particular bend angle and bend radius. Now only bend deduction measured from the tangent point is supported (**BendDeductionTangent**). Bend deduction (BD) is the difference between the sum of the lengths of 2 flanges measured to the tangent point on a 3D model and the length of the same fragment in the unfolded state:

  In depth: methods to compute tangential bend deduction for acute and obtuse angles.

- Within the defined header, several tables can exist. For each table, the first key is the **Thickness** of the sheet metal part. The second key, **DieWidth**, is optional. The **Thickness**/**DieWidth** pairs must be unique. If the **DieWidth** is not defined, only one unique **Thickness** value key is allowed, but multiple thickness values can be added. When a bend table is used to control the unfolding process, the **Thickness** and **DieWidth**, defined in the sheet metal part, must exist in the bend table. If the table is not found, the default K-factor policy is used.
- Once the bend table is found in the .CSV, it is used to unfold the sheet metal part. Using the sample bend table above, let us check a few cases:
- T = Thickness, R = Internal Bend Radius, A = Bend Angle
  - T = 1, R = 2, A = 45. Then after unfolding it will obtain the BD = 0.466, because both R and A are found in the bend table for the given thickness.
  - T = 1, R = 2.5, A = 45. A = 45 is found, but R = 2.5 is missing: the interpolation of BD between R=2 and R=3 is used.

- T = 1, R = 2.5, A = 75. Neither R nor A is found in the table: the following cells are used to calculate the value of BD: (R=2, A=60), (R=3, A=60), (R=2, A=90) and (R=3, A=90).

The algorithm is used to calculate the interpolation. For example, linear interpolation on adjacent BD values render unnatural results, which does not match the real bending results. Also keep in mind that for a given T, R and A not all BD values make sense: a random one might place the neutral surface outside the sheet metal part, or expressed in K-factor (which can be deduced from BD), the K-factor can be out of range [0, 1]. In this case, the BD for the default K-factor is used.

- If you produce your sheet metal parts with different tools, you can describe them in one bend table using the **DieWidth** parameter. A sample bend table is provided for your convenience in the*[BricsCAD Installation folder]\Samples\Mechanical\bend_tables*.

  Note: Writing rights to this folder might be limited. In such case you need to copy the bend tables to a different folder in which you have full access rights.

30.8.2 Setting the bend table for your sheet metal part

1 Select the root node in the **Mechanical Browser**.

2 Select the **Bend table** field.

3 Click the **Browse** button (...).

4 Select the *.CSV file.

5 Double-click or press the **Open** button.

30.8.3 Updating the bend table set previously

1 Select the root node in the **Mechanical Browser**.

2 Select the **Bend table** field.

3 Click the **Reload file** button ().

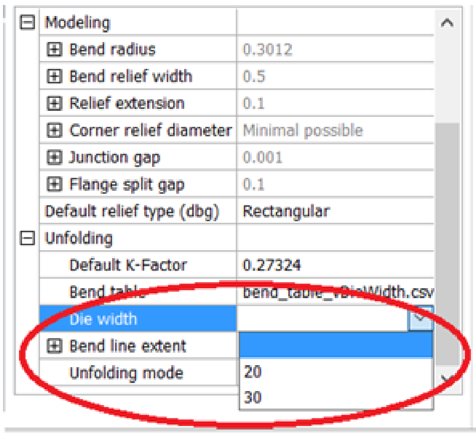
If the bend table is not assigned to the document or has no **DieWidth** secondary key, the **DieWidth** parameter can not be assigned to a single bend or all the bends. In order to make the assignment, perform the following steps:

	A	B	C	D
1	BricsCAD			
2	Version	1		
3	AngleType	Internal		
4	LengthType	BendDeductionTangent		
5				
6	Thickness	1		
7	AngleValues		90	
8	Radius	1	2.1	
9				
10	Thickness	1		
11	DieWidth	20		
12	AngleValues		90	
13	Radius	1	1.7	
14				
15	Thickness	1		
16	DieWidth	30		
17	AngleValues		90	
18	Radius	1	1.8	

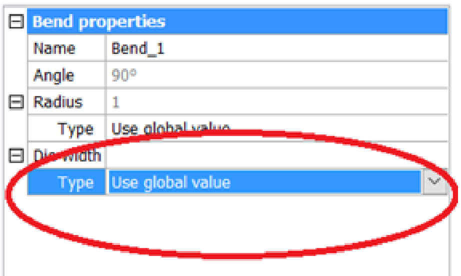
1 Select .CSV file **DieWidth** secondary keys and thickness matching the current part. On given example we assume that part has thickness of 1 mm and bend radius also equals to 1. We will vary bend deduction to different tools: the first sub-table is when the tool is not specified, second sub-table is for the case when tool **DieWidth** is equal to 20, and the last one is for a tool with **DieWidth** equal to 30.

2 Assign the bend table to the sheet metal part.

3 Check that in the **Mechanical Browser** new fields appear:

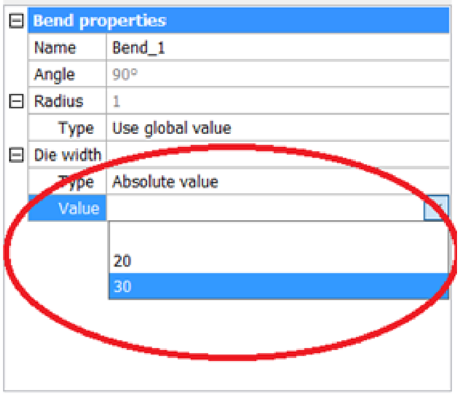


In this global setting we can select any default **DieWidth** value suitable for the part thickness. The value is assigned by default to the bends in the part, which can be checked by the property of the bend (click on the bend in the **Mechanical Browser**):



4 However, we can select a specific tool which will be used for the given bend by expanding the drop-

down list:



5 With the given selection, an unfolding of Bend_1 uses a third sub-table from our bend table for bend deduction and uses the value 1.8. For the rest of the bends, unless **DieWidth** is set for them, the default no-die-width value will be used, which will trigger first the bend table and uses value 2.1 for 90-degrees with a bend radius equal to 1.

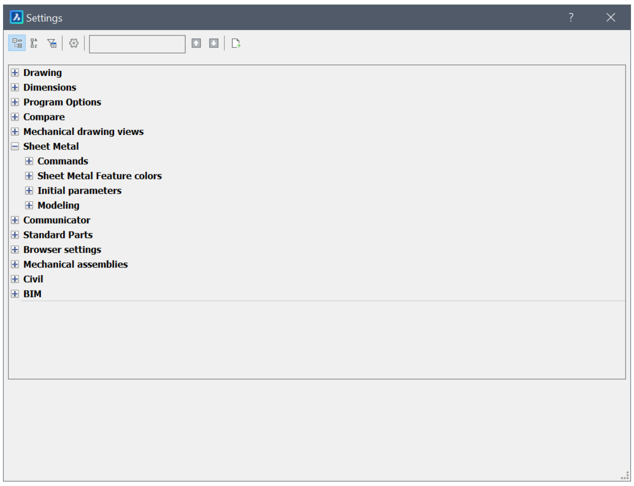
Note: After assigning the **DieWidth** to the bends and saving the document, the value is saved to the bends. However, next time the drawing is opened, in order to assign the **DieWidth** value to another bend, you have to reload the bend table first (press the **Reload file** button ( ), see **Updating the bend table set previously**. After the reload, the values from the bend table will be available.

Note: You can alter bend tables for different solids in model space by defining a local Sheet Metal Context per each solid.

30.9 Sheet Metal Settings

30.9.1 Overview

The Sheet Metal section in the *Settings* dialog box controls the different properties of Sheet Metal in BricsCAD. Properties are split into 4 groups: **Commands**, **Feature colors**, **Initial parameters**, and **Modeling**.

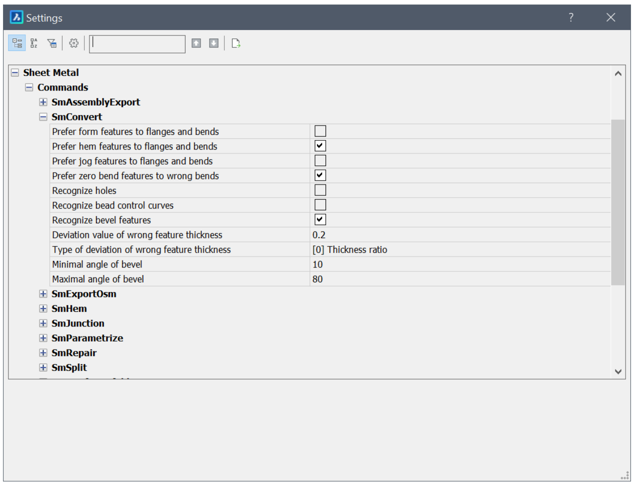


Note: Most of the sheet metal settings are stored in the drawing. For best results always start with one of the sheet metal template files DWT or create your own.

30.9.2 Commands

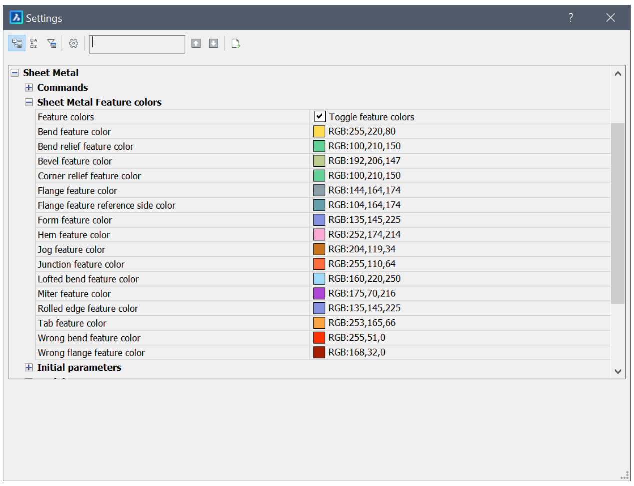
The **Commands** section contains settings that control the behavior of the various Sheet Metal commands.

Note: Before BricsCAD V19 these controls were available as options within the individual commands. They were moved to the settings dialog to simplify the command interface.



The Layers used for Sheet Metal Unfolding determine the appearance of unfolded parts.

30.9.3 Feature colors



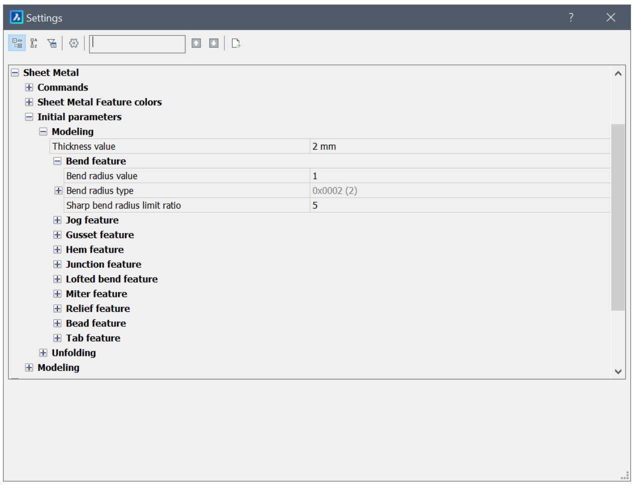
The **Feature colors** settings visually distinguish different types of sheet metal features by color. You can adjust the colors for each feature type.

30.9.4 Initial parameters

These parameters are responsible for the initialization of the **Sheet Metal Context** associated with a drawing.

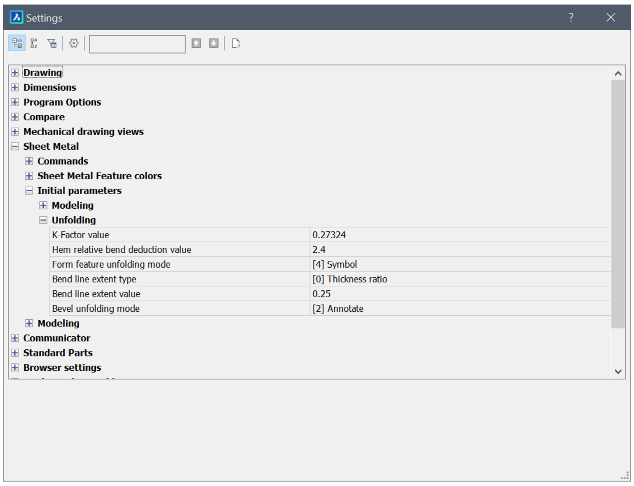
Modeling

Modeling settings control the behavior of sheet metal features. Settings are grouped by feature type.



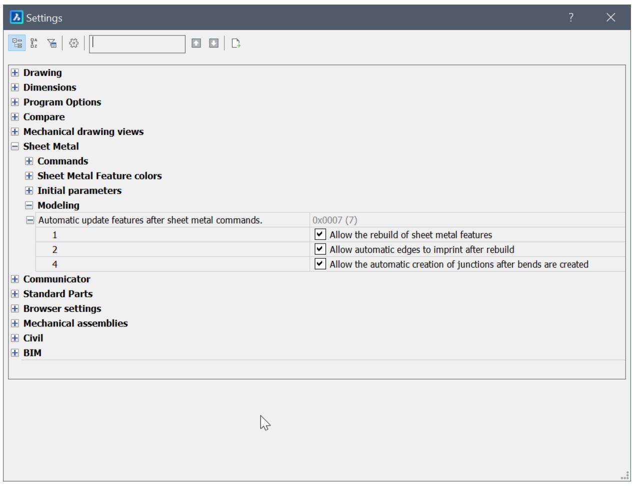
Unfolding

Unfolding settings control the behavior of **K-factor value**, **Form feature unfolding mode**, and **Bend line extent type/value**.



30.9.5 Modeling

The Modeling settings include the *SMSMARTFEATURES* user preference that controls the local, intelligent update of the geometry after Sheet Metal commands are executed.



30.10 Creating Flanges

30.10.1 Overview

A flange is the main feature of most sheet metal parts. In BricsCAD you can create flanges using many different techniques:

- Create a base flange from a closed 2D profile, which defines its boundary.
- Create one or several edge flanges by pulling edges of existing flanges.
- Create several flanges at once by extruding a 2D polyline (every linear edge of the polyline becomes a flange).

Tip: There are also other techniques you can use to create flanges. These are described in:

- Flanges can be created as elements of lofted sheet metal parts. See **Creating and Splitting Lofted Bends**.
- Flanges can be recognized in existing geometry. See **Converting to Sheet Metal and Repairing**.

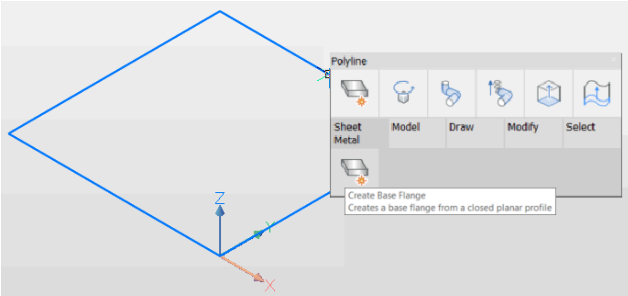
In contrast to history-based MCAD, BricsCAD applies the same set of tools to modify any flange - independently.

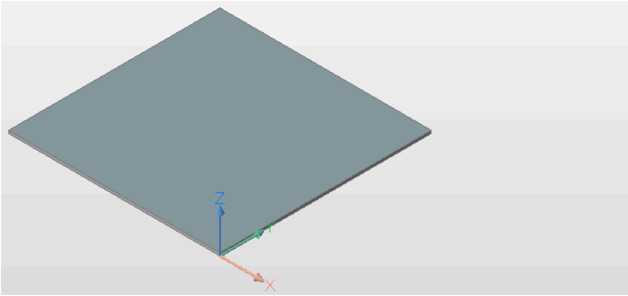
30.10.2 Create a Base Flange

In general, you create a base flange to start your sheet metal part design. To create a base flange, select a closed planar profile and run the SMFLANGEBASE command.

Do one of the following:

- Click the **Create Base Flange** tool button () on the **Sheet Metal** toolbar.
- Choose **Create Base Flange** in the **Sheet Metal** menu.
- Choose **Create Base Flange** in the **Sheet Metal** group of the **Quad** cursor menu:





To create a base flange the SMFLANGEBASE command extrudes a profile.

The height of the base flange is the default **Thickness** property of the sheet metal part. To change the thickness of your sheet metal part, type the appropriate value in the **Thickness** field of the Mechanical Browser.

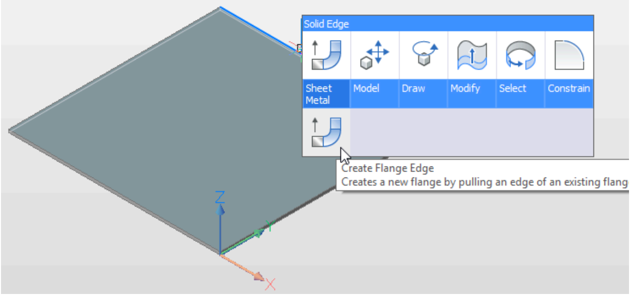
30.10.3 Create Edge Flanges

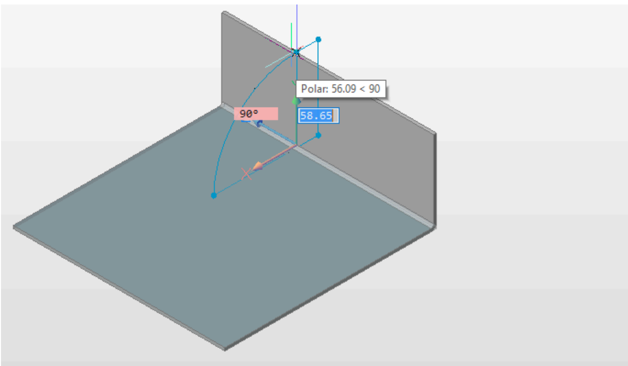
You can add an additional flange to your sheet metal:

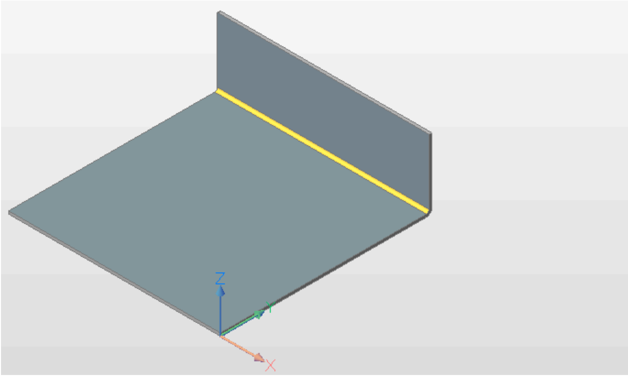
1 Select a linear edge.

2 Select the SMFLANGEEDGE command in the **Sheet Metal** section of the **Quad** cursor menu.

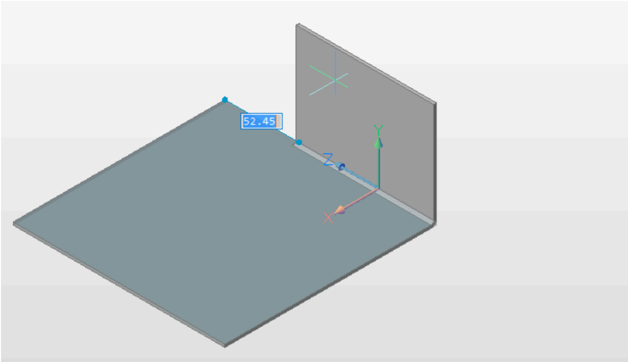
3 Move your mouse pointer to define the desired length of the edge flange and the angle between the 2 flanges. You can also use the dynamic dimensions fields to enter the required values.

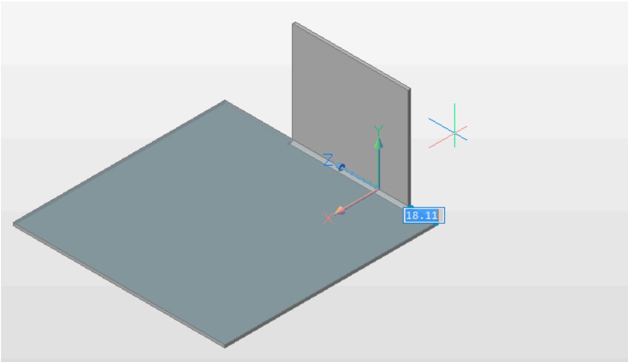


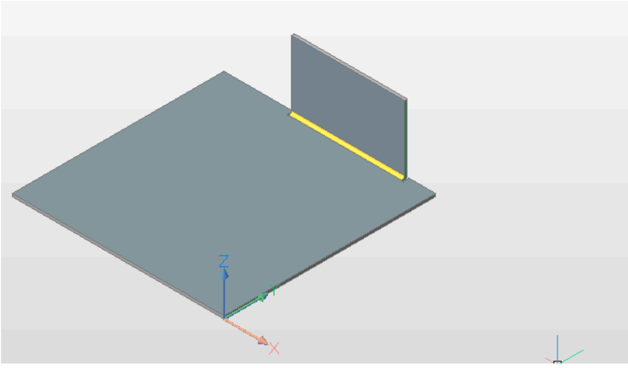




To create an edge flange where the width is different to the width of the edge, select the **Width** option of the SMFLANGEEDGE command and define the offset distance from one or both sides of the edge.



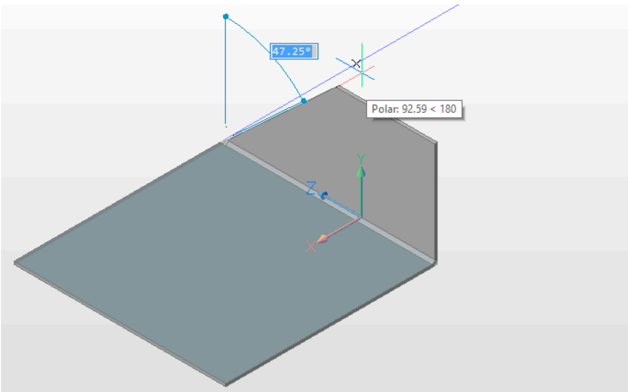


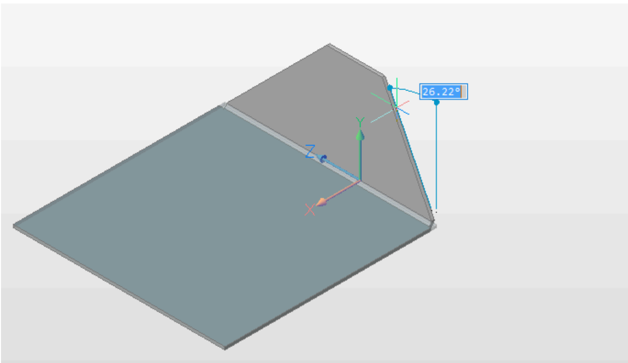


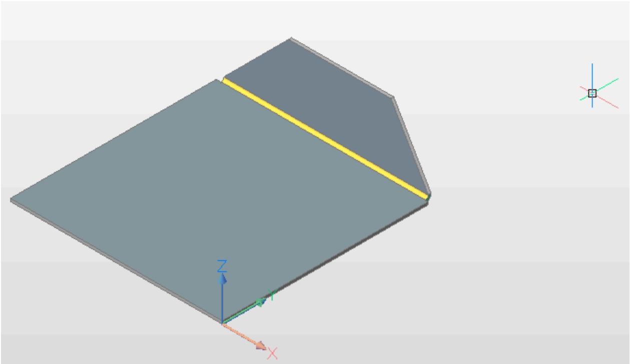
Note: When you create an edge flange of a different width to the edge, bend reliefs are created automatically.

30.10.4 Create Edge Flange with Taper Angle(s)

You can create trapezoidal edge flanges with the **Taper angle** option of the SMFLANGEEDGE command:

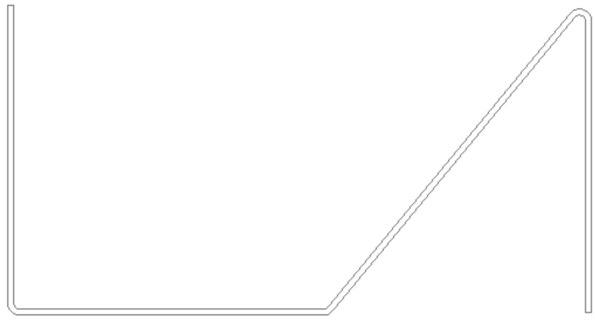






30.10.5 Create Multiple Flanges by Extruding a Polyline

There is a simple but important class of sheet metal parts which are seen as a thickened polyline if we view on the part from the "front" side:

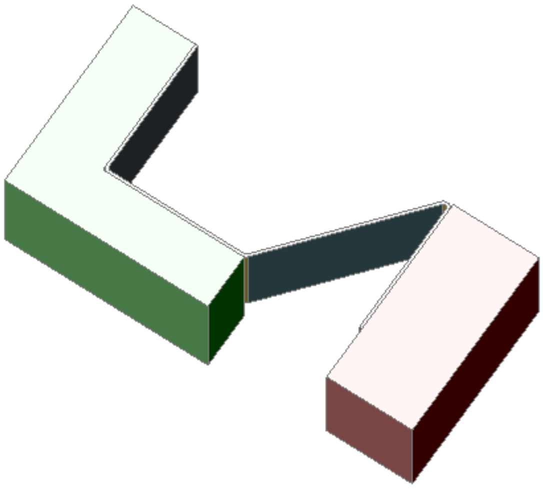


To create this part you can:

- use sequential calls of the SMFLANGEEDGE command, or
- draw a polyline and use the DMEXTRUDE command, followed by the DMTHICKEN, SMCONVERT, and SMBENDCREATE commands.

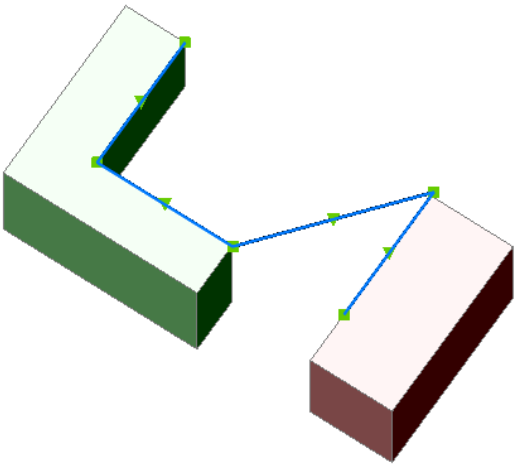
However, we recommend the following procedure:

BricsCAD allows you to draw a part in a complex assembly with other parts so that it does not interfere with other entities in the assembly.



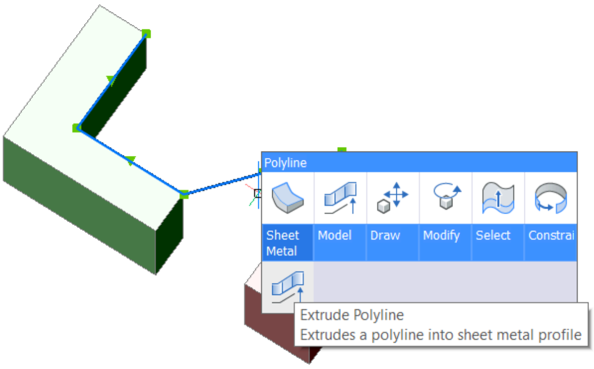
You can control the thickening direction of every line segment individually with SMEXTRUDE.

1 Draw a polyline that represents a sketch of the future sheet metal part:

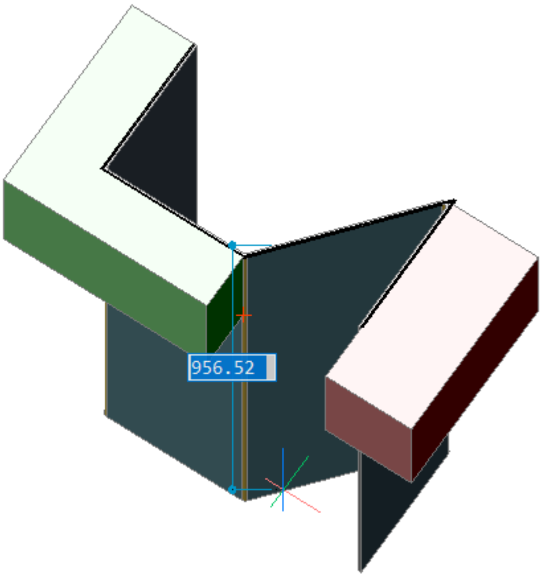


Do one of the following:

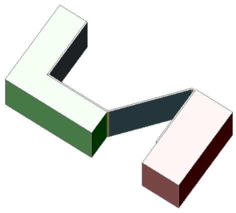
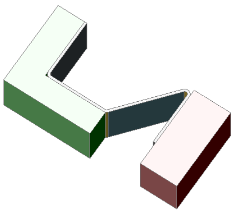
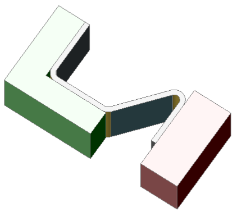
- Click the **Extrude Polyline** tool button ( ) on the **Sheet Metal** toolbar.
- Choose **Extrude Polyline** in the **Sheet Metal** menu.
- Choose **Extrude Polyline** in the **Sheet Metal** group of the **Quad** cursor menu:



2 Hover your cursor to select **extrusion height** or enter the value and press **Enter** to accept:



The body is created and the extensions are respected. The sheet metal part does not exceed the dimensions of the original polyline. The behavior is the same, even with different thicknesses:

Thickness 1 mm	Thickness 2 mm	Thickness 4 mm
		

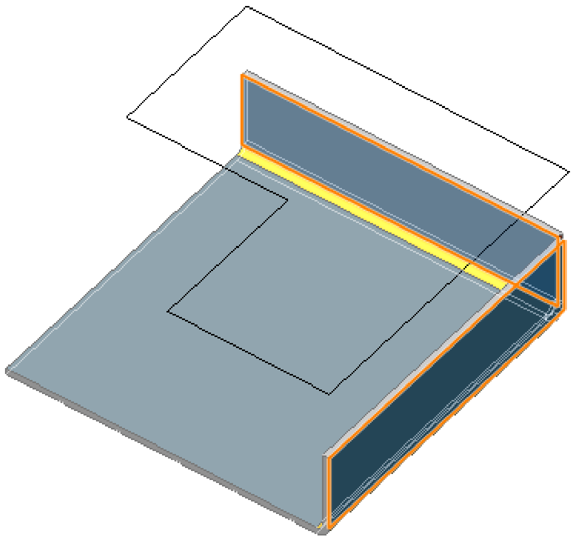
Note: The policy of preserving external dimensions also works for sheet metal parts created by other workflows: sequential flange building with SMFLANGEEDGE, part bends with SMFLANGEBEND and sheet metal design recognition with SMCONVERT. If you want to change the extrusion direction for a flange, apply the SMFLIP command to it.

30.10.6 Extrude a Contour to Create a Flange

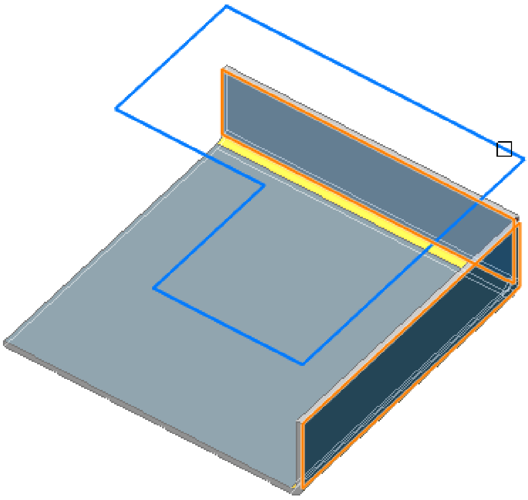
The SMFLANGECONTOUR command allows you to extrude a given contour to a new flange and attach it to the model. It can be thought of as a sequence of the SMFLANGEBASE, SMFLANGECONNECT, and SMBENDCREATE commands. In reality, it intelligently chooses the thickness faces on selected flanges and connects them with the thickness faces on a new flange.

Do one of the following:

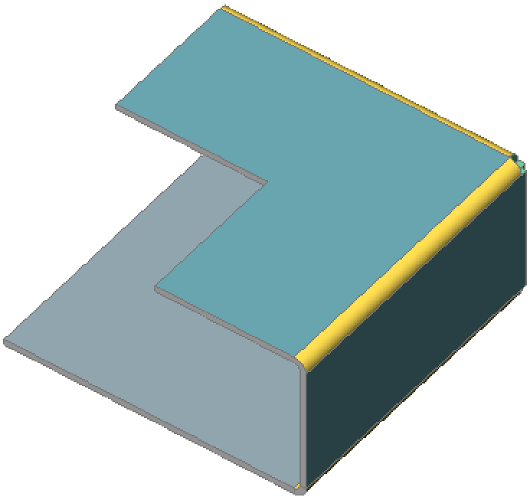
- Click the **Create Flange from closed Contour** tool button ( ) on the **Sheet Metal** toolbar.
- Choose **Create Flange from Contour** in the **Sheet Metal** menu.
- Choose **Create Flange from Contour** in the **Sheet Metal** group of the **Quad** cursor menu:



Once all the required flanges are selected, click on the contour:

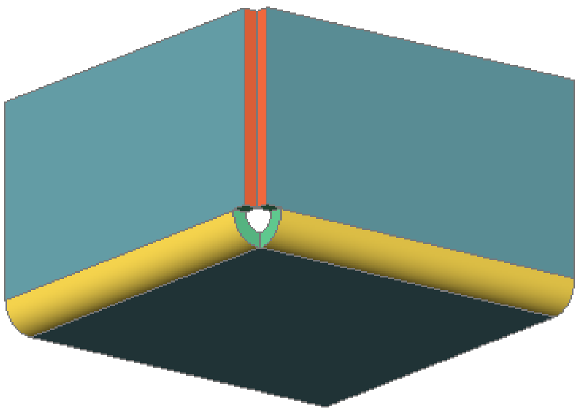
- 

The command creates a new flange and merges it with the selected flanges on the part:

- 

30.10.7 Bends, Corner Reliefs and Junctions

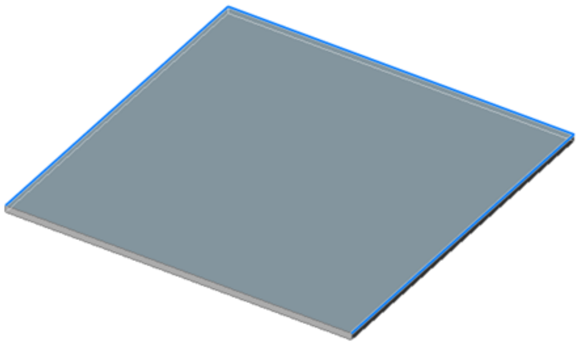
When you pull an edge flange with the SMFLANGEEDGE command, a bend feature between the edge flange and the base flange is automatically created. A corner relief feature is automatically created in a corner where 3 flanges meet together. This operation also creates a junction between 2 flanges, which are not connected with a bend.



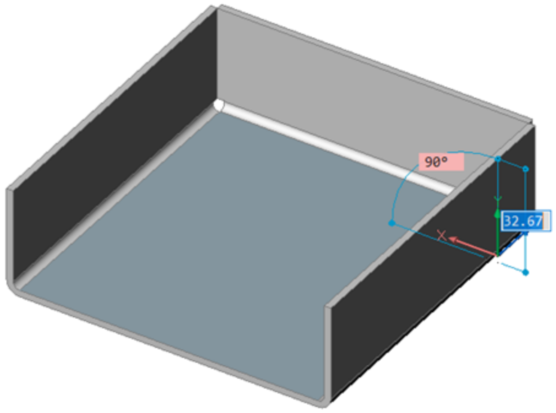
30.10.8 Create Multiple Flanges

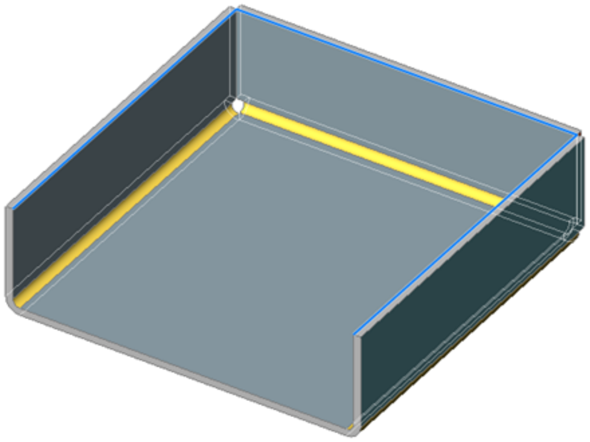
You can create several flanges simultaneously.

Do one of the following:

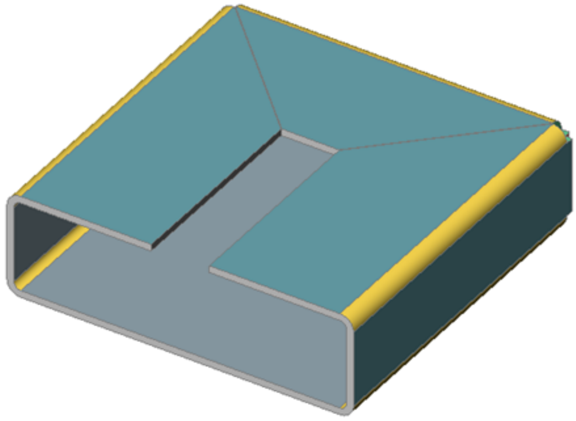
- Select multiple edges, then choose **Create Flange Edge** () in the **Sheet Metal** command group of the **Quad** cursor menu.
- Launch the SMFLANGEEDGE command and select multiple edges.
- 
- Create the edges dynamically.

Define the angle and the flange height using the dynamic entry fields. Press the **TAB** key to switch between the **Angle** and the **Distance** field.

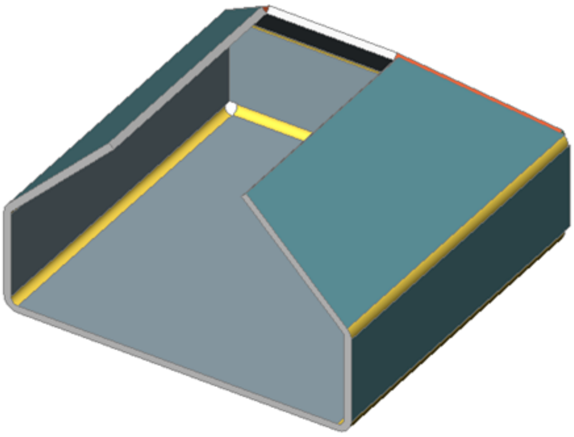
- 

- Select the inside edges of the newly created flanges:
- 
- Create the flanges.

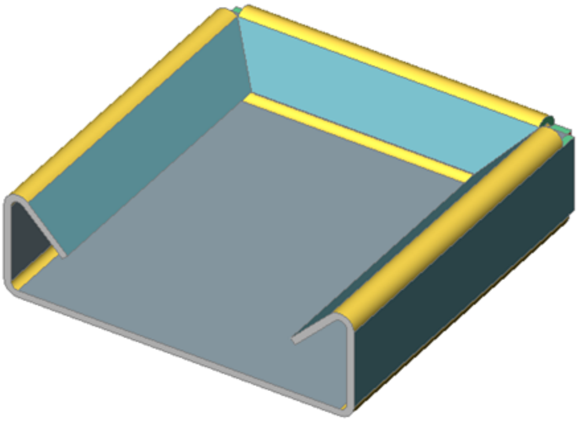
  The command automatically calculates the intersection of the flanges and creates a junction or miter features. At 90 degrees miters are created:

  - 

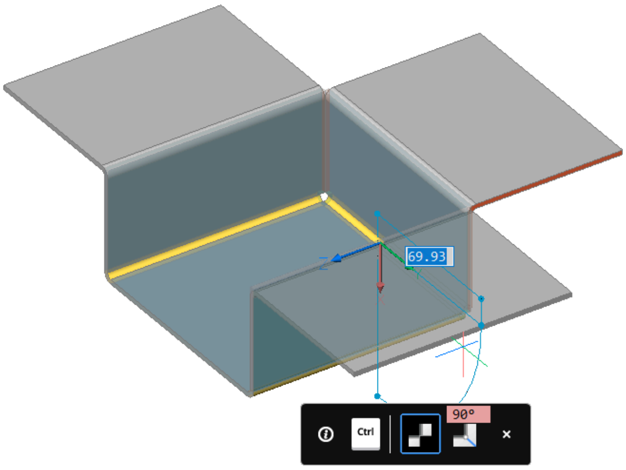
For other angles junctions are created:

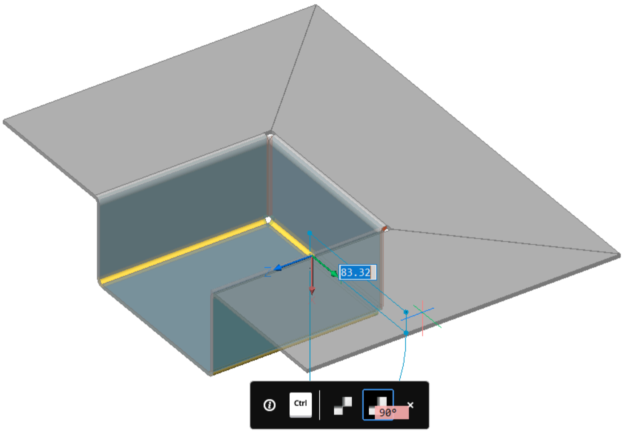
- 

In all cases material conflicts are avoided:

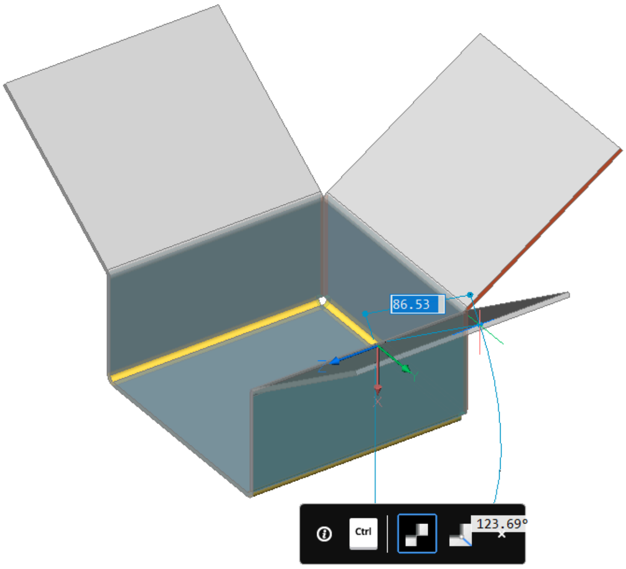
- 

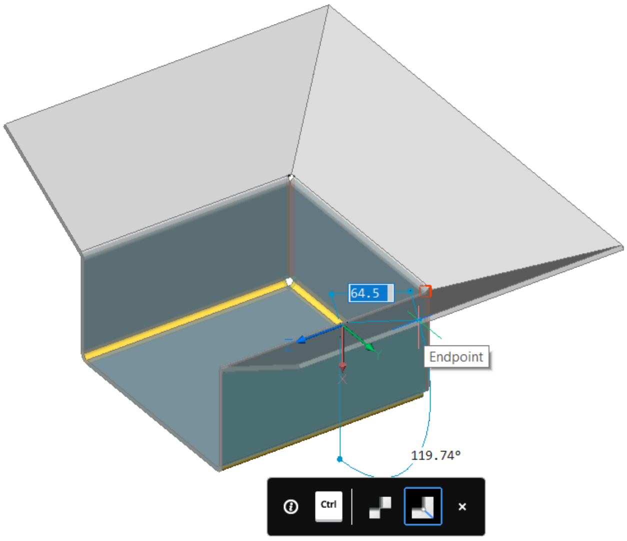
If the flanges are created externally, use the **Toggle** option (as well as **widget control**) to control the connection type for miters:

- 

- 

and junctions:

- 

- 

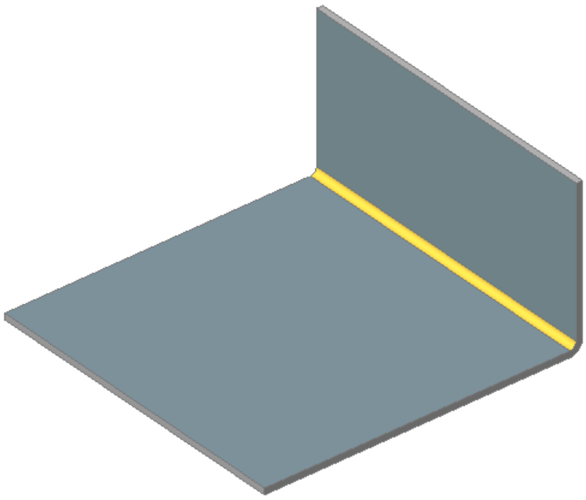
30.11 Modifying Flanges

BricsCAD provides several tools to modify flanges in sheet metal parts. These tools can be applied consistently to any part - independent of its design history. In this article we use Solid Modeling tools and specific sheet metal tools. Almost all solid modeling tools apply to sheet metal parts, while preserving the design intelligence provided by Sheet Metal Features.

30.11.1 Changing flange size and shape

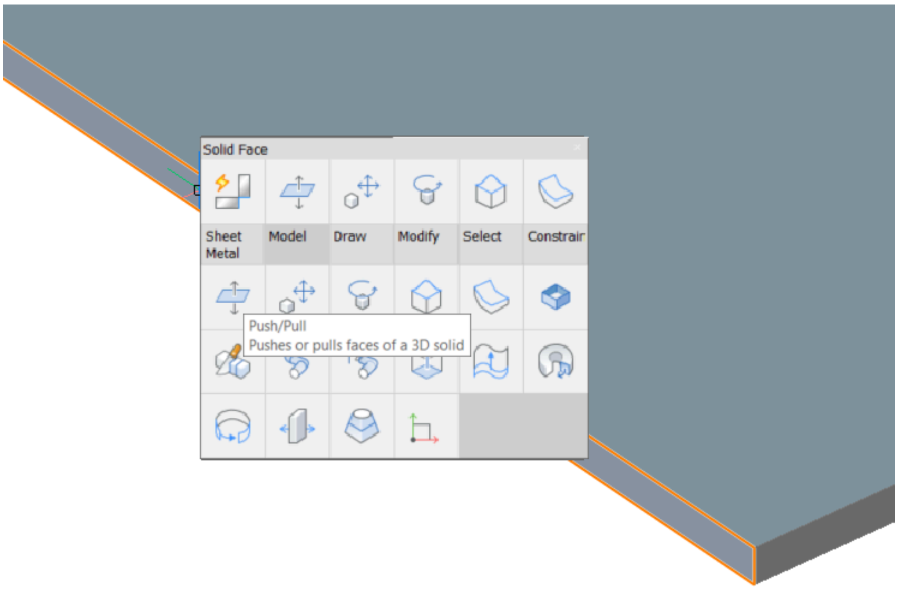
On a given simple model we will apply several Solid Modeling operations to modify the shape of its flanges.

Note: We work on thickness faces of the model only

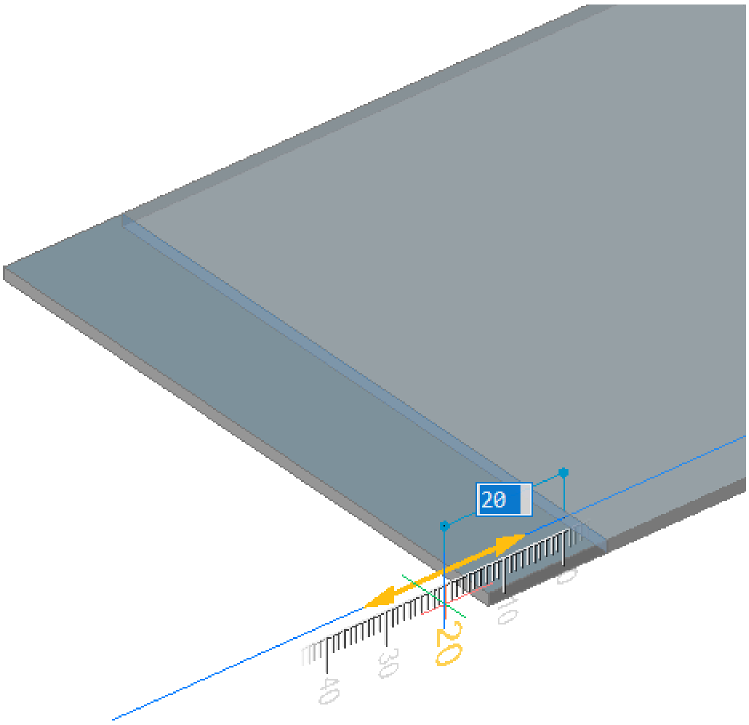


30.11.2 Stretching flanges

1 Hover over the thickness face, then select **PushPull** in the **Model** command group in the **Quad**



menu.

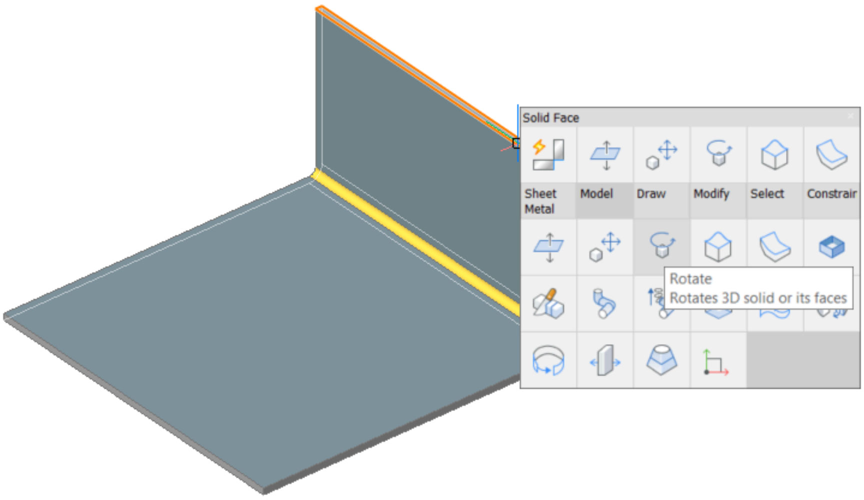


2 Drag the face, then right-click.

The changes will be applied to the part, the flange is stretched over the specified distance.

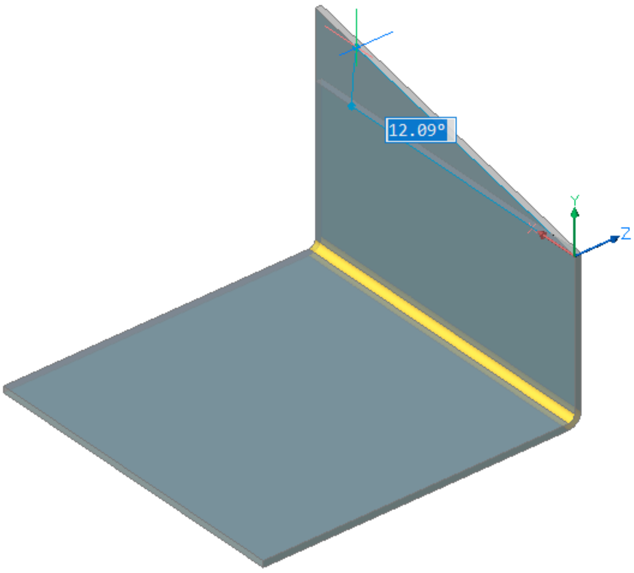
30.11.3 Rotating a thickness face

1 Hover over the face and choose **Rotate** in the **Model** command group in the **Quad**



menu.

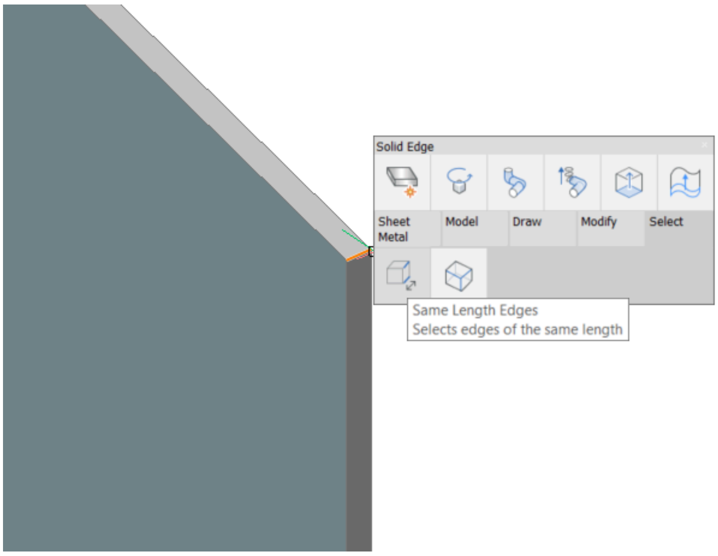
2 Specify the rotation axis, base point and angle (for more details, see DMROTATE in the Command



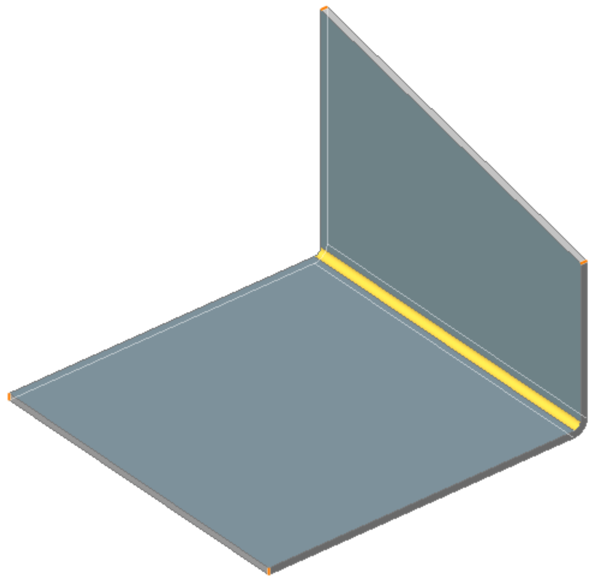
Reference):

3 Fillet the sharp edges.

Hover over a thickness edge and choose **Same Length Edges** () in the **Select** command group in the **Quad** menu.



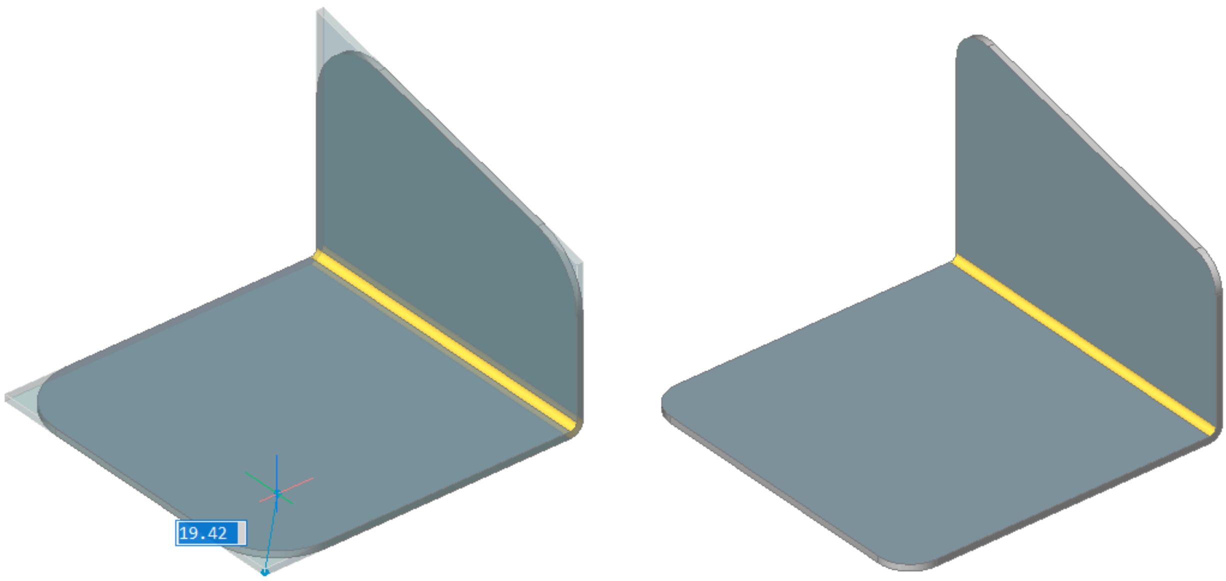
4 Press **Enter**. Edges with the same length are



selected.

5 Choose **Fillet** in the **Model** command group in the **Quad** menu.

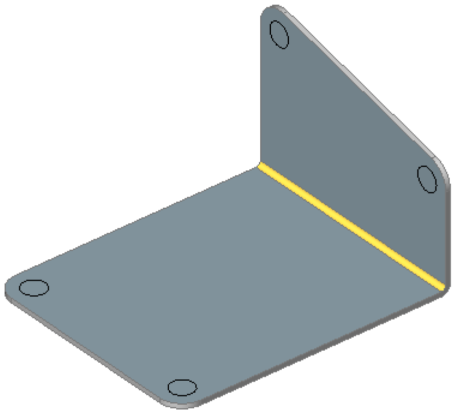
6 Specify the desired fillet



radius:

30.11.4 Making a hole in a flange

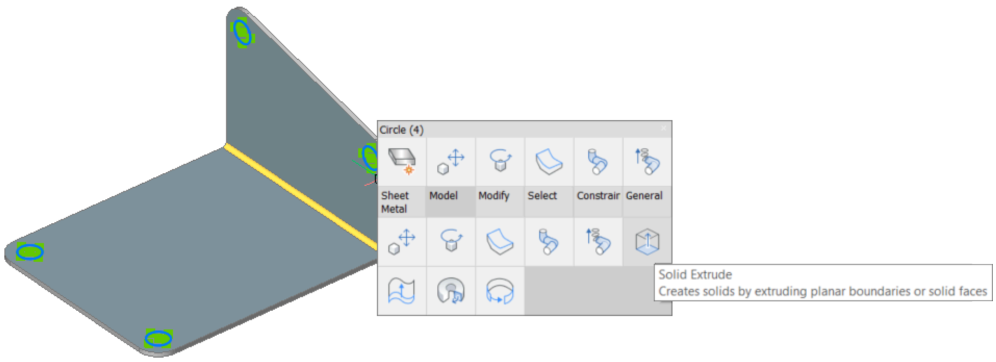
1 Create 4 circles as indicated in the image below. Use the Snap to Center entity snap to define the



center points.

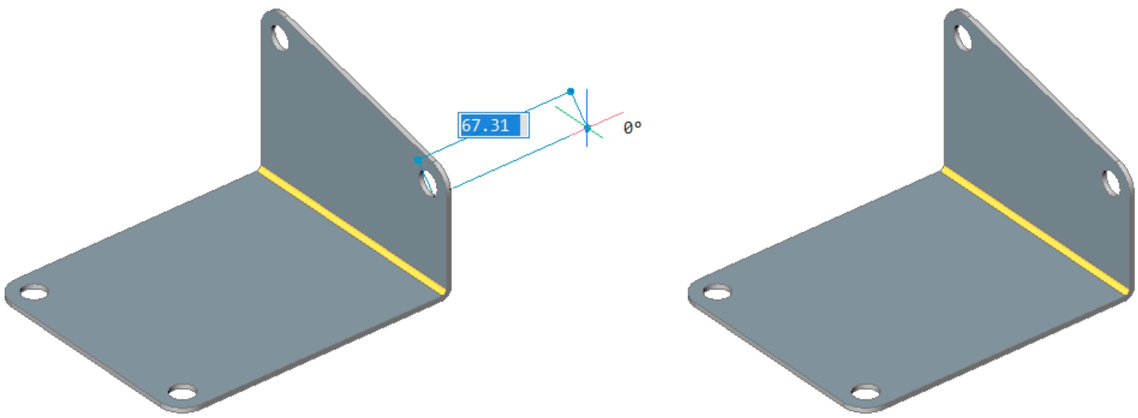
2 Select the circles.

3 Choose **Solid extrude** in the **Model** command group in the **Quad**



menu.

4 Move the cursor to create the



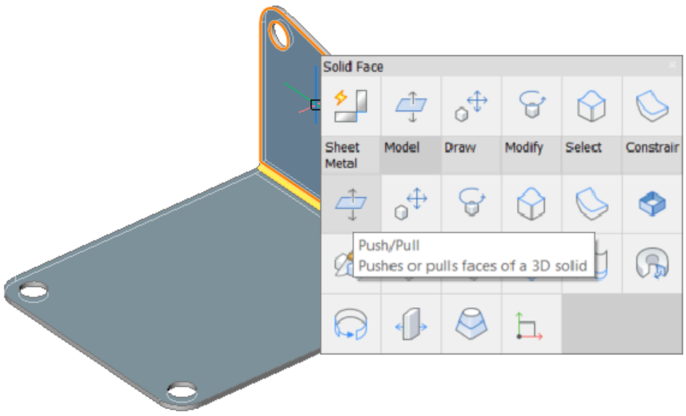
holes.

30.11.5 Moving a flange

1 Do one of the following:

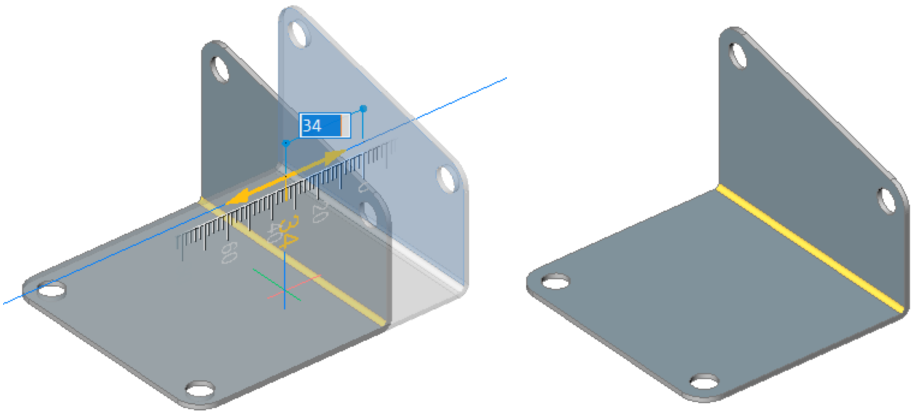
- Click the **Push/Pull** tool button ( ) on the **Direct Modeling** toolbar.
- Choose **Direct Modeling > Push/Pull** in the **Model** menu.

- Choose **Push/Pull** () in the **Edit** panel of the **Sheet Metal** tab in the ribbon.
- Run the *DMPUSHPULL* command for given face.
- Hover over the face you want to move and choose **Push/Pull** in the **Model** command group in the **Quad** menu.



2 Select new position of the flange.

Note: Due to presence of **Sheet Metal Features**, part thickness is preserved.



30.11.6 Rotating a flange

Although it is possible to rotate a flange using the DMROTATE command (), it is recommended to use SMFLANGEROTATE () instead, because this command automatically selects the rotation axis to respect the design intent of the sheet metal part.

1 Launch the SMFLANGEROTATE () command.

You are prompted: Select a flange face to rotate:

Flange faces under the cursor highlight.

2 Click to select a flange face.

The flange rotates dynamically.

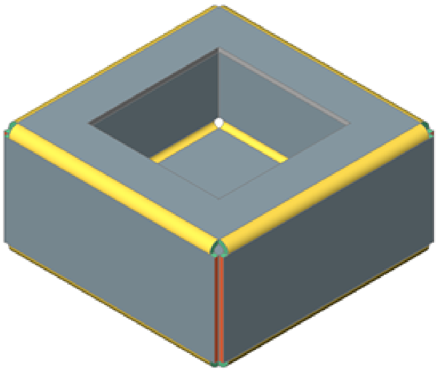
A dynamic dimension displays the current angle with respect to the base flange.

3 Do one of the following:
- Select a point.
- Type a value in the dynamic dimension field.
- Press the **TAB** key, then specify the absolute rotation angle in the dynamic dimension field.

Note: When the DMROTATE command is used to rotate a flange, which is connected to other flanges by junction features, these features are automatically dissolved before the rotation.

30.11.7 Splitting a flange

In some cases flanges are better split to minimize the material consumption, for example, when a flange has a large hole, as shown in the image below:



1 Launch the SMSPLIT ( ) command.

You are prompted: Select lines or edges to split the flange or [SMart split/draw a New line/selection options (?)] <draw a New line>:

2 Press Enter to accept the default option.

You are prompted: Start point of the line:

3 Hover over the face you want to split (make sure Dynamic UCS (DUCS) is ON).

The face highlights.

4 Specify the start point of the line.

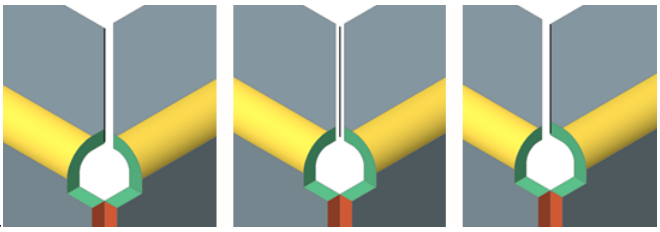
You are prompted: End point of the line:

5 Specify the end point of the line:

You are prompted: Make split Center/Left/Right/<Accept model>:

6 To adjust the position of the cut with respect to the line, do one of the following:
- Hit the **CTRL** key to toggle the **Center**/**Left**/**Right** options.

  If **TIPS** is ON, the currently selected option is indicated in the Tips Widget.
- Select **Center** , **Left** or **Right** in the Prompt Menu.
- Type an option in the Command line, then press

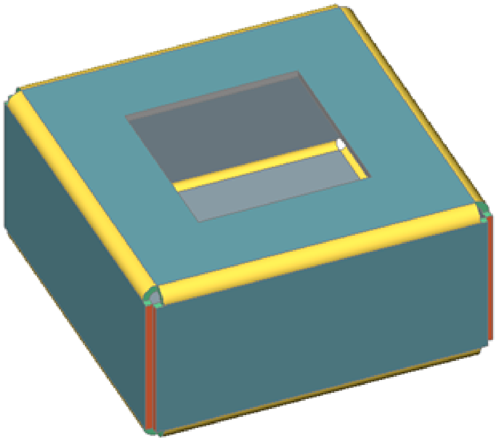
Enter. 

The flange is cut by the line: left, center or right.

Note: A split operation creates a miter feature. You can restore the initial flange by removing the miter feature from the feature list in **Mechanical Browser**.

30.11.8 Smart tools to split a flange

The scenario below illustrates the other capabilities of the SMSPLIT command:

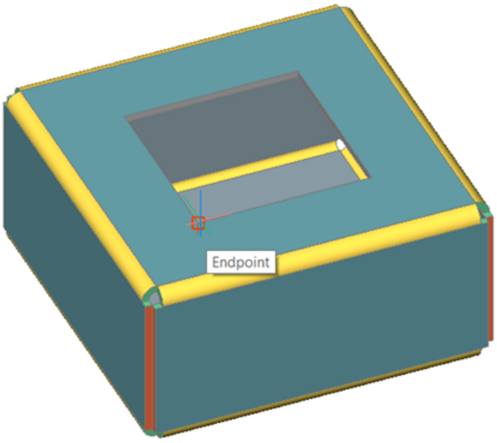
- Smart split by selecting a single vertex in the corner.
- Split off the bend and convert the part adjacent to the corner into a  junction.

1 Do one of the following:
   - Click the **Smart Split** tool () on the **Sheet Metal** ribbon tab.
   - Choose **Smart Split** in the **Mechanical** > **Sheet Metal** > **Split** menu.
   - Type SMSPLIT in the Command line.
   - You are prompted: Select lines or edges to split the flange or [SMart split/draw a New line/ selection options (?)] <draw a New line>:

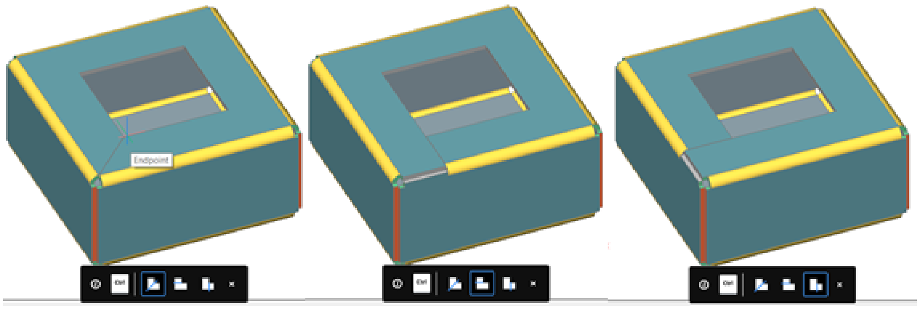
     Choose the **SMart split** option.

2 You are prompted: Select point on flange or lofted bend border for smart split or [Flange propagate/ vertex Propagate] <vertex Propagate>:

3 Select a corner vertex or snap to a point on an edge off the flange you want to split:



You are prompted: Make split [Center/Left/Right/Accept] <Accept>:

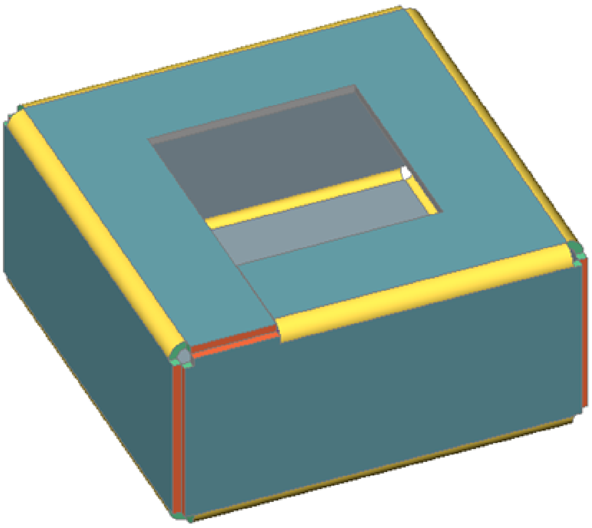


4 Do one of the following:

- Press Enter to accept the split.
- Repeatedly press the **CTRL** key.

  If **TIPS** is ON, the currently selected option is indicated in the Tips Widget.

  The flange is cut by a line through the specified point. The bend is partially converted into the junction:



In case of the **Center** option if **Smart Features** are turned on, corner reliefs adjacent to the split will be converted into bend reliefs.

Note: The conversion of part of the bend into a junction is unrelated to the **Smart Split** mode.

30.11.9 Bending a flange

The SMFLANGEBEND command bends one or more flanges using a line. You can either draw the line first, then start the command, or use the **New Line** option of the command.

To bend multiple flanges simultaneously

1 Launch the SMFLANGEBEND command.

You are prompted: Select a flange face [selection options (?)]:

2 Select the flange you want to bend.

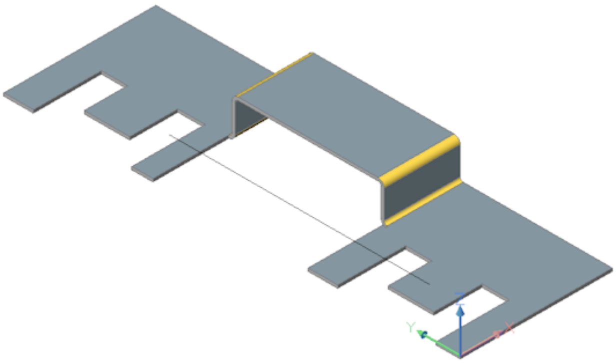
You are prompted: Select line or edge to bend the solid or draw a [New line] <New line>:

3 Do one of the following:

- Select a line that overlaps the flange(s) to be bent.
- Choose the **New Line** option.

  You are prompted to draw a line on top of the flange(s) to be bent.

1 You are prompted: Select flange position [Angle/Radius/Switch side/Toggle bend extents/ACcept] <Accept>:



Pick a point at the side of the line you want to move.

The selected part is bent dynamically.

2 (option) Type a value to define the bend angle, using the default bend radius.

3 (option) Select the **Radius** option.

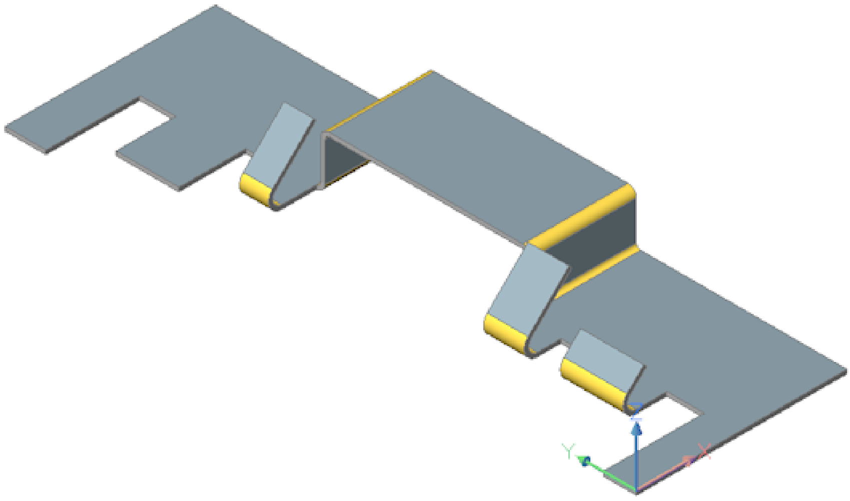
You are prompted: Enter bend radius [Back] <Back>:

Type a value to override the default bend radius and press **Enter**.

4 (option) Select the **Angle** option.

You are prompted: Enter bend angle [Back] <Back>:

Type a value to define the bend angle.



30.11.10 Connecting flanges

The SMFLANGECONNECT command closes gaps between 2 arbitrarily oriented flanges.

1 Launch the SMFLANGECONNECT command.

You are prompted: Select planar thickness faces of two flanges:

Flange faces under the cursor highlight.

2 Select the thickness face of the first flange.

You are prompted:

Entities/subentities in set: 1

Select planar thickness faces of two flanges:

3 Select the thickness face of the second flange.

The selected thickness faces are connected.

Note: After connecting 2 flanges, you can convert the junction feature to a bend.

30.12 Working with Bends and Junctions

Two adjacent flanges of sheet metal parts are connected by a bend or a junction. Different from many other mechanical CAD systems, bends and junctions are separate features in BricsCAD. A bend can be switched to a junction and vice versa, which allows you to easily fix or optimize the unfolded representation of a sheet metal part.

30.12.1 Converting Hard Edges to Bends or Junctions

If adjacent flanges of a sheet metal part are not connected by a bend or junction, the part cannot be unfolded or exported to a CAM system. Therefore, such hard edges must be converted to bends or junctions.

Converting hard edges to bends

1 Do one of the following:

- Hover over a hard edge and click the **Create Bend** () tool in the **Quad** menu.
- Click the **Create Bend** () tool button in the **Sheet Metal** ribbon tab.
- Choose **Create Bend** in the **Sheet Metal** menu.
- Type *SMBEND* in the Command line.

  You are prompted:

  Select hard edges or junctions or flanges or 3D solids or [Entire model/selection options (?)] <Entire model>:

2 Select a flange face or a hard edge.

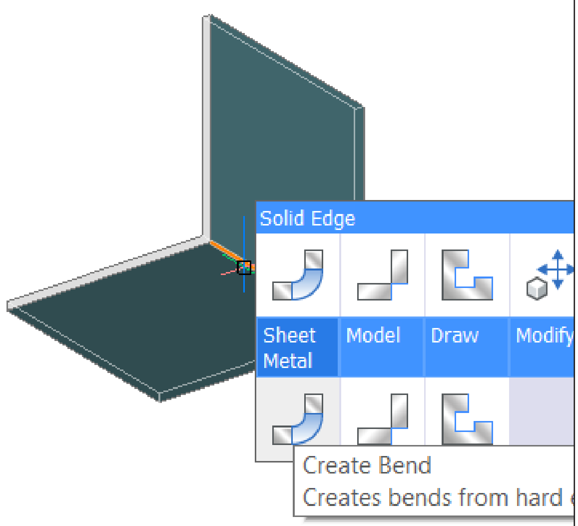
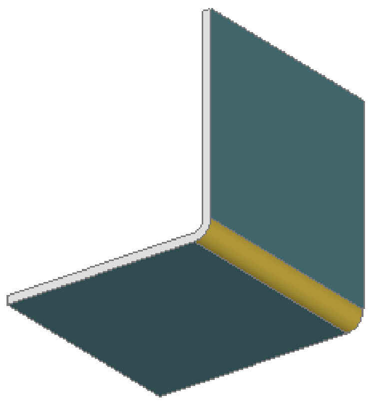
You are prompted:

Entities in set: 1

Select flange faces or hard edges:

3 Do one of the following:

- Select more faces or hard edges.
- Right-click or press **Enter** to create the bends or junctions on the selected items.

	
Selected a hard edge	A bend is created

Converting Hard Edges to Junctions

1 Do one of the following:

- Hover over a hard edge and click the **Create Junction** () tool in the **Quad** menu.
- Click the **Create Junction** () tool buttons in the **Sheet Metal** ribbon tab.

- Choose **Create Junction** in the **Sheet Metal** menu.
- Type SMJUNCTION in the Command line.

  You are prompted: Select hard edges or bends or flanges or 3D solids [Entire model/selection options (?)] <Entire model>:

2 Select a flange face or a hard edge.

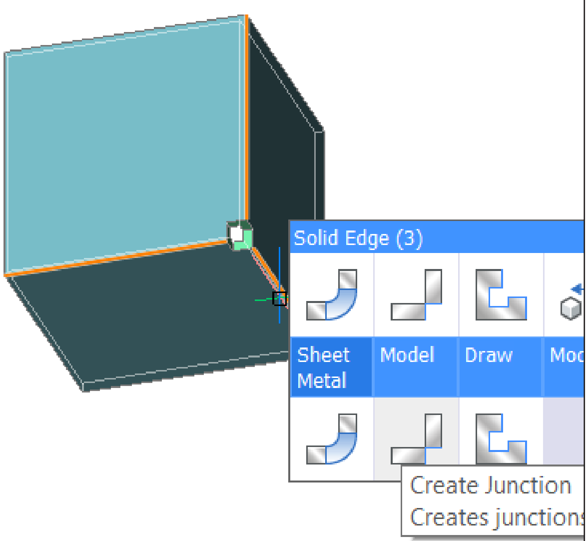
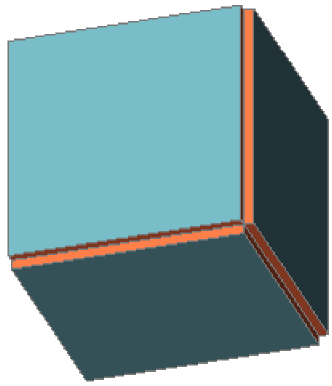
You are prompted:

Entities in set: 1

Select flange faces or hard edges:

3 Do one of the following:

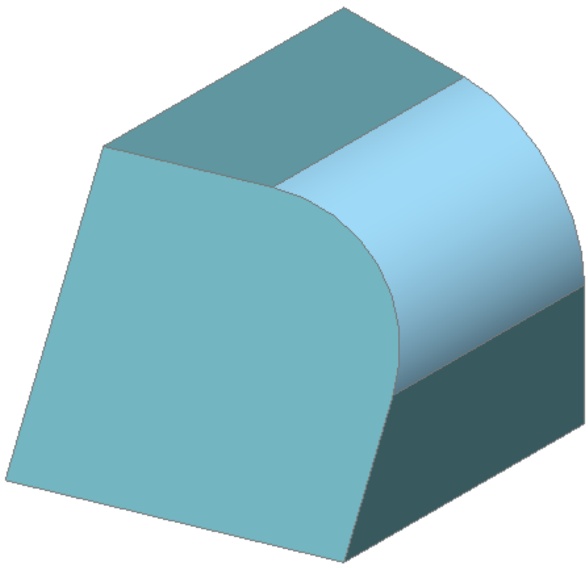
- Select more faces or hard edges.
- Right-click or press **Enter** to create the bends or junctions on the selected items.

	
Selected hard edges. Notice that a corner relief must be created first.	Junctions created. Notice that unnecessary material cuts have been removed due to **Smart Features**.

30.12.2 Creating Curved Junctions

Lets have a look at a more complex scenario where a junction has to be cut over a non-linear edge. To make the part unfoldable, you need to:

- Cut the bend relief for one of the linear hard edges.
- Cut the junctions for the remaining linear and curved hard edges.
- Create a bend on the remaining hard edge.



1 Create bend reliefs.

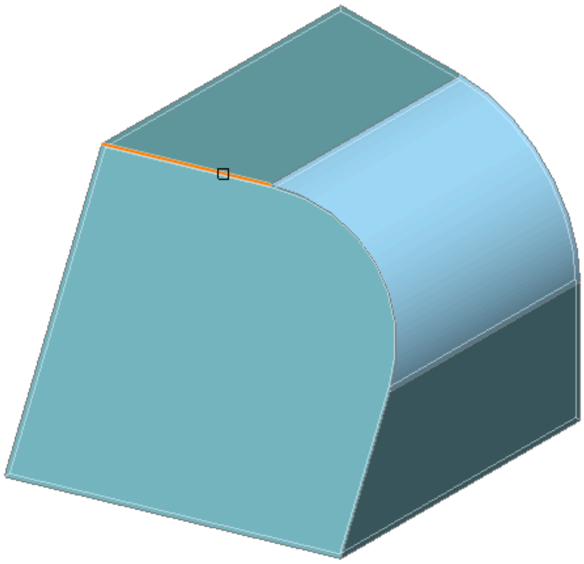
Do one of the following:

- Click the **Create Relief** tool button ( ) on the **Sheet Metal** ribbon tab.
- Choose **Create Relief** in the **Sheet Metal** menu.
- Type SMRELIEF in the Command line.

  You are prompted:

  Select a hard edge or bend face, flange face, 3D solid [Entire model/selection options (?)] <Entire model>:

2 Select a hard edge.



You are prompted:

Select another hard edge or bend face for the corner relief:

3 Press **Enter** or right-click.

You are prompted:

Enter relief size by bend radius ratio or [force Bend reliefs/Auto] <Auto>:

4 Press **Enter** or right-click.

2 bend reliefs are created for the edge:

2 bend reliefs are created because a hard edge side with a **free** vertex needs a relief in order to make orthogonal thickness face.

A bend relief on the side adjacent to the lofted bend is created because this feature is different from a regular bend: creating a corner relief there would be too big.

5 Create 2 junctions.

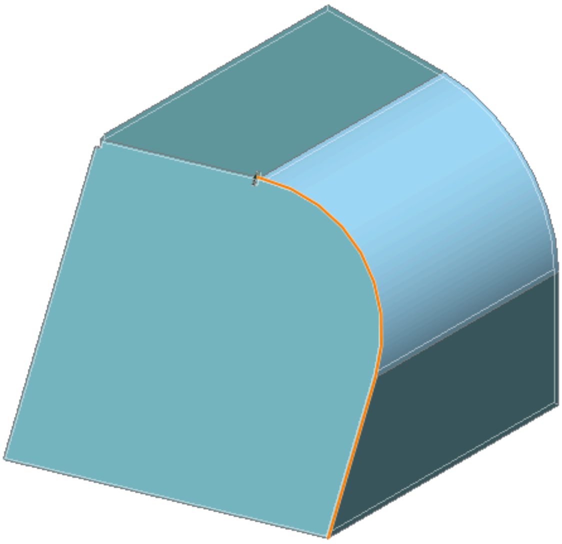
Do one of the following:

- Click the **Create Junction** () tool buttons in the **Sheet Metal** ribbon tab;
- Choose **Create Junction** in the **Sheet Metal** menu;
- Type SMJUNCTION in the Command line.

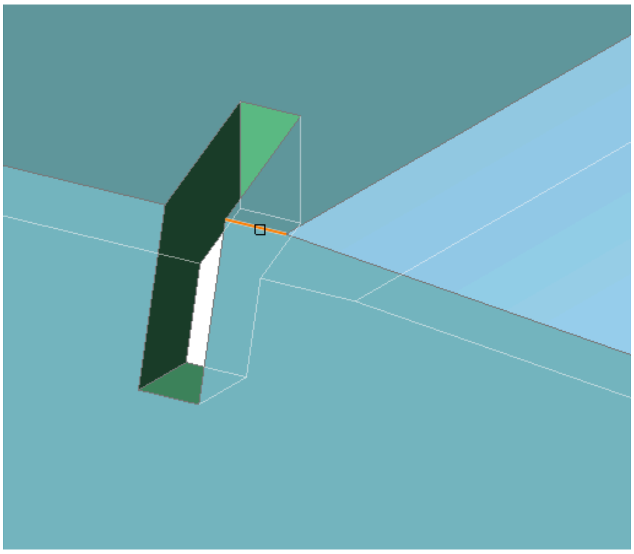
  You are prompted:

  Select hard edges or bends or flanges or 3D solids [Entire model] <Entire model>:

6 Select the following 3 hard edges:



Consider the small hard edge:



The cut is created. Parts which corresponded to linear edge get regular junction features.

7 Create a bend on the hard edge. Because it is the only hard edge left on the model, apply the command to the entire model.

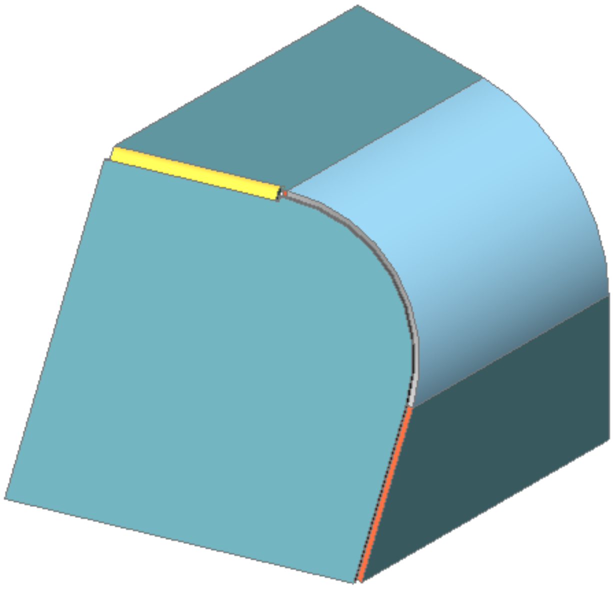
- Click the **Create Bend** ( ) tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Create Bend** in the **Sheet Metal** menu.
- Type SMBEND in the Command line.

  You are prompted:

  Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>:

8 Press **Enter**.

The bend is created.



9 Unfold the part.

- Click the **Unfold Body** ( ) tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Unfold Body** in the **Sheet Metal** menu.
- Type SMUNFOLD on the Command line.

  You are prompted:

  Select a flange or lofted bend face to start unfolding [Associative]:

10 Select some flange face.

You are prompted:

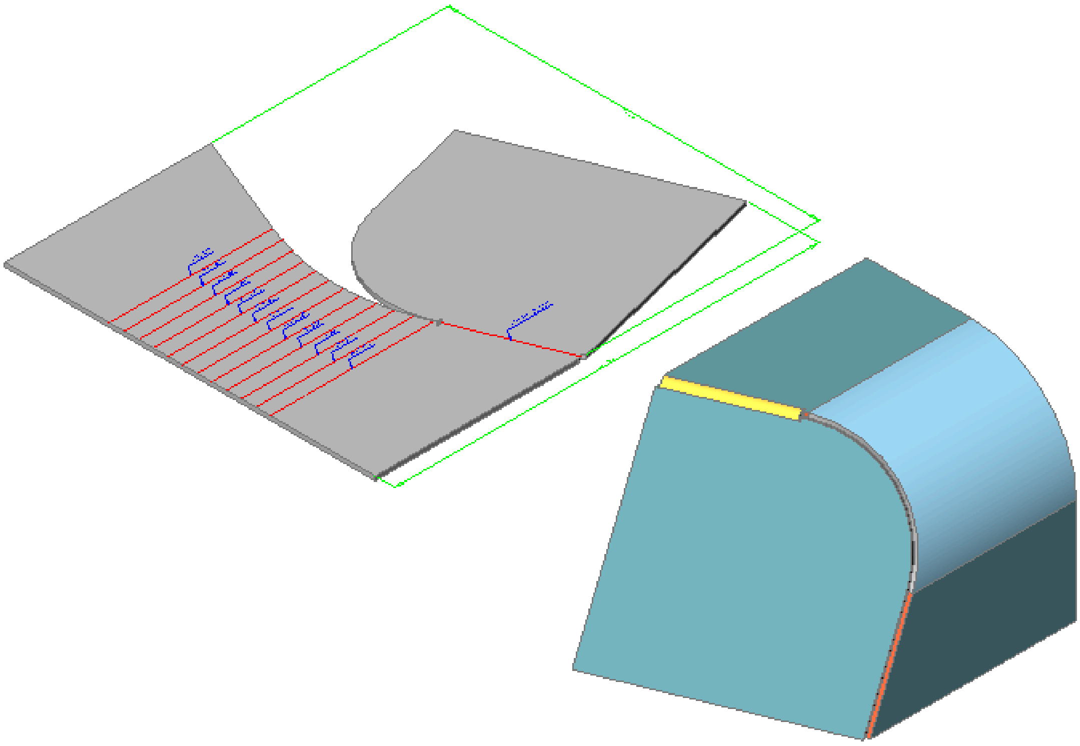
Select position of the unfolded body:

11 Place the unfolded part.

You are prompted:

Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>:

12 Press **Enter** to keep the result.



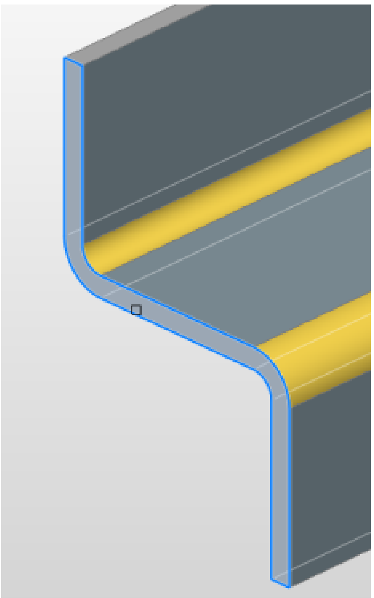
30.12.3 To Split a Thickness Face

To extend a flange which shares the same thickness face with other flanges, the thickness face must be split.

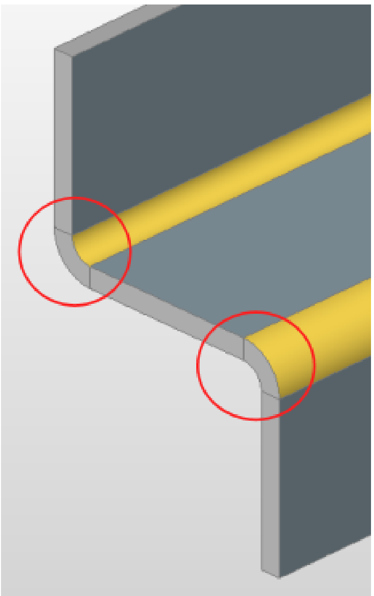
1 Launch the SMIMPRINT command ( ).

Prompts you: Select a thickness face:

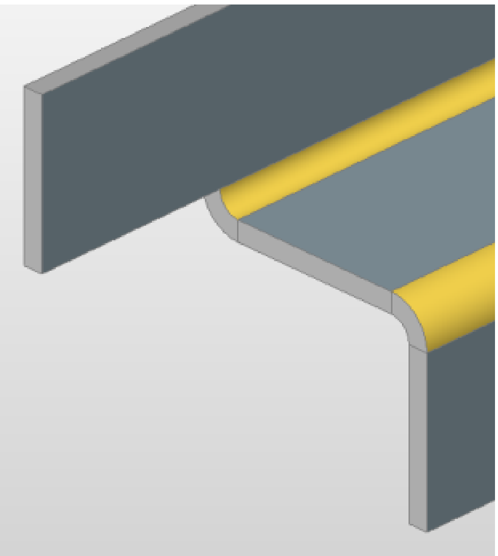
2 Select the shared thickness face.

3 

4 Right-click or press **Enter** to split the face.



5 The flanges can now be extended separately:

6 

7 Execute the DMSIMPLIFY command to remove the unneeded splits, thus unifying the coplanar thickness faces.

30.12.4 Changing the Bend Radius

To change the overall bend radius:

1 Select the root node in the **Mechanical Browser**.

2 Find the **Bend radius** group in the bottom part of the **Mechanical Browser**.

3 Do one of the following:

- Set **Type** to **Absolute Value**, then type the radius value in the **Value** field.

  The **Bend Radius** field changes accordingly.

- Set **Type** to **Thickness ratio**, then type a value in the **Ratio** field.

  The **Bend Radius** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

To change the bend radius for a particular bend:

1 Select the bend node in the **Mechanical Browser**. The selected bend highlights in the model. The **Bend properties** display in the bottom part of the **Mechanical Browser**.

2 Do one of the following:

- Set the **Type** to **Use global value**.

  The current global bend radius is applied to the selected bend.

- Set the **Type** to **Absolute value**, then type the radius value in the **Value** field.

  The **Radius** field changes accordingly.

- Set the **Type** to **Thickness ratio** and type a value in the **Ratio** field.

  The **Bend Radius** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

30.12.5 Changing the Junction Gap

To change the overall junction gap:

1 Select the root node in the **Mechanical Browser**.

2 Locate the **Junction gap** group in the bottom part of the **Mechanical Browser**.

3 Do one of the following:

- Set **Type** to **Absolute Value**, then type the gap value in the **Value** field.

  The **Junction gap** field changes accordingly.

- Set **Type** to **Thickness ratio**, then type a value in the **Ratio** field.

  The **Junction gap** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

To change the junction gap for a particular bend:

1 Select the bend node in the **Mechanical Browser**. The selected bend highlights in the model. The **Bend properties** display in the bottom part of the **Mechanical Browser**.

2 Do one of the following:

- Set the **Type** to **Use global value**.

  The current global junction gap is applied to the selected junction.

- Set the **Type** to **Absolute value**, then type the gap value in the **Value** field.

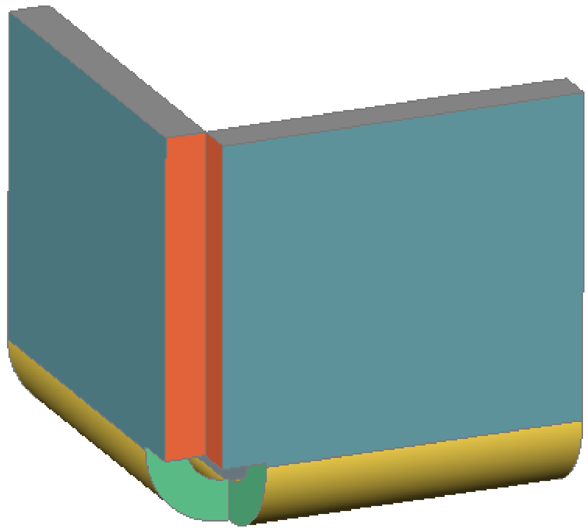
  The **Junction gap** field changes accordingly.

- Set the **Type** to **Thickness Ratio** and type a value in the **Ratio** field.

  The **Junction gap** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

30.12.6 Changing the Junction Type

By default, BricsCAD creates symmetric junctions, as shown in the image below:

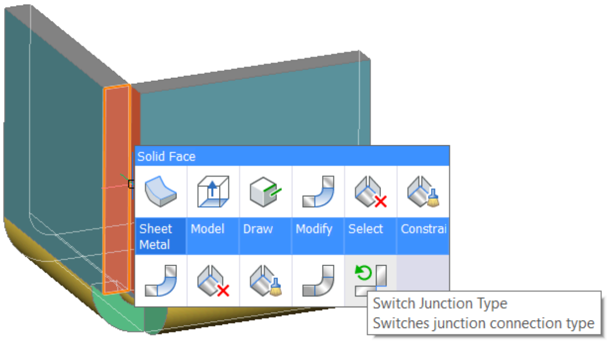


The SMJUNCTIONSWITCH command allows changing a symmetrical junction feature to one with overlapping faces.

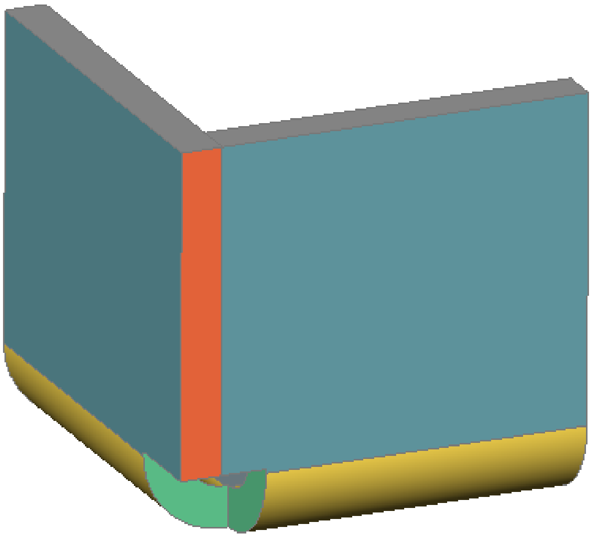
To change the junction type using the Quad menu:

1 Hover the cursor over one the thickness faces of this junction.

2 Select **Switch Junction Type** in the **Sheet Metal** command group in the **Quad** menu.



The selected thickness face is aligned with the flange face.



To change the junction type in the Mechanical Browser:

Method 1:

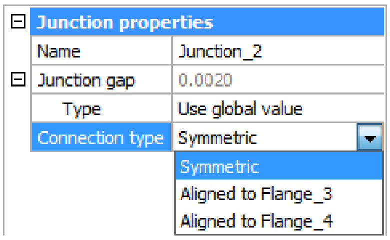
1 Select the corresponding junction in the tree.

The junction properties display in the bottom part of the **Mechanical Browser**.

2 Select the **Connection type** property.

3 Choose the desired junction type.

The options are: **Symmetric**, **Aligned to Flange_X** or **Aligned to Flange_Y**.

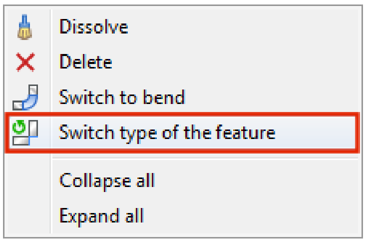


The junction is updated accordingly.

Method 2:

1 Select the corresponding junction in the tree.

A context menu displays:



2 Right-click and choose **Switch type of feature** in the context menu.

Note: Both symmetric and aligned junctions maintain the same gap value between connected flanges.

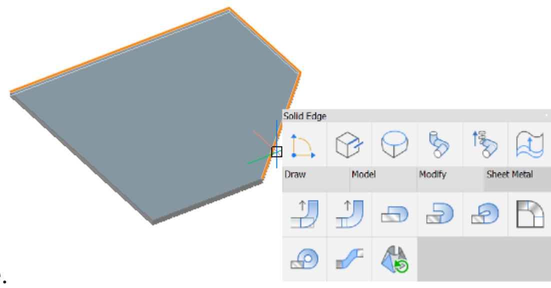
30.13 Working with Hems

A hem feature is produced when the edge of a sheet metal flange is folded back on itself. *SMHEMCREATE* simplifies the creation of sheet metal hems. In BricsCAD hems have their own set of properties depending on the type, "flange" and "bend" type of the geometry. Hem features are recognized by the *SMCONVERT* command and have a special symbol when unfolded with *SMUNFOLD*.

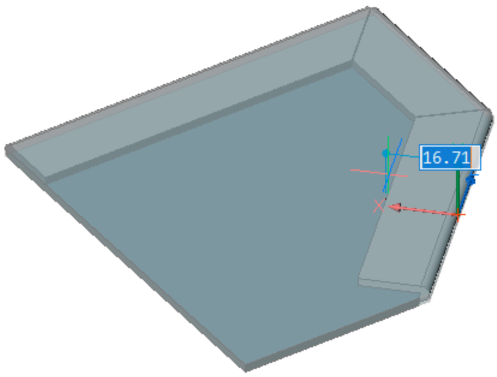
30.13.1 Create a Hem

1 Do one of the following:
- Select several linear edges on the flange.
- Hover the cursor over a linear edge of the flange.

2 Do one of the following:
- Select **Create a Closed Hem** () tool in the **Quad** menu.
- Click the **Create a Closed Hem** () tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Create a Closed Hem** in the **Sheet Metal** menu.

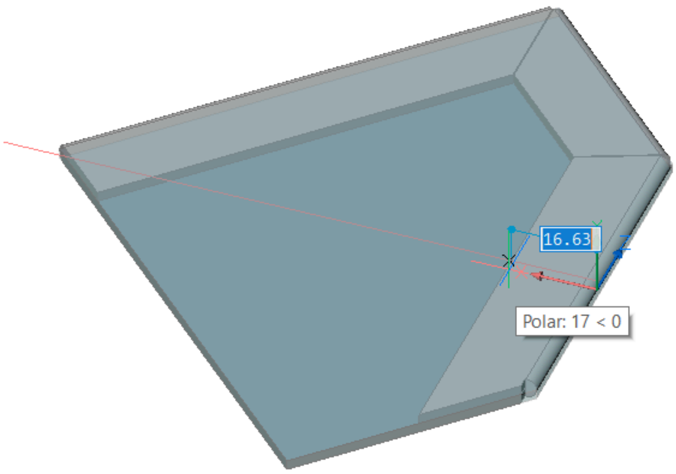


- Type *SMHEMCREATE* in the Command line.

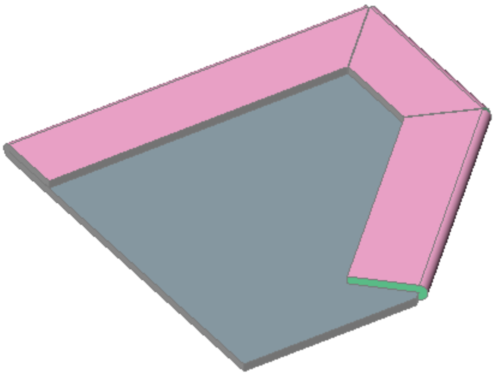


3 Hover the cursor to adjust the hem length:

4 Use the **Ctrl** widget or **Toggle tapering** option to flip the adjustment of a hem to the



edge:



5 When finished, press **Enter**:

Hems and auxiliary features (reliefs and miters) are created.

30.13.2 Modify Hem Parameters

To modify Hem parameters select the feature and use the **Mechanical Browser**. Different parameters are available, depending on the type of the hem.

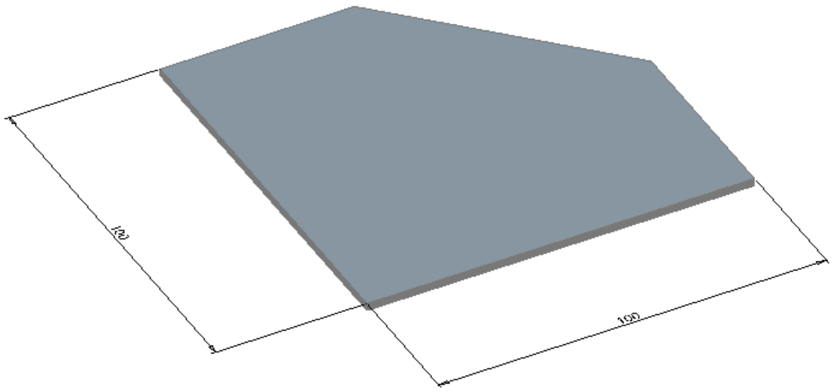
Type of hem	Available parameters
Closed	Radius Length
Open	Radius Length
Teardrop	Radius - change the angle of the hem Length - change hem length
Round	Radius - change the "planar" part of the hem

In the case of any parametric change, the hem type is preserved and the geometry is moved inside, to maintain external dimensions.

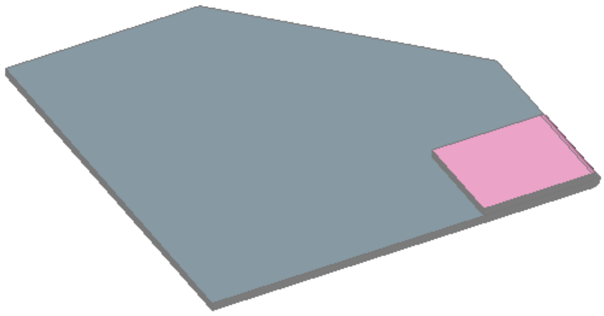
30.13.3 Erase a Hem

When a hem feature is erased, BricsCAD intelligently restores the geometry's dimensions and reliefs.

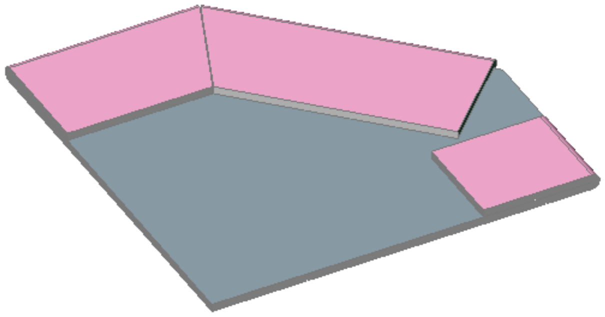
The flange part below has external dimensions equal to 100.



1 Use the Width option of the *SMHEMCREATE* command to create a partial hem on a single

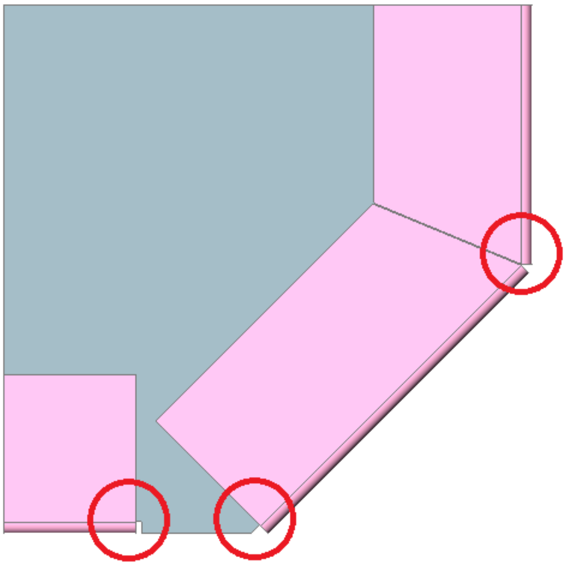


edge:

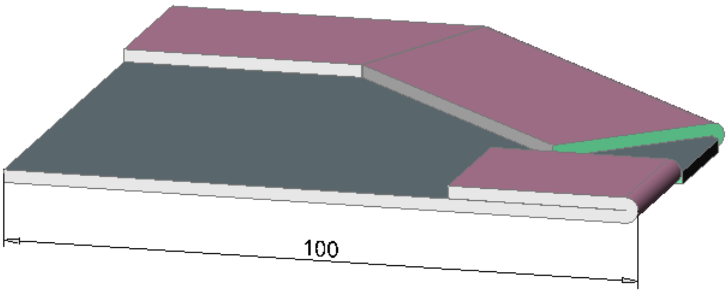


2 Create hems on 2 other edges:

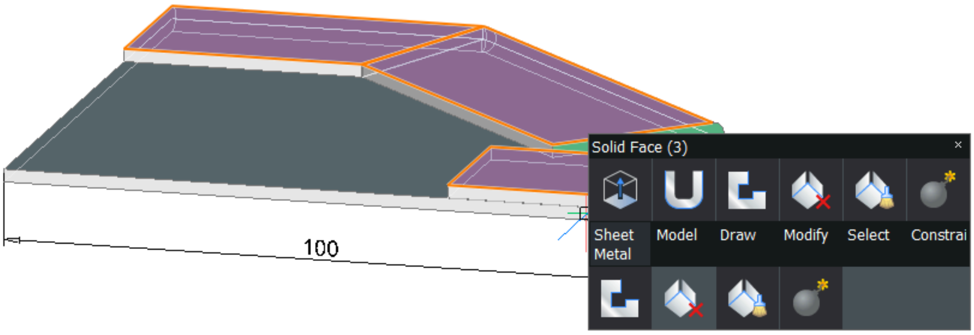
These operations lead to the appearance of complex relief geometry:



Note: The external dimensions of the part are preserved.

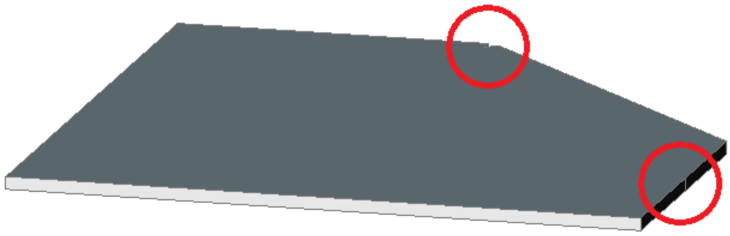


3 Select the hem features and delete

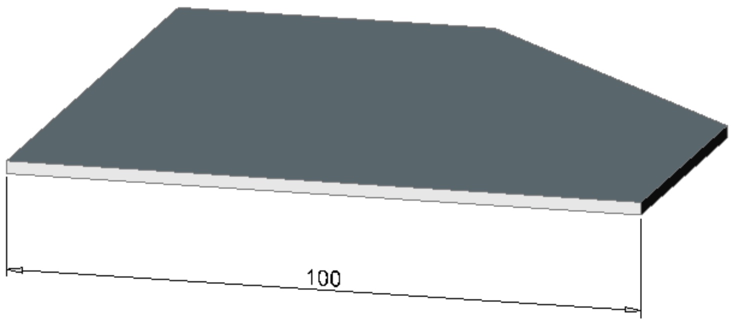


them.

Some imperfections are left on the corner relief feature and the imprinted edge:



4 Delete the corner relief feature and use *DMSIMPLIFY* to get the original part with original



dimensions.

30.13.4 Related commands

The *MECHANICALBROWSEROPEN* command displays the Mechanical Browser.

The *SMDELETE* command restores a hard edge between 2 flanges to remove a bend or a junction - removes a flange and all adjacent bends. The adjacent flanges are extended up to a junction configuration and the flange is deleted. Restores geometry removed by the miter.

The *SMHEMCREATE* command creates one or more hems to a sheet metal part by pulling one or more edges of an existing flange.

30.14 Working with Tabs

Tab features are used to create easy and robust connections between sheet metal parts.

Tab features are created from junctions.

30.14.1 Creating a Single Tab

A single tab is created by default by the SMTABCREATE command. This feature only has a single slot.

1 Do one of the following:

- Hover over a junction face and click the **Create single Tab** ( ) tool in the **Quad** menu.
- Click the **Create single Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create single Tab** in the **Sheet Metal** menu.
- Type SMTABCREATE in the Command line.

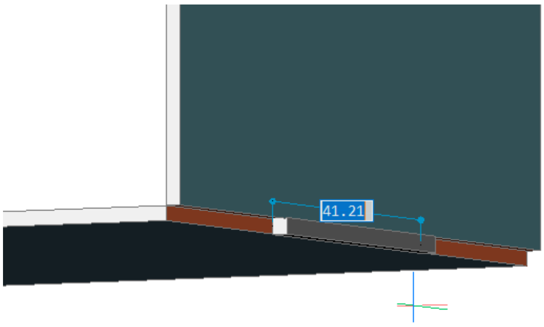
  You are prompted: Select junction to create single tab or [Array]:

2 Select a junction face.

You are prompted: Enter the length of the tab or [Central point]:

3 Define the length of the tab.

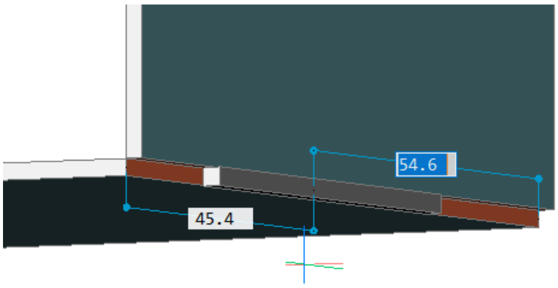
A tab is placed symmetrically in relation to a central point. Type values in the dynamic length entry fields to define the length of a tab.



4 Select the **Central point** option to position the tab.

You are prompted: Select the central point or [Length]:

2 Tab-controlled dynamic dimensions are used to create a tab relative to a junction:



5 Select the **Length** option to specify the length, or press **Enter** to finish the command.

30.14.2 Creating an Array Tab

An array tab feature has several slots, in contrast to Single tabs.

1 To create an array tab, do one of the following:

- Hover over a junction face and click the **Create array Tab** ( ) tool in the **Quad** menu.
- Click the **Create array Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create array Tab** from the **Sheet Metal** menu.
- Type SMTABCREATE in the Command line.

  You are prompted: Select junction to create single tab or [Array]:

2 Select the **Array** option.

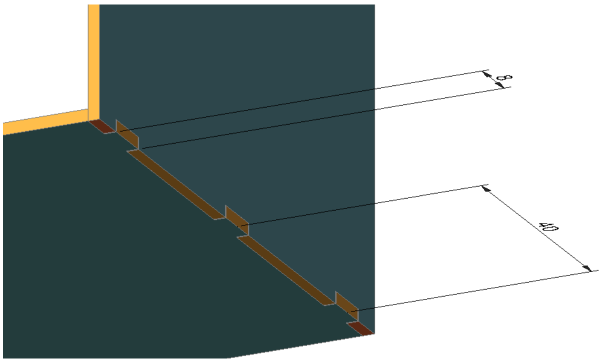
You are prompted: Select junction or curved hard edge to create array tab or [Single tab/selection options (?)]:

3 Select a junction face.

You are prompted: Accept tab configuration or [Flip/slot Number/Distance/Length/Round radius/CHamfer distance/Accept] <Accept>:

4 You can use options to modify the appearance of an array tab (see the example below).

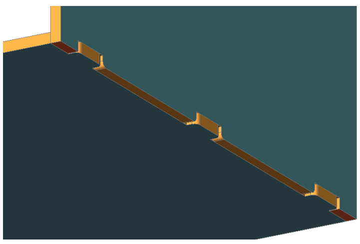
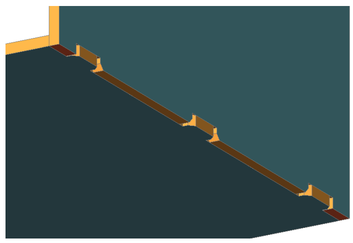
- **Slot Number = 3** - notice that the picture is not symmetric. If you click the **Flip** option, 3 slots will appear on the other side of the junction.
- **Length = 8** - the length of the slot.
- **Distance = 40** - the dimension between slot centers.



5 Change the **Round radius** and **Chamfer distance** to define the fillets and chamfers for the edges.

Tip: If you set fillets or chamfers by mistake, use zero values to turn them off.

6 Once you have set all the options, choose **Accept**, or press **Enter** to create the tab and finish the command.

Fillets	Chamfers
	

30.14.3 Creating a Curved Array Tab

This case is similar to creation of straight Array Tab, but the tabs are placed along a curved path.

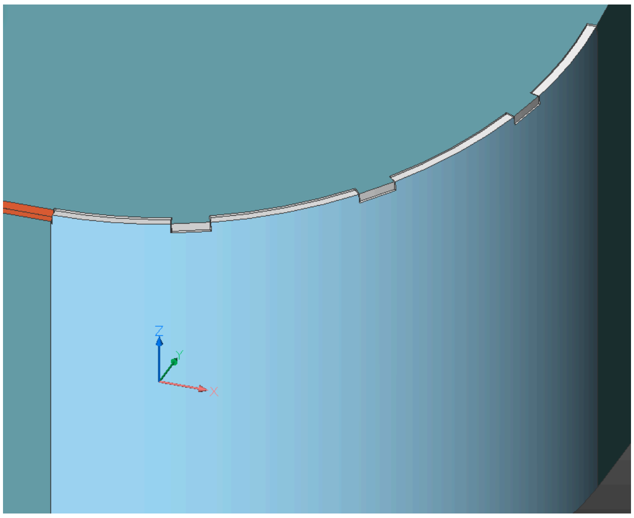
1 To create a curved array tab, do one of the following:

- Hover over a curved hard edge and click the **Create array Tab** () tool in the **Quad** menu.
- Click the **Create array Tab** () tool button in the **Sheet Metal** ribbon tab.

- Choose **Create array Tab** from the **Sheet Metal** menu.
- Type SMTABCREATE in the Command line.

  You are prompted: Select junction or curved hard edge to create array tab or [Single tab/ selection options (?)]:

2 Select a curved hard edge.

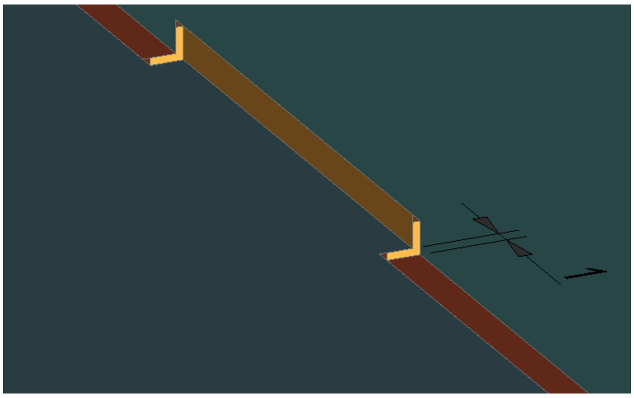


30.14.4 Modifying Tabs

Single and array tabs have different editable properties, which are available in the **Mechanical Browser**.

Click the tab feature name to edit.

A single tab has **Length**, **Clearance**, **Fillet radius**, and **Chamfer distance** parameters. **Tab clearance** defines the gap between the tab and the slot faces.



An array tab has **Slot number**, **Distance**, **Length**, **Clearance**, **Fillet radius**, and **Chamfer distance** parameters.

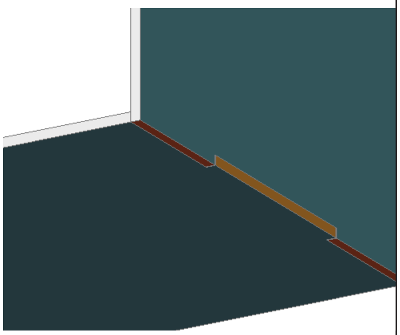
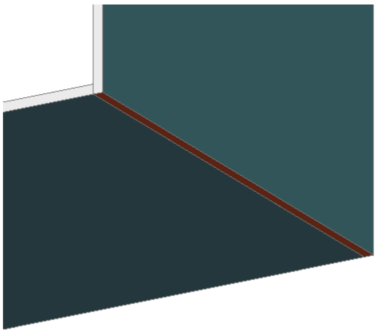
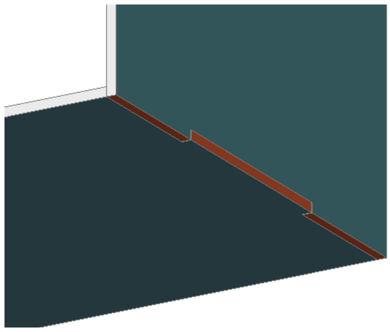
Note: Curved Array Tabs can not be modified.

30.14.5 Deleting Tabs

Use the SMDELETE command to delete tabs (not applicable for curved). A junction feature is created in place of the tabs.

30.14.6 Exploding Tabs

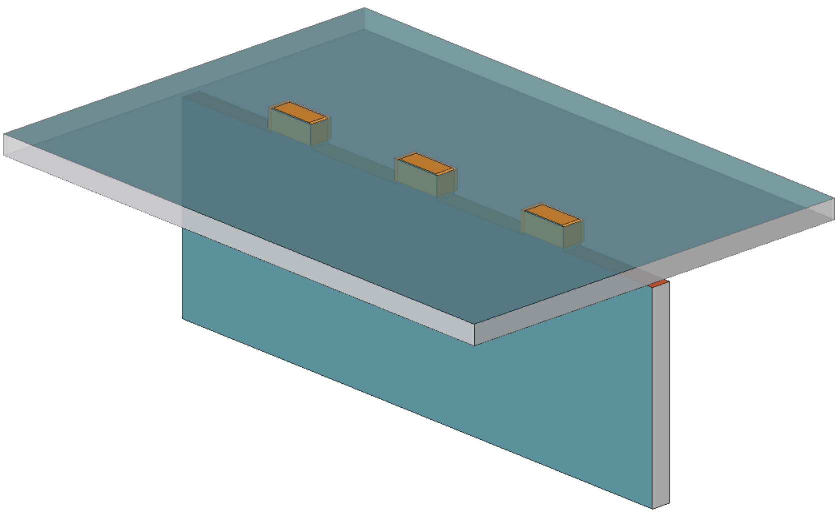
Use the SMEXPLODE command to explode a tabbed junction to a set of tab features. The tab geometry remains the same and its faces are distributed among several junction features.

Original tab feature	Result of deletion	Result of explode
		

30.15 Working with T-tabs and T-junctions

30.15.1 Overview

You can use T-tab features to create a connection with robust positioning between 2 sheet metal flanges. Unlike normal tabs, T-tabs form a T-like configuration, when one flange is attached to the other.



T-tabs can be created from T-junctions by the SMTABCREATE command. T-junction features are often used as an intermediate stage for making T-tabs.

30.15.2 Creating a T-junction

A T-junction can be created on existing separate flanges in the following way:

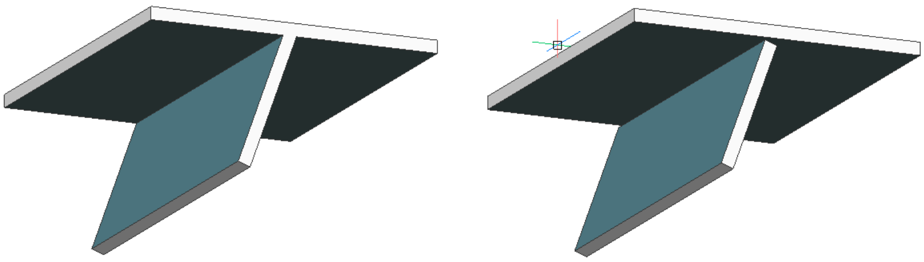
1 Select one thickness face on the flange.

2 Do one of the following:

- Click the **Connect Flanges** ( ) tool button in the **Sheet Metal | Junction** ribbon tab.
- Choose **Connect Flanges** in the **Sheet Metal | Junction** menu.
- Type SMFLANGECONNECT in the Command line.

3 Press **Enter**.

Alternatively, you can create a T-junction feature in place of a T-like hard edge between 2 flanges (image at the left):



1 Hover over a hard edge between the flanges and click the **Create Junction** ( ) tool in the **Quad** menu, or do one of the following:

- Click the **Create Junction** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create Junction** from the **Sheet Metal | Junction** menu.
- Type SMJUNCTIONCREATE in the Command line.

2 Select a hard edge.

30.15.3 Creating a Single T-tab

The SMTABCREATE command creates a single T-tab by default. This feature has only a single slot.

To create a single T-tab:

1 Hover over a T-junction face and click the **Create single Tab** ( ) tool in the **Quad** menu, or do one of the following:

- Click the **Create single Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create single Tab** from the **Sheet Metal | Tab** menu
- Type SMTABCREATE in the Command line.

2 Select a T-junction face.

30.15.4 Creating an Array T-tab

Different from Single T-tabs, array T-tabs features have multiple slots.

To create an array T-tab:

1 Hover over a T-junction face and click the **Create array Tab** ( ) tool in the **Quad** menu, or do one of the following:
   - Click the **Create array Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
   - Choose **Create array Tab** from the **Sheet Metal | Tab** menu.
   - Type SMTABCREATE in the Command line.

     You are prompted: Select junction to create single tab or [Array]: select **Array** option.

2 Select a T-junction face.

30.15.5 Modifying T-junction and T-tab Parameters

A T-junction has only the **Junction gap** parameter.

Array T-tabs and Single T-tabs have the following properties: **Distance**, **Length**, **Clearance**, **Fillet radius**, and **Chamfer distance**. Array T-Tabs also have the **Slot number** parameter.

You can set the value of all parameters in the **Mechanical Browser** panel.

30.15.6 Deleting a T-junction or T-tab

When a T-tab feature is deleted, BricsCAD restores a T-junction feature with the corresponding geometry. Erasing a T-Junction joins the flanges and creates a T-like hard edge between them.

30.15.7 Related commands

The MECHANICALBROWSEROPEN command displays the Mechanical Browser.

The SMDELETE command erases a feature.

30.16 Working with Bevels

30.16.1 Overview

Bevel features represent a chamfer operation made on Sheet Metal parts. On the level of BricsCAD V21 each bevel should be exactly on a single flange. There are no specific operations to create a bevel, so the DMCHAMFER command must be used on an edge of a flange, and then use DMROTATE to adjust the bevel angle. The SMCONVERT command recognizes the bevels and is processed by SMUNFOLD. Bevel features are also supported by the SMDELETE and SMREPAIR commands.

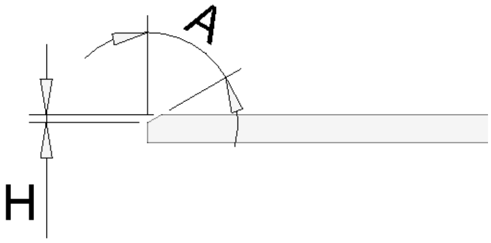
This procedure applies to 2 scenarios:

- Working with importing geometry, including batch export by SMASSEMBLYEXPORT.
- Modifying parts.

We distinguish between the following bevel types:

	One-side bevel
	Land bevel
	K-bevel
	X-bevel

In the **Mechanical Browser**, the properties of each chamfer of a bevel feature are available separately. These properties are:



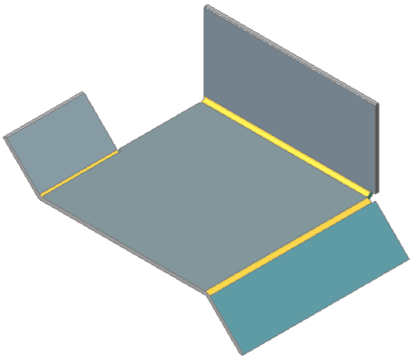
Depth (H): the depth of the chamfer operation, much like the DMCHAMFER command does.

Angle (A): the slope of the chamfered face.

Note: In the **Mechanical Browser** properties, Top or Bottom are not used to describe bevel features.

30.16.2 Recognizing bevels

Let us start with a sheet metal part with bevel geometry (chamfered flange edges).

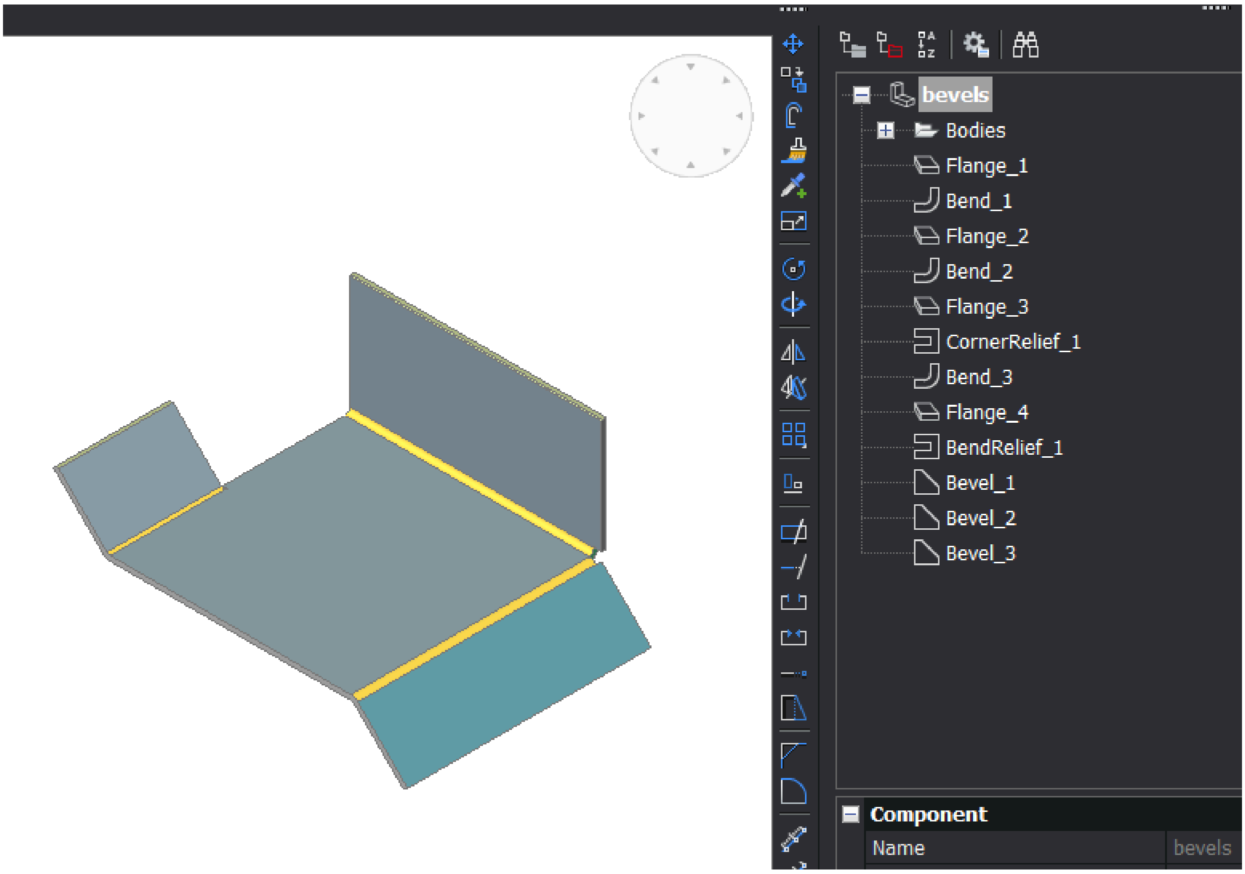


1 Click the **Convert to Sheet Metal** ( ) tool button on the **Sheet Metal / Create** Ribbon panel.

  You are prompted: Select 3D solids or seed faces or [Entire model/selection options (?)] <Entire model>:

2 Press **Enter** to convert the 3D solid to a sheet metal part.

The bevel features are recognized and appear in **Mechanical Browser** and show on the part:



30.16.3 Delete and Repair

The following round trip does not change the overall dimensions of the part: Create the bevel geometry and feature using DMCHAMFER/SMCONVERT, then delete the bevel using SMDELETE and restart the procedure.

When using SMDELETE to delete a bevel the volume is **always added** to the part because the dimensions of the part can not be reduced.

Support in the SMREPAIR command is also important, since this command is a part of the Import and Rework workflow.

Note: If you apply the SMREPAIR command to a part with chamfers but without bevel features, chamfers will be lost and dimensions will change, depending on the selected side of the part.

30.16.4 Unfold

Bevel features are unfolded according to the **Unfolding Mode** setting in the **Mechanical Browser**.

The options are:

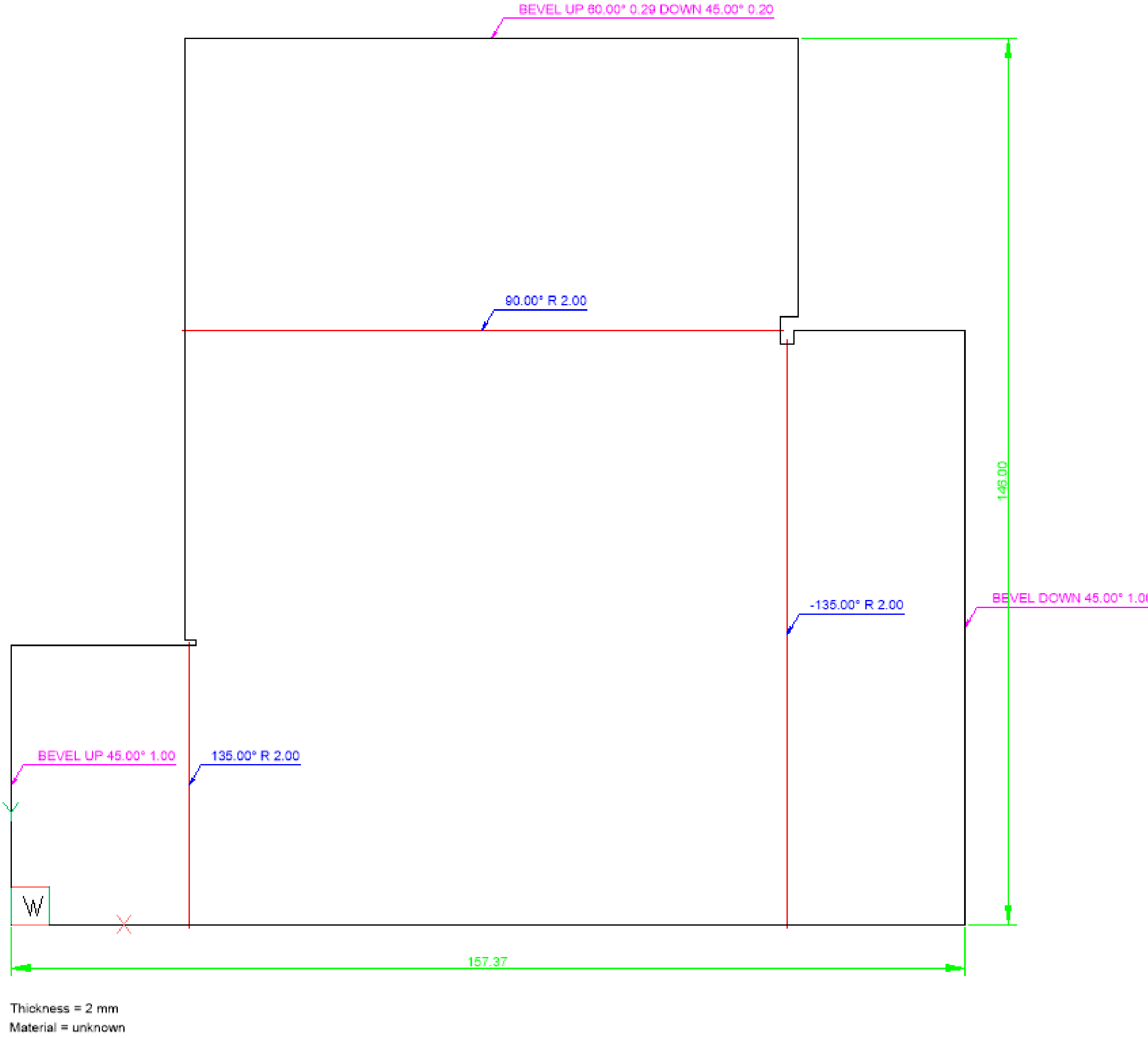
Keep: Does nothing with bevel geometry, resulting in a 3D solid with chamfers. These chamfered faces will likely produce artifacts in the 2D .DXF.

Remove: Same result as applying SMDELETE on all the bevel features: all the chops are filled.

Annotate: Each bevel feature is removed and the edge is annotated. This mode is recommended.

This is the result for the example part:

- Each bevel edge has been annotated with a pink text on the **Bevel Annotations** layer.
- UP and DOWN are with respect to the selected side for unfolding. When choosing the opposite side, all the UP annotations become DOWN, and vice versa.
- For **One-side** and **Land bevels** a single chamfer operation is needed only: a single UP or Down annotation is created.
- For **K-** and **X- bevels** 2 chamfer operations are needed: UP and DOWN annotations are created.



30.17 Creating and Splitting Lofted Bends

BricsCAD allows you to create or recognize lofted sheet metal parts with curvilinear geometry, which can be manufactured using multiple bending.

30.17.1 General Procedure to Create a Sheet Metal Part with a Lofted Bend

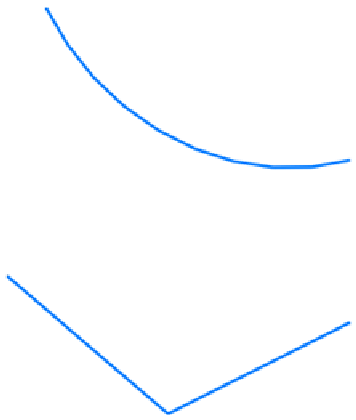
1 Launch the SMLOFT command.

2 Do one of the following:

- Click the **Create Lofted Bend** tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Create Lofted Bend** tool button ( ) on the **Sheet Metal** ribbon bar.
- Choose **Create Sheet Metal Loft** in the **Sheet Metal** menu.

  You are prompted: Select two cross sections:

3 Select 2 profile entities.

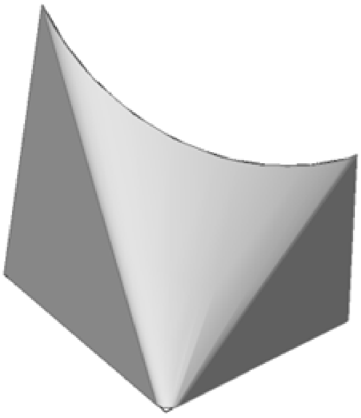


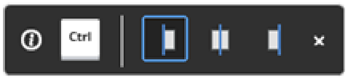
You are prompted: Set [fillets Radius/Thickness/Single bend/Flanges with bends] <Flanges with bends>:

4 Press **Enter** to accept the default settings.

A preview of the sheet metal part is created.

You are prompted: Set [fillets Radius/Thickness/Single bend/Flanges with bends/thicken Inside/Both sides/thicken Outside]:



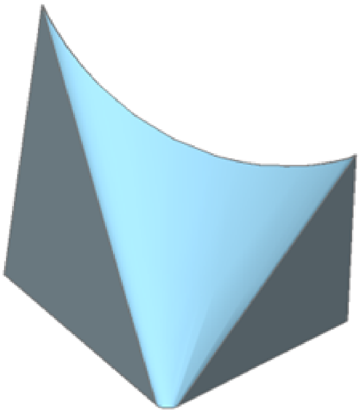


Note:

If the HKA field in the Status Bar is ON, the Hotkey Assistant widget displays.

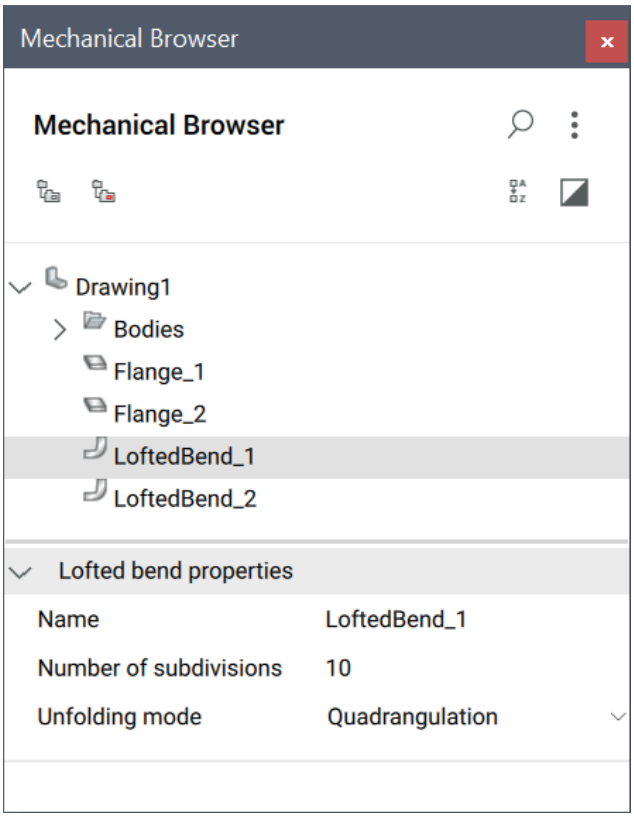
Press the CTRL key to choose the position of the sheet metal part thickness with respect to the source profiles: Thicken inside, Thicken to both sides or Thicken outside.

5 Press **Enter** to create the sheet metal part.



If **FEATURECOLORS** is ON, you can distinguish between the 2 flanges ( ) and the lofted bend ( ).

The properties of the lofted bend display in the **Mechanical Browser**:



Optionally select the lofted bend feature to edit its properties:

- **Number of subdivisions**: the number of flat regions in the unfolded representation of a lofted bend feature (the greater the number, the more bend lines will be added to the unfolded view for the lofted bend).
- **Unfolding mode**: most of lofted bends can be unfolded using an intelligent algorithm of quad tessellation. However, sometimes it might produce non-optimal results. In such cases set this property to **Triangulation** to switch to a more straight-forward algorithm, which creates triangle-like quads.

30.17.2 To Split a Lofted Bend

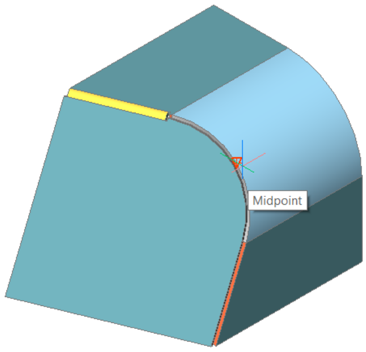
Lofted bends can split along a line on the lofted surface at an arbitrary point:

1 Do one of the following:
  - Click the **Split** tool ( ) on the **Sheet Metal** ribbon tab.
  - Choose **Split** in the **Mechanical** > **Sheet Metal** > **Split** menu.
  - Type SMSPLIT in the Command line.

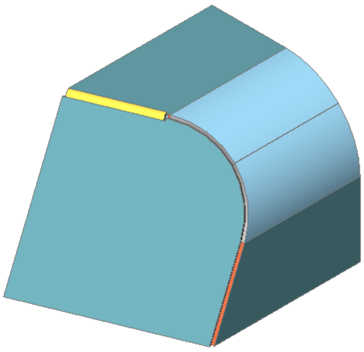
  You are prompted: Select lines or edges to split the flange or [SMart split/draw a New line/ selection options (?)]

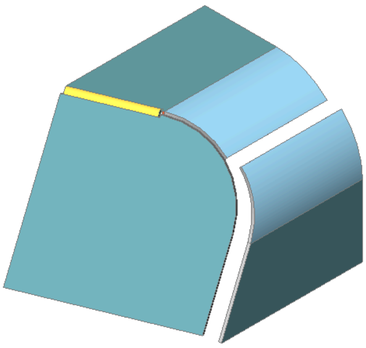
2 Choose **draw a New line** option.

3 Specify the start and end points of the line on the selected flange.



The lofted bend is split. If the connecting junction is dissolved, you can freely move the separated part.





30.18 Bend Relief and Corner Relief Feature Operations

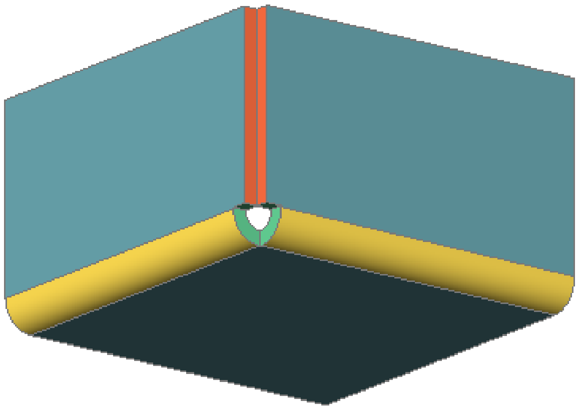
30.18.1 Overview

Use the SMFLANGEEDGE, SMRELIEFCREATE, SMJUNCTIONSWITCH, and SMRELIEFSWITCH commands.

Reliefs are needed in sheet metal parts to avoid material tear when bent on press breaks. Basically you must not care about reliefs, because BricsCAD automatically creates all needed reliefs when you create a bend (implicitly or explicitly). However, if you rework poor sheet metal parts, you may find it useful to create corner and bend reliefs with the dedicated tools.

30.18.2 Corner Reliefs and Junctions

When you create a flange using the SMFLANGEEDGE command, adjacent to a bend edge, a corner relief is created automatically in the corner where the 3 flanges meet. In addition, a junction is created between the 2 flanges, which are not connected with a bend.



30.18.3 Creating Missing Reliefs

1 Do one of the following:

- Click the **Create Relief** tool button () on the **Sheet Metal** ribbon tab.
- Choose **Create Relief** in the **Sheet Metal** menu.
- Type SMRELIEFCREATE in the Command line.

  You are prompted: Select a hard edge, bend face, flange face or 3D solid [Entire model]:

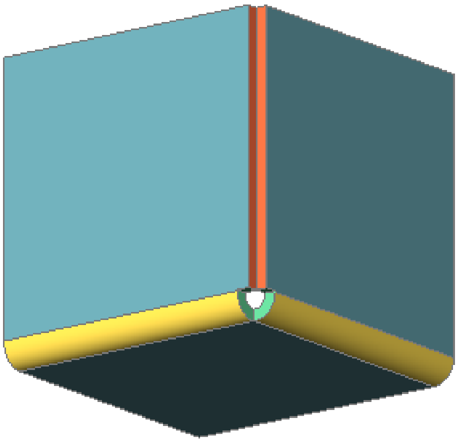
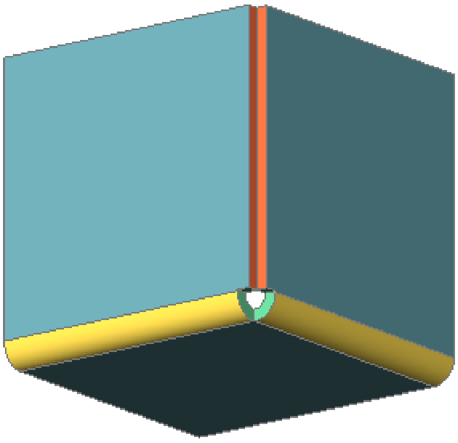
2 Select an item.

Enter relief size by bend radius ratio or [force Bend reliefs/Auto] <Auto>:

3 Do one of the following:

- Press **Enter** or right-click to accept the default bend radius ratio.
- Type a value at the command prompt, then press **Enter**.

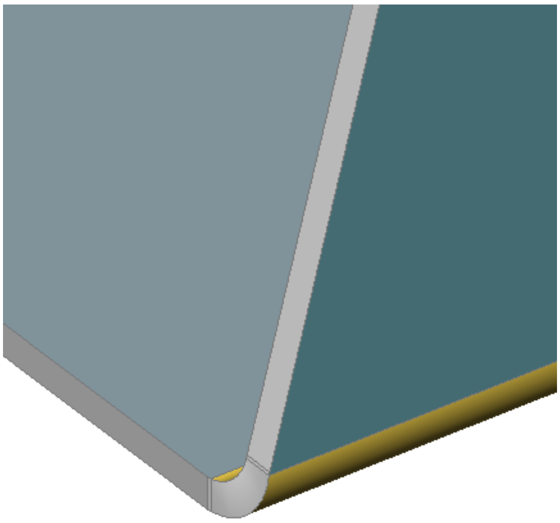
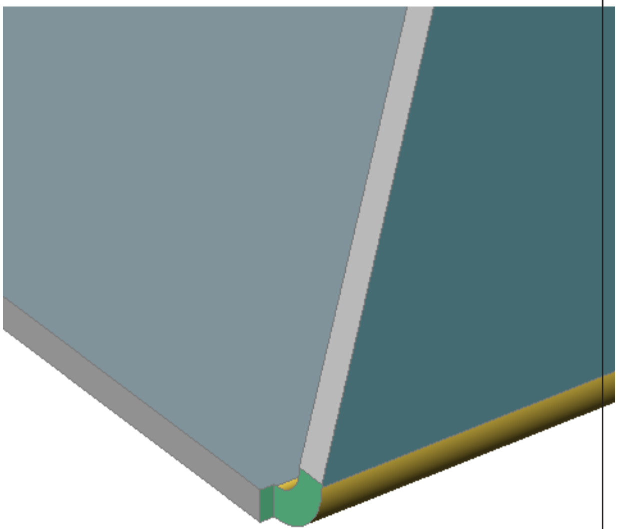
  The relief is created.

	
Selected bend faces	Created reliefs

Note:

- The selected items can be a 3D solid with sheet metal features, a face or a hard edge of the solid.
- To create a relief locally, you can select 1 or 2 hard edges, 1 or 2 bends or a bend and a hard edge.
- If a 3D solid is selected, the command recognizes the places where corner or bend reliefs are needed and creates them automatically.
- If 2 hard edges or 2 bends or a bend and a hard edge at a corner are selected, the command tries to create a corner relief.

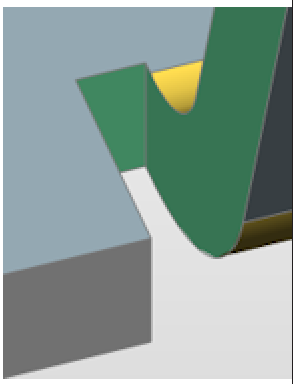
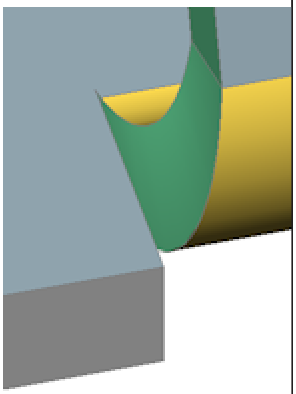
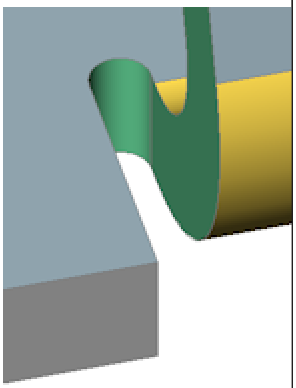
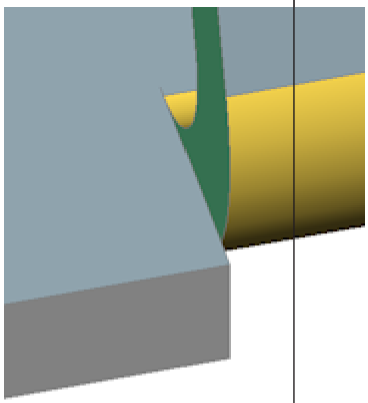
Option **force Bend reliefs** forces creation of the bend relief even if there is no need to create a bend relief. As shown in the image below, there is no need to create a bend relief, but the geometry is different from rectangular cut.

A bend relief is not needed because of a smooth transition of the geometry.	A rectangular bend relief is created only if the option **force Bend reliefs** is enabled.

30.18.4 Switching the Bend Relief Type

The bend relief type can be switched between 4 types:

			
Rectangular	Smooth	Round	Rip

Note:

- A **Rip** relief has a very small width of 0.001 * thickness. If you need to remove the relief completely, change its width to 0. This operation removes both the bend relief geometry and the bend relief feature.
- **Smooth**, **Round** and **Rip** bend relief types can be applied only if the current bend relief type is **Rectangular**.

Using the ribbon, menu or toolbar

1 To choose the desired relief type, do one of the following:
   - Click the **Relief** drop-down button in **Sheet Metal / Modify** panel on the ribbon and select **Switch to Rectangular**, **Switch to Smooth**, **Switch to Round** or **Switch to Rip**.
   - Click a tool button on the **SM Relief** toolbar: **Switch to Rectangular Relief** ( ), **Switch to Smooth Relief** ( ), **Switch to Round Relief** ( ) or **Switch to Rip Relief** ( ).
   - Choose the relief type in the **Sheet Metal / Relief** menu.

     You are prompted: Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>:

2 Do one of the following:
   - Press **Enter** to apply the selected relief type to the entire model.
   - Click a neighboring flange or bend or a face of the bend relief.

     The bend relief is converted.

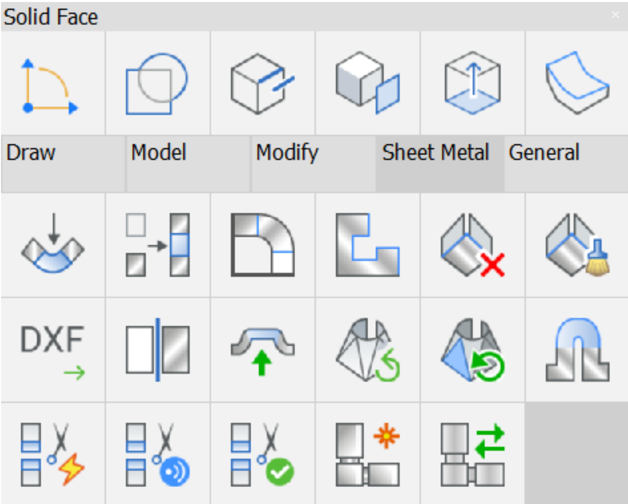
3 If the command has been called from the Ribbon, the command repeats the main prompt until you press the **ESC** key to stop.

Using the Quad

1 Hover over a neighboring flange or bend or a face of the bend relief.

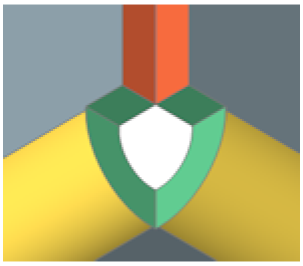
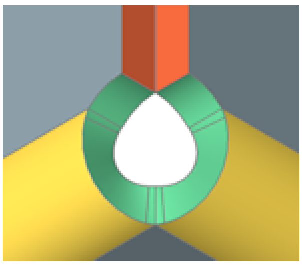
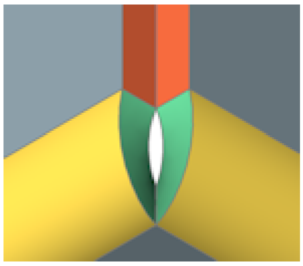
If the **Select Faces** option of the **SELECTIONMODES** system variable is selected, the flange or face highlights; otherwise hold the **CTRL** key down when hovering until the face highlights.

2 Select the desired relief type in the **Sheet Metal** tab in the **Quad** menu.



30.18.5 Switching the Corner Relief Type

The corner relief type can be switched between 3 types:

		
Rectangular	Circular	V-type

Using the ribbon, menu or toolbar:

1 To choose the desired relief type, do one of the following:
   - Click the **Relief** drop-down button in **Sheet Metal / Modify** tab on the ribbon and select a relief type.
   - Click a tool button on the **SM Relief** toolbar.
   - Choose the relief type in the **Sheet Metal / Relief** menu.

You are prompted: Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>:

2 Do one of the following:
- Press **Enter** to apply the selected relief type to the entire model.
- Click a neighboring flange or bend or a face of the corner relief.

  The corner relief is converted.

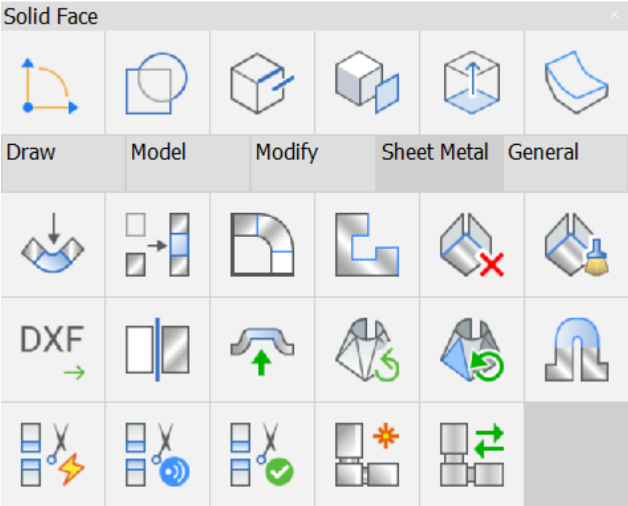
3 If the command has been called from the Ribbon, the command repeats the main prompt until you press the **ESC** key to stop.

Using the Quad:

1 Make sure the **Select Faces** option of the **SELECTIONMODES** system variable is selected.

2 Hover over a neighboring flange or bend or a face of the corner relief.

3 Select the desired relief type in the **Sheet Metal** command group in the **Quad** menu.



30.19 Working with form and bead features

30.19.1 Overview

Form and bead features are created by a forming tool. Bead features are produced by imprinting a wheel rolling over the flange following an arbitrary trajectory. We will use **form** to refer to both form and bead features. Form features can be recognized in imported geometry or inserted from built-in or user-defined libraries. Form features can be edited directly or parametrically. Form features are listed in the **Mechanical Browser** with their parameters. Parameters of form features are displayed and can be edited in the **Properties** panel when selected.

The following commands apply to form features:

Icon	Command	Description
	SMFORM	Convert a selected set of faces to a form feature. A drawing file with a user-defined form feature can be saved and then inserted with the BMINSERT command.
	SMCONVERT	Recognize form and bead features in imported geometry.
	SMREPLACE	Replace form features (including recognized ones) in sheet metal parts with form features from the built-in or user library.
	SMBEAD	Create associative Bead form features on sheet metal parts from 2D profiles. The feature is updated automatically when the defining profile has been modified.
	BMINSERT	Insert an existing mechanical component or sheet metal form feature into the current drawing.
	SMSELECT	Select comparable form features. The command works in 2 modes: • **Similar**: Features which have the same set of faces, but possibly different sizes. • **Same**: Identical features.

A library of parametrized sheet metal form features is available from the Form Features tab on the **Tool Palettes** panel. The library contains the most commonly used form feature types. To insert a form feature to a sheet metal 3D solid, drag the form feature icon from the **Tool Palettes** panel or the **Library** panel on a flange of the solid.

The SMDEFAULTFORMFEATUREUNFOLDMODE system variable controls the appearance of form features in 2D and 3D unfolded model representations.

30.19.2 Inserting a form feature

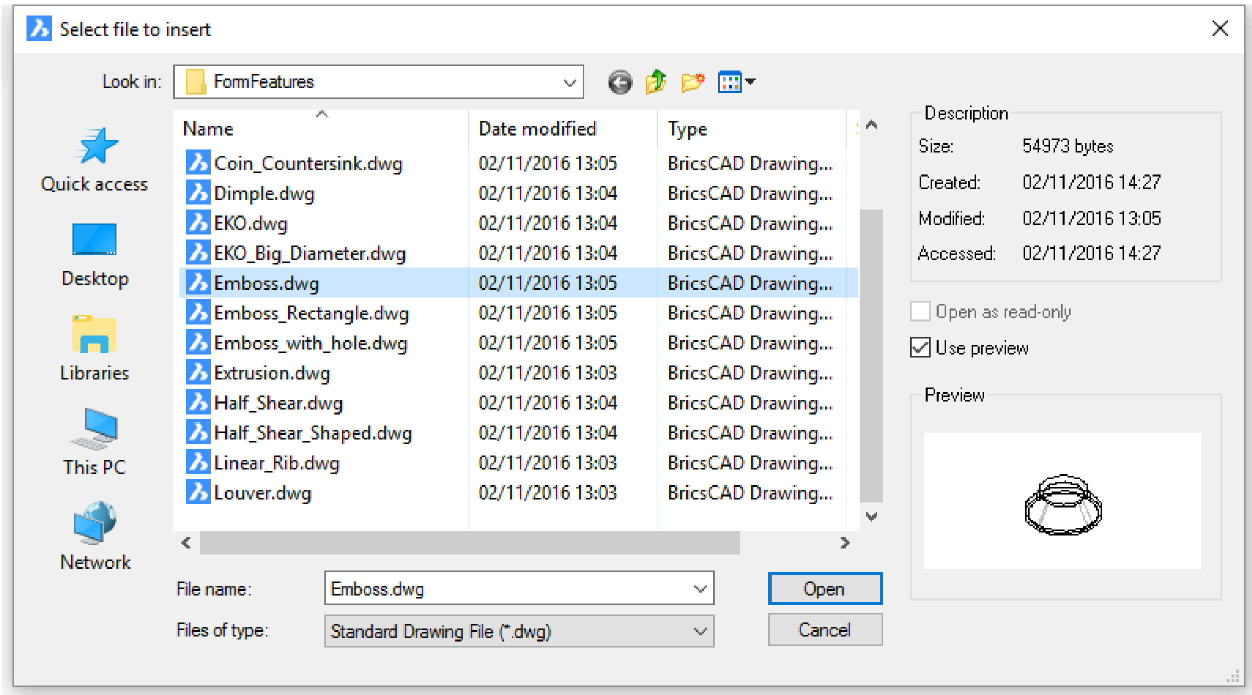
Form features are added to a sheet metal part by insertion on the existing flanges. The form features can be inserted with the BMINSERT command or by drag-and-drop from the **Form Features** tab of **Library**.

1 Make sure **Dynamic Dimensions** (DYN) and **Dynamic UCS** (DUCS) are active.

2 Do one of the following:

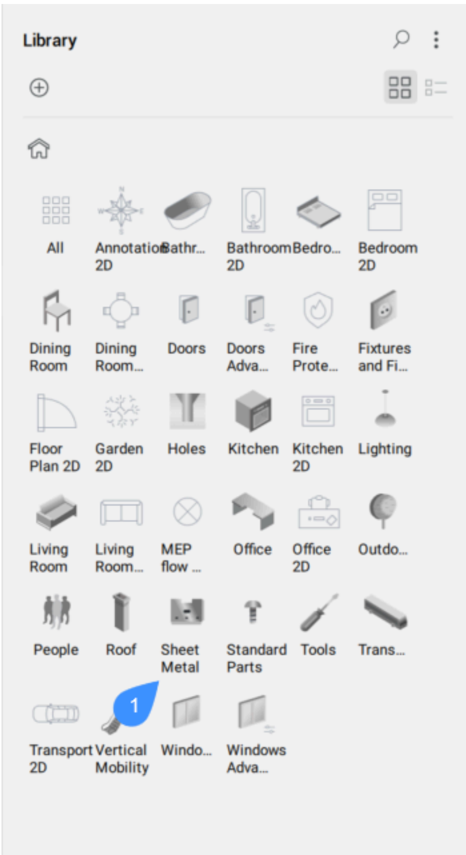
- Click the **Form Feature** tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Form Feature** tool button ( ) on the **Sheet Metal** ribbon bar.
- Choose **Insert Form Feature** in the **Sheet Metal** menu.
- Type BMINSERT in the Command line, then press **Enter**.

  The *Select file to insert* dialog box displays the **FormFeatures** folder *[BricsCAD Installation folder]\Support\DesignLibrary\SheetMetal\FormFeatures*:

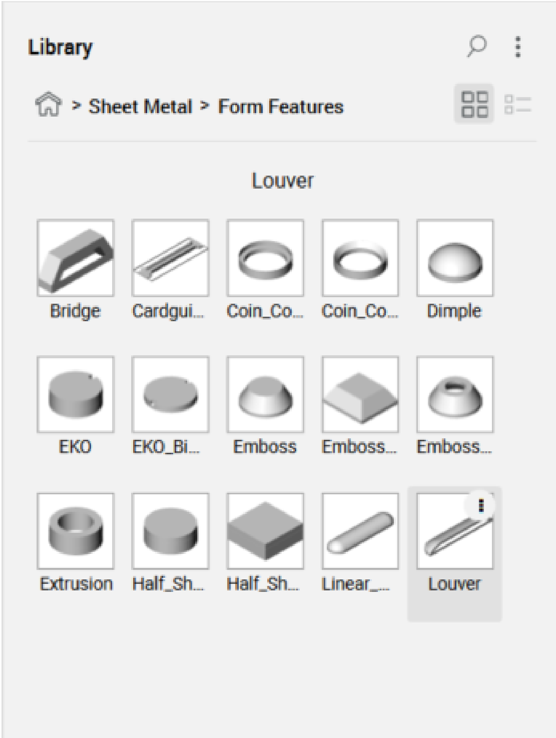


Select the desired form feature, then double-click the feature or click **Open**.

- Select the form feature from Form Features tab of **Library** panel:



- Select the form feature on the **Form Features** tab of the **Library** panel:



3 Move the cursor over the flange face you want to insert the feature in.

The dynamic UCS aligns to the selected face.

The orientation of the UCS depends on the edge the cursor enters the face.

Dynamic dimensions display from the origin of the UCS to the insertion point of the form feature.

You are prompted:

Select insertion point or [Edit inserted entity/Rotate component/set Base point/Name/insertion Type/Flip/mUltiple] <0, 0, 0>:

4 (Optional) Choose an option (see the BMINSERT command for more details):

- **Edit**: edits the parameters of the selected insert.
- **Rotate**: defines the rotation angle of the component.
- **Base Point**: specifies a different base point to insert the component.
- **Name**: defines the insert name of the component.
- **Insertion Type**: specifies the insertion type: Local or External.
- **Flip**: inserts the form feature in the opposite face of the flange.
- **Multiple**: inserts multiple inserts of the same component or create an array.

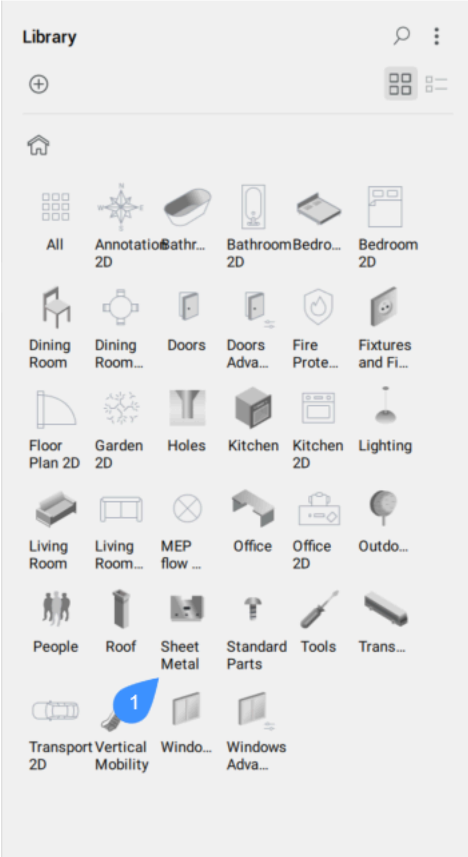
5 To insert a form feature do one of the following:

- Specify a point.
- Type a distance in the dynamic dimension fields. Press the **TAB** key to switch between the

entry fields.

6 Repeat step 5 to insert another copy, or press **Enter** to stop.

To have Form Features in the **Library** panel, create or use the **Mechanical Profile** in the **Get Started** dialog box.



30.19.3 Editing the parameters of a library form feature

Form features in the form feature library are parametric components. Several parameters control their dimensions. Parameters to control the Length, Width, Height and Radius of the form feature are named L, W, H and R correspondingly. The thickness of the form feature is automatically set equal to the thickness of sheet metal part.

The parameters of a single form feature can be edited either in the **Properties Panel** or in the **Mechanical Browser**. Parameters of multiple form features can be edited in the **Properties Panel** only.

Methods to select form features

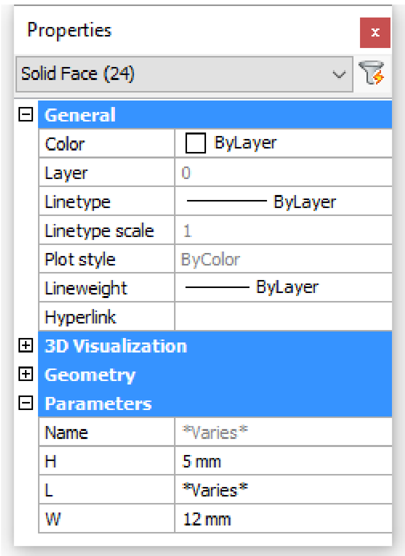
- Check the **Select Faces** option of the SELECTIONMODES system variable, then click the feature(s) in the model.
- Select the feature(s) in the **Mechanical Browser**, then press the space bar or right-click and choose **Select** in the context menu. Press and hold the Ctrl key to select multiple form features.

- A sheet metal part can have lots of different form features, which might complicate operations such as deletion or replacement, especially if only a selection of features, that meet certain conditions, needs to be processed. To select form features more easily, the SMSELECT command allows to select form features that are either identical to a selected feature or similar.

Editing parameters in the **Properties** panel.

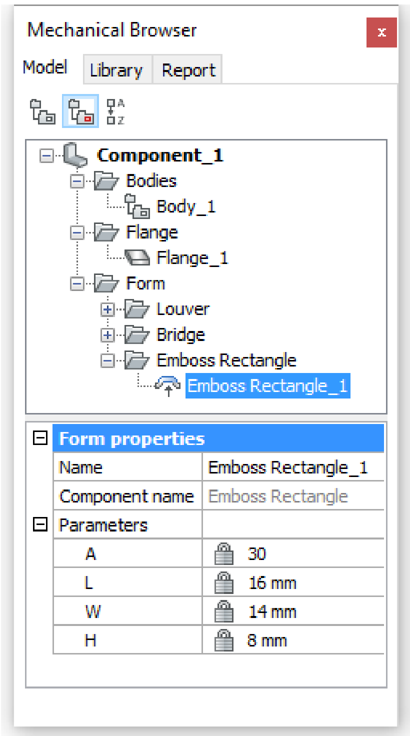
1 Select the form feature(s).

2 Select a parameter in the **Parameters** node in the **Properties** panel.



Editing parameters in the Mechanical Browser

1 Select the form feature in the **Form** node of the component structure tree.

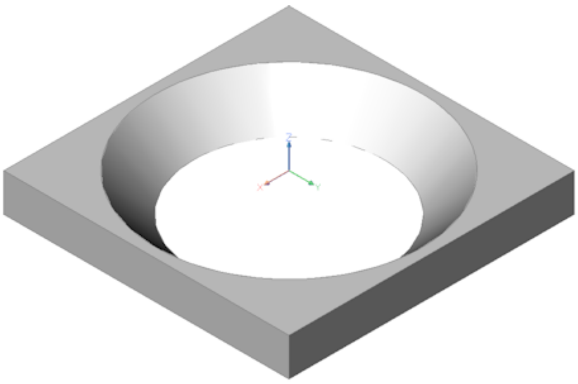


2 Select a parameter in the lower panel of the **Mechanical Browser** panel and type a new value.

30.19.4 Creating a custom form feature

1 Open a new drawing using **Mechanical-mm.dwt** as the template.

2 Create the 3D geometry of the form feature.



3 Set the SMCONVERTPREFERFORMFEATURES system variable to 1.

4 Run the SMCONVERT ( ) command.

5 Run the BMMECH ( ) command.

You are prompted: Name for mechanical component <Component_1>:

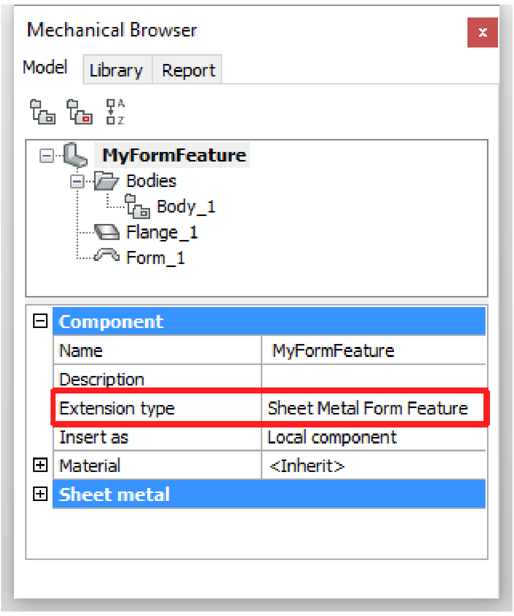
6 Type a name for the form feature.

You are prompted: Convert blocks and external references to mechanical components? [Yes/No]

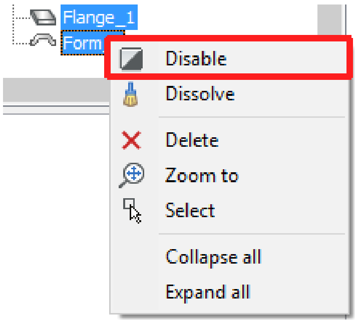
<Yes>:

7 Select **No**.

8 In the **Extension type** field in **Mechanical Browser**, select **Sheet Metal Form Feature**.



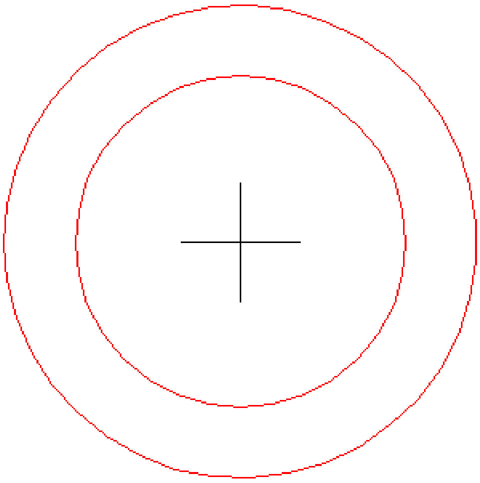
9 Select the Flange_1 and Form_1 features in the **Bodies** node, then right-click and select **Disable** in the context menu.



10 Create 2 new layers: UNFOLD_SYMBOL_UP and UNFOLD_SYMBOL_DOWN.

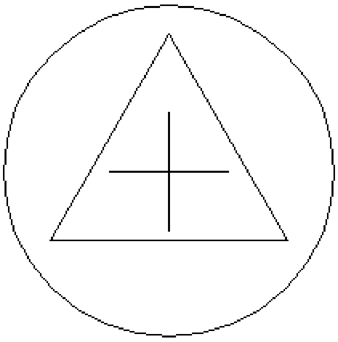
11 Turn layers 0 and UNFOLD_SYMBOL_DOWN **Off** and set layer UNFOLD_SYMBOL_UP **Current**.

12 Create the unfold symbol, which will be placed on the face the form feature is placed in.



13 Turn **Off** the UNFOLD_SYMBOL_UP layer and turn **On** the UNFOLD_SYMBOL_DOWN layer. Set the UNFOLD_SYMBOL_DOWN layer **Current**.

14 Create the unfold symbol, which will be placed on the opposite face the form feature is placed in.



15 Turn **On** layer 0 and turn the UNFOLD_SYMBOL_UP and UNFOLD_SYMBOL_DOWN layers **Off**. Set layer 0 as **Current**.

16 Save the drawing.

30.19.5 Creating a custom parametric form feature

We will now parametrize the custom form feature we created in the previous procedure.

2 parameters are needed:

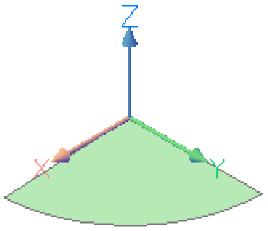
- **D** for a diameter of the bigger edge of the conical hole.
- **A** for an angle of the face of the conical hole.

In Sheet Metal, the default parameter **_SM_THICKNESS** controls the thickness of the sheet metal part.

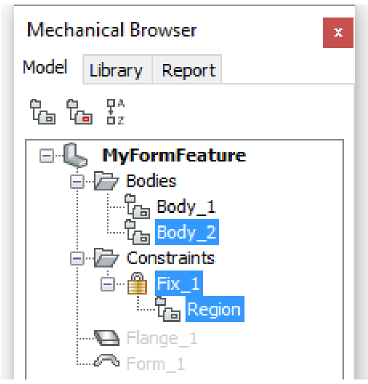
1 Create an auxiliary body for constraining on a new CONSTRUCTION layer.

Make this layer current and switch off all other layers.

2 Create a **Region** in the XY plane of the WCS on the CONSTRUCTION layer.

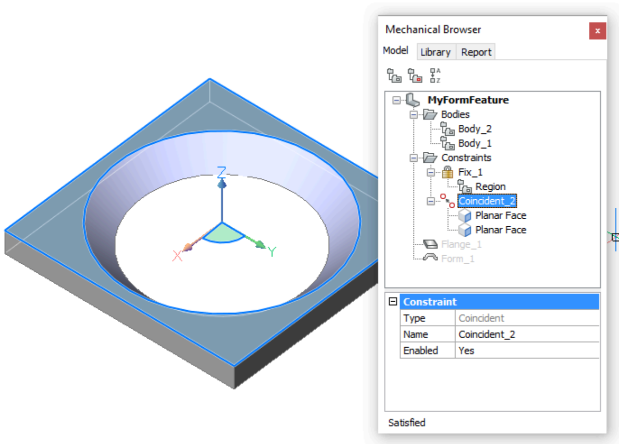


3 Apply a DMFIX3D constraint ( ) to the region.



4 Switch **ON** layer 0.

5 Apply a DMCOINCIDENT3D constraint ( ) between the region face and the flange top face.

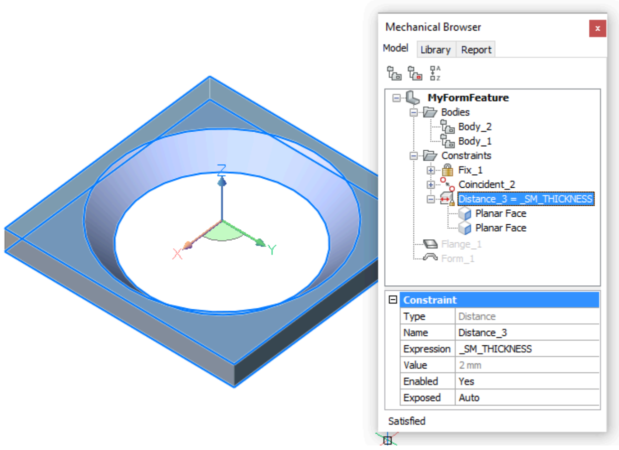


6 Apply a DMDISTANCE3D constraint ( ) between the flange top and bottom faces.

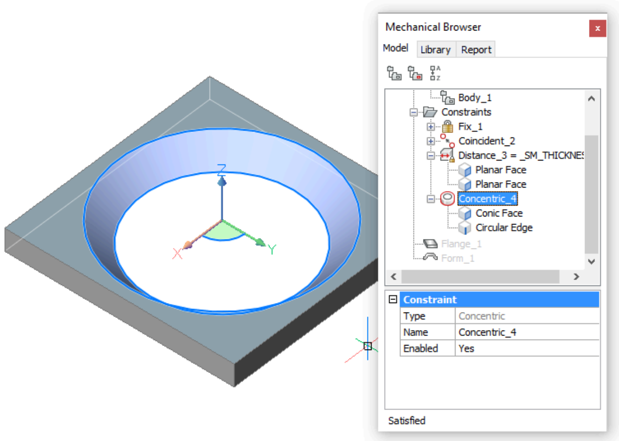
First select the top face, then press the Tab key to select the bottom face.

Accept the default value in the dynamic dimension.

7 Type _SM_THICKNESS in the **Expression** field of the Distance_3 constraint.



8 Since the Form Feature is conical, you can modify the flange from parallelepiped to cylinder.

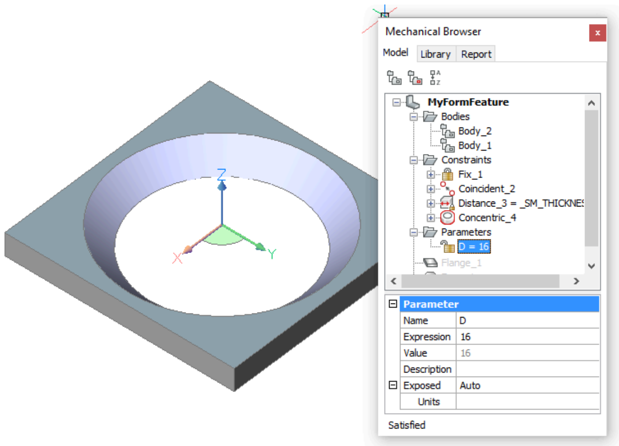


9 In the **Mechanical Browser**, right-click the feature name and select **Add new parameter** from the context menu.

Rename new parameter to **D**.

Select mm for the **Units** field.

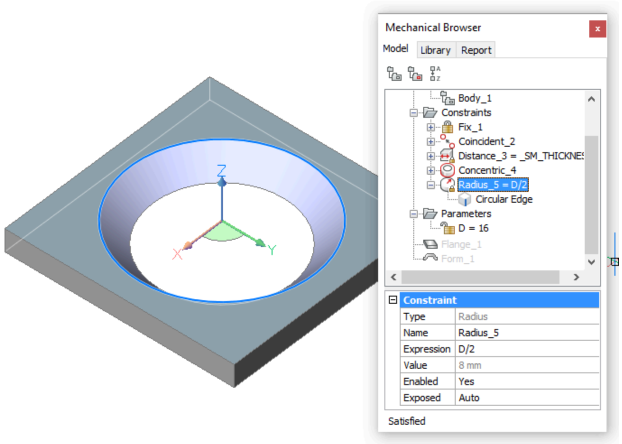
Type 16 in the **Expression** field.



10 Apply the DMRADIUS3D constraint ( ) to the top edge of the conical face.

Accept the default value in the dynamic dimension.

11 Type D/2 in the **Expression** field of the Radius_5 constraint.



12 In the **Mechanical Browser**, right-click the feature name and select **Add new parameter** from the context menu.

Rename new parameter to **A**.

Leave the **Units** field empty.

Type 45 in the **Expression** field.

13 Do one of the following to create a **Cone Angle constraint** to the conical face:

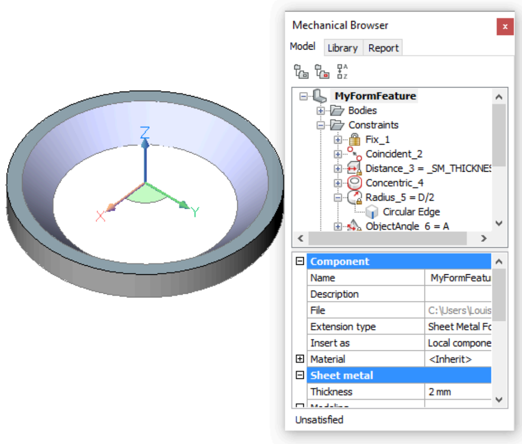
- Launch the DMANGLE3D command ( ). Choose the **Cone Angle constraint** option, then select the conical face.
- Hover over the conical face, then choose **Cone Angle constraint** ( ) in the **Constraints** command group in the **Quad** menu.

Accept the default value.

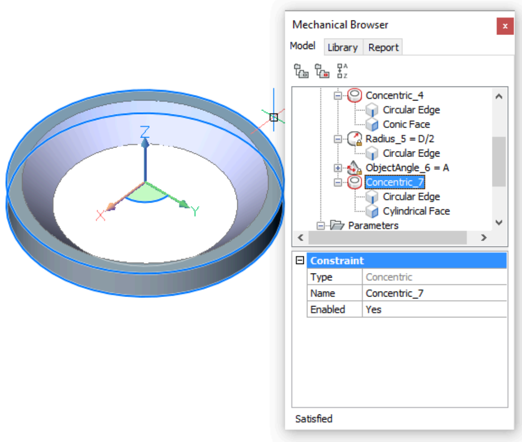
14 Type A in the **Expression** field of the ObjectAngle_6 constraint.

15 As the Form Feature is conical, modify the flange from parallelepiped to cylinder:

- Draw a circle on top of the flange. The radius of the circle is 9 mm.
- Select **Enable Boundary Detection** in the **Settings** panel on the **Home** tab in the Ribbon.
- Place the cursor over the area between the circle and the square edge of the flange, then choose **Extrude** ( ) in the **Quad** menu and move the cursor downwards.



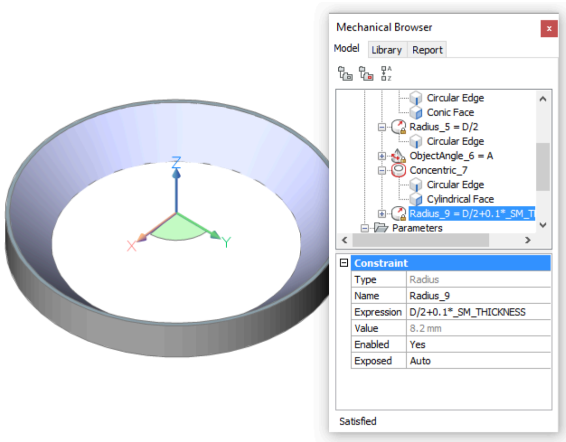
16 Apply the DMCONCENTRIC3D constraint ( ) between the circular edge of the region and the cylindrical face.



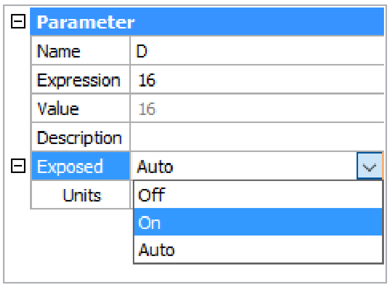
17 Apply the DMRADIUS3D constraint ( ) to the cylindrical face.

Accept the default value in the dynamic dimension.

18 Type D/2+0.1*_SM_THICKNESS in the **Expression** field of the Radius_8 constraint.



19 Set the **Exposed** field of both parameters A and D to **On**.



Create 2D constraints for the unfold symbols.

20 Make layer UNFOLD_SYMBOL_UP **current** and switch **Off** all other layers.

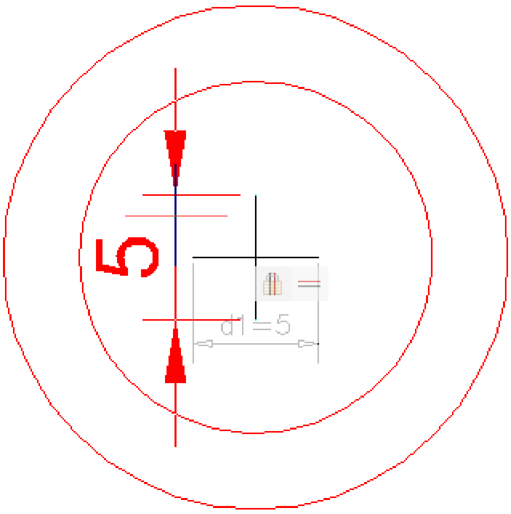
21 Apply the GCFIX constraint ( ) to both crossing lines indicating the center of the circles.

22 Apply the GCHORIZONTAL constraint ( ) to the horizontal center line.

23 Apply the GCVERTICAL constraint ( ) to the vertical center line.

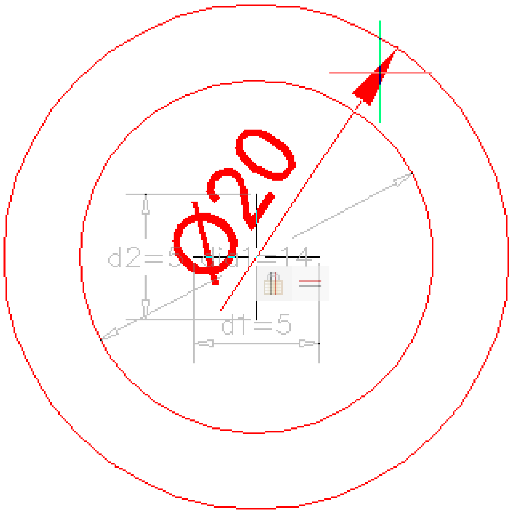
24 Apply the GCCONCENTRIC constraint ( ) between the 2 circles.

25 Apply the DCLINEAR constraint ( ) between the endpoints of the center lines. Accept the default value.



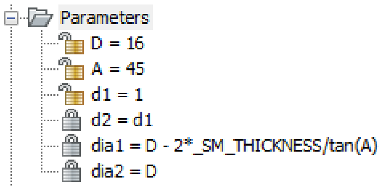
26 Apply the DCDIAMETER constraint ( ) to both circles.

Accept the default value for dia1 (=inner circle) and dia2 (= outer circle).

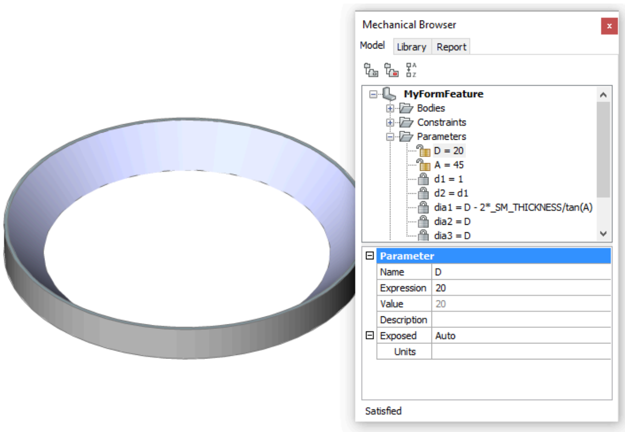


27 Adjust the **Expression** fields of the 2D constraints as follows:

- d1 = 1
- d2 = d1
- dia 1 = D - 2*_SM_THICKNESS/tan(A)
- dia 2 = D

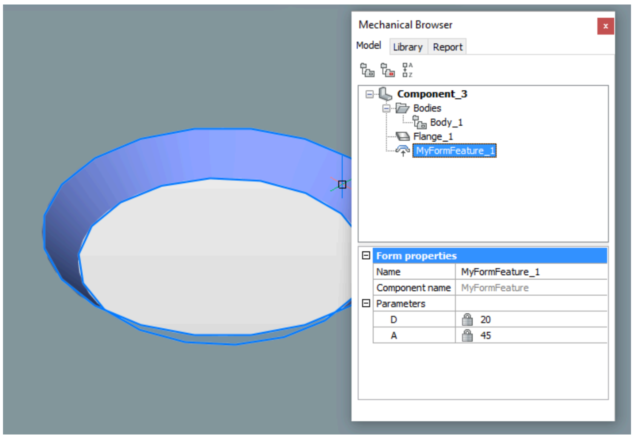


28 Switch **Off** the display of all layers, except layer 0, and save the file.



The form feature can now be used to make conical parametric openings in a sheet metal flange.

Use the **Insert Form Feature** tool ( ), which launches the BMINSERT command.



Optionally: add the form feature as a component to a tool palette.

30.19.6 Creating a bead feature

Bead features are created by the SMBEAD command from a 2D curve on flange face.

1 Do one of the following:

- Click the **Create Bead Feature** tool button ( ) on the **Sheet Metal** toolbar or the ribbon.
- Type SMBEAD in the Command line, then press **Enter**.

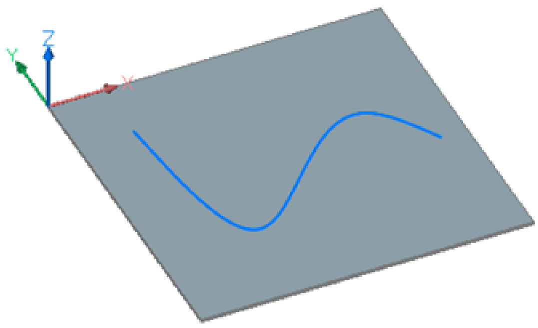
  You are prompted: Select flange face:

2 Select the flange face.

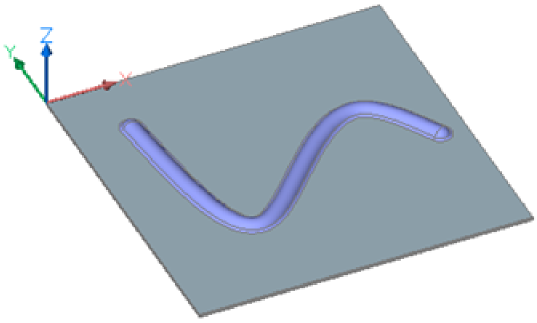
3 You are prompted: Select curve or [Profile radius/Round radius]:

4 Select a curve on the selected flange face.

5 Lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines are accepted. A 2D curve can be open or closed, but it cannot intersect itself or other form features.



After selection of the curve, a bead feature is created:



30.19.7 Modifying a bead feature

To modify a bead feature:

- Modify the generatrix curve using its grip points.
- Edit the **Profile** and **Round** radius in the **Mechanical Browser**.

Modify the generatrix curve

1 Select the curve.

1 Because the curve is hidden by the bead feature, press the Tab key to select obscured entities, or select the curve in the **Structure** panel.

2 Grip edit the curve.

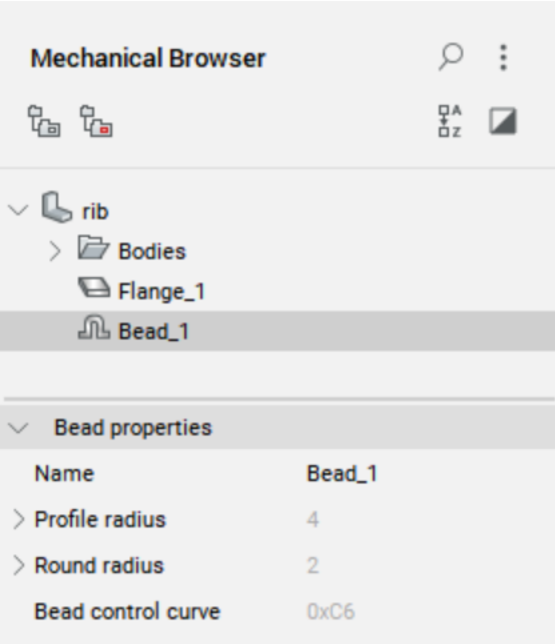
3 The bead feature updates according to the changes of the curve.

Edit the properties in the Mechanical Browser

1 Select the bead feature in the **Mechanical Browser**.

The bead feature highlights in the model.

2 Select the property in the **Bead properties** grid.



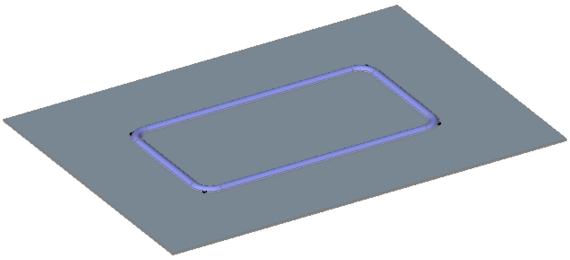
3 Type a value.

4 The selected bead feature is updated.

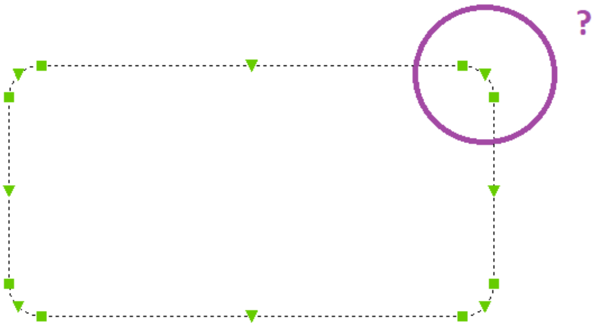
30.19.8 Understanding the difference between control and trajectory curves

When creating a rectangular bead feature, we first draw a rectangle which is then used to create a bead feature by the SMBEAD command. When this control curve is grip edited, the bead feature is updated accordingly.

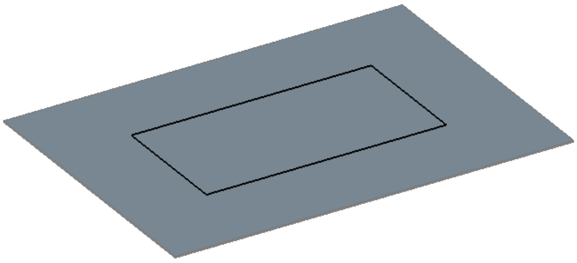
On the production side, the rectangular control curve is replaced by a filleted trajectory curve, according to the physical limitations of the wheel tool:



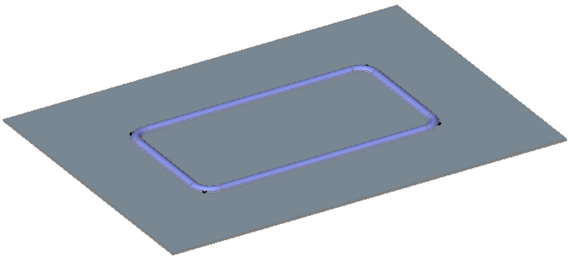
The trajectory curve, however correct, is hard to manipulate, for example when you want to move a corner.



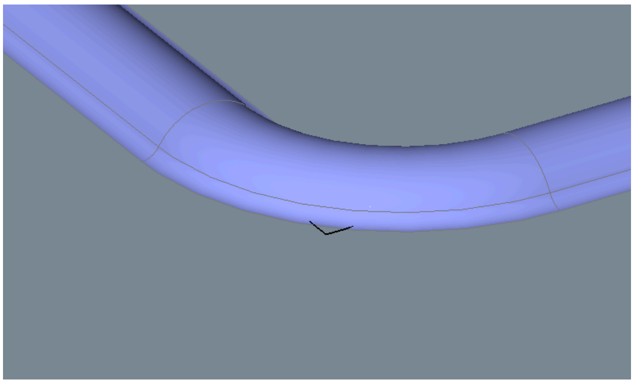
In the design stage we use the **Control Curve**:



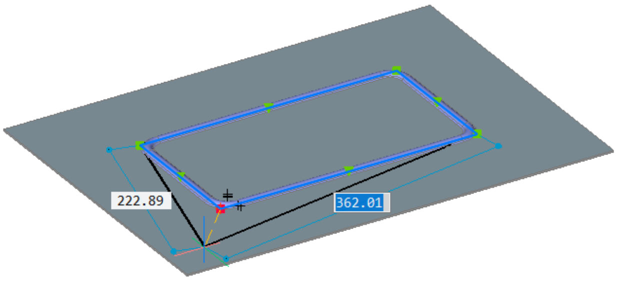
When creating the bead feature, SMBEAD adds fillets according to the value of the **Bead fillet radius** parameter in **Mechanical Browser**. This parameter is not controlled per feature, since it relates to the properties of the physical wheel tool:



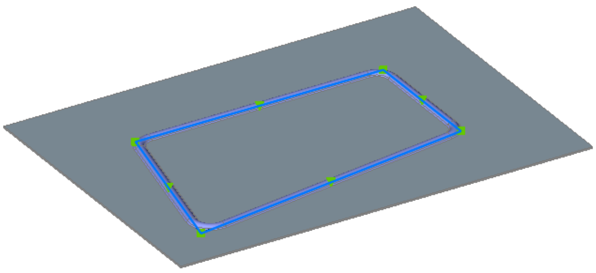
When zooming in closely, you notice the difference between the 3D geometry of the bead feature and the control curve:



When the Polyline is grip edited ...

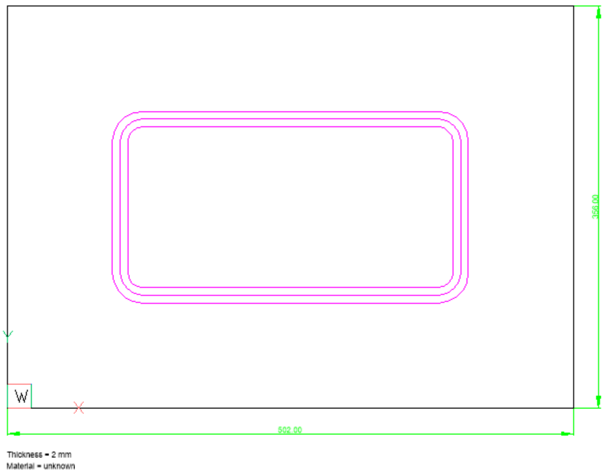


... the bead feature is adjusted accordingly:



This illustrates the associativity between the 2D curve and 3D bead feature. The recognition of bead features in SMCONVERT is controlled by the SMCONVERTRECOGNIZERIBCONTROLCURVES user preference.

When the part is unfolded by the SMUNFOLD command, the curves follow the 3D design and have the fillets. The middle curve is called **Trajectory Curve**:

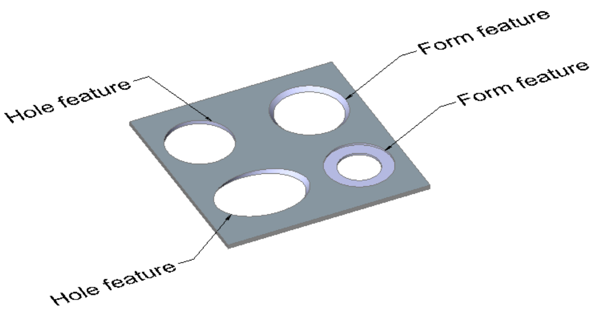


30.20 Working with Hole Features

30.20.1 Overview

Hole features are a simplified version of Form Features. In the image below you can see different features, which have only thickness faces (the SMCONVERT command respects these rules):

- Orthogonal holes are hole features.
- Simple non-orthogonal holes are hole features.
- Chamfered orthogonal holes are form features.
- If the hole thickness face consists of 2 different regions, it is a form feature.



Hole features are recognized by the SMCONVERT command if the SMCONVERTRECOGNIZEHOLES user preference is ON.

The following basic operations are allowed for hole features:

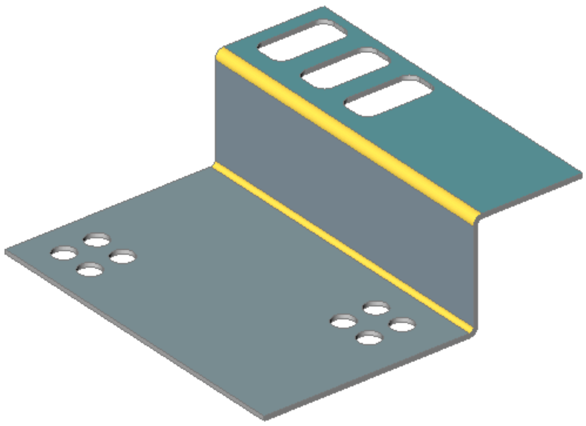
- SMDELETE, SMDISSOLVE, disable and SMSELECT them using commands or in the **Mechanical Browser for Sheet Metal**.
- **Form Feature Modes** in the SMUNFOLD command, including **Symbol** mode.
- Associate your own 2D geometry in a flattened layout.
- Hole features are preserved by the SMREPAIR command. Plain hole faces become orthogonal. A hole feature of the same geometry is not affected by the SMREPAIR command.

30.20.2 Recognition of Arrays of Hole Features

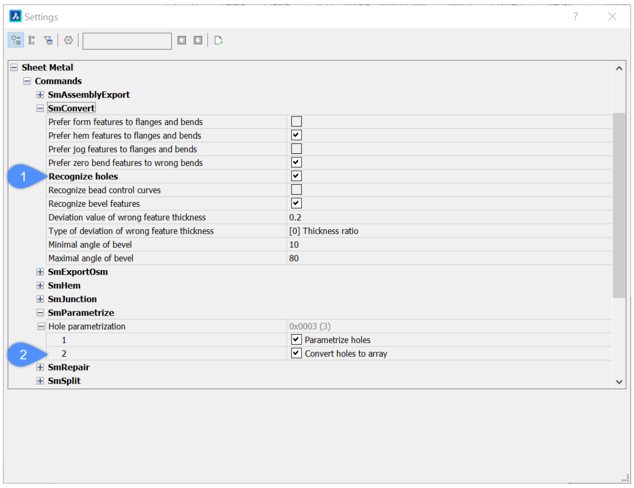
When hole features exist in a body, the SMPARAMETRIZE command detects rectangular arrays of holes on flanges, according to the following rules:

- Holes belong to the same flange.
- Holes are on a rectangular grid.
- There are no gaps (missing elements) in the array.
- Holes are equally oriented.
- The minimal array size is either 1x3 or 2x2.

Let us illustrate the workflow on a part with Flange and Bend features.



1. In the *Settings* dialog box, go to **Sheet Metal/Commands**.
2. Enable the **Recognize holes (1)** option for the SMCONVERT command.
3. Enable the **Convert holes to array (2)** and **Parametrize holes** options for the SMPARAMETRIZE command.



4. Run the SMCONVERT command.

5 Do one of the following:

- Click the **Convert to Sheet Metal** tool button () on the **Sheet Metal** toolbar.
- Click the **Convert to Sheet Metal** tool button () on the **Sheet Metal** ribbon panel.
- Choose **Convert to Sheet Metal** in the **Sheet Metal** menu.

  You are prompted: Select 3D solids or [Entire model] <Entire model>:

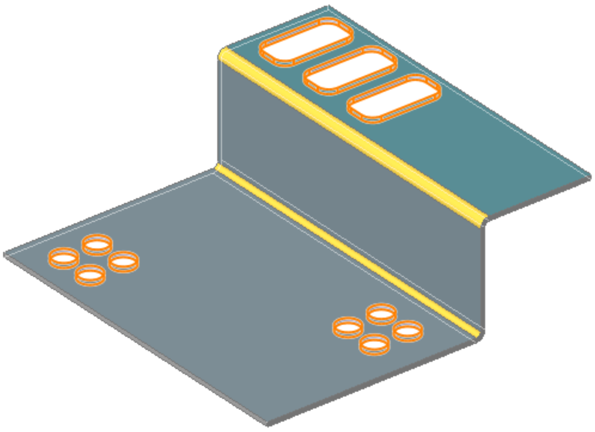
6 Select the 3D solid to convert.

You are prompted: Select 3D solids or [Entire model] <Entire model>:

7 Press **Enter** to convert the 3D solid to sheet metal part.

11 hole features are recognized on the part.

1



2 Run the SMPARAMETRIZE command.

3 Do one of the following:

- Click the **Parametrize** tool button () on the **Sheet Metal** toolbar.
- Click the **Parametrize** tool button () on the **Sheet Metal** ribbon panel.
- Choose **Parametrize** in the **Sheet Metal** menu.

  You are prompted: Select 3D solids to parametrize or [Entire model] <Entire model>:

4 Press **Enter** to process the sheet metal part.

The command reports must look like the one below:

Created 4 distance constraints

Created 3 fix constraints

Created 0 coincident constraints

Created 0 tangent constraints

Created 0 rigid set constraints

Created rectangular array 2x2 basing on Hole_3 feature

Created rectangular array 2x2 basing on Hole_7 feature

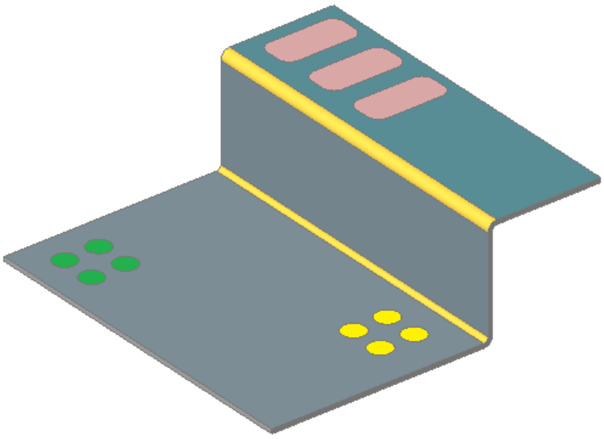
Created rectangular array 3x1 basing on Hole_9 feature

Total: 7 constraints and 3 array(s) created

Note: Feature names and constraint distribution may differ depending on the BricsCAD version.

To examine the arrays, you can temporarily unfreeze the **BC_SUBTRACT** layer.

In the image below the arrays are colored manually.

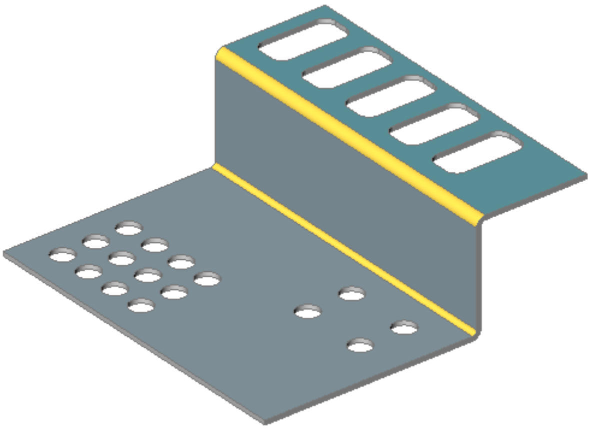


Note: The green and yellow arrays are not unified. Otherwise the array would have missing holes.

5 Edit the **Array Properties** in the **Mechanical Browser**.

In the image below the following has been modified:

- Green array: number of rows and columns from 2x2 to 3x4.
- Yellow array: X and Y spacing.
- Red array: number of rows and columns from 3x1 to 5x1.



30.21 Converting to Sheet Metal and Repairing

Use the *SMCONVERT* and *SMREPAIR* commands.

30.21.1 To convert 3D geometry to a valid Sheet Metal part

A sheet metal part can be created from any 3D solid geometry, whether created in BricsCAD or imported from another CAD system. However, the geometry might not be a valid sheet metal model: missing bends and reliefs or thickness faces which are not orthogonal to the flange faces. BricsCAD Sheet Metal contains a dedicated toolset to convert such geometry to a valid sheet metal part and add all missing sheet metal features.

1 Do one of the following:
   - Click the **Convert to Sheet Metal**tool button () on the **Sheet Metal** toolbar.
   - Click the **Convert to Sheet Metal** tool button () on the **Sheet Metal** ribbon bar.
   - Choose **Convert to Sheet Metal** in the **Sheet Metal** menu.

     You are prompted: Select 3D solids or [Entire model] <Entire model>:

2 Select the 3D solid to convert.

   You are prompted: Select 3D solids or [Entire model] <Entire model>:

3 Press **Enter** to convert the 3D solid to a sheet metal part.

The properties of the sheet metal part display in the **Mechanical Browser for Sheet Metal**.

Note: Converted bodies can be unfolded or exported to DXF or OSM file formats for processing by a CAM system.

In some cases it might be necessary to fix converted bodies before unfolding/exporting:

- Create missing reliefs: SMRELIEFCREATE command.
- Convert hard edges to bends or junctions: SMBENDCREATE and SMJUNCTIONCREATE commands.
- Split flanges and lofted bends: SMSPLIT command.
- Repair sheet metal bodies: SMREPAIR command.

30.21.2 To repair a Sheet Metal part

A sheet metal part may have inaccuracies in geometry and topology or sheet metal design issues. This is often the case for imported parts which are designed in geometrical kernels different from ACIS.

Examples of such problems are:

- Thickness faces, which are not perpendicular to the flange/bend faces.
- Lofted bends are not tangential to the adjacent flanges/bends.
- Wrong bends.
- General ACIS issues.

Such problems can prevent editing operations from being executed successfully.

The recommended workflow to fix such problems is:

- Run the DMAUDIT command to fix general modeling issues.
- Run the SMREPAIR command.

30.21.3 To repair a Sheet Metal body

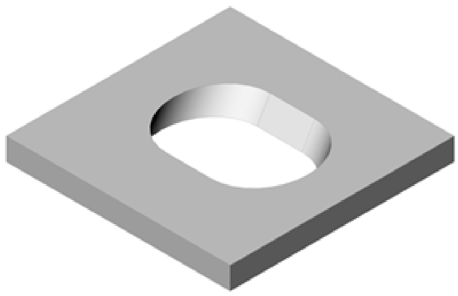
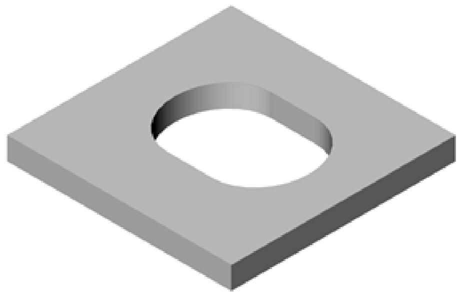
Sheet metal bodies with thickness faces, which are not perpendicular to the flange/bend faces, can not be correctly manufactured using traditional tools.

1 Do one of the following:

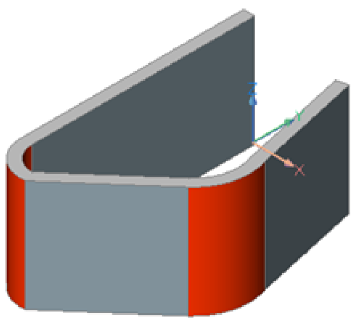
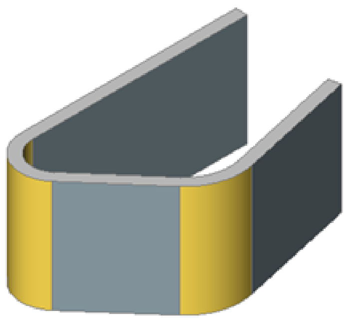
- Click the **Repair Sheet Metal** tool ( ) on the Heal panel of the Sheet Metal ribbon tab;
- Click the **Repair Sheet Metal** tool ( ) on the **Sheet Metal** toolbar;
- Choose **Repair Sheet Metal** in the **Sheet Metal** menu;
- Type SMREPAIR in the Command line.

  You are prompted: Select flange face or lofted bend face to repair or [switch on Local repairing/ Scan for lost details]:

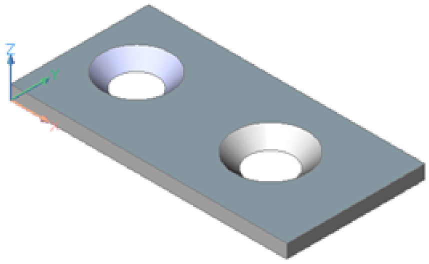
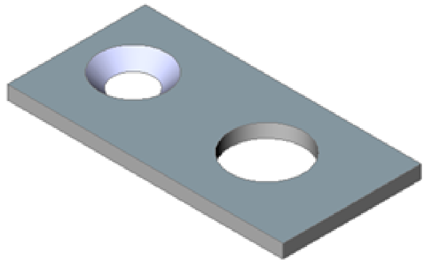
2 Select the face to be repaired.

	
The same model before ...	... and after applying a repair operation:

When the model has wrong bends SMREPAIR transforms them to regular bends:

	
Wrong bends, which have a non-tangential connection with the flanges.	Correct bends after repairing.

Form features are translated to the new repaired model:

	
Model before repairing.	The form feature has not been changed, while a hole, which is not a form feature, is orthogonalized.

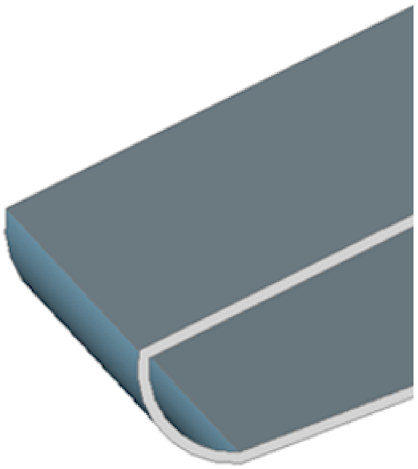
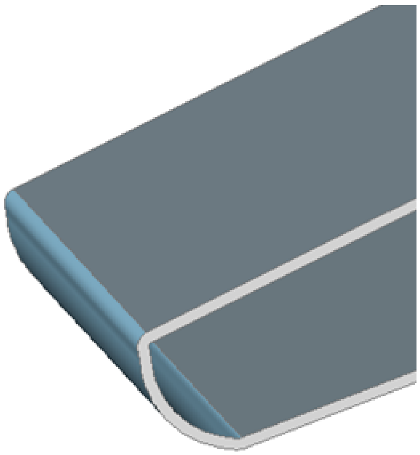
30.21.4 To repair a Sheet Metal body with lofted bends

1 Do one of the following:

- Click the **Repair Sheet Metal** tool ( ) on the **Heal** panel of the **Sheet Metal** ribbon tab.
- Click the **Repair Sheet Metal** tool ( ) on the **Sheet Metal** toolbar.
- Choose **Repair Sheet Metal** in the **Sheet Metal** menu.
- Type SMREPAIR in the Command line.

  You are prompted: Select flange face or lofted bend face to repair or [switch on Local repairing/Scan for lost details]:

2 Select a face of a flange or a lofted bend. The selected face provides the surface to be thickened.

Before	After
	

Before	After
The lofted bend is not tangent to the flange. Unfolding is impossible.	Notice the difference in the connection of the blue lofted bend with the top flange, which is smooth now. Unfolding is possible.

Option: If the **SmRepairLoftedBendMerge** user preference is ON, adjacent lofted bends with a tangential connection are merged to a single lofted bend.

30.22 Unfolding Sheet Metal Parts and Exporting to CAM

30.22.1 Overview

Use the SMUNFOLD command.

Sheet metal parts can be designed in many different ways. To manufacture the sheet metal part, a flat pattern with bend annotations is needed, which is then used in a CAM system to program CNC machines, such as laser cutters and press brakes. This flattening operation is called unfolding.

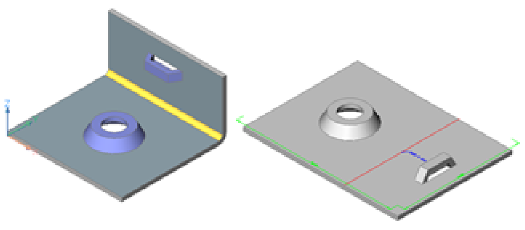
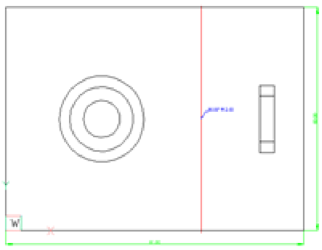
30.22.2 Unfolding a sheet metal body

The SMUNFOLD command unfolds sheet metal parts. BricsCAD automatically creates a solid body corresponding to the flat metal sheet needed to manufacture the sheet metal part using bending techniques. This sheet is placed on the XY-plane and is oriented along the coordinate axes similarly to the orientation of the initial body in 3D space. To change the orientation of the unfolding in Z axis, run the SMUNFOLD command again and select the opposite face of the initial body.

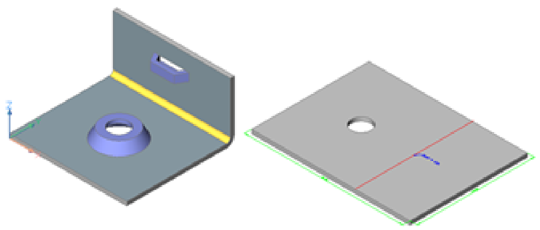
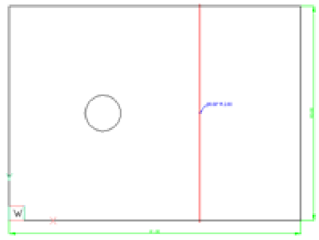
Form features are unfolded according to the **Unfolding Mode** setting in the **Mechanical Browser**.

The options are:

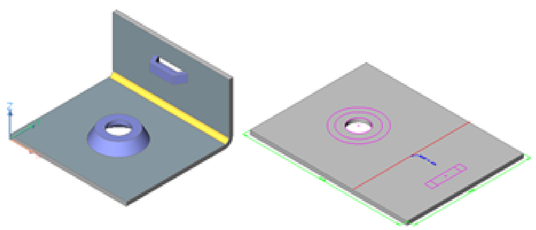
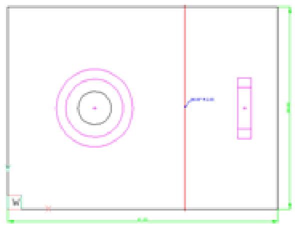
- **Keep** (0): allows to keep form features geometry on the unfolded representation.

3D model and unfolded body	2D
	

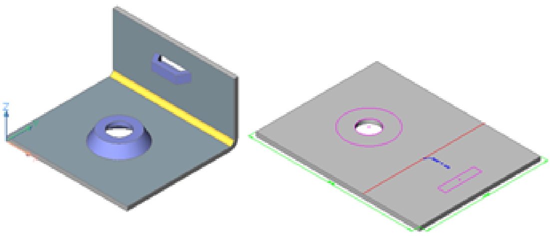
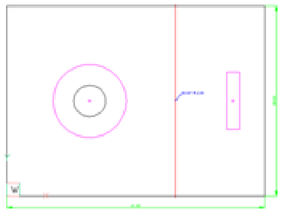
- **Remove** (1): removes the form feature geometry from the unfolded representation.

3D model and unfolded body	2D
	

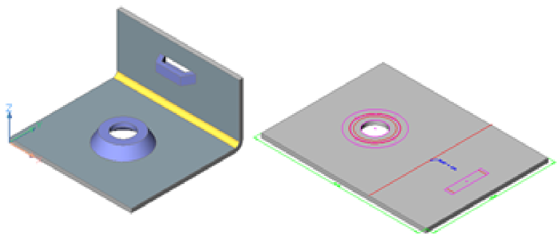
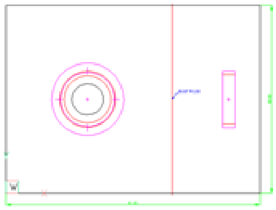
- **Project** (2): generates a form feature symbol by projecting the form feature edges onto the flange plane.

3D model and unfolded body	2D
	

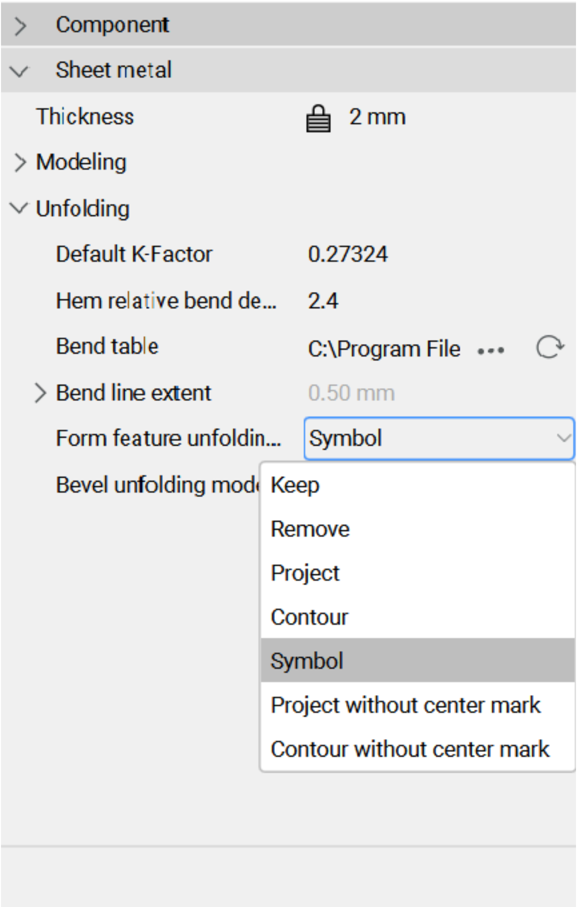
- **Contour** (3): generates a form feature symbol by projecting the external contour and holes of the form feature onto the flange plane.

3D model and unfolded body	2D
	

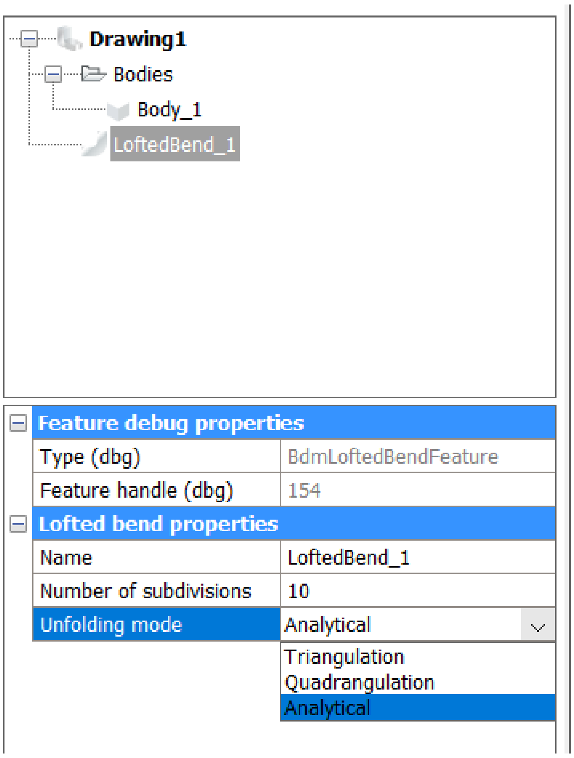
- **Symbol** (4): creates the unfold symbol on the **UNFOLD_SYMBOL_UP** or **UNFOLD_SYMBOL_DOWN** layer depending on the direction of the form feature.

3D model and unfolded body	2D
	

The unfold mode for the current sheet metal part can be set in **Mechanical Browser**.



Lofted bends lying on cylindrical and conical surfaces can be unfolded analytically: the dimensions and shape of the unfolded lofted bend will be as if its middle surface was mathematically unrolled on the plane. This shape can also be achieved by a highly increased number of samples for the lofted bend. You must set the **Unfolding mode** individually for each lofted bend feature. If analytical unfolding is allowed, the option will be present in a drop down list:



30.22.3 To unfold a sheet metal body

1 Launch the SMUNFOLD command.

You are prompted: Select a flange or lofted bend face to start unfolding [Associative]:

2 Select a face on sheet metal part.

You are prompted: Select position of the unfolded body:

3 Specify a point.

You are prompted: Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>

4 Do one of the following:

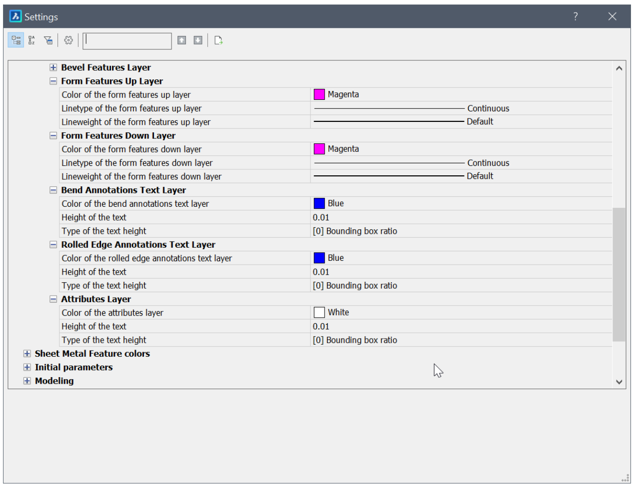
- Save the unfolded 3D solid in a separate drawing file.
- Convert the unfolded 3D solid to a 2D drawing file (*.DWG or *.DXF).
- Rearrange the bend annotations.
- Keep the block with the unfolded part in the sheet metal part model.

The SMUNFOLD command considers the deformation of the sheet metal material during bending. When a flat sheet of metal is bent into a 3D part with a bending tool (for example a press brake), the material is plastically deformed, which means it is compressed at the inside of the bend and stretched at the outside. As a result, the length of the part measured along its surface is different in the flat and bend states. BricsCAD can automatically compute the proper unfolded length of your part based on the material deformation properties. These properties can be defined by setting the value of **K-Factor** parameter or by attaching a Bend Table.

30.22.4 Managing layer properties of the 2D geometry

The 2D geometry generated by the SMUNFOLD, SMEXPORT2D, and SMASSEMBLYEXPORT commands has different colors for contours, bend lines up, bend lines down, overall dimensions, form features up, form features down, bend annotations and attributes.

You can set the colors differently in the **Sheet Metal/Commands/Layers for unfolding** settings group on the *Settings* dialog box. Each entity type is placed on a dedicated layer of which the color is controlled through a series of user preferences:



If the 2D geometry of a form feature was taken from a library component, its properties are controlled by the properties of this component at the moment of insertion. There are 2 other modes of **Unfolding Mode** which lead to generation of 2D geometry for form features: **Project** or **Contour**. In these modes 2D geometry is generated automatically from 3D geometry and its appearance is controlled by **Form Features Up Layer** and **Form Features Down Layer** settings groups.

30.22.5 To export a sheet metal body

Sheet metal parts created or edited in BricsCAD can be processed by different CAM systems (such as JETCAM or CADMAN-B) to generate NC code for cutting and bending machines.

The user preference controls the **save 2D geometry** option of the SMUNFOLD command to customize the .DXF file to be compatible with the list of popular CAM systems.

The SMEXPORT2D command exports the unfolded representation of a sheet metal body as a 2D profile in *.DXF or *.DWG file format.

The SMEXPORTOSM command exports a sheet metal solid to the *.OSM (Open Sheet Metal) file format (native for the CADMAN-B CAM system).

30.22.6 To export an assembly with sheet metal parts to DXF

The SMASSEMBLYEXPORT command searches the assembly tree for sheet metal design and exports it to a *.DXF file. As a result, a set of *.DXF files, sorted by thickness in different subfolders is created in the output folder. The command generates a HTML report which allows to control the output. Most of the sheet metal parts will be processed automatically. Poor sheet metal parts will be reported by the command to be reworked manually.

30.23 Associative Unfolded Representations of 3D Solids

30.23.1 Overview

BricsCAD offers bidirectional associativity between the 3D Sheet Metal part and its flattened appearance. In this mode both representations exist in separate documents: the Original part is stored in its native document and the Flattened part is stored in a temporary document. Associativity is temporary and exists while both documents are opened. In order to work in associativity mode in the next BricsCAD session or once the document was closed, you must repeat the steps again.

30.23.2 What does associativity mean?

Associative Unfolded allows you to make cuts on flattened geometry and propagates the changes back to the 3D model.

This helps simplify 2 scenarios:

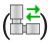
- Creating complex, non-standard Bends and Corner Relief cuts.
- Avoid self-intersections in unfolding.

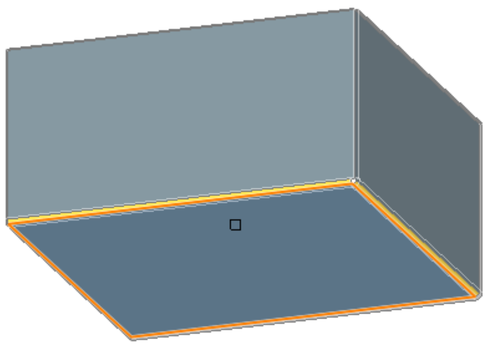
In these 2 scenarios both are easy to do in 2D rather than in 3D. In flattened mode Sheet Metal part is represented as set of separate flat solids (per-feature). Use the SMDELETE command to delete solid flanges and bends on flat patterns.

Associativity is bidirectional; changes can be propagated from flattened parts back to 3D models and vice versa. Changes made on one of the representations are propagated to the document with other representations once it becomes active.

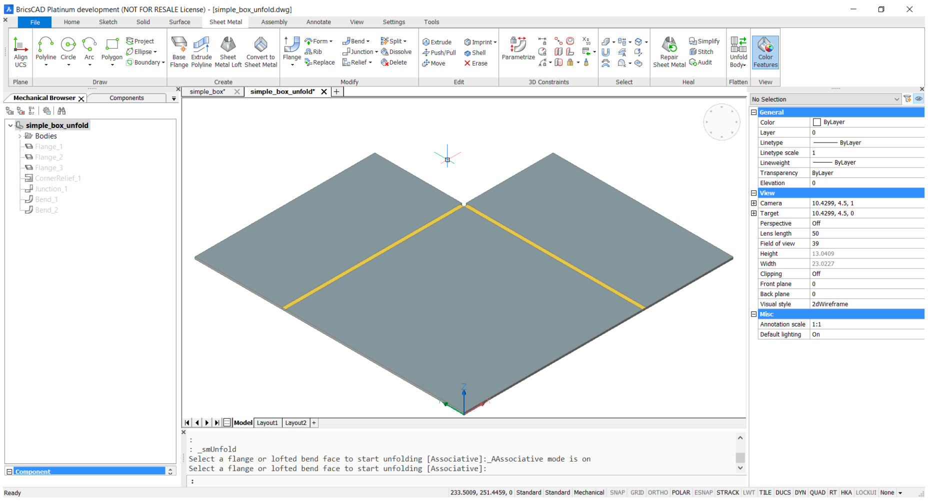
30.23.3 Switching to Associativity Mode

The associativity mode is turned on using the SMUNFOLD command. Make sure there are no faces selected.

From the **Sheet Metal** section of the Ribbon, select the **Associatively Unfold Body** icon () and select **Flange** face.



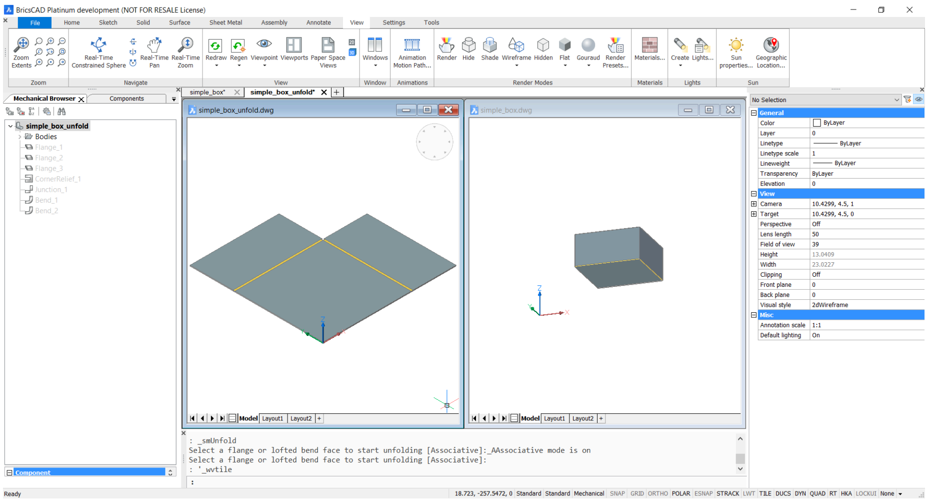
The application will compute flattening of the given part and switch it to associative mode. Condition of success of given operation is the same as for regular unfolding. Continue working in associativity mode only if no errors or warnings appeared during the operation. A new temporary document containing the flattened part will be created.



Note: There is no need to save this document. It is used to propagate editing operations from the flattened solid back to the 3D model.

Use **Vertical Tile** to view both the 3D model and associative unfolding documents.

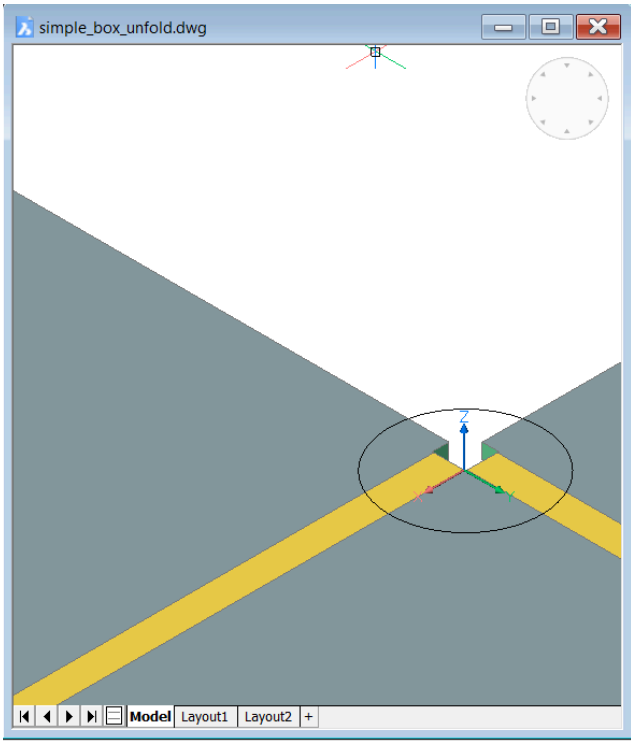
Click the **Vertical Tile** button on the Toolbar or Ribbon.



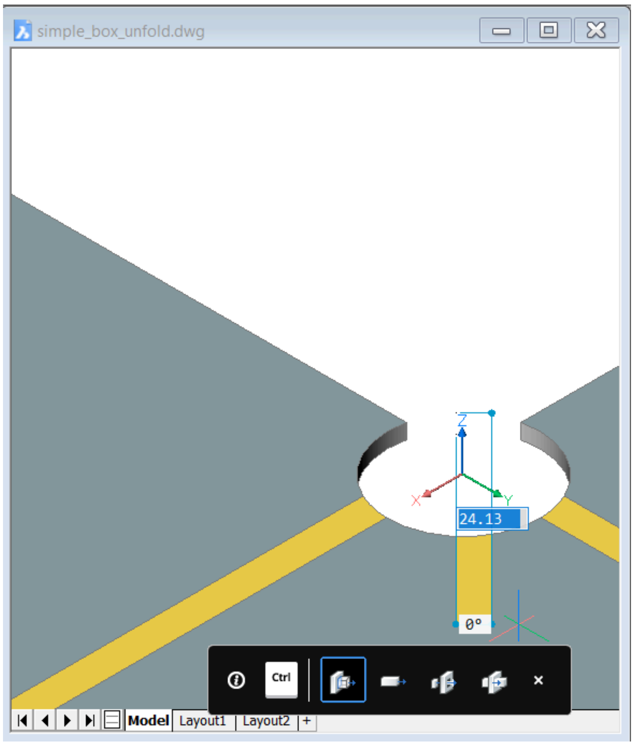
Note: In the **Mechanical Browser**, when **flattened geometry** is selected, it lists features that are disabled.

30.23.4 Editing the Geometry

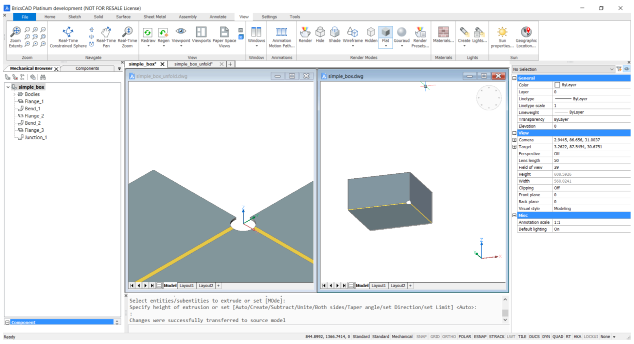
1 Working with flattened design document, draw a circle in the corner.

1 

2 Apply DMEXTRUDE command to the circle, cutting the volume.

3 

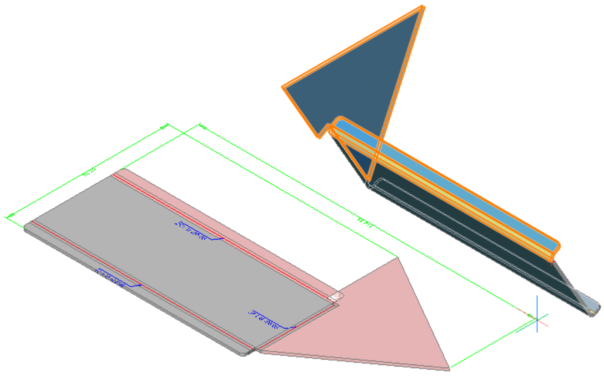
4 Click on the original document again to make it active.

5 

6 The model will be updated and the custom corner relief appears. Application writes to the Command line: Changes were successfully transferred to source model.

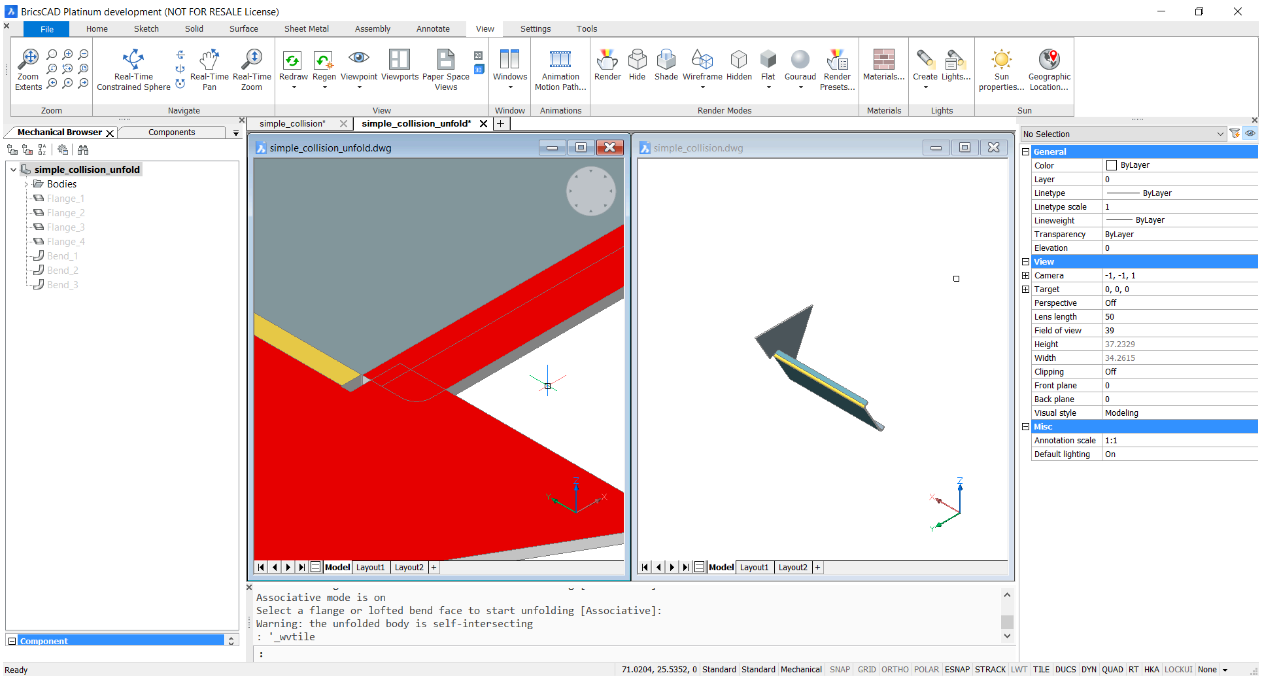
30.23.5 Avoiding Collisions in Flat Pattern

Another useful application of associative unfolding is fixing the not so obvious material collisions.



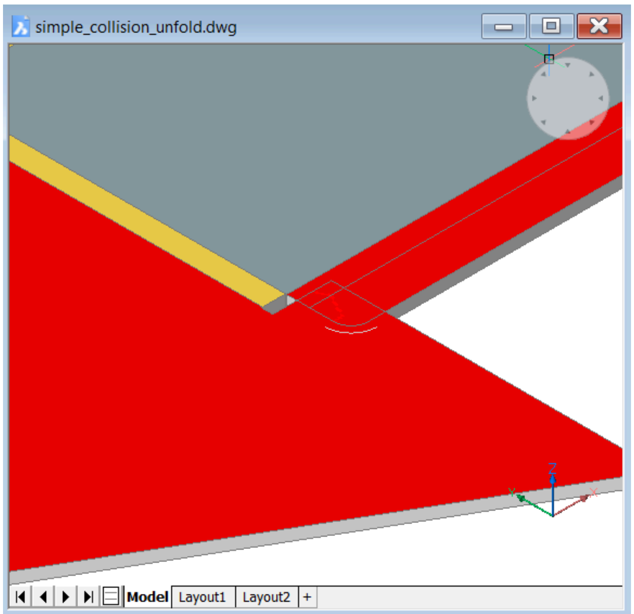
In this example, we do not know what changes to apply to the 3D model to avoid the collision when flattened. Let us rework the model with Associative Unfolding.

1 From the **Sheet Metal** section of the Ribbon, select the **Associatively Unfold Body** icon ( ), select **Flange** face, unfold and click the **Vertical Tile** button ( ) on the Toolbar or Ribbon to tile the opened documents vertically and zoom to the collision area.

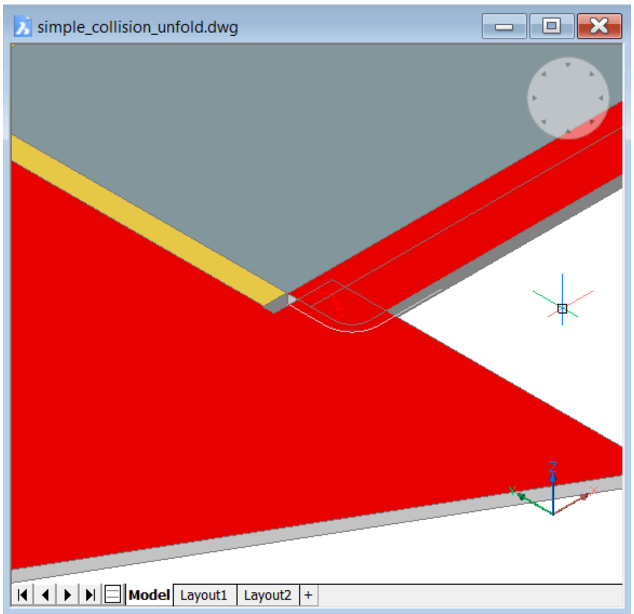


2 Select boundary circular edge and offset it by 1.

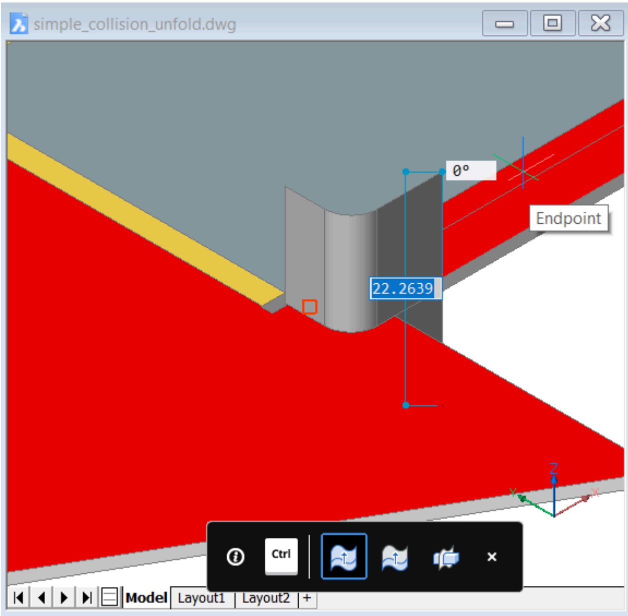
3



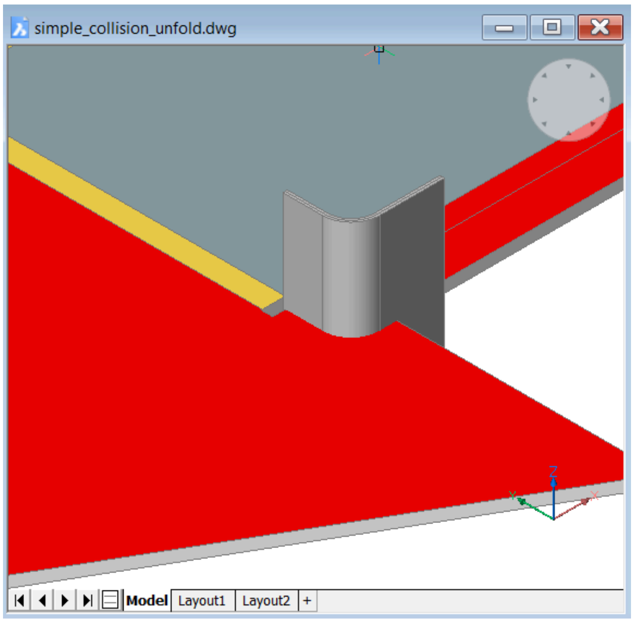
4 Draw extra straight lines adjacent to the edge and join them.

5 

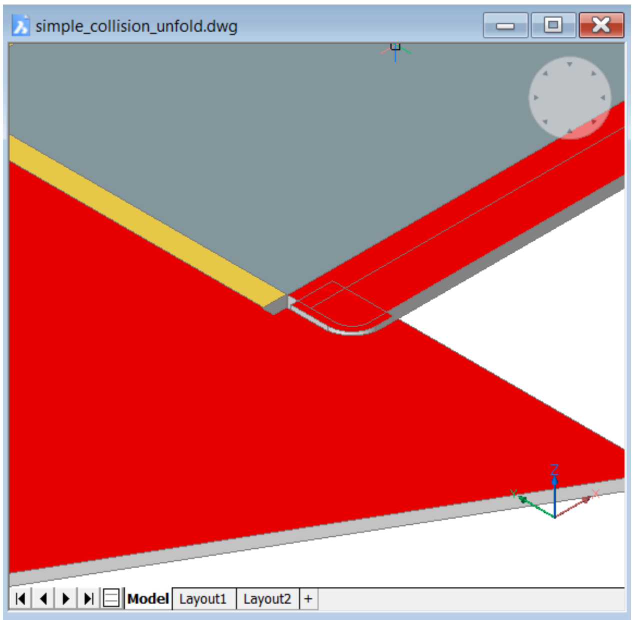
6 DMEXTRUDE the polyline on both sides.

7 

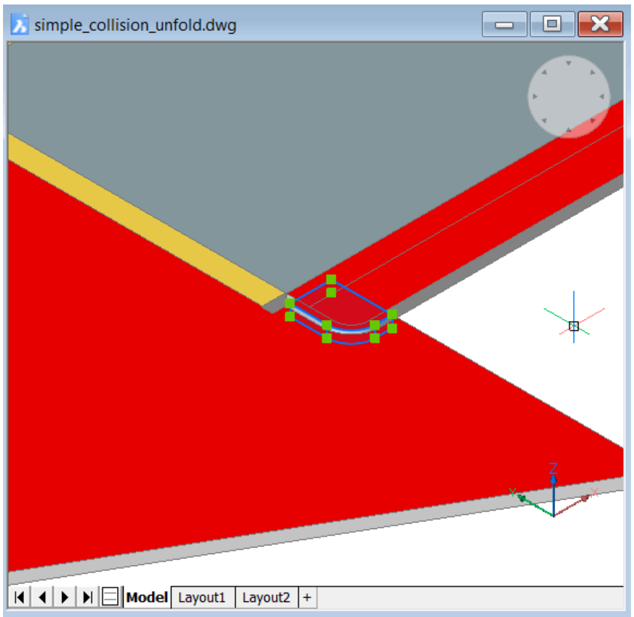
8 DMTHICKEN the surface by a bigger value.

9 

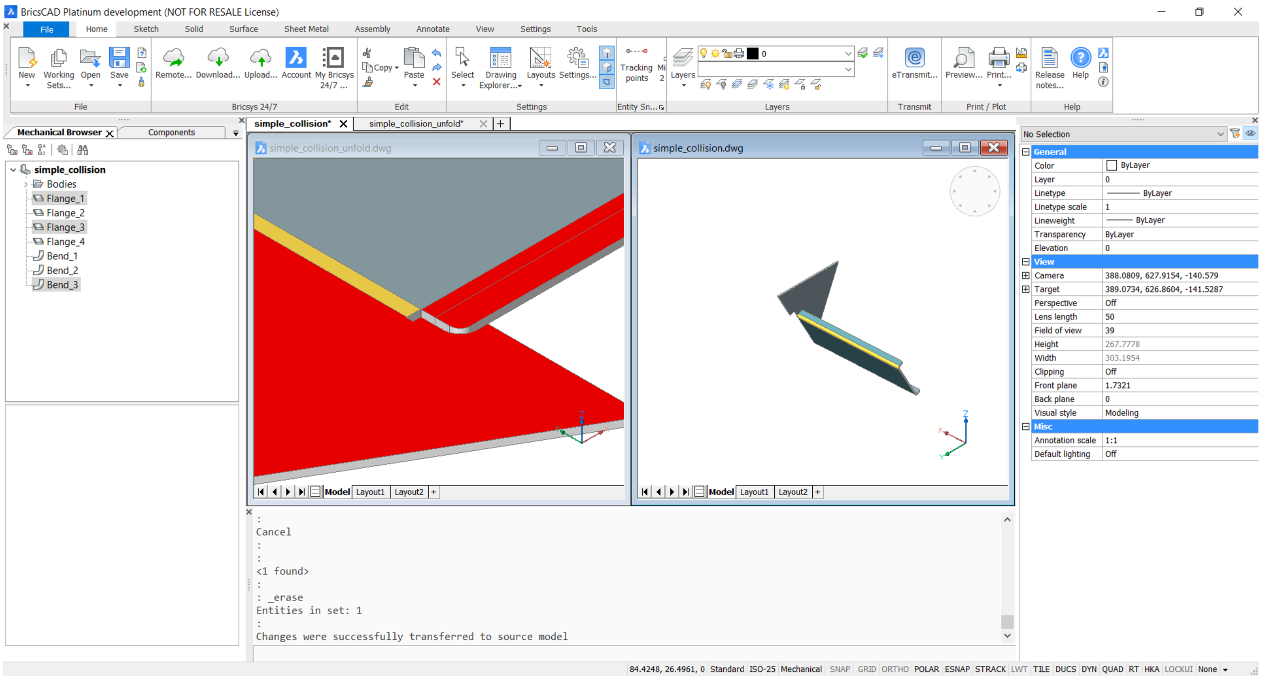
10 SUBTRACT the thickened body from unfolded pattern.

11 

12 ERASE the leftover pieces.

13 

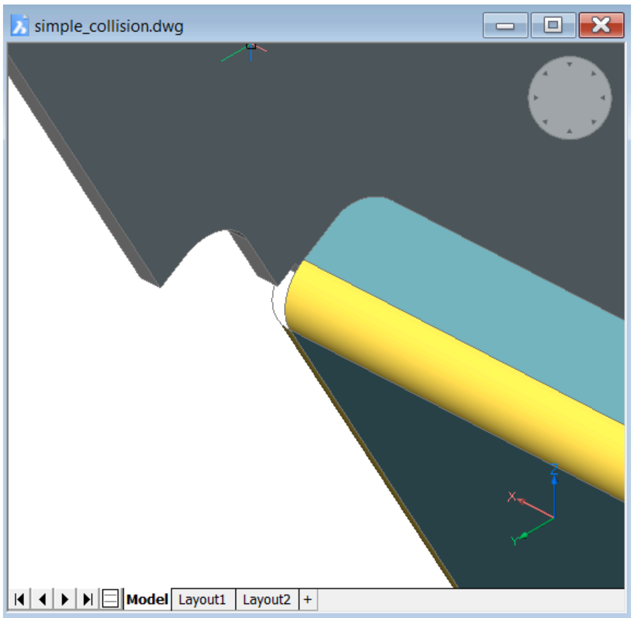
14 Switch back to the document with the 3D model.

15 

16 Make sure that the changes have been propagated.

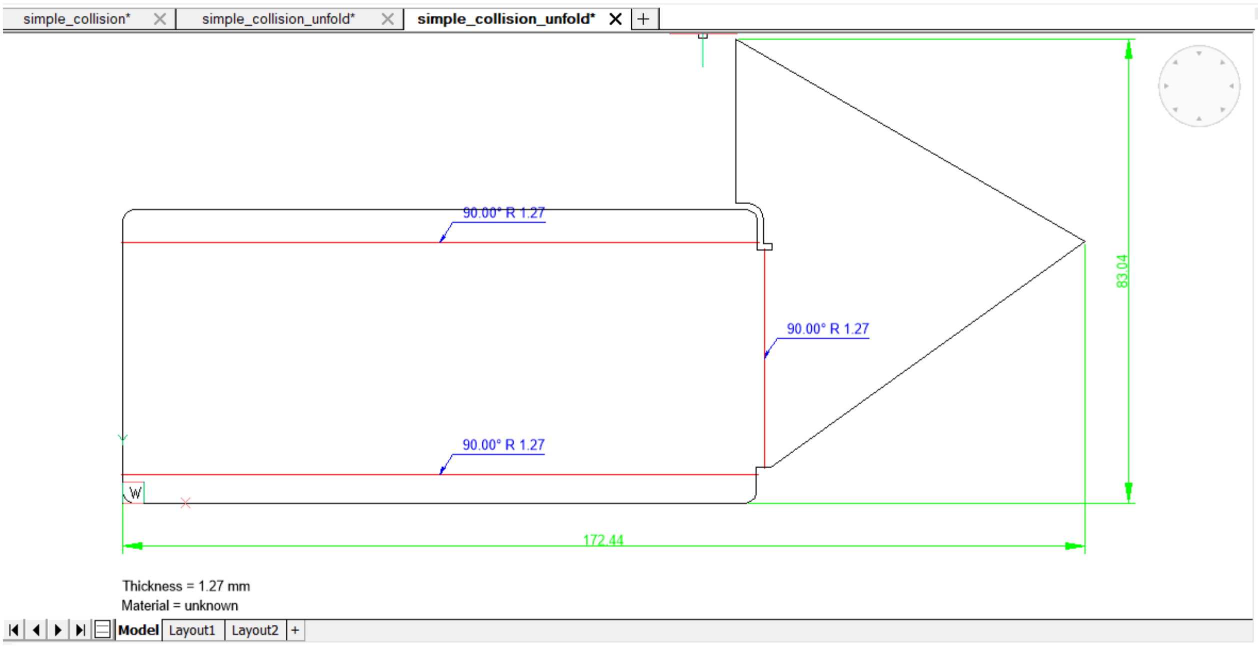
17 Changes were successfully transferred to the source model.

Zoom to the updated corner.



18 To test the 2D unfold, export the file as a DXF file.

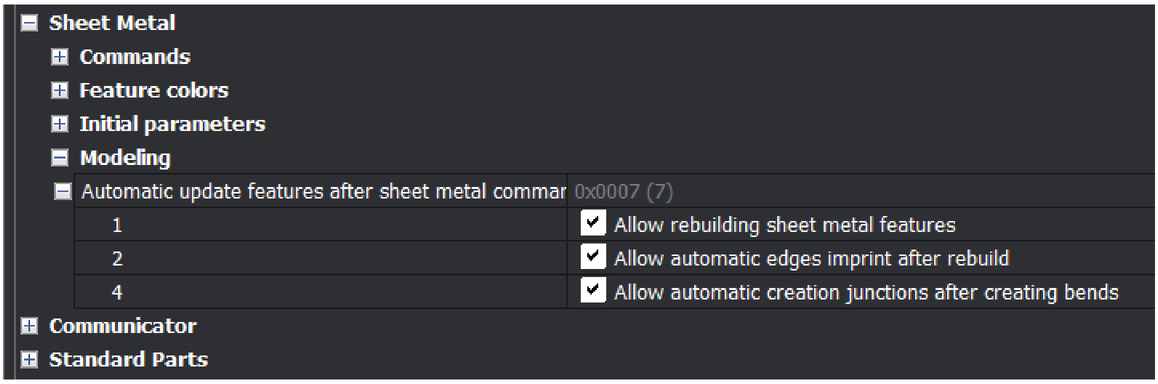
19



30.24 Smart Features

30.24.1 Overview

The Smart Features technology manages the local intelligent update of the geometry after executing Sheet Metal commands. The behavior is controlled through the **SmSmartFeatures** user preference.



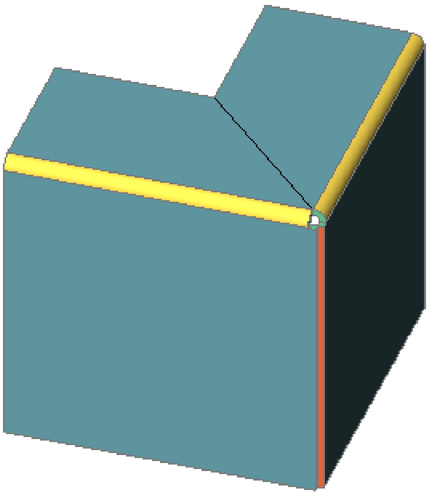
3 options are available:

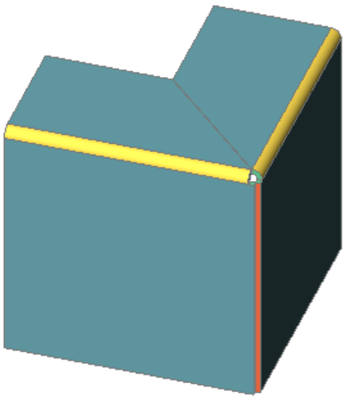
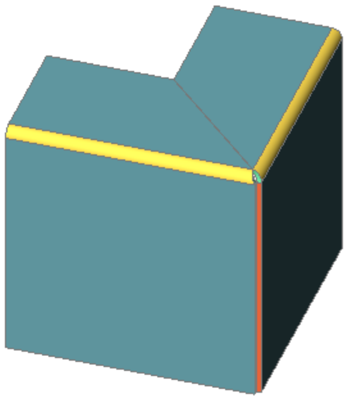
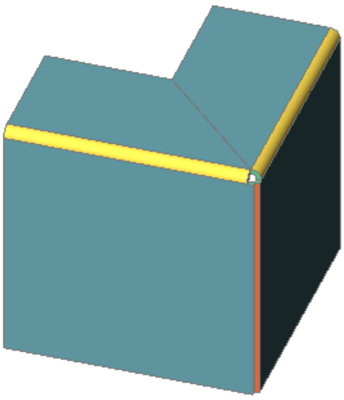
- Allow rebuilding sheet metal features:
  - Corner relief is transformed into 2 bend reliefs after executing *SMSPLIT* on a flange.
  - Unnecessary reliefs are removed.
- Allow automatic edges imprint after rebuild: automatic imprinting edges for thickness faces.
- Allow automatic creation of junctions after creating bends.

Note: Always keep Smart Features options ON, unless inconsistent behavior occurs.

30.24.2 Transformation of a corner relief into bend reliefs

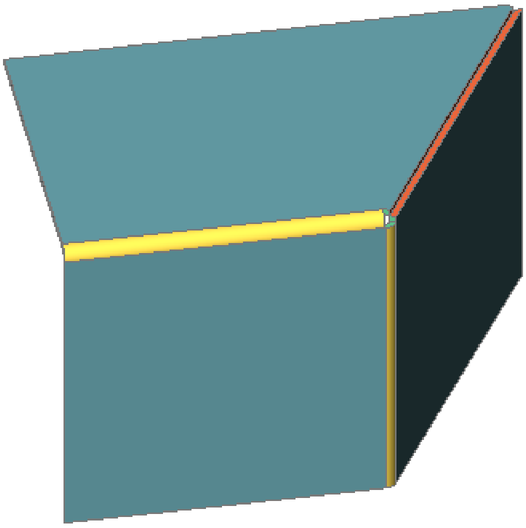
When splitting a top flange by a line into 2 separate parts, the corner relief is transformed into 2 bend reliefs.



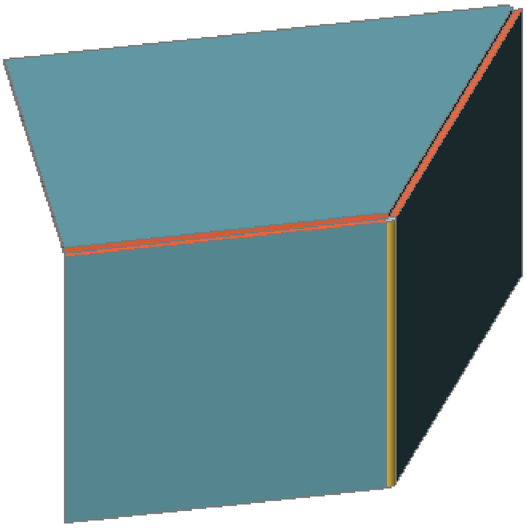
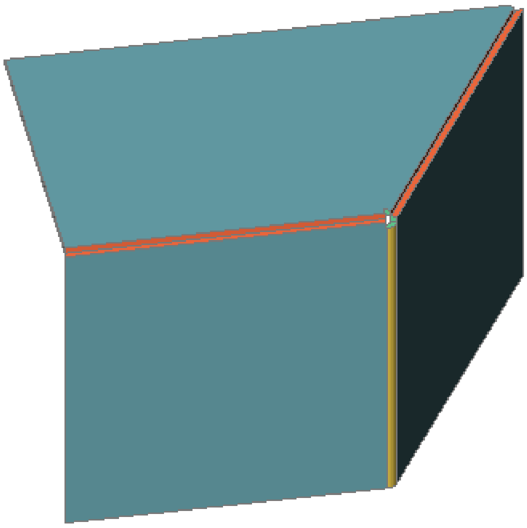
 	
Smart features create bend reliefs automatically, which can then be switched to smooth.	Without smart features, the corner relief is kept and cannot be switched.

30.24.3 Removing unnecessary cuts

In the model below, a corner relief cut exists, which is needed to prevent the material from being stretched.



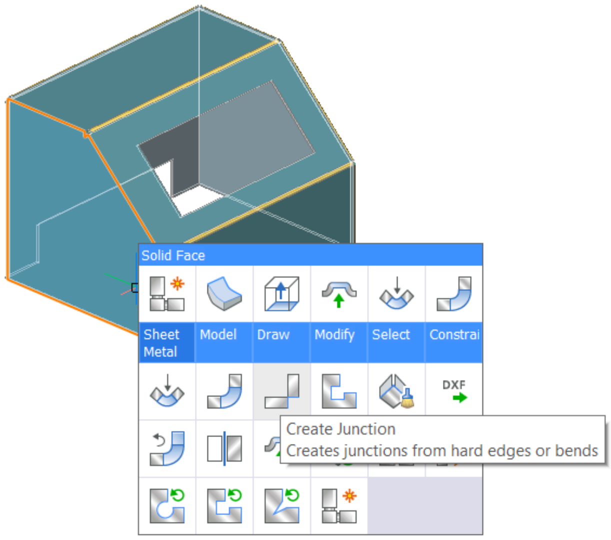
When the top bend is converted into a junction, we get to separate parts. The smart features automatically remove the now unnecessary relief.

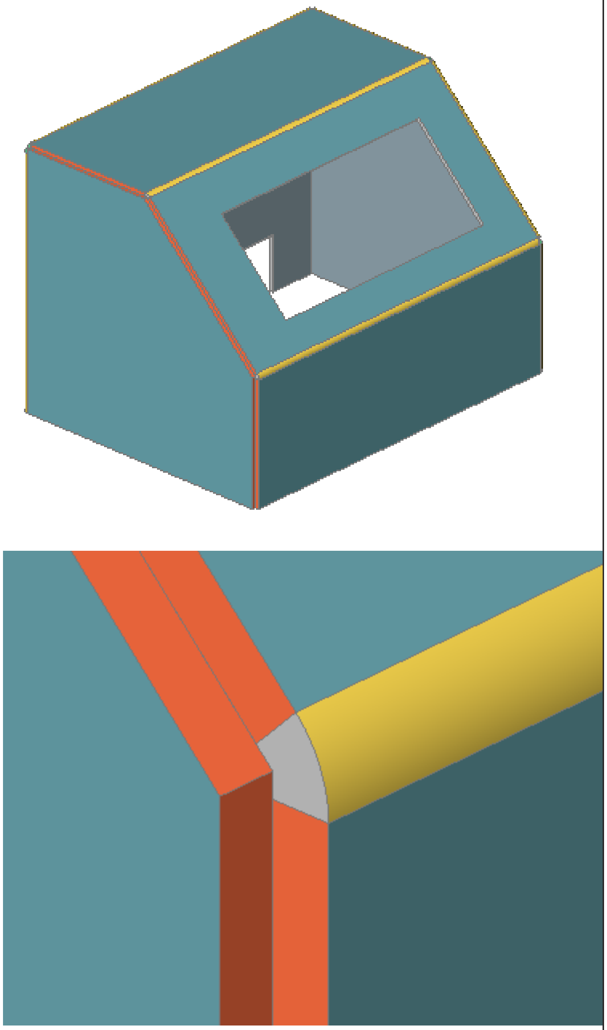
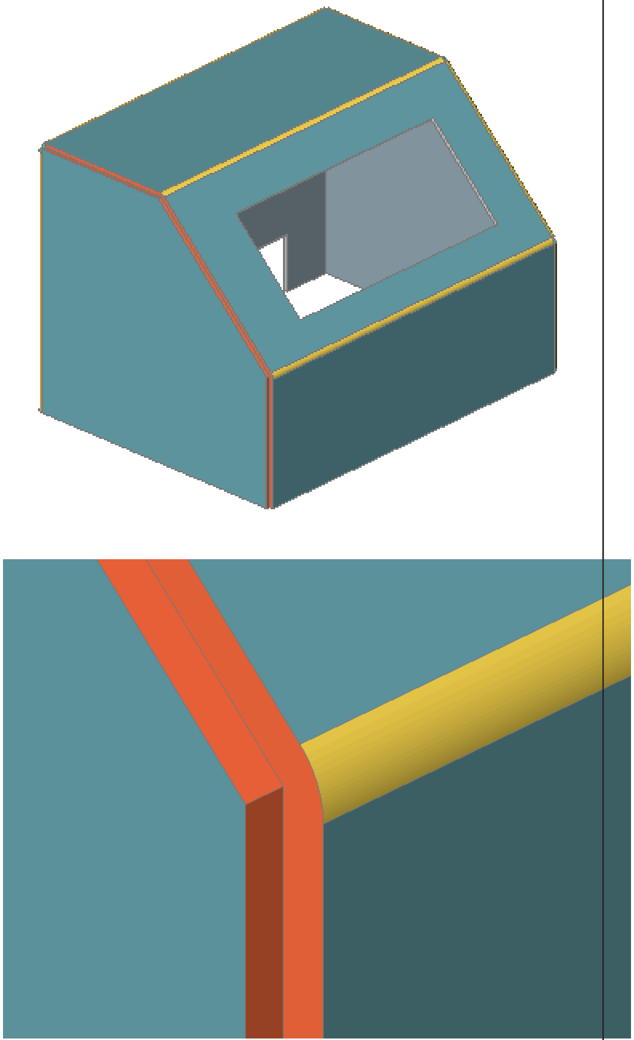
	
Smart features automatically fill unnecessary cuts, thus simplifying the geometry.	Without smart features, the corner relief still exists.

30.24.4 Automatic imprinting of edges

Imprinted edges on thickness faces separate them into areas which can be reworked automatically. That means localized junctions can be converted into bends. Without automatically imprinted edges the SMIMPRINT command is needed here.

In the example below, we will separate a flange using junctions. We turn **Rebuilding sheet metal features** ON, and then use **Automatic edges imprint after rebuild**.

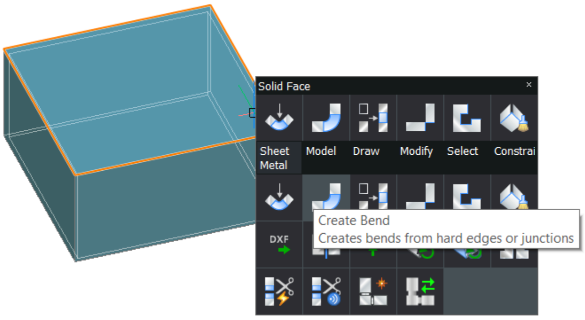


	
Allow automatic edges imprint after rebuild = ON. Junction faces are split correctly.	Allow automatic edges imprint after rebuild = OFF. Junction features overlap each other.

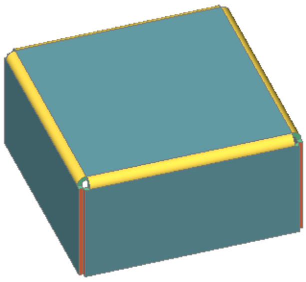
30.24.5 Automatic creation of junctions

In certain situations, when hard edges meet at the corner and we create a bend on all but one, which has to become a junction in order to make the part unfoldable.

Assume we create bends on the top face of a sheet metal box:



Then the remaining hard edges adjacent to the face must become junctions. When **SmSmartFeatures** = ON, this is performed automatically:



To be precise, the area of automatic junction creation is located to the common vertex of hard edges (or corner relief when bends are created). The option does not optimize the layout of the unfolded part.

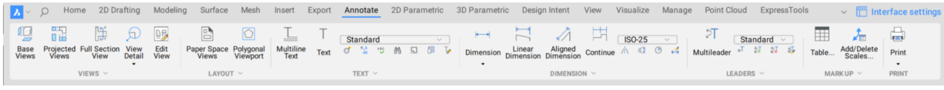
31. Generated Drawings

31.1 Drawing Views

31.1.1 Overview

BricsCAD Mechanical provides a toolset to generate drawing views from 3D models of parts and assemblies. Drawings views are created in the paper space of the same DWG file. The 3D model is created in model space. Drawing views can be annotated using dimensions, leaders, text and tables. Views and annotations are updated when the associative 3D model is changed.

The tools to generate, update and annotate drawing views are located on the **Annotate** tab - **Views** panel of the **Mechanical workspace**:



Views	Description
Base Views	Generate associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
Projected Views	Generate additional projected views from an existing drawing view.
Section View	Create a cross-section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.
Detail View	Create a detail view of a portion of a standard generated drawing at a larger scale.
Edit View	Change the scale and the hidden line visibility of drawing views.
Update Views	Updates a selection of drawing views obtained by VIEWBASE and VIEWSECTION commands when VIEWUPDATEAUTO system variable = 0.
Section Style	Predefined and user-defined styles.
Detail Style	
Export Views	Export the content of drawing views, obtained by the VIEWBASE and VIEWSECTION commands, to the Model Space of the drawing, or to a new drawing.

31.1.2 Draft-quality drawing views

By default, BricsCAD generates precise (high-quality) drawing views for any 3D model. The edges of such views (including silhouette edges) are represented by 2D entities; lines, circles, arcs and splines. The generation of precise views for a large number of 3D entities can take a significant amount of time. This is because every silhouette edge must be approximated with an analytical 2D curve.

You can reduce the time to generate drawing views by changing the DRAWINGVIEWQUALITY setting to 0. This enables draft-quality drawing views. Such drawing views are internally represented as 3D solids, which enables you to add annotations to their silhouette edges. However, they look very similar to a precise (high-quality) drawing view and you can use them to quickly create layouts.

You can switch a draft-quality drawing view to a high-quality drawing view by using its **Geometry** property (change it from Draft to 2D).

31.2 Drawing Annotations

Creating and editing annotations, review Dimensioning Tools Overview.

Dimensions and Multileader that are in the drawing view are updated upon change of the underlying 3D model.

Assembly drawings can be annotated with BOM balloons. Every balloon refers to a row in a BOM table generated from the assembly.

The BOM table can be placed in paper space (layouts) with the drawing view of an assembly.

Exploded Representations of assemblies can be used to generate drawing views and annotate them with balloons.

32. Communicator for BricsCAD

32.1 Communicator for BricsCAD® - Getting started

Communicator for BricsCAD® is an optional plugin for Pro and higher license levels of BricsCAD that imports and exports 3D data between major CAD formats. Its high-quality conversions facilitate the re-use of 3D CAD data to improve productivity and reduce project costs.

32.1.1 Activating Communicator for BricsCAD®

After you have installed Communicator for BricsCAD® on your computer, you need to activate the software.

1 Launch the LICENSEMANAGER command.

The **Bricsys License Manager** dialog box displays.

2 Next to **Communicator for BricsCAD**, click **Manage License**.

3 The **License Manager** > **Activate BricsCAD Communicator** dialog box displays.

- Click **Activate Trial** to activate Communicator for BricsCAD® trial license online.
- Click **Activate Now...**. You can:
- Type a valid Communicator for BricsCAD® license key into the **License Key** field if you are using a Single User or Volume license.
- Type the Server name or local IP address of the license server if you are using a Network license.

4 Click the **OK** button.

The license is registered, and a confirmation dialog box displays.

5 Read the text on the dialog box and click the **I understand** button.

Note:

- To activate Communicator for BricsCAD® license offline see: **Offline/manual activation (Trial)** or **Offline/manual activation (Single/Volume license)**.
- To activate Communicator for BricsCAD® network license on the license server see: **Online activation (Network license)** or **Offline/manual activation (Network license)**.

32.2 Communicator for BricsCAD® settings

The **Communicator** section in the **Settings** dialog box allows you to customize the behavior of the related commands:

- Setting environment options.
- Control the import/export of data.
- Toggling post-processing operations.

Settings are applicable when the EXPORT and IMPORT commands engage one of the Communicator for BricsCAD® file formats.

32.2.1 Perform import and export in background

Tick this option to enable user interaction while import/export is performed.

When the COMMUNICATORBACKGROUNDMODE system variable is ON and a file is imported, geometrical entities are inserted to the document after clicking the hyperlink with the filename from the notification balloon. This switches the document, regens viewport and zooms to extents.

Closing the notification balloon cancels the import session and deletes the memory allocated by the background import.

32.2.2 Import settings

System variable	Description
IMPORTPRODUCTSTRUCTURE	Controls the way a product structure is represented for an imported model. 'As mechanical components' automatically runs the BMMECH command after import.
IMPORTCOLORS	Controls how colors are converted during import.
IMPORTPMI	Enables the import of product and manufacture information.
IMPORTSTITCH	Automatically runs the DMSTITCH command on imported models. See the IMPORTIGESSTITCH system variable.
IMPORTREPAIR	Automatically runs the DMAUDITALL command on imported models.
IMPORTSIMPLIFY	Automatically runs the DMSIMPLIFY command on imported models. See also the IMPORTIGESSIMPLIFY system variable.
IMPORTHIDDENPARTS	Controls how hidden parts are imported.
CATIA V5 options	

System variable	Description
IMPORTCATIAV5REPRESENTATION	Controls the data that the Communicator imports during a Catia V5 import. Preview graphics are only imported and shown if the COMMUNICATORBACKGROUNDMODE system variable is on.
IMPORTCATIAV5SEARCHPATHSPREFERENCE	Controls the priority of file paths during a Catia V5 import.
IMPORTCATIAV5EDGEATTRIBUTES	Controls the import of edge attributes, by edge type, during a Catia V5 import.
Creo options	
IMPORTCREOALTERNATESEARCHPATHS	The alternate file used during a Creo import. Separate values with semicolons (;).
IGES options	
IMPORTIGESSTITCH	Automatically runs the DMSTITCH command during an IGES import. If on, overrides the IMPORTSTITCH system variable on IGES models.
IMPORTIGESSIMPLIFY	Automatically runs the DMSIMPLIFY command during an IGES import. If on, overrides the IMPORTSIMPLIFY system variable on IGES models.
Inventor options	
IMPORTINVENTORALTERNATESEARCHPATHS	Controls the list of alternate file system paths used during an Inventor file import. Separate values with semicolons (;).
IMPORTINVENTORSEARCHPATHSPREFERENCE	Controls the priority order of search paths during an Inventor file import.
JT options	
IMPORTJTREPRESENTATION	Controls the data to import during a JT import.

System variable	Description
NX options	
IMPORTNXALTERNATESEARCHPATHS	Controls the list of alternate file paths used during an NX import. Separate values with semicolons (;).
IMPORTNXSEARCHPATHSPREFERENCE	Controls the priority of file paths during an NX import.
IMPORTNXCONFIGURATION	Sets the name of the configuration that should be imported. If no configuration name is specified, then the part's default configuration will be imported.
SolidEdge options	
IMPORTSOLIDEDGEALTERNATESEARCHPATHS	Controls the list of alternate file paths used during a Solid Edge file import. Separate values with semicolons (;).
IMPORTSOLIDEDGESEARCHPATHSPREFERENC E	Controls the priority order of files paths during a Solid Edge file import.
SolidWorks options	
IMPORTSOLIDWORKSREPRESENTATION	Controls the data imported during a SolidWorks import. Preview graphics are only imported and shown if the COMMUNICATORBACKGROUNDMODE system variable is on.
IMPORTSOLIDWORKSALTERNATESEARCHPAT HS	Controls the list of alternate file system paths to search during a SolidWorks import. Separate values with semicolons (;).
IMPORTSOLIDWORKSSEARCHPATHSPREFEREN CE	Controls the priority order for search paths during a SolidWorks import.
IMPORTSOLIDWORKSCONFIGURATION	Sets the name of the configuration that should be imported. If no configuration name is specified, then the part's default configuration will be imported.

System variable	Description
IMPORTSOLIDWORKSROTATEYZ	Enables the conversion of a SolidWorks coordinate system to the current coordinate system.
STEP options	
IMPORTSTEPROTATEYZ	Enables conversion of a SolidWorks coordinate system to the current coordinate system, during a STEP import.

See more information about import settings in the articles dedicated to system variables.

32.2.3 Export settings

System variable	Description
EXPORTPRODUCTSTRUCTURE	Controls if a product structure is exported.
EXPORTHIDDENPARTS	Controls how hidden parts are exported.
EXPORTGEOMETRYFLAGS	Controls how geometry representations in IGES and STEP formats are exported.
ACIS options	
EXPORTACISFORMATVERSION	Controls the ACIS file version to export to.
EXPORTACISASSEMBLYWRITER	Controls the writer used to save ASAT/ASAB files. The internal ASAT/ASAB writer used if the Communicator is not installed.
CATIA options	
EXPORTCATIAV4FORMATVERSION	Controls CATIA V4 file version to export to.
EXPORTCATIAV5FORMATVERSION	Controls CATIA V5 file version to export to.
Parasolid options	
EXPORTPARASOLIDFORMATVERSION	Controls the Parasolid file version to export to.

System variable	Description
STEP options	
EXPORTSTEPFORMATVERSION	Controls the STEP file version to export to.
XCGM options	
EXPORTXCGMFORMATVERSION	Controls the XCGM file version to export to.
3D PDF options	
EXPORT3DPDFWRITER	Controls the writer used to save 3D PDF files.

See more information about export settings in the articles dedicated to system variables.

32.3 Compatibility between Communicator for BricsCAD® and BricsCAD

Communicator for BricsCAD® works with a BricsCAD Pro license or higher.

32.3.1 About

The two first product numbers must match. For example, BricsCAD 22.1.xx is only compatible with Communicator for BricsCAD® 22.1.xx. Previous BricsCAD versions (V21 or older) are not compatible.

There's one exception: For Communicator for BricsCAD® V21, all BricsCAD versions are compatible, provided that the major version number is 21.

Note:

- Communicator for BricsCAD® V22 installs and runs independently of previous versions.
- The compatibility between Communicator for BricsCAD® and BricsCAD is always mentioned in Release Notes for Communicator for BricsCAD®, under **Versioning**. The Release Notes for Communicator for BricsCAD® are available from the download page.

We strongly recommend using the latest product version of the major version number.

32.4 Communicator for BricsCAD® - Supported import file formats

32.4.1 Supported import file formats:

Windows

- **ACIS**: .asab, .asat, .sab, .sat
- **Autodesk Inventor**: .iam, .ipt
- **CATIA v4**: .model, .exp and .session
- **CATIA v5/v6**: .CATPart, .CATProduct, .CGR

- **Creo Elements / Pro Engineer**: .asm, .prt
- **IGES**: .iges, .igs
- **NX**: .prt
- **Parasolid**: .xmt_bin, .xmt_txt, .x_b, .x_t
- **Siemens**: .jt
- **SolidWorks**: .sldasm, .sldprt
- **STEP**: .step, .stp, .stpz
- **VDA-FS**: .vda
- **XCGM**: .xcgm
- **Solid Edge**: .asm, .par, .psm
- **STL**: .stl

Linux

- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4**: .model, .exp and .session
- **Creo Elements / Pro Engineer**: .asm, .prt
- **IGES**: .iges, .igs
- **Parasolid**:.xmt_bin, .xmt_txt, .x_b, .x_t
- **Siemens**: .jt
- **Solid Edge**: .asm, .par, .psm
- **SolidWorks**: .sldasm, .sldprt
- **STEP**: .step, .stp, .stpz
- **VDA-FS**: .vda
- **XCGM**: .xcgm

Mac

- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4**: .model, .exp and .session
- **Creo Elements / Pro Engineer**: .asm, .prt
- **IGES**: .iges, .igs
- **Parasolid**: .xmt_bin, .xmt_txt, .x_b, .x_t
- **STEP**: .step, .stp, .stpz
- **VDA-FS**: .vda

Note: Depending on the **Communicator** settings:

- You can import an assembly file as a plain geometry (a nonhierarchical set of entities) or map its product structure into native blocks or mechanical components, depends on the IMPORTPRODUCTSTRUCTURE system variable setting.
- You can define color conversion on import by setting the IMPORTCOLORS system variable.

- PMI (Product and Manufacturing Information) is supported for the following file formats: ACIS, CATIA V5, NX, Pro/E / Creo, SolidWorks, XCGM. When IMPORTPMI system variable is ON, PMI annotations are imported as graphical elements grouped in blocks.
- If the assembly contains hidden parts, you can decide how to represent them in BricsCAD: **import and hide**, **import and set visible** or just **do not import**. This feature is currently supported on CATIA V5 only, and it is controlled by IMPORTHIDDENPARTS system variable.
- You can set import options to automatically repair, stitch and simplify imported geometry, by setting the IMPORTSTITCH, IMPORTREPAIR and IMPORTSIMPLIFY system variables.
- The IGES format has also its own options to stitch and simplify (IMPORTIGESSTITCH and IMPORTIGESSIMPLIFY system variables).
- If you import a 3D model from SolidWorks or STEP, it makes sense to set an option to map Y axis into Z (IMPORTSOLIDWORKSROTATEYZ and IMPORTSTEPROTATEYZ system variables), because in SolidWorks the vertical axis is Y.
- For Creo, Inventor, NX, Solid Edge, and SolidWorks it is possible to specify an alternate search path (IMPORTCREOALTERNATESEARCHPATHS, IMPORTINVENTORALTERNATESEARCHPATHS, IMPORTNXALTERNATESEARCHPATHS, IMPORTSOLIDEDGEALTERNATESEARCHPATHS, and IMPORTSOLIDWORKSALTERNATESEARCHPATHS system variables). This will be used to look for parts absent in the imported assembly folder.
- For Creo, NX, and SolidWorks it is possible to specify the name of the configuration that should be imported (IMPORTCREOCONFIGURATION, IMPORTNXCONFIGURATION, IMPORTSOLIDWORKSCONFIGURATION system variables).
- For CATIA V5-6, Inventor, NX, Solid Edge, and SolidWorks it is possible to define priority order of search paths (IMPORTCATIAV5SEARCHPATHSPREFERENCE, IMPORTINVENTORSEARCHPATHSPREFERENCE, IMPORTNXSEARCHPATHSPREFERENCE, IMPORTSOLIDEDGESEARCHPATHSPREFERENCE, and IMPORTSOLIDWORKSSEARCHPATHSPREFERENCE system variables).
- When COMMUNICATORBACKGROUNDMODE system variable is ON, the Import/Export file is performed in the background. You can continue working on your CAD documents while Communicator for BricsCAD® imports/exports your file.

  When a file is imported, geometrical entities are inserted to the document after clicking the hyperlink with the filename from the notification bubble. This switches the document, regens viewport and zooms to extents.

Closing the notification bubble cancels the import session and deletes the memory allocated by the background import.

- For CATIA V5-6, JT, and SolidWorks format files the user can define which data representation should be imported by setting the IMPORTCATIAV5REPRESENTATION, IMPORTJTREPRESENTATION, and IMPORTSOLIDWORKSREPRESENTATION system variables.
- For CATIA V5 the user can define which edges attributes should be read during import by choosing an option for IMPORTCATIAV5EDGEATTRIBUTES system variable.

32.5 Communicator for BricsCAD® - Supported export file formats

32.5.1 Supported export file formats:

Windows

- **3D PDF**: .pdf
- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4**: .model
- **CATIA v5**: .CATPart, .CATProduct
- **IGES**: .iges, .igs
- **STEP**: .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS**: .vda
- **XCGM**: .xcgm
- **Parasolid**: .xmt_bin, .xmt_txt, .x_b, .x_t

  Export to Parasolid format is possible if:
  - pskernel.dll is available in InterOp, e.g. in a folder accessible through PATH enviroment variable.
  - P_SCHEMA environment variable points to the schema folder which can be found next to pskernel in its original installation. The folder name may slightly differ, e.g. in SolidWorks 2018 it is schema_18.
  - pskernel.dll version is earlier than the major version supported by InterOp, the EXPORTPARASOLIDFORMATVERSION system variable has to be set to a (lower) version that pskernel library supports.

Linux

- **3D PDF**: .pdf
- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4**: .model
- **CATIA V5**: CATProduct, .CATPart
- **IGES**: .iges, .igs
- **STEP**: .step, .stp

The following export protocols can be specified; AP203, AP214, AP242

- **VDA-FS**: .vda
- **XCGM**: .xcgm

Mac

- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4**: .model
- **IGES**: .iges, .igs
- **STEP**: .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS**: .vda

Note:

- When the EXPORTPRODUCTSTRUCTURE system variable is set, you can export a BricsCAD assembly with mechanical components as a file with product structure.
- If the assembly contains hidden parts, you can decide how to process the export hidden parts by setting the EXPORTHIDDENPARTS system variable.
- All other related system variables which controls the export file types are listed in the **Related variables**.

33. BricsCAD Shape

33.1 Introduction to BricsCAD Shape

33.1.1 What is BricsCAD Shape

BricsCAD Shape is the new conceptual modeling tool from Bricsys. It's made to help architects and engineers streamline their 3D schematic design work. BricsCAD Shape is the fastest way to get designs out of your head and in front of your clients. It's CAD-accurate, using a powerful direct modeling engine.

You can use it as a live presentation tool, too. Potentially best of all, BricsCAD Shape offers a clean, simple user interface that you can learn in 30 minutes or less.

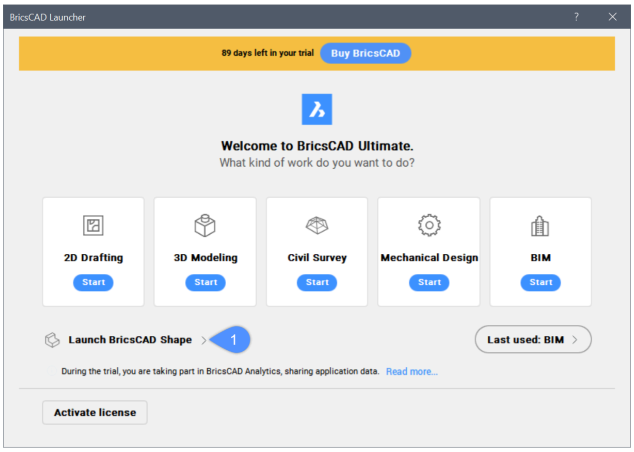
Important: BricsCAD V25 is the last version that contains the BricsCAD Shape product. BricsCAD Shape will be removed in the next major release of BricsCAD V26, which is currently planned for late 2025.

33.1.2 Activating BricsCAD Shape

1 Download BricsCAD.

2 Activate BricsCAD trial to be able to launch BricsCAD Shape.

3 In the **BricsCAD Launcher** click **Launch BricsCAD Shape** button (1).



Note: Once the BricsCAD trial period expires, BricsCAD Shape will remain active

33.1.3 BricsCAD Shape lets you create without limits

Shape's smart modeling tools make it easy to capture in 3D. The multifunctional Shape tool can be used to create walls, slabs, structural elements and more. Direct modeling features let you drag, connect, push/pull and extrude as you like. Wall connections are automatically mitered for you.

BricsCAD Shape also offers everything you need to present your designs. It comes with a library of materials, textures and pre-built 3D components. You can use the included collection of doors and windows, or create your own custom versions easily. Make your model look the way you want using the **Visual Styles** panel in Shape.

33.1.4 How is BricsCAD Shape different from other tools?

First, BricsCAD Shape is CAD-accurate from the start. Under the simple UI is the same fast and proven parametric, 3D direct modeling engine that's in BricsCAD Pro. Shape creates solid models, stored in industry-standard DWG – just like BricsCAD BIM. Unlike competing products, Shape's solid models are fully accurate – never approximated. Every element that you create in Shape can be modified deeply, anytime, on the fly. Your concept models open directly in BricsCAD BIM. You'll never lose time starting over again. Make the smart move – directly from concept to BIM with BricsCAD Shape with no loss of detail.

33.1.5 BricsCAD Shape accelerates your BIM workflow

The Bricsys BIM workflow starts in 3D with Shape, and stays in 3D – speeding 2D construction documentation with the world's best drafting tools. No breaks or transitions in your workflow, all in DWG, with a friendly, familiar user interface that you'll appreciate. It won't take long for you to be productive in BricsCAD Shape or BricsCAD BIM.

33.1.6 Stop sketching. Start shaping!

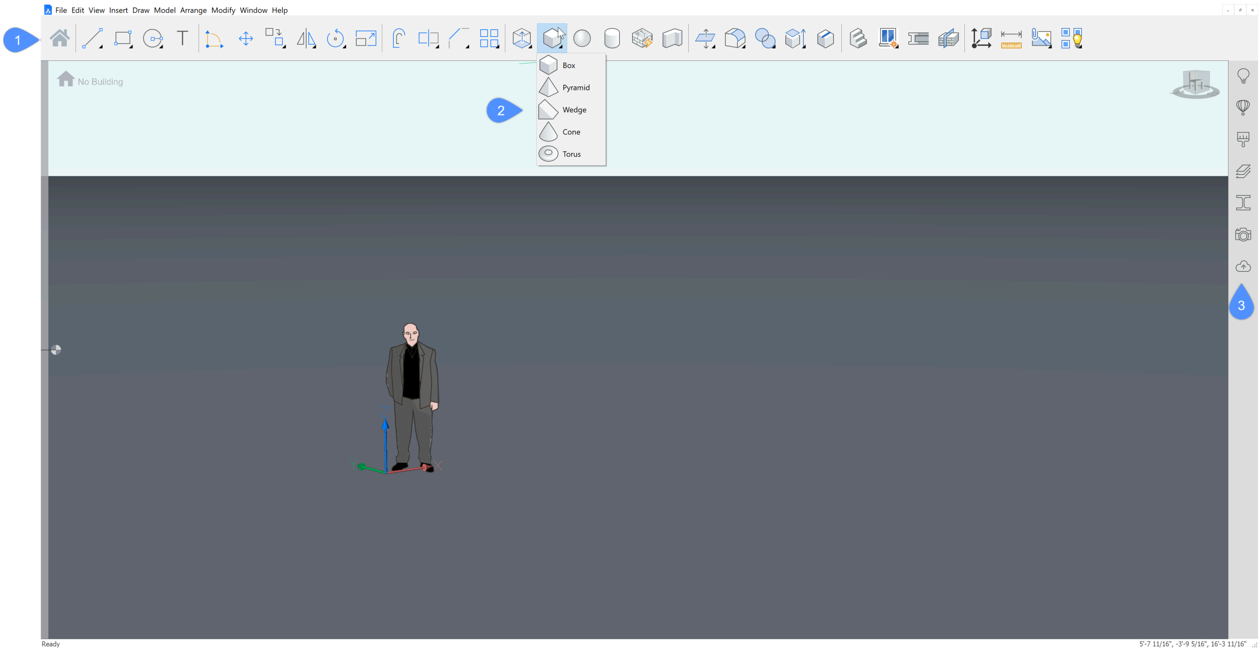
Get your free copy of BricsCAD Shape! BricsCAD Shape delivers the tools you need, and none that you don't. Even the learning materials are clean and simple – check out "10 Minutes to BricsCAD Shape" on YouTube, Bricsys channel.

33.2 BricsCAD Shape user interface

The toolbar at the top of your screen (1) gives you fast access to the most used tools. Left-click on any icon in the toolbar to use the tool.

Many top-level tools offer sub-menus (2). Hover over the icon, left click and hold. The sub-menu will fly out to show these additional tools.

The icons on the right side of your screen (3) give you access to a set of collapsible panels. The top-most icon opens the **Tips panel**. You'll find more details on how to use the tools in Shape here.



Note: Many of the modeling tools that are used in BricsCAD and BricsCAD BIM are also available in Shape. You can get an overview of all available commands in Shape by typing in COMMANDS in the Command line.

33.3 Basic concepts of modeling in BricsCAD Shape

33.3.1 Selection methods

Highlighting and selecting objects

Entities in BricsCAD Shape are solids, which are made up of edges and faces.

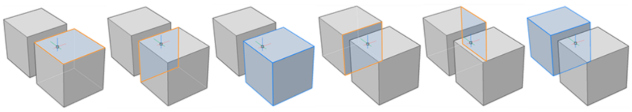
When hovering over an entity, a part of the entity is highlighted. To select the highlighted part of the entity, left-click the edge or face.

Note: To select or highlight the whole entity, hold down the **Ctrl** key and left-click.

For some operations, it's not necessary to select entities. When a surface or edge is highlighted, the **Quad** opens and several commands can be accessed.

Note: You can only highlight one entity at a time, while it is possible to select multiple entities at once by left-clicking them one-by-one, or using a window selection method.

Sometimes, the entity you want to select is hidden behind other entities. Place the cursor over the obscured entity, then repeatedly press the **TAB** key. All (sub)entities under the cursor highlight one by one. Click to select the currently highlighted (sub)entity.



To clear a selection of objects, press the **ESC** button.

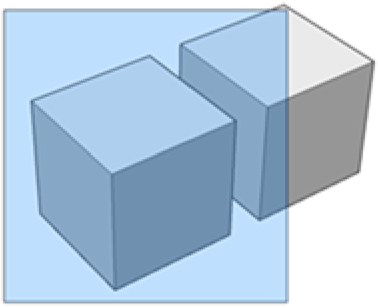
Selection windows

To select one or multiple elements at once, a selection window is used. Left click on an empty space in the model area. When you move the cursor a selection window displays.

> ***Note:*** The selection window selects entire solids by default. Press the **Ctrl** key to select faces, press the **Ctrl** key again to select edges and again to select solids. The current selection method is indicated in the **Hot Key Assistant** (HKA) at the bottom of the screen.

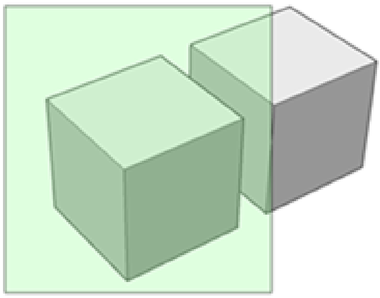
Blue selection window

Moving the cursor from left to right, a blue selection window displays: all entities that are completely inside the rectangle are selected. This selection method is called ‘Window Inside‘.



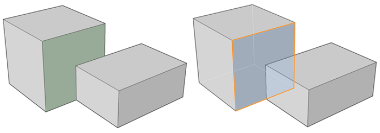
Green selection window

Moving the cursor from right to left, a green selection window displays: all entities that are completely inside or overlapping the selection rectangle are selected. This selection method is called ‘Window Overlap’.



Boundaries vs Faces

Boundaries can be defined by a set of 2D entities in the XY field of the current coordinate system or on a 3D solid face or by partially overlapping faces of 3D solids. BricsCAD Shape automatically detects and highlights closed boundaries, as you hover over different model elements. When a boundary is detected, it will be highlighted. To select it, just left-click inside the highlighted boundary. Boundaries can be used to create new solids or making holes or offsets and recesses.



> ***Note:*** To select the entire solid face instead of a boundary, press the **TAB** key.

33.3.2 Quad

The Quad cursor menu, or Quad for short, is an alternative to the Command line, toolbars or ribbon, offering a rich set of tools while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs. It includes rollover tips, which display a limited set of properties when selecting or hovering over an entity.

The Quad displays when:

- You hover over an entity.
- A selection set exists.
- You right click on an empty space on the screen. This will open the No Selection Quad.

The initial content of the Quad when hovering over an entity is:

- Entity type.
- The icon of the most recently used command on this entity type.
- Relevant properties.

When you move the cursor to the Quad, it expands. On top you can find the most relevant or most recently used commands for the entity. Below you see a series of command tabs:

- Move the cursor over a tab to expand it.
- Place the cursor over an item to see a tool tip.
- Click an icon to launch a command.
- Click the Quad title bar to collapse, click again to expand.

33.3.3 Manipulator

The manipulator allows you to move, rotate scale and mirror selected entities and optionally create copies.

To display the Manipulator do one of the following:

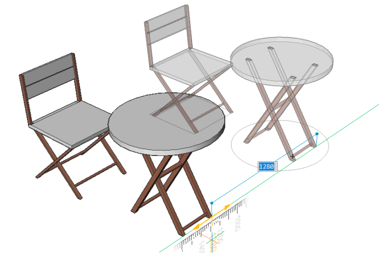
- Select the entity or subentity and choose **Manipulator** in the **Modify** tab on the **Quad**.
- Hold down the left mouse button a little bit longer when selecting.

Note: The Manipulator aligns along the edge that you enter the entity by. Its origin sits at the closest endpoint of that edge.

Select an axis

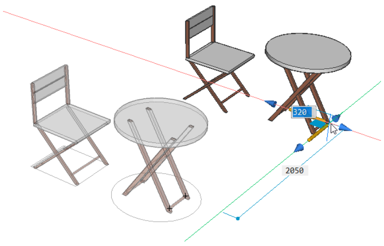
By selecting an axis of the manipulator you are able to move the entity. Specify the value of displacement by entering a value or picking a point. The step size of the ruler depends on the zoom factor. Zoom in to decrease the step size, zoom out to increase.

Note: To copy the entity, hold down the **CTRL** key before clicking the axis.



Select a plane

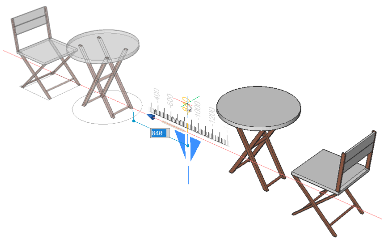
By selecting a plane of the manipulator you are able to move the entity in the selected plane after specifying the value of displacement by entering a value or picking a point.

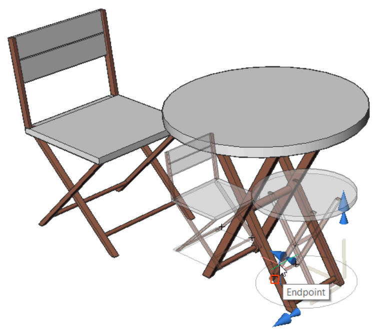


Select an arrowhead

By selecting an arrowhead of the manipulator you are able to scale or rotate the selected entity. By right clicking when hovering over the entity, a contexts menu with other options is displayed.There the function of an arrowhead can be changed between 3D mirror and 3D scale.

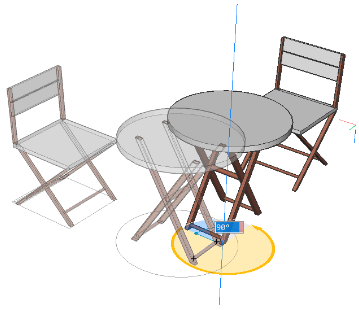
Select one of the arrowheads and specify the mirror axis or scale.





Select a rotation arc

By selecting an arc of the manipulator you are able to rotate the entity after entering the rotation angle value or specifying by clicking on a point in the drawing.



Select the anchor handle

By selecting the anchor of the manipulator you are able to relocate the manipulator.

33.4 Navigation in BricsCAD Shape

33.4.1 Panning

You can move the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space.

There are several ways to pan:

- Hold down the middle mouse button.
- Press and hold the **CTRL+SHIFT** and right mouse button.
- Type the *PAN* in the Command line.
- Type the *RTPAN* in the Command line.
- Click the **Real-Time Pan** icon in the toolbar.

To abort the *PAN* command, do one of the following:

- Right click and choose **Exit** in the context menu.
- Press **Enter** key, **space bar** key or **Esc** key on the keyboard.

33.4.2 Rotating a view

You can rotate a view freely, in a constrained manner.

To rotate in real-time:

- Hold down the **SHIFT** and middle mouse button.
- Type *RTROT* in the Command line.
- Click the **Real-Time Constrained Sphere** icon in the toolbar.

To abort the *RTROT* command, do one of the following:

- Right click and choose **Exit** in the context menu.
- Press **Enter** key, **space bar** key or **Esc** key on the keyboard.

33.4.3 Zoom in/out

You can change the magnification of your drawing at any time by zooming.

Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail.

Changing the magnification of the drawing affects only the way the drawing is displayed. It has no effect on the dimensions of the entities in your drawing.

To zoom in/out:

- Scroll the mouse wheel.
- Press and hold the **CTRL+SHIFT** and left mouse button to zoom in or out.
- Type **ZOOM** in the Command line.
- Click the **Zoom In** or **Zoom Out** icons in the toolbar.

Note: **Zoom Extents** displays the drawing to the extents of entities

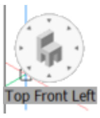
33.4.4 Look From control

The **Look From** control allows you to select a number of preset views such as orthographic views and isometric views. By default the **Look From** control displays in the top right corner of the graphic screen.

1. Move the cursor to the **Look From** control.

   The **Look From**control becomes active.
2. Position the cursor on the **Look From** control to choose a view orientation.

   A preview image and a tooltip indicate the currently selected view orientation.

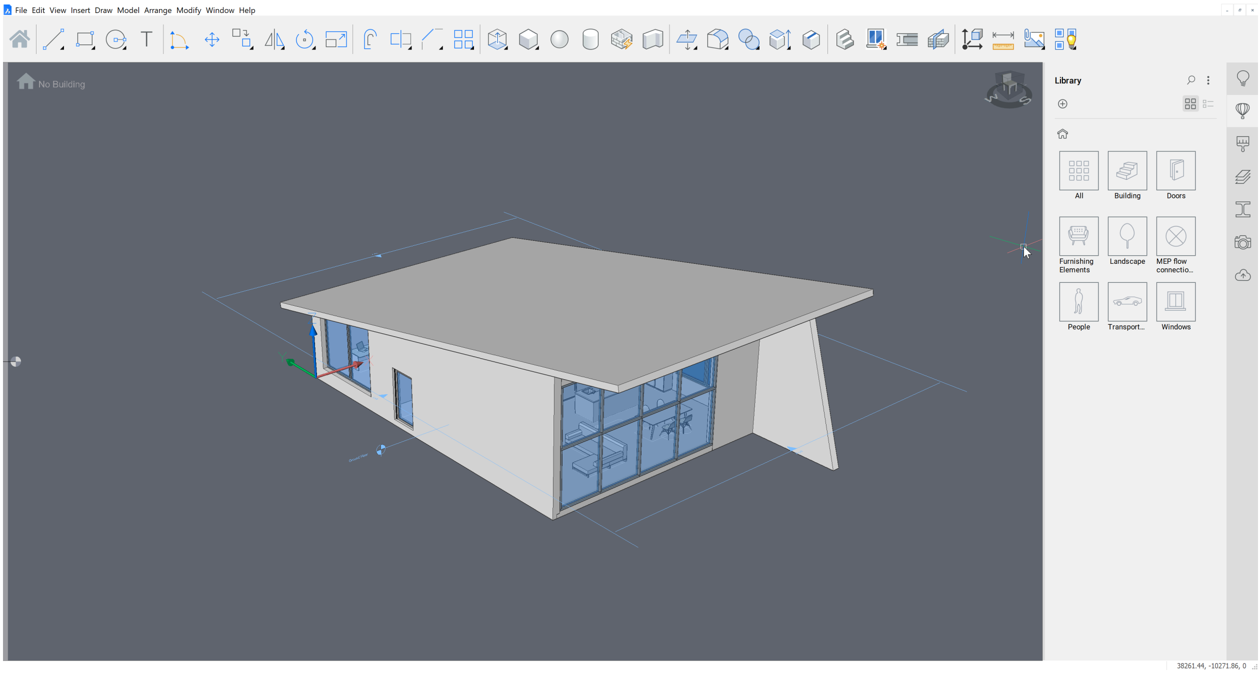
   

3. Click to confirm.
4. The view is updated.

33.5 Components in BricsCAD Shape

BricsCAD Shape includes a collection of components in the **Library** panel.

Open the library by clicking the airballoon icon on the right hand edge of the screen. Several categories are available.



Click the menu button at the top right corner of the panel to select or hide the available libraries:

- **Bricsys BIM library**: content of *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support \en_US\Bim\Components*.
- **Bricsys 2D library**: content of *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support \en_US\Blocks2D*.
- **User library**: content of *C:\ProgramData\Bricsys\Components*.

Note: Choose **Manage Libraries** in the menu to add folders to the COMPONENTSPATH system variable in the **Settings** dialog box.

33.5.1 To insert a component

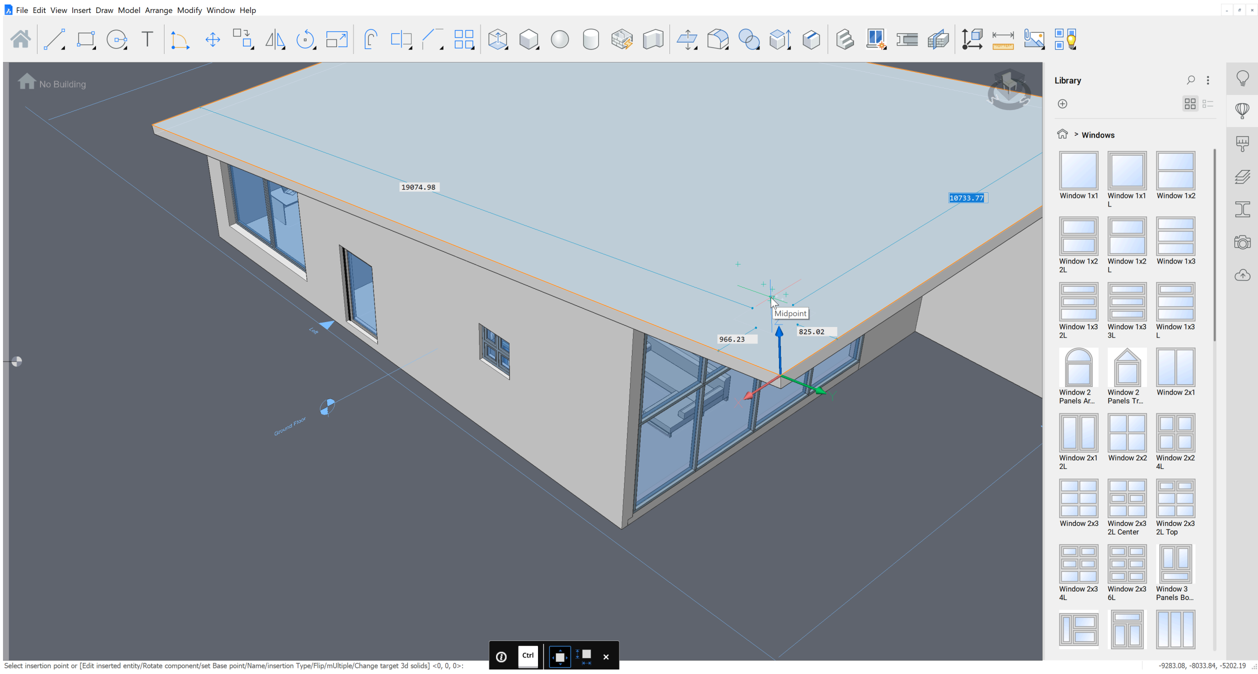
1. Click the component in the library.
2. Move the cursor into the model.

   Note: The selected component is attached to the cursor. The X and Y distances from the origin of the current coordinate system display dynamically.

3. Do one of the following:
   - Click to place the component.
   - Type a value in the X and Y distance fields.

     Note: Press the **TAB** key to jump to the other field. Press Enter to place the component.

33.5.2 To insert a window in the face of a solid



1 Click the window in the library.

2 Move the cursor into the model over the face of 3D solid.

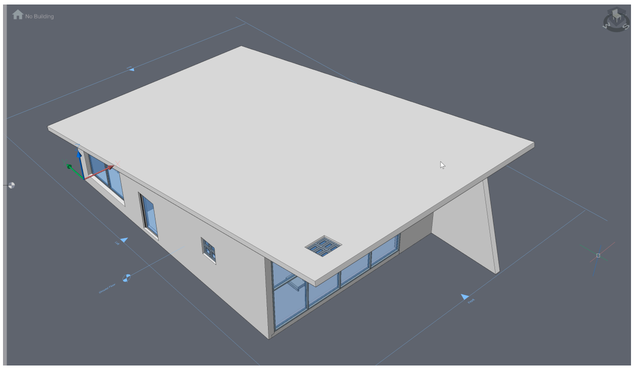
Note: The window aligns to the face under the cursor. Distance fields from the sides of the face display dynamically.

3 Do one of the following:

- Click to insert the window.
- Type a value in distance fields.

  Note: Press the **TAB** key to jump to the next field. Press Enter to place the component.

  The window automatically makes an opening in the solid.



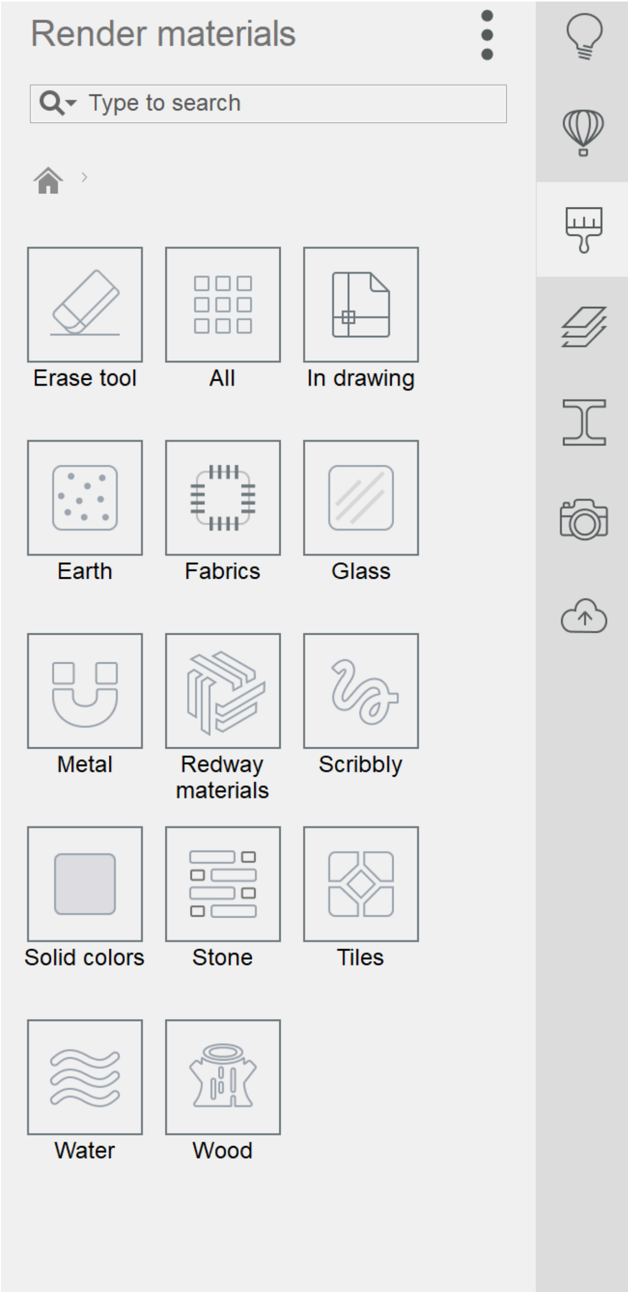
Note: When you delete a window, the opening is removed.

33.6 Materials in BricsCAD Shape

BricsCAD Shape includes a collection of materials in the **Render materials** panel.

Render materials are stored in: *C:\ProgramData\Bricsys\RenderMaterials\DefaultMaterials*.

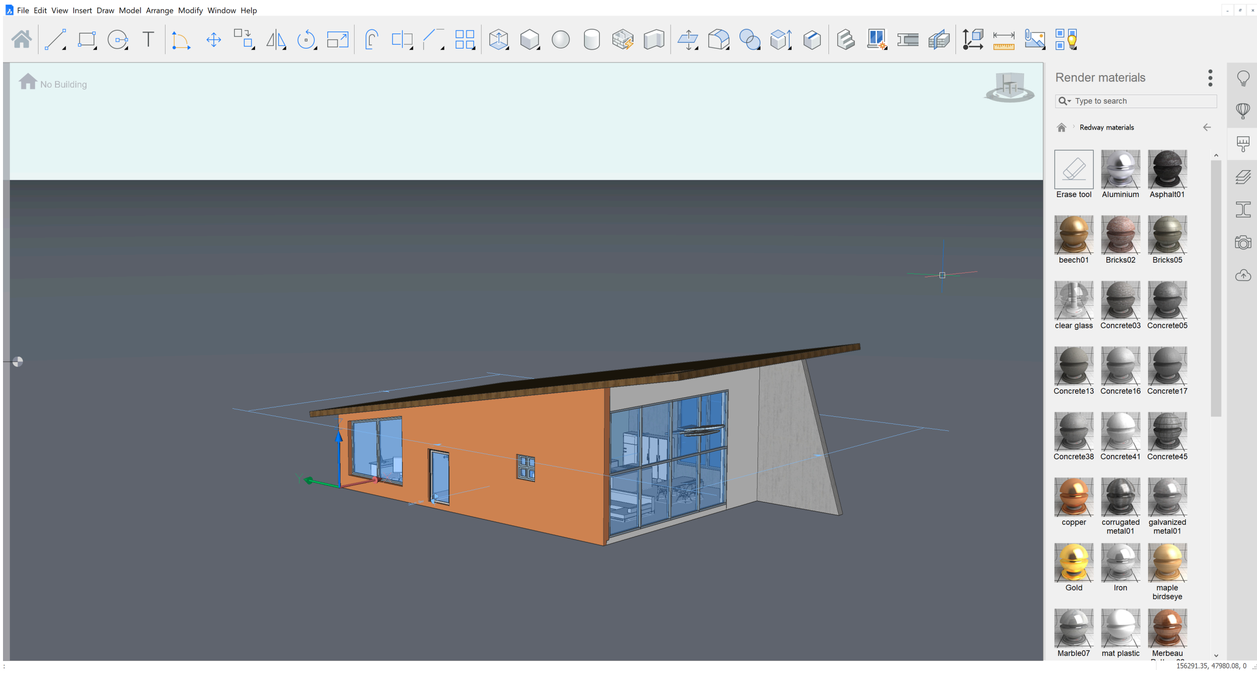
Open the **Render materials** panel by clicking the **paintbrush** icon on the right-hand edge of the screen. Several categories are available. To view all materials that are used in the drawing, select the **"In drawing"** category.



To assign a material to an entity in your model:

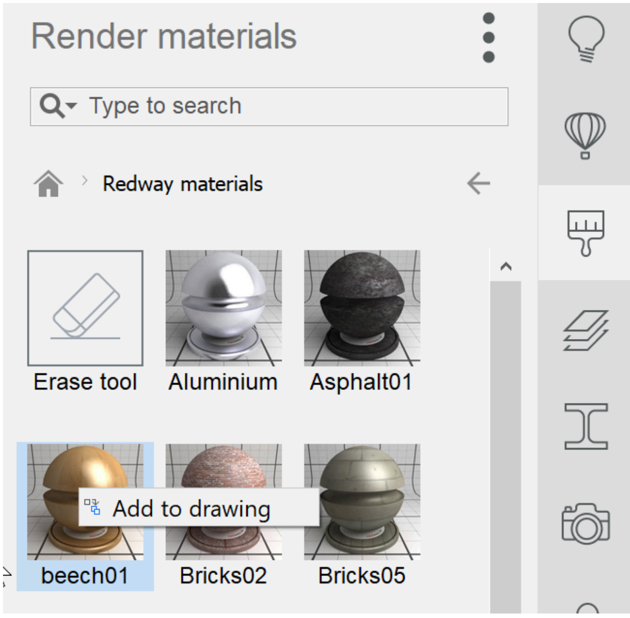
- Select the material in the **Render materials** panel.
- Select the entity you want to assign the material.

Note: You can apply a material to multiple entities at once by preselecting them.



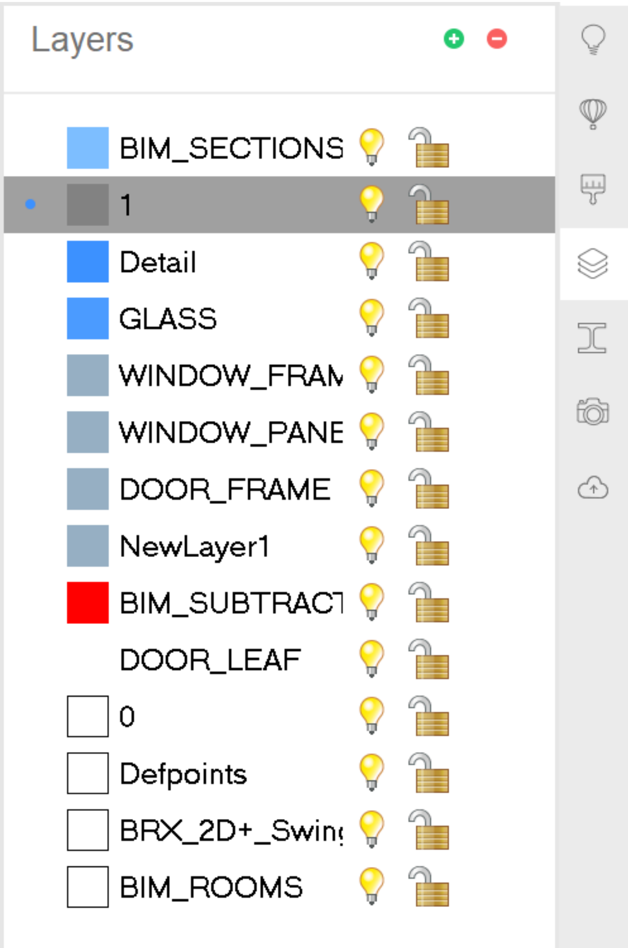
Note: To remove a material from an entity, use the erase tool and click any entity to remove the material from.

It you want to add materials to the drawing to use later, right-click the material icon and select the "**Add to drawing**" option.



33.7 Layers in BricsCAD Shape

Layers can be used to group entities to organize your 3D model. The **Layer** panel can be accessed through the stacked sheets icon on the right side of the screen.



Note: The current layer is indicated with the small blue dot. To change the current layer, click in the same column next to the layer you want to set as current layer.

33.7.1 Turn on/off

To turn on and of specific layers, toggle the lightbulb of the layer on and off. This way elements can be hidden or isolated. This can be of huge help with navigation.

- **On**: Displays entities that are on that layer.
- **Off**: Hides entities that are on that layer.

33.7.2 Lock/unlock

To lock or unlock layers, toggle the lock icon of the layer.

- **Unlocked**: Allows entities on that layer to be edited.
- **Locked**: Prevents entities on that layer from being edited.

33.7.3 Create/delete

To create a new layer, click the **green plus** sign. A new layer with a generic name is created. To delete a layer, make sure the layer to delete is selected and click the **red minus** sign.

33.7.4 Edit

Properties such as name and color of layers can be edited:

- To rename a layer, double-click the layer name or select **Rename** from the right-click menu.
- To change the color, select one or more layers and then click the current color for one of the selected layers. The **Color** dialog box is displayed. The color you select is applied to all selected layers.

33.8 Working with steel profiles

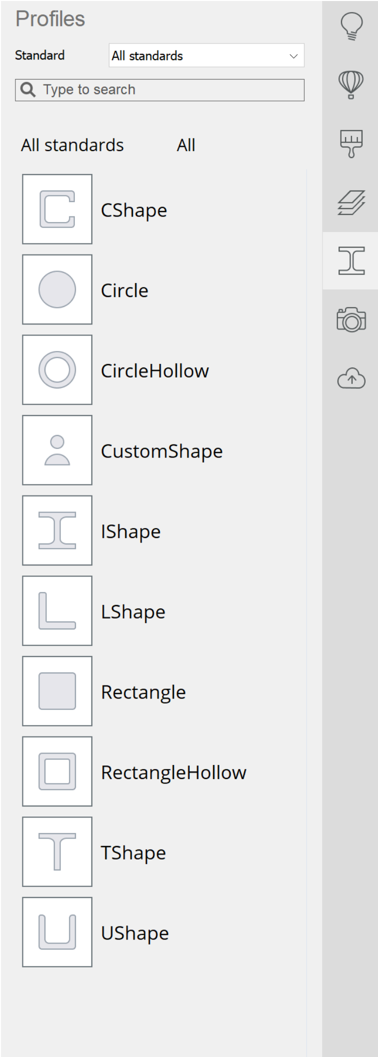
BricsCAD Shape offers standard structural steel profiles which enable you to easily create structural elements. Multiple standards are available, including American, British and European standards.

Steel profiles are created as linear solids.

33.8.1 Creating a steel profile

1 Click the **Profile** icon the vertical toolbar, on the right-hand edge of the screen.

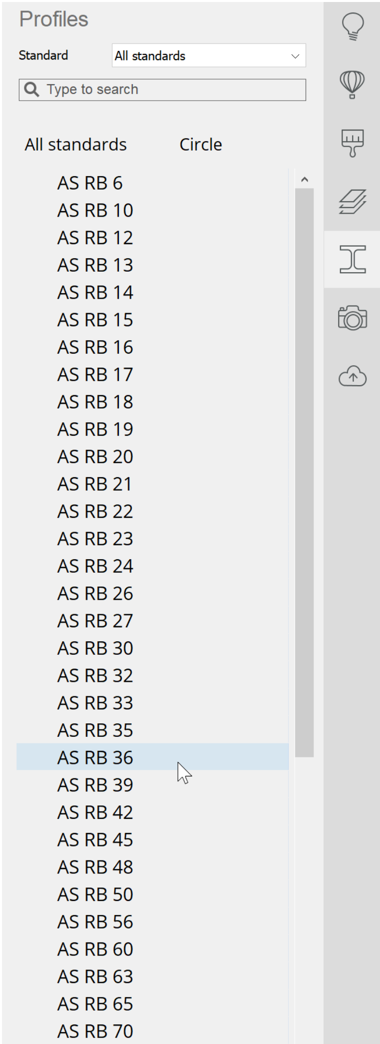
The **BIM Profiles** panel displays.



2 Optionally select a standard.

3 Select a profile shape.

4 A list of all profiles of the selected shape displays.



5 Click a profile in the list, then move the cursor to the model.

6 The outline of the selected profile is attached to the cursor.

7 Specify a point.

8 A linear solid displays dynamically.

A length and angle field display the current length an angle of the profile.

When the current direction is close to the X, Y or Z direction of the current coordinate system a colored tracking line displays. Press the **Shift** key to lock the tracking line. Press again to unlock.

9 Do one of the following to create the profile:

- Specify a point.
- Type a value in the length field, then press **Enter** or press the **TAB** key to activate the angle field.

The next profile displays dynamically.

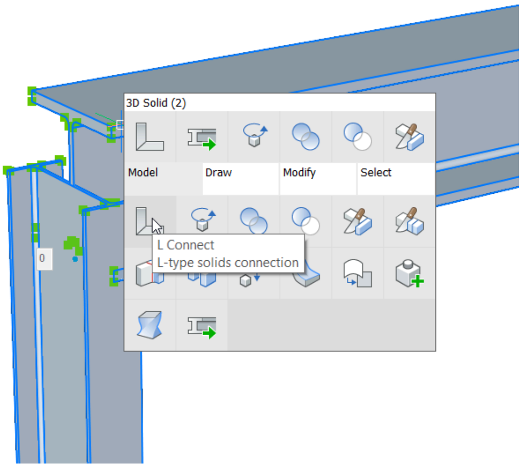
10 Repeat the previous steps to create more profiles or right click to stop.

33.8.2 Connecting Profiles

The L-Connect tool creates various types of L-connections between two selected profiles.

1 Select both profiles.

2 Click the L-Connect tool in the Model tab on the Quad.



- A bisector L-connection is created.
- The **Hotkey Assistant** (HKA) displays at the bottom of the screen.

3 Press the **Ctrl** key to modify the connection type.

The current connection type highlights in the **HKA**.

4 Right click to accept the connection type.

33.8.3 Assigning a different profile

1 Place cursor over the new profile in the **BIM Profiles** panel. Do not click.

2 Hold down the left mouse button, then drag the profile onto the linear solid.

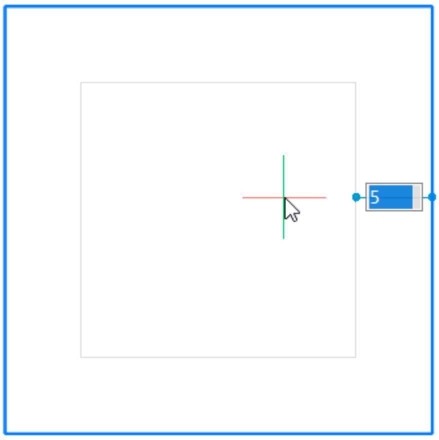
The selected profile is assigned to the linear solid.

33.8.4 Assigning a custom profile

1 Draw a custom profile shape using the tools in the **Draw** tab on the **Quad**:

- A closed polyline.
- A square or rectangle using the **Rectangle** tool.
- A circle.
- An ellipse.

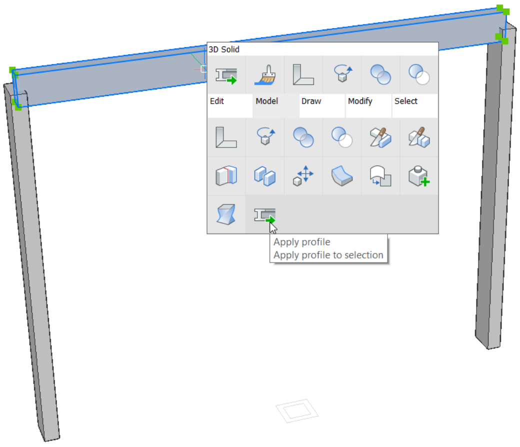
2 Optionally, use the **Offset** tool to create a hollow profile: select the outline, then specify the offset.



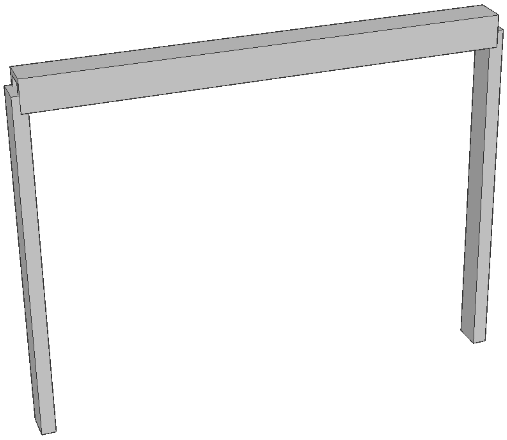
3 Hold down the **Ctrl** key, then select one or more linear solids. The linear solid(s) highlight(s).

4 Click the **Apply profile** tool in the **Model** tab on the **Quad**.

Prompts you: Select profile [selection options (?)]:



5 Do the following:
- Select the custom profile.
- Use the window overlap selection method to select the profile outline and the offset when applying a hollow profile.
- A preview of the result displays.
- Prompts you: Apply profile? [Ok/Quarter turn/Rotate/Delete clippings] <Ok>:



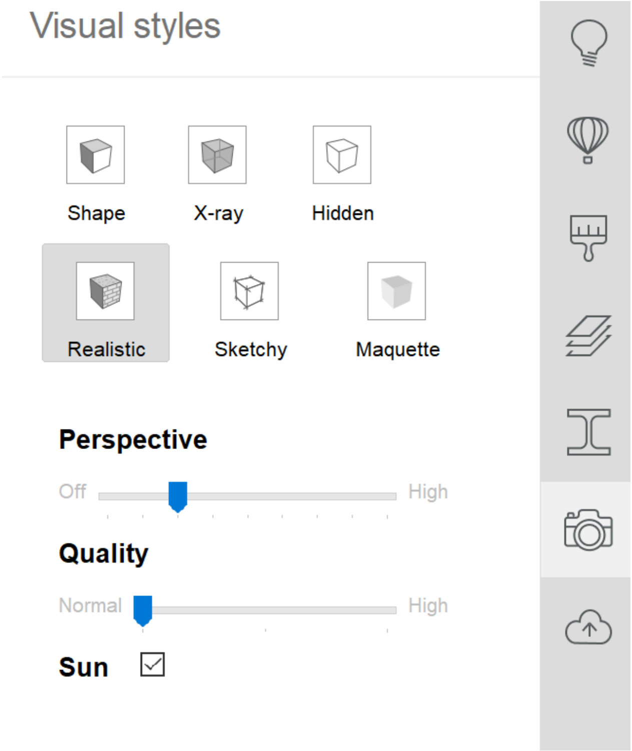
6 Do one of the following:
- Right click to apply the profile.
- Choose an option.
- Press the Esc key to quit.

33.9 Visualization in BricsCAD Shape

33.9.1 Visual styles panel

Visual styles control the appearance of your model.

The **Visual Styles** panel can be accessed by clicking the camera icon at the right side of the screen.



The look of the entire model will be changed when selecting another style.

The available visual styles are:

- Shape
- X-ray
- Hidden
- Realistic
- Sketchy
- Maquette

Perspective

You can change the perspective view by moving the **Perspective** slider.

Note: Moving the perspective slider all the way to the left will turn the perspective off.

Quality

To control the on-screen rendering quality of the model, move the **Quality** slider.

Sun

To make the shadows of the sun appear, toggle the **Sun** checkbox.

33.9.2 Hide/Isolate/Show entities

Hide

When model elements block your view, you can hide them easily.

To hide one or more elements:

- Select the entities you want to hide.
- Click the **Hide Entities** icon in the **Quad** or in the toolbar.

Isolate

The opposite of hiding an element is isolating it, which will hide all the non-highlighted elements in your model.

To isolate entities:

- Select the entities you want to isolate.
- Click the **Isolate Entities** icon in the **Quad** or in the toolbar.

Show

Hiding and Isolating are temporary functions that you can use to simplify your model while working on it.

When you want to see all the elements in your model:

- Select the **Show Entities** icon in the "No Selection" Quad or in the toolbar.

33.9.3 Sections

You can use Sections to cut through your model dynamically.

To create a section:

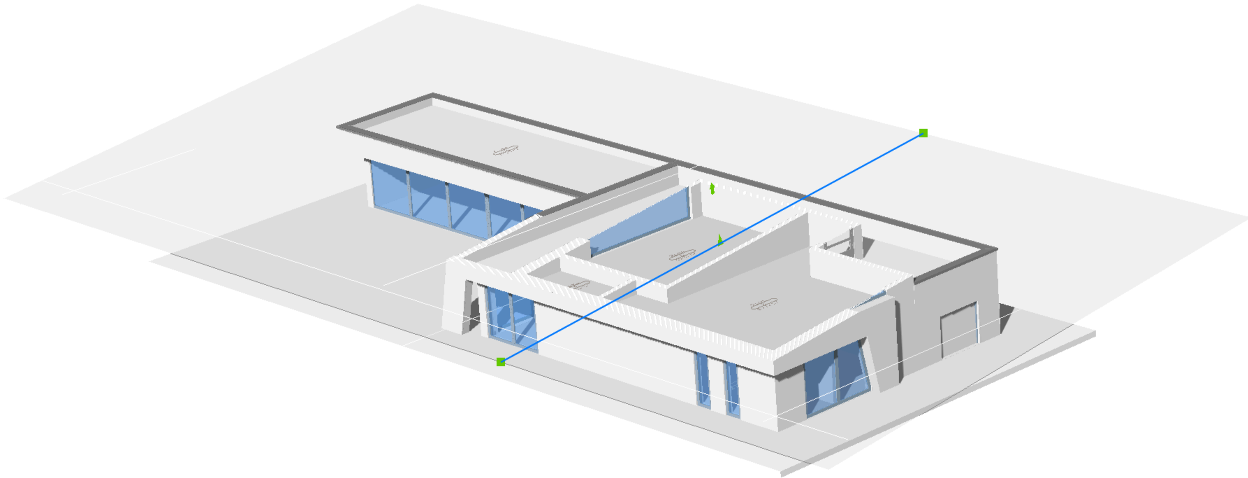
- Select the **Define Section** tool in the toolbar.
- Select any solid face in your model to create the section, parallel to the selected solid face.

Note: If you don't choose a surface to attach the Section, a horizontal Section plane will be created.

You can toggle a section on and off by double-clicking its handle.

A Section element has three grip edit points. Selecting the Section's handle turns the grips on for editing:

- You can move the start and end grips to change the orientation and width of the Section.
- You can move the midpoint grip to reposition the section plane.
- If you click the arrow displayed near the midpoint grip, you will flip the clipping direction of the Section.



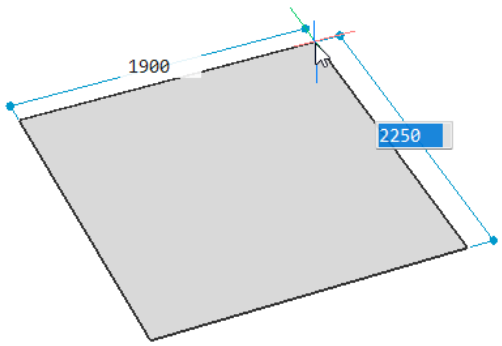
33.10 Solids in BricsCAD Shape

33.10.1 Drawing a box

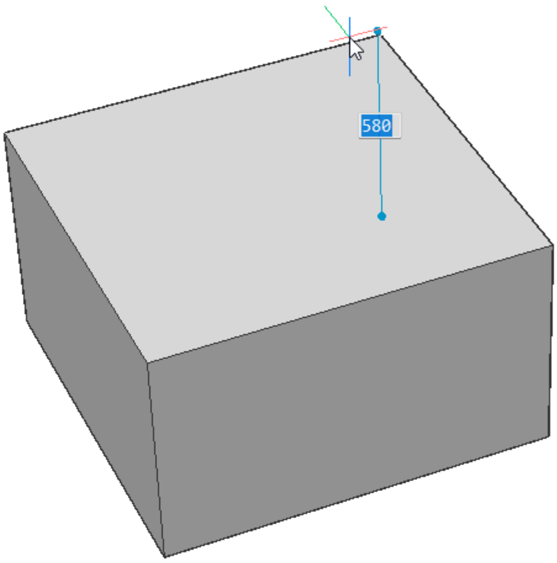
1. Start creating a box by accessing the *BOX* command through the command line or by clicking the **Box** icon in the Quad or toolbar.
2. Specify a corner for the base of the box.
3. Specify the opposite corner for the base of the box to apply both the length and width.
4. You can do this dynamically, by hovering the cursor at the opposite corner, or by entering the desired values in the dynamic input fields.

   The box is created parallel to the x and y axes.

   Note: The blue dynamic field is the active one.



5 Specify the height of the box.



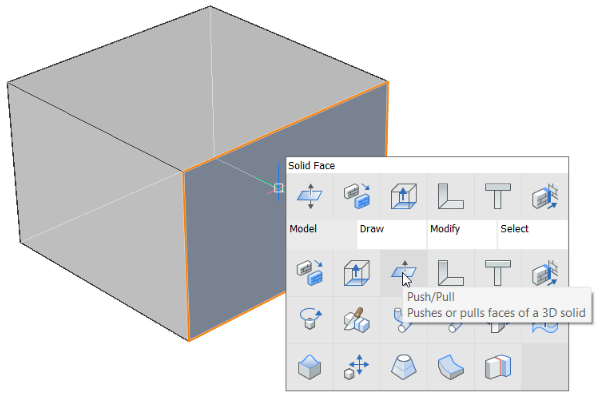
33.10.2 Manipulate a solid face

To manipulate a solid face:

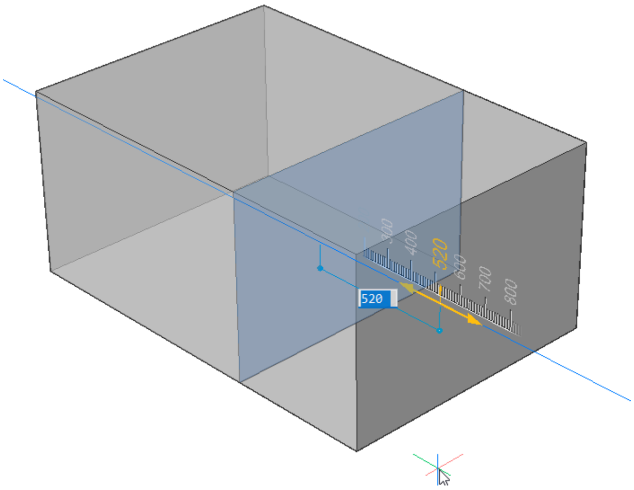
1 Hover the cursor over a face until highlighted.

2 Hover over the appearing icon to open the **Quad**.

3 Under the **Model** tab, the *DMPUSHPULL* command can be accessed.



4 Specify the displacement value or move with your cursor to define the end position of the face.



Note: When selecting multiple faces and accessing the command, both faces will be pushed or pulled over same distances.

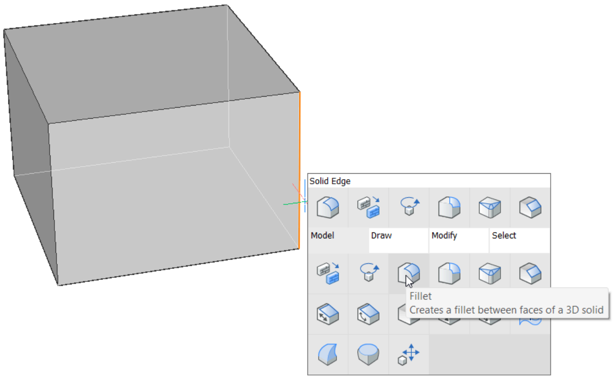
33.10.3 Manipulating a solid edge

To manipulate a solid edge:

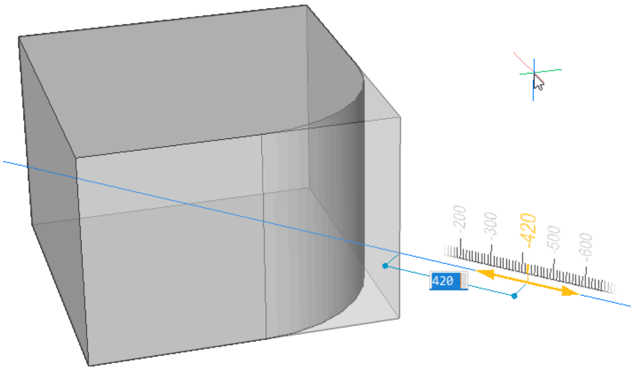
1 Hover your cursor over an edge while holding the **CTRL** key until highlighted.

2 Hover over the appearing icon to open the **Quad**.

3 Under the **Model** tab, the *FILLET* command can be accessed.

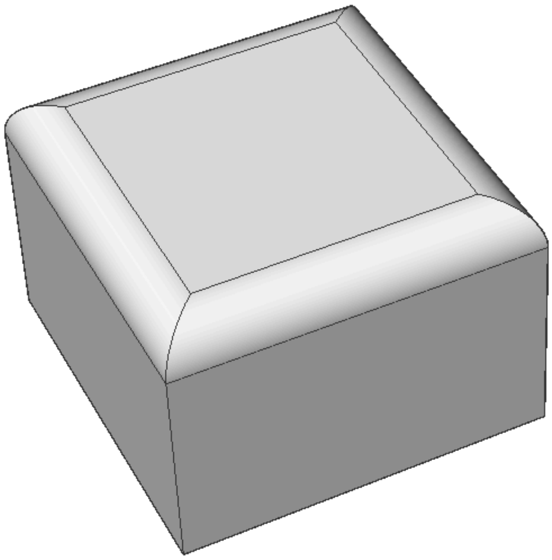


4 Specify the displacement value or move with your cursor to define the end position of the face.



Note: When selecting multiple edges and accessing the command, all selected edges will have a fillet of the same radius.

Note: When using a selection window to select edges, make sure you press the **CTRL** key twice so the edge selection icon is selected at the bottom of your screen.



33.10.4 Extrude solids

Selected boundaries can be extruded to create solids or surfaces. For example, a floor slab can be extruded between existing walls.

The extrude command can be accessed by the Quad or in the toolbar.

Note: The extrude command can also subtract geometry also.

33.11 Creating walls in BricsCAD Shape

There are two ways to create a simple wall:

- Using the POLYSOLID command.
- Using the BIMQUICKDRAW command.

Note: For a detailed explanation about BIMQUICKDRAW, read the article **Quickdraw in BricsCAD Shape**.

33.11.1 About the Polysolid tool

The POLYSOLID command allows you to create wall solids by selecting a start point, an endpoint and entering a height.

33.11.2 Procedure: Create a simple wall

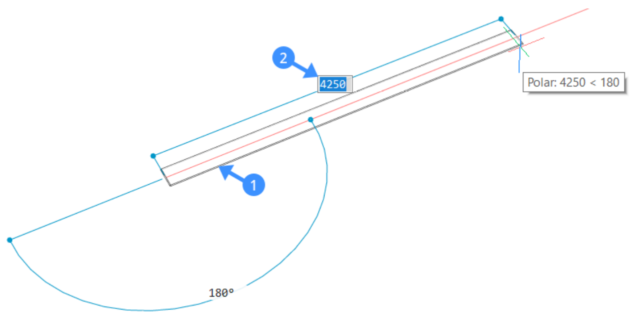
1 Launch the Polysolid command:
   - Click the **Polysolid** icon

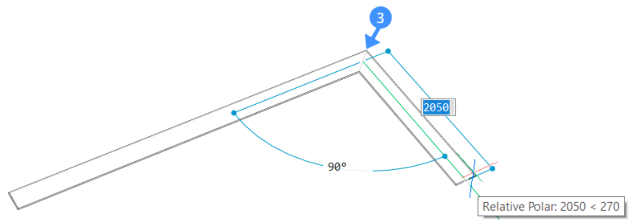
     in the toolbar.
   - Type POLYSOLID in the command line.

2 Select a start point, the footprint (1) of the wall is displayed automatically.

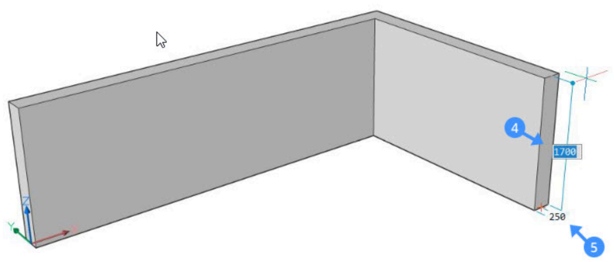
3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).



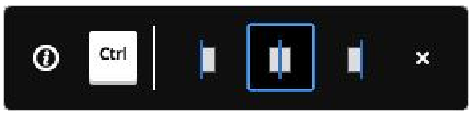
4 Select an endpoint or enter a value in the dynamic input field and press **Enter** key. The footprint (3) of the adjacent wall is displayed. Select a new endpoint or enter values in the dynamic input field to create adjoining wall segments.



5 Press **Enter** once more or right-click. The height of the wall is displayed dynamically in the height field (4). Here, you can change the default height of the wall. To change the width of the wall, use the width field (5). You can switch between the two dynamic input fields by pressing **Tab** key.



6 To change the justification of the wall hit **CTRL** while the 'Polysolid justification Hotkey Assistant widget' appears at the bottom of the screen.



Note: The width and height of the previous wall will be the new default values of the next Polysolid.

33.11.3 Procedure: Using an existing 2D plan to create walls

1 Launch the POLYSOLID command.

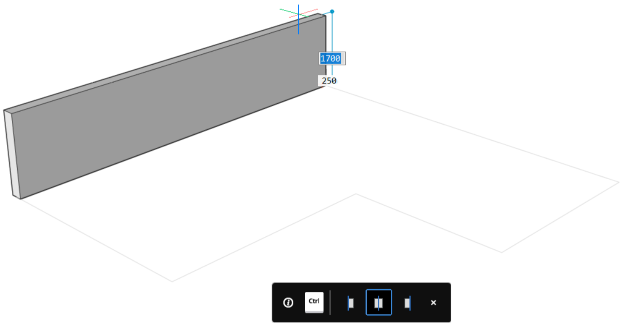
2 Type E in the command line or select **Entity** in the prompts menu.

3 Choose a 2D linear entity which will be the Polysolid base. Lines, open and closed polylines, arcs, circles, ellipses, elliptical arcs, and splines are accepted as a polysolid base.

4 Move the solid up or down and left-click to set the height of the wall, enter a value in the dynamic input field or right-click to accept the default value.

5 The current width of the wall appears in the **Width Field**. Press **Tab** switch between the Height and the Width field.

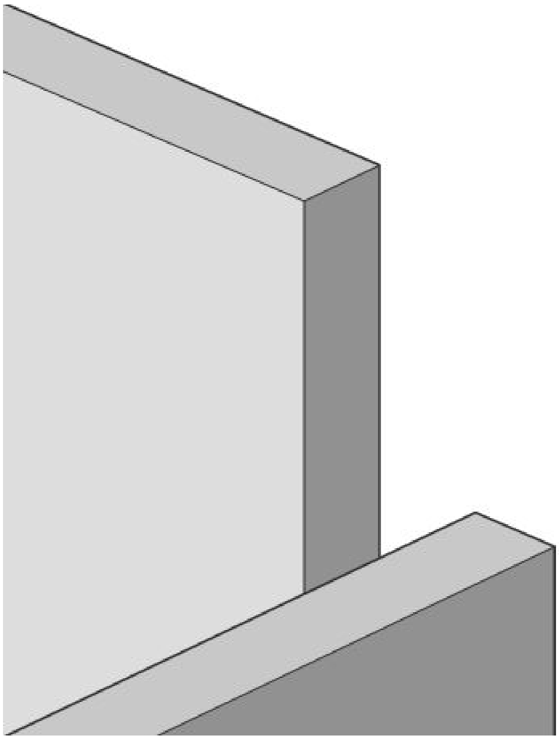
6 The **Hotkey Assistant** appears and displays the possible justification options. Press **CTRL** to cycle through the wall justification options.

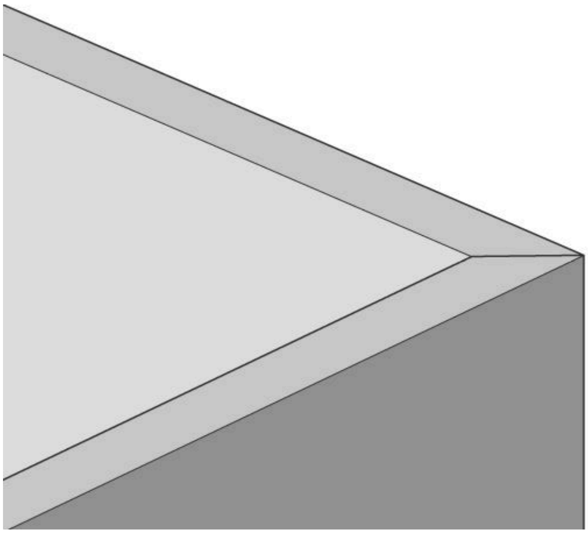


33.11.4 Procedure: Creating L-connections between two walls

1 Launch the **LConnect** tool, by typing LCONNECT in the Command line and press Enter.

Select the two wall elements you want to connect.

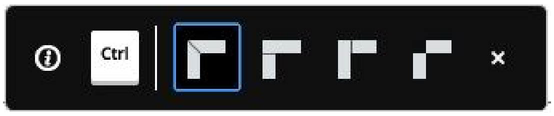




Note: When two walls intersect, a mitered connection will be created by default.

2 Change the layout of the wall connections using the ‘LConnect HotKeyAssistant widget’.

3 The selected connection type is indicated by a blue frame.



The connection can be (from left to right) a bisector L-connection , L parallel type 1 , L parallel type 2 or a disconnection.

Press **Enter** to accept the current connection type or press **CTRL** to cycle through the possible connection options.

33.11.5 Procedure: Creating T-connections between two elements

1 Launch the TCONNECT command.

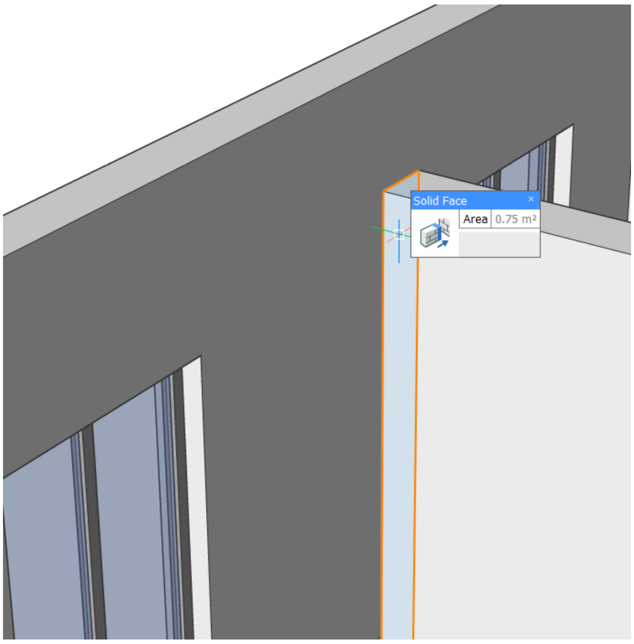
1 The tool allows you to connect the minor face of a wall to the major face of another element or wall. This tool can be used to connect the top face of a wall to a roof, the bottom face to a slab or side face of a slab to a wall.

2 Select the faces of the objects that you would like to connect and press **Enter**.

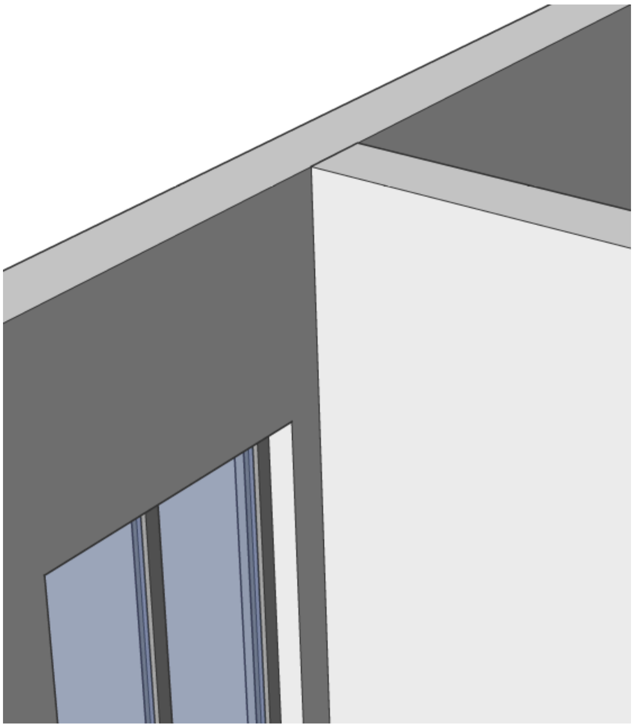
3 Select **Connect to nearest** in the prompt menu.

You can also use the **Quad** menu to create a T-connection between two walls or other elements:

- In the drawing area, highlight the face you want to connect to another wall.
- Use the **Connect with Nearest** tool from the **Quad**.



The **Connect with Nearest** tool automatically makes a T-connection between the two walls by extruding the face of the first wall to the face of the other wall.



33.11.6 Procedure: Creating a curved wall

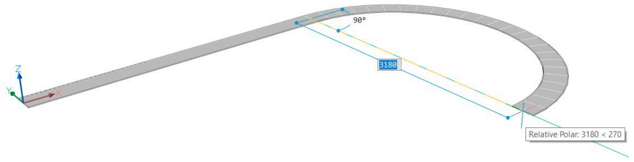
1 Launch the POLYSOLID command.

2 Select a start point and move the cursor to set the length or click to set the endpoint.

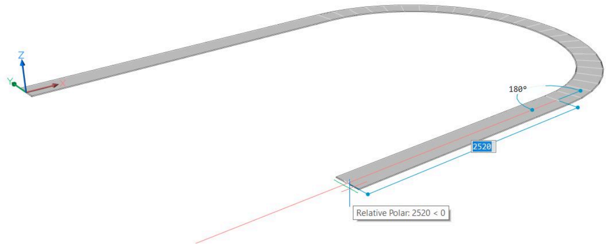
3 To create a curved wall, type A and press Enter to choose **Draw arcs** or select **Draw arcs** in the prompt

menu.

4 Now move the cursor in the desired direction to curve the wall. Enter a value in the dynamic input field to set the degree and length of the arc. Press **Enter**.

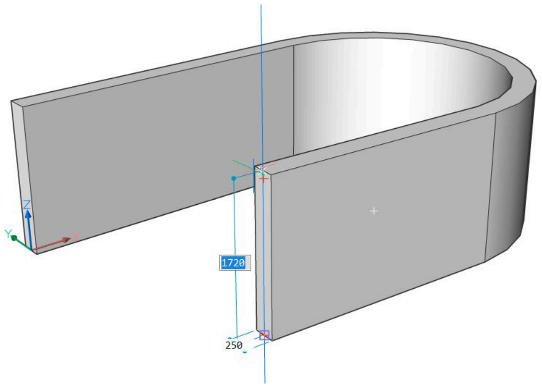


5 You can continue creating curved, adjoining wall segments by clicking or by entering values in the dynamic input fields. To go back to drawing straight wall segments, type L and press Enter or select **Draw lines** in the options dialog.



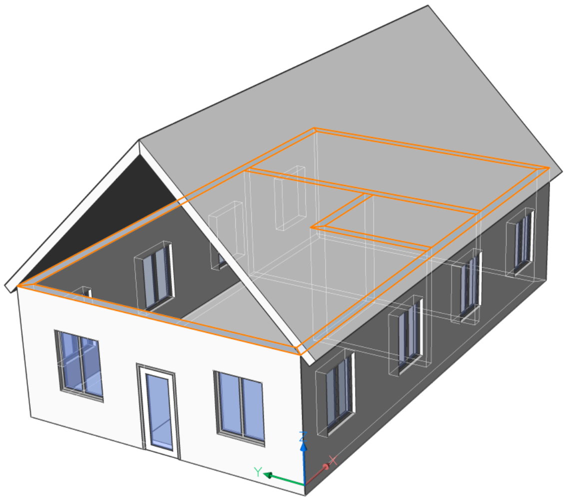
6 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field. Type a value to change the default height of the wall. To change the width of the wall, use the width field. You can switch between the two dynamic input fields by pressing Tab.

7

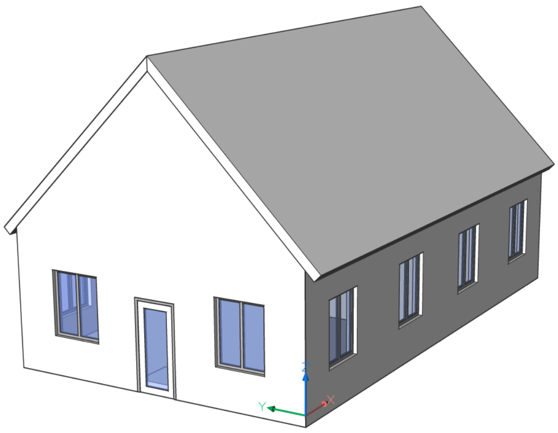


33.11.7 Procedure: Connecting multiple faces of the wall to the roof slab

1 To select all faces aligned to the current selection, highlight the top face of one of the walls and click the **Select aligned faces** tool in the **Select** tab of the Quad.



2 Launch the **Connect with Nearest** tool. The top faces of the wall are attached to the roof.



33.12 Quickdraw in BricsCAD Shape

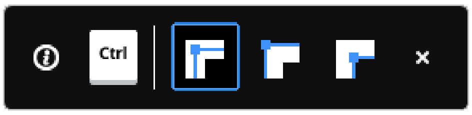
33.12.1 About the Quickdraw tool

The **Quickdraw** tool allows you to get started with conceptual modeling easily. You can use the **Quickdraw** tool to create a room or a whole building. Once in use, **Quickdraw** will remain active until you explicitly exit the command or until another command is executed.

33.12.2 Room and walls settings

The BIMQUICKDRAW command opens the **Bim Quickdraw** command context panel. This allows you to set the room dimensions by directly typing the needed value or by using the field arrows. You can also set the wall placement configuration and snapping options as well as adopt dimensions of nearby rooms by checking the **Auto adopt sizes** option.

To change the justification of the walls, press the **CTRL** key while the **Bim Quickdraw** justification **Hotkey Assistant** widget is displayed at the bottom of the screen.

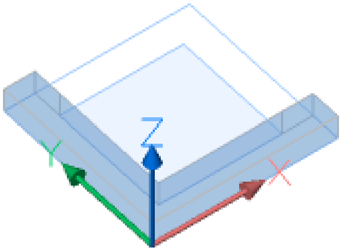


33.12.3 The magnetic snapping feature

With the magnetic snapping feature, you can easily align the **QuickDraw** cursor. When the command is active, move the cursor close to the object you want to align to. The **QuickDraw** cursor automatically snaps to the object. You have the following options:

1 Align to the origin

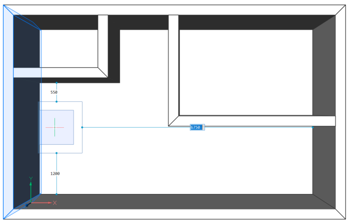
1



2 Align to existing walls

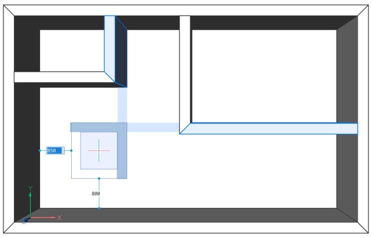
3 - To the wall itself:

-



- The extends of the wall:

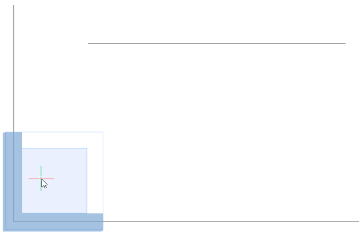
-

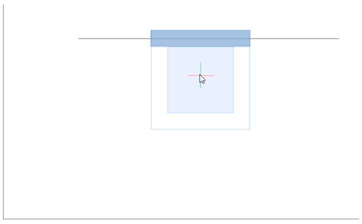


4 Align to lines

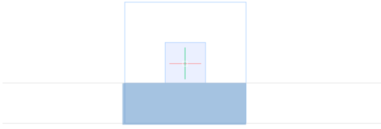
5 - To X- and Y- aligned lines and line segments

-





- To parallel lines
-



33.12.4 Procedure: Creating an L-shaped house

1 When the **Quad** cursor is in the **No Selection** state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the **Quickdraw** icon in the ribbon or type BIMQUICKDRAW in the Command line.

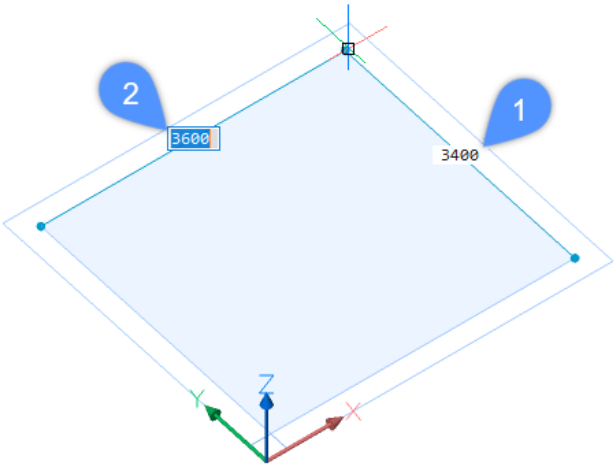
You are prompted: **Select first point:**

Note: Use the **Bim Quickdraw** command context panel to change the wall width, wall height and slab thickness of the room before you click to specify the first corner of a building.

2 Select any point in the drawing area to set the first corner.

3 Drag the mouse cursor to define the footprint of the building.

4



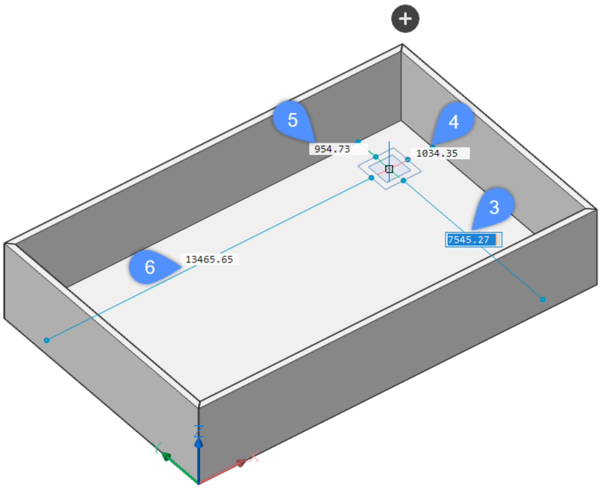
Specify the values of the room width and length in the Dynamic dimension field. Press the **Tab** key to switch between width and length and press **Enter** or just click freely. Press **Enter**or **Escape** to leave the

QUICKDRAW command or you can continue to create more rooms.

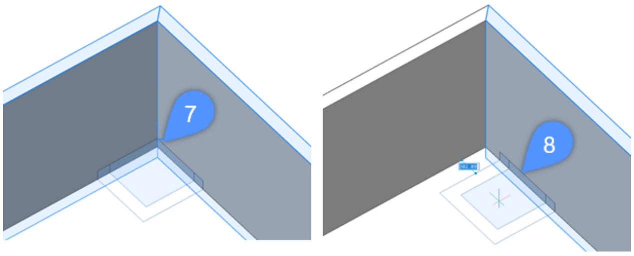
5 Place the **Quickdraw** cursor inside the existing building to create more rooms.

When you place the **Quickdraw** cursor inside the existing building, the dynamic dimension fields display the distances from the **Quickdraw** cursor to the adjacent walls (3,4,5,6).

Note: When the **Auto adopt sizes** option is checked, the dimensions of the walls and slab of the new room will be the same as the walls and slab you snap to. If you don't want this to happen, uncheck this option in the command context panel.



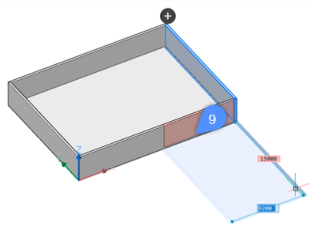
If the cursor aligns with one or two existing walls in the model, the layout of the cursor will adapt and depict the alignment.

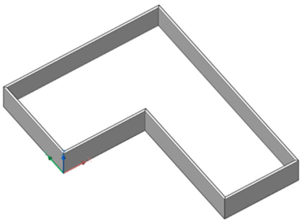


To actually draw new walls, choose the distances from the already existing walls to specify your first point. Move your cursor or type the values in the dynamic dimension fields to do this. Draw walls in the same way as explained in steps 2 and 3.

6 To remove a section of a wall from the existing room, first, align the **Quickdraw** cursor with an existing wall at the inner side of the building. Then click and drag the cursor outside of the existing building.

The outer wall will turn red (9) to indicate the part of the wall that will be removed.





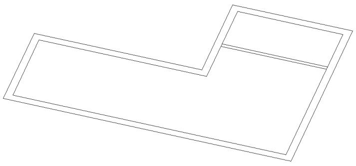
Note: You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

7 Select a second point. The wall highlighted in red will automatically be deleted and an L-shaped room will be created.

33.12.5 Procedure: using Quickdraw starting from a 2D layout

1 Open a 2D floorplan of a rectangular or L-shaped building.

1

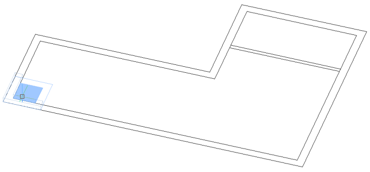


Note: It is also possible to draw a simple 2D floor plan with the POLYLINE command to start from. In this case, the default wall thickness of the BIMQUICKDRAW command will be used.

2 When the **Quad** cursor is in the **No Selection** state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab, click the **Quickdraw** icon in the ribbon or type BIMQUICKDRAW in the Command line.

3 Hover over one of the outer walls with the cursor.

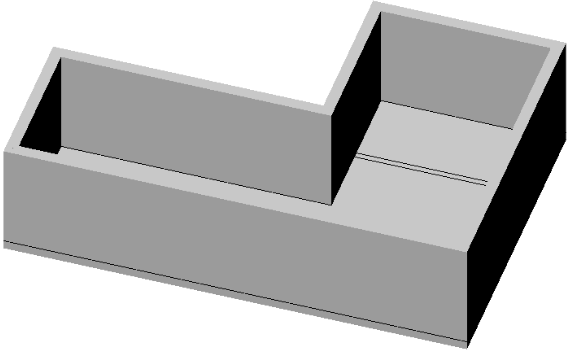
4



Note: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

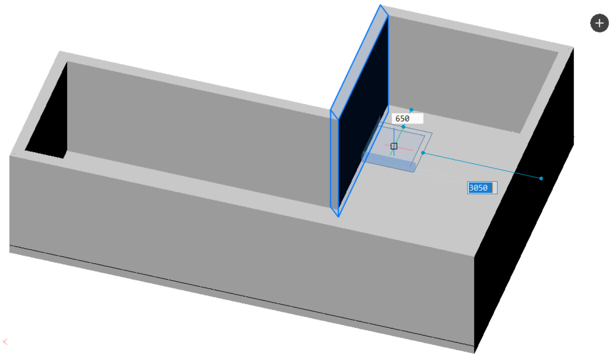
5 Draw the outer walls, using the magnetic snapping feature to snap to the lines of the 2D drawing.

6



7 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.

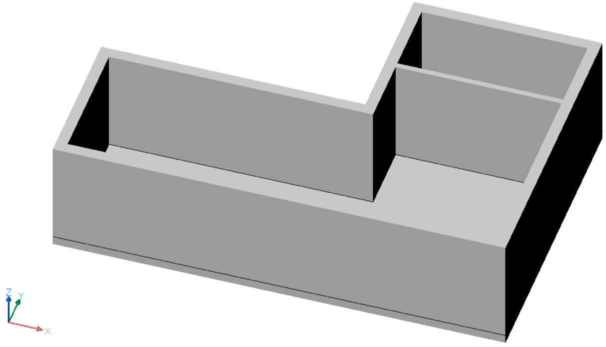
8



Note: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

9 The 2D plan has been transformed into a 3D model.

10



33.13 Creating windows and doors in BricsCAD Shape

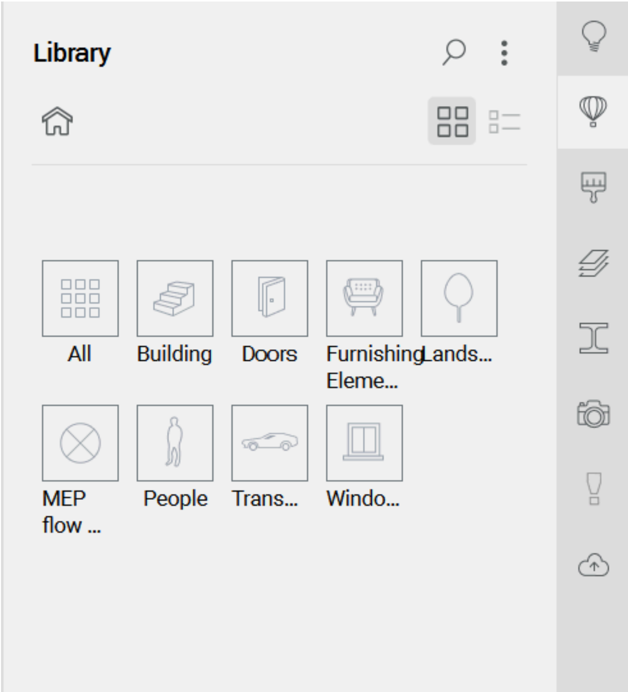
There are two ways to create windows or doors in BricsCAD Shape.

- Pick a door or window from the **Library** panel.
- Create your own custom window.

33.13.1 Procedure: Pick windows or doors from the library

1 Open the **Library** panel by clicking the airballoon icon on the right hand edge of the screen.

2 Click the **Doors** section in the **Library** panel.



3 Select a door from the panel.

4 Move the cursor into the model.

5 Click to place the component.

6 Press the **CTRL** key to toggle between door positioning and door size. Enter the desired values for the height and width of the door by pressing the **Tab** key. When you're done sizing and placing the door, press the **Enter** key.

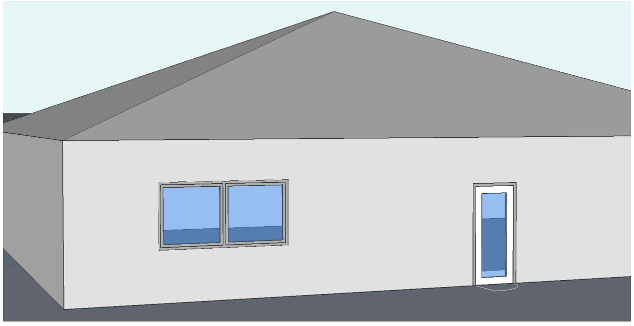
Note: This is the same for a window. Change Doors to Window section. Drag a window into the wall. You can change the size and location of a window just as you did with the door.

33.13.2 Procedure: Create a custom window

1 Use the either the **Polyline** tool, the **Polygon** tool or the **Rectangle** tool to draw the outline of the window on the wall face.

2 Move the cursor inside the window outline and, when the boundary is highlighted, choose **Create window** from the Quad.

3 The available window styles are shown in a dialog box. Move your cursor over the window design of your choice, and left-click it. The window is automatically created.



You can also create a custom window from a wall boundary:

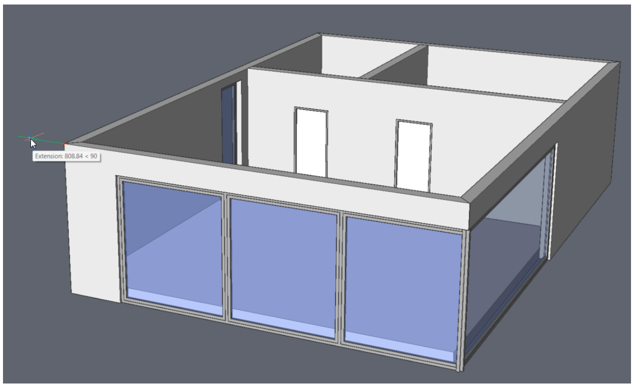
1 Select face of the outer wall and use the **Offset** tool from the Quad.

2 Move the cursor to re-size the new boundary, or key in a value and press Enter. Then, highlight the offset boundary and select **Create window** from the Quad.

33.14 Creating roofs in BricsCAD Shape

Launch the POLYSOLID command through the toolbar or the "No selection" Quad.

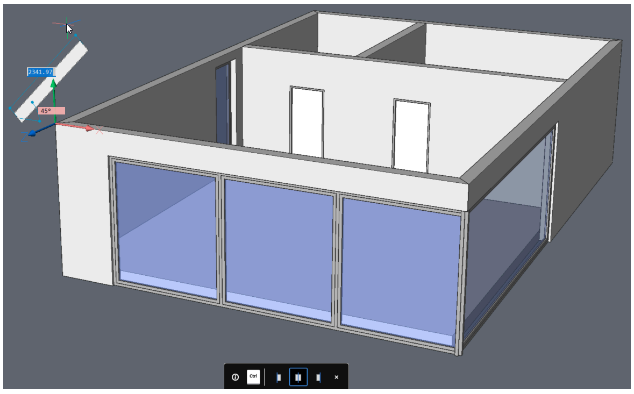
1 To align the drawing plane to the wall surface, press the **SHIFT** button while hovering over the wall surface. The alignment of the slab can be changed by pressing the **CTRL** button.



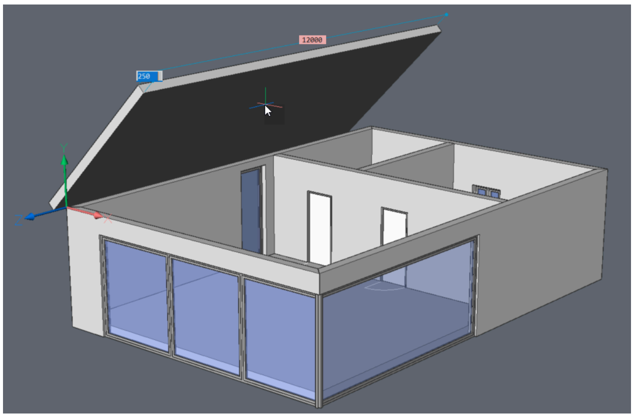
2 Pick an origin point to start drawing.

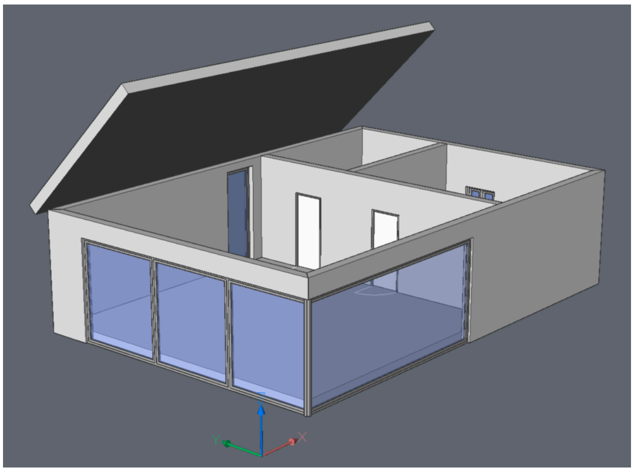
3 To lock the inclination, press TAB until the dynamic dimension of the inclination turns blue. Enter a

value and press Enter.

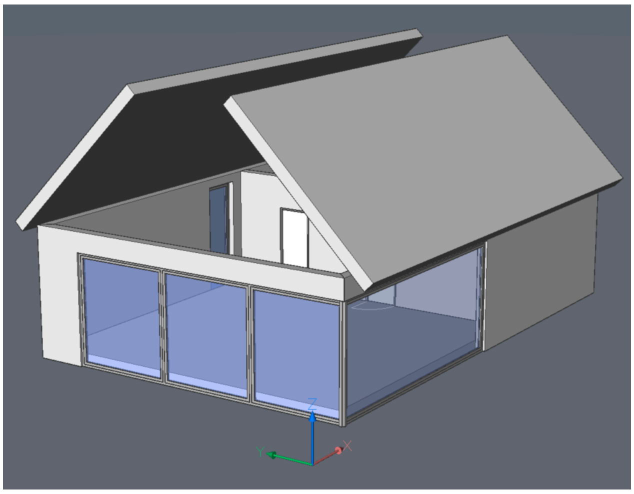
4 

5 The profile thickness can be changes by pressing the TAB key and entering the desired value.

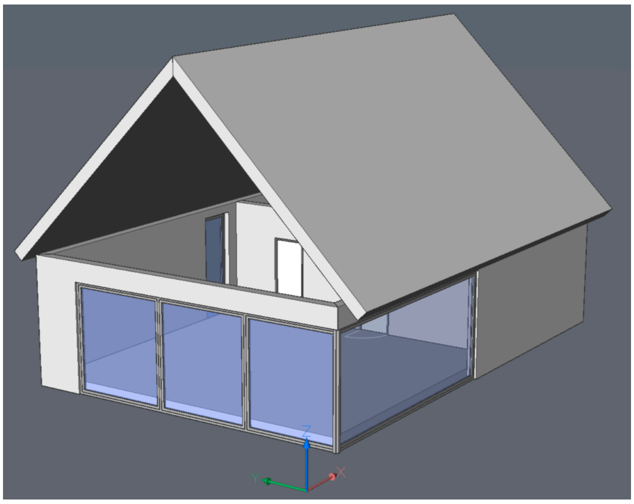
6 



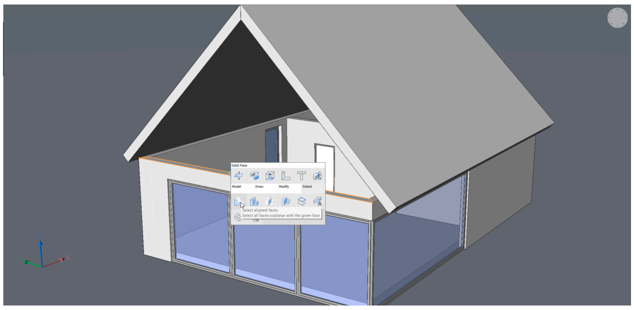
7 Repeat the same process on the other side of the building. Another way to have identical slabs on both sides is by 3D mirroring the first slab.

8 

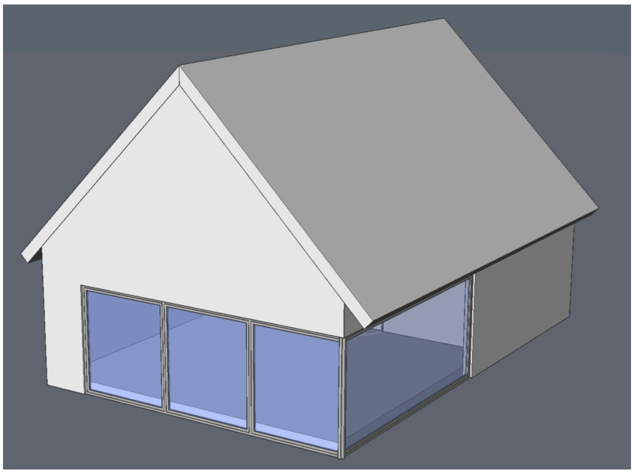
9 Access the **L connect** tool to connect both slabs.



10 To connect the walls to the roof, select one of the top surfaces of a wall. Through the Quad, the **Select aligned faces** option can be accessed in the **Select** tab.



11 Once the top surfaces are selected, the **Connect with nearest** command can be accessed through the Quad in the **Model** tab. Walls are extended up to the roof.



34. Dialog boxes

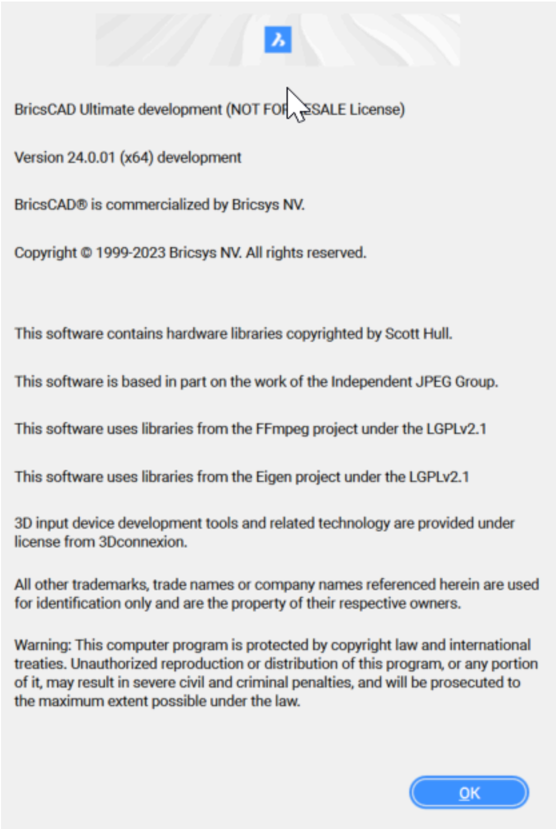
34.1 A

34.1.1 About BricsCAD dialog box

Opens via: ABOUT command

The **About BricsCAD** dialog box displays BricsCAD version, copyright and product information.

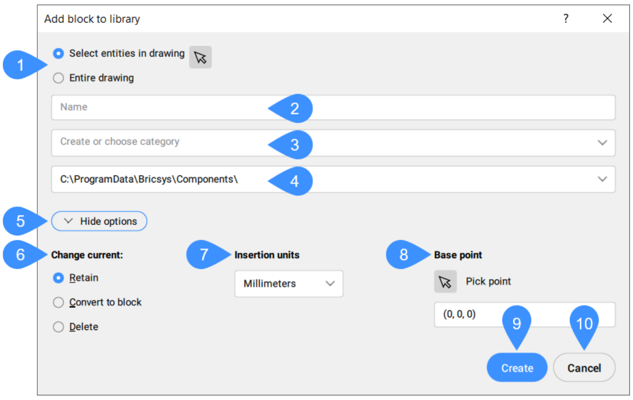
The product information includes the license type, version and revision number.



34.1.2 Add block to library dialog box

Opens via: CREATELIBRARYBLOCK command

The **Add block to library** dialog box creates a block definition and adds it to the block library.



1 Selection set

2 Name

3 Category

4 Location

5 Options

6 Change current

7 Insertion units

8 Base point

9 Create

10 Cancel

34.1.2.1 Selection set

Specifies which entities are included in the block definition. You can select entities in the drawing or use the entire drawing.

34.1.2.2 Name

Specifies the name of the block.

34.1.2.3 Category

Specifies a category for the block. You can choose from a list of existing categories or enter a new category name.

34.1.2.4 Location

Specifies the location where the block DWG file is saved. Available locations are controlled by the **Library directory path** setting (COMPONENTSPATH system variable).

34.1.2.5 Show/Hide Options

Shows or hides additional options. The available options depend on the selection set.

34.1.2.6 Change current

Specifies what happens to selected entities after the block is created. You can choose to retain the individual entities, convert them to a block insertion using the new definition, or delete them. This option is only available if you define the selection set by choosing to select entities in drawing.

34.1.2.7 Insertion units

Specifies the units for the block insertion. You can choose from the following list of standard units:

- Inches
- Feet
- Miles
- Millimeters
- Centimeters
- Meters
- Kilometers
- Microinches
- Mils
- Yards
- Angstroms
- Microns
- Decimeters
- Dekameters
- Hectometers
- Gigameters
- Astronomical Units
- Light-years
- Parsecs
- US Survey Feet
- US Survey Inch
- US Survey Yard
- US Survey Mile
- Base point

Specifies the base point for block definition. You can pick a point in the drawing or enter X,Y,Z coordinates. This option is only available if you define the selection set by choosing to select entities in drawing.

34.1.2.8 Create

Creates a DWG file for the block definition in the specified location, adds the block to the **Library** panel under the specified category and closes the dialog box.

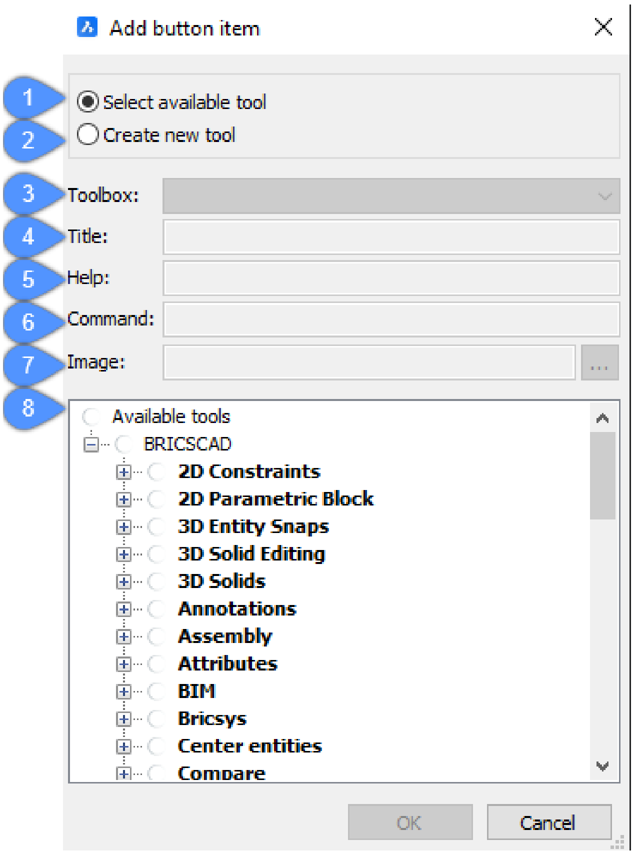
34.1.2.9 Cancel

Closes the dialog box without creating the block definition.

34.1.3 Add button item dialog box

Opens via: **Customize** dialog box

The **Add button item** dialog box allows you to add a button item by selecting an available tool or by creating a new tool.



1 Select available tool

2 Create new tool

3 Toolbox

4 Title

5 Help

6 Command

7 Image

8 Available tools

34.1.3.1 Select available tool

Assigns an existing command to the new button or button group. If this option is selected, the other options are grayed out, except for Available tools.

34.1.3.2 Create new tool

Assigns a new command/macro to the new button or button group. If this option is selected, Available tools is grayed out.

34.1.3.3 Toolbox

Specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list.

34.1.3.4 Title

Specifies the name of the new tool.

34.1.3.5 Help

Specifies the help string displayed on the status bar.

34.1.3.6 Command

Specifies the command(s) or macros.

34.1.3.7 Image

Specifies the image to display for the new tool. The browse icon opens the **Tool Image** dialog box. There are five options to select an image.

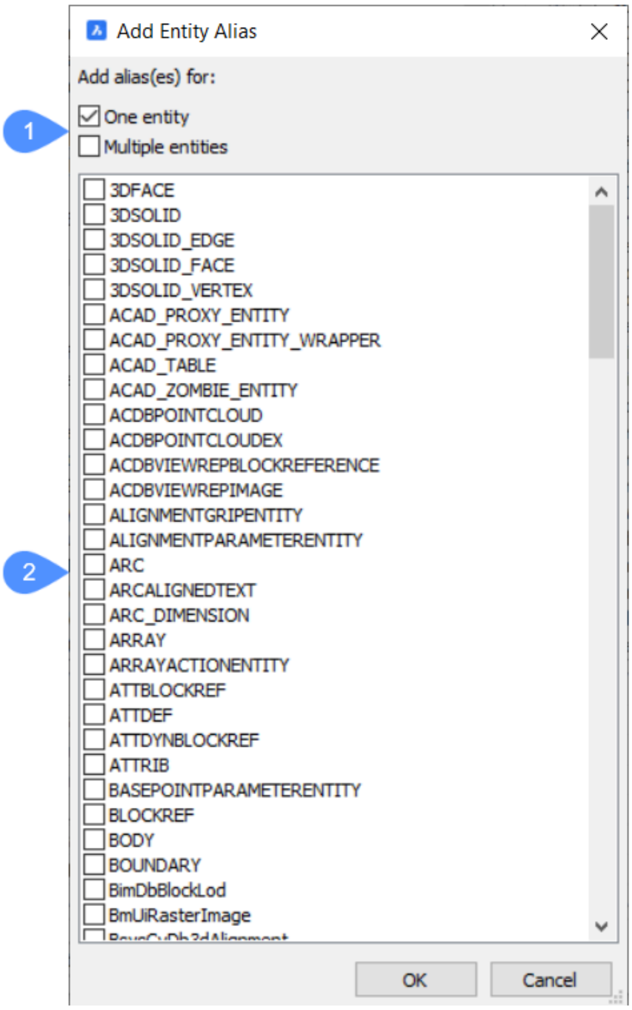
34.1.3.8 Available tools

Selects an existing tool.

34.1.4 Add Entity Alias dialog box

Opens via: **Edit Quad Button Filter** dialog box

The **Add Entity Alias** dialog box allows you to add alias(es) for the selected entity(ies). See the **Customizing the Quad** article).



1 The number of entities in the filter.

2 The list of entity types.

34.1.4.1 The number of entities in the filter

The selected tool will be displayed in the Quad depending on the number of entities that are selected.

The behavior of these options is like this:

Option	Behavior
One entity = Yes Multiple entities = No	The tool is displayed when one single entity of the specified type is selected or hovered. If multiple entities of the specified type are selected, the tool is not displayed.
One entity = Yes Multiple entities = Yes	The tool is displayed when the selection contains either one single entity or multiple entities of the specified type.

Option	Behavior
One entity = No Multiple entities = Yes	The tool is displayed only if many entities of the specified type are selected.

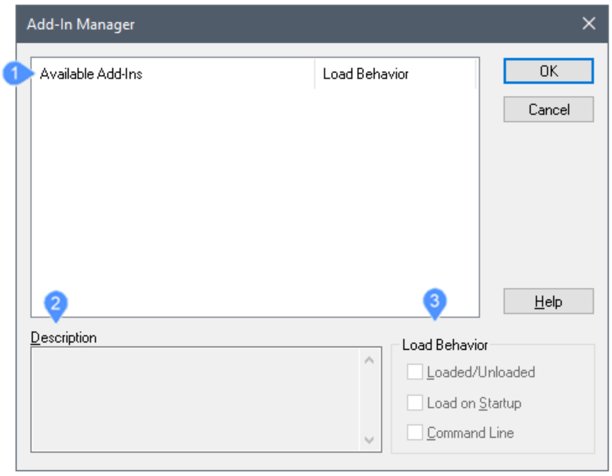
34.1.4.2 The list of entity types

The list with types of entities that can be added to the filter.

34.1.5 Add-In Manager dialog box

Opens via: ADDINMAN command

The **Add-In Manager** dialog box controls how Visual Basic for Applications COM (Common Object Model) modules are loaded. You can either toggle the modules between loaded and unloaded or start modules when BricsCAD starts up.



1 Available Add-ins

2 Description

3 Load Behavior

34.1.5.1 Available Add-ins

Lists the names and status of VBA COM modules available to BricsCAD. "VBA COM" is short for Visual Basic for Applications Common Object Model.

34.1.5.2 Description

Describes the selected VBA COM module, if any.

34.1.5.3 Load Behavior

Determines how modules are loaded.

Loaded/Unloaded
Toggles the module between loaded and unloaded.

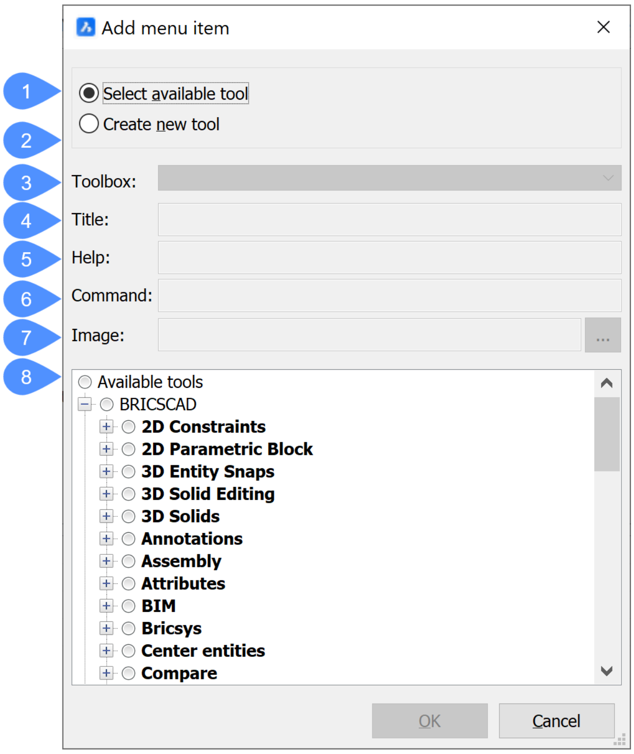
Load on Startup
Loads modules when BricsCAD starts up.

Command line
Loads modules through the Command line.

34.1.6 Add menu item dialog box

Opens via: **Customize** dialog box

The **Add menu item** dialog box allows you to add a menu item by selecting an available tool or by creating a new tool. Opens via the **Customize** dialog box.



1 Select available tool

2 Create new tool

3 Toolbox

4 Title

5 Help

6 Command

7 Image

8 Available tools

34.1.6.1 Select available tool

Assigns an existing command to the new button or button group. If this option is selected, the other options are grayed out, except for Available tools.

34.1.6.2 Create new tool

Assigns a new command/macro to the new button or button group. If this option is selected, Available tools is grayed out.

34.1.6.3 Toolbox

Specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list.

34.1.6.4 Title

Specifies the name of the new tool.

34.1.6.5 Help

Specifies the help string displayed on the status bar.

34.1.6.6 Command

Specifies the command(s) or macros.

34.1.6.7 Image

Specifies the image to display for the new tool. The browse icon opens the **Tool Image** dialog box. There are five options to select an image.

34.1.6.8 Available tools

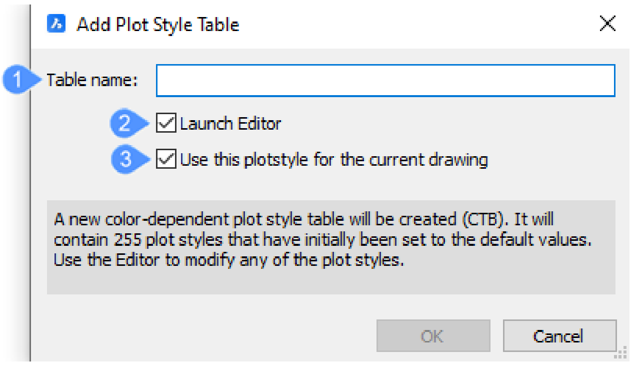
Selects an existing tool.

34.1.7 Add Plot Style Table dialog box

Opens via: **Print** dialog box

The **Add Plot Style Table** dialog box allows you to create new plot styles.

If the drawing uses named plot styles, the wizard creates a new named plot style table (STB file). If the current drawing does not use named plot styles, the wizard creates a new color-dependent table (CTB file).



1 Table name

2 Launch Editor

3 Use this Plotstyle for the current drawing

34.1.7.1 Table name

Enter the name you want to give to the plot style table.

34.1.7.2 Launch Editor

Opens the Plot style table editor dialog box after you enter the name and press the OK button.

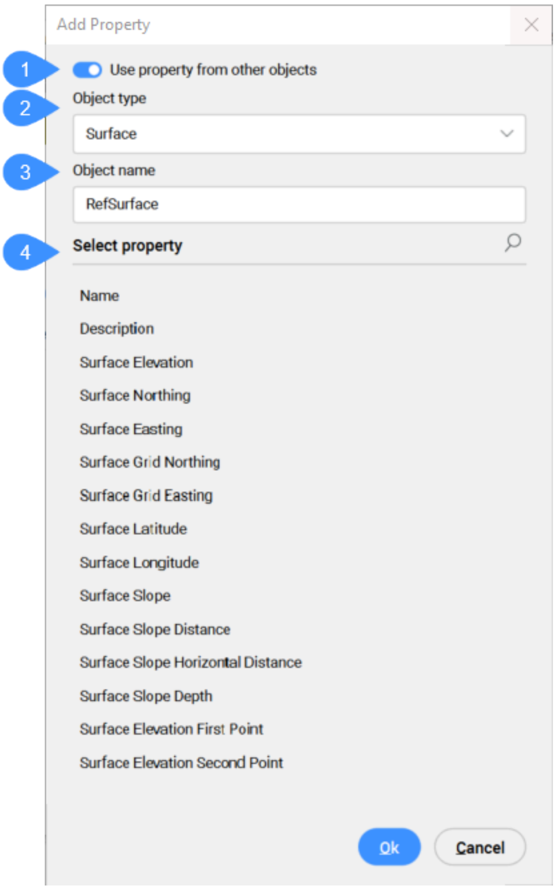
34.1.7.3 Use this Plotstyle for the current drawing

If you do not check the Launch Editor option, it saves and sets the table name you have entered as current plot style table in the current layout.

34.1.8 Add Property dialog box

Opens via: **Text Content Editor** dialog box

The **Add Property** dialog box allows you to add properties to text contents for label styles.



1 Use property from other objects

2 Object type

3 Object name

4 Select property

34.1.8.1 Use property from other objects

Toggle this option if you want to add properties from other Civil objects to this Label Style.

34.1.8.2 Object type

Select the object type from which you want to select the property. Properties can be added from Civil Points, Surfaces, Horizontal Alignments, or Vertical Alignments.

34.1.8.3 Object name

Displays the object name.

34.1.8.4 Select property

Select the property you wish to include in the Label Style. It is possible to include the following attributes:

From Civil Point:

- Name
- Description
- Point Easting
- Point Northing
- Point Elevation
- Point Full Description
- Point Number
- *User-defined attribute for Civil Point*

From Surface:

- Name
- Description
- Surface Elevation
- Surface Northing
- Surface Easting
- Surface Grid Northing
- Surface Grid Easting
- Surface Latitude
- Surface Longitude
- Surface Slope
- Surface Slope Distance
- Surface Slope Horizontal Distance
- Surface Slope Depth
- Surface Elevation First Point
- Surface Elevation Seconds Point

From Horizontal Alignment:

- Name
- Description
- Alignment Station Value
- Alignment Offset Value

From Vertical Alignment:

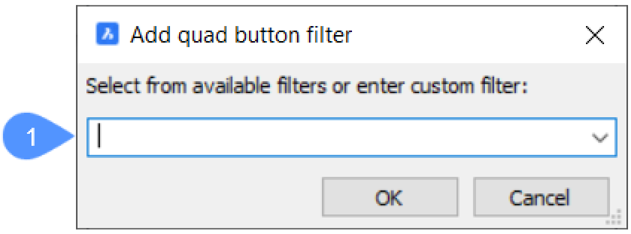
- Name
- Description

Note: When adding properties from Surface, Vertical or Horizontal Alignment to be shown in the Label Style, make sure to select the appropriate references in the properties panel of Civil Points.

34.1.9 Add quad button filter dialog box

Opens via: **Edit Quad Button Filter** dialog box

The **Add quad button filter** dialog box allows you to select a filter for the quad button (see the **Customizing the Quad** article).



1 Selection drop-down list with filters.

34.1.9.1 Selection drop down list with filters

The list contains three general filters that apply to any object type.

NO_SELECTION	The tool is displayed when nothing is selected.
OBJECT_ANY	The tool is displayed when one entity (regardless its type) is selected.
OBJECTS_ANY	The tool is displayed when many entities (regardless their type) are selected.

Note: It is possible to type in a custom filter, however it is recommended to use the **Add Entity Alias** dialog box to create a custom filter.

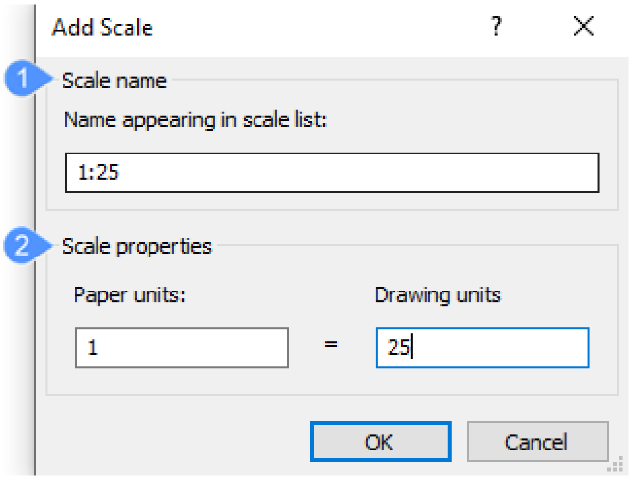
Note: The custom filter format consists of one of the three general filters with object type suffix like _LINE, _CIRCLE, etc. For example: OBJECT_CIRCLE, OBJECTS_SURFACE_EDGE, etc.

34.1.10 Add Scale dialog box

Opens via:

- **Default Scale List** dialog box
- **Edit Scale List** dialog box

The **Add Scale** dialog box allows you to add a new scale factor to the list.



1 Scale name

2 Scale properties

34.1.10.1 Scale name

Specifies the scale name as it will appear in the scale list.

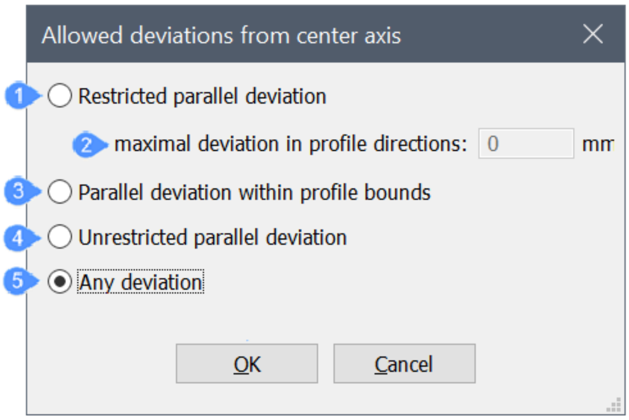
34.1.10.2 Scale properties

Specifies the scale factor in paper (size when plotted) and drawing units.

34.1.11 Allowed deviations from center axis dialog box

Opens via: BIMANALYTICALMODEL command

The **Allowed deviation from center axis** dialog box allows you to set the level of freedom used when generating an analytical model.



1 Restricted parallel deviation

2 Maximal deviation in profile directions

3 Parallel deviation within profile bounds

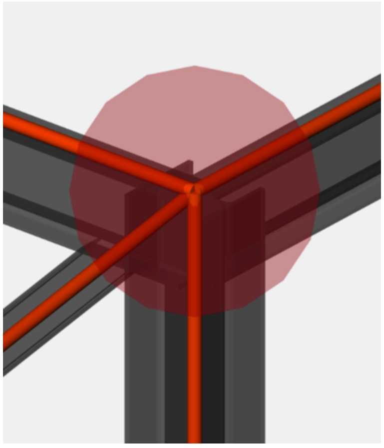
4 Unrestricted parallel deviation

5 Any deviation

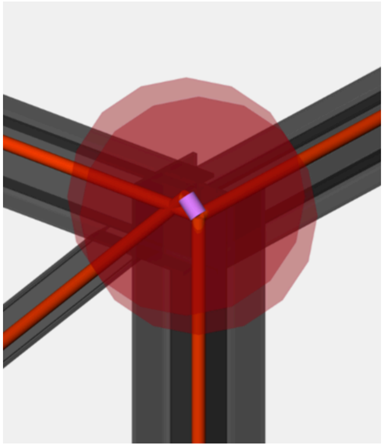
34.1.11.1 Restricted parallel deviation

Axes remain parallel to the center axis and within the specified distance from the center axis in both profile directions. You can specify the distance in the window.

Maximum deviation in profile directions: 0mm



Maximum deviation in profile directions: 20mm



34.1.11.2 Parallel deviation within profile bounds

Parallel deviation within profile bounds axes remain parallel to the center axis and do not exceed profile bounds.

34.1.11.3 Unrestricted parallel deviation

Axes remain parallel to the center axis, but can be placed outside the profile bounds if this is useful to avoid node splits or rigid links.

34.1.11.4 Any deviation

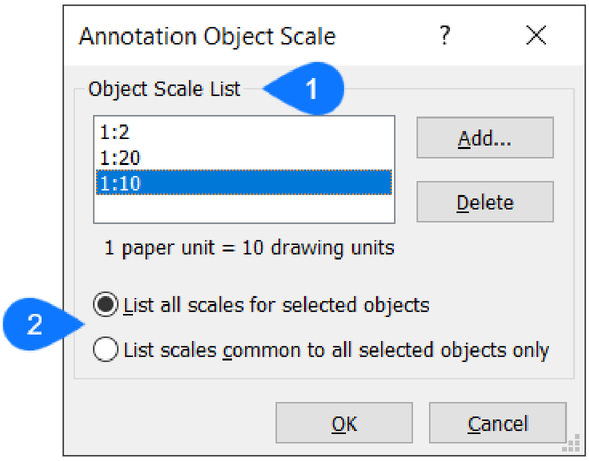
The main axes remain parallel to the center axis, but tilted axes (with respect to vertical and main horizontal directions) can be allowed to deviate from parallel if it useful to avoid node splits and rigid links. Other axes can move outside of profile bounds.

Click **OK** to open the **Visual settings** dialog box.

34.1.12 Annotation Object Scale dialog box

Opens via: OBJECTSCALE dialog box

The **Annotation Object Scale** dialog box allows you to add or remove scale factors used with annotative entities.



1 Object Scale List

2 Options

34.1.12.1 Object Scale List

Shows all the scales supported by the selected annotative objects.

Add
Adds annotation scale factors to the selected entities.

Delete
Removes annotation scales from the selected entities. Choose **No** in the **Annotative** field on the Properties Bar to remove the annotative property.

Note: This option does not work if there is just one scale factor attached to the entities.

34.1.12.2 Options

List all scales for selected objects
Lists all the scale factors.

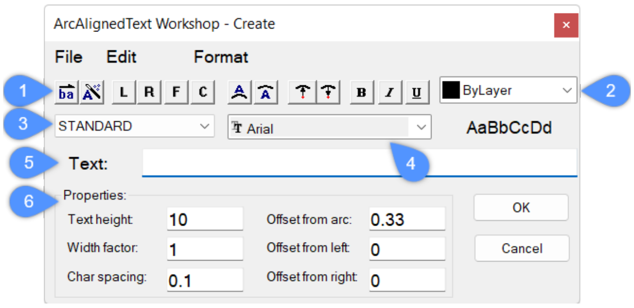
List scales common to all selected objects only
Lists only the scale factor shared by the selected entities.

Note: At least two entities must be selected.

34.1.13 ArcAlignedText Workshop dialog box

Opens via: ARCTEXT command (Express Tools)

The **ArcAlignedText Workshop - Create** dialog box allows you to design arc aligned text entities.



1 Text options

2 Text color

3 Text style

4 Text font

5 Text content

6 Properties

34.1.13.1 Text options

- : reverses text reading order.
- : controls the text behavior when the arc is moved.
- : aligns the text to the left.
- : align the text to the right.
- : fits the text along the arc.
- : centers the text along the arc.
- : places the text on the convex side.
- : places the text on the concave side.
- : places the text outwards from center.
- : places the text inwards to center.
- : makes the text bold.
- : makes the text italic.
- : makes the text underlined.

34.1.13.2 Text color

Sets the color of the text.

34.1.13.3 Text style

Specifies the text style.

34.1.13.4 Text font

Specifies the text font.

34.1.13.5 Text content

Allows you to type the text to be displayed.

34.1.13.6 Properties

Text height
Specifies the text height.

Width factor
Specifies the width of the text in proportion to the length of the arc.

Char spacing
Specifies the space between characters.

Offset from arc
Specifies the offset distance from the arc to the text.

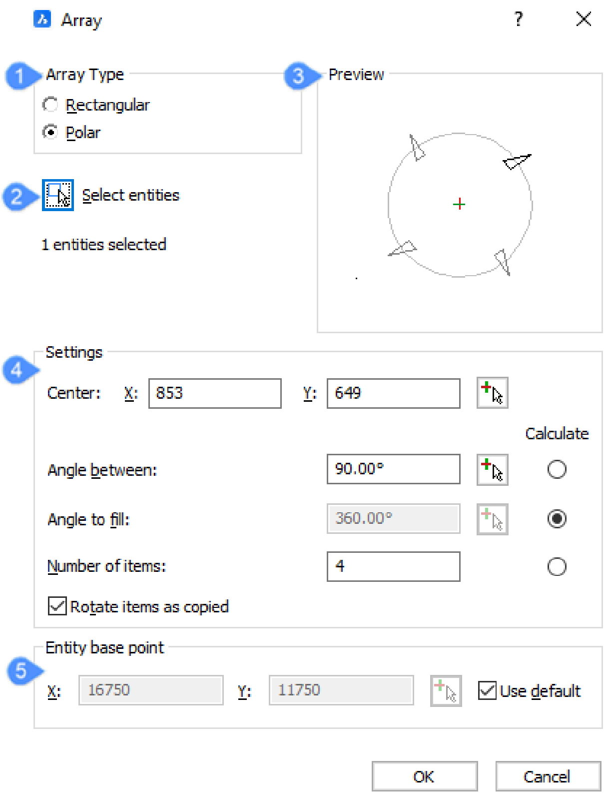
Offset from left
Specifies the offset distance from the left end of the arc.

Offset from right
Specifies the offset distance from the right end of the arc.

34.1.14 Array dialog box

Opens via: ARRAYCLASSIC command

The **Array** dialog box allows you to create a non-associative 2D rectangular or polar array of entities.



1 Array Type

2 Select entities

3 Preview

4 Settings

5 Entity base point (Polar only)

34.1.14.1 Array Type

Specifies whether a rectangular or polar array will be created.

34.1.14.2 Select entities

Temporarily closes the **Array** dialog box and prompts you to select entities. Press Enter to complete your select and return to the **Array** dialog box.

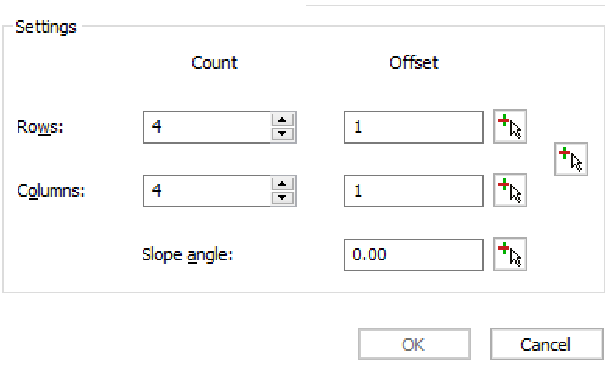
34.1.14.3 Preview

Displays a preview of the array based on the specified settings.

34.1.14.4 Settings

Specifies the number and location of items in the array. The available settings differ based on the array type.

Rectangular
Copies the selected entities in a rectangular pattern.



Count

Specifies the number of rows and columns in the array.

Offset

Specifies the distance between rows and columns. You can type a distance in the Offset fields or click the buttons to pick the offset distance in the drawing.

Offset field

Enables you to type distances for the offsets. Positive distances add rows along the positive Y axis and add columns along the positive X axis. Negative distances add rows along the negative Y axis and add columns along the negative X axis.

Offset buttons

Enables you to specify the offset distance by picking two points in the drawing. Clicking one of these buttons temporarily closes the **Array** dialog box and prompts you to pick points.

Pick Row Offset

Specifies the distance between rows. A temporary line displays between the two points you pick. The length of the line determines the offset distance. The angle of the line determines if rows are added along the positive or negative Y axis.

Pick Column Offset

Specifies the distance between columns. A temporary line displays between the two points you pick. The length of the line determines the offset distance. The angle of the line determines if columns are added along the positive or negative X axis.

Pick Both Offsets

Specifies the distance between rows and the distance between columns. A temporary rectangle displays between the two points you pick. The length of the rectangle determines the distance between columns and the width of rectangle determines the distance between rows.

Slope Angle

Specifies the angle of the array from the X axis. You can type an angle in the Angle field or click the button to pick an angle in the drawing.

Angle field

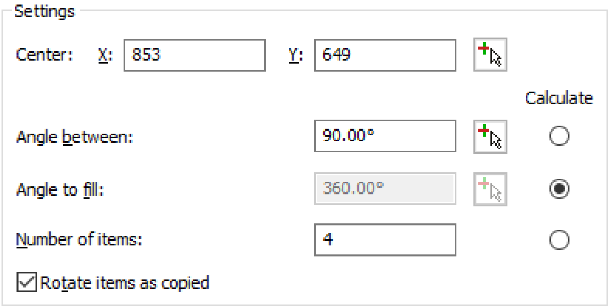
Enables you to type an angle for the slope.

Pick Angle of Array button

Enables you to specify the angle of the array by picking two points in the drawing. Clicking the button temporarily closes the **Array** dialog box and prompts you to pick points.

Polar

Copies the selected entities in a circular pattern.



Center

Specifies the center point of the polar array. You can type coordinates in the X and Y fields or click the button to pick the center in the drawing.

X field

Enables you to type a coordinate to place the center of the array along the X axis.

Y field

Enables you to type a coordinate to place the center of the array along the Y axis.

Pick Center Point button

Enables you to specify the center of the array by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point.

Angle between

Specifies the angle between each item in the polar array. You can type an angle in the field, click the button to pick an angle in the drawing, or choose the Calculate option.

Angle field

Enables you to type a value to specify the angle between each item of the array.

Pick Angle Between Items button

Enables you to specify the angle between each item by picking a point in the drawing. Clicking the button temporarily closes the **Array** dialog box and prompts you to pick a point. A temporary line displays between the center point of the array and the point you pick. The angle of the line determines the angle between items.

Calculate

Enables you to specify the angle between each item based on the Angle to fill and number of items. Choosing the Calculate radio button disables the Angle between options and enables the Angle to fill options.

Angle to fill

Specifies the fill angle and direction for a polar array. You can type an angle in the field, click the button to pick an angle in the drawing, or choose the Calculate option.

Angle field

Enables you to type a value to specify the fill angle of the array. Enter 360 for a full circle of copies. Enter a smaller number to draw a partial polar array. Positive values draw the array counterclockwise and negative values draw the array clockwise.

Pick Angle to Fill button

Enables you to specify the fill angle of the array by picking a point in the drawing. Clicking the button temporarily closes the **Array** dialog box and prompts you to pick a point. A temporary line displays between the center point of the array and the point you pick. The angle of the line determines the fill angle.

Calculate

Enables you to specify the fill angle based on the angle between items and the number of items. Choosing the Calculate radio button disables the Angle to fill options and enables the Angle between options.

Number of items

Specifies the number of items in the polar array. You can type a number in the field or choose the Calculate option.

Number field

Enables you to enter the number of items to include in the array.

Calculate

Enables you to specify the number of items based on the angle between items and the fill angle. Choosing the Calculate radio button disables the Number of items options and enables the Angle Between and Angle to Fill options.

Rotate items as copied

Specifies whether array items are rotated when they are copied to form the polar array.

- No: does not rotate items; all copies have the same orientation as the original item.
- Yes: rotates copies.

34.1.14.5 Entity base point (Polar only)

Specifies the base point of items in a polar array.

X field

Enables you to type a coordinate to specify the basepoint of array items along the X axis.

Y field

Enables you to type a coordinate to specify the basepoint of array items along the Y axis.

Pick base point button

Enables you to specify the base point of array items by picking a point in the drawing. Clicking the button temporarily closes the **Array** dialog box and prompts you to pick a point.

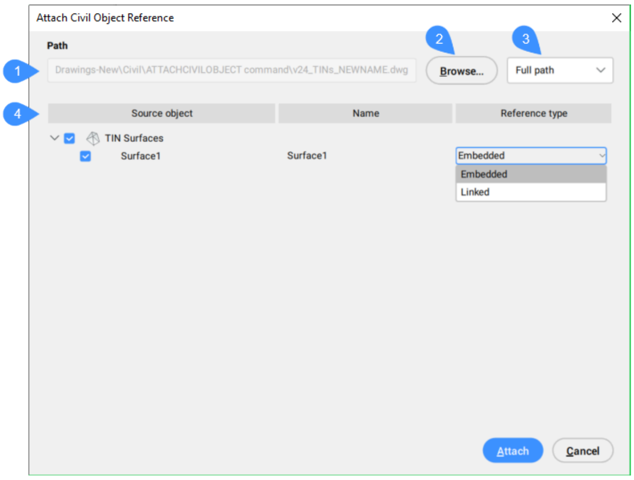
Use Default

Uses the default basepoint for array entities. Choosing the Use default option disables the other entity base point options.

34.1.15 Attach Civil Object Reference dialog box

Opens via: ATTACHCIVILOBJECT command (after selecting a Reference civil object file)

The **Attach Civil Object Reference** dialog box allows you to attach civil object references to the current drawing.



1 Path

2 Browse

3 Path type

4 Attached civil object

34.1.15.1 Path

Displays the path of the drawing file.

34.1.15.2 Browse

Opens the **Open file** dialog box that allows you to choose a different civil object DWG file.

34.1.15.3 Path type

Determines how much of the path is stored.

Full path
Stores the drive and folder names of the DWG file as an absolute reference, such as *C:\cad\dwg \filename.dwg*.

Relative path
Stores the path from the parent drawing's location to the referenced drawing's location, such as *.. \filename.dwg*. The **..** refers to the folder above the current one. The drawing must be saved before you can use this option.

34.1.15.4 Attached civil object

Lists the properties of the civil object.

Note: These properties are available for editing in the **Info** tab of the **Civil Explorer** panel.

Source object
Displays the source civil object name.

Note: A warning icon displays if the civil object is already referenced in the drawing.

Name
Displays the name of the attached civil object.

Reference type
Defines how the civil object will be attached in the current drawing.

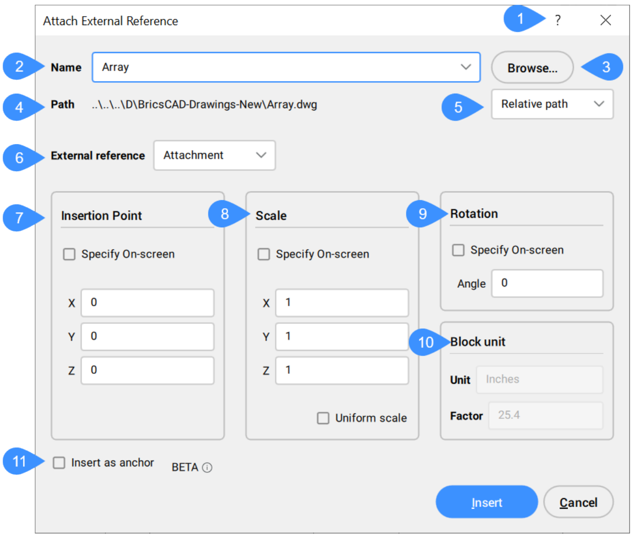
Embedded
The geometry of the reference surface is saved in the drawing. The drawing will be larger but open faster unless the source surface has been changed. Reference surfaces will be visible even if the source drawing is not available.

Linked
The geometry of the reference surface is not saved in the drawing.

34.1.16 Attach External Reference dialog box

Opens via: XATTACH command (after selecting a Reference file)

The **Attach External Reference** dialog box allows you to attach externally referenced drawings to the current drawing.



1 Help
2 Name
3 Browse
4 Path
5 Path type
6 External reference
7 Insertion Point
8 Scale
9 Rotation
10 Block Unit
11 Insert as anchor

34.1.16.1 Help

Opens the Bricsys Help article about the XATTACH command.

34.1.16.2 Name

Specifies the name of the DWG file attached to the current drawing.

34.1.16.3 Browse

Opens the **Select Reference File** dialog box to allow you to choose a different DWG file.

34.1.16.4 Path

Displays the path of the drawing file.

34.1.16.5 Path type

Determines how much of the path is stored:

Full path
Stores the drive and folder names of the DWG file as an absolute reference, such as *c:\cad\dwg \filename.dwg*

Relative path
Stores the path from the parent drawing's location to the referenced drawing's location, such as *.. \filename.dwg*. The .. refers to the folder above the current one. The drawing must be saved before you can use this option.

No path
Strips the drive and folder names, leaving just the DWG file name, such as **filename.dwg**.

34.1.16.6 External reference

Specifies how the external reference is to be attached.

Attachment
Attaches the Xref and nested Xrefs.

Overlay
Attaches only the first level of Xref.

34.1.16.7 Insertion Point

Specifies the location of the Xrefs lower-left corner:

Specify On-screen
Determines how to specify the insertion point.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box using the X, Y, and Z fields.

X, Y, or Z
Specifies the X, Y, and Z coordinates for the Xrefs insertion point. Use 0,0,0 to insert the Xref at the drawing's origin.

34.1.16.8 Scale

Specifies the size of the Xref.

Specify On-screen
Determines how to specify the scale factors.

- On: specify the scale factors in the drawing after the dialog box is closed.
- Off: specify the scale factors in the dialog box using the X, Y, and Z fields.

X, Y, or Z
Specifies the X, Y, and Z scale factors of the Xref in the X, Y, and Z directions. Use 1,1,1 to keep the Xref at its original size.

Uniform Scale
Makes the Y and Z scale factors equal to X.

34.1.16.9 Rotation

Specifies the rotation angle of the Xref.

Specify On-screen
Determines how to specify the rotation angle.

- On: specify the angle in the drawing after the dialog box is closed.
- Off: specify the angle in the dialog box using the Angle field.

Angle
Specifies the rotation angle about the insertion point. Positive angles rotate the Xref counterclockwise. Negative angles rotate the Xref clockwise. Use 0 to keep the xref at its original orientation.

34.1.16.1 Block Unit
0

Specifies the units with which the drawing will be inserted, usually a form of Imperial or metric units, such as inches or mm:

Unit
Reports the INSUNITS setting of the drawing being attached.

Factor
Reports the calculated scaling factor with respect to the INSUNITS setting of the drawing being attached and the current drawing.

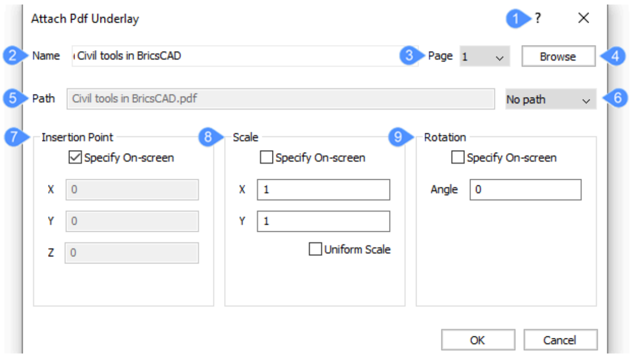
34.1.16.1 Insert as anchor
1

Anchors an external reference to a face of a solid. This anchor links the external reference to the face. The external reference will stay on the face when the host solid is moved or edited.

34.1.17 Attach PDF underlay dialog box

Opens via: PDFATTACH command

The **Attach PDF Underlay** dialog box allows you to attach a PDF file as an underlay into the current drawing.



1 Help

2 Name

3 Page

4 Browse

5 Path

6 Path type

7 Insertion Point

8 Scale

9 Rotation

34.1.17.1 Help

Opens the Bricsys Help article about the -PDFATTACH command.

34.1.17.2 Name

Displays the file name of the PDF file to attach. When more than one PDF file has been attached to the drawing, their names are listed in the drop-down list.

34.1.17.3 Page

Allows you to display the page you want from the whole PDF file. You can change the number of the page from the drop-down arrow list.

34.1.17.4 Browse

Opens the **Select PDF underlay file** dialog box from where you can select another PDF file than the one already selected.

34.1.17.5 Path

Displays the path of the file according to the path type you choose.

34.1.17.6 Path type

Choose the way the program remembers the path to the PDF file. You can select from the Path drop-down list one of the following options:

Full path
Stores the drive and the folder(s) name to the PDF file as an absolute reference such as: C:\Users\user \PDFATTACH\filename.pdf

Relative path
Stores the part of the path to PDF file's folder, such as .\filename.pdf. This option does not work until the drawing has been saved at least once. (..\ means one folder higher)

No path
Strips the drive and folder names, leaving just the PDF file name, such as filename.pdf.

34.1.17.7 Insertion Point

Specifies the location of the underlay's lower-left corner.

Specify On-screen
Determines how the insertion point is specified.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box, using the X, Y and Z fields.

X, Y, Z
Specifies the x, y, z coordinates for the PDF underlay's insertion point. Use 0,0,0 to insert the underlay at the drawing's origin.

34.1.17.8 Scale

Specifies the size of the PDF underlay:

Specify On-screen
Determines how the scale factors are specified:

- On: specify the scale factor in the drawing after the dialog box is closed.
- Off: specify the scale factor in the dialog box using the X and Y fields.

X,Y
Specifies the scale factors along the X and Y axes.

Uniform scale
Makes the Y scale factor equal to X.

34.1.17.9 Rotation

Specifies the rotation angle of the PDF underlay:

Specify On-screen
Determines how the rotation angle is specified:

- On: specify the rotation angle in the drawing after the dialog box is closed.
- Off: specify the rotation angle in the dialog in Angle field.

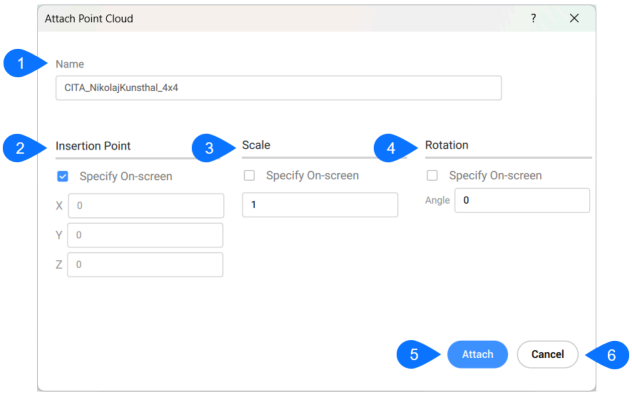
Angle
Specifies the rotation angle about the insertion point. Positive angles rotate the underlay counterclockwise. Negative angles rotate the image clockwise. Use 0 to keep the underlay at its original orientation.

34.1.18 Attach Point Cloud dialog box

Opens via:

- POINTCLOUDATTACH command
- **Point Cloud Reference Manager** dialog box

The **Attach Point Cloud** dialog box allows you to set the options for the attachment of a point cloud file.



1 Name

2 Insertion point

3 Scale

4 Rotation

5 Attach

6 Cancel

Name
Represents the name of the selected point cloud file that will be attached

Insertion point
Defines the point where the origin of the point cloud file will be attached to.
If the **Specify On-screen** option is ticked, the insertion point will be specified in the model space.

Scale
Defines whether the original point cloud file is scaled or not.
If the **Specify On-screen** option is ticked, the scale will be specified in the model space.

Rotation
Defines the rotation of the original point cloud file.
If the **Specify On-screen** option is ticked, the rotation will be specified in the model space.

Attach
Attaches the point cloud file according to the defined properties.

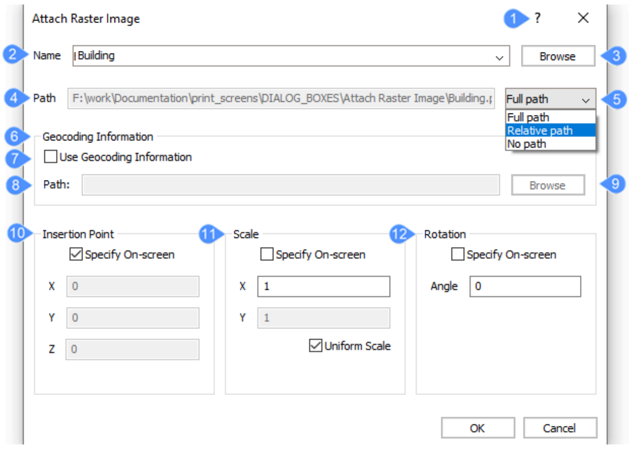
Cancel
Cancels the attachment operation.

Note: The attached point cloud has the **Locked** property in the **Properties** panel. When it is ON, the point cloud can still be selected but not deleted, moved, or edited.

34.1.19 Attach Raster Image dialog box

Opens via: IMAGEATTACH command

The **Attach Raster Image** dialog box allows you to attach raster images to the drawing.



1 Help

2 Name

3 Browse

4 Path

5 Path type

6 Geocoding Information

7 Use Geocoding Information

8 Path

9 Browse (image position file)

10 Insertion Point

11 Scale

12 Rotation

34.1.19.1 Help

Opens the Bricsys Help article about the IMAGEATTACH command.

34.1.19.2 Name

Specifies the name of the file to attach.

34.1.19.3 Browse

Opens the **Select Image File** dialog box to allow you to choose a different raster file.

34.1.19.4 Path

Displays the path of the image file.

34.1.19.5 Path type

Specifies how much of the path is stored.

Full path
Stores the full path to the image file as an absolute reference such as: *C:\foldername\imagename.jpg*

Relative path
Stores the path from the drawing's location to the image's location. For example: *..\foldername\imagename.jpg*. The .. refers to the folder above the current one. The drawing must be saved before you can use this option.

No path
Strips the drive and folder names, leaving just the image file name such as *imagename.jpg*.

34.1.19.6 Geocoding Information

Position files or world files have an extension that depends on what kind of image file they are associated with.

Use Geocoding Information
Toggles use of a position file, which specifies the size, location and rotation of the image file.

- Yes: uses a position file
- No: does not use a position file.

Path
Displays the path of the position file.

Browse (image position file)
Opens the **Select Image Position File** dialog box.

34.1.19.7 Insertion Point

Specifies the location of the image's lower left corner.

Specify On-screen
Determines how you specify the insertion point.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box using the X, Y and Z fields.

X,Y,Z
Specifies the x, y, and z coordinates for the image's insertion point.

34.1.19.8 Scale

Specifies the size of the image.

Specify On-screen
Determines how you specify the scale factor.

- On: specify the scale factors in the drawing after the dialog box is closed.
- Off: specify the scale factors in the dialog box using the X and Y fields.

XY
Specifies the scale factors along the X and Y axes.

Uniform scale
Makes the Y scale factor equal to X.

34.1.19.9 Rotation

Specifies the rotation angle of the image.

Specify On-screen
Determines how you specify the rotation angle.

- On: specify the rotation angle in the drawing after the dialog box is closed.
- Off: specify the angle in the dialog box using the Angle field.

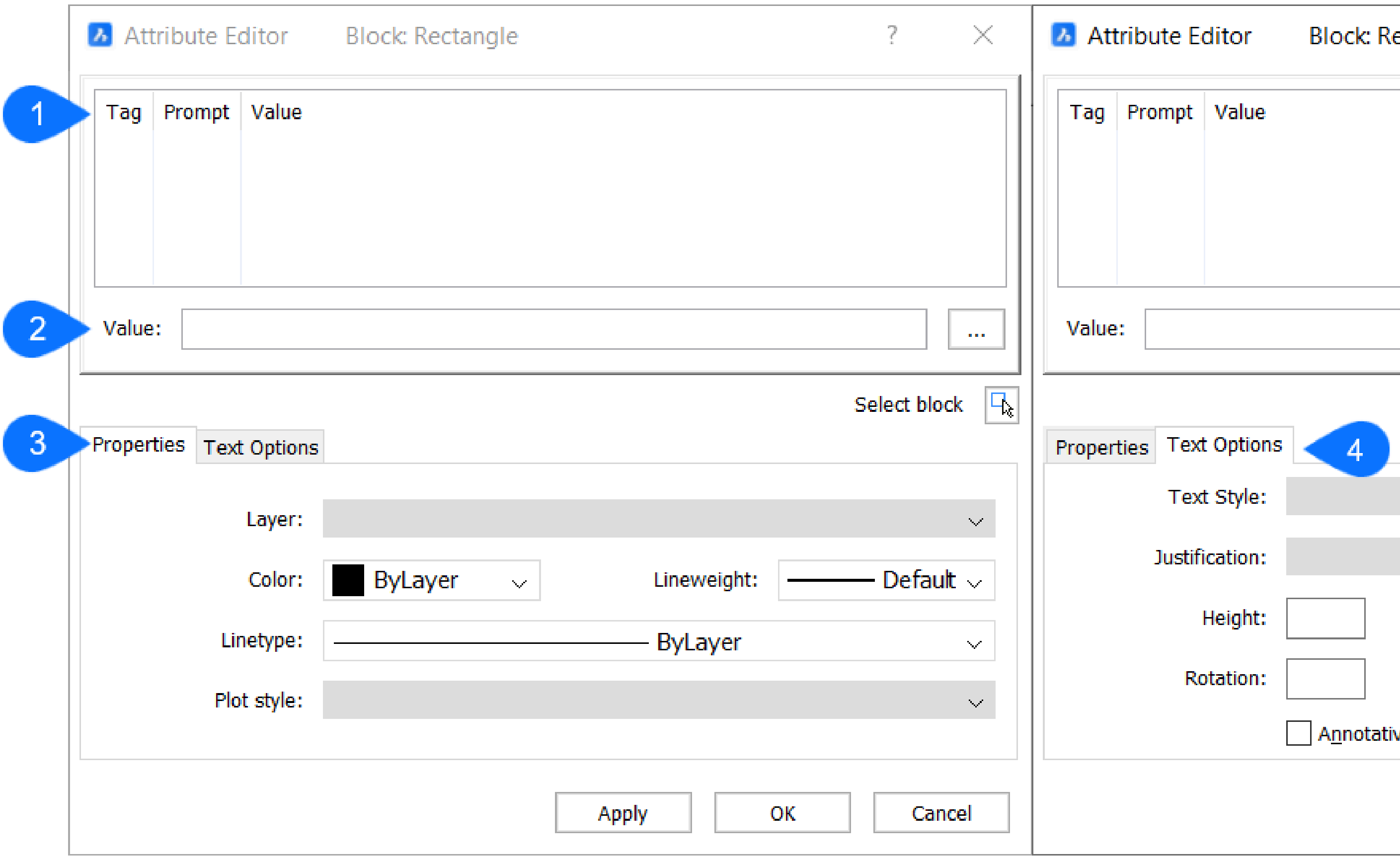
34.1.19.1 Angle
0

Specifies the rotation angle of the image about the insertion point. Positive angles rotate the image counterclockwise. Negative angles rotate the image clockwise. Use 0 to keep the image at its original orientation.

34.1.20 Attribute Editor dialog box

Opens via: EATTEDIT command

The **Attribute Editor** dialog box allows you to edit the values and most properties of all attributes contained in a single block.



1 Overview list

2 Value

3 Properties

4 Text Options

34.1.20.1 Overview list

Displays an overview of all attributes used in the block.

34.1.20.2 Value

Displays the currently selected attribute from the list. A new value can be typed.

34.1.20.3 Properties

Specifies properties of the selected attribute.

Layer
Specifies the layer of the selected attribute.

Color
Specifies the color of the selected attribute.

Linetype
Specifies the linetype of the selected attribute.

Plot style
Specifies the plotstyle of the selected attribute.

34.1.20.4 Text Options

Text style
Specifies the font of the text in the selected attribute.

Justification
Specifies the alignment of the text in the selected attribute.

Upside down
When checked, text is displayed upside down.

Backwards
When checked, text is displayed backwards.

Height
Specifies the height of the text in the selected attribute.

Rotation
Specifies the rotation angle of the text in the selected attribute.

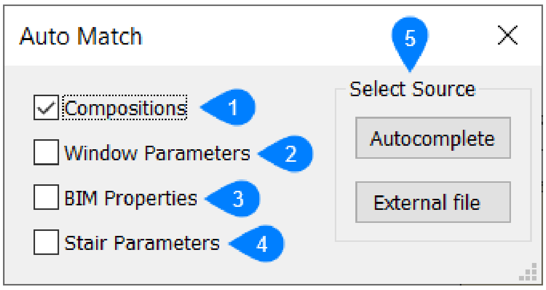
Width factor
Specifies the width of the text in the selected attribute.

Oblique angle
Specifies the oblique angle of the text in the selected attribute.

34.1.21 Auto Match dialog box

Opens via: BIMAUTOMATCH command

The **Auto Match** dialog box defines which characteristics of example entities are matched and lets you select a source.



1 Compositions

2 Window Parameters

3 BIM Properties

4 Stair Parameters

5 Select Source

34.1.21.1 Compositions

When checked on, the BIM properties of entities will be copied.

Entities without a composition will be matched with the other compositions in your drawing, based on type, spatial location and orientation of the entity, as well as on the 'Is external'-property.

If all entities have compositions, you can overwrite some by choosing a source entity. AutoMatch will then copy this source's composition to the rest of the valid entities.

34.1.21.2 Window Parameters

When checked on, the parameters of the windows will be copied.

If there are unchanged windows in your drawing, AutoMatch will match the properties based on width, height, type, spatial location and orientation of the windows and type and composition of the entities they are in.

If all windows have changed parameters, you can overwrite some by choosing a source window. AutoMatch will then copy this source's parameters to the rest of the valid windows.

Note: Some window parameters will never be copied along.

- W (= width of the window)
- H (= height of the window)

34.1.21.3 BIM Properties

When checked on, the BIM properties of entities will be copied.

If there are entities with unchanged properties in your drawing, AutoMatch will match the properties based on BIM type, composition and spatial location of the other entities in your drawing.

If all entities have a changed property, you can overwrite it by choosing a source entity. AutoMatch will then copy this source's property to the rest of the valid entities.

Some general BIM properties will never be copied along:

- GUID
- Composition
- Profile
- SpatialLocation
- IsExternal

Some beam and column related BIM properties will never be copied along:

- eomHash
- athString
- XAxis
- YAxis
- ZAxis
- ProfileName
- EndPoint

- StartPoint
- Valid
- InstancePropertySets

Some window and door related BIM properties will never be copied along:

- Name
- SubtractAllSolids
- Translucency
- OverallWidth
- OverallHeight

Some slab related BIM properties will never be copied along:

- RoomBounding

Some room related BIM properties will never be copied along:

- Number
- RoomArea
- RoomRepresentation

Some stair related BIM properties will never be copied along:

- StairType
- NumberOfRisers
- NumberOfTreads

34.1.21.4 Stair Parameters

When checked on, the parameters of a stair will be copied.

If there are unchanged stairs in your drawing, AutoMatch will match the properties based on number of treads, number of risers, height, width, length and type of the stair.

If all stairs have changed parameters, you can overwrite some by choosing a source stair. AutoMatch will then copy this source's parameters to the rest of the valid stairs.

Note: Some stair parameters will never be copied along:

- Height_Staircase
- Stair_Width
- Length_Staircase

34.1.21.5 Select Source

Autocomplete
Matches all chosen options for entities in the open drawing with similar entities in the open drawing.

External file
Opens the **Select file to insert** dialog box. Here you can choose a file to match all chosen options for entities in the chosen drawing with similar entities in the open drawing.

34.2 B

34.2.1 Background dialog box

Opens via: BACKGROUND command

The **Background** dialog box enables you to apply a background to the current viewport.

You can choose from four types: **None**, **Solid**, **Gradient**, and **Image**.

Note: Set Backgrounds = On in **Visual Styles** in **Drawing Explorer** dialog box to apply the changes you may make in **Background** dialog box. This option is available in all visual styles, except 2DWIREFRAME.

34.2.1.1 None

When background type is set to **None**, the **Background** dialog box includes 2 areas:

1 Type

2 Preview

Type

Specifies the background type for the current viewport. When the background type is **None**, default colors are applied to the current viewport.

Preview

Displays a preview of the background. When set to **None**, the background depends on the PERSPECTIVE variable:

- **Off**: background is the color specified by the BKGCOLOR variable, most often black or white.
- **On**: two-color gradient that simulates the ground with the sky; the colors are specified by variables that begin with Gradient-, such as GRADIENTCOLORTOP, GRADIENTCOLORMIDDLE and GRADIENTCOLORBOTTOM.

Note: After changing the variables' values, rerun the GRADIENTBKGON command.

34.2.1.2 Solid

When the background type is set to **Solid**, the **Background** dialog box includes 3 areas:

1 Type

2 Choose a color

3 Preview

Type
Specifies the background type for the current viewport. When the background type is **Solid**, a single color is applied to the current viewport.

Choose a color
Specifies the current color for the solid background. You can click on the color swatch to display the **Select Color** dialog box and specify a different color.

Preview
Displays a preview of the background using the color you selected.

34.2.1.3 Gradient

When the background type is set to **Gradient**, the **Background** dialog box includes 3 areas:

1 Type

2 Colors

3 Preview

Type

Specifies the background type for the current viewport. When the background type is **Gradient**, a gradient between 2 or 3 colors is applied to the current viewport.

Gradient settings

Specifies the rotation and colors for the gradient background.

Rotate field

Specifies the rotation angle of the gradient background. You can type an angle or use the arrows to increase or decrease the angle.

Colors

Specifies the colors for the gradient background.

Two

Applies a two-color gradient to the background. When you select this option, the dialog box displays two color swatches. One is for the top color and one is for the bottom color. You can click on the color swatch to display the **Select Color** dialog box and specify a different color.

Three

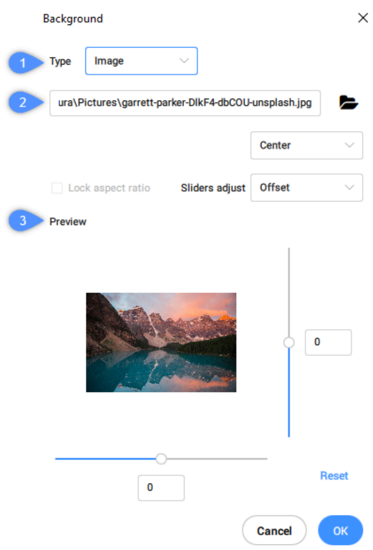
Applies a three-color gradient to the background. When you select this option, the dialog box displays three color swatches. One is for the top color, one is for the middle color, and one is for the bottom color. You can click on the color swatch to display the **Select Color** dialog box and specify a different color.

Preview

Displays a preview of the background using the colors you have selected.

34.2.1.4 Image

When the background type is set to **Image**, the **Background** dialog box includes 3 areas:



1 Type

2 Choose image and settings

3 Preview

Type

Specifies the background type for the current viewport. When the background type is **Image**, a raster image is applied to the current viewport.

Image settings

Specifies the file and position for the image background.

Path field

Specifies the name and location of the image file. You can enter the path by typing or choose the browse button to open the **Select Image File** dialog box.

Position

Specifies the position of the image within the current viewport. Options include **Center**, **Stretch** and **Tile**.

Center

Centers the image in the viewport.

Stretch

Stretches small images to fit the size of the viewport.

Tile
Tiles the image as needed to fill the viewport.

Lock aspect ratio
Specifies whether the aspect ratio of the image is maintained when the image is scaled. This option is only available when the **Sliders adjust** option is set to **Scale**.

- **ON**: maintains the image's aspect ratio; the x and y sliders move together.
- **OFF** : allows the image to be distorted; the x and y sliders move independently of each other.

Sliders adjust
Specifies the function of the 2 slides that display in the **Preview** pane.

- **Offset**: moves the image in the viewport.
- **Scale**: resizes the image; allows you to make the image the same size as the viewport.

Preview
Displays a preview of the background using the image file and options you specify.

X and Y sliders
Adjusts the offset and/or the scale of the image in the viewport. Which one is adjusted, offset or scale, is determined by the **Slides adjust** setting.

- **X**: moves the slider left and right. The X field updates accordingly. You can also type a value into the X field.
- **Y**: moves the slider up and down. The Y field updates accordingly. You can also type a value into the Y field.

Reset
Resets the offset to a default value of 0 and resets the scale to a default value of 1.

34.2.2 BIM BCF panel - Custom login dialog box

Opens via: **BIM BCF** panel

The BIM BCF **Custom log in** dialog box allows you to connect to a cloud service to start collaborating on issues in the drawings.

1 Source url

2 Log in

34.2.2.1 Source url

It is generally easier to use cloud services to manage BCF issues. If you wish to connect to these services from within BricsCAD BIM, you need an account to these services first. This allows you to create issues and manage them in real-time from anywhere.

To connect to a cloud service, insert the URL in the **Source url** field.

Note: The displayed URL represents the web address of the last used external BIM collaboration service which is stored by the BCFSOURCEURL system variable.

34.2.2.2 Log in

Connects you to the cloud service. Insert the e-mail address and the password to connect.

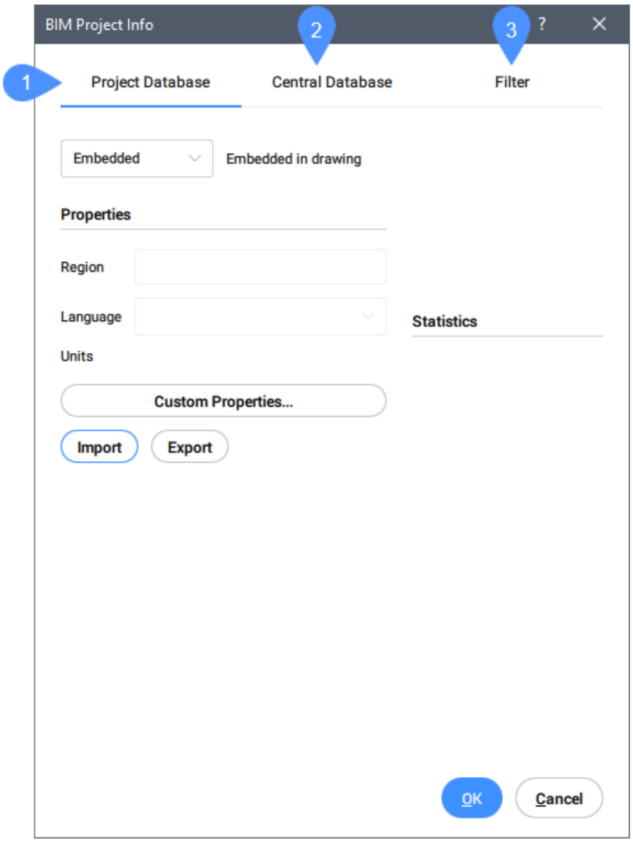
After logging in, you can close the browser window and start collaboration on issues in BricsCAD.

34.2.3 BIM project info dialog box

Opens via:

- BIMPROJECTINFO command
- **BIM Compositions** panel

The **BIM Project Info** dialog box shows information about properties, materials, compositions, profiles and spatial structures of the Project and Central databases. It allows you to import or export the database and set the project database as embedded or external. When you launch the BIMPROJECTINFO command in BricsCAD, the **BIM project info** displays in a dialog box that includes certain components and tabs. Project and central databases of BIM models can be changed using this dialog box.



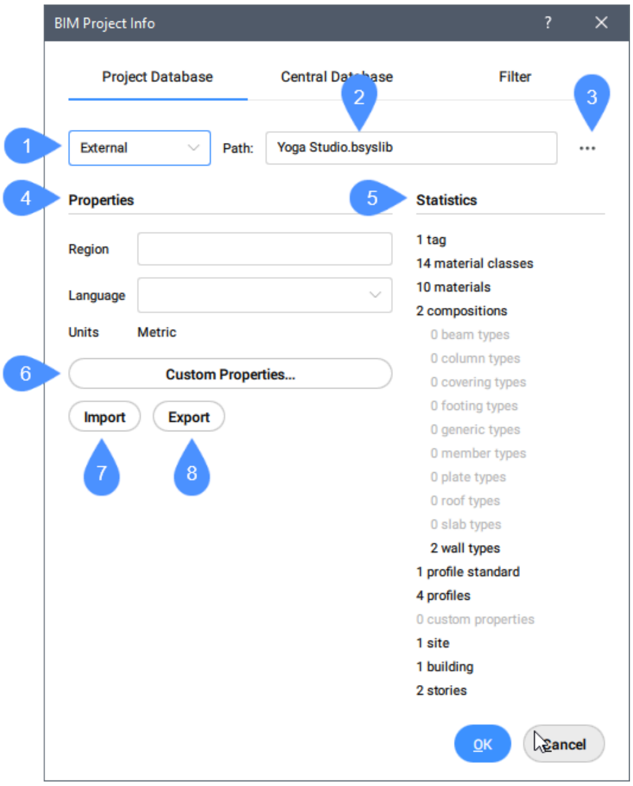
1 Project Database

2 Central Database

3 Filter

34.2.3.1 Project Database

Shows the BIM project info.



1 Project database drop-down list

2 Path

3 Browse

4 Properties

5 Statistics

6 Custom properties...

7 Import

8 Export

Project database drop-down list

Choose between **Embedded** or **External**.

Embedded

Saves the project database in the drawing file.

External

Saves the project database in a BIM database file (.bsyslib).

Note: The external database keeps the database independent from the current drawing. This is especially useful if you want to share the database between multiple models, e.g. when a model consists of several Xref drawings.

When switching from **External** to **Embedded**, the content of the external database is copied to the embedded project database.

Path
Displays the path of the project database.

Browse
Opens the **Select or create a Bricsys Library** dialog box.

Properties
Region, Language and Units are the properties of the BIM project.

Region
Fill out the region field.

Language
You can choose one of the language options from the drop-down list.

Units
Displays the system of measurement used in the BIM project (metric, imperial).

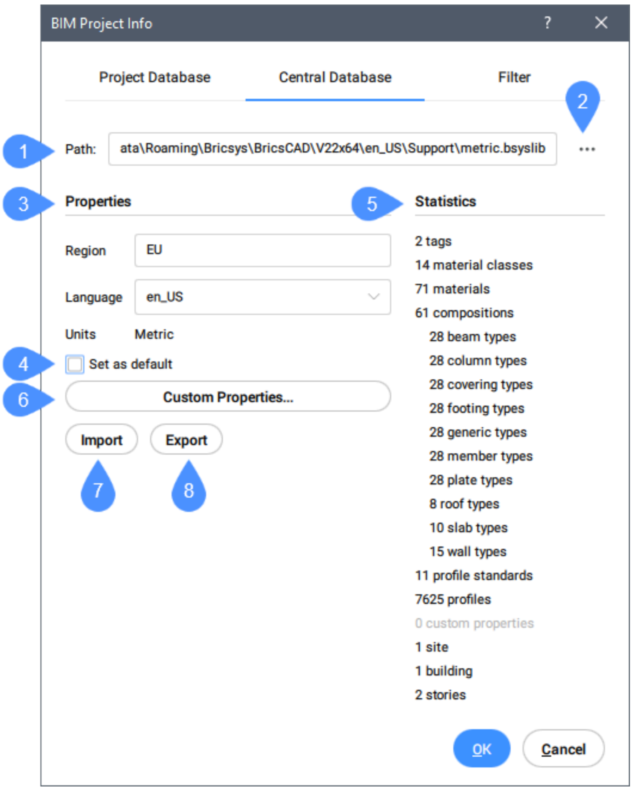
Statistics
Shows the location, properties, spatial structures (the amount of buildings, stories, etc.) and the content of the current project database of the BIM model.

Custom properties…
Adds custom properties to materials and compositions. Opens the **Edit Custom Properties** dialog box (see **Edit Custom Properties dialog box** related article).

Import
Imports the selected *.xml or *.csv library files to enlarge the project database, for example, to add profiles. Opens the **Select xml/csv file to import** dialog box.

Export
Saves .xml library files in the defined location. Opens the **Select location to save xml file** dialog box.

34.2.3.2 Central Database



1 Path

2 Browse

3 Properties

4 Set as default

5 Statistics

6 Custom Properties…

7 Import

8 Export

Path

Displays the path of the central project database.

Browse

Opens the **Select or create a Bricsys Library** dialog box.

Properties

Region, Language and Units are the properties of the project.

Region

Fill out the region field.

Language

You can choose one of the language options from the drop-down list.

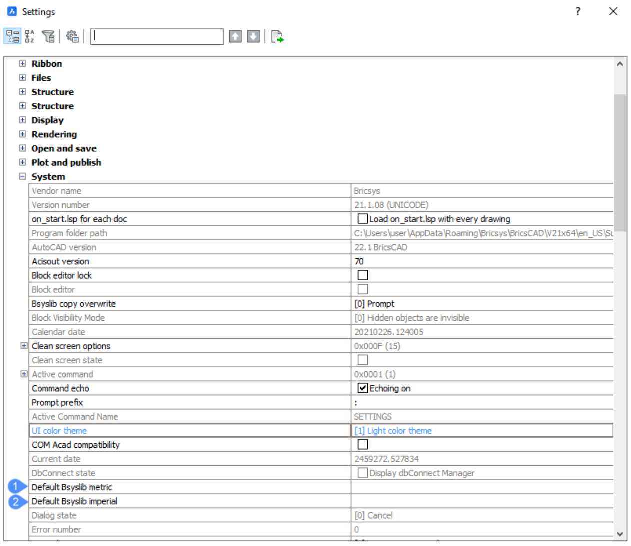
Units

Displays the system of measurement used in the BIM project (metric, imperial).

Set as default

This option allows you to set the selected database as default library database for next projects.

Note: You can edit the default library database settings in the **Settings** dialog box under **Program options** > **System**.



Statistics

Shows the location, properties, spatial structures (the amount of buildings, stories, etc.), the content of the central project database of the BIM model.

Custom Properties…

Adds custom properties to materials and compositions. Opens the **Edit Custom Properties** dialog box.

Import

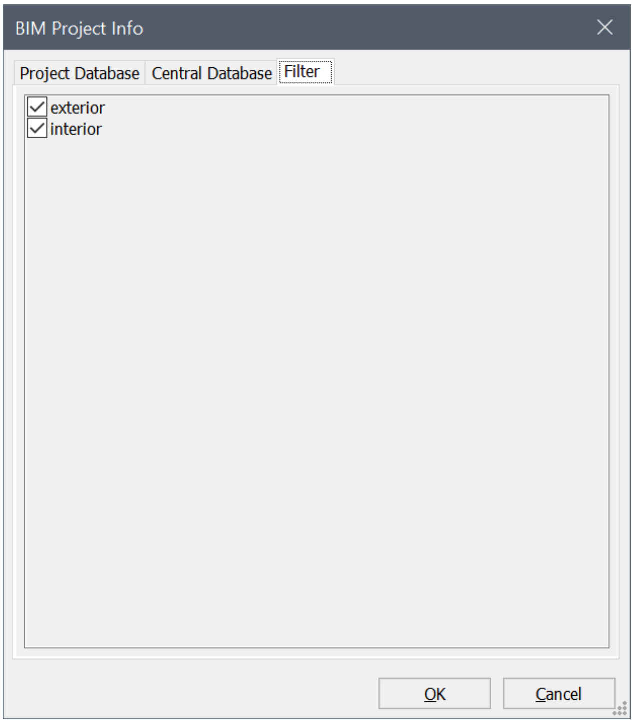
Imports the selected *.xml library files. Opens the **Select xml/csv file to import** dialog box.

Export

Saves .xml library files in the defined location. Opens the **Select location to save xml file** dialog box.

34.2.3.3 Filter

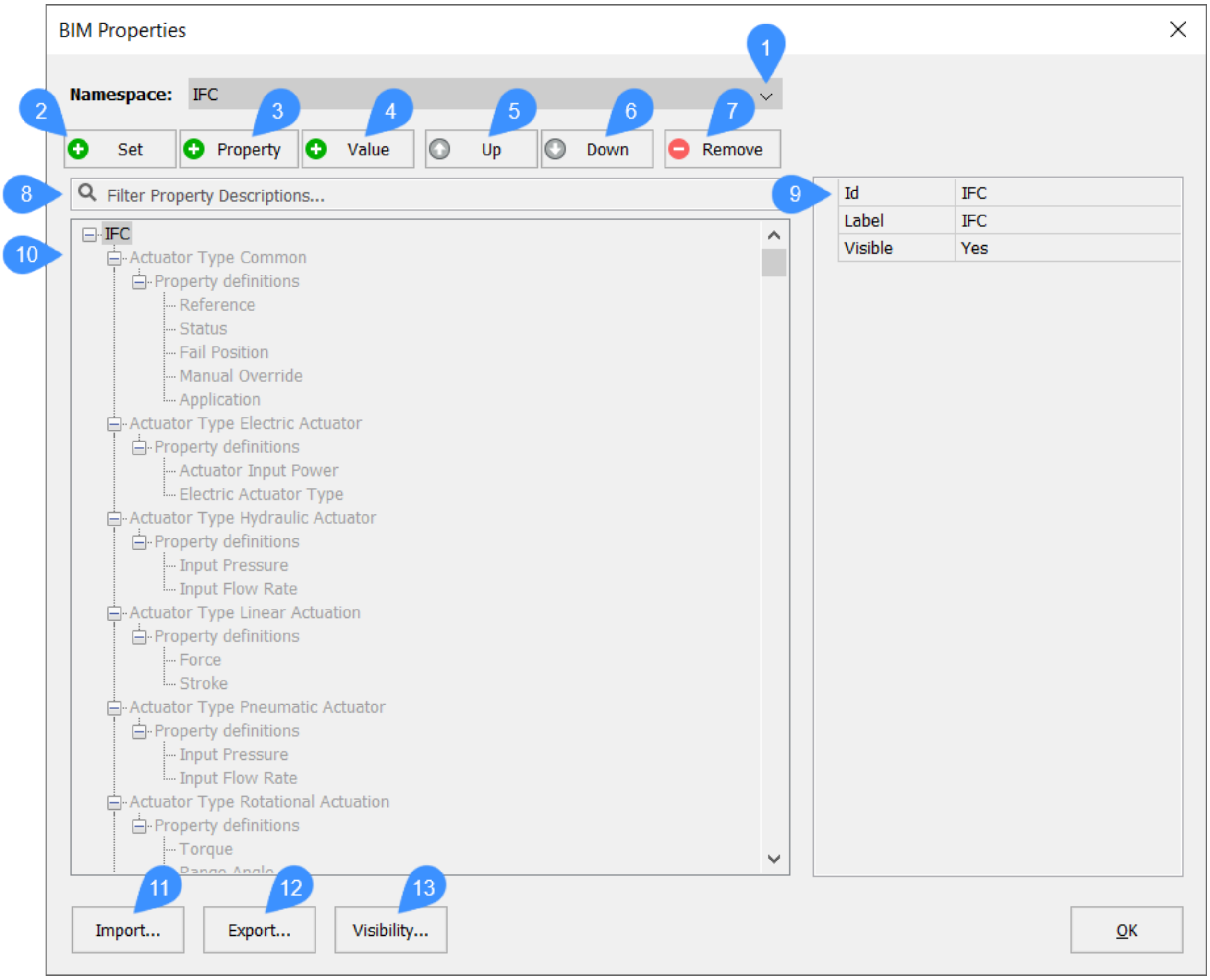
Check the tags that you want to use in the filter.



34.2.4 BIM Properties dialog box

Opens via: BIMPROPERTIES command

The **BIM Properties** dialog box allows you to create, edit and delete property definitions and to organize properties in property sets.



1 Namespace

2 Add Set

3 Add Property

4 Add Value

5 Up

6 Down

7 Remove

8 Filter

9 Properties

10 Properties Tree

11 Import

12 Export

13 Visibility

34.2.4.1 Namespace

By default, the below namespaces are available:

- **User**: allows you to create user defined properties.
- **IFC**: shows the IFC2x3 and IFC4 properties merged under a single namespace.

  Note:

  - Differences between the IFC schemas will be dealt with in the background during IFC Import or Export.
  - Static attributes from the BIM namespace are marked as deprecated not to be used instead of dynamic ones.
- **Quantity**: shows the quantities properties. It is read-only.

It is also possible to import namespaces, using the **Import** button.

- **Classification System**: allows you to organize your BIM models with standard classification codes that are used in the construction industry.

34.2.4.2 Add Set

Creates a new property set.

34.2.4.3 Add Property

Adds a new property to the currently selected property set.

34.2.4.4 Add Value

Creates a new value definition.

34.2.4.5 Up

Moves up the selected property or value.

34.2.4.6 Down

Moves down the selected property or value.

34.2.4.7 Remove

Removes the selected property set, property definition value definition, or namespace.

Note: In case of removing a namespace, a warning message displays.

34.2.4.8 Filter

Displays only the properties of which the Id and Label contain the search string typed. The results in the properties tree are filtered as you type.

34.2.4.9 Properties

Displays the properties of the selected property sets, property definitions and value definitions.

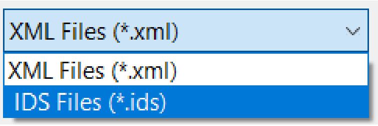
Note: Visibility rules for the property definitions are available. It is possible to make the visibility of a property or property set dependent on the value of another property.

34.2.4.10 Properties Tree

Displays the property sets, property definitions and value definitions.

34.2.4.11 Import

Imports property definitions based on the XML or IDS-XML schema definitions. Opens the **Select XML file to import** dialog box, where you can choose the file to import, be it XML or IDS.



An XSD (XML Schema Definition) file has been created to generate XML files for BIM properties. The file can be accessed here.

The IDS-XML property definition contains IFC properties (defined by buildingSMART®), custom properties, and IFC attributes. An IDS namespace is created. When there are properties in the IDS file that are not part of the IFC Pset properties, they will be added as new property definition to the IDS namespace.

Note:

- When an IDS file is imported, the SHOWIDSPROPERTIESONLY system variable is automatically set to ON. This toggles on the **IDS Properties** option in the **Visibility** dialog box.
- When the SHOWIDSPROPERTIESONLY system variable is ON, only the custom properties, added to the IDS namespace, as well as the IFC properties in the IFC namespace that were part of the IDS file are displayed in the **Properties** panel (hiding all IFC properties that are not specified in the imported IDS file).

34.2.4.12 Export

Exports to an XML file only the active namespace (selected in the drop-down menu) with its property sets (properties/quantities) as specified by the IFCEXPORTIDSPROPERTIESONLY system variable:

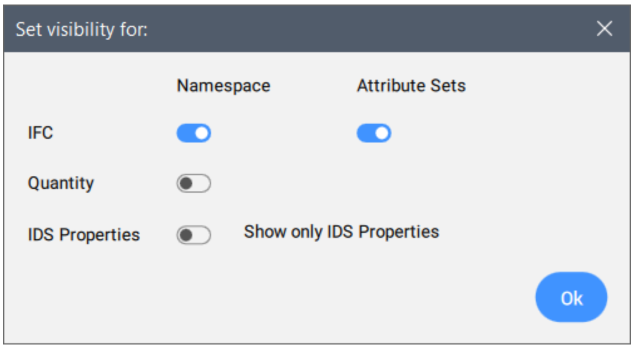
- Export all active namespace's properties (OFF).
- Export only the active namespace's IFC and custom properties mentioned in the imported IDS file (ON).

This file can then be imported in other drawings.

Note: If the IFCEXPORTMULTIPLYELEMENTSASAGGREGATED system variable is set to ON, value definitions for visible Ply-elements are also exported.

34.2.4.13 Visibility

Enables you to toggle on and off the visibility of IFC **Namespace** and **Attribute Sets**, **Quantity** and **IDS Properties**.



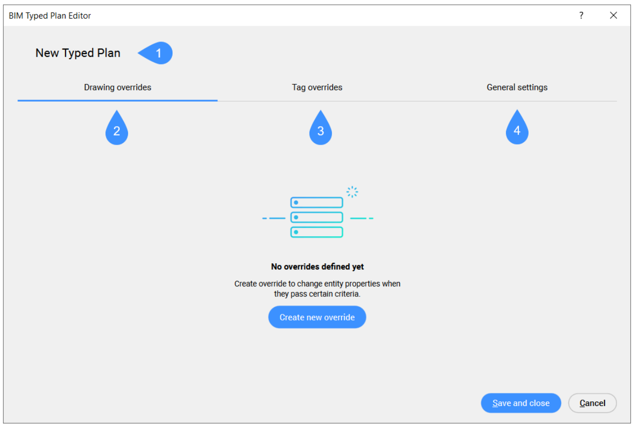
Note: The **IDS Properties** toggle changes the value of the SHOWIDSPROPERTIESONLY system variable.

34.2.5 BIM Typed Plan Editor dialog box

Opens via:

- double clicking a Typed Plan listed in the **Files** tab of the **BIM Project Browser** panel.
- clicking a Typed Plan listed in the **BIM Typed Plans** dialog box.

The **BIM Typed Plan Editor** dialog box allows you to edit the existing Typed Plans.



1 Typed Plan name

2 Drawing overrides

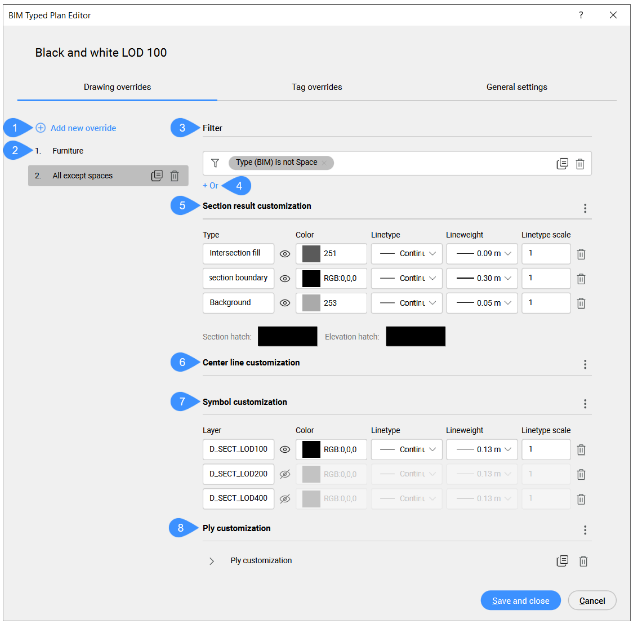
3 Tag overrides

4 General settings

34.2.5.1 Typed Plan name

Specifies the name of the selected Typed Plan to be edited. You can click on the selected Typed Plan name to rename it, then press **Enter**.

34.2.5.2 Drawing overrides



1 Add new override

2 Drawing overrides list

3 Filter

4 Add new filter

5 Section result customization

6 Center line customization

7 Symbol customization

8 Ply customization

Add new override

Adds a new override element to the list that functions as a top down entity filter. When an entity fits a criterion in the list, the rest of the overrides are no longer checked.

> ***Note:*** Since the order of the list is important, it is best to start with the most specific filter lines and then add more broader filter lines. For example you can leave your last filter line so generic to encompass all other elements, by not specifying any filter expression in it.

Drawing overrides list

Displays the hierarchically organized list of drawing overrides. You can rearrange the position of an override in the list by dragging and dropping it to the desired position.

Filter

Allows you to edit filter rules for drawing overrides by adding one or more parameter filters from the drop down list. Filter rows can be added to filter by multiple parameter combinations. These rows are separated using the logical "OR" operation. The rows can be copied by clicking the copy symbol or deleted by clicking the delete symbol.

> ***Note:*** User defined properties can be used as filter in Typed Plans only when the BIM Project uses a Project database stored as an external BSYSLIB file. For more information about the Project and Central databases, see the articles **BIM data structure** and **BIM project info**.

Add new filter row

Click the **+ Or** button to add a new filter row.

Section result customization

Allows you to customize how the lines of each element that is being processed by the section generation algorithm will turn out.

Click the context menu button to open the menu options:

Add customization

Click the arrow next to **Add customization** to open the drop-down list with customization types: **Background**, **Intersection boundary**, **Intersection fill**, **Elevation fill**, **Hidden lines**, **Cut away geometry**. Click a type to add it to the **Section result customization** so you can customize the line's visibility, colour, type, weight and scale.

Override hatch

Allows you to specify the entity's hatch appearances (section and elevation), thus not to use the default hatch patterns already defined within the **Physical Materials** library. When selected, the options **Section hatch** and **Elevation hatch** appear for the **Section result customization**. Click the hatch patterns here to open the **Material Pattern Editor** dialog box that allows you to further edit the hatches.

Note: When you create any new **Physical Material** entry in the **In Project** category as part of the **Typed Plan** template, your desired configurations are independent of the drawing project and stored in the template instead. You can apply this template later to other project models as you wish.

Apply for plies

You can further ensure the customization applies to the plies within the entity composition, if appropriate, by checking the **Apply for plies** option.

Center Line customization

Determines the appearance of the center of planar or linear entities. The resulting center lines, for example from cutting a wall's center plane in a floor plan, can be customized.

Note: There is no default setting to adjust the visibility of these center elements, and they are not normally displayed in section results.

Symbol customization

Click the context menu button to open the menu options:

Choose external file...

Opens the **Set external symbol source.** dialog box where you can select an external symbol drawing in DWG file format. This allows you to apply a saved symbol customization to specific layers within your external symbol source.

Create new external file...

Opens the **New external symbol source.** dialog box where you can save an external symbol drawing in DWG file format. This allows you to create a new symbol drawing on a default symbol template.

Add customization

Adds a new symbol customization line. This allows you to filter on the names of the layers which contain symbols and edit those layers' lines appearance.

Ply customization

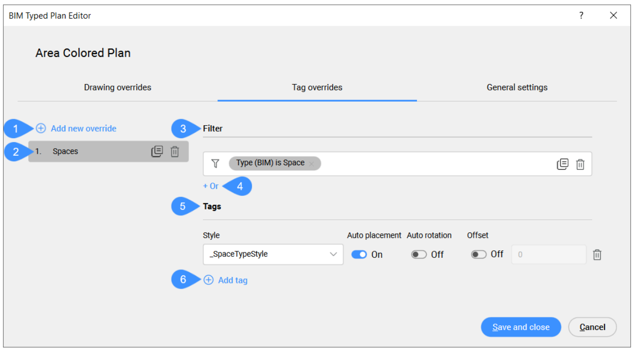
Can be used to show only a certain type of plies in a plan view. An easy example is the visibility of only structural plies in a structural plan. Click the context menu button to open the menu options.

Add ply customization.
Allows to filter entity plies by adding one or more parameter filters from the drop down list. A main categorization for the customization of plies is predefined as Structure function and No structure function. Filter rows can be added to filter by multiple parameter combinations.
For each Filter combination, ply customizations can be defined.

34.2.5.3 Tag Overrides

Tag overrides allow you to define the visualization of entities containing specific tags. In the **Tag overrides** tab, preferences can be specified.

Note: When there is no tag defined for an object in the Typed Plan, the rules set in _TagTypeToStyle.xml are used as fallback rules.



1 Add New Override

2 Tag overrides list

3 Filter

4 Add new filter row

5 Tags

6 Add tag

Add new Override
Creates a new tag override element to the list. By default, the template is named **New override** with the text highlighted. This highlighted override title can be modified by entering an custom name and pressing **Enter** to save and apply.

Note: Created overrides can be copied by clicking the copy icon or deleted by clicking the delete icon. To rename the override, double-click its name.

Tag overrides list
Displays the hierarchically organized list of tag overrides. You can rearrange the position of an override in the list by dragging and dropping it to the desired position.

Filter
Allows you to edit filter rules for tag overrides. It is similar with changing the filter rules for drawing overrides described above.

Note: User defined properties can be used as filter in Typed Plans only when the BIM Project uses a Project database stored as an external BSYSLIB file. For more information about the Project and Central databases, see the articles **BIM data structure** and **BIM project info**.

Add new filter row

Click the **+ Or** button to add a new filter row.

Tags

Allows you to add and edit tag customizations. You can also deleted tag customizations. By adding customizations for different entity types, customized styles can be defined including several characteristics:

Style

Allows you to select the tag style.

Auto placement

Specifies whether the placement of the tag should be defined automatically or not.

Auto Rotation

Specifies whether the rotation of the tag should be defined automatically or not.

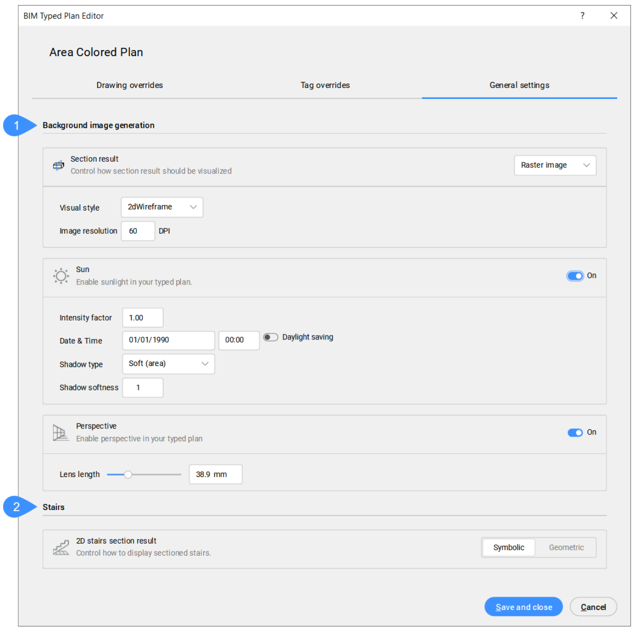
Offset

Specifies whether the tag should be placed either with offset or not as well as the numerical offset value.

Add tag

Allows you to add a new tag customization.

34.2.5.4 General settings



1 Background image generation

2 Stairs

Background image generation

Section result

Allows you to control how the section result should be visualized by selecting one option from the drop-down menu:

- **Line drawing**: the section result is visualized as a line drawing.

- **Line drawing + Raster image**: the section result is visualized as a line drawing to which a background raster image is added.
- **Raster image**: the section result is visualized as a raster image.

Visual style
Allows you to choose a visual style from the drop-down menu.

Image resolution
Sets the DPI resolution for the image to be generated.

Note: **Visual style** and **Image resolution** settings are only available for **Line drawing + Raster image** and **Raster image** section result visualization types.

Sun
Allows you to toggle **On** and **Off** the sun settings for the background image. When toggled **On**, the following settings are available:

- **Intensity factor**: specifies the intensity or brightness of the sun; the higher the number, the brighter the light.
- **Date &Time**: edits the date to show the sun at a different time of year. Edit the time to show the sun at a different time of day. You can also toggle **On/Off** the effect of daylight saving time.
- **Shadow type**: allows you to select an option from the drop-down menu.
  - **Soft (area)**: displays realistic shadows with softer edges based on extended light sources, according to the **Softness** property.
  - **Sharp**: displays shadows with sharp edges.
- **Shadow softness**: determines the sampling size of shadows. Larger numbers are more accurate but take longer to render.

Perspective
Allows you to enable the perspective mode for the background image:

- **Lens length**: sets the perspective lens length.

Stairs

2D stairs section result
Allows you to control the generation of 2D representations of BIM stair entities during a section generation:

- **Symbolic**: a symbolic 2D representation is used for BIM stair section generation.
- **Geometric**: a geometric 2D representation is used for BIM stair section generation.

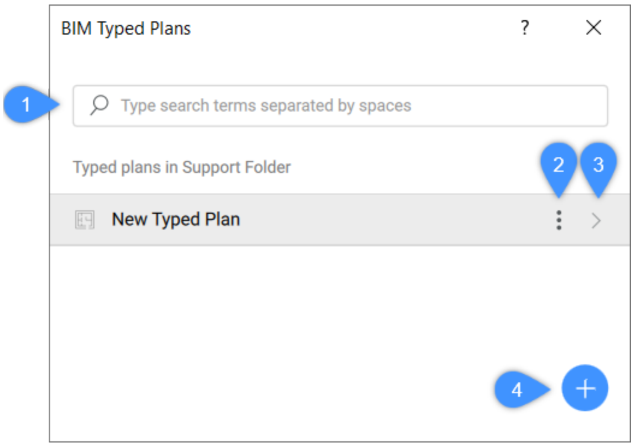
Note: This setting overrides the AUTOMATICSTAIRSECTIONBEHAVIOR system variable.

For more information, see the Command reference article **BIMGENERATE2DSTAIR command**.

34.2.6 BIM Typed Plans dialog box

Opens via: TYPEDPLANS command

The **BIM Typed Plans** dialog box lists all Typed Plans in the support folder.



1 Search

2 Context menu

3 Edit typed plan

4 Add typed plan

34.2.6.1 Search

Searches all the Typed Plans that matches the words you enter in the search box.

34.2.6.2 Context menu

Clone
Creates a copy of the selected Typed Plan.

Remove
Removes the selected Typed Plan.

Rename
Edits the name of the selected typed plan. Press Enter to save and apply the changes.

34.2.6.3 Edit typed plan

Opens the **BIM Typed Plan Editor** dialog box which allows you to edit the selected Typed Plan.

Note: You can also open the **BIM Typed Plan Editor** for any Typed Plan by clicking on it's name in the list.

34.2.6.4 Add typed plan

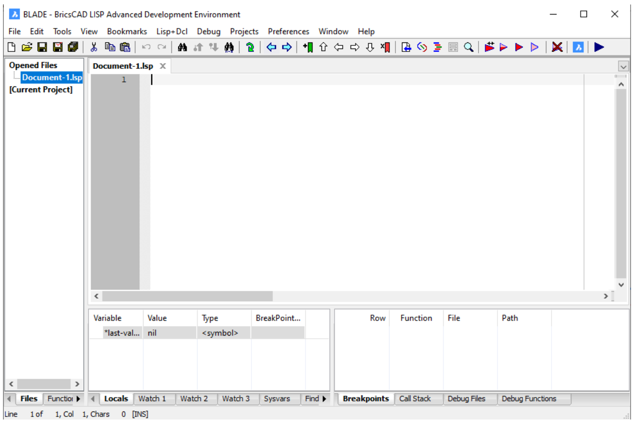
Creates a new Typed Plan.

You can create a template by clicking on the button. By default, the Typed Plan is named **New Typed Plan** with the text highlighted. After you have added a relevant name for the Typed Plan, press **Enter** to save and apply the changes. The inserted Typed Plans will be sorted alphabetically. Use the scroll bar to navigate through the list.

34.2.7 BLADE - BricsCAD LISP Advanced Development Environment dialog box

Opens via: BLADE command

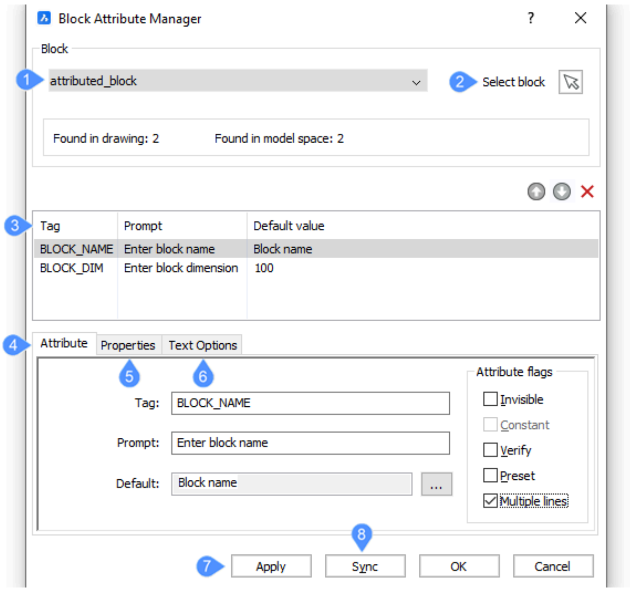
The **Blade - BricsCAD LISP Advanced Development Environment** dialog box allows you to edit and debug LISP applications.



34.2.8 Block Attribute Manager dialog box

Opens via: BATTMAN command

The **Block Attribute Manager** dialog box allows you to edit all aspects of attributes in a block definition and then optionally apply the changes to all blocks of the same name in the drawing.



1 Block name

2 Select block

3 Attribute list

4 Attribute tab

5 Properties tab

6 Text Options Tab

7 Apply

8 Sync

34.2.8.1 Block name

Select a block from the drawing whose attributes you want to edit.

You can select a block from the drop-down list.

34.2.8.2 Select block

Allows you to select a block in the drawing.

34.2.8.3 Attribute list

Select an attribute from the list. It includes the tag, the prompt and the default value.

: moves the attribute up the list.

: moves the attribute down the list.

: deletes the attribute up the block.

34.2.8.4 Attribute tab

Allows you to edit the definition of the selected attribute.

There are 5 attribute flags options:

- **Invisible**: hides the attributes from view; they are not displayed and not printed. Hidden attributes can, however, be displayed with the *ATTDISP* command.
- **Constant**: specifies a default value that the user cannot change.
- **Verify**: forces the user to enter the value a second time; this helps to ensure the value is entered correctly.
- **Preset**: inserts attributes without prompting the user; the attributes can be changed later with the ATTEDIT command.
- **Multiple lines**: when checked, allows the use of multiline text.

34.2.8.5 Properties tab

The **Properties** tab allows you to edit the properties of the selected attribute.

34.2.8.6 Text Options Tab

The **Text Options** tab allows you to edit the text properties of the selected attribute.

34.2.8.7 Apply

Applies the changes to the block definition, and keeps the dialog box open; choose another block whose attributes you wish to edit.

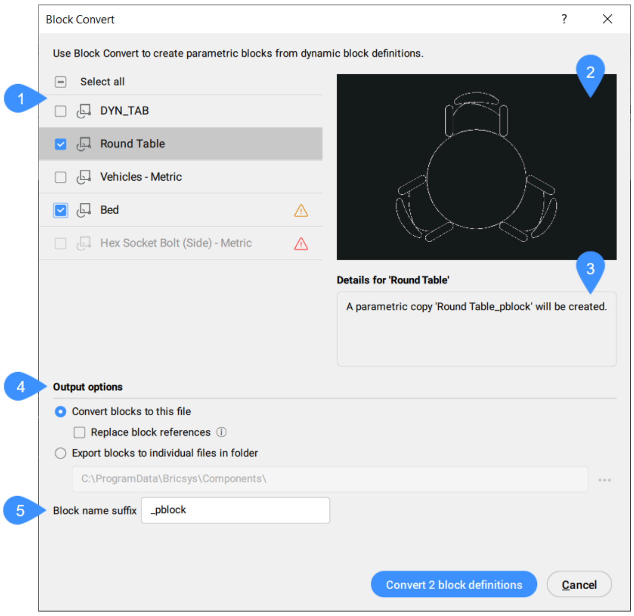
34.2.8.8 Sync

Applies the changes to all instances of the same block in the drawing.

34.2.9 Block Convert dialog box

Opens via: BLOCKCONVERT command

The **Block Convert** dialog box allows you to select the AutoCAD® Dynamic Blocks in the current drawing to be converted to BricsCAD Parametric Blocks.



1 List of blocks

2 Preview

3 Details for

4 Output options

5 Block name suffix

List of blocks

All AutoCAD® Dynamic blocks that are present in the current drawing are listed here. Individually check the blocks you want to convert to BricsCAD Parametric Blocks or check all by ticking the **Select all** option.

Note: Blocks that cannot be converted or can be partially converted are marked with a red or yellow warning sign respectively.

Preview

Displays a preview of the currently selected block in the list.

Details for

Displays information on whether the selected block can be converted or not. If the block cannot be converted, the features that are not supported are listed.
Some blocks can be converted with partial loss of information. In this case warning messages are shown in the details section.

Output options

Convert blocks to this file

Converts the dynamic blocks into the parametric blocks inside the current file.

Replace block references
If checked, all references to the selected dynamic blocks will be replaced with references to parametric blocks. All parameter values will be preserved.

Note: This option is only available if the **Export blocks to individual files in folder** option is not checked.

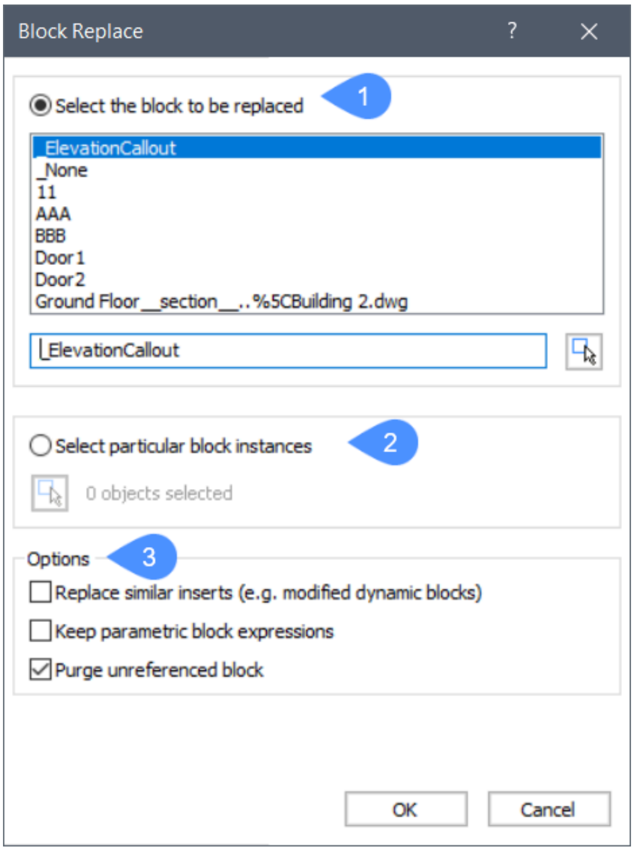
Export blocks to individual files in folder
Check this box to export the converted blocks to the specified folder. Exporting blocks to the default *C:\ProgramData\Bricsys\Components* folder makes blocks accessible through the **Library** panel. Click the **Browse...** button that opens the **Select a folder for saving exported blocks** to specify a different folder.

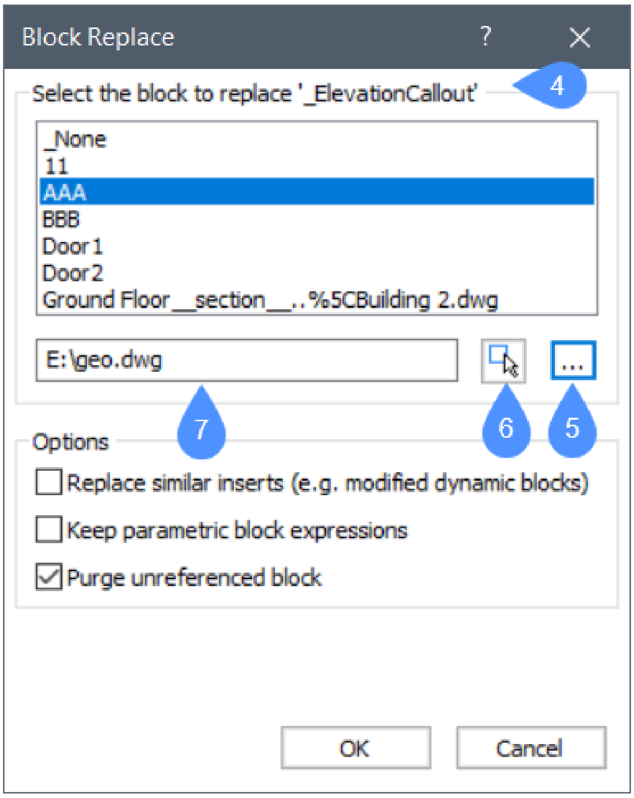
Block name suffix
Allows you to modify the default block name suffix. The last used name for suffix is remembered.

34.2.10 Block Replace dialog box

Opens via: BLOCKREPLACE command

The **Block Replace** dialog box allows you to replace specific block instances with other blocks.





1 Select the block to be replaced.

2 Select particular block instances.

3 Options.

4 Select the block to replace selected block instances.

5 Choose a .dwg or a .dxf file instead of a block name.

6 Select a block by clicking on it.

7 The blockName.

34.2.10.1 Select the block to be replaced

Selects a block by name to be replaced.

Note: The modified (anonymous) dynamic blocks cannot be selected by name.

34.2.10.2 Select particular block instances

Selects any block instances, including modified (anonymous) dynamic block instances to be replaced.

34.2.10.3 Options

Replace similar inserts

Replaces all modified (anonymous) and unmodified instances of dynamic block and all instances of non-dynamic block selected.

Keep parametric block expressions
Keeps the parameters' expressions of the old block if the replacement block has parameters with identical names.

Purge unreferenced block
Removes unreferenced block instances.

Select the block to replace selected block instances by

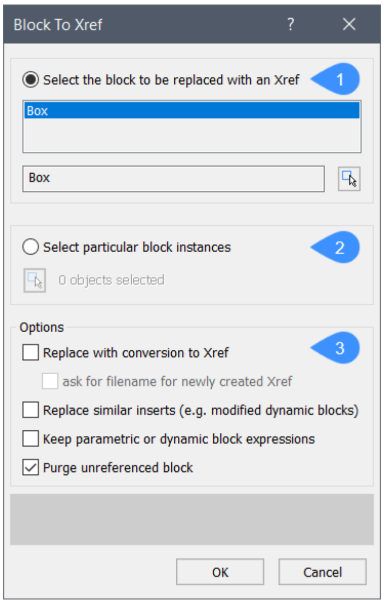
- Choosing a block name from the list.
- (5) Browsing for a .dwg or a .dxf file (the file name will be the block name).
- (6) Selecting a block by clicking on it.
- (7) Typing a file name of an external .dwg or .dxf that will be used as a new block name (e.g. fileName.dwg) or typing "blockName=fileName.dwg" when the new block name should be different than the file name.
- ***Note:*** fileName.dwg can have blocks with the same names as in the current drawing. In this case, blocks from the file do not re-define blocks with the same names already present in the current drawing.

  Note: If fileName.dwg file is not located in the support file search path, you have to indicate the complete path.

34.2.11 Block To Xref dialog box

Opens via: BLOCKTOXREF command

The **Block To Xref** dialog box allows you to replace specific block instances with Xrefs.



1 Select the block to be replaced.

2 Select block instances.

3 Options.

34.2.11.1 Select the block to be replaced

Selects all block instances to replace.

34.2.11.2 Select particular block instances

Selects only particular block instances to replace.

34.2.11.3 Options

Replace with conversion to Xref
ON: saves the block first and then replaces the block with the newly created Xref.
OFF: the block is replaced with an Xref.

ask for filename for newly created Xref
ON: the **Save Block** dialog box displays to save the block before replacing it with an Xref.

Note: If the selected block name is a dynamic block name and the **Replace similar inserts** option is On, all modified dynamic blocks are replaced by one Xref. If the selected block is an instance of a modified dynamic block and the **Replace with conversion to Xref** option is On, the newly created Xref will contain the default (unmodified) dynamic block definition (similar with WBLOCK command which cannot save anonymous block definitions).

OFF: a new drawing is created next to the original drawing with the same name as the block name.

Replace similar inserts (e.g. modified dynamic blocks)

Note: This option acts like the BLOCKREPLACE command.

ON: replaces all instances of a block and all its modified (anonymous) dynamic blocks.

Note:

- If the **Select particular block instances** is On and a non-dynamic block is selected, all instances with the same block name are replaced with the Xref.
- This option cannot be changed when the **Replace with conversion to Xref** option is also On and **Select particular block instances** is selected, because multiple Xref files and multiple block definitions can be created for one block.

OFF: anonymous (modified) dynamic blocks cannot be replaced with Xref.

Note: If **Select particular block instances** is selected, only selected instances are replaced with Xref.

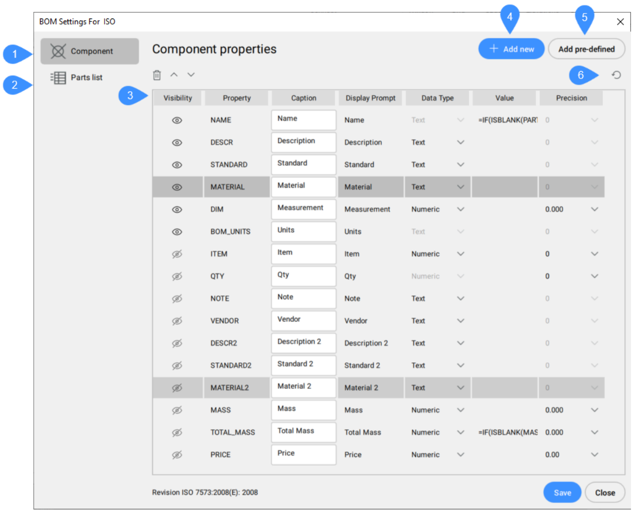
Keep parametric or dynamic block expressions
Keeps expressions of the replaced block when the replacement block has parameters with the same names.

Purge unreferenced block
Removes unreferenced block instances.

34.2.12 BOM Settings dialog box

Opens via: AMBOMSETTINGS command

Configures part references (Part References) properties and set up default properties for parts lists, and data capture for the BOM.



1 Component

2 Parts list

3 List of properties

4 Add new

5 Add pre-defined

6 Restore defaults

34.2.12.1 Component

Contains a list of component properties that are visible in Part Reference. More properties can be selected by pressing the Ctrl / Shift key.

Above this list there are delete, move up and move down icons, which are available only when a row is selected or the selected row(s) can be moved up or down.

34.2.12.2 Parts list

Contains a list of **Parts list** properties that are visible in the Parts Lists.

34.2.12.3 List of properties

Visibility
Toggles the visibility of the property in the **Part Reference** and **Parts List** dialog boxes.

Property
Displays the default name of a component property. This unique name is the one by which a BOM column is referred to in formulas.

Caption
Defines the column header displayed in the Parts List, overriding the default property name. It is editable and will appear as the component property name in the **Part Reference** and **Parts List** dialog boxes.

Note: The changes are saved in the drawing. To use them later, save the drawing as a template (DWT) file and use this template when you start a new drawing.

Display prompt
Displays the chosen column header to be displayed for this property in a BOM table or parts list.

Data type
Defines the data type for the property (Numeric or Text)

Value
Defines the default value for the property. You can also use a formula to define the property.

Precision
Defines the precision for the property.

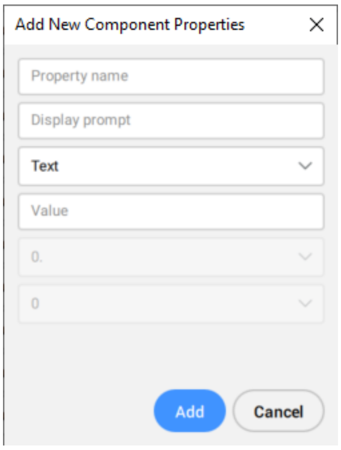
Caption Align
Defines the column header position in the cell.

Value Align
Defines the value position in the cell.

Width
Defines the width of the cell.

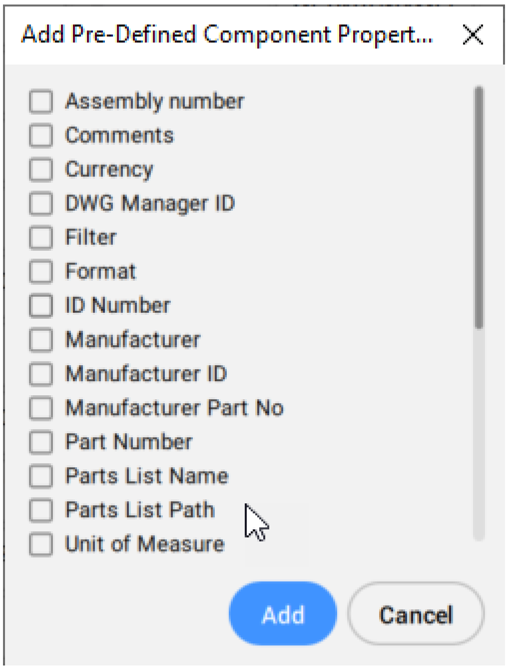
34.2.12.4 Add new

Adds new component properties as columns to BOM. Opens **Add New Component Properties** dialog box.



34.2.12.5 Add pre-defined

Adds new pre-defined component properties columns to BOM. Opens **Add Pre-Defined Component Properties** dialog box.



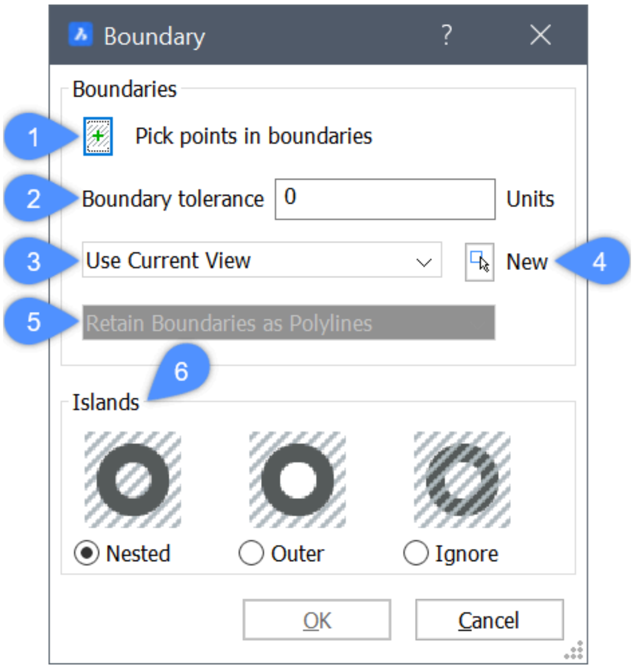
34.2.12.6 Restore defaults

Resets all settings to the default value for the current drafting standard.

34.2.13 Boundary dialog box

Opens via: BOUNDARY command

The **Boundary** dialog box allows you to create closed polylines, defined by surrounding entities.



1 Pick points in boundaries

2 Boundary tolerance

3 Boundary set

4 New

5 Retain Boundaries as Polylines

6 Islands

Pick points in boundaries

Allows you to specify a point inside an enclosed area for which you want to create a boundary. It temporarily dismisses the **Boundary** dialog box.

Boundary tolerance

Specifies the largest gap that can exist in the boundary for BricsCAD to consider the area closed.

Note: When boundary tolerance is 0, no gaps can exist.

Boundary set

Specifies where BricsCAD should search for entities that make up the boundary.

- **Use Current View**: searches all entities in the current viewport.
- **Use Boundary Set**: searches only the current selection set for possible boundaries.

New

Allows you to create a new selection set of entities that make up the boundary. It temporarily dismisses the **Boundary** dialog box.

Retain Boundaries as Polylines

Indicates how boundaries are retained. This option is read only.

Islands

Specifies island detection. An island is an enclosed area inside a boundary.

Nested

Treats every island as a boundary.

Outer

Creates a boundary only of the outermost entities.

Ignore

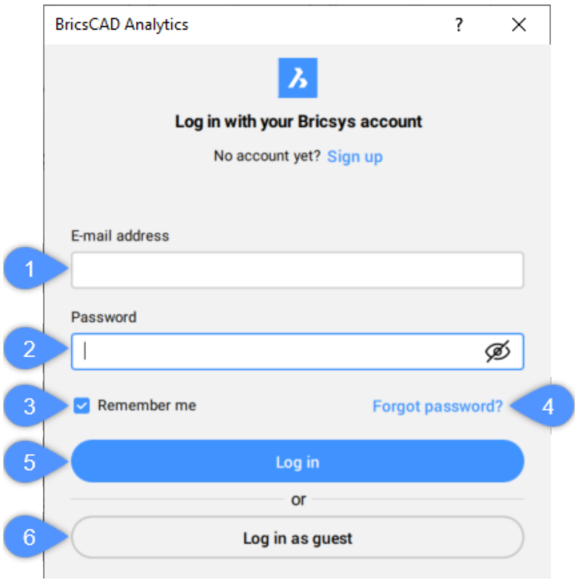
Ignores the innermost areas. A boundary is created between the outermost area and islands.

34.2.14 BricsCAD Analytics dialog box

Opens via:

- -LOGIN command
- -LOGINUSAGEDATAPROGRAM command

The **BricsCAD Analytics** dialog box allows you to login to BricsCAD with your personal Bricsys account.



1 E-mail address

2 Password

3 Remember me

4 Forgot password?

5 Log in

6 Log in as guest

34.2.14.1 E-mail address

Specifies the email address connected to your Bricsys account.

34.2.14.2 Password

Specifies the password.

34.2.14.3 Remember me

Check this box to keep being logged in.

34.2.14.4 Forgot password?

Redirects you to an internet browser to set up a new password.

34.2.14.5 Log in

Logs you in with your personal Bricsys account.

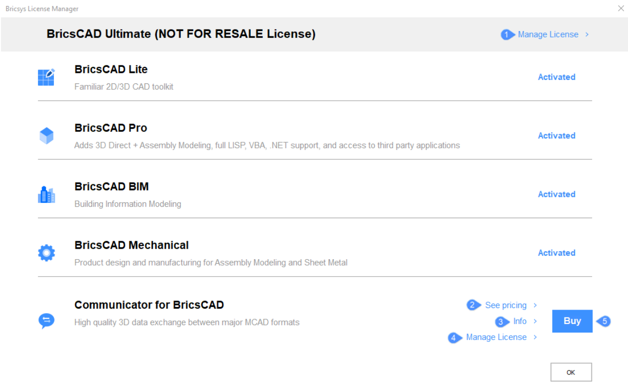
34.2.14.6 Log in as guest

Allows you to log in as guest.

34.2.15 Bricsys License Manager dialog box

Opens via: LICENSEMANAGER command

The **Bricsys License Manager** dialog box allows you to activate/deactivate the software licenses for BricsCAD and Communicator for BricsCAD®.



1 Manage License

2 See pricing

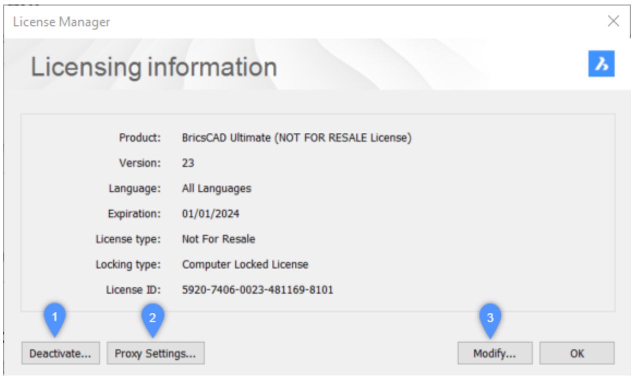
3 Info

4 Manage License for the selected product

5 Buy

34.2.15.1 Manage License

Opens the **License Manager** dialog box.



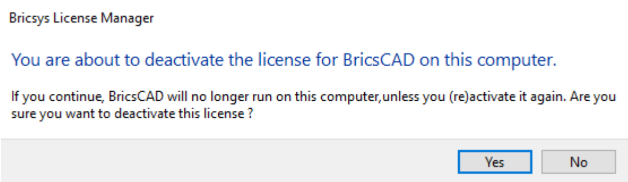
1 Deactivate

2 Proxy Settings...

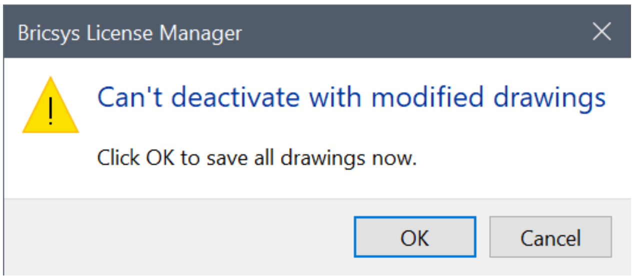
3 Modify...

Deactivate

Opens the **Bricsys License Manager** dialog box for confirming deactivation.



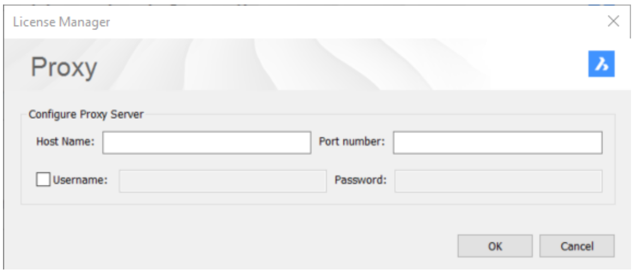
Note: If you have unsaved drawings, a warning message displays. You have to save all drawings before deactivating the license.



Important: The BricsCAD session closes after deactivating the license.

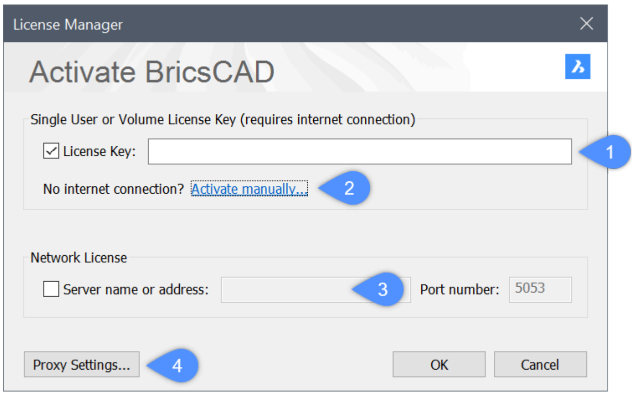
Proxy Settings…

Opens the **License Manager** dialog box where you can configure the proxy server.



Modify…

Opens the **License Manager** dialog box for activating BricsCAD where you can insert your license key, your network license or activate it manually.



1 License Key

2 Activate manually…

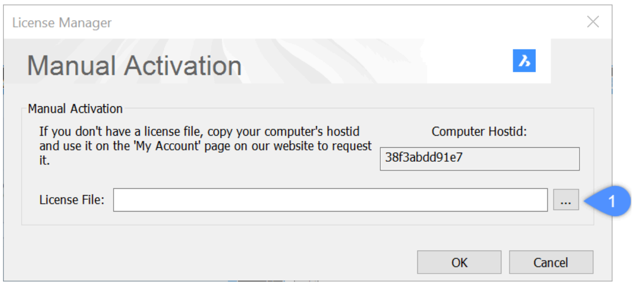
3 Network License

4 Proxy Settings…

License Key

If you are connected to the internet, you can enter in this field a single user or volume license key.

Activate manually...
Opens the **License Manager** dialog box for **Manual Activation**, where you can select the license file. The LIC file can be found in this path: *C:\ProgramData\Bricsys\BricsCADV25.lic*



1 Select license file

Select license file
Opens the **Select license file** dialog box.

Network License
If you are using a network license, enter the host name or the IP address of the Network License Server.

Proxy Settings...
Opens the **Bricsys License Manager** dialog box to configure the proxy server.

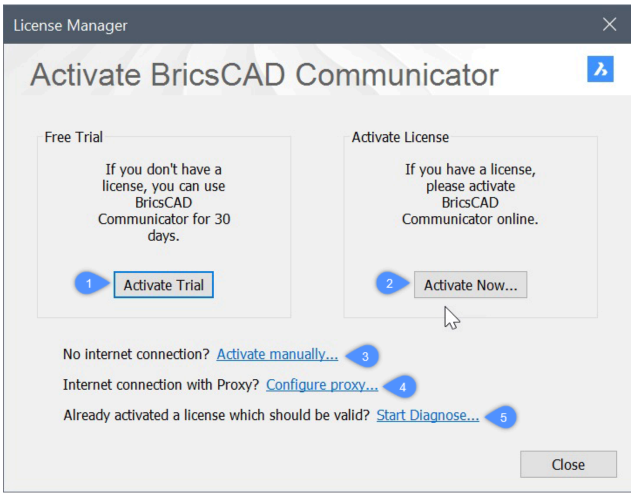
34.2.15.2 See pricing

Opens the Bricsys official website (Bricsys), where you can see the prices for Communicator for BricsCAD® and 1 year/3 years/lifetime BricsCAD subscription.

34.2.15.3 Info

Opens the **Communicator Info** dialog box. See **Communicator Info dialog box** article.

34.2.15.4 Manage License for the selected product



1 Activate Trial

2 Activate Now

3 Activate manually…

4 Configure proxy…

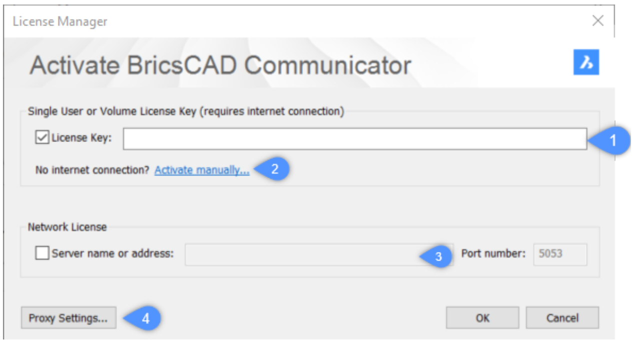
5 Start Diagnose…

Activate Trial

Activates the 30 days free trial of Communicator for BricsCAD®.

Activate Now

Opens the **License Manager** dialog box for **Activate BricsCAD Communicator**.



1 License Key

2 Activate manually

3 Network License

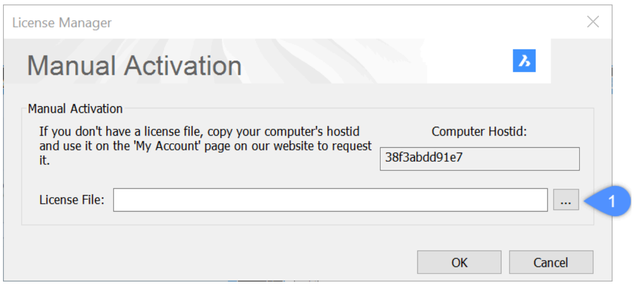
4 Proxy settings

License Key

If you are connected to the internet, you can enter in this field a single user or volume license key.

Activate manually…

Opens the **License Manager** dialog box for **Manual Activation**, where you can select the license file. The LIC file can be found in this path: *C:\ProgramData\Bricsys\BricsCADV25.lic*



1 Select license file

Select license file

Opens the **Select license file** dialog box.

Network License

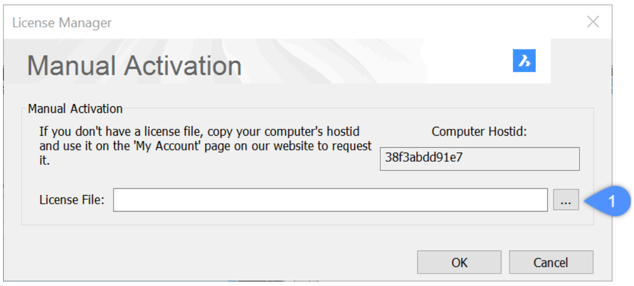
If you are using a network license, enter the host name or the IP address of the Network License Server.

Proxy Settings…

Opens the **License Manager** dialog box to configure the proxy server.

Activate manually…

Opens the **License Manager** dialog box for activating Communicator for BricsCAD®.



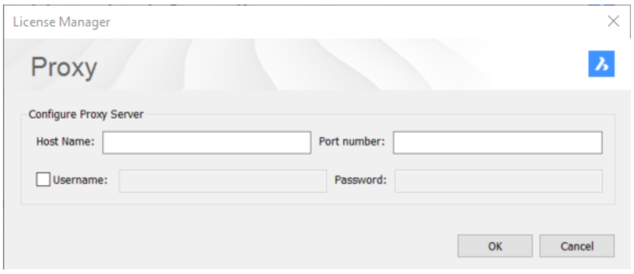
1 Select license file

Select license file

Opens the **Select license file** dialog box.

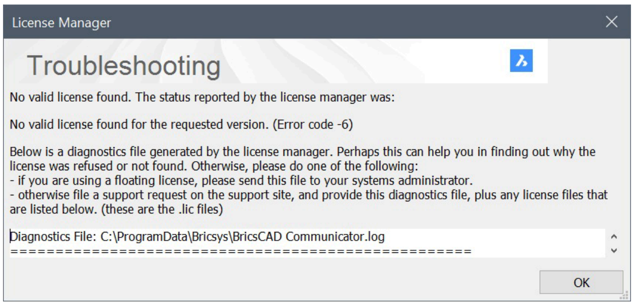
Configure proxy…

Opens the **License Manager** dialog box where you can configure the proxy server.



Start Diagnose…

Opens the **License Manager** dialog box for **Troubleshooting**.



34.2.15.5 Buy

Opens the Bricsys official website (Bricsys) where you can buy or subscribe to BricsCAD.

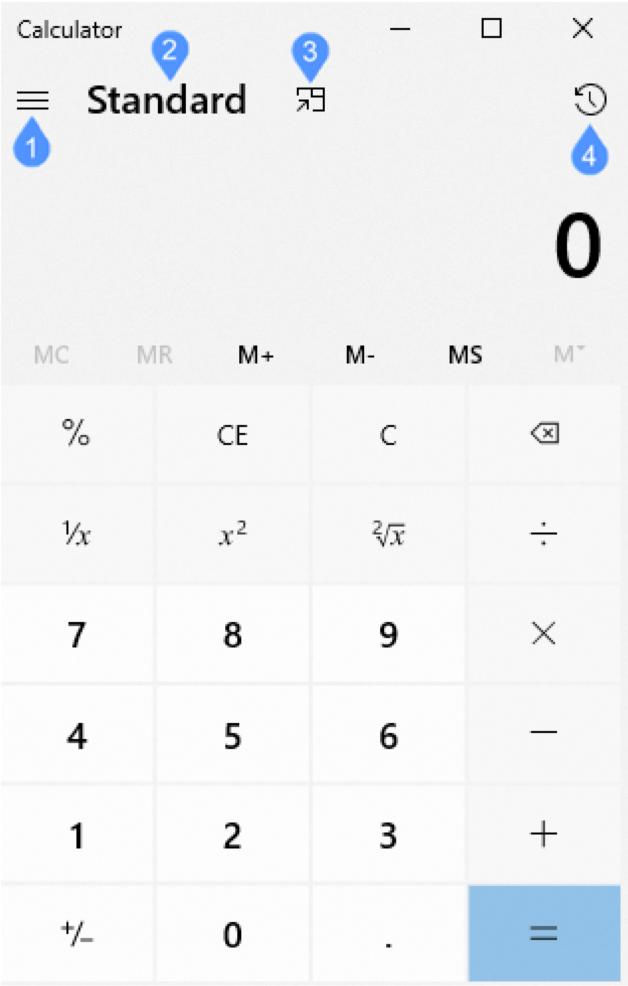
34.3 C

34.3.1 Calculator dialog box

Opens via: CAL command

The **Calculator** dialog box displays the operating system's software calculator.

When you are using the Standard view, it includes 4 areas.



1 View

2 View's name

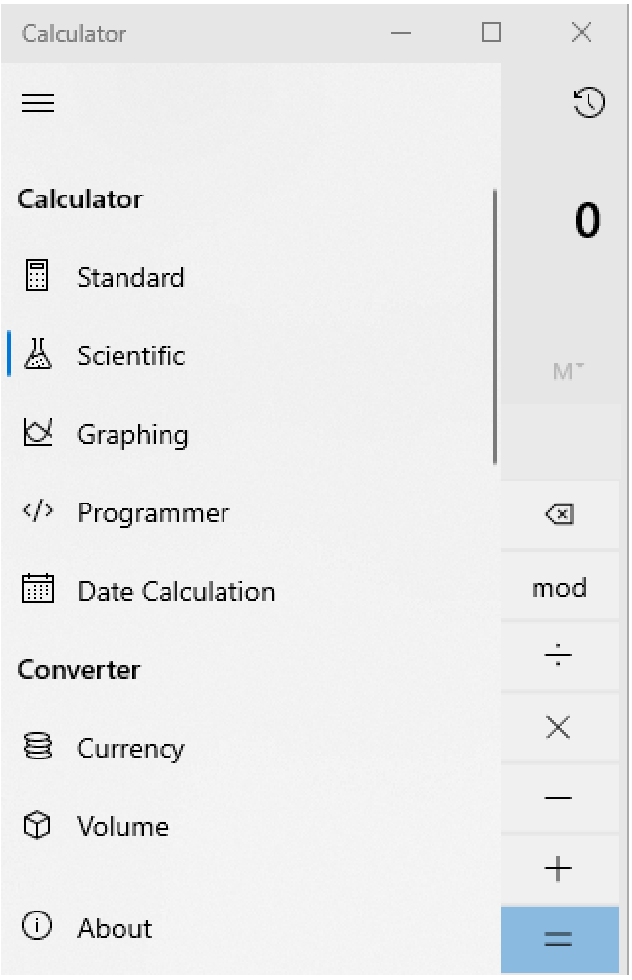
3 Keep on top

4 History

34.3.1.1 View

Displays the possible views you can choose from.

It has 2 categories to choose from: **Calculator** (Standard, Scientific, Graphic, Programmer, Date Calculation) and **Converter** (Currency, Volume, Length, Weight and Mass, Temperature, Energy, Area, Speed, Time, Power, Data, Pressure, Angle).



34.3.1.2 View's name

Displays the current view name.

34.3.1.3 Keep on top

You can also use the keyboard shortcut ALT + Up to access this function.

Note that this function is available only for the Standard view.

34.3.1.4 History

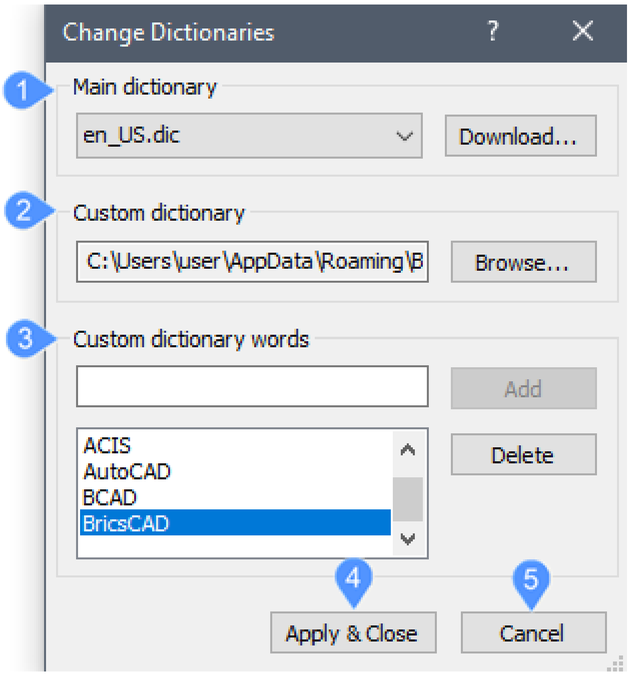
Displays the history of your calculations done in the current session.

Note: This function is available only for the Standard and Scientific calculator.

34.3.2 Change Dictionaries dialog box

Opens via: **Spell Checking** dialog box

The **Change dictionaries** dialog box allows you to change dictionaries.



1 Main dictionary

2 Custom dictionary

3 Custom dictionary words

4 Apply & Close

5 Cancel

34.3.2.1 Help

Opens the Bricsys Help article for the SPELL command.

34.3.2.2 Main dictionary

Specifies the main dictionary to use for spell checking.

File name
Specify the dictionary file to use as the main dictionary. Available files are displayed in the drop-down list.

Download
Download additional dictionaries from OpenOffice.org. Opens the Web browser at: https://extensions.openoffice.org/dictionaries

34.3.2.3 Custom dictionary

Specifies a custom dictionary to use for spell checking. Use custom dictionaries for discipline-specific words, such as medical or mechanical terms.

File location
Specifies the file name and location of a custom dictionary to use in addition to the main dictionary.

Browse
Open the **Select Custom Dictionary** dialog box to select a custom dictionary file.

34.3.2.4 Custom dictionary words

Specifies words to be included in the custom dictionary.

Word field
Allows you to enter a word to be included in the custom dictionary.

Add
Add the word you entered in the word field to the custom dictionary.

Words list
Displays all the words included in the custom dictionary.

Delete
Delete a selected word from the custom dictionary.

34.3.2.5 Apply & Close

Apply your changes and close the dialog box. The custom dictionary file is only updated with added or deleted words after you choose Apply and Close.

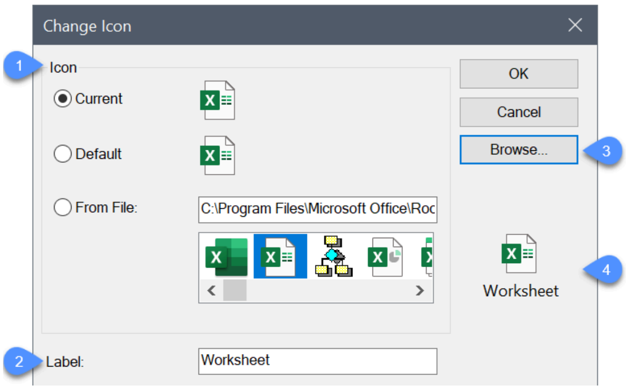
34.3.2.6 Cancel

Close the dialog box without applying changes.

34.3.3 Change Icon dialog box

Opens via: **Insert Object** dialog box

The **Change Icon** dialog box allows you to change the icon of an OLE Frame entity before inserting it into the drawing. The icon is related to the document type and it is defined by the source applications.



1 Icon

2 Label

3 Buttons

4 Preview

34.3.3.1 Icon

Specifies an icon to represent an embedded or linked document.

Current
Displays the current icon.

Default
Displays the default icon.

From File
Choose another icon.

34.3.3.2 Label

Optionally edit the label text.

34.3.3.3 Buttons

The Buttons area contains OK, Cancel and the Browse button.

OK
Click OK to accept the changes and to return to the previous dialog box (Insert Object).

Cancel
Click Cancel to discard the changes and to return to the previous dialog box (Insert Object).

Browse
Displays the **Browse** dialog box (a standard file selection dialog box), in which you can select an icon from a file.

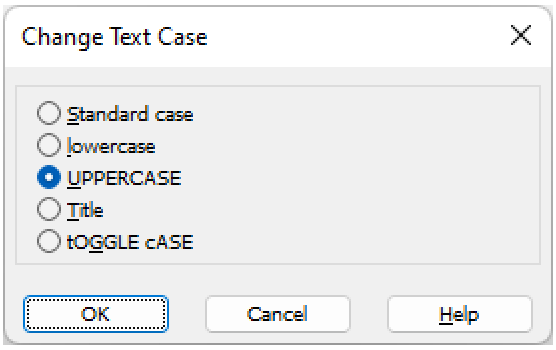
34.3.3.4 Preview

Displays the icon preview.

34.3.4 Change Text Case dialog box

Opens via: TCASE command

The **Change Text Case** dialog box allows you to change the case of selected text entities. You can choose between **Standard case**, **lowercase**, **UPPERCASE**, **Title** and **tOGGLE cASE**.



Standard case
Capitalizes the first letter of the first word.

lowercase
Changes all letters of the selected text to lowercase.

UPPERCASE
Capitalizes all letters of the selected text.

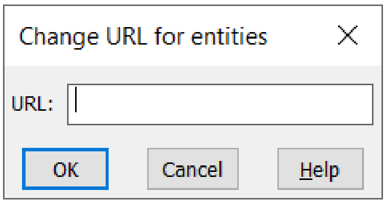
Title
Capitalizes the first letter of each word.

tOGGLE cASE
Changes the first letter of each word to lowercase and capitalizes the rest of the letters.

34.3.5 Change URL for entities dialog box

Opens via: CHURLS command (Express Tools)

The **Change URL for entities** dialog box allows you to change the URL previously attached to entities.

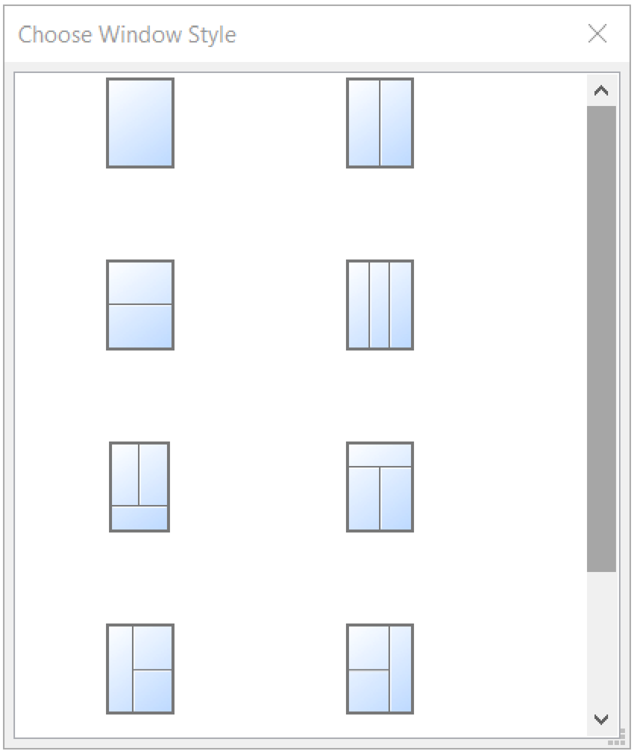


34.3.6 Choose Window Style dialog box

Opens via: BIMWINDOWCREATE command

The **Choose Window Style** dialog box allows you to specify the style of a window.

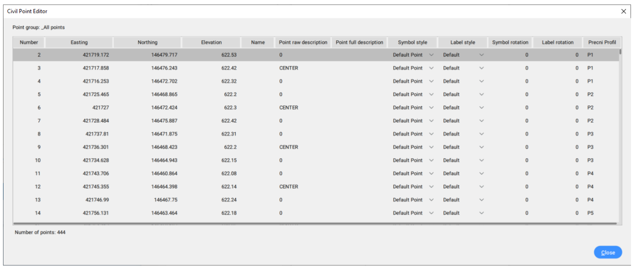
A series of parameters is created, which can be edited in the Parameters section of the Properties panel. Rectangular windows and openings have a W (Width) and H (Height) parameter.



34.3.7 Civil Point Editor dialog box

Opens via: CIVILPOINTEDIT command

The **Civil Point Editor** dialog box displays Civil points and their properties in a table.



Note:

- Click a cell to edit the content.
- It is possible to sort the Civil points by some of their properties.
- The **Civil Point Editor** dialog box is modeless. This means you can work in the drawing while the **Civil Point Editor** dialog box is available on the screen.

Right-click the selected point(s) to open a context menu:

Zoom to
Zooms in to the selected civil point.

Pan to
Pans to the selected civil point in the drawing.

Select
Selects the civil point(s).

Copy to clipboard
Copies the civil point(s) to the clipboard.

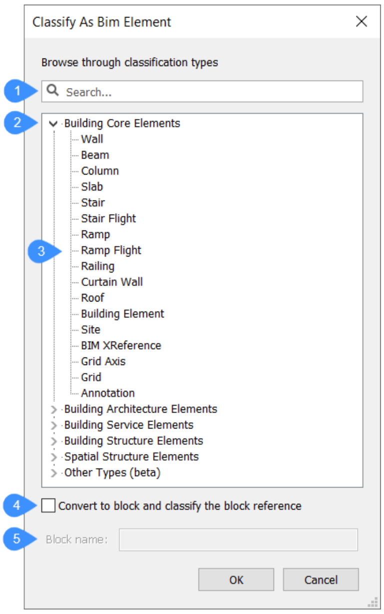
Delete point
Removes the civil point(s).

Note: A warning dialog box opens to specify whether you are sure to delete the points.

34.3.8 Classify As Bim Element dialog box

Opens via: BIMCLASSIFY command

The **Classify As Bim Element** dialog box specifies the types of classification or converts the selection to a block and classifies the block reference.



1 Search bar

2 Bim Element categories

3 Selected category elements

4 Convert to block and classify the block reference

5 Block name

Search bar
Allows you to search for an element in all listed element categories.

Bim Element categories
Displays all available Bim Element categories.

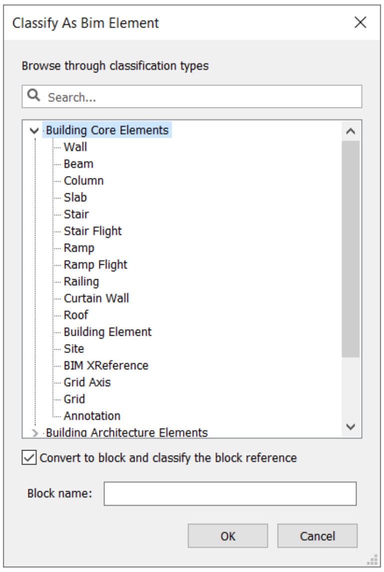
Selected category elements
Displays all the elements available for the selected category.

Convert to block and classify the block reference
Converts selection to block that then classifies as Bim Element. Selecting it enables the **Block name** field.

Block name
Allows you to enter a name for the soon to be created block reference.

Note: All BIM types can be exported and imported to IFC, and can be used in data extraction, filtering (for Typed Plans or Graphic Override).

34.3.8.1 Building Core Elements



Wall

Vertical construction that subdivides spaces.

Example: separation wall, bearing wall, interior wall.

Beam

Structural member that carries loads between or beyond points of support, usually narrow in relation to its length and (nearly) horizontal.

Example: I-beam, L-beam, T-beam, double T-beam.

Column

Structural member of slender form, usually vertical, that transmits to its base the forces, primarily in compression, that are applied to it.

Example: Doric column, Ionic column, Corinthian column.

Slab

Component of the construction that normally encloses a space vertically. The slab may provide the lower support (floor) or upper construction (roof slab) in any space in a building.

Example: floor slab, roof slab.

Stair

Construction comprising a succession of horizontal stages (steps or landings) that make it possible to pass on foot to other levels.

Example: stairs, escalator, spiral staircases interrupted with landings.

Stair Flight

Assembly of building components in a single "run" of stair steps (not interrupted by a landing). The stair steps and any stringers are included in this object.

Example: stairs, escalator, spiral staircases NOT interrupted with landings.

Ramp

Inclined way or floor joining 2 surfaces at different levels.

Example: Slope/Ramp interrupted with landings.

Ramp Flight

Inclined slab segment, normally providing a human circulation link between 2 landings, floors or slabs at different elevations.

Example: Slope/Ramp NOT interrupted with landings.

Railing

The railing is a frame assembly adjacent to human circulation spaces and at some space boundaries where it is used instead of walls or to complement walls. It is designed to aid humans, either as an optional physical support, or to prevent injury by falling.

Example: handrail, balustrade, grab bar, guard rail.

Curtain Wall

Exterior wall of a building which is an assembly of components, hung from the edge of the floor/roof structure rather than bearing on a floor.

Roof

Construction enclosing the building from above.

Example: flat roof, gable roof, gambrel roof, hip roof.

Building Element

The building element comprises all elements that are primarily part of the construction of a building.

Example: structural and space separating system like walls, beams, doors.

Site

A defined area of land, possibly covered with water, on which the project construction is to be completed.

BIM XReference

The identification of information that is not explicitly represented in the current model or in the project database. Such information may be contained in classifications, documents or libraries within the external source.

Grid Axis

An individual axis that is defined in the context of a design grid. The axis definition is based on a curve of dimensionality 2.

Example: X-axis, Y-axis, Z-axis.

Grid

A planar design grid defined in 3D space used as an aid in locating structural and design elements.

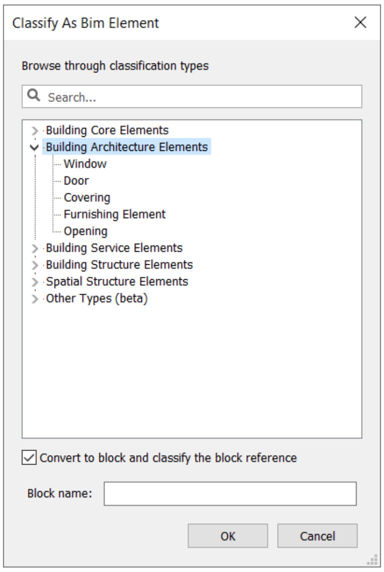
Example: rectangular grid, radial grid, triangular grid.

Annotation

A graphical representation within the geometric context of a project, that adds a note or meaning to the objects of the project.

Example: additional line drawings, text, dimensioning, hatching, ...

34.3.8.2 Building Architecture Elements



Window

Construction for closing a (near) vertical opening in a wall or pitched roof that will admit light and may admit fresh air. The window is a building element that is predominately used to provide natural light and fresh air. A window consists of a lining and one or several panels.

Example: skylights, light domes, swinging, pivoting, sliding, or revolving panels and fixed panels.

Door

Construction for closing an opening. The door is a building element that is predominately used to provide controlled access for people and goods. A door consists of a lining and one or several panels.

Example: hinged, pivoted, sliding, revolving, folding doors.

Covering

A covering is an element which covers some part of another element and is fully dependent on that other element.

Example: wall claddings, floorings, suspended ceilings, finish trim, base molding.

Furnishing Element

Generalization of all furniture related objects.

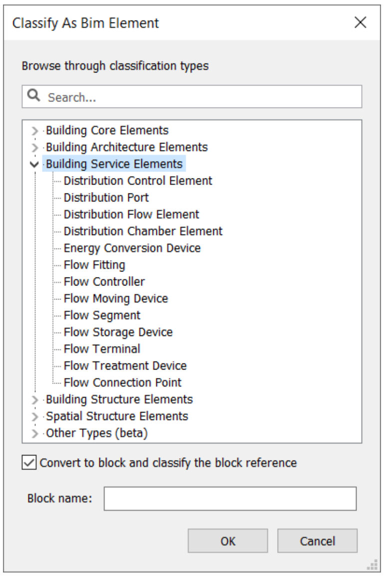
Example: closet, table, chair, bed.

Opening

Represents a void within any element that has physical manifestation.

Example: opening for a door, opening for a window.

34.3.8.3 Building Service Elements



Distribution Control Element

The occurrence elements of a building automation control system that are used to impart control over elements of a distribution system. It senses elements and measures changes in the controlled variable such as temperature, humidity, pressure, or flow.

Example: thermostat

Distribution Port

The occurrence of a specialized port for use within the context of distribution elements.

Example: A gas-powered hot water heater may have 3 ports: GAS, DOMESTICCOLDWATER, and DOMESTICHOTWATER. The heater is a member of 2 systems (GAS and DOMESTICCOLDWATER) and hosts 1 system (DOMESTICHOTWATER) at the corresponding port.

Distribution Flow Element

The occurrence elements of a distribution system that facilitate the distribution of energy or matter, such as air, water or power.

Example: ducts, pipes, wires, fittings

Distribution Chamber Element

A place at which distribution systems and their constituent elements may be inspected or through which they may travel.

Example: sump, trench

Energy Conversion Device

The occurrence of a device used to perform energy conversion or heat transfer and typically participates in a flow distribution system.

Example: boiler, chiller, or a cooling coil

Flow Fitting

The occurrence of a junction or transition in a flow distribution system.

Example: elbow, tee, a junction box in an electrical distribution system

Flow Controller

The occurrence of elements of a distribution system that are used to regulate flow through a distribution system.

Example: damper, valve, switch, relay

Flow Moving Device

The occurrence of an apparatus used to distribute, circulate or perform conveyance of fluids, including liquids and gases, and typically participates in a flow distribution system.

Example: pump, fan

Flow Segment

A flow segment is a section of a distribution system that typically has only 2 ports.

Example: section of a duct, pipe, conduit, etc.

Flow Storage Device

Device used for the temporary storage of a fluid such as a liquid or a gas, the voltage potential induced by the induced electron flow.

Example: tank, battery

Flow Terminal

The occurrence of a permanently attached element that acts as a terminus or beginning of a distribution system. A terminal is typically a point at which a system interfaces with an external environment.

Example: air outlet, drain, water closet, sink

Flow Treatment Device

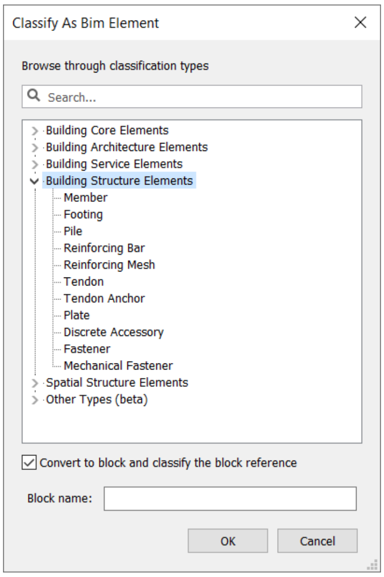
Device used to change the physical properties of the medium, flow treatment types (or the instantiable subtypes) may be exchanged without being already assigned to occurrences.

Example: air, oil or water filter, duct silencer

Flow Connection Point

The connection point is used to describe the geometric constraints that facilitate the physical connection of 2 objects at a point.

34.3.8.4 Building Structure Elements



Member

Structural member designed to carry loads between or beyond points of support. It is not required to be load bearing. The orientation of the member (being horizontal, vertical or sloped) is not relevant to its definition.

Example: diagonal element of a truss construction

Footing
A part of the foundation of a structure that spreads and transmits the load directly to the soil.
Example: wall footing, stepped footing, sloped footing, raft footing

Pile
A slender timber, concrete or steel structural element, driven, jetted or otherwise embedded on end in the ground for the purpose of supporting a load. A pile is also characterized as deep foundation, where the loads are transferred to deeper subsurface layers.
Example: pile foundation

Reinforcing Bar
A steel bar, usually with manufactured deformations in the surface, used in concrete and masonry construction to provide additional strength.
Example: reinforced concrete

Reinforcing Mesh
A series of longitudinal and transverse wires or bars of various gauges, arranged at right angles to each other and welded at all points of intersection.
Example: reinforced concrete floor slab

Tendon
A steel element used to impart pre-stress to concrete when the element is tensioned.
Example: wire, cable, bar, rod, strand

Tendon Anchor
A tendon anchor is the end connection for tendons in pre-stressed or post-tensioned concrete.

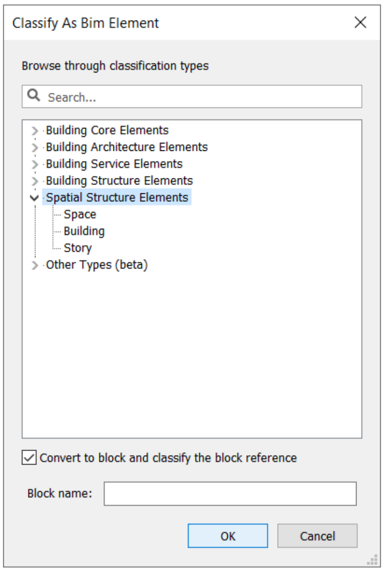
Plate
A Plate is a planar and often flat part with constant thickness. A plate can be a structural part carrying loads between or beyond points of support, however it is not required to be load bearing. The location of the plate (horizontal, vertical or sloped) is not relevant to its definition.

Discrete Accessory
A discrete accessory is a representation of different kinds of accessories included in or added to elements.
Example: visible steel corbel (corbel system made from steel components protruding from the master element), corner fixing plate (fixing plate attached to the corner of an element), steel plate (steel plate used as an accessory in a joint)

Fastener
Representations of fixing parts which are used as fasteners to connect or join elements with other elements.
Example: jointing mortar, glue, weld

Mechanical Fastener
Fasteners connecting building elements mechanically.
Example: bolt, screw, nail, nut, washer, rivet

34.3.8.5 Spatial Structure Elements



Space

BIM Space entities from enclosed boundaries (3D solids or linear 2D entities).

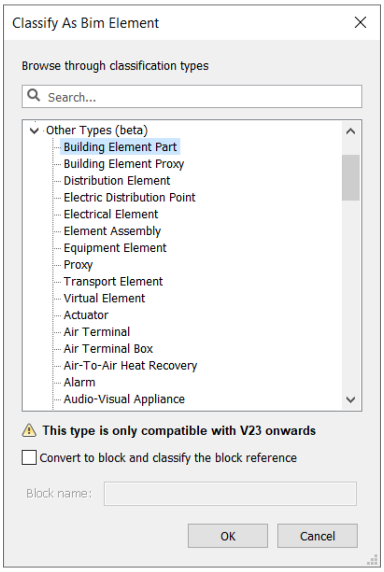
Building

Buildings in the model.

Story

Story of the building / spatial elements.

34.3.8.6 Other Types (beta)



Note: Please note that these BIM types are not backward compatible with older BricsCAD versions. In order to keep your project backward compatible, make sure not to use the new BIM types and use the IFC attributes.

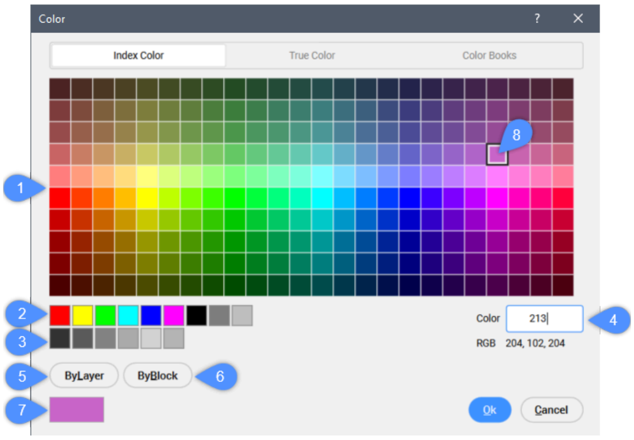
Information about each element type in this category can be found by accessing: https://standards.buildingsmart.org/IFC/RELEASE/IFC4/ADD2_TC1/HTML/annex/annex-b/en/alphabeticalorder_entities.htm

34.3.9 Color dialog box

Opens via: COLOR command

The **Color** dialog box allows you to select a color for layers, grids, dimensions, background, etc. It contains 3 tabs: **Index Color**, **True Color** and **Color Books**.

34.3.9.1 Index Color tab



Color palette (1)
Displays 240 colors.

Basic colors (2)
Displays the basic colors.

Grey colors (3)
Displays shades of gray.

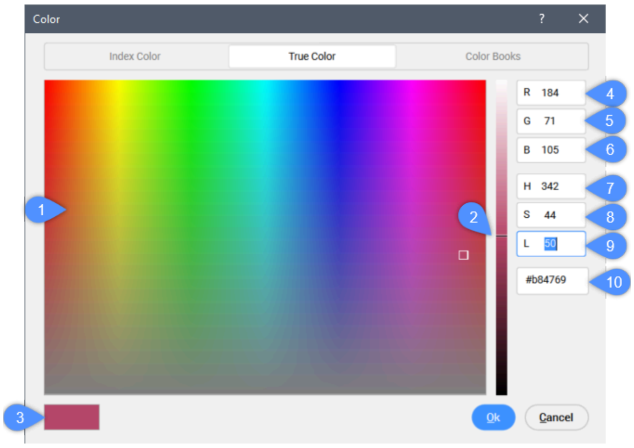
Color index (4)
Displays the index of the selected color. You can insert an index and preview the color in field (7). The corresponding color is marked with a square (8).

ByLayer (5)
Sets the color to ByLayer in which entities adopt the color defined by their layer's property.

ByBlock (6)
Sets the color to ByBlock in which entities adopt the color defined by their block.

Color sample (7)
Previews the selected color.

34.3.9.2 True Color tab



Color (1)
Specifies hue (left and right) and saturation (up and down).

Luminosity bar (2)
Specifies the luminosity of the color.

Color sample (3)
Previews the selected color.

Red (R) - (4)
Specifies the amount of red in the color. It ranges from 0 (no red) to 255 (full red).

Green (G) - (5)
Specifies the amount of green in the color. It ranges from 0 (no green) to 255 (full green).

Blue (B) - (6)
Specifies the amount of blue in the color. It ranges from 0 (no blue) to 255 (full blue).

Note: Hue, saturation and luminosity work together, while red, green and blue work separately.

Hue (H) - (7)
Specifies the color’s hue. It ranges from red to green, blue and pink.
It ranges from 0 to 359.

Saturation (S) - (8)
Specifies the intensity of the color. It ranges from 0 to 100. Saturation is the intensity of the color, where 0 = gray and 100 = full color.

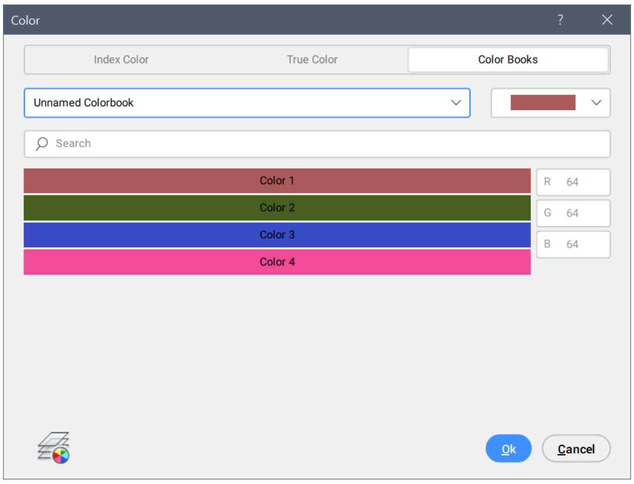
Luminosity (L) - (9)
Specifies the brightness of the color. It ranges from 0 to 100. Luminosity is the lightness or darkness of the color, where 0 = black and 100 = white.

Note: When the current color is white (L=100) or black (L=0) and a true color is picked on the **COLOR** dialog box, the luminosity is set to 50. Also, the picked true color is not synced with the luminosity.

Color code (10)
Displays the selected color code.

34.3.9.3 Color Books tab

Choose a color book that is present on your system.



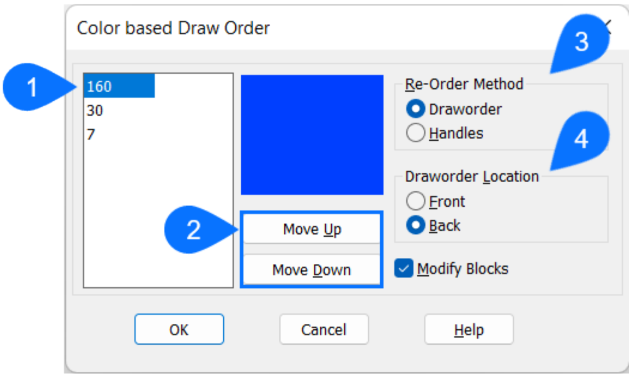
Note:

- The COLORBOOKPATH system variable specifies the folder in which BricsCAD should look for color book files.
- Encrypted color books are supported. The encrypted color books are not included within BricsCAD.

34.3.10 Color based Draw Order dialog box

Opens via: CDORDER command (ExpressTools)

The **Color based Draw Order** dialog box allows you to specify the draw order of the selected entities based on their index color.

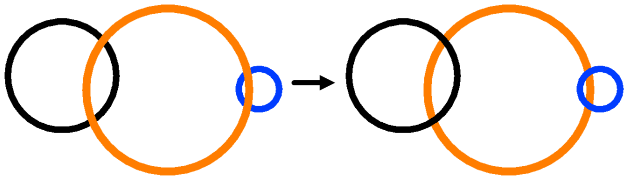


1 Color list

2 Move up and down buttons

3 Re-Order Method

4 Draworder Location



34.3.10.1 Color list

Lists the index colors of the selected entities.

Note: The first color on the list places the entities in the front, while the last color places the entities in the back.

34.3.10.2 Move up and down buttons

Move up
Moves the selected color up in the list.

Move down
Moves the selected color down in the list.

34.3.10.3 Re-Order Method

Draworder
Changes the display order of the selected entities using the DRAWORDER command.

Handles
Changes the display order of the selected entities by reordering entities in the drawing database.

34.3.10.4 Draworder Location

Front
Places the selected entities in front of the entities that are not specified in the color list.

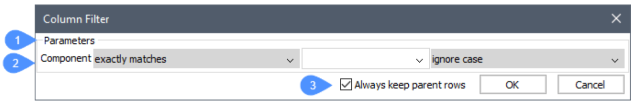
Back
Places the selected entities behind the entities that are not specified in the color list.

Modify Blocks
Changes the display order of the entities in blocks.

34.3.11 Column Filter dialog box

Opens via: **BOM Manager** panel > **Set column filter** option

The **Column Filter** dialog box configures the column filter.



1 Parameters

2 Column name

3 Always keep parent rows

34.3.11.1 Parameters

Defines the parameters to use in the column filter.

34.3.11.2 Column name

Displays the name of the column you put a filter on. Selects the type of filter that should be applied to this column and configures it.

Note: The list of possible filters depends on the column content type.

Note: For all text filters, you can also specify whether they should be case-sensitive or case-insensitive.

Always accepted
All column values are accepted.

Equal to
Value should be equal to the target one.

Note: Appears with numerical columns.

Not equal to
Value should be not equal to the target one.

Note: Appears with numerical columns.

Less than
Value should be less than the target one.

Note: Appears with numerical columns.

Greater than
Value should be greater than the target one.

Note: Appears with numerical columns.

Less than or equal to
Value should be less than or equal to the target one.

Note: Appears with numerical columns.

Between
Value should be in the given range, both ends excluded.

Note: Appears with numerical columns.

Not in range
Value should not be in the given range, both ends excluded.

Note: Appears with numerical columns.

Exactly matches
The content of the column should exactly match the target one.

Note: Appears with columns containing text.

Does not match
The content of the column should be different from the target one.

Note: Appears with columns containing text.

Starts with
The content of the column should start with the given text.

Note: Appears with columns containing text.

Does not start with
The content of the column should not start with the given text.

Note: Appears with columns containing text.

Ends with
The content of the column should end with the given text.

Note: Appears with columns containing text.

Does not end with
The content of the column should not end with the given text.

Note: Appears with columns containing text.

Contains
The content of the column should contain the given text.

Note: Appears with columns containing text.

Does not contain
The content of the column should not contain the given text.

Note: Appears with columns containing text.

Matches pattern
The content of the column should match the given wildcard template.

Note: Appears with columns containing text.

Does not match pattern
The content of the column should not match the given wildcard template.

Note: Appears with columns containing text.

Included in
The content of the column should be in the given set of values.

Note: Appears with columns containing text and numerical columns.

Not included in
The content of the column should not be in the given set of values.

Note: Appears with columns containing text and numerical columns.

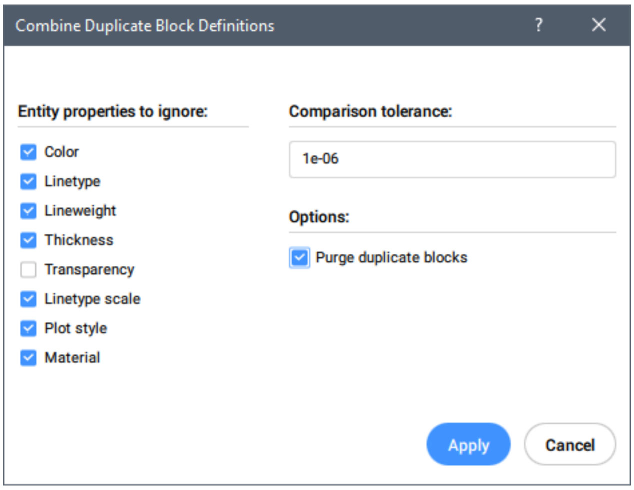
34.3.11.3 Always keep parent rows

Decides whether to keep and display also the parent rows if only the child rows match the filter criteria. If a parent row is removed, then its child rows will be promoted to its level.

34.3.12 Combine Duplicate Block Definitions dialog box

Opens via: OVERKILL command

The **Combine Duplicate Block Definitions** dialog box allows you to search the drawing for duplicate block definitions and replace all such block instances by the most recent of these duplicates. See also the related article **OVERKILL command**.



1 Entity properties to ignore

2 Comparison tolerance

3 Options

34.3.12.1 Entity properties to ignore

Specifies the block properties to ignore when searching for duplicate block definitions.

34.3.12.2 Comparison tolerance

Sets the tolerance for overlapping blocks. If set to 0 (zero), entities must match completely before being evaluated in the overkill process.

34.3.12.3 Options

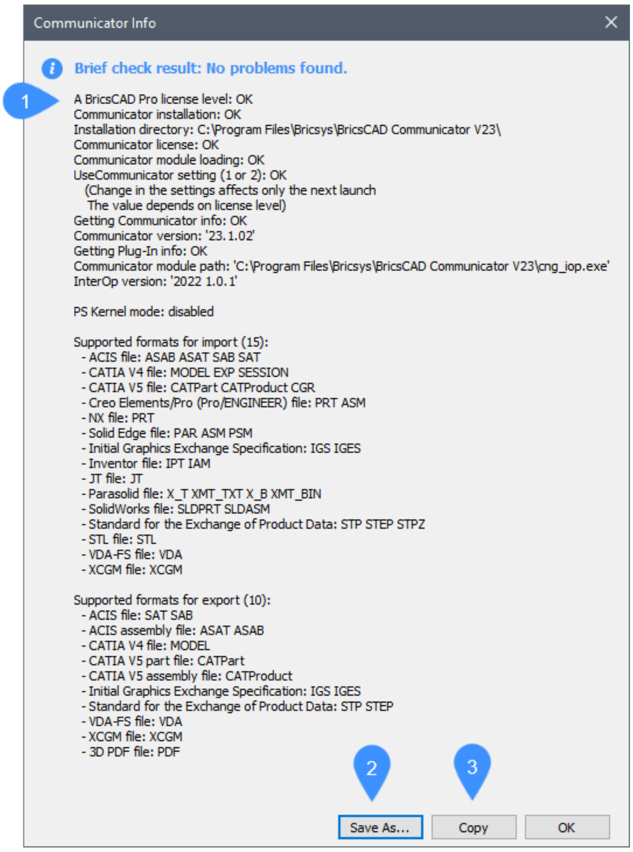
Purge duplicate block definitions
Deletes duplicate block definitions.

34.3.13 Communicator Info dialog box

Opens via: **Bricsys License Manager** dialog box

The **Communicator Info** dialog box checks whether the Communicator for BricsCAD® add-on is correctly installed and provides a diagnostic report.

Note: The command is available on Windows only.



1 Diagnostic

2 Save As...

3 Copy

34.3.13.1 Diagnostic

Displays a diagnostic report for the Communicator for BricsCAD® add-on installation, supported formats to import and to export.

34.3.13.2 Save As...

Opens the **Save Communicator Info As** dialog box.

34.3.13.3 Copy

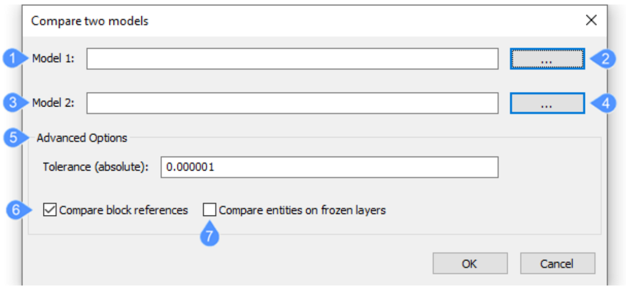
Copies the text displayed in this dialog box to clipboard.

34.3.14 Compare two models dialog box

Opens via: 3DCOMPARE command

The **Compare two models** dialog box allows you to perform a geometric comparison of solids and surfaces between two drawing files.

The result is shown in a new drawing and reported in the **Mechanical Browser** panel.



1 Model 1

2 Select 1^{st} drawing

3 Model 2

4 Select 2^{nd} drawing

5 Advanced Options

6 Compare block reference

7 Compare entities on frozen layers

34.3.14.1 Model 1

Specifies the file name of the first drawing to compare.

34.3.14.2 Select 1^{st} drawing

Opens the Select first drawing for comparison dialog box to choose the drawing you want.

34.3.14.3 Model 2

Specifies the file name of the second drawing to compare.

34.3.14.4 Select 2^{nd} drawing

Opens the **Select second drawing for comparison** dialog box to choose the drawing you want.

34.3.14.5 Advanced Options

The tolerance (absolute) expresses the threshold value for geometry comparison: determines how close two models must be to be considered unchanged.

34.3.14.6 Compare block reference

Determines if the content of blocks should be compared:

- **Yes** - compares solids and surfaces in blocks.
- **No** - ignores solids and surfaces in blocks.

This command works with regular blocks and inserts of arbitrary depth, which contain solids or surfaces, including assemblies that use block structures.

34.3.14.7 Compare entities on frozen layers

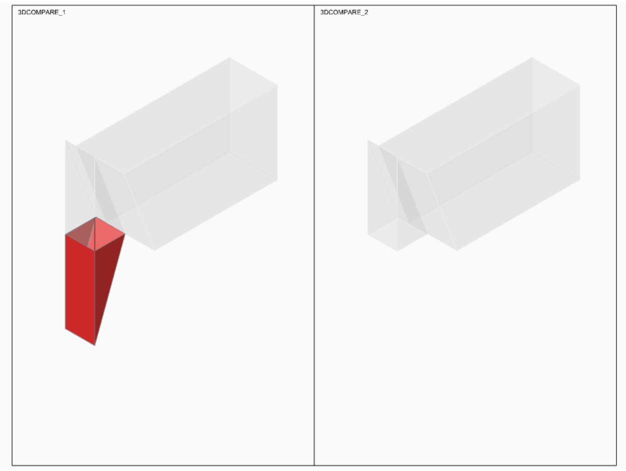
Determines if entities on frozen layers should be compared:

- **Yes** - solids and surfaces on frozen layers are compared.
- **No** - solids and surfaces on frozen layers are ignored.

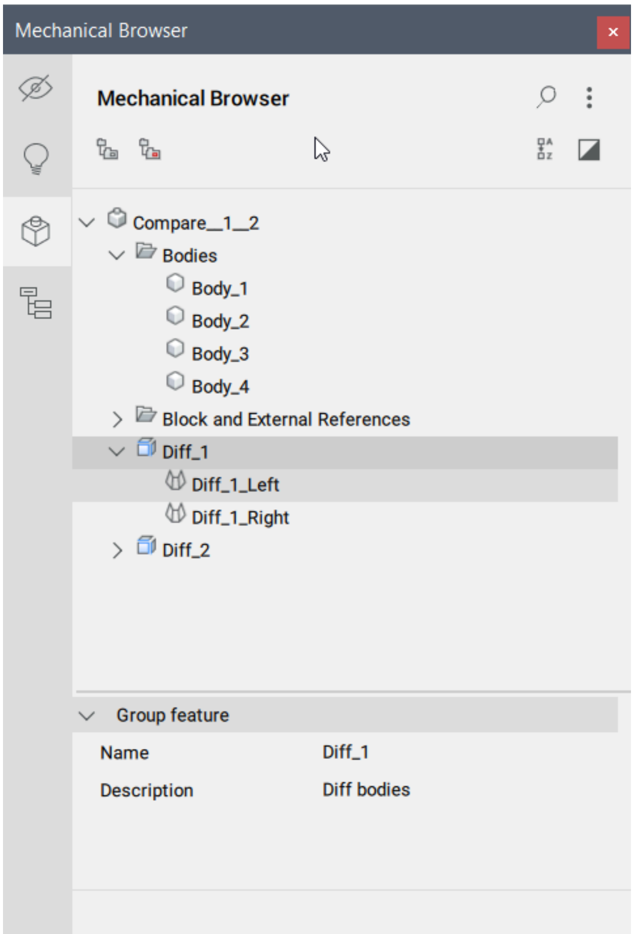
The value of the 3DCOMPAREMODE system variable controls the display in the viewports of the Comparison layout; in the **Settings** dialog box, search for "3dcomparemode".

The selected files are attached to a new drawing that is named Compare_<File_1>_<File_2>.dwg

In this drawing, a layout named “Comparison” is created automatically with two viewports, one for each drawing, illustrating the differences between the drawing using colors.



At the same time, the **Mechanical Browser** panel opens automatically and lists the differences between the two drawings.

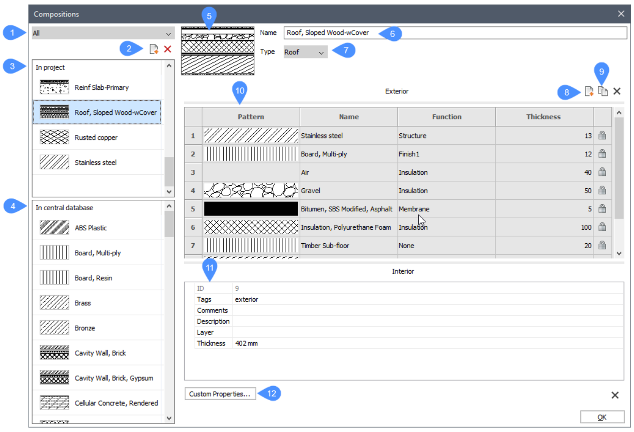


34.3.15 Compositions dialog box

Opens via:

- BLCOMPOSITIONS command
- BIMBEAM command
- BIMCOLUMN command
- BIMROOF command
- BIMSLAB command
- BIMWALL command
- **BIM Compositions** panel

The **Compositions** dialog box allows you to create, modify and delete compositions.



1 Composition category selection

2 New composition

3 In project

4 In central database

5 Preview pane

6 Name

7 Type

8 Add ply

9 Duplicate ply

10 Structure grid

11 Properties grid

12 Custom properties

34.3.15.1 Composition category selection

Composition categories are used to filter the list of the compositions.

Click the drop-down button and choose one of the available categories: **All**, **Generic**, **Roof**, **Slab**, and **Wall**.

34.3.15.2 New composition

Click this button to create a new composition.

34.3.15.3 In project

Displays the compositions of the selected category in the project database.

Right-click a composition to add a new composition, duplicate, delete or copy to the database (library).

Note: Compositions that are used in the project cannot be deleted from the project database.

34.3.15.4 In central database

Displays the compositions of the selected category in the central database.

Right-click a composition to add a new composition, duplicate, delete or copy to a project.

34.3.15.5 Preview pane

Displays the preview of the selected composition.

34.3.15.6 Name

Displays the selected composition name. You can change the default name.

34.3.15.7 Type

Displays the category for the selected composition. You can click the drop-down button and select a new category. The available categories are: **Generic**, **Roof**, **Slab**, and **Wall**.

34.3.15.8 Add ply

Opens the **Physical materials** dialog box which allows you to choose a material from the library or project database.

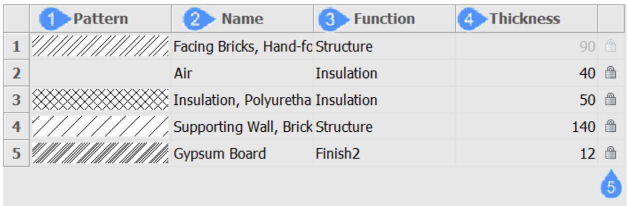
34.3.15.9 Duplicate ply

Inserts a copy of the selected ply.

34.3.15.1 Structure grid
0

Displays the material and thickness of the composition plies. The ply on top (exterior) is applied to the reference of the solid.

The plies of a composition are ordered from Exterior (top) to Interior (bottom). You can drag the ply number on the desired position to change the ply order.



	1 Pattern	2 Name	3 Function	4 Thickness
1		Facing Bricks, Hand-fc	Structure	90
2		Air	Insulation	40
3		Insulation, Polyuretha	Insulation	50
4		Supporting Wall, Brick	Structure	140
5		Gypsum Board	Finish2	12

1 Pattern

2 Name

3 Function

4 Thickness

5 Lock/Unlock thickness

Pattern
Displays a preview of the ply. Double click the **Pattern** field to change the appearance in the **Physical Materials** dialog box.

Name
Displays the name of the ply. Double click the **Name** field to change it in the **Physical Materials** dialog box.

Function
You can select a function from the drop-down list. The available functions are: **None**, **Structure**, **Substrate**, **Insulation**, **Finish1**, **Finish2**, and **Membrane**.

Thickness
Click the **Thickness** field to modify the thickness of a ply. Use the BIMUPDATETHICKNESS command to update the thickness of the solids in the model.

Note: You can only modify the ply thickness if the **Variable Thickness** property in the **Physical Materials** dialog box is set to **Yes**.

Lock/Unlock thickness
You can click the **Lock/Unlock** button to change its status.

Note: Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

- **Fixed**: all plies have locked thicknesses.
- **Minimal**: the composition contains at least two plies and one ply has an unlocked thickness. The minimum thickness is equal to the sum of the thicknesses of the locked plies.
- **Free**: the composition contains a single ply that has an unlocked thickness.

Note: The thickness of a ply can only be unlocked if the **Variable Thickness** property in the **Physical Materials** dialog box is set to **Yes**.

34.3.15.1 Properties grid

1

Displays the properties of the selected composition like tags, comments, description, layer, thickness and custom.

Tags
Select the **Tags** field and type new tags, separated by commas. Tags are case sensitive.
Click the **Browse** button next to the **Tags** field to open the **Tags** dialog box.

Comments
Optionally, you can add comments to the selected composition.

Description
Optionally, you can add a description to the selected composition.

Layer
Optionally, you can assign a layer from the drop-down list to the selected composition.

Thickness
Displays the overall thickness of a composition, which is equal to the sum of the thicknesses of the composing plies.

34.3.15.1 Custom properties

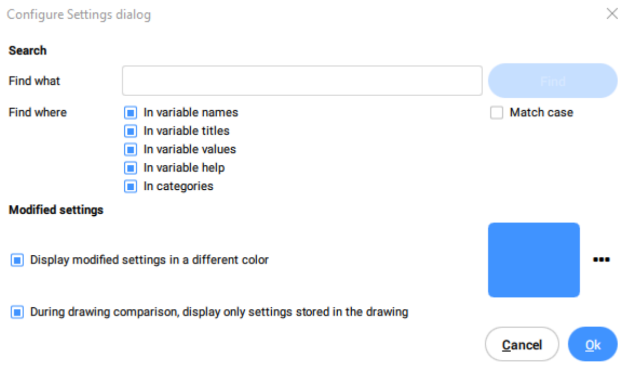
2

Opens the **Select/Create a Custom Property** dialog box. You can add custom properties for compositions.

34.3.16 Configure Settings dialog box

Opens via: **Settings** dialog box

The **Configure Settings** dialog box allows you to control how BricsCAD searches for variables and how the differences are handled.



1 Find what

2 Find

3 Find where

4 Match case

5 Display modified settings in a different color

6 Select color

7 During drawing comparison, display only settings stored in the drawing

34.3.16.1 Find what

Type the name of the setting you are looking for.

34.3.16.2 Find

Searches in the Settings dialog box for the word you have typed.

34.3.16.3 Find where

Determines which part of the Settings dialog box should be searched.

In variable names
Thick this checkbox to search in variable names.

In variable titles
Thick this checkbox to search in variable titles.

In variable values
Thick this checkbox to search in variable values.

In variable help
Thick this checkbox to search in variable help.

In categories
Thick this checkbox to search in categories.

34.3.16.4 Match case

Thick this checkbox to specify that the capitalization of the searched phrase must match that of the words in the **Settings** dialog box.

34.3.16.5 Display modified settings in a different color

Thick this checkbox to see the modified settings in a different color.

34.3.16.6 Select color

Click this button to select another color.

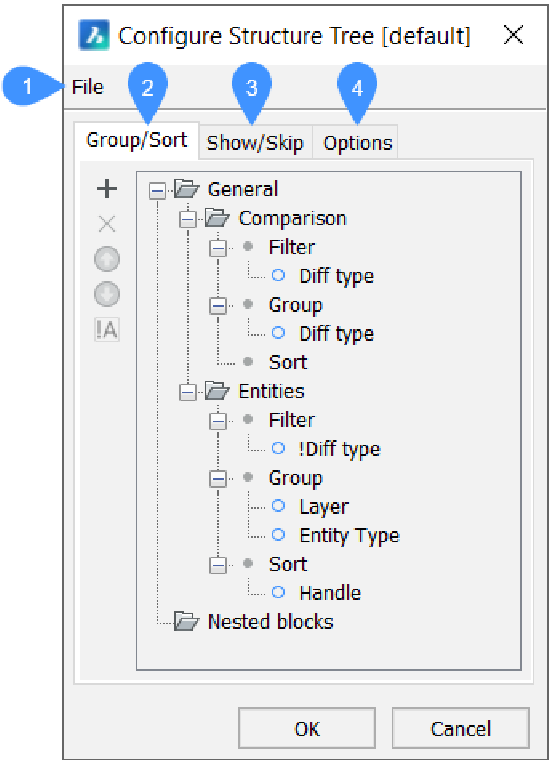
34.3.16.7 During drawing comparison, display only settings stored in the drawing

Thick this checkbox to see the differences for settings stored in the current drawing and another drawing that you open by running the DWGCOMPARE command.

34.3.17 Configure Structure Tree dialog box

Opens via: **Structure** panel > **Configure structure tree** option

The **Configure Structure Tree** dialog box allows you to define the **Structure Tree Configuration** of the **Structure** panel.



1 File

2 Group/Sort tab

3 Show/Skip tab

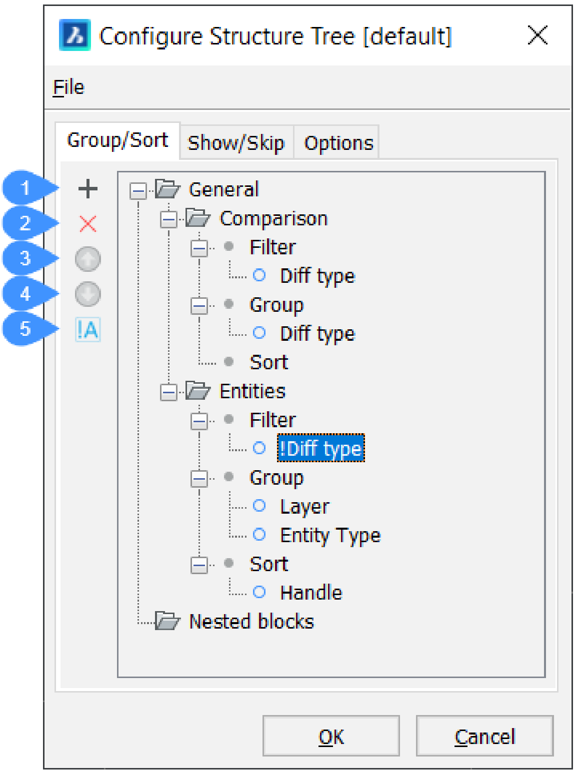
4 Options tab

34.3.17.1 File

Click the **File** menu in the dialog box and choose one of the available options in the list:

- **Open...**: opens the **Select Structure Tree Configuration File** dialog box to select another CST configuration file.
- **Save**: saves the current configuration.
- **Save As...**: saves the current configuration under a different name.

34.3.17.2 Group/Sort Tab



1 Add rule or property

2 Delete rule or property

3 Move property up

4 Move property down

5 Toggle negation (filter properties only)

Add rule or property (+)

Adds a new rule or a new property for **Filter**, **Group**, or **Sort**.

Note: The position of the rules and properties is essential, as the rule structure is hierarchical.

- Select an existing rule, then click the **Add** icon (+) or right-click and choose **Add Rule** in the context menu.

  The new rule is added below the selected rule (or at the end of the rule hierarchy when no rule is selected).

  Note: To rename the rule, select it and click its name or right-click and choose **Rename** in the context menu.

- Select the **Filter** node of the rule, then click the **Add** icon or right-click and choose **Add filter property** in the context menu.
- Select the **Group** node of the rule, then click the **Add** icon or right-click and choose **Add grouping property** in the context menu.
- Select the **Sort** node of the rule, then click the **Add** icon or right-click and choose **Add sorting property** in the context menu.

The **Select Property** dialog box displays to let you choose a property for filtering, grouping, or sorting.

- Select a property, then double-click or click **OK**.

  Note: Properties and value definitions that are defined with the BIMPROPERTIES command can be selected to configure the structure tree.

- Optionally, type a search string in the **Search** field, then click the **Find previous** (**Shift+F3**) ( ) or **Find next** (**F3**) ( ) arrow buttons to look up the properties containing the search string.
- Repeat the previous step to add more filter/ grouping/ sorting properties.

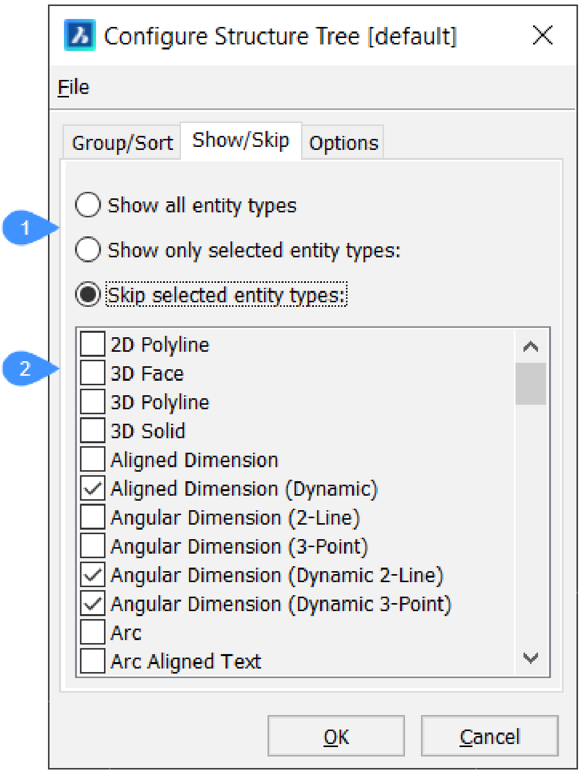
Delete rule or property ( )
Deletes the selected rule or property. You can also use the **Delete rule**/**Delete property** option from the context menu of the selected rule/property.

Move property up ( )
Moves up the selected rule or property.

Move property down ( )
Moves down the selected rule or property.

Toggle negation (filter properties only)
Toggles negation only for filter properties.

34.3.17.3 Show/Skip Tab



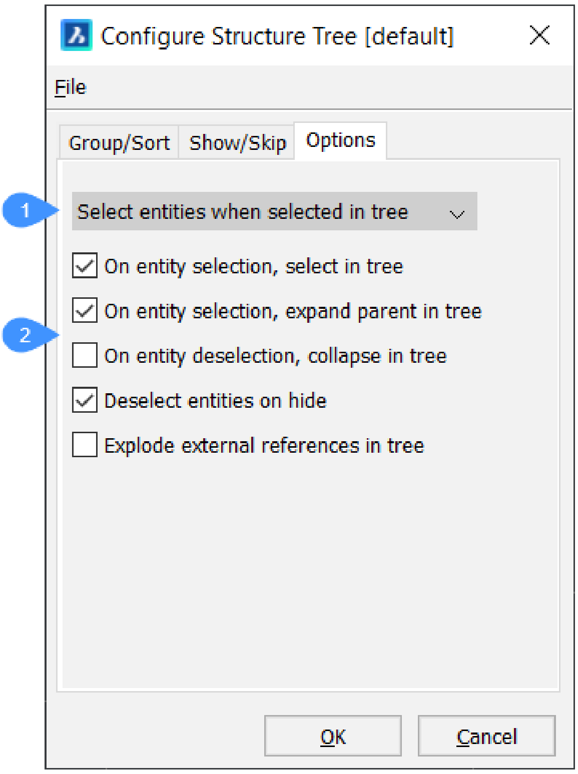
1 Show/skip options

2 Selection list

Show/skip options
Specifies whether selected entities should be shown or not.

Selection list
Allows you to create a selection of entity types to show or exclude.

34.3.17.4 Options Tab



1 Selection options

2 Additional options

Selection options

Defines what happens in the model when entities are selected in the structure tree. Several options are available:

- **Ignore tree selection**: entities are not selected in the drawing.
- **Highlight entities when selected in tree**: entities in the drawing are highlighted, but not selected.
- **Select entities when selected in tree**: matching entities are selected in the drawing.

Additional options

Defines what happens in the structure tree when an entity is selected/deselected in the model.

On entity selection, select in tree

When on, selects matching entity(ies) in the structure tree when you select them in the drawing.

On entity selection, expand parent in tree

When on, expands the node related to the entity(ies) selected in the drawing.

On entity deselection, collapse in tree

When on, collapses the node related to the entity(ies) selected in the drawing.

Deselect entities on hide

When on, deselects hidden entities.

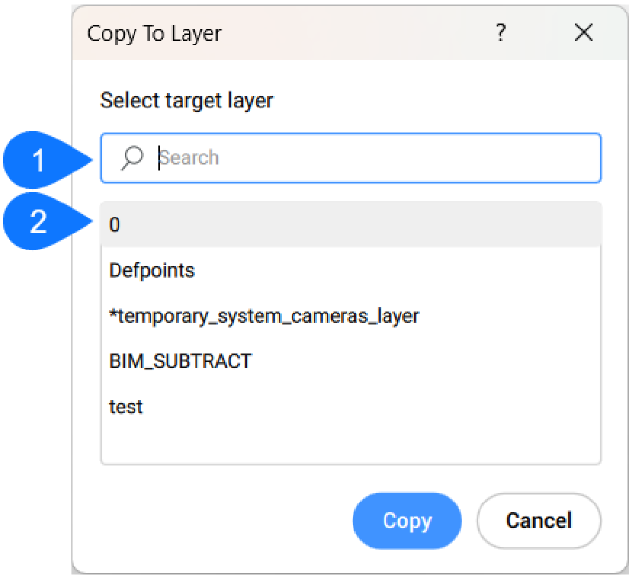
Explode external references in tree

When on, lists the contents of externally-referenced drawings in the structure tree.

34.3.18 Copy To Layer dialog box

Opens via: COPYTOLAYER command (Express Tools)

The **Copy To Layer** dialog box allows you to specify a target layer.



1 Search bar

2 Search report

34.3.18.1 Search bar

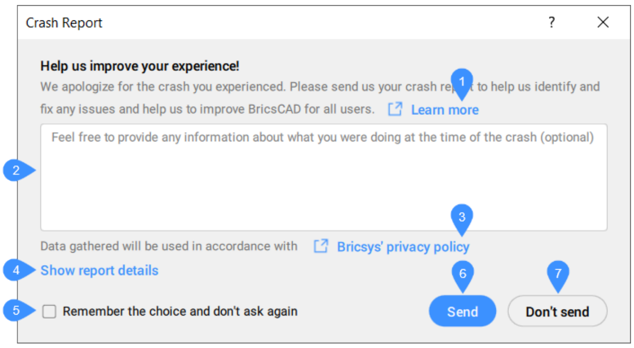
Allows you to search for specific layer names.

34.3.18.2 Search report

Lists all layer names that comply with the entered search.

34.3.19 Crash Report dialog box

The **Crash Report** dialog box allows you to examine and manually submit a crash report. (**Windows only**)



1 Learn more

2 Crash info field

3 Bricsys' privacy policy

4 Show report details

5 Remember the choice and don't ask again

6 Send

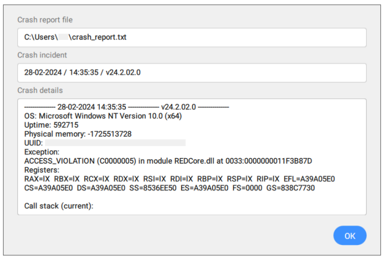
7 Don't send

Learn more
Opens the Help Center's **Sending crash reports via Crash Report dialog box** page.

Crash info field
(Optional) Type in additional information about how the crash occurred.

Bricsys' privacy policy
Opens Bricsys' **Privacy Statement** page.

Show report details
Opens a new dialog box containing information about the automatically generated **crash_report.txt** file. Click **OK** to close the dialog box.



Remember the choice and don't ask again
The **Crash Report** dialog box will no longer be displayed.

Send
Sends the report.

Don't send
Does not send the report.

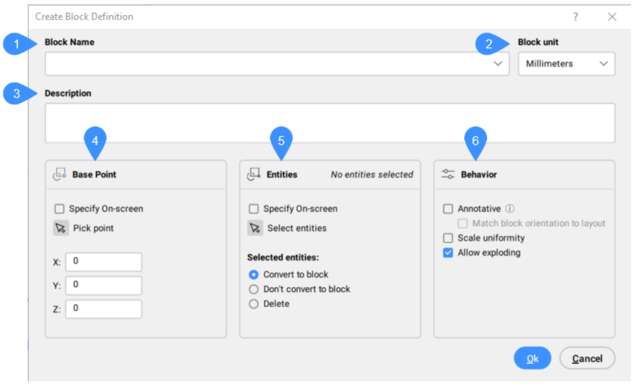
Note:

- The preferences submitted through the **Crash Report** dialog box are stored by the CRASHREPORTSENDING system variable. The default value is 0, for which the dialog box is always displayed. Ticking the **Remember the choice and don't ask again** check-box sets the value of the CRASHREPORTSENDING system variable to 1 if the **Send** option is used, or to 2 if the **Don't send** option is used. For more details, see **Sending crash reports via Crash Report dialog box** article.
- The value of the CRASHREPORTSENDING system variable can be easily changed by typing its name in the Command line and setting the new value, or via the **Settings** dialog box.

34.3.20 Create Block Definition dialog box

Opens via: BLOCK command

The **Create Block Definition** dialog box allows you to create a new block definition.



1 Block Name

2 Block unit

3 Description

4 Base Point

5 Entities

6 Behavior

34.3.20.1 Block Name

Specifies the name of the block.

34.3.20.2 Block unit

Scales the block correctly when the block definition is inserted in a drawing whose units are different from the drawing in which the block was created. More specifically, in which the INSUNITS variable differs.

34.3.20.3 Description

Provides a description of the block (optional).

34.3.20.4 Base Point

Modifies the properties of the base point of the block definition. At this point, the block is inserted with the INSERT command.

Specify On-screen
Allows you to define the base point in the model space after you click the **OK** button, if the box is ticked.

Pick point
Allows you to pick a point in the drawing for the block's base point. You can also enter the X,Y,Z coordinates in the drawing.

X/Y/Z
Defines the coordinates where the block should be inserted. The Z coordinate is usually optional.

34.3.20.5 Entities

Selects the entities that make up the block.

Specify On-screen
Allows you to select the entities in the model space after you click **OK**, if the box is ticked.

Note: Optionally, include Reference Curves in the selection, which allows to automatically align a block during insertion.

Select entities
Selects one or more entities in the drawing.

Selected entities
Defines how the entities are turned into a block.

Convert to block
Selected entities are converted to a block. This is the default option.

Don't convert to block
Selected entities are retained as entities, and the block definition is created in the drawing.

Delete
Selected entities are deleted after the block definition is created.

Note: The **Convert to block** option is the most efficient one.

34.3.20.6 Behavior

Annotative
Sets the annotative property of the block. This type of block should be created when the annotation scale in model or paper space is 1:1. By being annotative, the block automatically scales itself according to the current annotation scale factor.
Choose whether you want the block to follow annotative scaling:

- **On**: the block will scale itself to whatever annotative scale is in effect. The Scale Uniformly option is grayed out (unavailable).
- **Off**: the block follows the scale factor you give the block during the INSERT command.

Note: You can modify supported annotation scales of a selected block attribute in the **Properties** panel, even when its owning block reference is not annotative.

Match block orientation to layout
Determines if annotative blocks match the orientation of the layout.

- **On**: annotative blocks display upright, regardless of the orientation of the viewport.
- **Off**: annotative blocks match the orientation of the viewport.

Scale uniformity
Determines if blocks can be scaled non-uniformly. This option is not available to annotatively-scaled blocks.

- **On**: X, Y, and Z scale factors of blocks are the same. This prevents blocks from being distorted.
- **Off**: blocks can be inserted with different X, Y, and Z scale factors. This is useful for objects that can have different dimensions, such as differently sized table tops.

Allow exploding
Determines if users can explode the block after it is inserted. When a block is exploded, it loses its block status, and the individual entities can be edited.

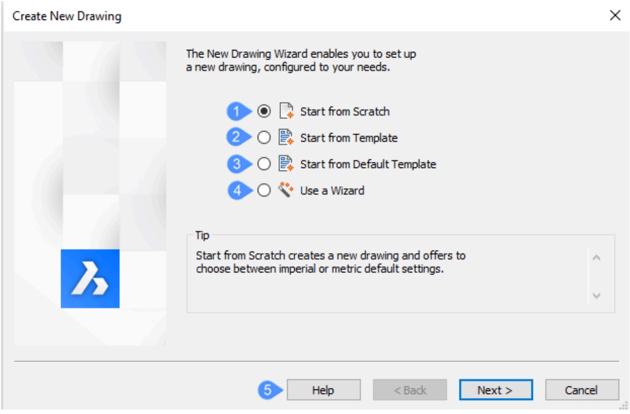
Tip: To edit the entities of an unexploded block, use the BEDIT command.

- **On**: blocks can be exploded after being inserted, using the EXPLODE command.
- **Off**: blocks cannot be exploded. You can change this property with the EXPBLOCKS command, Blocks section.

34.3.21 Create New Drawing dialog box

Opens via: NEWWIZ command

The **Create New Drawing** dialog box allows you to create new drawings by several methods with the New Drawing Wizard.



1 Start from Scratch

2 Start from Template

3 Start from Default Template

4 Use a Wizard

5 Help

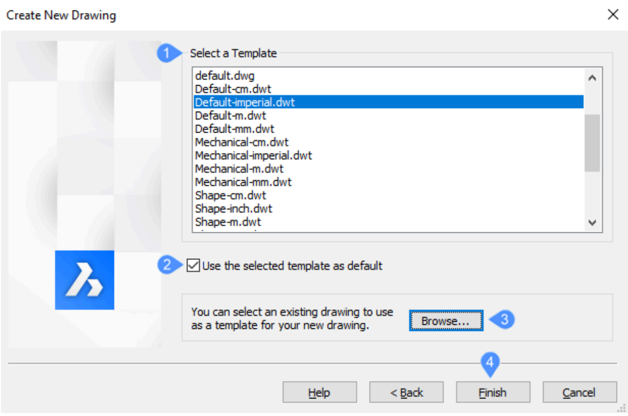
34.3.21.1 Start from Scratch

Starts the new drawing using the default template file specified by the BASEFILE system variable. Click the **Next** button and choose a measurement system:

- **Imperial (feet and inches)** - new drawing is based on the default-imperial.dwt file and so uses Imperial measurements.
- **Metric** - new drawing is based on the default-mm.dwt file and so uses metric measurements.

After you click the **Finish** button, the program opens a new drawing whose content depends on the default template file.

34.3.21.2 Start from Template



1 Select a Template

2 Use default template

3 Browse...

4 Finish

Select a Template
Allows you to choose a file to act as the template. From the Template list, choose one of the DWT file names. These files are stored in the folder specified by the TemplatePath variable; the default location is *C:\Users\%username%\AppData\Local\Bricsys\BricsCAD\V25x64\en_US\Templates*.

Use default template
Turn on **Use the selected template as default** option when you want to use this file as the template from now on. If so, then the program updates the value of the BASEFILE system variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

Browse...
Opens the **Select a template file** dialog box from where you can select the following kind of file type for a template: DWG - drawing files, DWT - template files, DXF - drawing interchange files.

Finish
Click the **Finish** button after selecting a Template from the list or from the dialog box.

34.3.21.3 Start from Default Template

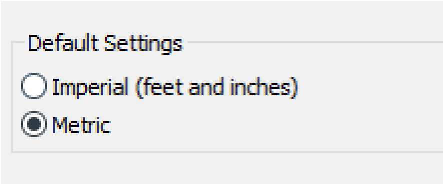
Starts new drawings with the default template defined by the BASEFILE system variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

34.3.21.4 Use a Wizard

Starts the new drawing by stepping through a wizard that guides you along the way:

1 Click the **Next** button.

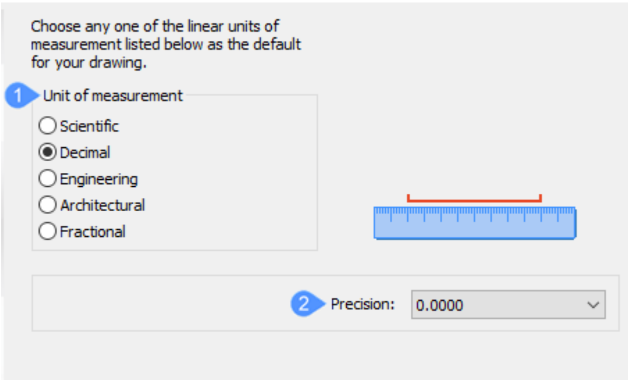
2 Choose a measurement system:



- Imperial (feet and inches) - new drawing is based on the default-imperial.dwt file and so uses Imperial measurements.
- Metric - new drawing is based on the default-mm.dwt file and so uses metric measurements.

3 Click the **Next** button.

4 Select a linear unit of measurement and display precision.



- Unit of Measurement
- Precision

Unit of Measurement

Choose one from the list. If you are unsure, choose Decimal; you can always change the units any time later with the UNITS command.

Unit Name	Measures	Example Display
Architectural	Feet, inches, fractional inches	4'-6 1/16"
Engineering	Feet, inches, decimal inches	4'-6.0625"
Fractional	Inches, fractional inches	54 1/6"
Decimal	(default) Units and decimal units	2128.4449
Scientific	Units, decimal units, exponents	2.1284E+03

Precision

Click the drop-down list and choose a level of precision. If you are unsure, choose the defaults, as you can always change the precision any time later with the UNITS command.

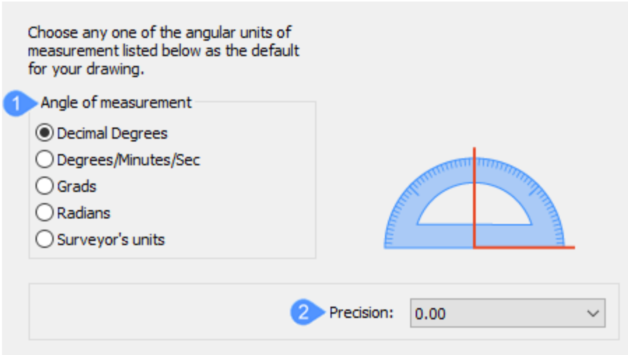
Note: This is a display precision: the program makes calculations internally always to eight decimal places.

Note:

1 Click the **Next** button.

2 Select an angular unit of measurement and display precision.

3



- Angle of measurement
- Precision

Angle of Measurement

Choose one from the list. If you are unsure, choose Decimal/Degrees; you can always change the units any time later with the UNITS command.

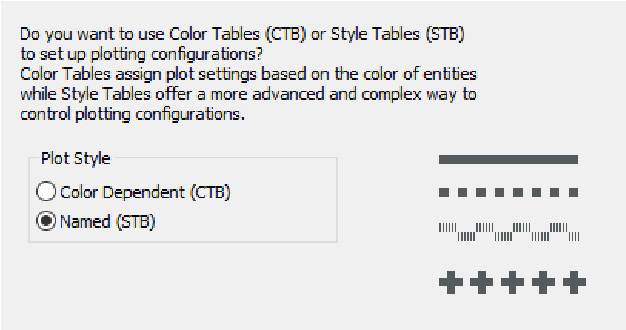
Angle Name	Measures In	Example
Decimal degrees	Degrees and decimals	12.3456
Degrees/Minutes/Seconds	Degrees,minutes,seconds,decimal secs	12d34'56"
Grads (g)	Grads and decimals	123.45g
Radians (r)	Radians and decimals	1.23r
Surveyor's Units	North/South degrees towards East/West	N12d34'56"E

Precision

Click the drop-down list and choose a level of precision. If you are unsure, choose the defaults, as you can always change the precision any time later with the UNITS command.

1 Click the **Next** button.

2 Determine which kinds of plot style the drawing should employ:

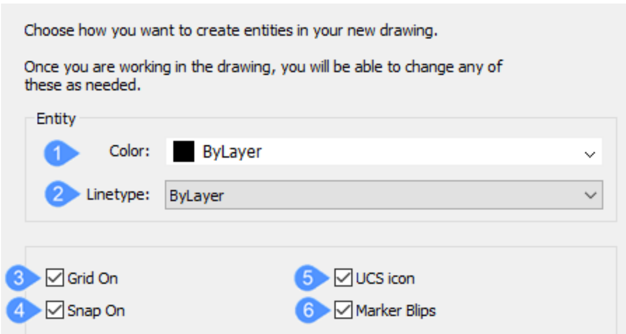
3 

- **Color Dependent (CTB)** - these plot styles are simpler to employ, but less flexible.
- **Named (STB)** - plot styles are more flexible, but complex, because they control every aspect of every entity property during plotting.

If you are unsure, choose Color-dependent.

4 Click the **Next** button.

5 Choose the default working color, the linetype and the visual aids you want to be displayed by the drawing:

6 

a Color
b Linetype
c Grid On
d Snap On
e UCS icon
f Marker Blips

Color
You can choose any color from the drop-down list. However, this is not recommended. Choose ByLayer so that layers control the colors of entities. See COLOR and LAYER commands.

Linetype
You can choose any linetype from the drop-down list, this again is not recommended. Choose ByLayer, for the same reason. See LINETYPE command.

Grid On
When on, displays a grid of dots (or lines) that helps you see the unit spacing, as well as drawing extents, such as every 1m; see the GRID command. It is recommended you turn this on.

Snap On
When on, sets the movement increment of the cursor, such as every 0.5m; see SNAP command. It is recommended you turn this on.

UCS icon
When on, displays the UCS icon for showing the orientation of the x, y, and z axes; see the UCSICON command. It is recommended that you turn this on for 3D drawings, but turn it off for 2D ones.

Marker Blips
When on, draws a tiny + each time you click in the drawing; see the BLIPMODE command. It is recommended you leave this one off, as it can turn the screen messy.
After you click the **Finish** button, the program opens a new drawing whose content depends on the default template file and the options you choose during the wizard.

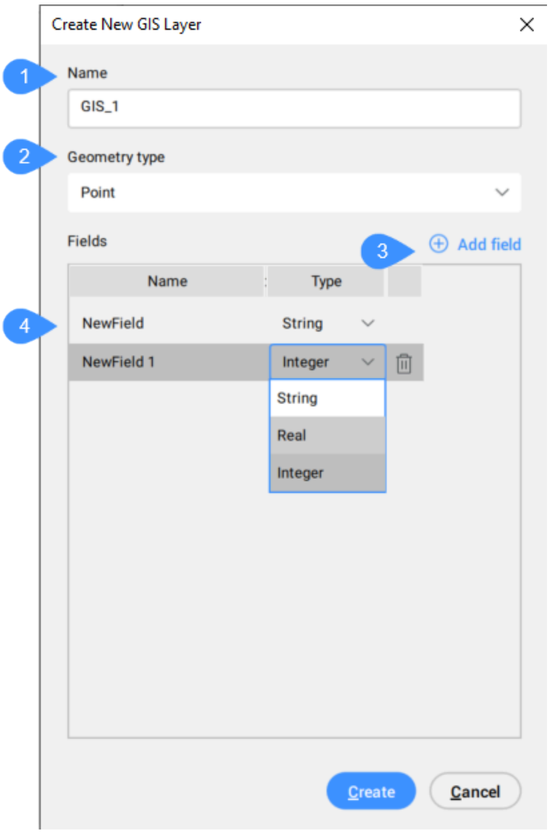
34.3.21.5 Help

Opens the Bricsys Help article about the NEWWIZ command.

34.3.22 Create New GIS Layer dialog box

Opens via: **Civil Explorer** panel

The **Create New GIS Layer** dialog box allows you to create a new GIS layer.



1 Name

2 Geometry type

3 Add field

4 Fields

34.3.22.1 Name

Displays the name chosen for the GIS layer.

34.3.22.2 Geometry type

Displays the geometry type chosen from the geometry types available list, which opens by clicking the right-side arrow.

34.3.22.3 Add field

Adds a new GIS Layer field.

34.3.22.4 Fields

Lists all GIS Layer fields added. Each field has a **Name** and **Type**.

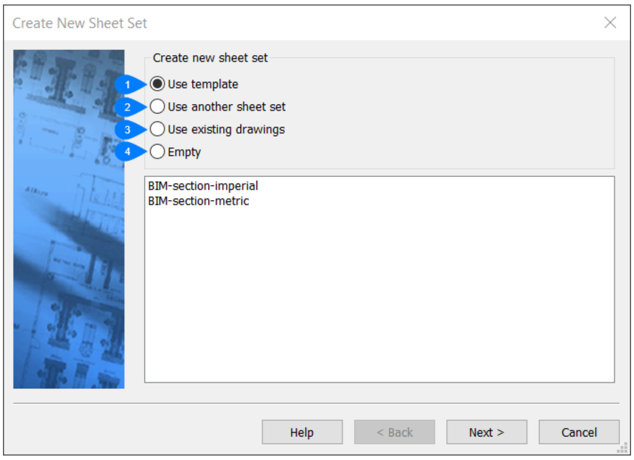
Click the trashcan icon to delete GIS Layer fields.

34.3.23 Create New Sheet Set dialog box

Opens via:

- NEWSHEETSET command
- **Sheet Sets** panel

The **Create New Sheet Set** dialog box provides 4 methods to create a new sheet set.



1 Use template

2 Use another sheet set

3 Use existing drawings

4 Empty

34.3.23.1 Use template

Use template
Creates a new sheet set from a template.

Sheet set templates are stored as DST files in the folder set by the SheetSetTemplatePath preference (default path *C:\Users\%username%\AppData\Local\Bricsys\BricsCAD\V[Version]\en_US\Templates*)

1 Select a template sheet set from the list shown in the dialog box.

1 ***Note:*** All metric projects use by default a sheet set template in mm, and will honor the setting of the SECTIONSHEETSETTEMPLATEMETRIC system variable, if not empty. The cm and meter sheet set and sheet template files are no longer included in the installation.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

Note: By default, the current drawing's location folder is open.

Description

1 (Optional) Type a description for the sheet set file.

2 Click **Next**.

Sheet Set Preview

Review the summary of the sheet set that will be created, then click **Finish** to create the new sheet set file. The program opens the **Sheet Set Manager** panel with the new sheet set.

34.3.23.2 Use another sheet set

Creates a new sheet set from the settings in an existing sheet set.

Select sheetset to copy

1 Click **Browse** and select an existing sheetset file from the **Select a Sheet Set file** dialog box.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder for the sheet set and save it.

Important: You cannot progress until you have selected a folder that stores the new file.

Description

1 (Optional) Type a description for the sheet set file.

2 Click **Next**.

Sheet Set Preview

Review the summary of the sheet set that will be created, then click **Finish** to create the new sheet set file. The program opens the **Sheet Set** panel with the new sheet set.

34.3.23.3 Use existing drawings

Creates a new sheet set from an existing set of drawings.

1 Select **Use existing drawings**.

2 Click **Next**.

Select Folder

1 Select a folder from the **Choose a folder** dialog box where the new sheet set will be stored.

Note: The dialog box fills with names of drawings and their layouts.

2 Choose the options (it is recommended to keep both options turned on).

Create subsets from folder structure

Uses the folders in the main folder to determine how the sheet set is structured.

Prefix sheet titles with drawing file names

1 Add the drawing file's name to the start of the sheet set titles. This makes it easier to track the source of the files.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder for the sheet set and save it.

Important: You can not progress until you select a folder.

Description

(Optional) Type a description for the sheet set file.

Default drawing template for new sheets

1 Select a template sheet set from the **Select Drawing Template** dialog box.

2 Click **Next**.

Sheet Set Preview

Review the summary of the sheet set that will be created, then click **Finish** to create the new sheet set file. The program opens the **Sheet Set** panel with the new sheet set.

34.3.23.4 Empty

Creates a new sheet set from scratch.

1 Select **Empty**.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder for the sheet set and save it.

Important: You can not progress until you select a folder.

Description

(Optional) Type a description for the sheet set file.

Default drawing template for new sheets

1 Select a template sheet set from the *Select Drawing Template* dialog box.

2 Click **Next**.

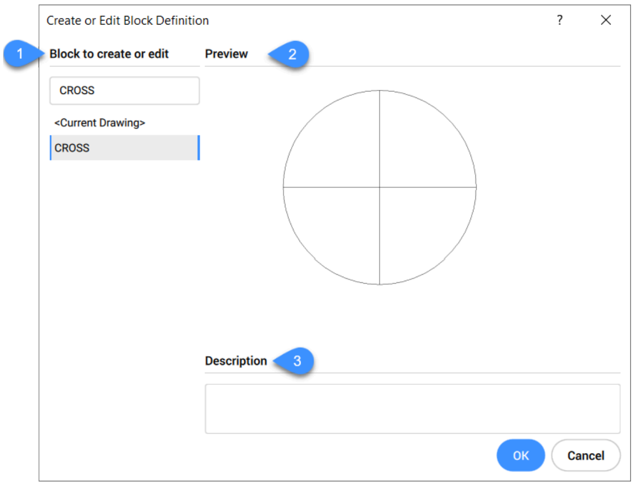
Sheet Set Preview

Review the summary of the sheet set that will be created, then click **Finish** to create the new sheet set file. The program opens the **Sheet Set** panel with the new sheet set.

34.3.24 Create or Edit Block Definition dialog box

Opens via: BEDIT command

The **Create or Edit Block Definition** dialog box allows you to select an existing block definition to edit, or to enter a new name to create a new block definition.



1 Block to create or edit

2 Preview

3 Description

34.3.24.1 Block to create or edit

Specifies the name of the block you want to create or edit. To edit a block, you can select a block from the list below with the blocks available in the drawing.

34.3.24.2 Preview

Provides a preview of the selected block.

34.3.24.3 Description

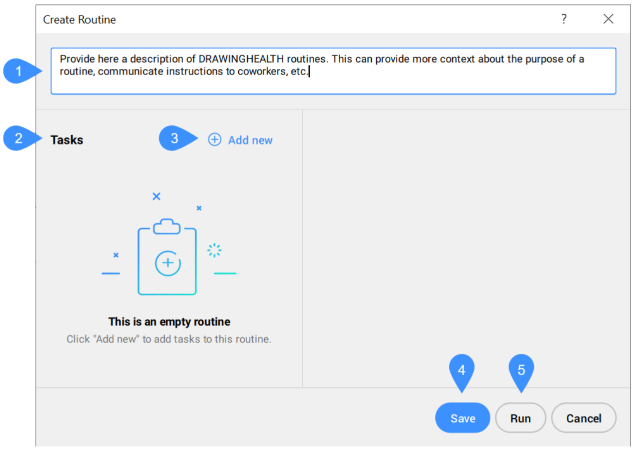
Displays the description of the selected block.

Click **OK**, to open the **Block Editor**.

34.3.25 Create Routine dialog box

Opens via: **Drawing Health** command context panel

The **Create Routine** dialog box allows you to define the tasks to be performed within a routine during the DWGHEALTH command.



1 Description

2 Tasks

3 Add new

4 Save

5 Run

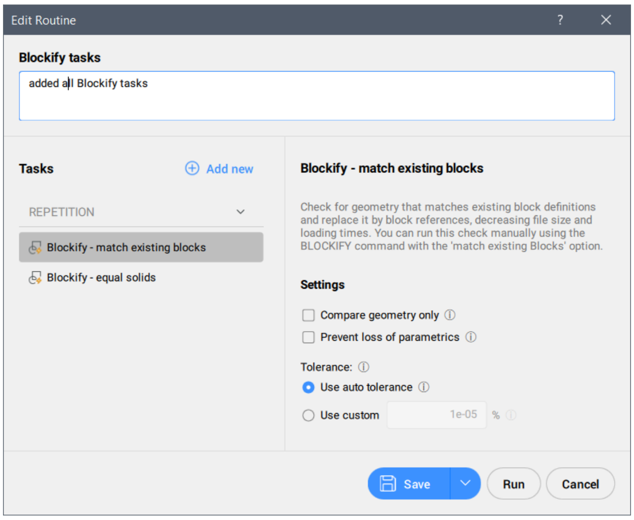
34.3.25.1 Description

Allows you to add a description of the DRAWINGHEALTH routines. This can provide more context about the purpose of a routine, communicate instructions to coworkers, etc.

34.3.25.2 Tasks

Lists the tasks selected for the current routine to be performed during the DWGHEALTH command.

Note: A working routine requires at least one task.



34.3.25.3 Add new

Adds new tasks to the current routine from the drop-down list that contains all available tasks.

Available tasks are grouped as follows:

BASIC

Purge
Checks and removes various types of unused objects, shown in the below picture. You can decide the object types that will be included in the check. You can run this check manually using the PURGE command.

Audit
Analyzes the integrity of the current drawing and fixes the errors. You can run this check manually using the AUDIT command.

Find Outliers
Finds any outlier outside of the valid area in order to move or remove them. All found outliers will be removed when running the routine in Express mode. You can run this check manually using the FINDOUTLIERS command.

2D GEOMETRY

Simplify
Simplifies or smoothens polylines, splines and hatches by removing redundant vertices, accordingly with the user's preferences. You can run this check manually using the SIMPLIFY command.

3D GEOMETRY

Audit 3D
Analyzes and automatically fix the problems in 3D geometry, accordingly with the user's preferences. You can run this check manually using the DMAUDIT command.

Solidify
Converts meshes to solids. You can run this check manually using the SOLIDIFY command.

Simplify 3D
Simplifies 3d solid entities by removing unnecessary edges and vertices, merging seam edges, and replacing the geometry of faces and edges with analytic surfaces and curves within the use-specified tolerance. You can run this check manually using the DMSIMPLIFY and DMSIMPLIFYALL commands.

REPETITION

Overkill - duplicate geometry
Deletes duplicate entities. Optionally, move duplicates to a dedicated layer. You can run this check manually using the OVERKILL command with the **Combine duplicate block definitions** option.

Overkill - duplicate blocks
Checks the block definitions with identical contents, and (optionally) removes duplicates, according to the user's preferences. You can run this check manually using the OVERKILL command with the **Combine duplicate block definitions** option - **Purge duplicate blocks** button.

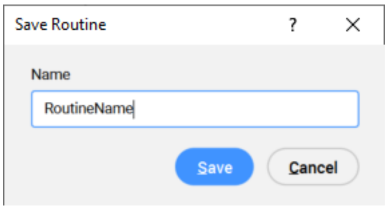
Blockify - match existing blocks
Checks for geometry that matches existing block definitions and replaces it with block references, decreasing file size and loading times, according to the user's preferences. You can run this check manually using the BLOCKIFY command with the **match existing Blocks** option.

Blockify - equal solids
Replaces equal solids with block references to add structure to the drawing and decreases file size and loading times, according to the user's preferences. You can run this check manually using the BLOCKIFY command with the **Equal Solids** option.

Array Detect
Adds structure to the drawing and reduces file size by replacing sets of entities by arrays. You can run this check manually using the ARRAYDETECT command.

34.3.25.4 Save

Opens the **Save Routine** dialog box. Type in a name for the created routine.



34.3.25.5 Run

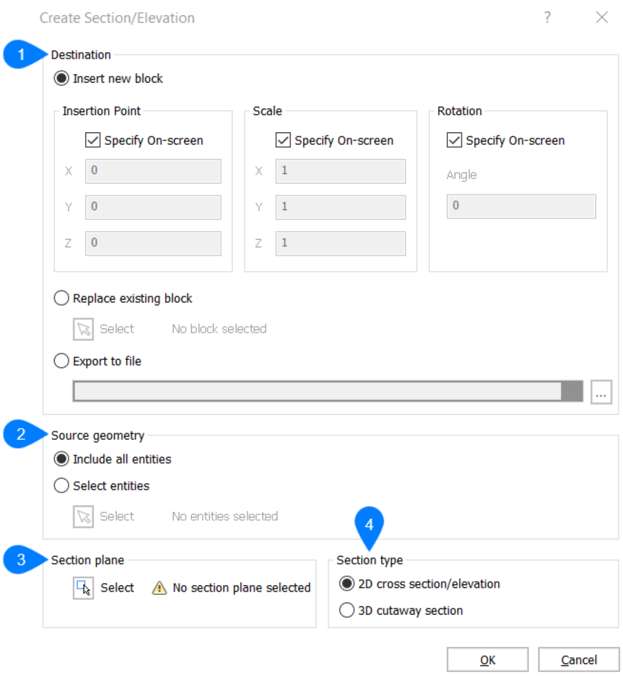
Allows you to run the created routine under the **New Routine*** name, without saving it.

Note: The **Drawing Health** command context panel displays a **Save as** button after you run an unsaved routine via the **Run** button in the **Create Routine** dialog box or **Edit Routine** dialog box. Click it to open the **Save Routine** dialog box and save the custom routine with the desired name.

34.3.26 Create Section/Elevation dialog box

Opens via: SECTIONPLANETOBLOCK command

The **Create Section/Elevation** dialog box saves section planes as blocks inserted in the current drawing or saved to disk as DWG files.



1 Destination

2 Source geometry

3 Section plane

4 Section type

34.3.26.1 Destination

Defines where the block will be placed in the drawing.

Insert new block

Inserts the created section as a new block in the drawing. The insertion point, scale and rotation can be specified.

Insertion point

Allows to specify the insertion point in the model space after you click the OK button, if the Specify on-screen button is checked.

X/Y/Z

Defines the coordinates where the block should be inserted, only available if the Specify on-screen button is checked off.

Scale

Allows to specify the scale factor in the model space after you click the OK button, if the Specify on-screen button is checked.

X/Y/Z

Defines the individual X,Y,Z direction scale factors, only available if the Specify on-screen button is checked off.

Rotation
Allows to specify the rotation angle in the model space after you click the OK button, if the Specify on-screen button is checked.

Angle
Defines the rotation angle of the inserted section/elevation.

Replace existing block
Replaces an existing block in the drawing. This block will be replaced by the created section

Select
Allows to select the block to replace in the model space.

Export to file
Opens the Select export file dialog box. Here you can select a file to export the created section to.

34.3.26.2 Source geometry

Specifies the entities that will be included from the drawing.

Include all entities
Selects all 3D geometry in the drawing (includes 3D solids, 3D surfaces, polyface meshes, and 3D faces).

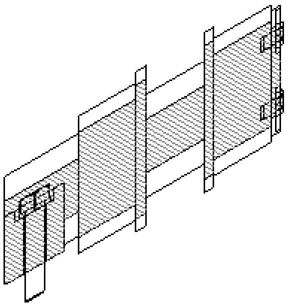
Select entities
Allows to select the entities to be included in the model space.

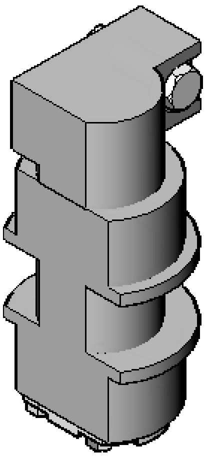
34.3.26.3 Section plane

Selects the section plane to be converted to a block.

34.3.26.4 Section type

Determines the kind of section plane block to create.





2D cross section/elevation
Uses the 2D section settings defined by the **SectionPlaneSettings** dialog box to create the block.

3D cutaway section
Uses the 3D section settings defined by the **SectionPlaneSettings** dialog box to create the block.

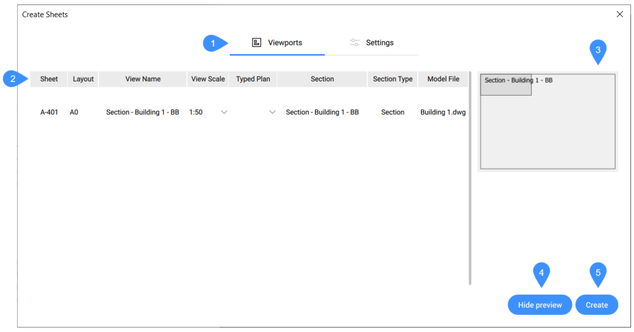
34.3.27 Create Sheets dialog box

Opens via: **BIM Project Browser** panel

The **Create Sheets** dialog box allows you to create sheets for a section or a set of sections. The dialog box can be launched from the **BIM Project Browser** panel by right-clicking a previously added section and selecting **Create Sheets**.

Note: It is also possible to select multiple sections to create sheets. The subset of sections which do not yet generate a result will be listed.

Note: If the selected sections already contain sheets, a dialog box pops up mentioning you can go to the **BIM Project Browser** panels **Sheet** tab to edit them. This can be done by right-clicking on a sheet and selecting **Edit Sheets** from the context menu.



1 Tabs

2 Tab options

3 Preview

4 Hide preview

5 Create

34.3.27.1 Tabs

Viewports

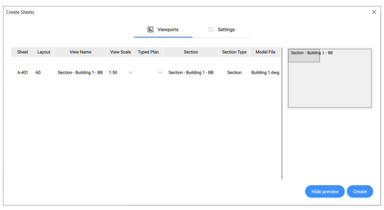
Allows you to set the layout, scale and typed plan of the viewports that will appear on the newly created sheet.

Settings

Allows you to change the name prefix and the margins of the sheets that will be created.

34.3.27.2 Tab options

Viewports tab options:



Sheet
Displays the sheet name.

Note: The name cannot be changed in the **Viewports** tab, it can be edited in the **Settings** tab.

Layout
Opens a drop-down menu where you can select a layout that is available in the sheet template. As many layouts as possible are positioned on one sheet. There is only one layout possible per sheet. If there are multiple viewports on one sheet and the layout of one of them is changed, the viewport automatically is transferred to another sheet.

Note: When a sheet view size is larger than the sheet's layout, the layout preview is displayed in red. In this case you can adjust the view scale to fit the layout.

View Name
Displays the name of the view.

Note: The view name cannot be changed in the **Viewports** tab. You can do this from the **Project Browser** after the sheet is created, by selecting the corresponding view in **Sheets** tab and typing a new name in the **Details Title** field.

View Scale
Opens a drop-down menu where you can select a predefined view scale, or select **Edit scalelist...**. This opens the **Edit Scale List** dialog box where you can create or edit scales.

Typed plan
Opens a drop-down menu from which you can assign a typed plan to the selected viewport.

Note: Typed plans can be added in the **File** tab of the **BIM Project Browser** panel. Afterwards, they will appear in the drop-down menu in the viewports tab.

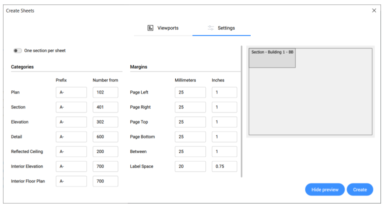
Section
Displays the section name.

Note: The section name cannot be changed in the **Viewports** tab. You can change the name by right-clicking the corresponding section in **Sections and views** tab and selecting the **Rename** option. This opens the **Rename** dialog box where you can type the new name.

Section Type
Displays the section type.

Model File
Displays the model file name in which the section is located.

Settings tab options:



One section per sheet
Toggle **Off** to place more than one section on a sheet. Toggle **On** to create one sheet per section.

Categories
Defines the naming of the categories when sheets are created. You can change the prefix and the numbering of the sheets that will be created.

Margins
Defines the margins of the sheet's layout.

34.3.27.3 Preview

Shows a preview of the sheet's layout.

34.3.27.4 Hide preview

Hides the preview of the sheet's layout. To show the preview again, click the **Show preview** button.

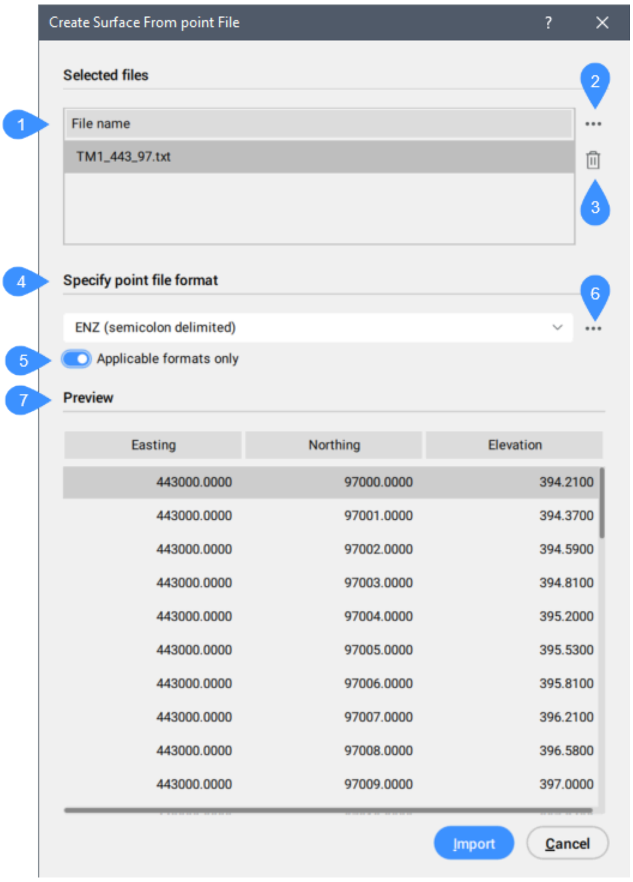
34.3.27.5 Create

Press **Create** to create the new sheets.

34.3.28 Create Surface From Point File dialog box

Opens via: TIN command > **Import from file** option

The **Create Surface From Point File** dialog box allows you to select one or more point files to create a TIN surface.



1 List of selected point files

2 Open point file
3 Delete point file
4 Specify point file format
5 Applicable formats only
6 Manage Surface Point File Formats
7 Preview

34.3.28.1 List of selected point files

Displays the selected input point files to create a TIN Surface.

34.3.28.2 Open point file

Opens the **Open file** dialog box to select a point file. The input point file can be any ASCII file of points with any extension, any separator between columns and arbitrary arrangement of data in columns. Multiple point files could be selected.

34.3.28.3 Delete point file

Removes the point file from the list.

34.3.28.4 Specify point file format

Selects a point file format from the opened drop-down list.

34.3.28.5 Applicable formats only

If checked, shows only formats that are applicable to all files in the list.

34.3.28.6 Manage Surface File Formats

Opens the **Manage Surface File Formats** dialog box (see the related article **Manage Surface File Formats dialog box**).

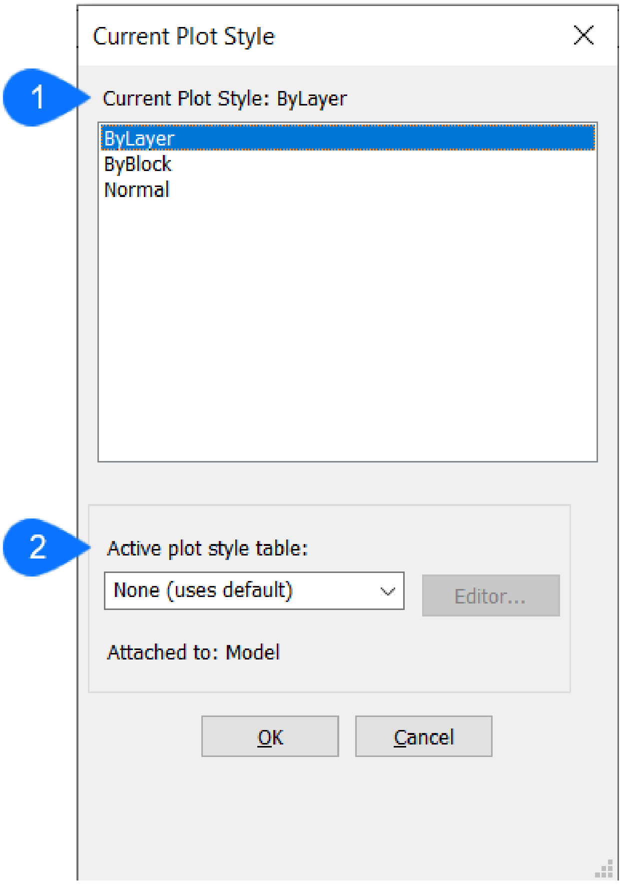
34.3.28.7 Preview

Displays all point's coordinate according to the specified point file format.

34.3.29 Current Plot Style dialog box

Opens via: PLOTSTYLE command

The **Current Plot Style** dialog box allows you to set the current plot style.



1 Current Plot Style

2 Active plot style table

34.3.29.1 Current Plot Style

Sets the current plot style for the drawing. You can choose between:

- **ByLayer**: uses the plot style of the layer of the entity.
- **ByBlock**: uses the plot style of the block the entity is assigned to.
- **Normal**: uses the normal plot style. The default properties of the entity are used.

34.3.29.2 Active plot style table

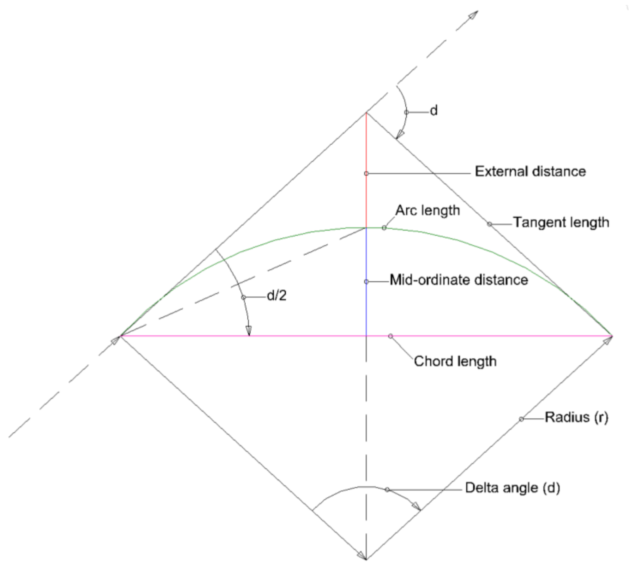
Lists the names of STB files (short for "style table"). To edit the chosen plot style table, click 'Editor...'. The **Plot Style Table Editor** dialog box will be displayed.

34.3.30 Curve Calculator dialog box

Opens via: CURVECALC command

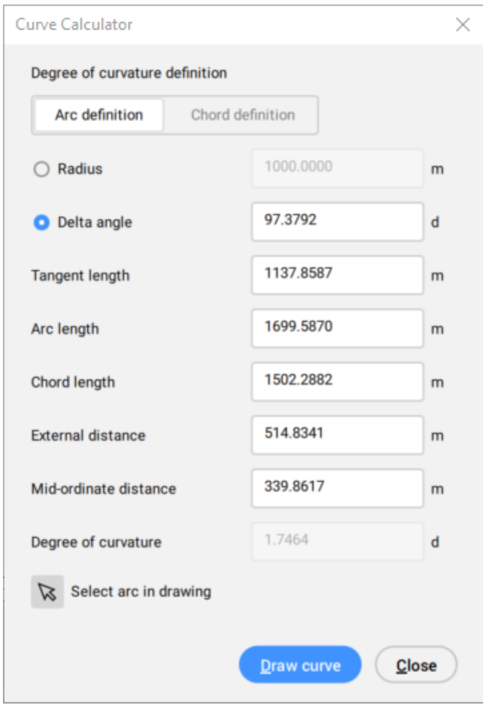
The **Curve Calculator** dialog box allows you to calculate curve parameters from inputs, using the **Arc definition** or **Chord definition** to define the **Degree of curvature**.

The curve parameters are presented in the following picture:



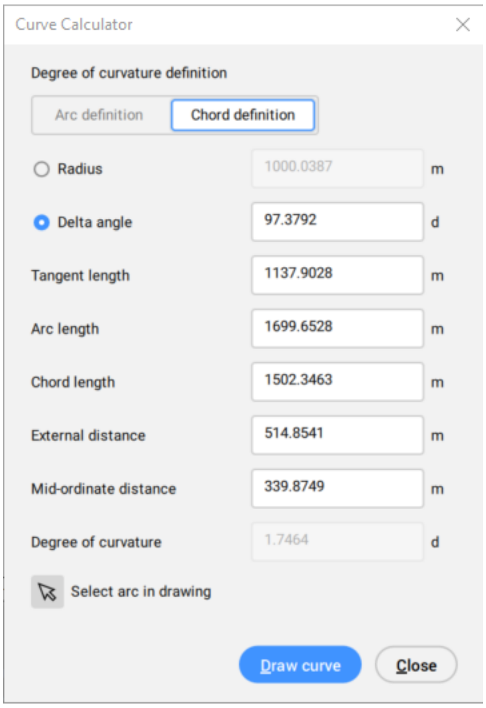
34.3.30.1 Arc definition

The **Degree of curvature** defined by the **Arc definition** means that the arc determines the degree of curvature. This definition is commonly used for roadway curves, where the degree of the curve corresponds to an angle of an arc of 100 units.



34.3.30.2 Chord definition

The **Degree of curvature** defined by the **Chord definition** means that the chord determines the degree of curvature. This definition is commonly used for railway curves, where the degree of the curve corresponds to a chord of 100 units.



The calculation is performed based on the **Radius** or the **Delta angle**.

Radius
If checked, the calculation starts from a fixed radius.

Delta angle
If checked, the calculation starts from a fixed delta angle.

Tangent length
Sets the tangent length of the curve.

Arc length
Sets the arc length of the curve.

Chord length
Sets the chord length of the curve.

External distance
Sets the external length of the curve.

Mid-ordinate distance
Sets the mid-ordinate length of the curve.

Degree of curvature
Sets the degree of curvature. Based on **Arc definition**, the degree of curvature is determined by the arc, and based on **Chord definition**, the degree of curvature is determined by the chord.

Select arc in drawing
Allows you to select an arc from the drawing.

Draw curve
Temporarily dismisses the **Curve Calculator** dialog box to draw the calculated curve in the drawing.

34.3.31 Customization Groups dialog box

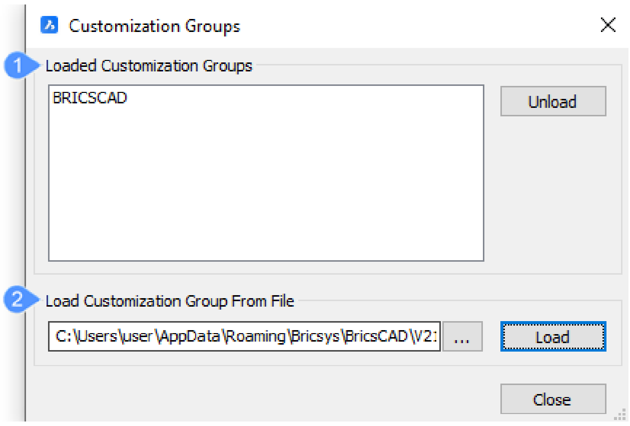
Shape Lite Pro Mechanical BIM

Opens via: MENULOAD command

The **Customizations Groups** dialog box allows you to load or to unload a partial CUI file.

When you install BricsCAD, the default CUI files are copied to the **Support** folder of your Roamable root folder. (e.g. *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support*).

When you launch BricsCAD the first time, the classic-ribbon.cui file is loaded automatically.



1 Loaded Customization Groups

2 Load Customization Group from File

34.3.31.1 Loaded Customization Groups

A list with the names of the customization files already loaded displays.

To unload a CUI file, select it from the list and click **Unload**.

34.3.31.2 Load Customization Group from File

Click the browse button to select a partial CUI file and click **Load** to load it.

Note: There are three standard CUI files available by default:

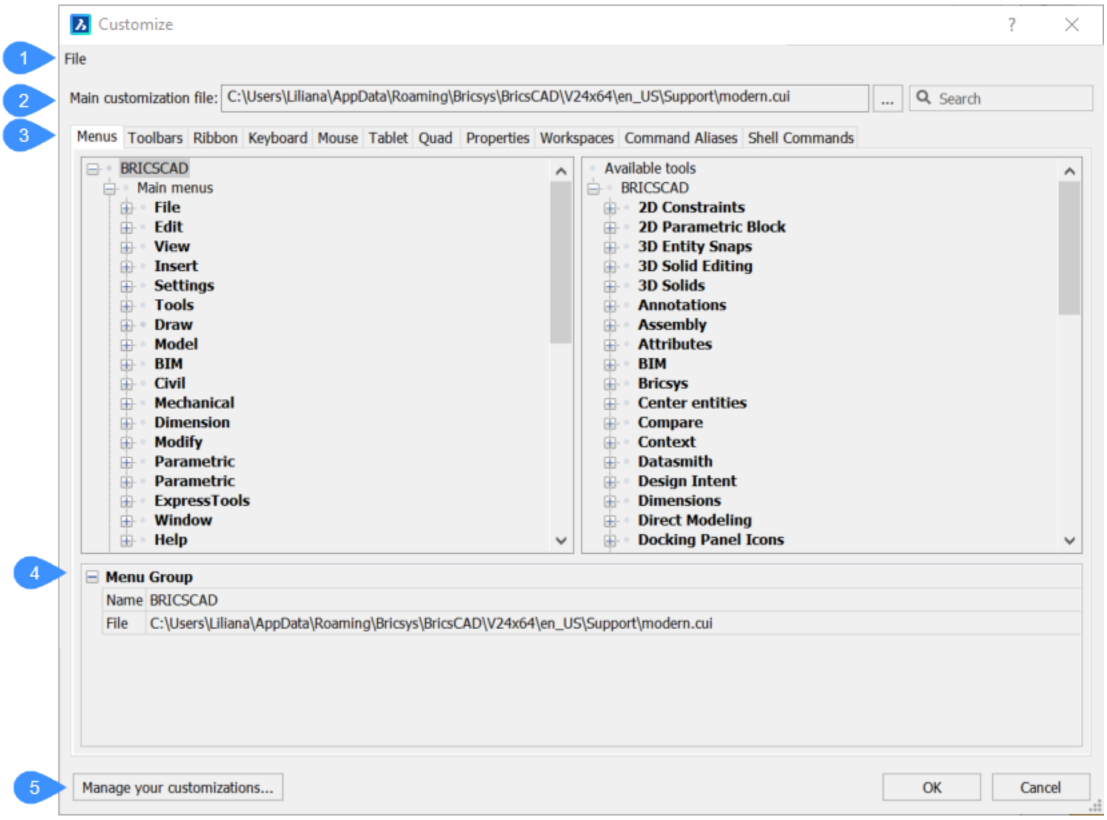
- Modern
- Classic-ribbon
- Classic-toolbars

34.3.32 Customize dialog box

Shape Lite Pro Mechanical BIM

Opens via: CUI command

The **Customize** dialog box allows you to customize the user interface of BricsCAD.



1 File

2 Main customization file

3 Tab options

4 Menu Group

5 Manage your customizations

34.3.32.1 File

Shows a menu to select customization files or import workspaces.

Load main CUI file...
Opens the **Select main CUI file** dialog box to select a customization file. See the CUILOAD command.

Save main CUI file as...
Opens the **Save main CUI file as** dialog box to save the current main CUI file.

Note: You can save the main CUI file under a different name to copy the adapted main CUI file.

Load Partial CUI file...
Opens the **Choose a Customization File** dialog box to open a partial CUI file.

Note: See chapter **Loading a partial CUI file** in **Loading and unloading CUI files** for a detailed explanation about the partial CUI files.

Create new partial CUI file...
Opens the **Create a Customization File** dialog box to create an empty CUI file.

Import workspace(s)…
Opens the **Choose a Customization File** dialog box to load one or more workspaces from another CUI file.

34.3.32.2 Main customization file

Specifies the CUI file that defines customization of the menu, toolbar, and other elements of the user interface.

Note: Aliases and shell commands are defined in a PGP file.

Browse
Opens the **Select main CUI file** dialog box to load a different main CUI file.

Note: There are three standard CUI files available by default:

- Modern
- Classic-ribbon
- Classic-toolbars

Search
Searches the Available Tools list for command names.

34.3.32.3 Tab options

Menus
Adds, edits, moves and deletes menu items.
When right clicking a menu, menu item or submenu in the **Menus** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: If you select **Append item** or **Insert item** from the context menu, the **Add menu item** dialog box displays.

Menu Group/Menu/Menu Item/Submenu
Displays the options of the selected menu, submenu or menu item.

Title
Specifies the name that is displayed by the menu, menu item or submenu.

Note: You can prefix a letter with '&' to create an Alt-key shortcut. For example: &Line.

ID
Unique identifier for each Menu item. (The ID is assigned by BricsCAD).

Alias
Defines the aliases of the menu. The browse icon opens the **Edit Aliases** dialog box where you can edit, create or delete aliases.

Diesel
Specifies the Diesel code to be employed by the menu item or submenu.

Tool ID
Identifies the menu item or sub menu to other elements in the CUI file. (The Tool ID is assigned by Bric-sCAD).

Help
Specifies the help string displayed on the status bar.

Command
Specifies the command(s) or macros.

Image
Specifies the image to display for the menu item or submenu. The browse icon opens the **Tool Image** dialog box. There are five options to select an image.

Toolbars
Adds, moves and deletes toolbar items.
When right clicking a toolbar, tool, control, flyout or separator in the **Toolbars** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: If you select **Append tool** or **Insert tool** from the context menu, the **Add tool** dialog box displays. For options description, see the **Add menu item** dialog box.

Menu Group/Toolbar/Toolbar Button
Displays the options of the selected toolbar, toolbar button, toolbar control or toolbar flyout.

Title
Specifies the name that is displayed by the tooltip.

ID
Unique identifier for each toolbar item. (The ID is assigned by BricsCAD).

Alias
Defines the aliases of the selected item. The browse icon opens the **Edit Aliases** dialog box where you can edit, create or delete aliases.

Position
Specifies the position of the selected toolbar. You can choose between **Floating**, **Top**, **Left**, **Bottom**, **Right**.

Default Display
Determines whether the selected item will be added to the workspace.

Rows
Specifies the number of rows for an undocked toolbar.

Xval Yval
Specifies the distance in pixels measured from the top left corner of the screen to the toolbar. It applies to undocked toolbars.
The values are taken from the X,Y options of the toolbar as they are set in the **Workspace** tab.

Diesel
Specifies the Diesel code to be employed by the selected item.

Tool ID
Identifies the toolbar item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

Help
Specifies the help string displayed on the toolbar.

Command
Specifies the command(s) or macros.

Image
Specifies the image to display for the toolbar button. The browse icon opens the **Tool Image** dialog box. There are five options to select an image.

Ribbon
Manages the ribbons and/or adds panels to a ribbon tab.
When right clicking a ribbon tab or panel in the **Ribbon** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: If you select **Add launcher**, **Insert toggle button** or **Insert command button** from the context menu, the **Add ribbon dialog launcher**, **Add ribbon toggle button** or **Add ribbon command button** dialog boxes display:

- **Select available tool**: assigns an existing command to the new item. If this option is selected, the other options are grayed out, except for **Available tools**.
- **Create new tool**: assigns a new command/macro to the new item. If this option is selected, **Available tools** is grayed out.
- **Toolbox**: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop down list.
- **Title**: specifies the name of the new tool.
- **Help**: specifies the help string displayed on the status bar.
- **Command**: specifies the command(s) or macros.
- **Image**: specifies the image to display for the new tool. The **browse** icon opens the **Tool Image** dialog box. There are five options to select an image
- **Available tools**: selects an existing tool.

Menu Group/Ribbon Tab/Ribbon Panel Reference
Displays the options of the selected **Ribbon Tab** or **Ribbon Panel Reference**.

ID
Identifies the item in the CUI file. (The ID is assigned by BricsCAD).

Collapse
Allows you to choose if the panel will collapse automatically or if it will never collapse.

Label
Specifies the name that is displayed by the selected **Ribbon Tab** or **Ribbon Panel Reference**.

Title
Specifies the name of the selected **Ribbon Tab** or **Ribbon Panel Reference**.

Key Tip

Note: Ribbon keytips are not yet implemented in BricsCAD.

Menu Group/Ribbon Panel/Split Button/Ribbon Row Panel/Command Button/Toggle Button
Displays the options of the selected **Ribbon Panel**, **Split Button**, **Ribbon Row Panel**, **Command Button** or **Toggle Button**.

ID
Identifies the item in the CUI file. The ID is assigned by BricsCAD.

Label
Specifies the name that is displayed by the selected Ribbon item.

Title
Specifies the name of the selected **Ribbon Panel**.

Button Style
Specifies the display mode for the selected button. You can choose between **Small with text**, **Small without text**, **Large with text (vertical)**, **Large with text (horizontal)** and **Large without text**.

Note: The icon size for the small options is 16×16 pixels, while for the large options is 32×32 pixels.

Behavior
Determines how the topmost button behaves when you click on it. You can choose between **Drop down**, **Drop down with recent**, **Split**, **Split with recent**, **Split with recent (static text)**. The options determine whether the button displays the default command (the first one in the list of buttons), or the most recently used one.

Note: Split buttons let you click the upper half to execute the most-recently-used command, or lower half to display the drop-list (flyout).

List Style

Note: This option is not yet implemented in BricsCAD.

Grouping

Note: This option is not yet implemented in BricsCAD.

Image
Specifies the image to display for the selected Ribbon item.

Resize Style

Note: This option is not yet implemented in BricsCAD.

Resize Priority

Note: This option is not yet implemented in BricsCAD.

Justify Top

Note: This option is not yet implemented in BricsCAD.

Tool ID
Identifies the ribbon item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

Help
Specifies the help string displayed on the Ribbon.

Command
Specifies the command(s) or macros.

Keyboard
Assigns different keyboard shortcuts to different commands.
When right clicking a keyboard shortcut in the **Keyboard** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: If you select **Append shortcut** or **Insert shortcut** from the context menu, the **Add keyboard shortcut** dialog box displays.

- **Select available tool**: assigns an existing command to the new keyboard shortcut. If this option is selected, the other options are grayed out, except for **Available tools**.
- **Create new tool**: assigns a new command/macro to the shortcut. If this option is selected, **Available tools** is grayed out.
- **Toolbox**: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list.
- **Title**: specifies the name of the new tool.
- **Help**: specifies the help string displayed on the status bar.
- **Command**: specifies the command(s) or macros.
- **Image**: specifies the image to display for the new tool. The browse icon opens the **Tool Image** dialog box. There are five options to select an image.
- **Available tools**: selects an existing tool.

Menu Group/Keyboard Shortcut
Displays the options of the selected keyboard shortcut.

Key
Specifies the used keyboard shortcut.

Tool ID
Identifies the keyboard item to other elements in the CUI file. The Tool ID is assigned by BricsCAD.

Help
Specifies the help string displayed on the status bar, even though help strings are not displayed by keyboard shortcuts.

Command
Specifies the command(s) to associate with the keyboard shortcut.

Image
Specifies the image to associate with the command, even though images are not used with keyboard shortcuts.

Mouse
Modifies the different actions that are connected to the mouse buttons.
When right clicking a button or button group in the **Mouse** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: If you select **Append button** or **Insert button** from the context menu, the **Add button item** dialog box displays.

Alias
Defines the aliases of the selected item. The browse icon opens the **Edit Aliases** dialog box where you can edit, create or delete aliases.

Button
Specifies the used mouse button.

Tool ID
Identifies the button item to other elements in the CUI file. The Tool ID is assigned by BricsCAD.

Title
Specifies the name that is displayed by the button item.

Help
Specifies the help string displayed on the status bar.

Command
Specifies the command(s) or macros.

Image
Specifies the image to display for the button item. The browse icon opens the **Tool Image** dialog box. There are five options to select an image.

Tablet
Customizes the tablet options.
When right clicking a button or button group in the **Tablet** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: If you select **Append button** or **Insert button** from the context menu, the **Add button item** dialog box displays.

Menu Group/Button Group
Displays the options of the selected button group.

Alias
Defines the aliases of the selected item. The browse icon opens the **Edit Aliases** dialog box where you can edit, create or delete aliases.

Quad
Adds commands to the quad tab or move commands to other tabs.
When right clicking a quad tab or quad button in the **Quad** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: If you select **Append quad button** or **Insert** from the context menu, the **Add quad button** dialog box displays.

- **Select available tool**: assigns an existing command to the new quad button. If this option is selected, the other options are grayed out, except for **Available tools**.
- **Create new tool**: assigns a new command/macro to the new quad button. If this option is selected, **Available tools** is greyed out.
- **Toolbox**: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list.
- **Title**: specifies the name of the new tool.
- **Help**: specifies the help string displayed on the status bar.
- **Command**: specifies the command(s) or macros.
- **Image**: specifies the image to display for the new tool. The browse icon opens the **Tool Image** dialog box. There are five options to select an image.

- **Available tools**: selects an existing tool.

ID
Identifies the Quad item in the CUI file. The ID is assigned by BricsCAD.

Title
Specifies the name that is displayed by the Quad button.

Help
Specifies the help string displayed on the Quad.

Command
Specifies the command(s) or macros related to the Quad button.

Image
Specifies the image to display for the Quad button. The browse icon opens the **Tool Image** dialog box. There are five options to select an image.

Entity Filter
Specifies which items will be displayed in the Quad, depending on the type of the entity.

Properties
Edits which properties are shown for different kinds of entities.

Note: These properties will only be shown when the ROLLOVERTIPS system variable is ON.

Rollover
Specifies which properties should be displayed.

Workspaces
Controls which menu tabs are visible in different workspaces.

When right clicking an item in the **Workspace** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Name
Shows the name of the workspace displayed in the workspace list in the status bar and by the Workspace toolbar (reported by the WSCURRENT system variable).

Display
Determines if the workspace name is displayed by the status bar and toolbar drop lists.

Description
Help-like description displayed in the status bar.

ID
Identifies the workspace item in the CUI file. The ID is assigned by BricsCAD.

Menu Bar
Toggles the display state of the menu bar. See also MENUBAR (EXCEPT OS X) system variable.

Scrollbars
Toggles the display of scroll bars. See also the WNDLSCRL system variable.

Default
Toggles whether this workspace is the default one when BricsCAD starts.

- **Yes**: shows this workspace when BricsCAD starts.

- **No**: does not show this workspace.

Stack Type
Determines how panels are displayed. See also the STACKPANELTYPE system variable.

Panel Button Size
Specifies the initial size of buttons on panels.

Quad Button Size
Specifies the initial size of buttons on Quad.

Ribbon Tool Size
Specifies the initial size of buttons on Ribbon.

Tool Button Size
Specifies the initial size of buttons on tools.

Note:

- **Small**: displays 16×16 icons.
- **Large**: displays 24×24 icons.
- **Extra large**: displays 32×32 icons.

Ribbon Panel Margin
Specifies the size of blank space at ribbon panel edges, measured in pixels. See also the RIBBONPANELMARGIN system variable.

Toolbar Margin
Specifies margin above and below each toolbar, measured in pixels. See also the TOOLBARMARGIN system variable.

Tool Icon Padding
Specifies margin between icons on each toolbar, measured in pixels.

Delete Tool
Specifies what to do with tool entities following the SUBTRACT command. See also the DELETETOOL system variable.

DMAUDIT Detail Level
Specifies messages to display. See also the DMAUDITLEVEL system variable.

DMPUSHPULL Subtract
Specifies whether to enable subtract mode in the DMPUSHPULL command. See also the DMPUSHPULLSUBTRACT system variable.

Extrude Inside
Determines how to modify the parent entity when it intersects with the extruded/revolved entity when the Auto option of the EXTRUDE and REVOLVE command is selected. See also the EXTRUDEINSIDE system variable.

Extrude Outside
Determines how to modify the parent entity when it touches the extruded/revolved entity when the Auto option of the EXTRUDE and REVOLVE command is selected. See also the EXTRUDEOUTSIDE system variable.

Generate Associative Attributes
Specifies whether associative attributes are generated for 3D entities. See also the GENERATEASSOCATTRS system variable.

Generate Associative Drawings
Specifies associativity between the 3D model and generated views (VIEWBASE) and calculated drawings (BIMSECTIONUPDATE). See also the GENERATEASSOCVIEWS system variable.

Intersected Entities
Determines how to modify entities that intersect with the extruded/revolved entity when the Auto option of the EXTRUDE or REVOLVE command is selected. See also the INTERSECTEDENTITIES system variable.

Report Panel Mode
Specifies the mode of report panel. Applicable only for commands which support it. See also the REPORTPANELMODE system variable.

Selection Modes
Specifies the subentities to highlight during entity selection. See also the SELECTIONMODES system variable.

Sheet Set Manager Auto Open
Toggles whether the **Sheet Sets** panel is opened automatically when a drawing from a sheet set is opened.

Structure Tree Config
Names the CST structure tree configuration file to use.

Unite Surfaces
Determines whether adjacent extruded/revolved surfaces are united or not. See also the UNITESURFACES system variable.

Components Config
Names the CCF components configuration file to use.

Menu Group
Allows you to choose from the available menu groups.

Title
Shows the name of the menu displayed on the menu bar.

Diesel
Executes Diesel code when you select the menu.

Command Aliases
Customize aliases and add them to different commands.
When right clicking an item in the **Command Aliases** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: If you select **Add alias** or **Edit alias** from the context menu, the **Add alias** or **Edit alias** dialog box displays where you can add or edit an alias.

Alias
Specifies the name of the alias, a command abbreviation.

Note: The list of Aliases and commands will be ordered alphabetically according to the aliases when you click on **Alias**.

Command
Specifies the command to which the alias is assigned.

Note: The list of Aliases and commands will be ordered alphabetically according to the commands when you click **Command**.

Add
Adds an alias.

Edit
Edits the selected alias.

Delete
Deletes the selected alias.

Commands
Lists all the available commands to add an alias to.

Shell Commands
Create and edit shell commands. Shell commands run programs external to BricsCAD.
When right clicking an item in the **Shell Commands** tab, a context menu opens. See the paragraph **Options Within the Context Menu** for further explanation.

Note: If you select **Add shell command** or **Edit shell command** from the context menu, the **Add shell command** or **Edit shell command** dialog box displays where you can add or edit a shell command.

Alias
Specifies the name of the shell command; this is the command you enter at the ' : ' prompt.

Shell Command
Specifies the operating system command to execute, or program to run.
To run a program, use the Start prefix, as in Start Notepad.exe.

Prompt
Specifies the prompt to display in the Command line, for example: **Enter file to edit:**.

Flags
Determines how the shell command operates:

- **Don't wait**: BricsCAD immediately returns to the command prompt without waiting for the application to finish.
- **Minimized**: the application runs minimized.
- **Hidden**: the application does not appear on the screen.
- **Quotes**: the shell command uses quotations when the command string uses spaces, such as *C:\cad programs\file name.exe*.

Add
Adds a shell command through a dialog box.

Edit
Edits the shell command through a dialog box.

Delete
Deletes the selected shell command. Gives a warning where you can confirm you to delete the selected item.

Options Within the Context Menu
The following options appear when you right click on an item:

Append…
Adds the mentioned item below the last item.

Insert…
Adds the mentioned item above the selected item.

Add…
Adds the mentioned item in alphabetical order.

Delete…
Deletes the menu, submenu, item or separator selected item. Gives a warning where you can confirm you to delete the selected item.

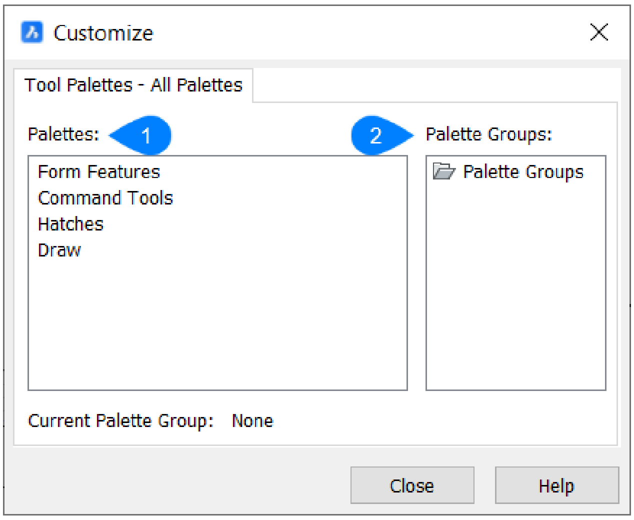
34.3.32.4 Manage your customizations

Displays the **Manage Customizations** dialog box to confirm and reverse changes made to the user interface.

34.3.33 Customize (Tool Palettes) dialog box

Opens via: **Tool Palettes** panel

The **Customize** (Tool Palettes) dialog box allows you to customize your own tabs and tools to use in the tool palettes panel.



1 Palettes

2 Palette Groups

34.3.33.1 Palettes

Displays a list of all palettes available in the tool palettes panel. By right clicking on one of the palettes, several options may be available in the context menu.

Rename
Renames the Palette by entering the new name.

New Palette
Adds a new Palette to the list with a generic name that can be changed right away.

Delete
Deletes the selected palette

Export
Exports a palette through the **Export Palette** dialog box.

Import
Imports a palette through the **Import Palette** dialog box.

34.3.33.2 Palette Groups

Displays the loaded palette groups. By right clicking on one of the groups, several options may be available in the context menu.

New group
Adds a new sub-group to the list with a generic name that can be changed right away.

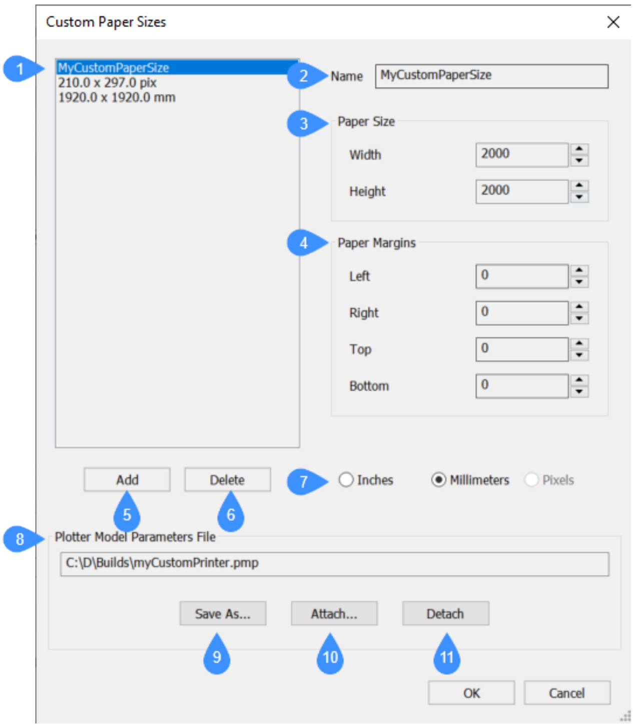
Note: For the additional sub group, the available options in the context menu are the same as those for palettes.

Import
Imports a palette through the **Import Palette Group** dialog box.

34.3.34 Custom Paper Sizes dialog box

Opens via: **Print** dialog box

The **Custom Paper Sizes** dialog box allows you to create custom paper sizes for a non-system plotter (such as Print as PDF.pc3) selected in the **Printer / Plotter Configuration** list or change a standard paper size.



1 Custom Paper Sizes list
2 Name
3 Paper Size
4 Paper Margins
5 Add
6 Delete
7 Units
8 Plotter Model Parameters File (PMP file)
9 Save As
10 Attach
11 Detach

34.3.34.1 Custom Paper Sizes list

Displays the list of all custom paper sizes created by the user.

34.3.34.2 Name

Allows you to name the custom paper size.

34.3.34.3 Paper Size

You can insert the width and the height of the page in the appropriate fields.

34.3.34.4 Paper Margins

You can set the paper margins by entering the desired values in the fields. Only positive values are allowed.

34.3.34.5 Add

Allows you to create the new custom paper size and adds it to the list.

34.3.34.6 Delete

Allows you to select a custom paper size from the list and delete it. You can delete one item at a time.

34.3.34.7 Units

You can choose whether to use inches or millimeters for your custom paper size.

34.3.34.8 Plotter Model Parameters File (PMP file)

Displays the path of the attached PMP file, if any.

34.3.34.9 Save As

Saves the selected custom paper sizes configuration to a PMP file that can be used for future plotting.

34.3.34.1 Attach 0

Attaches the PMP file to the plotter configuration (PC3) file. Opens **Calibration data file** dialog box where you have to choose a PMP file. The custom paper size will be available in the drop-down paper size list.

34.3.34.1 Detach 1

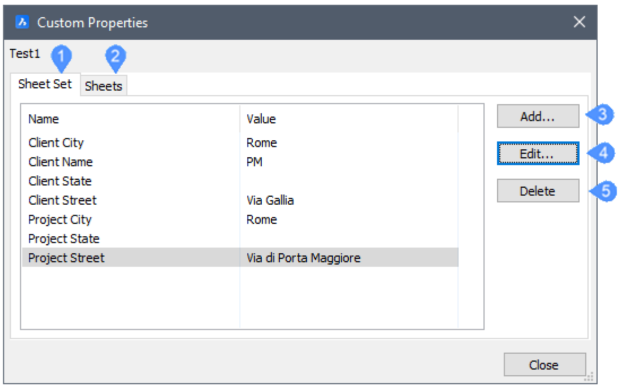
Detaches the PMP file from the plotter configuration (PC3) file.

34.3.35 Custom Properties dialog box

Opens via:

- **BIM Project Browser** panel
- **Sheet Sets** panel

The **Custom Properties** dialog box allows you add custom properties for sheet set and sheets.



1 Sheet Set Tab

2 Sheets Tab

3 Add

4 Edit

5 Delete

34.3.35.1 Sheet Set Tab

Manages the custom sheet set properties.

34.3.35.2 Sheets Tab

Manages the custom sheets properties.

34.3.35.3 Add

Adds a custom property. Displays the **Add Custom Property** dialog box in which you add a property name and its initial value.

34.3.35.4 Edit

Allows to edit the value of a selected custom property. Displays the **Custom Sheet Property** or **Custom Sheet Set Property** dialog box in which you edit the name of the property.

34.3.35.5 Delete

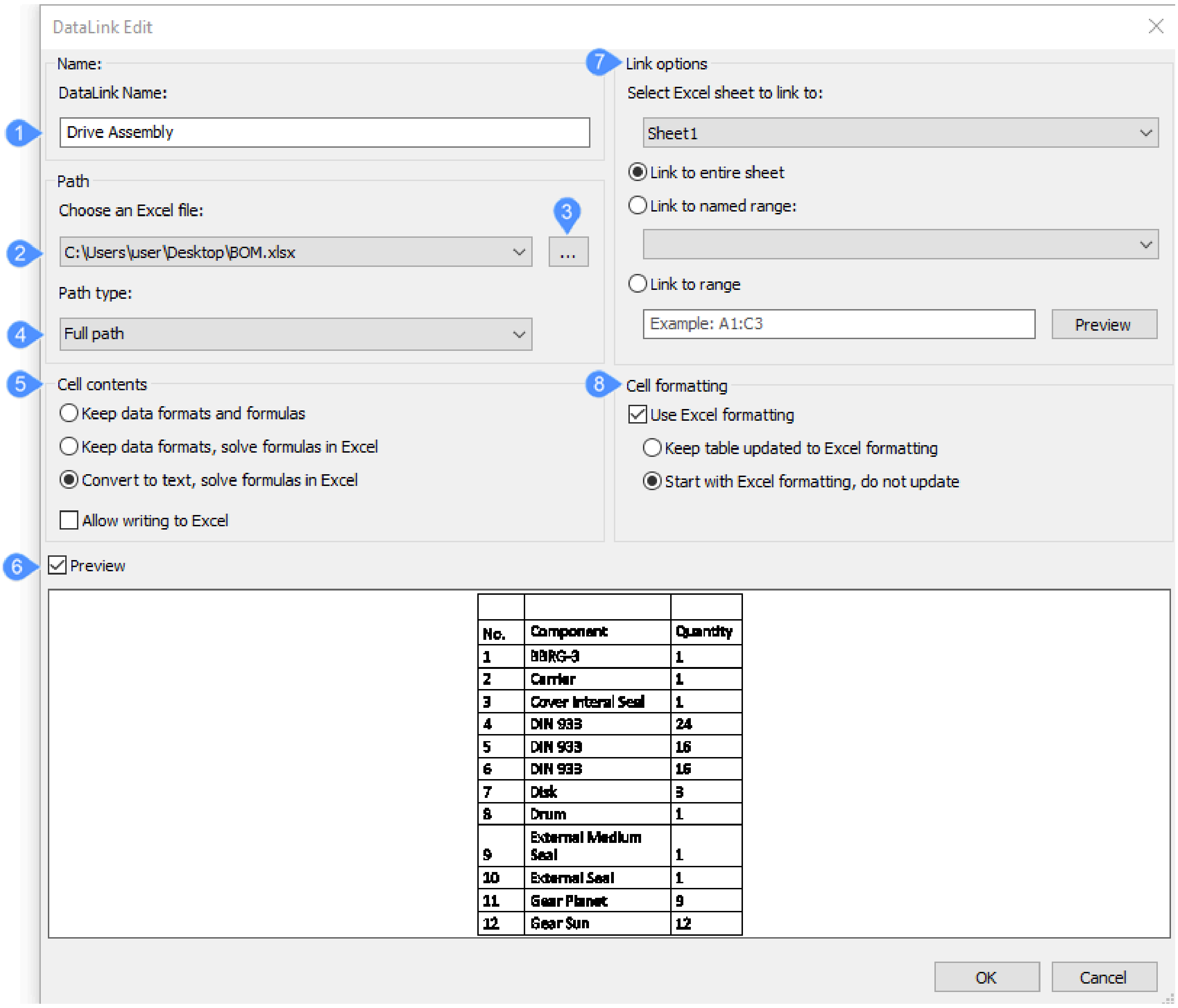
Deletes a selected custom property.

34.4 D

34.4.1 DataLink Edit dialog box

Opens via: **Datalink Manager** dialog box

The **DataLink Edit** dialog box allows you to link data from an Excel spreadsheet directly into drawing tables for bi-directional updating.



1 Name

2 Path

3 Browse

4 Path type

5 Cell contents

6 Preview

7 Link options

8 Cell formatting

34.4.1.1 Name

You can type a name in the **DataLink Name** field.

34.4.1.2 Path

Displays the path where your Excel file is located.

34.4.1.3 Browse

Allows you to select a data file. *.xls, *.xlsx and *.csv files are accepted.

34.4.1.4 Path type

Allows you to choose a path type option:

- **Full Path**: BricsCAD searches for the data file in the specified folder only.
- **Relative Path**: the data file is searched for in a subfolder of the drawing file folder.
- **No Path**: the data file is searched for in the drawing file folder only.

34.4.1.5 Cell contents

- **Keep data formats and formulas**: imports data with formulas and supported data formats.
- **Keep data formats, solve formulas in Excel**: imports data formats. Data is calculated from formulas in Excel.
- **Convert to text, solve formulas in Excel**: imports data as text with data calculated from formulas in Excel.
- **Allow writing to Excel**: if checked, the DATALINKUPDATE command uploads changes to linked data in your drawing to the original external spreadsheet.

34.4.1.6 Preview

If this button is selected, a preview of the data file is displayed.

34.4.1.7 Link options

Allows you to select Excel sheet to link to:

- **Link to the entire sheet**: all cells that contain data are linked.
- **Linked to named range**: links to a named range of cells. Click the button to select a named range.
- **Link to range**: specify a range of cells. Click the **Preview** button to display the selected range.

34.4.1.8 Cell formatting

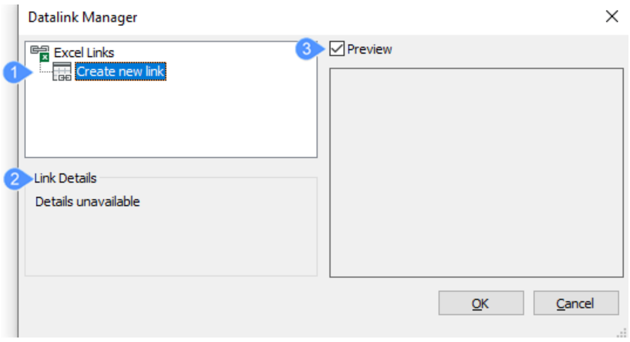
You must select the **Use Excel formatting** option to be able to select one of the following options:

- **Keep table updated to Excel formatting**: updates any changed formatting when the DATALINKUPDATE command is executed.
- **Start with Excel formatting. Do not update**: imports the formatting specified in the original data file. The DATALINKUPDATE command does not affect any changes made to the cell formatting.

34.4.2 Datalink Manager dialog box

Opens via: DATALINK command

The **Datalink Manager** dialog box links data from an Excel spreadsheet directly into drawing tables for bi-directional updating.



1 Create new link

2 Link details

3 Preview

34.4.2.1 Create new link

Opens the **DataLink Edit** dialog box. See the related article **DataLink Edit dialog box**.

34.4.2.2 Link details

Displays link details.

34.4.2.3 Preview

Display a preview of the datalink.

34.4.3 Datum Identifier dialog box

Opens via: AMDATUMID command

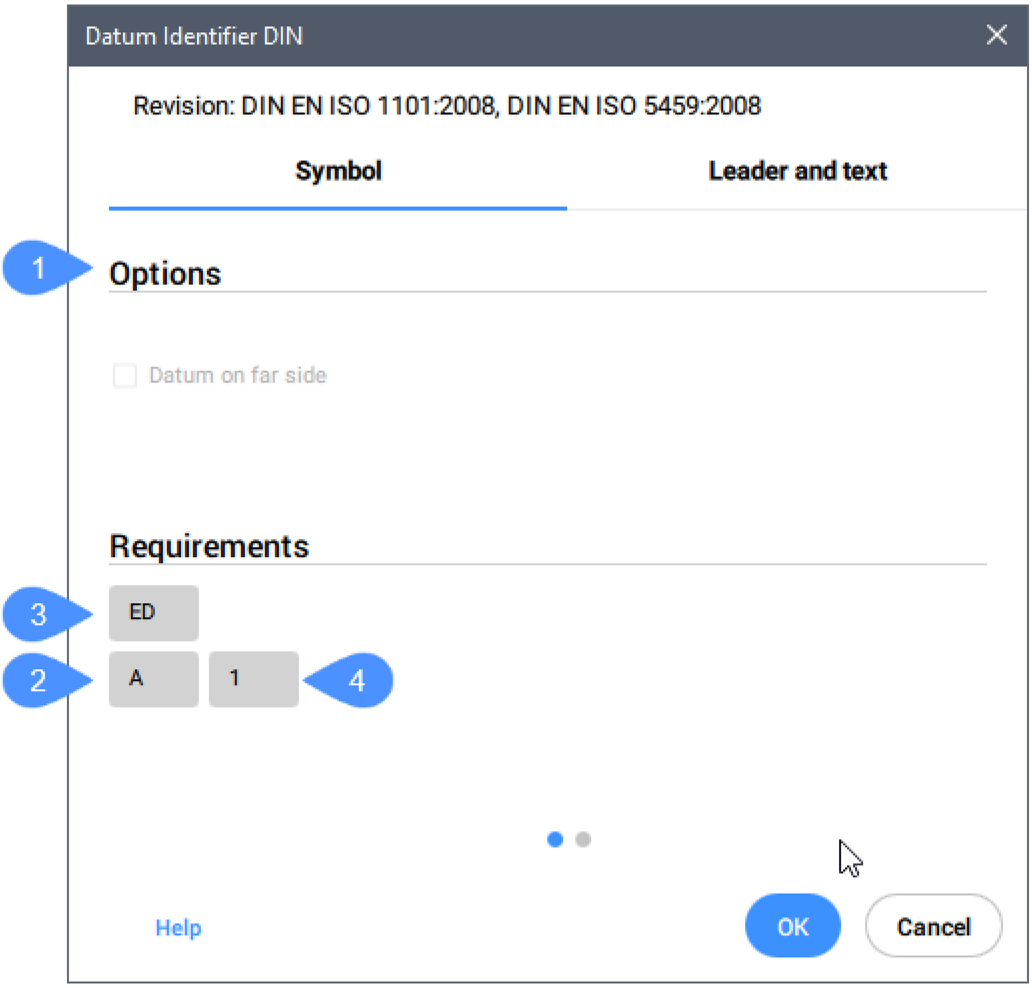
The **Datum Identifier** dialog box allows you to set all parameters for the datum identifier symbol.

The parameters entered for the datum identifier symbol will be stored temporarily and used as default values for the next datum identifier setting.

The most recent revision of the standard to which this symbol conforms is displayed at the top of the dialog box.

The dialog box contains 2 main tabs, **Symbol** and **Leader and text**.

34.4.3.1 Symbol



1 Options

2 Identifier

3 Thread note

4 Datum note

Options (1)
Toggles the datum type.

Identifier (2)
Defines the identifier, which can contain a maximum of two characters that is automatically assigned in alphabetical order, starting with A, followed by B, C, and so on. After reaching Z, the sequence continues with AA, AB, AC, etc. If an identifier is deleted, the AMDATUMID command will attempt to fill the gap starting from the top of the alphabetical list. For example, if identifiers A, B, and C are in use and B is deleted, the next time an identifier is needed, it will reuse B to fill that gap.

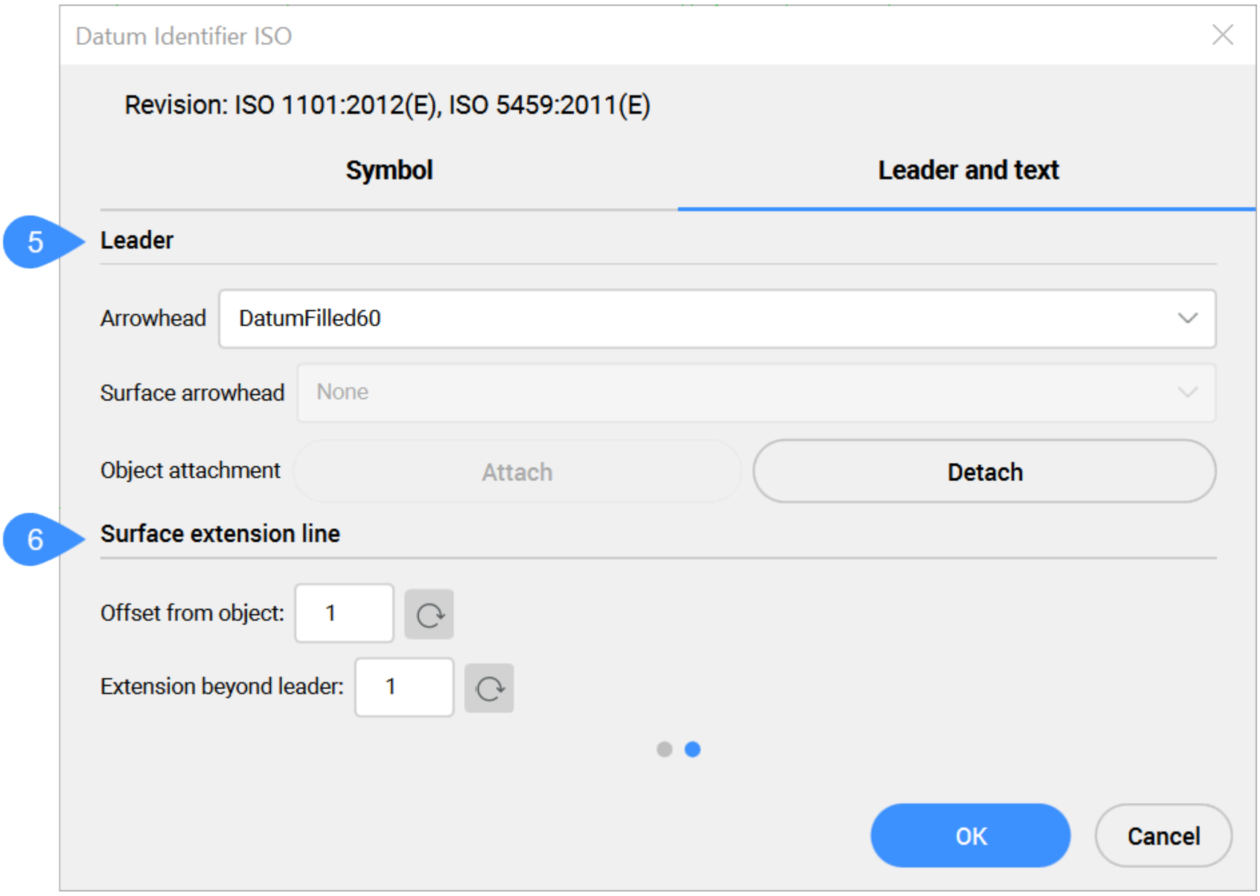
Thread note (3)
Defines the thread notes, which are placed on symbols attached to gears or screw threads. They specify which diameter to use as the datum.

Datum note (4)
Defines the reference datum targets which correspond to points on a surface. Generally, it contains a series of datum targets separated by commas (",").

Note: Thread and datum notes are not available for all standards.

34.4.3.2 Leader and text

Specifies the Leader and text characteristics.



Leader (5)
Sets the leader characteristics.

Arrowhead
Sets the default leader arrowhead type.

Surface arrowhead
Sets the arrowhead for surface indication leaders.

Note: This option is only available for the standards which allow surface indicator leaders.

Object attachment
Defines if the symbol leader is attached or detached to the object.

Attach
Attaches the symbol to the selected object.

Detach
Detaches the symbol from the object.

Surface extension line (6)
Sets the surface extension line characteristics.

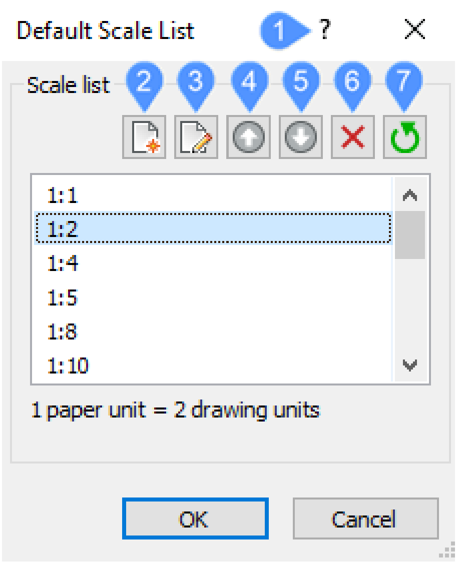
Offset from object
Sets the offset from the object.

Extension beyond leader
Sets the extension beyond leader.

34.4.4 Default Scale List dialog box

Opens via: DEFAULTSCALELIST command

The **Default Scale List** dialog box allows you to edit the list of scale factors displayed by commands, such as PRINT and PAGESETUP and by annotative scale factors.



1 Command reference

2 Add new scale

3 Edit scale

4 Move up

5 Move down

6 Delete

7 Reset

34.4.4.1 Command reference

Opens the Bricsys Help article about the DEFAULTSCALELIST command.

34.4.4.2 Add new scale

Allows you to add a new scale factor to the list. Opens the **Add Scale** dialog box.

34.4.4.3 Edit scale

Allows you to edit the factors of an existing scale. Opens the **Edit Scale** dialog box.

34.4.4.4 Move up

Moves the selected scale factor up the list.

34.4.4.5 Move down

Moves the selected scale factor down the list.

34.4.4.6 Delete

Deletes the selected scale factor(s). Press the **Ctrl** key to select and delete more than one factor at a time. Scale 1:1 cannot be deleted.

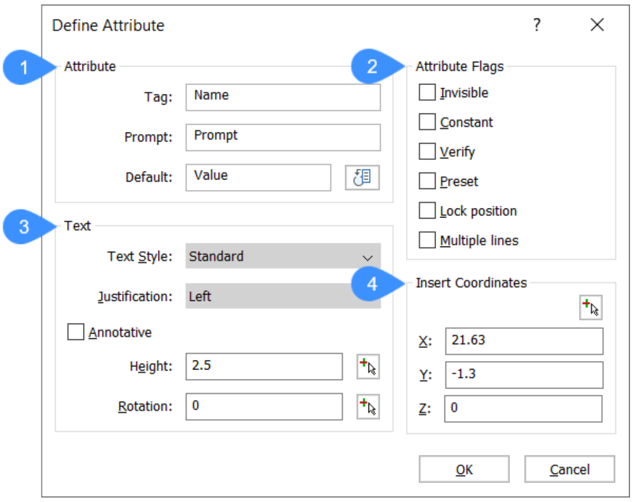
34.4.4.7 Reset

Returns the list to its original condition, adding back in the deleted factors and removing those that were added, but are not being used.

34.4.5 Define Attribute dialog box

Opens via: ATTDEF command

The **Define Attribute** dialog box defines the options for the attribute values.



1 Attribute options

2 Attribute Flags options

3 Text options

4 Insert Coordinates options

34.4.5.1 Attribute options

- **Tag**

  Specify the name of the attribute. This is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers and punctuation.
- **Prompt**

  Specify the user prompt. This is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank. BricsCAD then uses the tag as the prompt during attribute insertion.
- **Default**

  Specify the default value. This is displayed in angle brackets, for example <360>. Press **Enter** to accept the value.

  Click the **Insert Field** icon to assign a field value to the attribute (see the FIELD command).

34.4.5.2 Attribute Flags options

- **Invisible**

  Hide the attribute from view; they are not displayed and not printed. However, hidden attributes can be displayed with the ATTDISP command.
- **Constant**

  Specify a default value that cannot be changed.
- **Verify**

  Forces to enter the value a second time. This helps to ensure that the value is entered correctly.
- **Preset**

  Inserts attributes without prompting. The attributes can be changed later with the ATTEDIT command.
- **Lock position**
  - **On**: locks the position of the attribute within the block reference.
  - **Off**: unlocked attributes can be moved relative to the rest of the block using grip editing.
- **Multiple lines**
  - **On**: allows the use of multiline text.
  - **Off**: limits to a single line.

  When the **Multiple lines** box is checked, the button icon displays next to the dimmed **Default** text field.

  When you click the button, the **Define Attribute** dialog box temporarily closes to let you define a multiline text entity.

  The MTEXT command launches. Type the text for the attribute, then click the **OK** button on the **Text Formatting** toolbar to return to the **Define Attribute** dialog box.

34.4.5.3 Text options

- **Text style**

  Specify the text style for the attribute text. Choose from one of the styles defined in the drawing. (You can create additional text styles with the STYLE command.)

- **Justification**

  Specify the justification of the attribute text, such as **Center** or **Fit**.

- **Height**

  Specify the height of the attribute text, unless overridden by the text style. Enter a value, or specify two points in the drawing by clicking the button. The last option dismisses the dialog box temporarily.

- **Rotation**

  Specify the rotation angle of the rotation text. Enter a value, or specify two points in the drawing by clicking the button. The last option dismisses the dialog box temporarily.

- **Annotative**

  Specify the **Annotative** property of the attribute text.

  Note: Supported annotation scales of a selected block attribute can be modified from the **Properties** panel, even when its owning block reference is not annotative.

34.4.5.4 Insert Coordinates options

- **Select insertion point**

  Specify the insertion point directly in model.

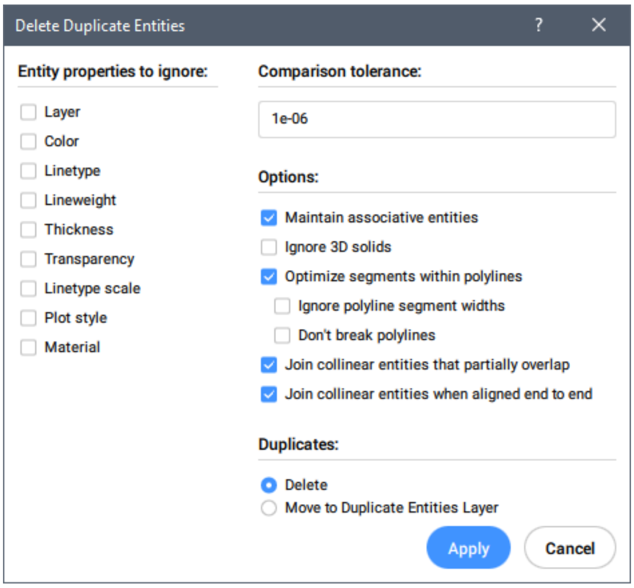
- **X/Y/Z**

  Specify the coordinates of the attribute's insertion point.

34.4.6 Delete Duplicate Entities dialog box

Opens via: OVERKILL command

The **Delete Duplicate Entities** dialog box allows you to delete duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones.



1 Entity properties to ignore

2 Comparison tolerance

3 Options

4 Duplicates

34.4.6.1 Entity properties to ignore

Ignores selected properties when comparing duplicate or overlapping entities.

34.4.6.2 Comparison tolerance

Sets the tolerance for the comparison process. If set to 0 (zero), entities must match completely before being evaluated in the overkill process.

34.4.6.3 Options

Determines how special cases are handled.

Maintain associative entities
Associative entities are not processed.

Ignore 3D solids
Determines if duplicate 3D solids are ignored or removed.

Optimize segments within polylines
Evaluates segments of selected polylines individually, duplicate vertices and segments are removed.

- **Ignore polyline segment widths**: ignores the Width property of overlapping polyline segments.
- **Don’t break polylines**: polylines entities are kept intact.

Join collinear entities that partially overlap
Unifies partially overlapping entities into a single entity.

Join collinear entities when aligned end to end
Unifies collinear lines and polylines having coinciding endpoints into a single entity or segment.

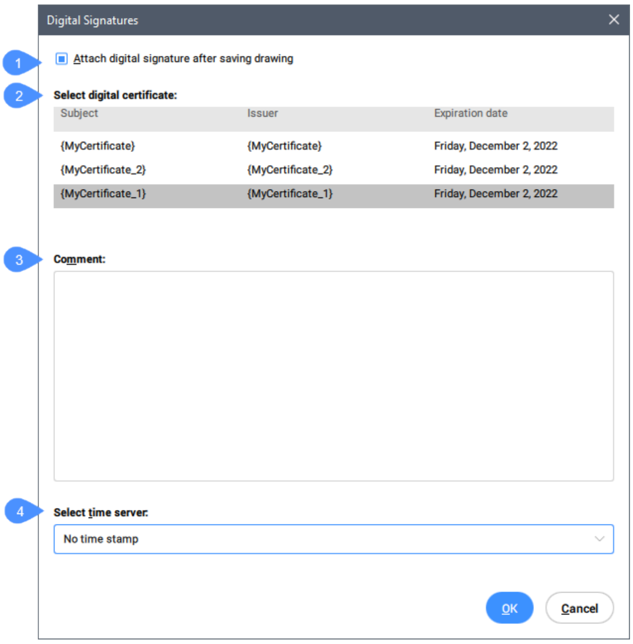
34.4.6.4 Duplicates

Determines what to do with duplicates: either delete them or move them to a Duplicate Entities layer.

34.4.7 Digital Signatures dialog box

Opens via: DIGITALSIGN command

Applies a digital signature to the drawing.

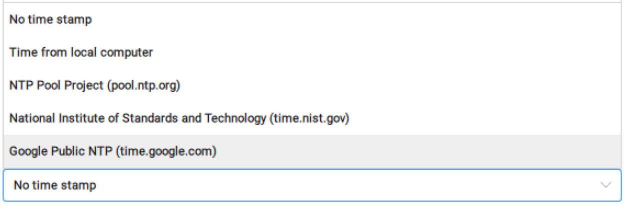


Attach digital signature after saving drawing
Tick the box to attach the digital signature after saving the drawing.

Select digital certificate
Select the digital certificate that you want to attach to the current drawing.

Comment
Adds a comment to the signature.

Select time server
Sets a time server from the pop-up menu:

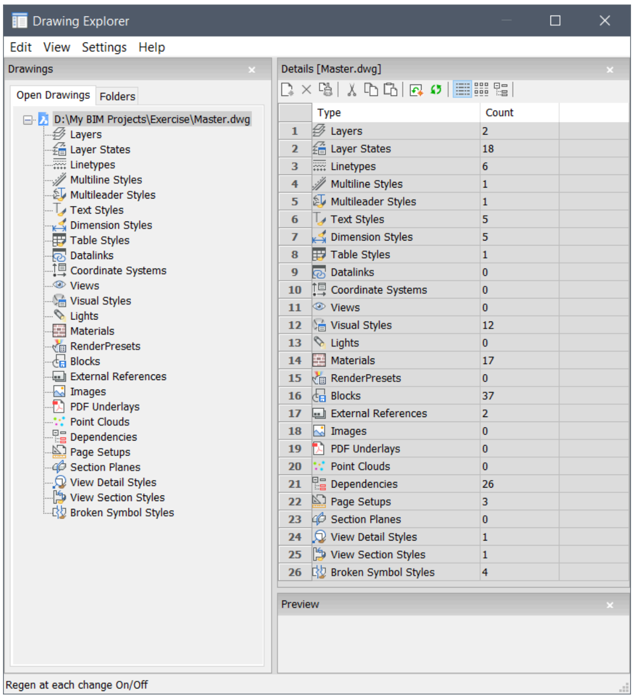


The selected time server will be used to obtain the precise time of the drawing signing. This means that when the signature is attached to the drawing during saving, the date and time from the selected time server will be inserted in the Signature date and time field of the Digital Signature Content. See the SIG-VALIDATE command.

34.4.8 Drawing Explorer dialog box

34.4.8.1 Opening the Drawing Explorer dialog box

- Launch the EXPLORER command.
- Launch a definition category command, such as EXPLAYERS or EXPBLOCKS.
- Choose a definition category in the **Tools** > **Drawing Explorer** menu.
- Choose **Drawing Explorer** in the **General** tab of the No Selection Quad.



Categories	Description
Layers	Reports the number of layers defined in the selected drawing. Click to explore the layers; see the EXPLAYERS command.
Layer States	Reports the number of layer states defined in the selected drawing. Click to explore the layer states; see the LAYERSTATE command.
Linetypes	Reports the number of linetypes loaded into the selected drawing. Click to explore the linetypes; see the LINETYPE command.
Multiline Styles	Reports the number of multiline styles defined in the selected drawing. Click to explore multiline styles; see the MLSTYLE command.
Multileader Styles	Reports the number of multileader styles defined in the selected drawing. Click to explore multileader styles; see the MLEADERSTYLE command.

Categories	Description
Text Styles	Reports the number of text styles defined in the selected drawing. Click to explore text styles; see the STYLE command.
Dimension Styles	Reports the number of dimension styles defined in the selected drawing. Click to explore dimension styles; see the DIMSTYLE command.
Table Styles	Reports the number of table styles defined in the selected drawing. Click to explore table styles; see the TABLESTYLE command.
Data Links	Reports the number of data in the drawing linked to external spreadsheet files. Click to explore data links; see the DATALINK command.
Coordinate Systems	Reports the number of named user coordinate systems (UCS) defined in the selected drawing. Click to explore coordinate systems; see the EXPUCS command.
Views	Reports the number of named views in the selected drawing. Click to explore saved views; see the VIEW command.
Visual Styles	Reports the number of visual styles defined in the selected drawing. Click to explore visual styles; see the VISUALSTYLES command.
Lights	Reports the number of lights placed in the selected drawing. Click to explore lights; see the LIGHTLIST command.
Materials	Reports the number of materials defined in the selected drawing. Click to explore materials; see the MATERIALS command.
Render Presets	Reports the number of render presets defined in the drawing. Click to explore render presets; see the RENDERPRESETS command.
Blocks	Reports the number of blocks defined in the selected drawing. Click to explore blocks; see the EXPBLOCKS command.
External References	Reports the number of external references (xrefs) attached to the selected drawing. Click to explore xrefs; see the EXPXREFS command.
Images	Reports the number of images attached to the selected drawing. Click to explore the images; see the EXPIMAGES command.
PDF Underlays	Reports the number of PDF underlays attached to the selected drawing. Click to explore PDF underlays; see the EXPPDFS command.

Categories	Description
Point Clouds	Reports the number of point clouds attached to the selected drawing. Click to explore point cloud; see the POINTCLOUD command.
Dependencies	Reports the number of external files that the selected drawing depends on. Click to explore dependencies; see the ETRANSMIT command.
Page Setups	Reports the number of the page setups attached to the selected drawing. Click to explore page setups; see the PAGESETUP command.
Section Planes	Reports the number of section planes defined in the selected drawing. Click to explore the section planes; see the SECTIONPLANESETTINGS command.
View Detail Styles	Reports the number of view detail styles defined in the selected drawing. Click to explore the view detail styles; see the VIEWDETAILSTYLE command.
View Section Styles	Reports the number of view section styles defined in the selected drawing. Click to explore the view section styles; see the VIEWSECTIONSTYLE command.
Broken Symbol Styles	Reports the number of available styles for the VIEWBREAK command. Click to explore the broken symbol styles.

34.4.8.2 The Drawing Explorer layout

The **Drawing Explorer** window consists of a number of sub-windows or panes.

You can select which panes you want to have open or closed in the **View** menu. BricsCAD remembers the display settings for each category.

Item	Description
Drawing	Displays the **Drawing** pane. • **Open Drawings**: a list of all drawings that are currently open. • **Folders**: - **Local Folders**: your favorite local drawing folders. - **Bricsys 24/7**: your drawing folders on Bricsys 24/7.
Details	Displays the **Details** pane. Shows the details of the selected drawing or the details of the selected category.
Preview	Displays the **Preview** pane. Shows a preview of the selected drawing, layer, block, view, dimension style, table style, XRef, image or material.

Item	Description
Editor	Displays the **Editor** pane. Allows you to define or edit the properties of the selected category.
Status bar	Displays the Status bar at the bottom of the **Drawing Explorer** dialog box.

	View options for the Details pane
Details	Displays the details of definitions in a grid.
Icons	Displays definitions as an array of icons; applies only to blocks and external references.
Tree	Displays definitions in tree form; applies only to layer states, data links, dimension styles, external references, images, PDF underlays and dependencies.

Regen	Updates the drawing display to reflect changes made in this dialog box.
Regen at each change	Toggles the automatic updating of the display at each change.

Hide Xref symbols	Do not display definitions in Xrefs.

Note:

- If you click a definition category of an open drawing in the **Drawings** sub-window when the **Details** sub-window is closed, the **Details** window is re-opened, showing the content of the selected category.
- To close a sub-window, click the close button ( ) in its title bar.
- To restore the default layout, choose **Restore Default Explorer Layout** in the **Settings** menu.

34.4.8.3 The Settings menu

Item	Description
Restore Explorer Default Layout	Restores the layout of the Drawing Explorer to the default layout for the selected category. When you make changes to the layout of the Drawing Explorer, the program remembers them. This option resets all your changes.

Item	Description
Options	Applies to Blocks and External References only. Specifies default settings for inserting blocks. See the EXPBLOCKS command.

34.4.8.4 The Details toolbar

Icon	Name	Shortcut	Description
	New	Ctrl + N	Creates a new definition with a generic name, such as NewLayer 1 or NewStyle1. Use the Rename button to rename it.
	Delete	Ctrl + D	Erases the selected definition from the drawing. When the item is in use, such as a layer with entities, you have to option to retain the entities from the layer to be erased. Some items cannot be erased, such as layer 0, Global material, visual style 2D Wireframe, ...
	Purge	Ctrl + P	Purges unused definitions from the drawing. Some definitions cannot be erased, such as layer 0 and text style Standard. See the PURGE command.
	Cut	Ctrl + X	Copies the selected definition(s) to the Clipboard, and then deletes it from the drawing following the same procedure as the Delete tool.
	Copy	Ctrl + C	Copies the definition(s) to the Clipboard for pasting elsewhere in the Drawing Explorer.
	Paste	Ctrl + V	Pastes the definition(s) from the Clipboard, into the Drawing Explorer. When the item on which the Clipboard content will be pasted is in use, such as a layer with entities, you have options: • **Copy and Replace** - replaces the existing item with the pasted one • **Don't Copy** - cancels the copy operation • **Copy, but keep both** - pastes the Clipboard content with an incremented name, usually suffixed by a 1 • **Cancel** - also cancels the operation
	Details View		Displays the details of definitions in a grid.
	Icon View		Displays definitions as an array of icons; applies only to blocks and external references.

Icon	Name	Shortcut	Description
	Tree View		Displays definitions in tree form; applies only to layer states, data links, dimension styles, external references, images, PDF underlays and dependencies.
	Regen		Updates the drawing display to reflect changes made in this dialog box.
	Toggle Regen		Toggles the automatic updating of the display at each change.

34.4.8.5 The Context menu

Select a definition in a category, then right-click.

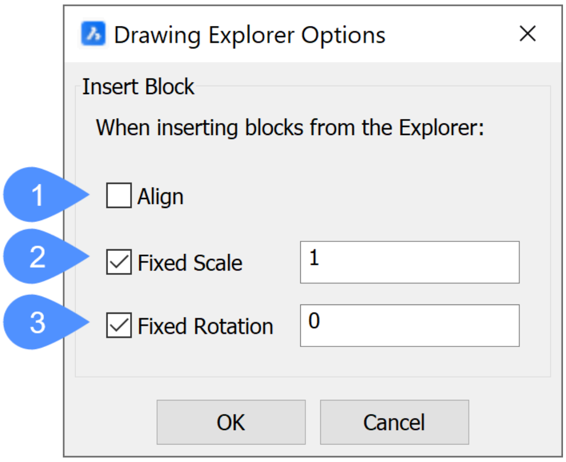
Item	Shortcut	Description
New	Ctrl + N	Creates a new definition as a copy of the selected definition.
Delete	Ctrl + D	Deletes the selected definitions. Definitions that are in use cannot be deleted.
Purge	Ctrl + P	Deletes unused definitions in the current selection.
Cut	Ctrl + X	Cuts the selected definitions to the clipboard.
Copy	Ctrl + C	Copies the selected definitions to the clipboard.
Paste	Ctrl + V	Pastes definitions from the clipboard.
Rename	F2	Renames the selected definition; alternatively, click on the name twice, slowly, to rename it.
Select All	Ctrl + A	Selects all definitions in the current category.
Invert Selection		Inverts the selected definition(s), for example, select a few layers, then use this to select all other layers.
Set current		Makes the definition current, such as making the selected layer the current (working) layer.

Item	Shortcut	Description
Bind		Binds the selected definition(s) to the drawing.

34.4.9 Drawing Explorer Options dialog box

Opens via: **Drawing Explorer** dialog box

The **Drawing Explorer Options** dialog box allows you to align or set the scale and the rotation of the inserted blocks.



1 Align

2 Fixed Scale

3 Fixed Rotation

34.4.9.1 Align

Tick the checkbox to align blocks with the snapped entity.

34.4.9.2 Fixed Scale

Tick the checkbox to specify the scale factor to be used for all block insertions.

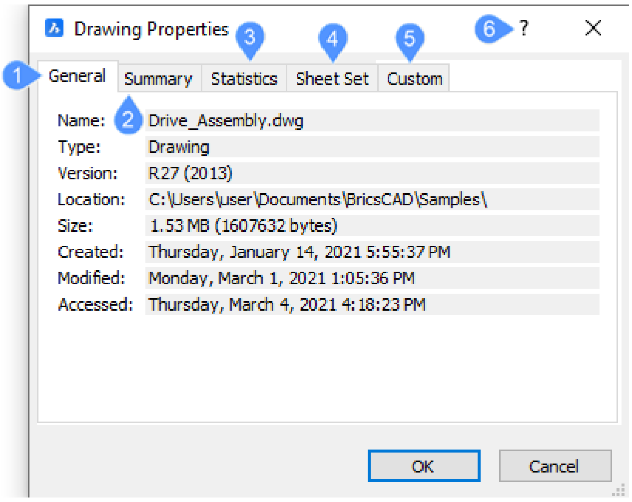
34.4.9.3 Fixed Rotation

Tick the checkbox to specify the rotation angle to be used for all block insertions.

34.4.10 Drawing Properties dialog box

Opens via: DWGPROPS command

The **Drawing Properties** dialog box allows you to view and modify general information about the drawing and user-defined properties.



1 General

2 Summary

3 Statistics

4 Sheet Set

5 Custom

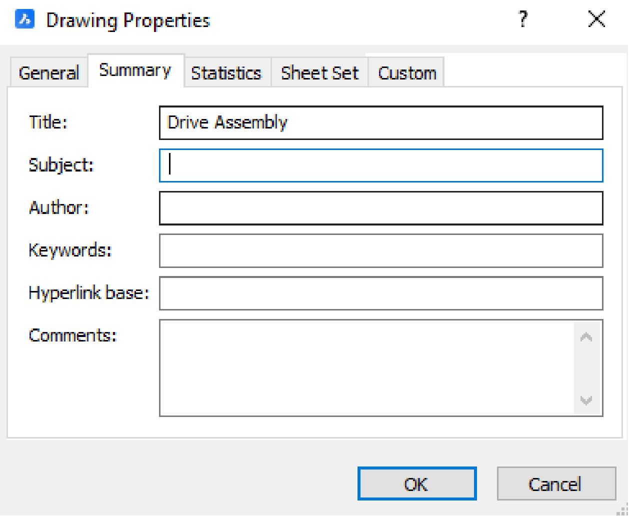
6 Command reference

34.4.10.1 General

Displays the general properties of the drawing, such as creation date and time.

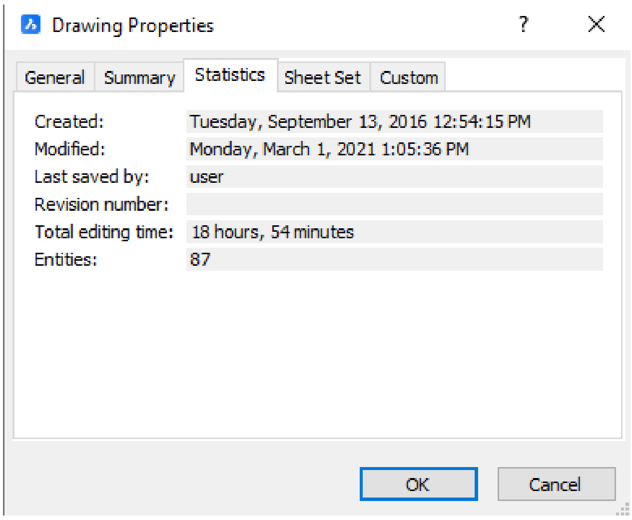
34.4.10.2 Summary

Lets you define the summary properties of the drawing, such as subject and keywords. This data can be searched for by EDMS (engineering data management software) to find drawings in large collections.



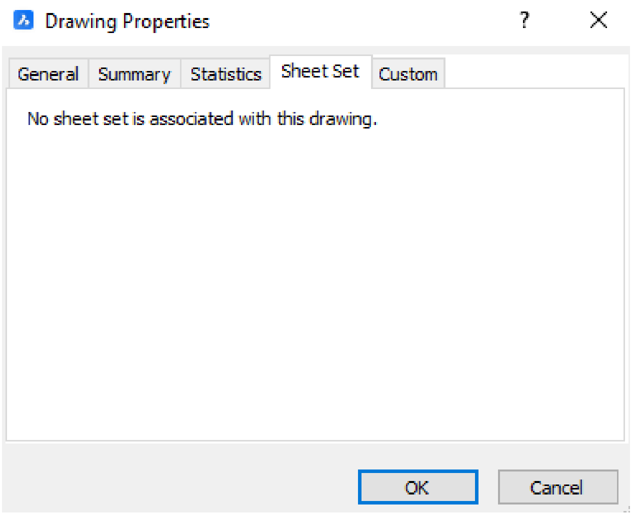
34.4.10.3 Statistics

Displays statistical information about the drawing, such as the creation and modification time.



34.4.10.4 Sheet Set

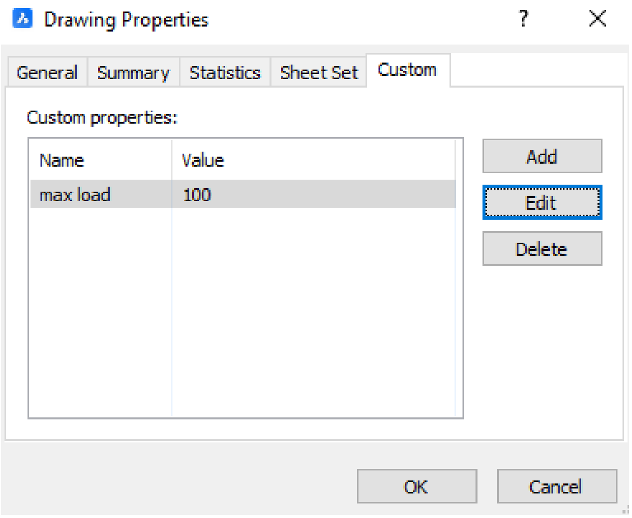
Displays sheet set data when the drawing is associated with a sheet set.



34.4.10.5 Custom

Displays the custom properties of the drawing. Click the **Add** button to add a new property or the **Edit** button to edit a property.

The **Custom Property** dialog box appears when clicking the **Add** and **Edit** buttons.



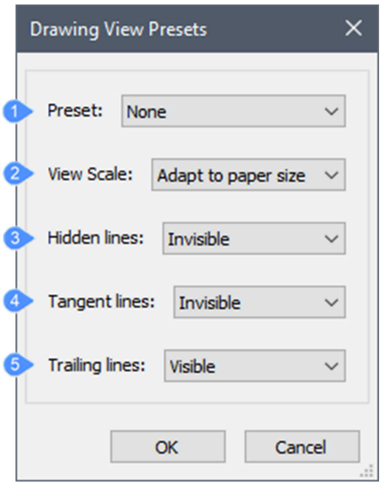
34.4.10.6 Command reference

Opens the Bricsys Help article about the DWGPROPS command.

34.4.11 Drawing View Presets dialog box

Opens via: VIEWBASE command > **Presets** option

Specifies the types of generated drawings and their placement in the layout.



1 Preset

2 View Scale

3 Hidden lines

4 Tangent lines

5 Trailing lines

34.4.11.1 Preset

Select a preset view arrangement:

- **None**: no preset affects the views generated.
- **Architectural Views**: generates:
    - Five standard views: **Front**, **Right**, **Back**, **Left** (used for elevations) and **Top** views.
    - Three section views: two horizontal sections (for plans) and 1 vertical section.
- **Mechanical Views**: generates:
    - Three standard views: **Front**, **Top** and **Left** (first angle projection) or **Right** (third angle projection).

The layout of the mechanical views is controlled by the projection type: first or third angle. The current projection type is saved in the PROJECTIONTYPE system variable.

34.4.11.2 View Scale

Sets the default scale factor for the drawing views from drop-down list. The scale list can be edited by the SCALELISTEDIT command.

34.4.11.3 Hidden lines

Sets the default setting for hidden lines in drawing views.

34.4.11.4 Tangent lines

Sets the default for **Tangent Lines** in drawing views.

34.4.11.5 Trailing lines

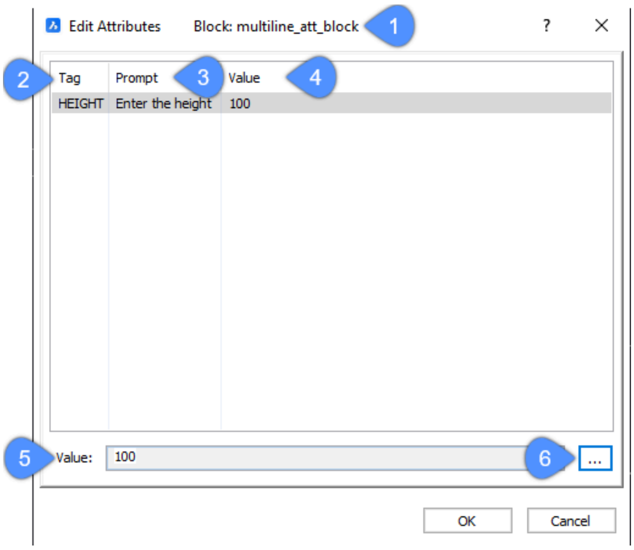
Sets the default for **Trailing Lines** in drawing views.

34.5 E

34.5.1 Edit Attributes dialog box

Opens via: DDATTE command

The **Edit Attributes** dialog box allows you to edit the values of attributes.



1 Block name

2 Tag

3 Prompt

4 Value

5 Value

6 Text Formatting

34.5.1.1 Block name

Displays the name of the selected block.

34.5.1.2 Tag

Specifies the name of the attribute.

Note: This is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers, and punctuation.

34.5.1.3 Prompt

Specifies the user prompt.

Note: This is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank, BricsCAD uses the tag as the prompt during attribute insertion.

34.5.1.4 Value

Displays the value that belongs to a certain attribute.

34.5.1.5 ... Text Formatting

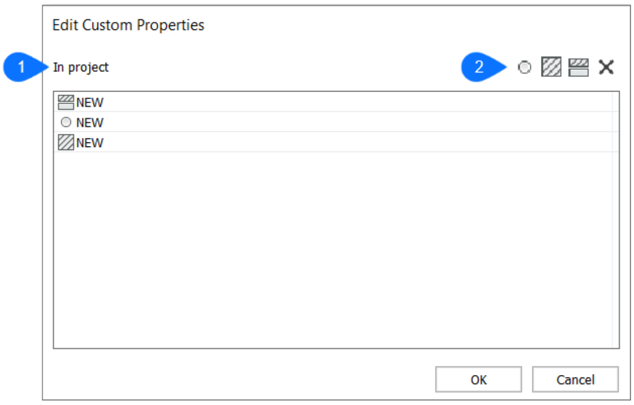
Changes the value of the attribute text.

Note: To be able to use this option, the Multiple lines attribute flag must be selected, in the **Define Attribute** dialog box, when defining an attribute.

34.5.2 Edit Custom Properties dialog box

Opens via: **BIM Project Info** dialog box

The **Edit Custom Properties** dialog box allows you to add custom properties to materials and compositions.



1 In project

2 Add Generic/ Material/Composition/Delete properties

34.5.2.1 In project

Displays the custom properties available in the project.

34.5.2.2 Add/Delete Properties

Add Generic Custom Property
Creates a new generic custom property, which can be assigned to building elements in the project.

Add Material Custom Property
Creates a new material custom property, which can be assigned to materials in the project.

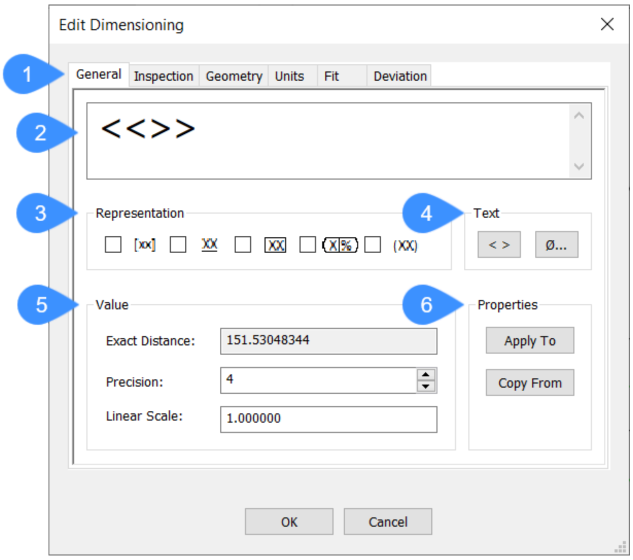
Add Composition Custom Property
Creates a new composition custom property, which can be assigned to compositions in the project.

Delete
Deletes the selected property.

34.5.3 Edit Dimensioning dialog box

Opens via: AMPOWEREDIT command

The **Edit Dimensioning** dialog box allows you to edit the properties of a dimension.



1 Editor tabs

2 Graphical representation

3 Representation

4 Text

5 Value

6 Properties

34.5.3.1 General

Graphical representation

Represents how the dimension text will look like.

Representation

Defines how the dimension is represented.

Note: A tooltip with explanation about the representation appears when you hover over the checkboxes of the options.

Text

Adds text elements to the displayed dimension text. The left symbol toggles on and off whether the dimension value will be displayed and you can select **Special Characters** from the **Special Characters** dialog box if you click the character box.

Value

Specifies the settings of the value that is displayed.

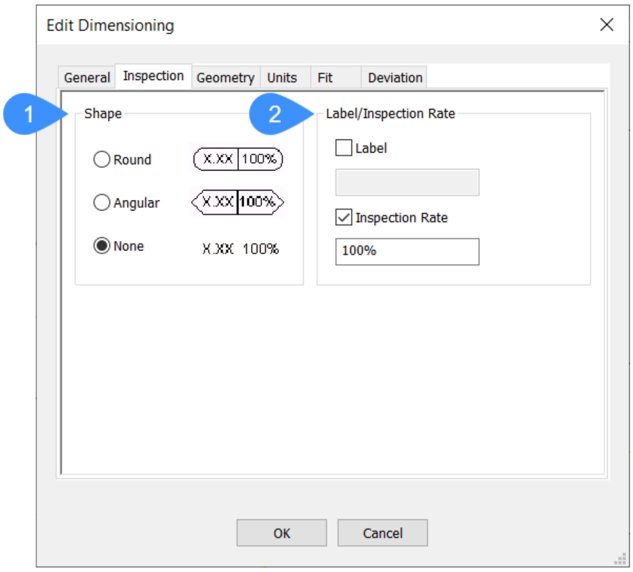
- **Exact Distance**: displays the exact distance of the selected dimension.
- **Precision**: specifies the number of decimal digits that are displayed. By default this is set to 2.

- **Linear Scale**: specifies the linear scale of the dimension value. By default this is set to 1.

Properties

- **Apply to**: allows you to apply the specified properties to other entities. Click the **Apply to** button, then select the entities you want to apply the dimension settings to. The **Properties** dialog box opens, where you can select the properties that you want to apply to the selected entities.
- **Copy From**: allows you to copy properties from a selected entity to the **Edit Dimensioning** dialog box. click the **Copy From** button and select an entity you want to copy properties from. This opens the **Properties** dialog box, where you can select the properties that you want to copy to the **Edit Dimensioning** dialog box.

34.5.3.2 Inspection



1 Shape

2 Label/Inspection Rate

Shape

Defines the shape of the frame around the dimension text. You can toggle between the options.

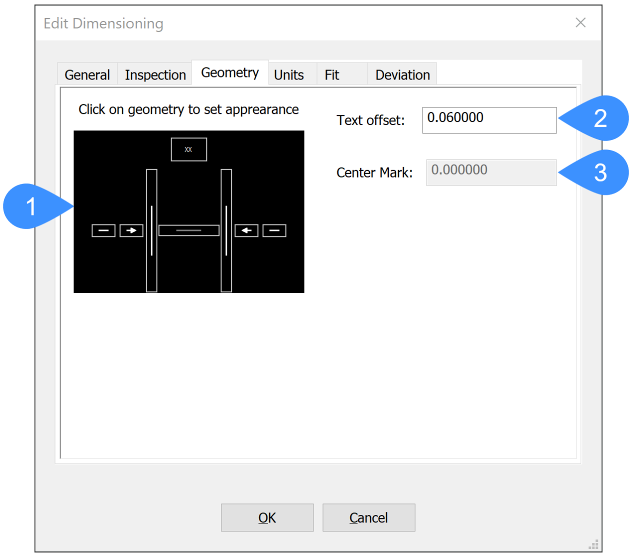
Label/Inspection Rate

Specifies the settings for the inspection dimension.

- **Label**: toggles the label on or off which identifies individual inspection dimensions. You can type a label in the Label box if Label is toggled on.
- **Inspection Rate**: toggles the display of the **Inspection Rate** on or off. You can enter an inspection rate, this is the frequency the part is inspected with.

  Note: The inspection rate is expressed as a percentage.

34.5.3.3 Geometry



1 Geometry

2 Text offset

3 Center Mark

Geometry

Displays how the dimension will look like. You can toggle on and off the First Dimension line, the First Extension Line, the Second Extension Line, the Second Dimension Line, Display Center and Force Line Inside.

If you click the Dimension Text, the dialog box switches to the **General** tab where you can edit how the dimension text is displayed.

If you click an arrowhead, the **Arrowhead Dialog** dialog box is opened that allows you to select the type of arrowhead that is displayed.

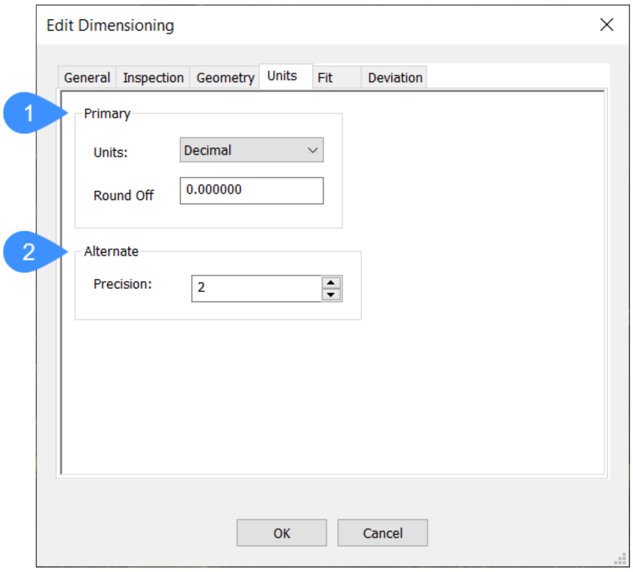
Text offset

Specifies the text offset from the dimension line.

Center Mark

Specifies the size of the Center Mark. This option is only enabled with a radius or diameter dimension, when the Display Center option is toggles on in the Geometry box.

34.5.3.4 Units



1 Primary

2 Alternate

Primary

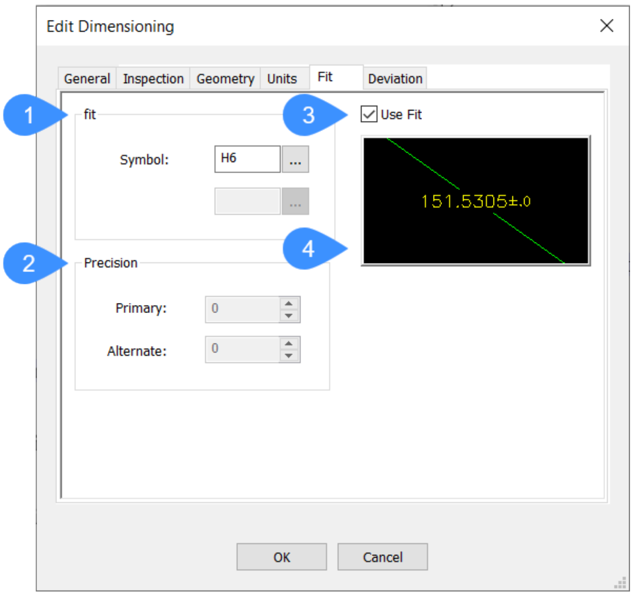
Specifies the properties for the primary units.

- **Units**: opens a drop-down list where you can select the unit type. By default this is set to Decimal.
- **Round Off**: specifies the number of decimals to round off the dimension values.

Alternate

Specifies the properties for alternate units. The precision sets the number of decimals for the alternate units.

34.5.3.5 Fit



1 Fit

2 Precision

3 Use Fit

4 Preview

Fit

Sets the symbol to display the fit. This is only displayed if the **Use Fit** option is toggled on. Click the three dots to open the **Fits** dialog box to add fits to dimensions.

Precision

Sets the number of decimals of the Primary and Alternate units.

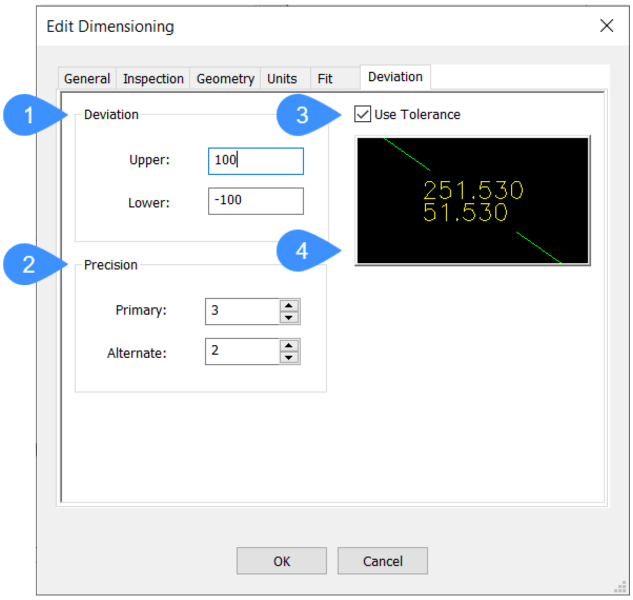
Use Fit

Toggles on and off whether the fit is displayed in the dimension text.

Preview

Displays how the fit is displayed in the dimension text.

34.5.3.6 Deviation



1 Deviation

2 Precision

3 Use Tolerance

4 Preview

Deviation
Sets the upper and lower deviation limit between the nominal value and the lower and upper tolerance value that are displayed.

Precision
Sets the number of decimals of the Primary and Alternate units.

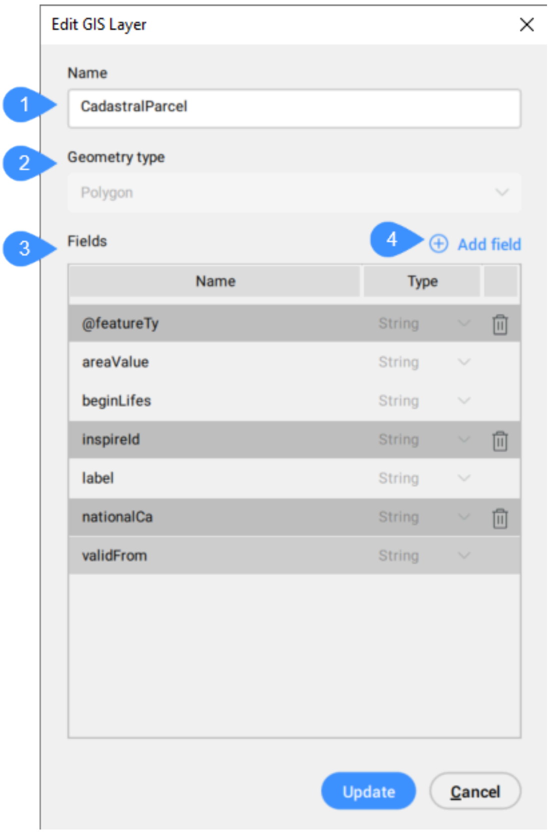
Use Tolerance
Toggles on and off whether the dimension text is displayed as a tolerance value.

Preview
Displays how the tolerance values are displayed as dimension text.

34.5.4 Edit GIS Layer dialog box

Opens via: **Civil Explorer** panel

The **Edit GIS Layer** dialog box allows you to add, edit, or delete fields from an existing GIS layer.



1 Name

2 Geometry type

3 Fields

4 Add field

Name
Allows you to change the name of the GIS layer.

Geometry type
Allows you to change the geometry type of the GIS layer.

Fields
Allows you to change the name and type of a field or remove it.

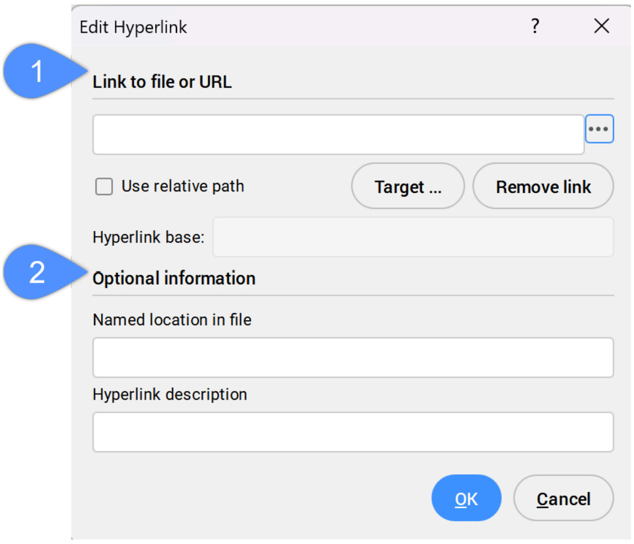
Note: Multiple selection is accepted by using **Shift** or **Ctrl** keys.

Add field
Allows you to add a field for GIS layer.

34.5.5 Edit Hyperlink dialog box

Opens via: HYPERLINK command

The **Edit Hyperlink** dialog box allows you to add or edit a hyperlink to a file or a web page.



1 Link to file or URL

2 Optional information

34.5.5.1 Link to file or URL

Specifies the link to a file or a URL to a web page.

- Click the Browse button (...) to open the **Select file** dialog box. Then, select the file and click **Open** to add the link to the **Link to file or URL** field.
- Type a URL to a web page in the **Link to file or URL** field.

Note: To remove the attached hyperlink, click the **Remove link** button.

Use relative path
If the file path in the **Link to file or URL** filed is absolute, the **Use relative path** option is greyed out.
You can manually shorten the file path of the attached hyperlink to make it relative. The **Use relative path** option becomes available and it is automatically ticked.

Hyperlink base
Displays the file path for all relative hyperlinks in the drawing.

Note: The base path is set by the **HYPERLINKBASE** system variable.

34.5.5.2 Optional information

Specifies additional information.

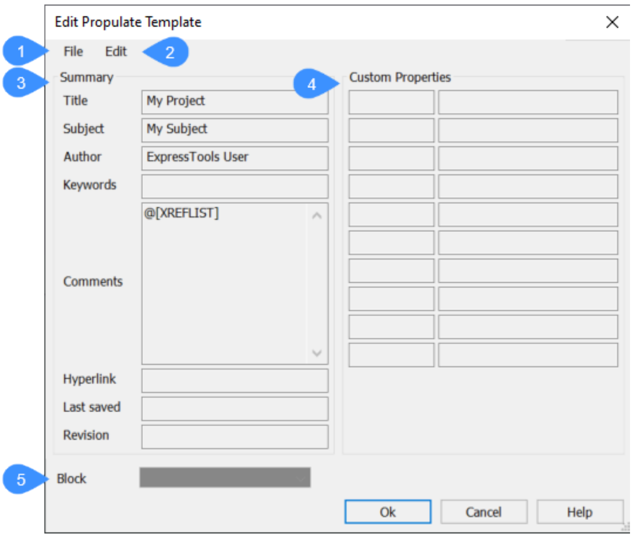
Named location in file
Specifies a bookmark name. The # character specifies locations (bookmarks) within files or web pages.

Hyperlink description
Adds an additional description of the hyperlink.

34.5.6 Edit Propulate Template dialog box

Opens via: PROPULATE command (Express Tools)

Allows you to save, load and edit a propulate template.



1 File

2 Edit

3 Summary

4 Custom Properties

5 Block

34.5.6.1 File

Displays a menu to save or open new propulate template files:

New...
Clears all the fields of the current propulate template.

Open...
Opens the **Select Propulate Template File** dialog box to open a propulate template file.

Save...
Saves the current propulate template.

Save As...
Opens the **Select Propulate Template File** dialog box to save the propulate template file.

34.5.6.2 Edit

Displays a menu to insert lists or adjust the template fields:

Insert Xref list
Adds the **@[XREFLIST]** syntax at the current cursor location.

Insert Font list
Adds the **@[FONTLIST]** syntax at the current cursor location.

Insert Image list
Adds the **@[IMAGELIST]** syntax at the current cursor location.

Insert Attribute
Adds the selected value in the **Block Attributes** section at the current cursor location.

Make active template
Sets the currently edited template as the default template.

Fill from Current Drawing
Clears all edited fields and fills them with the **Drawing Properties** values from the current drawing.

34.5.6.3 Summary

Lists all standard elements of the propulate template.

Title
Specifies the title of your project.

Subject
Specifies the subject of your project.

Author
Specifies the author of your project.

Keywords
Keywords concerning your project.

Comments
Allows you to add comments to your projects.

Hyperlink
Links a hyperlink to your drawing.

Last saved
Specifies the person or institution that last saves the project.

Revision
Specifies the revision number.

34.5.6.4 Custom Properties

Allows you to add custom properties by defining the **Name** and the **Value**.

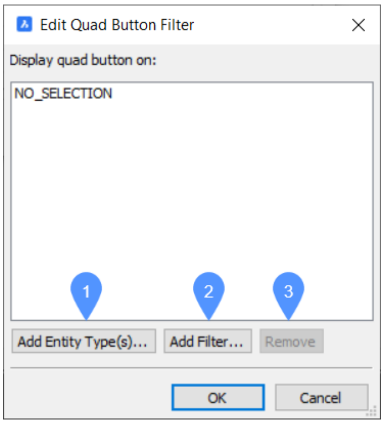
34.5.6.5 Block

Lists the values of block attributes from the current drawing.

34.5.7 Edit Quad Button Filter dialog box

Opens via: **Customize** dialog box

The **Edit Quad Button Filter** dialog box allows you to edit the filters of a quad button (see the **Customizing the Quad** article).



1 Add Entity Type(s)

2 Add Filter

3 Remove

34.5.7.1 Add Entity Type(s)

Opens the **Add Entity Alias** dialog box.

34.5.7.2 Add Filter

Opens the **Add quad button filter** dialog box.

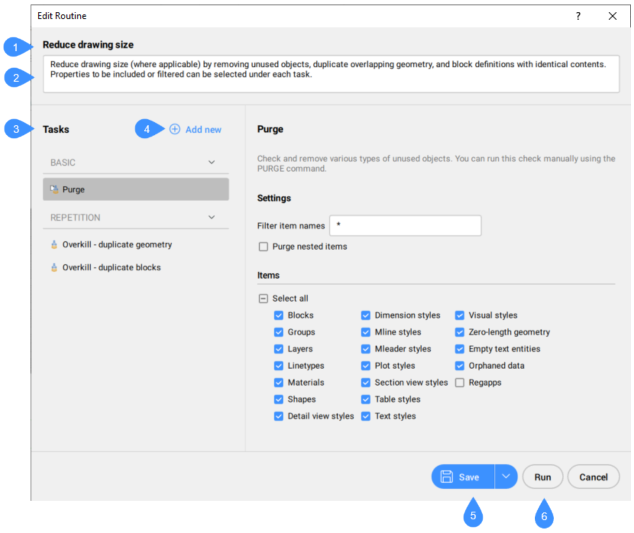
34.5.7.3 Remove

Deletes the highlighted filter that is highlighted in the list.

34.5.8 Edit Routine dialog box

Opens via: **Drawing Health** command context panel

The **Edit Routine** dialog box allows you to edit the tasks to be performed within a routine during the DWGHEALTH command.



1 Routine name

2 Routine description

3 Tasks

4 Add new

5 Save

6 Run

34.5.8.1 Routine name

Displays the name of the routine that is being edited.

34.5.8.2 Routine description

Allows you to add a description of the routine. This can provide more context about the purpose of a routine, communicate instructions to coworkers, etc.

34.5.8.3 Tasks

Lists the tasks selected for the routine to be performed during the DWGHEALTH command.

34.5.8.4 Add new

Adds new tasks to the routine from the drop-down list.

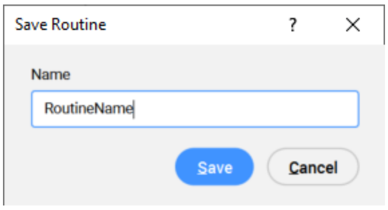
34.5.8.5 Save

Saves the routine.

Press the arrow from the right to open a drop-down list with two options for saving:

Save: saves the routine with the same name.

Save as: opens the **Save Routine** dialog box to save the routine with a different name.



34.5.8.6 Run

Allows you to run the edited routine under the **New Routine*** name, without saving it.

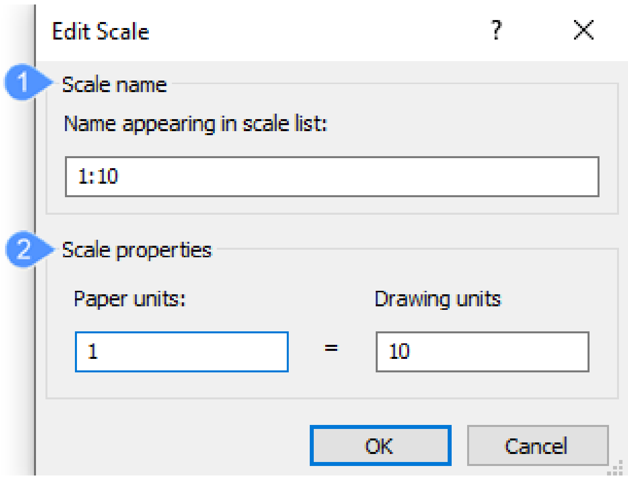
Note: The **Drawing Health** command context panel displays a **Save as** button after you run an unsaved routine via the **Run** button in the **Create Routine** dialog box or **Edit Routine** dialog box. Click it to open the **Save Routine** dialog box and save the custom routine with the desired name.

34.5.9 Edit Scale dialog box

Opens via:

- **Default Scale List** dialog box
- **Edit Scale List** dialog box

The **Edit Scale** dialog box allows you to edit the name and the scale properties of a scale factor.



1 Scale name

2 Scale properties

34.5.9.1 Scale name

Edit the name by typing the new name.

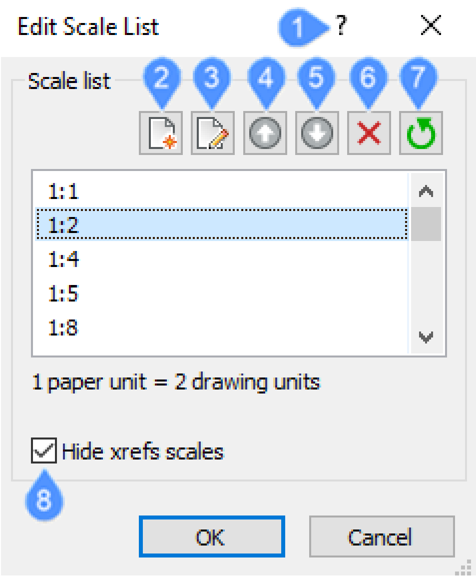
34.5.9.2 Scale properties

Change the scale factor in paper (size when plotted) and/or drawing units.

34.5.10 Edit Scale List dialog box

Opens via: SCALELISTEDIT command

The **Edit Scale List** dialog box allows you to add, edit and delete scales.



1 Command reference

2 Add a new scale

3 Edit scale

4 Move up

5 Move down

6 Delete the selected scale

7 Reset

8 Hide xrefs scales

34.5.10.1 Command reference

Opens the Bricsys Help article about the SCALELISTEDIT command.

34.5.10.2 Add a new scale

Opens the **Add scale** dialog box.

34.5.10.3 Edit scale

Opens the **Edit Scale** dialog box.

34.5.10.4 Move up

Moves the selected scale up the list.

34.5.10.5 Move down

Moves the selected scale down the list.

34.5.10.6 Delete the selected scale

Deletes the selected scale. Press the **Ctrl** key to select and delete more than one scale at a time.

Note that the scale 1:1 cannot be deleted.

34.5.10.7 Reset

Returns the list to its original condition, adding back in the deleted scales and removing all added but unused scales.

34.5.10.8 Hide xrefs scales

Determines if the scale factors from xrefs are displayed in the list:

- **Yes**: scale factors from externally referenced drawings are not shown in the list.
- **No**: scales in the attached xrefs not in the current drawing are listed here.

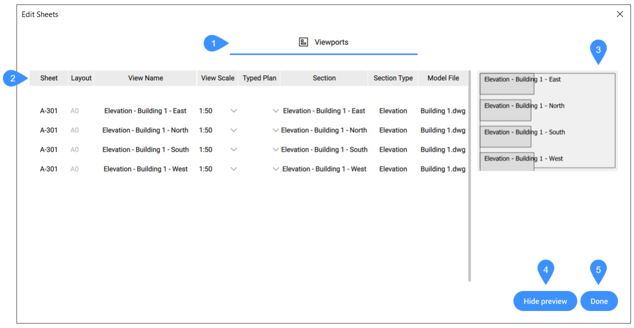
Toggles the HIDEXREFSCALES variable.

34.5.11 Edit Sheets dialog box

Opens via: **BIM Project Browser** panel > right-click a sheet > select **Edit Sheets** option

The **Edit Sheets** dialog box allows you to edit one or multiple sheets.

Note: If there are no sheets created yet, you can go to the **Sections and views** tab in the **BIM Project Browser** panel, right-click on a section and select **Create Sheets**.



1 Tabs

2 Viewports tab options

3 Preview

4 Hide preview

5 Done

34.5.11.1 Tabs

Viewports

Allows to set the scale and typed plan of the viewports.

34.5.11.2 Viewports tab options

Sheet
Displays the sheet name.

Note: You cannot change the name in the **Viewports** tab. You can change it in the **Sheets** tab of the **BIM Project Browser** panel, **Properties** section.

Layout
Displays the layout that is used for the selected sheet.

View Name
Displays the name of the view.

Note: You cannot change the name in the **Viewports** tab. You can change the name by right-clicking a section and selecting rename. This opens the **Choose name** dialog box where you can change the name.

View Scale
Opens a drop-down menu where you can select a predefined view scale, or select **Edit scalelist...**. This opens the **Edit Scale List** dialog box where you can create or edit scales.

Typed Plan
Opens a drop-down menu from which you can assign a typed plan to any viewport.

Note: Typed plans can be added in the **File** tab of the **BIM Project Browser** panel. Afterwards they will appear in the drop-down menu in the **Viewports** tab.

Section
Displays the section name. You cannot change the name in the **Viewports** tab. You can change the name by right-clicking a section and selecting rename. This opens the **Choose name** dialog box where you can change the name.

Section Type
Displays the section type.

Model File
Displays the model file name in which the section is located.

34.5.11.3 Preview

Shows a preview of the sheet that is edited.

34.5.11.4 Hide Preview

Hides the preview of the sheet that is edited. After clicking the button, it turns to **Show preview** to display the preview.

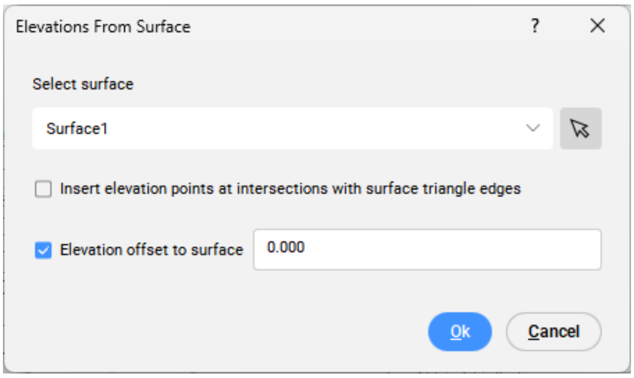
34.5.11.5 Done

Closes the **Edit Sheets** dialog box.

34.5.12 Elevations From Surface dialog box

Opens via: STRINGELEVATIONFROMSURFACE command

The **Elevations From Surface** dialog box allows you to set the String elevations from TIN surface.



34.5.12.1 Select surface

Displays the selected surface. You can select a surface from the drop down list or click to pick it from the drawing area.

34.5.12.2 Insert elevation points at intersections with surface triangle edges

If ticked, inserts the elevation point at each point where Civil String crosses a TIN surface triangle edge.

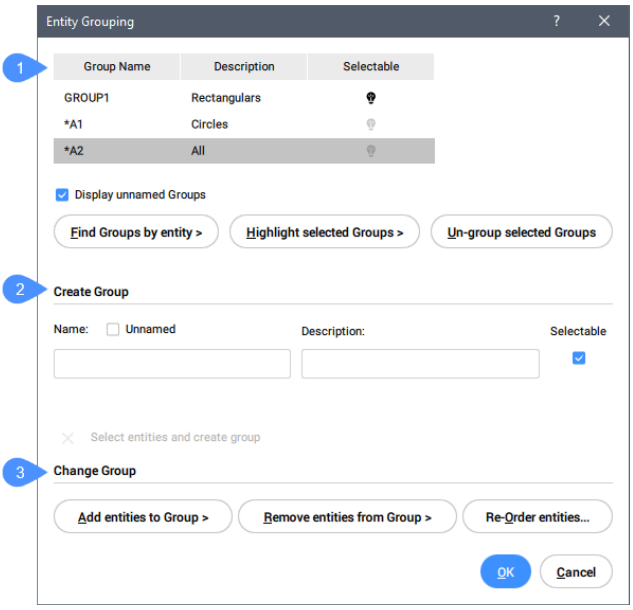
34.5.12.3 Elevation offset to surface

If ticked, you are able to define whether Civil String entity is created above or below the TIN surface elevation by specifying a value for an elevation offset to the surface.

34.5.13 Entity Grouping dialog box

Opens via: GROUP command

The **Entity Grouping** dialog box allows you to view, create, modify and delete named groups of entities in the current drawing.



1 Existing groups

2 Create Group

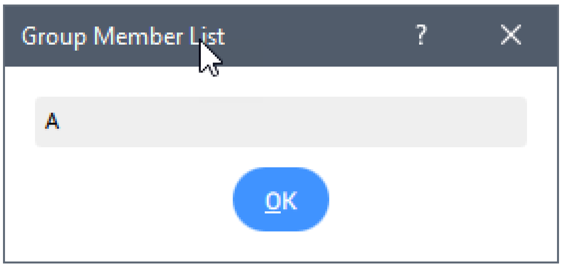
3 Change Group

34.5.13.1 Existing groups

Lists all existing groups with the Group Name and a short description.

Display unnamed Groups
Toggles the displaying of unnamed groups.

Find Groups by entity
Reports the name(s) of groups in the **Group Member List** dialog box when entities in the drawing are selected.



Highlight selected Groups
Highlights all entities that belong to the selected group.

Un-group selected Groups
Removes the group status from selected entities.

34.5.13.2 Create Group

Name
Specifies the name of a group.

Unnamed
Toggles whether the group has a specific name.

Description
Adds an optional description of the group.

Selectable
Determines how groups are selected when the PICKSTYLE system variable is set to 1 or 3.

Select entities and create group
Selects entities in the workspace which should be part of the group by pressing the right side cross button.

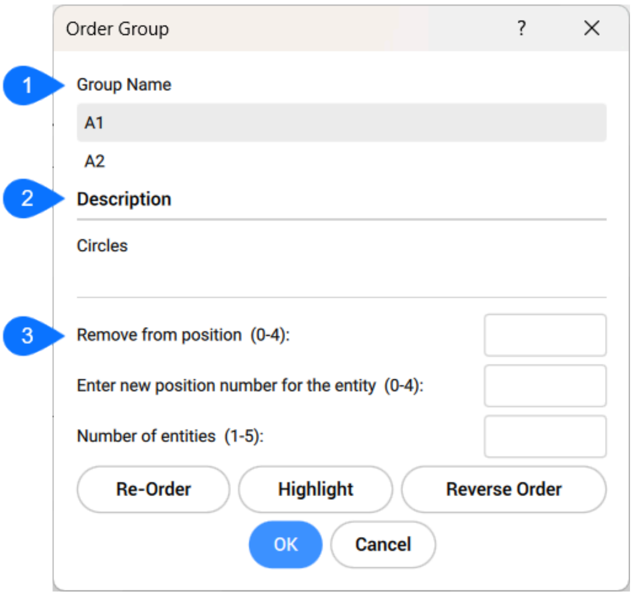
34.5.13.3 Change Group

Add entities to group
Adds entities to the selected group.

Remove entities from group
Removes entities from the selected group.

Re-Order entities

Changes the order of entities in groups through the **Order Group** dialog box.



1 Group Name

2 Description

3 Options

Group Name
Lists the names of named and unnamed groups.

Description
Displays the description of the selected group.

Options
Specifies several options

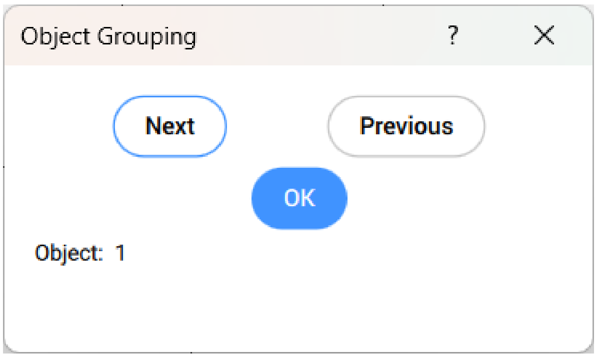
Remove from position
Specifies the position number of the entity to be reordered.

Enter new position number for the entity
Specifies the new position number for the group.

Number of entities
Specifies the range of entities to reorder.

Re-order
Applies the reordering dictated by the previous three fields.

Highlight
Highlights entities in the group one by one by pressing the **Next** and **Previous** button from the **Object Grouping** dialog box.
Pressing **OK** button returns to the **Order Group** dialog box.



Reverse Order

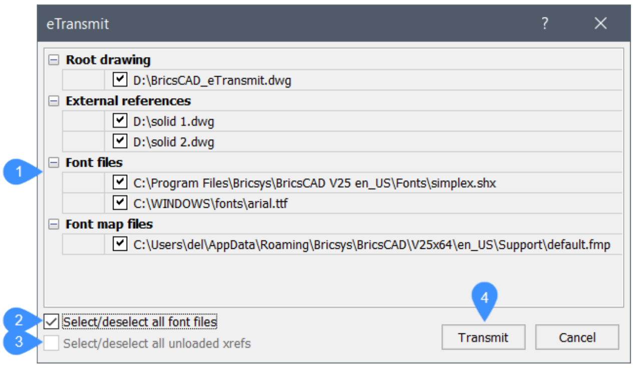
Reverses the order of entities in the group.

34.5.14 eTransmit dialog box

Opens via:

- ETRANSMIT command
- **Sheet Sets** panel

The **eTransmit** dialog box creates a package consisting of the drawing file and all its dependencies, such as external references, images, font files, plot configuration files, plot style tables, and font map files.



1 Available files

2 Select/deselect all font files

3 Select/deselect all unloaded xrefs

4 Transmit

34.5.14.1 Available files

Select the files to be included in the eTransmit package (usually all of them).

34.5.14.2 Select/deselect all font files

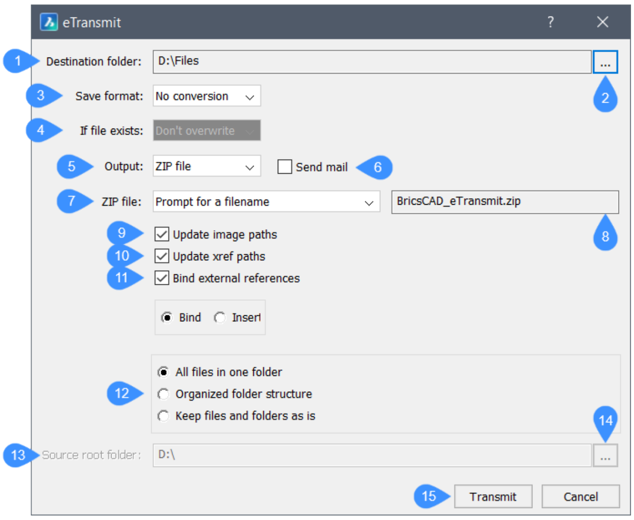
Tick the checkbox to automatically select all font files.

34.5.14.3 Select/deselect all unloaded xrefs

Tick the checkbox to automatically select all xref files.

34.5.14.4 Transmit

Opens the following dialog box:



1 Destination folder

2 Browse (destination folder)

3 Save format

4 If file exists

5 Output

6 Send mail

7 ZIP file

8 ZIP file name

9 Update image paths

10 Update xref paths

11 Bind external references

12 Path options

13 Source root folder

14 Browse (root folder)

15 Transmit

Destination folder

Specifies the output folder for the eTransmit package. If the destination folder does not exist, you are prompted to create the folder.

Browse (destination folder)

Opens the **Select the destination folder** dialog box to allow you to choose the destination folder for the transmitted package.

Save format

Specifies the format of the drawing files in the eTransmit package.

If file exists
Specifies what to do if a file with the same name already exists in the output folder.
This option is available when **Folder** is chosen as output.
Select an option from the drop-down list:

- **Overwrite**: replaces the existing files with new ones.
- **Don't overwrite**: all files are transmitted, except files with the same file name as an existing file in the folder.
- **Abort**: the operation is aborted and no files are saved if a file with the same name already exists in the destination folder.

Output
Specifies whether to create a ZIP file or copy the files to the output folder.

Send mail
Tick the checkbox to send a mail with the zipped eTransmit package in attachment using your default mail application.
This option is available when **ZIP file** is chosen as output.

ZIP file
Specifies what to do when a ZIP file with the same name already exists in the output folder.
Choose an option from the drop-down list:

- **Prompt for a file name**: opens a dialog box where you can choose a name and a path.
- **Overwrite if necessary**: replaces the existing file with the new one if a file with the same name already exists.
- **Increment name if necessary**: adds a suffix to the file name.

ZIP file name
Allows you to edit the ZIP file name.

Update image paths
Tick the checkbox to update the saved paths for image files according to the new location of the transmitted files.

Bind external references
Adds the XREF files into the drawing as block references. Subsequently, only the drawing itself is exported without including the Xref drawing.

Bind
Binds all DWG files external references to the files they were originally attached to. The properties of the XREF entities are kept. The XREF entity definitions are added to the drawing with a prefix.

Insert
Inserts all DWG files external references to the files they were originally attached to. The properties of the XREF entities are overwritten. The XREF entity definitions are merged into the drawing without adding prefixes.

Update xref paths
Tick the checkbox to update the saved paths to externally-referenced files according to the new location of the transmitted files.

Path options

All files in one folder

All files in the eTransmit package are copied to the destination folder.

Organized folder structure

Creates a hierarchical folder tree with subfolders for all drawing file dependencies.

Keep files and folders as is

Keeps the folder structure of all files in the eTransmit package. This procedure allows you to easily install the package on another computer using the folder structure of the source system.

Source root folder

Specifies the source root folder for relative paths of drawing-dependent files, such as xrefs or image files.

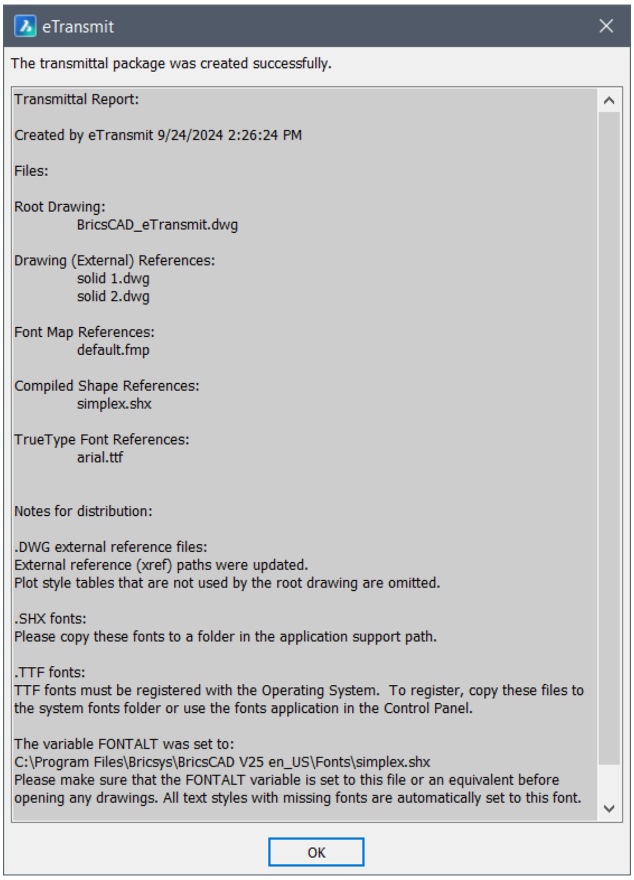
This option is available when **Organized folder structure** is checked.

Browse (root folder)

Opens the **Select the source root folder** dialog box to allow you to choose a root folder.

Transmit

Transmits the files according to the selected options and displays a report of the transmission. The report contains a complete list of all files included and notes for distribution.



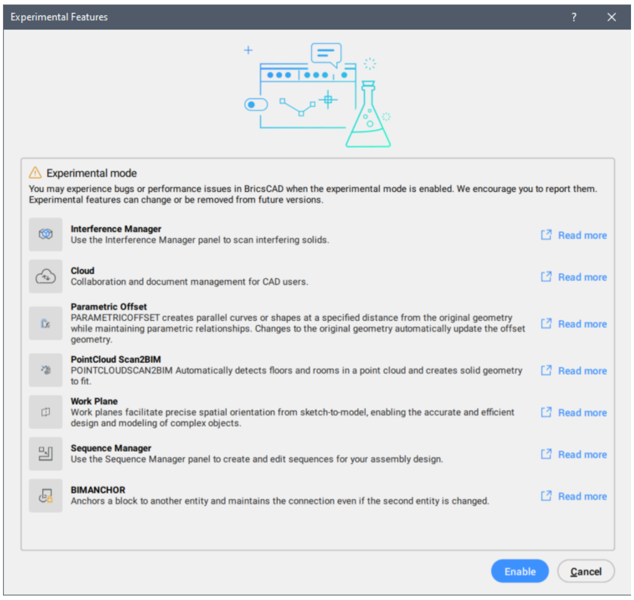
34.5.15 Experimental Features dialog box

Opens via: MANAGEEXPERIMENTALFEATURES command

The **Experimental Features** dialog box allows you to enable or disable the new features introduced as experimental.

Note:

- The **Experimental mode** is disabled by default. Enabling the mode gives you access to early features, and provides you with the opportunity to help the BricsCAD team in defining the direction of these features.
- Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.



Note: Pressing the **Read more** button opens the relevant help page.

Important: You may experience bugs or performance issues in BricsCAD when the **Experimental mode** is enabled. We encourage you to report them. Experimental features can change or be removed from future versions.

The experimental features available at the moment are:

Interference Manager
Use the **Interference Manager** panel to scan interfering solids.

Cloud
Collaboration and document management for CAD users.

Parametric Offset
PARAMETRICOFFSET creates parallel curves or shapes at a specified distance from the original geometry while maintaining parametric relationships. Changes to the original geometry automatically update the offset geometry.

PointCloud Scan2BIM
Automatically detects floors and rooms in a point cloud and creates solid geometry to fit.

Work Plane
Work planes facilitate precise spatial orientation from sketch-to-model, enabling the accurate and efficient design and modeling of complex objects.

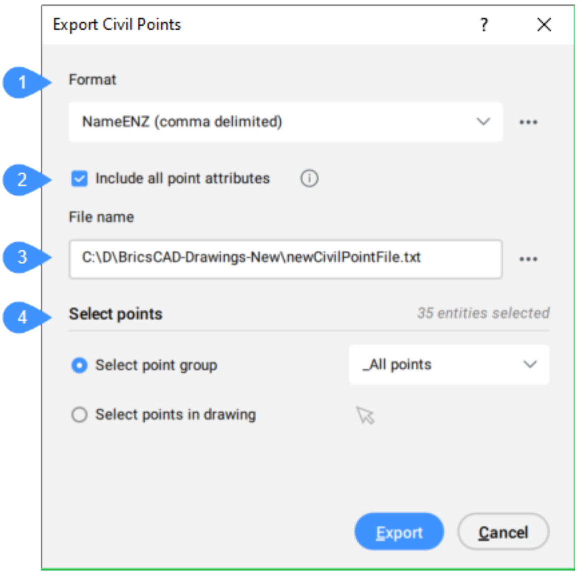
Sequence Manager
Use the **Sequence Manager** panel to create and edit sequences for your assembly design.

BIMANCHOR
Anchors a block to another entity and maintaining the connection even if the second entity is changed.

34.5.16 Export Civil Points dialog box

Opens via: CIVILPOINTEXPORT command

The **Export Civil Points** dialog box allows you to export the Civil Points to an external file.



1 Format

2 Include all point attributes

3 File name

4 Select points

Format
Allows you to select from the drop-down list a format for the exported file.

Note: Press the three points from the right side to open the **Manage Point File Formats** dialog box to create a new file format.

Include all point attributes
If checked, includes additional point attributes that are not defined in the selected format for export.

File name
Defines the path and file name.

Select points
Allows you to select Civil Points for export.

Select point group
Allows you to select a point group from the drop-down list.

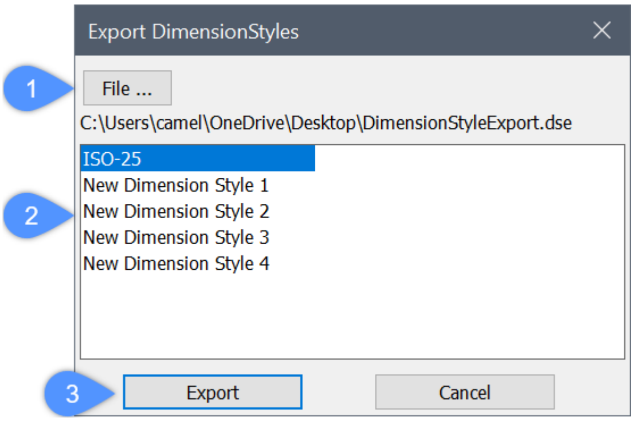
Select points in drawing

Click the select button () to select Civil Points in the drawing for export.

34.5.17 Export DimensionStyles dialog box

Opens via: DIMEX command (Express Tools)

The **Export DimensionStyles** dialog box allows you to export dimension styles and their settings to an external file.



1 File...

2 Available Dimension Styles

3 Export

34.5.17.1 File...

Opens the **Select export file** dialog box that allows you to choose the location for the DSE file.

34.5.17.2 Available Dimension Styles

Displays the available dimension styles and allows you to select the dimension styles to write to the DSE file. The selected styles are written in the file indicated under the **File...** button.

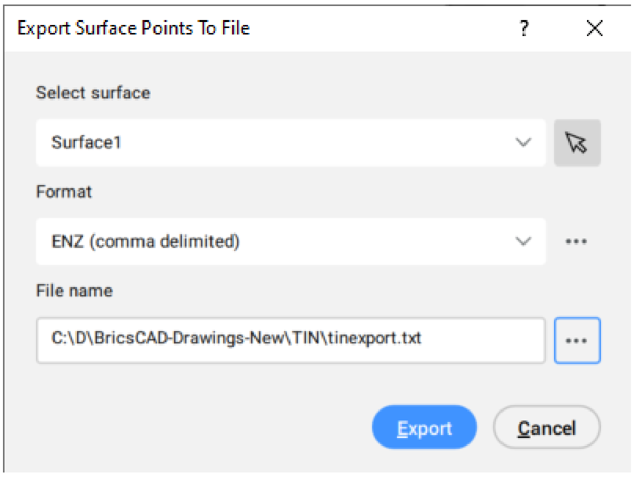
34.5.17.3 Export

Exports the selected dimension styles to the indicated DSE file.

34.5.18 Export Surface Points To File dialog box

Opens via: TINEXPORT command

The **Export Surface Points To File** dialog box allows you to choose which surface to export.



34.5.18.1 Select surface

Allows you to select a TIN surface available in the drop-down list or pick a TIN surface from the drawing.

34.5.18.2 Format

Allows you to select a file format from the drop-down list or configure a new file format in the **Manage Surface File Formats** dialog box, which opens when you press the three dots from the right side.

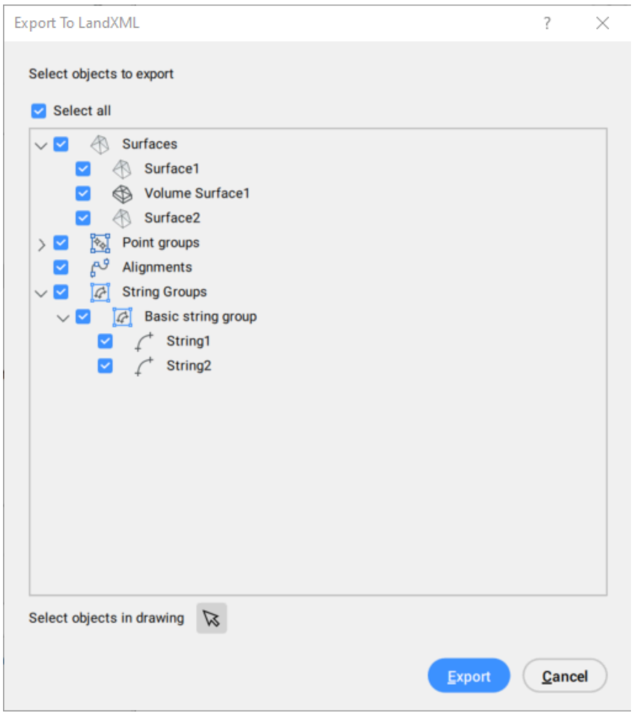
34.5.18.3 File name

Allows you to specify the file's name and the folder where you want the TIN surface to be exported.

34.5.19 Export To LandXML dialog box

Opens via: LANDXMLEXPORT command

The **Export To LandXML** dialog box allows you to select which entities you want to export by their type or by selecting them in the drawing.



34.5.19.1 Select all

When checked, all entities in the drawing are selected for export.

34.5.19.2 Select objects in drawing

Click the select icon ( ) to temporarily dismiss the **Export To LandXML** dialog box and select entities in the drawing to export.

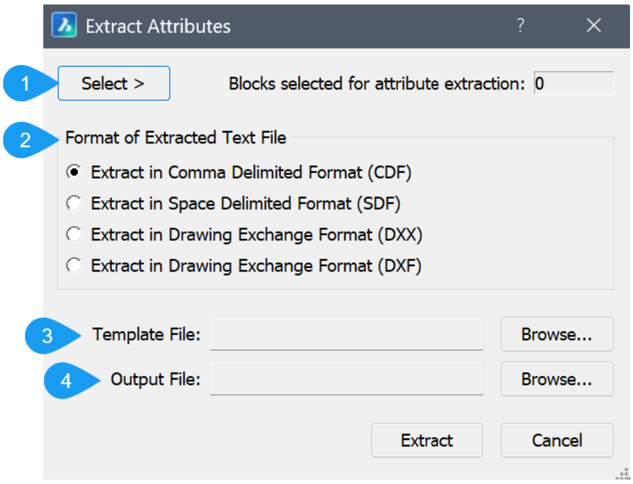
34.5.19.3 Export

Opens the **Save LandXML file** dialog box where you can specify the location and name of the output LandXML file.

34.5.20 Extract Attributes dialog box

Opens via: ATTEXT command

The **Extract Attributes** dialog box allows you to export data from block attributes to a text file.



1 Select

2 Format of Extracted Text File

3 Template File

4 Output File

34.5.20.1 Select

Specifies the set of blocks with attributes to copy data from.

Note: If blocks without attributes are selected, BricsCAD ignores them.

34.5.20.2 Format of Extracted Text File

Specifies the format of the extracted text file. Several formats are available:

- CDF: Comma Delimited Format
- SDF: Space Delimited Format
- DXX: Drawing Exchange Format
- DXF: Drawing Exchange Format

34.5.20.3 Template File

Specifies the location path of the template file by entering the path or through the **Attribute Extract Template File** dialog box after pressing the **Browse...** button.

34.5.20.4 Output File

Specifies the location path of the output file by entering the path or through the **Attribute Extract Output File** dialog box after pressing the **Browse...** button.

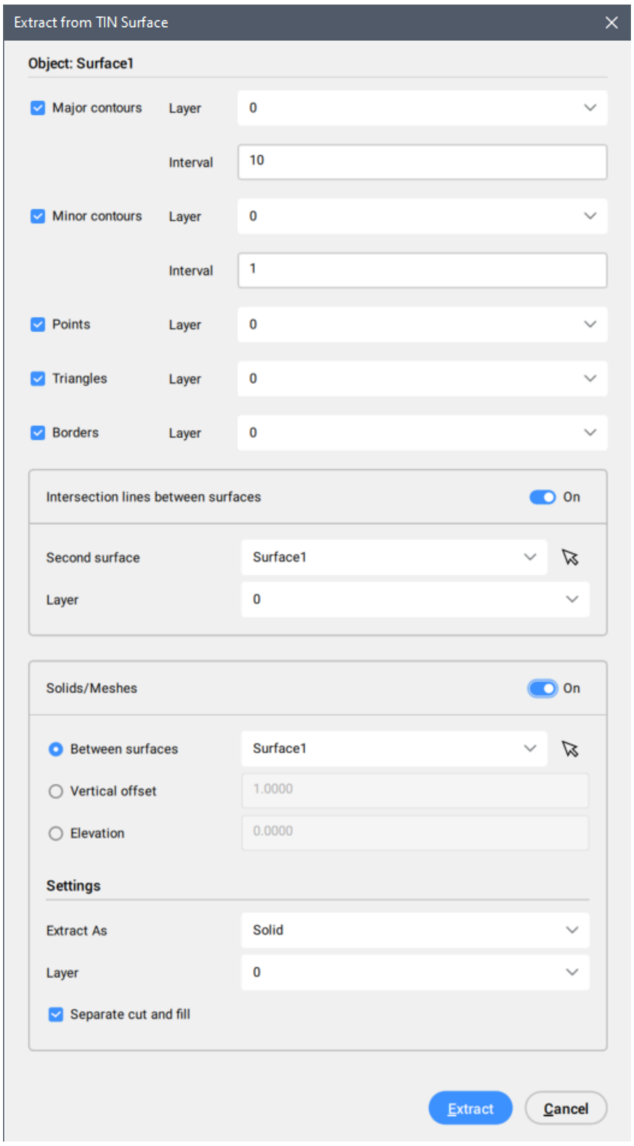
34.5.20.5 Extract

Exports the attribute values to the output file.

34.5.21 Extract from dialog box

Opens via: TINEXTRACT command

Depending on the selected entity (TIN surface, Grading, or TIN Volume Surface), the TINEXTRACT command opens the **Extract from TIN Surface / Grading / TIN Volume Surface** dialog box.



34.5.21.1 Major contours

Extracts the contours of a TIN surface, Grading, or TIN Volume Surface for an elevation according to major contours interval set.

Layer

Sets the layer where the major contours will be placed.

Interval

Defines the major contours interval.

34.5.21.2 Minor contours

Extracts the contours of a TIN surface, Grading, or TIN Volume Surface for an elevation according to minor contours interval set.

Layer
Sets the layer where the minor contours will be placed.

Interval
Defines the minor contours interval.

34.5.21.3 Points

Extracts all points on the surface.

Layer
Sets the layer where the points will be placed.

34.5.21.4 Triangles

Extracts all triangular faces of the surface.

Layer
Sets the layer where the triangular faces will be placed.

34.5.21.5 Borders

Extracts the outer boundary of the surface.

Layer
Sets the layer where the outer boundary will be placed.

34.5.21.6 Intersection lines between surfaces

Extracts intersection lines between the selected surfaces.

Second surface
Allows you to select one of the available surfaces in the drawing from the drop down list.

Note: You can also select the surface from the drawing area.

Layer
Sets the layer where the intersection line entity will be placed.

34.5.21.7 Solids/Meshes

Extracts solids or meshes between the selected surfaces.

Between surfaces
Allows you to select one of the available surfaces in the drawing from the drop down list.

Note: You can also select the surface from the drawing area.

Vertical offset
Defines the vertical offset value.

Elevation
Defines the elevation value.

Extract As
Sets the type of the extracted entity, solid or mesh.

Layer
Sets the layer where the solid or mesh will be placed.

Separate cut and fill
Toggles the creation of separate meshes for cutting and filling.

34.6 F

34.6.1 Feature Control Frame dialog box

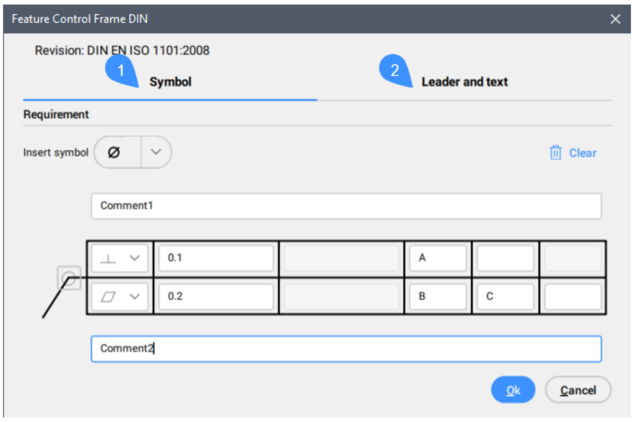
Opens via: AMFCFRAME command

The **Feature Control Frame** dialog box allows you to set the symbol for characteristics, their tolerance values, and the letter identifying the datum relative to which the tolerance is specified. Also, it is possible to add notes.

The parameters entered for the feature control frame will be stored temporarily and used as default values for the next feature control frame symbol.

The most recent revision of the standard to which this symbol conforms is displayed at the top of the dialog box.

The dialog box contains two main tabs, **Symbol** and **Leader and text**.

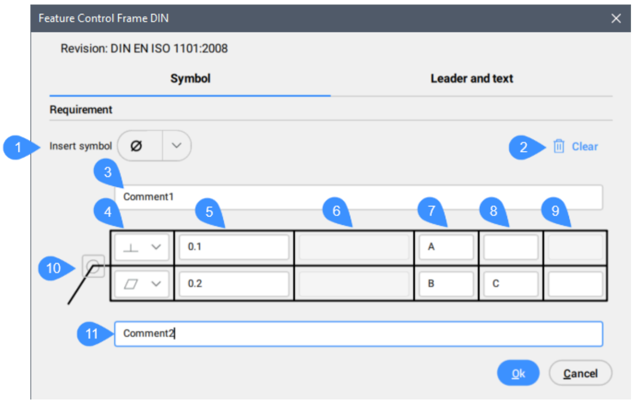


1 Symbol

2 Leader and text

34.6.1.1 Symbol

Specifies the feature control frame requirements. The drafting standard, ISO, DIN, ANSI, or JIS is written in the Revision field.



1 Insert symbol

2 Clear

3 Top Note

4 Geometric symbol

5 Tolerance 1

6 Tolerance 2

7 Datum 1

8 Datum 2

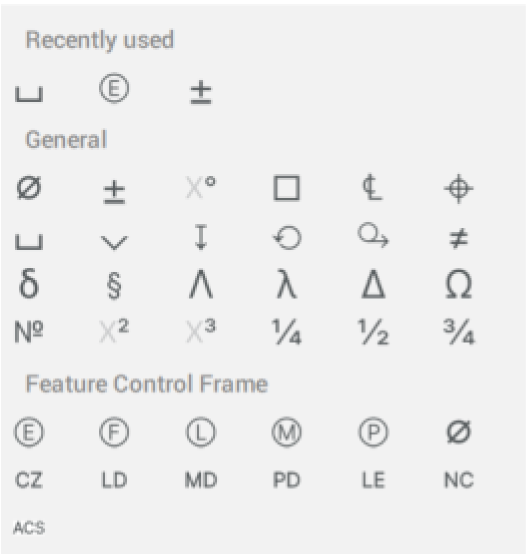
9 Datum 3

10 All around

11 Bottom Note

Insert symbol

Inserts a symbol from the drop-down list into the selected parameter field.



Clear

Clears the last inserted parameters. The cells become empty.

Top Note
Inserts a note to add above the symbol.

Geometric Symbol
Allows you to choose a geometric symbol from the drop-down list.

Tolerance 1
Allows you to type in the tolerance to appear next to the geometric symbol.

Tolerance 2
Allows you to type in the tolerance to appear next to the geometric symbol.

Datum 1/2/3
Allows you to type in additional letters identifying the datum relative to which the tolerance is specified.

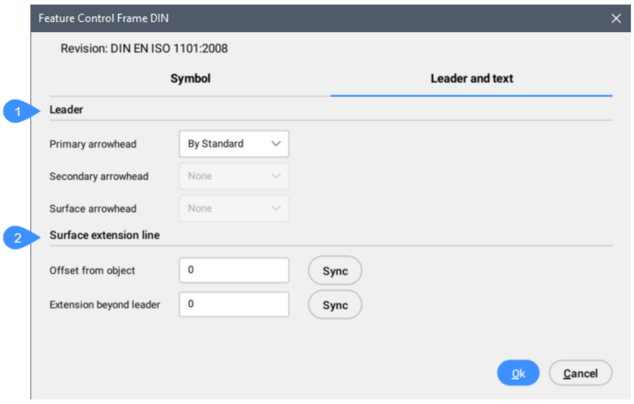
Note: The second and third fields will become available for writing only after the previous one is filled in.

All around
Toggles the visibility of the all around symbol.

Bottom Note
Inserts a note to add below the symbol.

34.6.1.2 Leader and text

Specifies the Leader and text characteristics.



1 Leader

2 Surface extension line

Leader
Sets the leader characteristics.

Primary arrowhead
Defines the default leader arrowhead type from the drop-down list.

Surface extension line
Sets the surface extension line characteristics.

Offset from object
Defines the distance between the extension line's start point and the attached object.

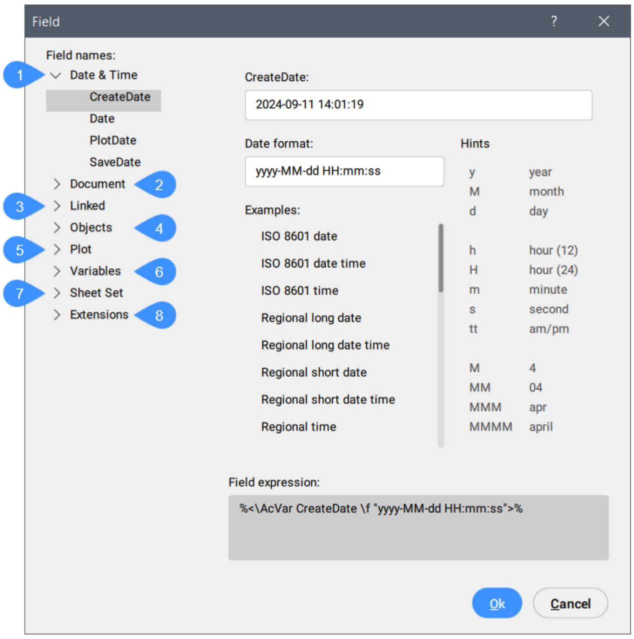
Extension beyond leader
Defines the distance between the symbol's start point and the surface extension line's end point.

34.6.2 Field dialog box

Opens via: FIELD command

The **Field** dialog box allows you to insert a field in a single or multiline text, table cells, attributes, and so on.

Note: A field is variable text that uses a coding system which the program updates automatically, displaying data as drawing properties (for example a file name or the saved date), entity properties (like length, area, or layer), user-defined properties, plot settings, variables... If a field does not have value, the program displays hyphens (----). If the information referenced by the selected property is not valid, the value for the field is displayed as ####.



Note: Expand a category in **Field Names**, then select a field name.

1 Date & Time

2 Document

3 Linked

4 Objects

5 Plot

6 Variables

7 Sheet Set

8 Extensions

9 Field expression

34.6.2.1 Date & Time

CreateDate
Displays the last revision date of the drawing.

Date
Displays the current date.

PlotDate
Displays the date when the drawing was last plotted.

SaveDate
Displays the date when the drawing was last saved.

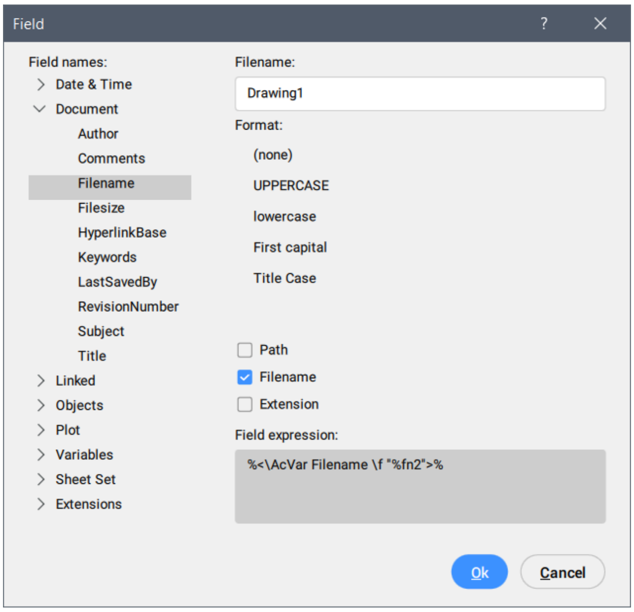
Date format
Select a date format from the list or type a format in this field.

Examples
Displays date format examples.

Hints
Explains the meaning of the digits used in the date format acronym/abbreviation definition.

Field expression
Displays the field expression of the selected date. An operand in an expression can be a DisplayProperty-Name surrounded with arrow brackets (<>).

34.6.2.2 Document



Author
Displays the author of the document.

Comments
Displays the comments in the document.

Filename
Shows the file name. It can include the path and the extension of the file.

Filesize
Displays the file size expressed in Bytes, Kilobytes and Megabytes.

HyperLinkBase
Displays the default path for the relative hyperlink in drawings.

Keywords
Displays the keywords for the opened file.

LastSavedBy
Displays the name of the person that last saved the file.

RevisionNumber
Displays the drawing revision number.

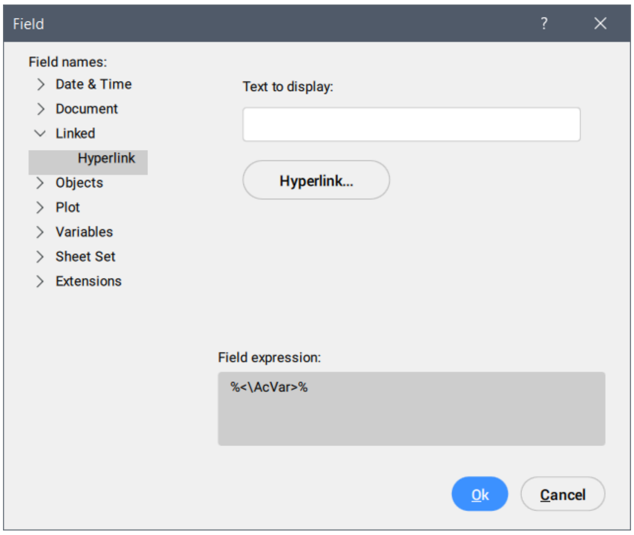
Subject
Displays the subject of the drawing.

Title
Displays the title of the drawing.

Format
Displays the format of the selected category.

Field expression
Displays the field expression of the selected date. An operand in an expression can be a DisplayProperty-Name surrounded with arrow brackets (<>).

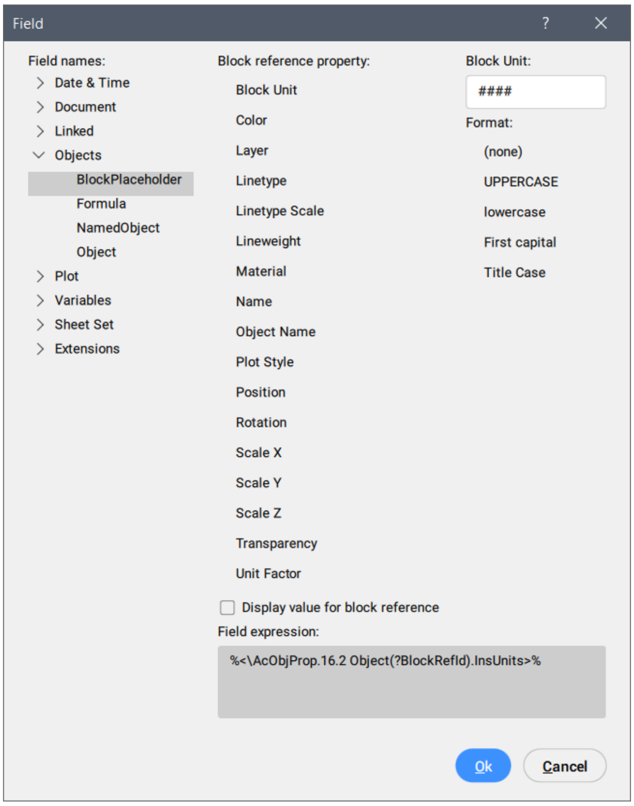
34.6.2.3 Linked



Text to display
Enter the text to be displayed for this hyperlink.

Hyperlink
Opens the **Edit Hyperlink** dialog box.

34.6.2.4 Objects



BlockPlaceholder

See the ATTDEF command to create block property fields in an attribute. When such an attribute is included in a block definition, the field displays the current value of the block property.

Block reference property

Displays the block reference property.

Block name

Displays the name of the property.

Format

Specifies the format for displaying the property.

Formula

Creates a formula field. You can use the values of table cells in a formula field. Click the **Average**, **Sum**, **Count** or **Cell** button. The **Field** dialog box closes to allow you to select cells in a table in the drawing. Also, you can enter the formula manually, for example, if you know the table handle.

NamedObject

Creates a field that shows the current name of a named entity.

- Click the **Named object type** field, then select the object type in the list.

  All named objects of the selected type are listed in the **Name** list box.
- Select a name in the list.
- Choose a format.
- Click the **OK** button to place the field.

Object

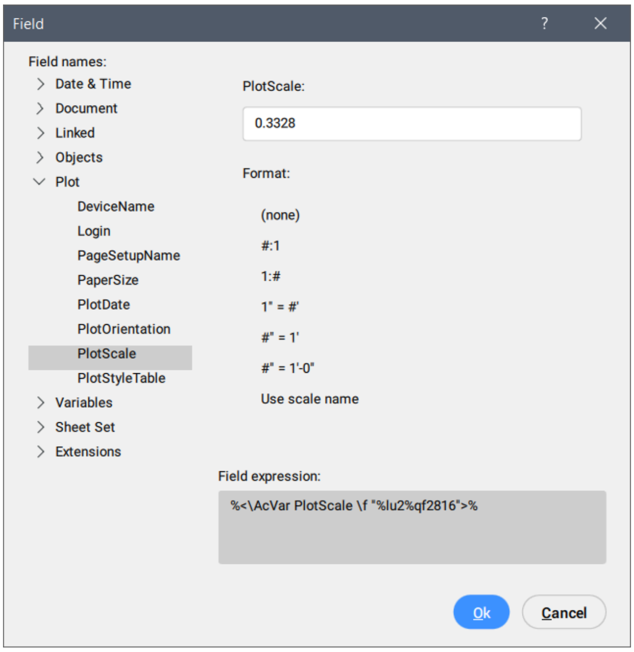
Creates a field that shows a property of a selected entity.

- Click the select button, next to the **Object type** field.

  The **Field** dialog box temporarily closes.
- Select an entity in the drawing.
- The properties of the selected entity are listed in the **Property** list.
- Select a property in the list. Custom properties of all parameter types and visibility states for dynamic blocks and block references are also available.
- Choose a format.
- Click the **OK** button to place the field.

  Note: Attributes with fields are updated after REGEN and UPDATEFIELD commands.

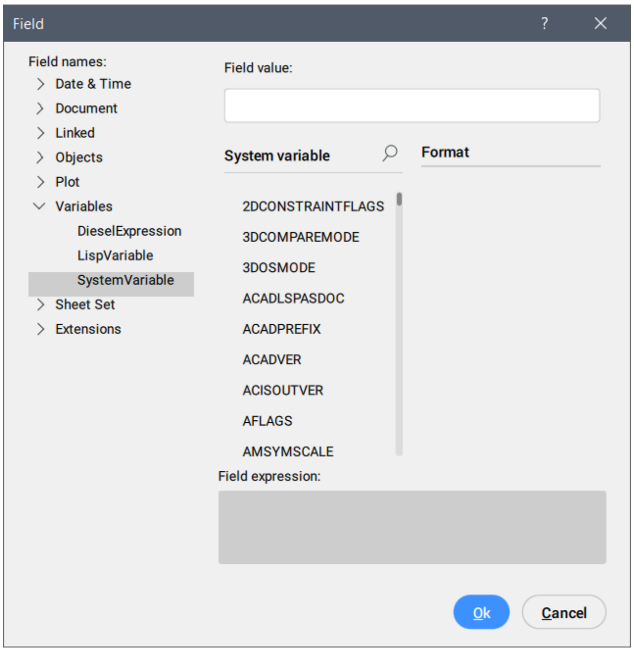
34.6.2.5 Plot



Plot fields apply to paper space layouts.

Allows you to create fields which show the current print settings of a layout.

34.6.2.6 Variables



Allows you to create fields which show the current value of **Diesel Expressions**, **Lisp Variables** and **System Variables**.

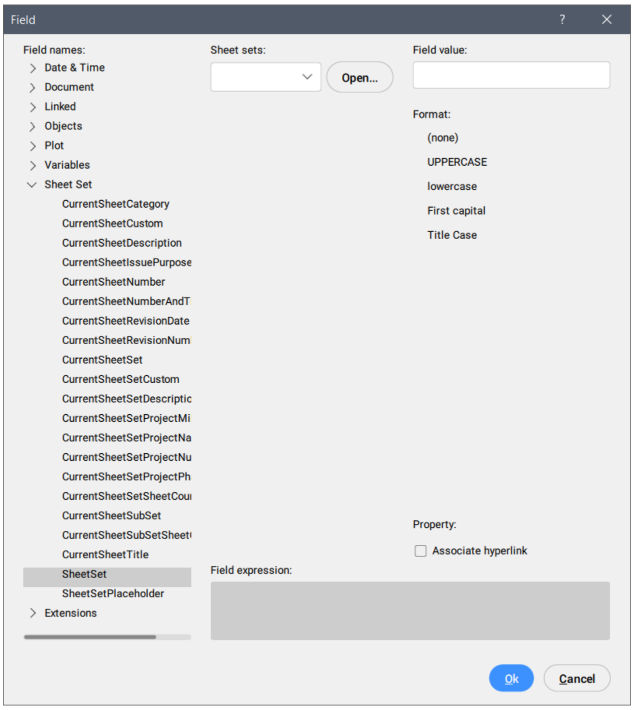
It is possible to search for a specific variable in the **Lisp Variables** and **System Variables** lists using:

- **Key navigation**: press the up/down arrows to navigate in the list.
- **Search bar**: allows you to search for specific variables. The list is shortened to display only the variables that contain the entered sequence of characters.

  Note: To enable the search bar:

  - Click the icon.
  - Press the **Crtl+F** key. The focus needs to be in the variable list. You can use the TAB key to cycle focus.

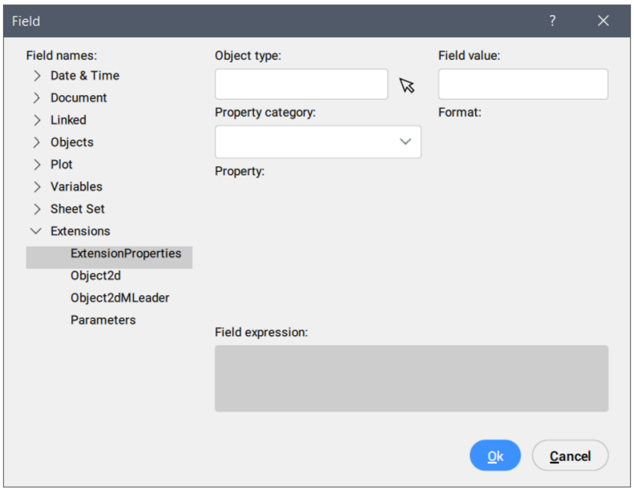
34.6.2.7 Sheet Set



Allows you to create fields which show default and custom sheet set properties. With the **Sheet Set** property, a sheet navigation tree is opened.

Note: Sheet Set fields are updated with the UPDATEFIELD command.

34.6.2.8 Extensions



Allows you to create fields that show properties of entities specific to BricsCAD, such as BIM or Mechanical entities, displaying cached values in AutoCAD®.

ExtensionProperties

Allows you to create fields with properties of a selected BIM entity.

Object2d
Allows you to create fields with properties of 3D entities by selecting the associated 2d entity inside BIM section block.

Object2dMLeader
Allows you to create fields inside MLeader with properties of 3D entities by selecting the associated 2d entity inside BIM section block using MLeader's arrow.

Parameters
Allows you to create fields referencing parameters in the drawing.

Note:

- For block parameters, use the FIELD command inside a BEDIT session.
- After changing parameter values (from the **Properties**, **Parameters Manager**, or **Mechanical Browser** panel) use the REGEN, REGENALL, or UPDATEFIELD command to update the value displayed in the created field.

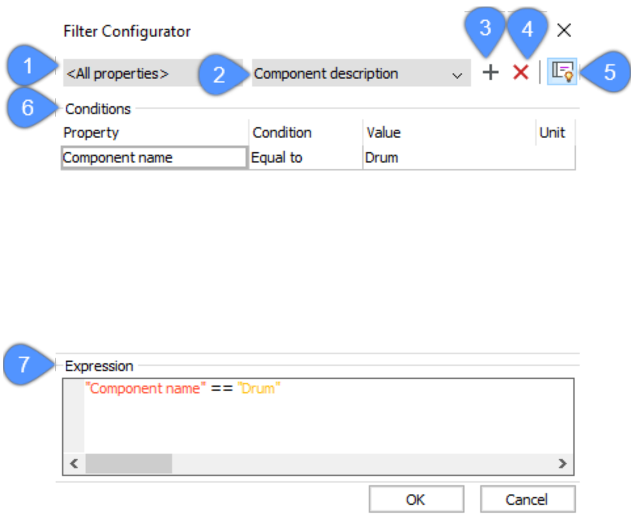
34.6.2.9 Field expression

Displays the expression of the field. You can learn how fields are structured by reading this code.

34.6.3 Filter Configurator dialog box

Opens via: **BOM Manager** panel > **Set filter** option

The **Filter Configurator** dialog box allows you to specify the conditions to filter components in the BOM table.



1 Columns

2 Properties

3 Add

4 Delete

5 Expression

6 Conditions

7 Expression Editor

34.6.3.1 Columns

Selects columns to apply the filter to.

34.6.3.2 Properties

Specifies the available properties that can be used to apply the filter.

Note: When specified the options above, the condition can be added by pressing the plus sign.

Filter conditions

Displays the conditions that were added. This includes the properties, it's condition, value and unit.

34.6.3.3 Add

Adds a condition for the selected property.

34.6.3.4 Delete

Deletes the selected condition(s) from the filter.

34.6.3.5 Expression

Shows or hides the expression editor.

34.6.3.6 Conditions

Displays all the added conditions that will be used to filter the content of the BOM table.

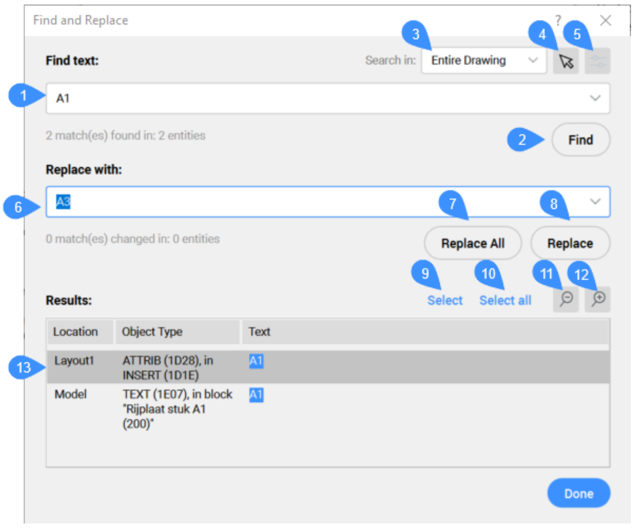
34.6.3.7 Expression Editor

Displays the Expression Editor where you can configure and preview the filtering expression.

34.6.4 Find and Replace dialog box

Opens via: FIND command

The **Find and Replace** dialog box allows you to find and optionally replace strings of text in the current drawing. It searches for text in blocks, attributes, dimensions, and hyperlinks.



1 Find Text

2 Find

3 Search In

4 Select Entities

5 Options

6 Replace With

7 Replace All

8 Replace

9 Select

10 Select all

11 Zoom Out Drawing

12 Zoom In Drawing

13 Search report

34.6.4.1 Find Text

Specifies the text to find.

34.6.4.2 Find

Finds the next occurrence of the text string.

Shortcut Keystroke: Crtl+F.

34.6.4.3 Search In

Specifies the range of search:

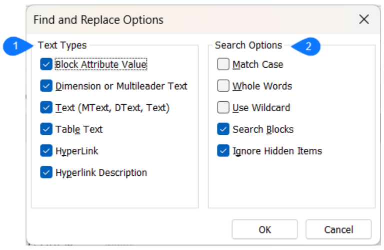
- **Entire Drawing**: searches all text in the drawing.
- **Current Layout**: only searches in the current Layout.
- **Current Selection**: only searches in the current selection that can be made using button no. 5.

34.6.4.4 Select Entities

Dismisses the dialog box so that you can select the entities to search. After choosing one or more entities, press Enter to return to the dialog box.

34.6.4.5 Options

Opens the **Find and Replace Options** dialog box that allows you to specify the types of text entities to search.



Text Types

Specifies the types of text to include in the search:

- Block Attribute Value
- Dimension or Multileader Text
- Text
- Table Text
- HyperLink
- Hyperlink Description

Search Options

Specifies the search options:

- **Match Case**: only reports if string matches the use of capital letters.
- **Whole Words**: only reports if whole word matches the entered text string. If a long word contains the entered string, this will not be reported in the results.
- **Use Wildcard**: filters the search using wildcard characters as comparison criteria.
- **Search Blocks**: includes blocks in the search.
- **Ignore Hidden Items**: does not include hidden items in the search.

34.6.4.6 Replace With

Specifies the text that will replace the found text. Do not fill in this field when you're only searching for text.

34.6.4.7 Replace All

Replaces all occurrences of the text string with the replacement string.

Shortcut Keystroke: Ctrl+A.

34.6.4.8 Replace

Replaces the found text string with the replacement string.

Shortcut Keystroke: Ctrl+R.

34.6.4.9 Select

Creates a selection set from the selected entities in the list and closes the dialog box.

34.6.4.10 Select All

Selects all occurrences of the text string.

34.6.4.11 Zoom Out Drawing

Zooms out from the center of the window with a factor of ½.

34.6.4.12 Zoom In Drawing

Zooms in on the center of the window with a factor of 2.

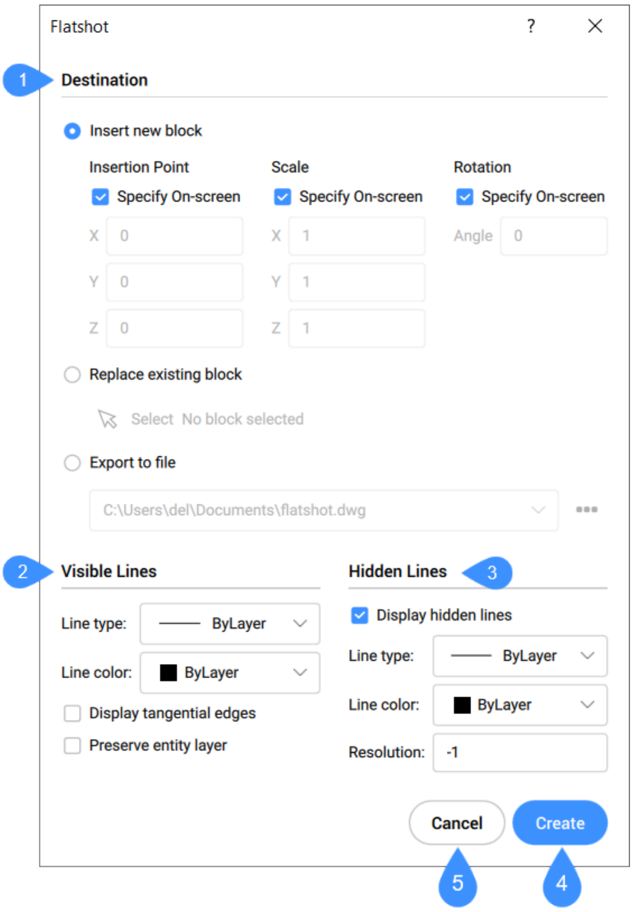
34.6.4.13 Search report

Displays the number of matches found in the selected entities, and the number of matches and entities changed after replacement. The search result can be sorted by each column.

34.6.5 Flatshot dialog box

Opens via: FLATSHOT command

The **Flatshot** dialog box allows you to create a flattened, hidden line representation of 3D solids, and 3D surfaces as a block or a new drawing.



1 Destination

2 Visible lines

3 Hidden lines

4 Create

5 Cancel

34.6.5.1 Destination

Determines where the block should be placed. Several options are available.

Insert new block

Inserts the flatshot block as a new block in the current drawing by specifying some characteristics:

- Insertion point
- Scale
- Rotation

Replace existing block

Replaces a block that is already inserted in the drawing. By choosing this option, you can further select a block from the current drawing.

Export to file

Saves the flatshot block as a DWG file through the **Select Export File** dialog box.

34.6.5.2 Visible lines

Sets the linetype and color for the visible lines. Choose the linetype and color from the drop-down lists.

Note: You can load a new linetype into the drawing through the **Load Linetypes** dialog box by selecting **Load...**. You can select a new line color through the **Color** dialog box by selecting **Select color...**.

Display tangential edges
Toggles the display of tangential edges. A tangential edge is the imaginary transition line between two tangent faces.

Preserve entity layer
When activated, the layers of entities in the block are maintained.

34.6.5.3 Hidden lines

Toggles the display of hidden lines and sets their properties.

Display hidden lines
Toggles the display of hidden lines.

Resolution
Specifies the short entity that will be represented by a hidden line. Enter a negative value so the program will determine this automatically. The range is from 0.01 to 0.000000000001.

34.6.5.4 Create

Creates the Flatshot.

Note: The corresponding keyboard shortcut for this button is **ALT + R**.

34.6.5.5 Cancel

Cancels the Flatshot creation.

Note: The corresponding keyboard shortcut for this button is **ALT + C**.

34.6.6 Font Substitution dialog box

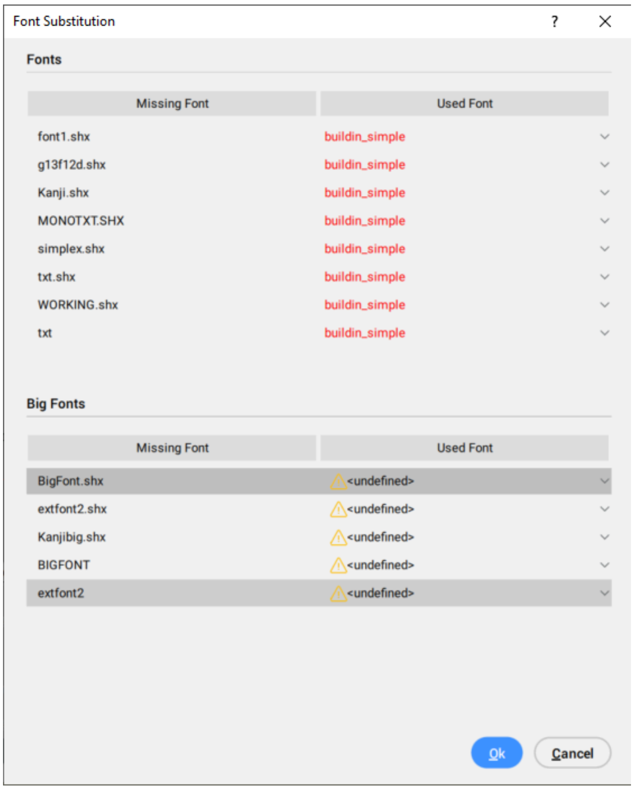
When opening a file containing a substituted Big font, the **Missed font (s)** warning message appears on the bottom right side of the drawing. Pressing the **Specify font replacement** button opens the **Font Substitution** dialog box.

Note: The warning message displays when the **Tray Icon Items** button is active in the **Status bar** (the TRAYNOTIFY and TRAYICONS system variables are ON).

The **Font Substitution** dialog box allows you to automatically replace unknown fonts or fonts with substitutions in the font mapping file (default.fmp). It acts as an extended font mapping of the default.fmp file. See the FONTMAP system variable.

Note: The substitution fonts are applied to all the opened drawings in the session, so all drawings that use these fonts will show differently. The font substitution is done the same way as with the fontmap default.fmp, so it does not modify any Text Styles in the drawings.

Note: You can view all this automatic font mapping (substitutions) in **Drawing Explorer**/**Text Styles** dialog box any time after opening the file. Also, the **Font Substitution** dialog box opens when you choose the **Fonts substitution...** option from the context menu to show all fonts associated with Text Styles in the current session drawing for which you can make substitutions (without modifying the Text Styles).



34.6.6.1 Fonts

Lists the fonts that are not recognized or have substitutions in the font mapping file (default.fmp).

34.6.6.2 Big Fonts

Lists the Big Fonts (SHX bigfont files) that are not recognized.

34.6.6.3 Missing Font

Lists the fonts not recognized or have substitutions in the font mapping file (default.fmp).

34.6.6.4 Used Font

Displays the font name used for substitution according to the mapping, which is actually used in the drawing. This can be different from the font the Text style refers to.

Note: Click the drop-down arrow and scroll down the list to choose another available font from the default font mapping (default.fmp) file.

Note: The default font, **Simplex.shx**, is used as substitution when BricsCAD cannot recognize the original font in the drawing. See FONTALT system variable.

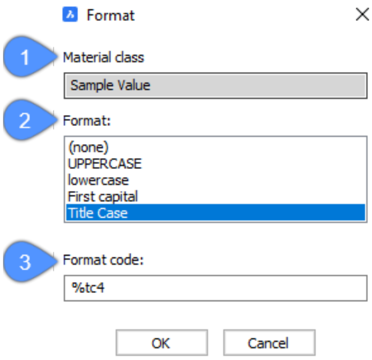
Note: If an MTEXT uses a font directly instead of through a Text Style, this font will not be listed anywhere in the interface, and it will not be listed when the font is absent.

Also, when a font is present, but an MTEXT uses characters that the font does not support, the program will try to use an alternative font for those characters. This alternative font is not listed in the Font Substitution dialog box.

34.6.7 Format dialog box

Opens via: **BOM Manager** panel > **Format** option

The **Format** dialog box allows you to specify the format of text strings in the specified column of a BOM table.



1 Material class

2 Format

3 Format code

34.6.7.1 Component

Specifies a component.

34.6.7.2 Format

Specifies the desired format for the selected component from the list.

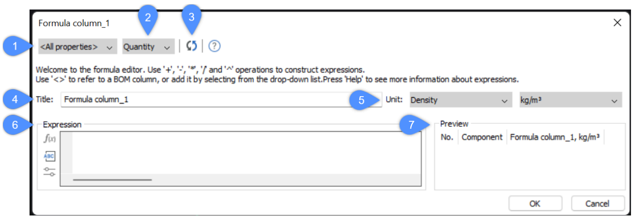
34.6.7.3 Format code

Changes depending on the format type you choose for all the materials in the BOM table.

34.6.8 Formula column dialog box

Opens via: **BOM Manager** panel

The **Formula column** dialog box allows you to configure the title, unit and expression for the formula column. It also displays a preview of the formula results.



1 Category selector

2 Column reference

3 Refresh

4 Title

5 Unit

6 Expression

7 Preview

34.6.8.1 Category selector

Lists all categories of columns.

34.6.8.2 Column reference

Displays the available columns to use.

34.6.8.3 Refresh ( )

Refreshes the preview.

34.6.8.4 Title

Displays the name of the new column.

34.6.8.5 Unit

Lists all measurements and units.

34.6.8.6 Expression

Displays the template expression.

Function (fx)
Inserts the selected function at the cursor position.

Variable ( )
Inserts the selected variable at the cursor position.

Configure variable ( )
Allows you to configure the selected variable.
In formula expressions, the syntax was extended to allow you to set target unit for variables, if the variable supports a unit conversion. It allows to write unit-dependent formula (such as density computation) without bothering about the initial unit of the variable.

If a formula unit is known to BricsCAD, then unit conversions will be supported for formula values.
The default unit mode for formula columns is set to be user units.

Note: The **Expression** field supports autocomplete.

34.6.8.7 Preview

Displays a preview of the formula results.

34.7 G

34.7.1 Generic Open file dialog box

The **Generic Open file** dialog box allows you to open or import a file.

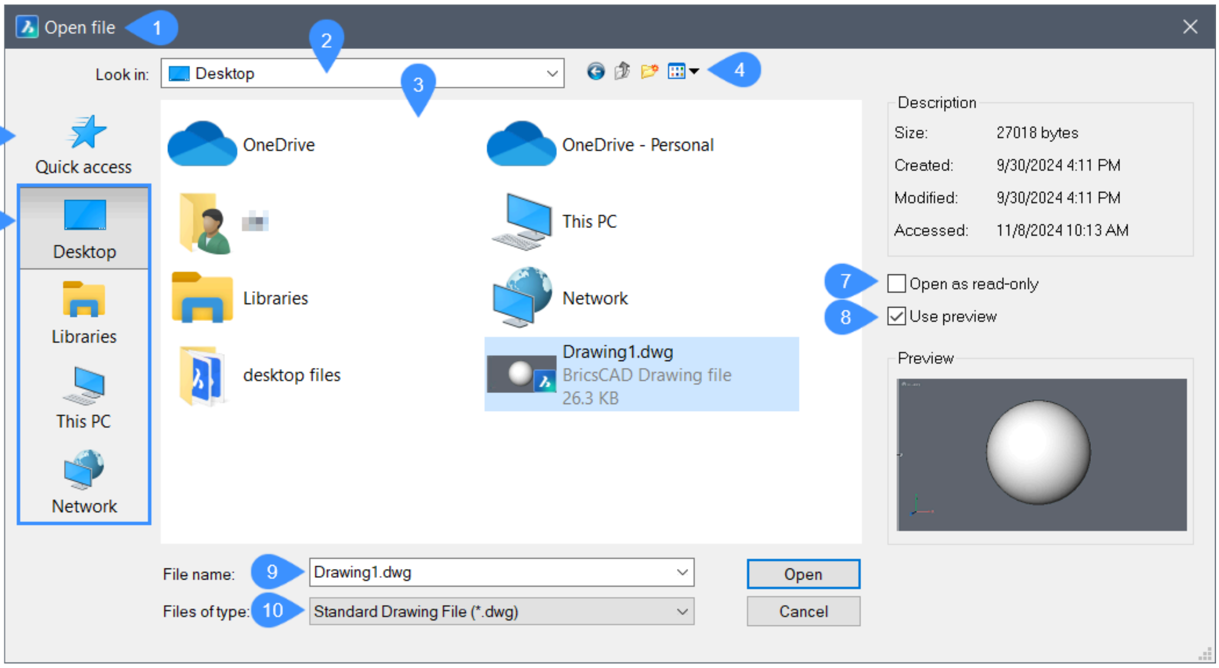
This dialog box can be accessed using multiple commands, and its name reflects each case.

The type of dialog box opened depends on the value of the USESTANDARDOPENFILEDIALOG system variable (default value is 0):

- Non-standard for USESTANDARDOPENFILEDIALOG ==0
- Standard for USESTANDARDOPENFILEDIALOG ==1

Note: For Windows 11, the default homepage of File Explorer is called **Home** instead of **Quick access**.

34.7.1.1 Standard Open file dialog box



1 Dialog name

2 Look in

3 Folders and files list

4 View menu

5 Quick access

6 Standard folders

7 Open as read-only

8 Use preview

9 File name

10 Files of type

Dialog name

Displays the dialog name depending on the command used.

Look in

Allows you to specify the location of your file. To change it to a different disk drive or folder, click the arrow and select another disk drive or folder from the drop-down list.

To scroll the list, press the F4 key.

Folders and files list

Lists the folders and files in the current folder.

View menu

Sets how the folders and files are displayed in the list.

Quick access

You can populate the Quick access with folders that you want to be always available, not only with folders that have being used recently. The folders shown in Quick access can be configured in Windows Explorer.

The Quick access folders are available during both open and save operations.

Standard folders
The four standard folders in the places bar are: **Desktop**, **Libraries**, **This PC**, and **Network**.

Note: The folders in the places bar of the **Standard Open file dialog box** are not customizable. For advanced customization options, use the **Non-standard Open file dialog box** (USESTANDARDOPENFILEDIALOG == 0).

Open as read-only
Tick the checkbox to open the file as read-only. Consequently, you do not have permission to write new information or edit the data in the file.

Use preview
Tick the checkbox to view a part of the file before opening.

File name
Specifies the name of the selected file.

Files of type
Displays a list of valid file types. You can select a file type to filter the files list so that it only shows files matching that type.

34.7.1.2 Non-standard Open file dialog box

The non-standard Open file dialog box provides extra functionality in addition to that of the standard dialog box:

- **Search in folder**: Allows you to search for folders and files in the current folder. Start to type in a string to filter the results.
- **Navigation pane**: Allows you to view the folder structure of the computer, navigate and access files.

  You can populate the **Quick access** list with folders that you want to be always available, not only with folders that have being used recently. To do this, locate the desired folder, then drag and drop it in the **Quick access** list.

  To change the order of the custom folders in the **Quick access** list, drag and drop folders in the list to the desired position.

  The **Quick access** custom folders are available during both open and save operations.
- **Preview pane**: Displays a preview of the selected file.

34.7.2 Generic Save file dialog box

The **Generic Save file** dialog box allows you to save or export a file.

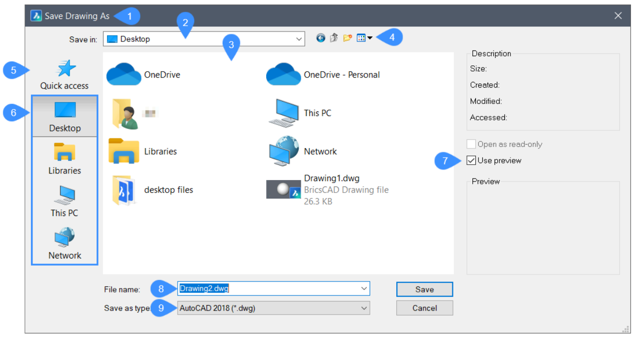
This dialog box can be accessed using multiple commands, and its name reflects each case.

The type of dialog box opened depends on the value of the USESTANDARDOPENFILEDIALOG system variable (default value is 0):

- Non-standard for USESTANDARDOPENFILEDIALOG ==0
- Standard for USESTANDARDOPENFILEDIALOG ==1

Note: For Windows 11, the default homepage of File Explorer is called **Home** instead of **Quick access**.

34.7.2.1 Standard Save file dialog box



1 Dialog name

2 Save in

3 Folders and files list

4 View menu

5 Quick access

6 Standard folders

7 Use preview

8 File name

9 Files of type

Dialog name
Displays the dialog name depending on the command used.

Save in
Allows you to specify the location of your file. To change it to a different disk drive or folder, click the arrow and select another disk drive or folder from the drop-down list.
To scroll the list, press the F4 key.

Folders and files list
Lists the folders and files in the current folder.

View menu
Sets how the folders and files are displayed in the list.

Quick access
You can populate the Quick access with folders that you want to be always available, not only with folders that have being used recently. The folders shown in Quick access can be configured in Windows Explorer.
The Quick access folders are available during both open and save operations.

Standard folders
The four standard folders in the places bar are: **Desktop**, **Libraries**, **This PC**, and **Network**.

> ***Note:*** The folders in the places bar of the **Standard Open file dialog box** are not customizable. For advanced customization options, use the **Non-standard Open file dialog box** (USESTANDARDOPENFILEDIALOG == 0).

Use preview
Tick the checkbox to view a part of the file before opening.

Note: A preview is available when selecting an existing file to replace it.

File name
Specifies the name of the file. You can accept the predefined name or you can specify a different name.

Files of type
Displays a list of valid file types. You can select a file type to filter the files list so that it only shows files matching that type.

34.7.2.2 Non-standard Save file dialog box

The non-standard Save file dialog box provides extra functionality in addition to that of the standard dialog box:

- **Search in folder**: Allows you to search for folders and files in the current folder. Start to type in a string to filter the results.
- **Navigation pane**: Allows you to view the folder structure of the computer, navigate and access files.

  You can populate the **Quick access** list with folders that you want to be always available, not only with folders that have being used recently. To do this, locate the desired folder, then drag and drop it in the **Quick access** list.

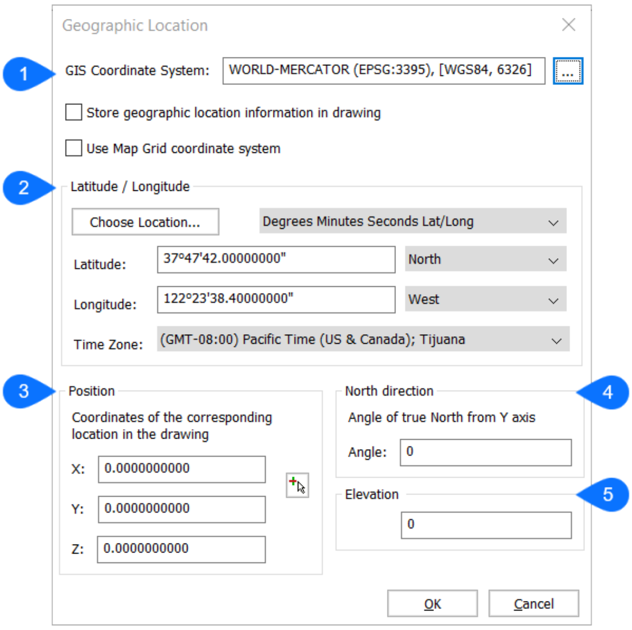
  To change the order of the custom folders in the **Quick access** list, drag and drop folders in the list to the desired position.

  The **Quick access** custom folders are available during both open and save operations.
- **Preview pane**: Displays a preview of the selected file.

34.7.3 Geographic Location dialog box

Opens via: GEOGRAPHICLOCATION command

The **Geographic Location** dialog box allows you to set the geographic location of the drawing by applying the longitude and latitude to a point in the drawing.



1 GIS Coordinate System

2 Latitude / Longitude

3 Position

4 North direction

5 Elevation

34.7.3.1 GIS Coordinate System

Reports the GIS system being used by the drawing.

Note: Click the Browse button (...) to change the GIS Coordinate System.

Store geographic location information in drawing

When checked, the geographic marker (Survey location marker) is displayed to indicate the position of the Geographic Location point in the drawing.

Use Map Grid coordinate system

When enabled, the drawing is associated with real-world coordinates, the location data from GIS Coordinate System.

Note: When disabled, the origin point is determined by the values entered for X, Y, and Z, which correspond with the latitude and longitude.

34.7.3.2 Latitude / Longitude

Choose Location

Specifies the geographic location through the **Choose Geographic Location** dialog box.

Degrees Minutes Seconds Lat/Long

Displays the location in a Degrees, Minutes and Seconds format.

Decimal Lat/Long
Displays the location in a decimal format.

Latitude
Sets the latitude. The valid range is 0 to 90.

Longitude
Sets the longitude. The valid range is 0 to 180.

Time zone
Specifies the time zone from the Time Zone drop list.

Note: This setting is saved to the TIMEZONE system variable.

34.7.3.3 Position

Specifies the position of the geographic location point in the drawing by entering coordinate values.

Note: You can also select a specific point in the workspace to set the position.

34.7.3.4 North direction

Specifies the angle of the sun from north, in the context of the world coordinate system (the angle between the True North and the Y-axis, measured clockwise).

Note: The setting is saved to the NORTHDIRECTION system variable.

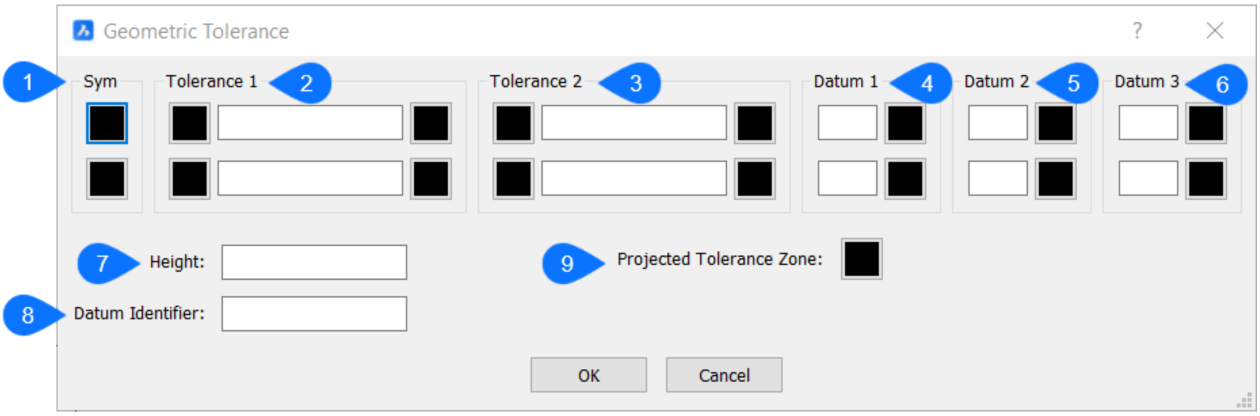
34.7.3.5 Elevation

Specifies an elevation for the long-lat point. The value can be positive or negative.

34.7.4 Geometric Tolerance dialog box

Opens via: TOLERANCE command

The **Geometric Tolerance** dialog box allows you to place tolerance symbols in drawings. Opens via the TOLERANCE command.



1 Symbol

2 Tolerance 1

3 Tolerance 2

4 Datum 1

5 Datum 2

6 Datum 3

34.7.4.1 Symbol

Specifies a tolerance symbol through the **Symbol** dialog box.

34.7.4.2 Tolerance 1 & 2

Specifies the tolerance specifications (diameter, value and material condition).

Diameter
Toggles the diameter symbol.

Value
Specifies the tolerance value.

Material condition
Specifies the material condition through the **Material condition** dialog box.

34.7.4.3 Datum 1, 2 & 3

Specifies the datum reference (value and material condition).

Value
Specifies the datum value.

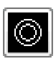
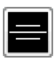
Material condition
Specifies the material condition through the **Material condition** dialog box.

Height
Specifies the height of the tolerance symbols.

Datum Identifier
Specifies datum identifier, such as Datum A.

Projected Tolerance Zone
Toggles the projected tolerance zone symbol.

Tolerance Symbols

Symbol	Characteristic	Type
	Position	Location
	Concentricity or Coaxiality	Location
	Symmetry	Location

Symbol	Characteristic	Type
∥	Parallelism	Orientation
⊥	Perpendicularity	Orientation
∠	Angularity	Orientation
⌭	Cylindricity	Form
⏥	Flatness	Form
○	Circularity or Roundness	Form
—	Straightness	Form
⌓	Profile of a surface	Profile
⌒	Profile of a line	Profile
↗	Circular Run-out	Run-out
⌰	Total Run-out	Run-out

Material Condition Symbols

Symbol	Definition
Ⓜ	At maximum material condition (MMC), a feature contains the maximum amount of material stated in the limits.
Ⓛ	At least material condition (LMC), a feature contains the minimum amount of material stated in the limits.

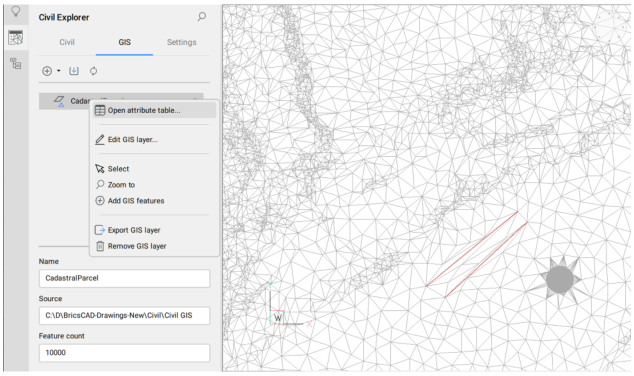
Symbol	Definition
Ⓢ	Regardless of feature size (RFS) indicates that the feature can be any size within the stated limits.

34.7.5 GIS Attribute Table dialog box

Opens via: **Civil Explorer** panel

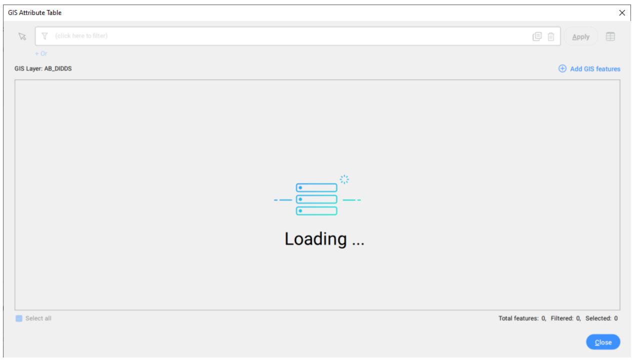
The **GIS Attribute Table** is a dialog box that allows interaction of a GIS attribute in the table with GIS entities (geometry) in the drawing.

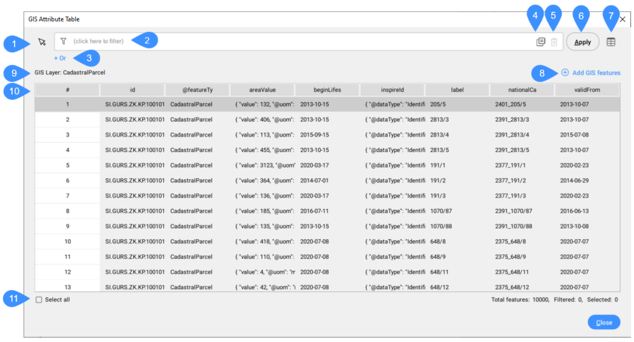
Once GIS layer is imported in a drawing, it is also listed in the **GIS** tab of the **Civil Explorer** panel. The attributes of the imported GIS layer are listed in the **GIS Attributes Table** dialog box.



The **GIS Attribute Table** dialog box can be opened via the **Civil Explorer** panel either by double-clicking on an individual GIS layer or by right-clicking on the layer to open the context menu and select the **Open attribute table** option.

When loading large GIS data sets into table view and for filtering data, a loading screen displays.





1 Select GIS features in the drawing

2 Filter attribute

3 Add a filter

4 +Or

5 Delete a filter

6 Apply

7 Change column layout

8 Add GIS Features

9 GIS Layer name

10 GIS data

11 Select all

34.7.5.1 Select GIS features in the drawing

Allows you to select GIS features in the drawing to highlight them in the table.

34.7.5.2 Filter attribute

Allows you to find rows/features using an expression. All the rows that meet the specified filter rules are listed in the table.

Two types of filters are available: And & Or.

Note: Consider the **And** operator when the filters are in the same row.

34.7.5.3 Add a filter

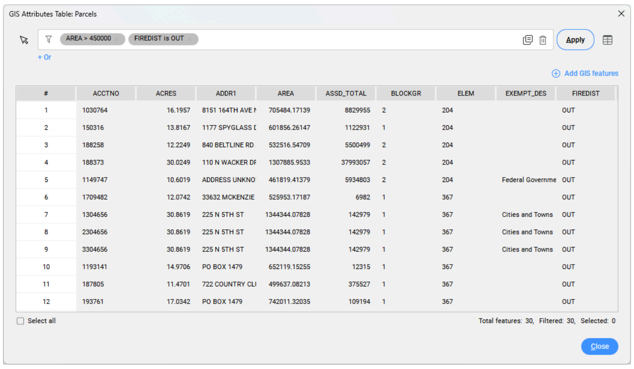
Adds one expression field for finding GIS attributes that meet all the criteria specified in that expression.

To use the And filter, specify at least one or a set of filter rules in the one expression field. To do this, click the filter field and select the GIS attribute from the drop-down list. Then choose a comparison symbol and enter a value. The available comparison symbols depend on the type of data and can be selected from the drop-down list:

- numbers (integers, doubles):
  - = equal
  - ≠ not equal to

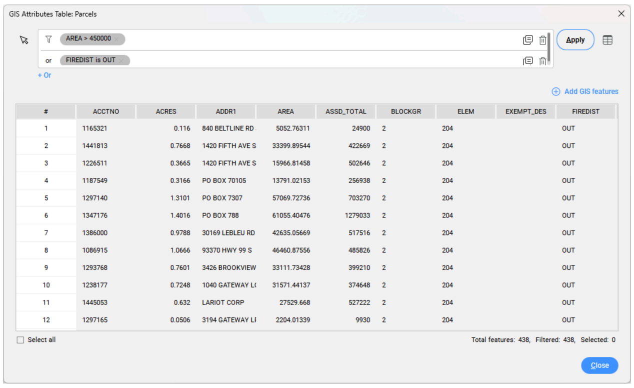
- > greater than
- < less than
- ≥ greater than or equal to
- ≤ less than or equal to

- words (strings):
  - is
  - is not
  - contains
  - does not contain



34.7.5.4 + Or

Use the Or filter to get GIS attributes that meet at least one criteria in a set of filter rules, specified in the multiple Filter fields.



34.7.5.5 Delete a filter

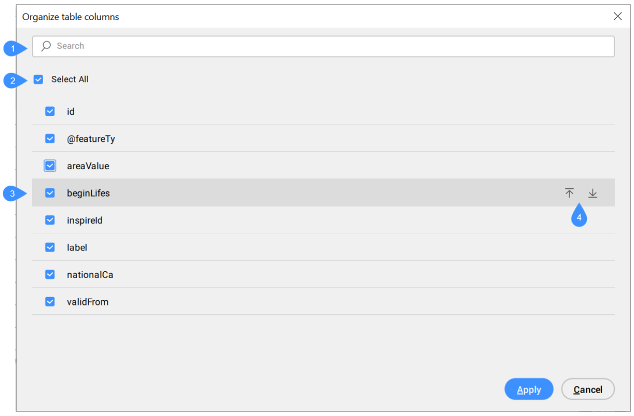
Deletes an expression field for the filter.

34.7.5.6 Apply

Displays the filtering result.

34.7.5.7 Change column layout

Opens **Organize table columns** dialog box, which allows you to change several columns behavior at once, unhide a column, or change the order of columns.



1 Search

2 Select All

3 Check box

4 Up and down arrows

Search
Allows you to search from a letter or a whole word to find appropriate attributes.

Select All
Allows you to select or deselect all fields at once.

Check box
Allows you to select or deselect a single field.

Up and down arrows
Allows you to move a field up or down in the list.

34.7.5.8 Add GIS features

Allows you to add CAD elements from a drawing to a GIS layer. Attributes from the associated GIS Layer are assigned to such a GIS features, so you can edit their values in the **GIS Attribute table** dialog box.

34.7.5.9 GIS Layer name

Displays the GIS Layer name.

34.7.5.10 GIS data

Displays the attributes of the GIS data.

First column (#) in a dialog is mandatory and shows the row number. When you click a field in the first column, the whole row is selected. When you click a field in the other columns, only this field is selected and you can edit its value. Click a column name to sort the column contents alphabetically.

To select multiple rows, select the first field, hold down the **Shift** key and select the last field.

A context menu opens when you right-click the selected field(s).

Zoom to
Zooms to selected features/entities in the drawing.

Pan to
Pans to the selected features/entities in the drawing.

Select
Selects features/entities in the drawing.

Copy to clipboard
Copies selected content to the clipboard.

Delete feature
Deletes selected row/feature from the table. Allows you to remove only the attribute data or the geometry in the drawing as well.

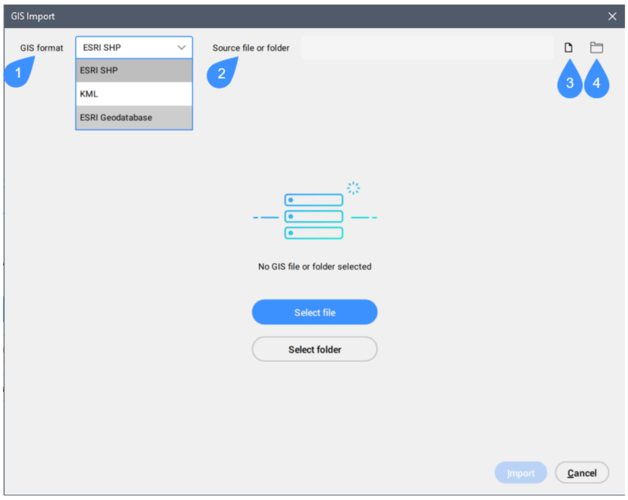
34.7.5.11 Select all

Selects all GIS data in the drawing.

34.7.6 GIS Import dialog box

Opens via: GISIMPORT command

The **GIS Import** dialog box allows you to import different GIS Features (Layers) with shapes and attributes from SHP, KML/KMZ, or Geodatabase files into the current drawing. You can also specify the properties for each imported Feature.



1 GIS format

2 Source file or folder

3 Select the source file

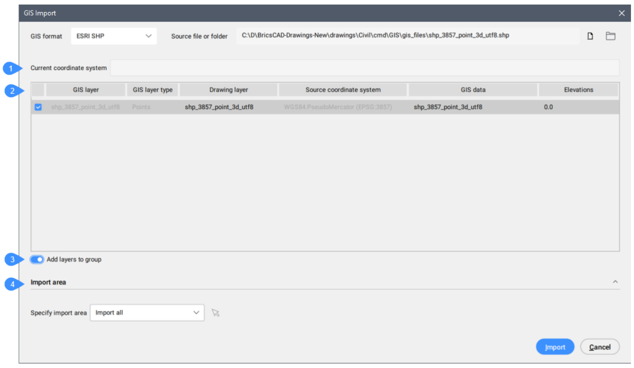
4 Select the source folder

GIS format
Defines the GIS format.

Source file or folder
Displays the file source path.

Select the source file
Opens the **Open file** dialog box to select a GIS file.

Select the source folder
Opens the **Choose input folder** dialog box to select a folder that contains the GIS files.



1 Current Coordinate System

2 Table of imported GIS layers

3 Add layers to group

4 Import Area

5 Import

34.7.6.1 Current Coordinate System

Reports the GIS coordinate system being used by the current drawing.

34.7.6.2 Table of imported GIS Features

Lists the GIS Layers to import and allows you to define their properties.

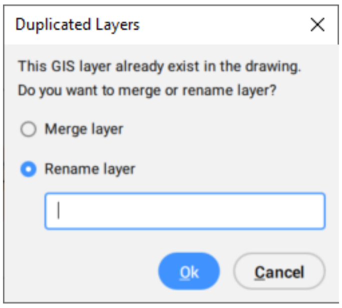
Selected
Allows you to select/deselect GIS features to import.

Note: A warning message appears when nothing is selected for import.

GIS layer
Specifies the name of the GIS layer.

Note: Importing a duplicate GIS layer by renaming or merging it is possible. Follow the indications from the warning message.

Press the ⚠ button to open the Duplicated Layers dialog box.



GIS layer type
Specifies the layer type, for example lines, polygons, points, etc.

Drawing layer
Specifies the drawing layer on which the entities are created.
Click the **Drawing layer** field of the GIS feature to open the **Drawing Layer** dialog box to set the layer.

Note: Importing a duplicate GIS layer by renaming or merging it is possible. Follow the indications from the warning message.

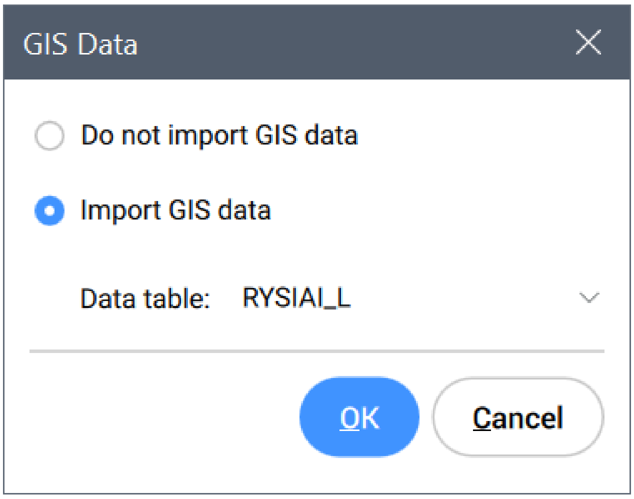
Existing layer
Allows you to choose an existing layer from current drawing.

New layer
Allows you to create a new layer.

Layer from data field
Allows you to choose a layer from the data field.

Source Coordinate System
Specifies the source coordinate system that is used in the imported GIS file, which is read-only.

GIS Data
Specifies the GIS Data.
Click the **GIS Data** field of the GIS feature to open the **GIS Data** dialog box where you can choose whether or not to import the GIS data.



Do not import GIS data
If selected, the GIS data is not imported.

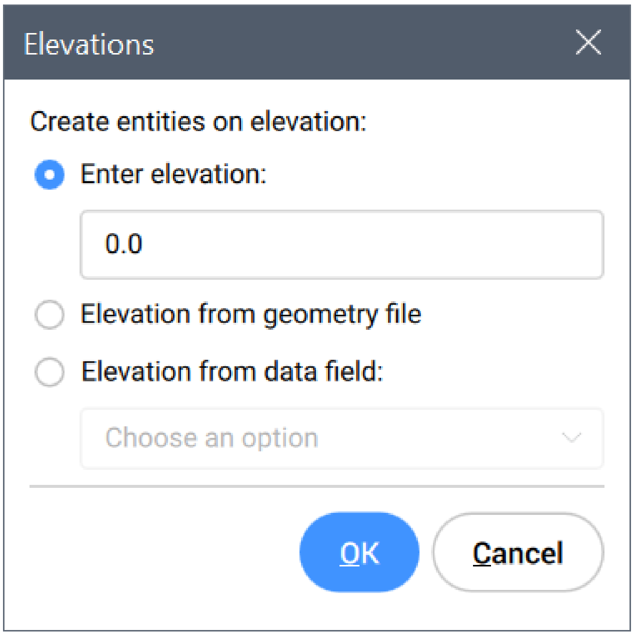
Import GIS data
If selected, it imports the GIS data from the specified data table.

Note: When an entity with GIS data is copied, the GIS data is transferred with it.

GIS data linked to entities is maintained when:

- Entity with GIS data is copied in the current drawing.
- You create a block definition as an external DWG file using the WBLOCK command.
- Entity with GIS data is moved between drawings using the COPY and PASTE commands.

Elevations
Specifies the elevation at which entities are created.
Click the **Elevations** field of the GIS feature to open the **Elevations** dialog box to set the elevation.



Enter elevation
Allows you to enter the elevation value.

Elevation from geometry file
Uses the elevation from the geometry file.

Elevation from data field
Uses the elevation from the specified data field.

34.7.6.3 Add layers to group

Adds the imported GIS layers to a new group, visible under the **GIS** tab from the **Civil Explorer** panel.

34.7.6.4 Import Area

Allows you to specify the import area.

Specify import area

Import All

Uses the entire drawing as import area.

Select clipping polygon

Click the mouse icon to select a polygon in the drawing that defines the import area.

Draw clipping polygon

Click the mouse icon to specify an import area in the drawing.

34.7.6.5 Import

Imports the GIS features in the drawing and closes the **GIS Import** dialog box.

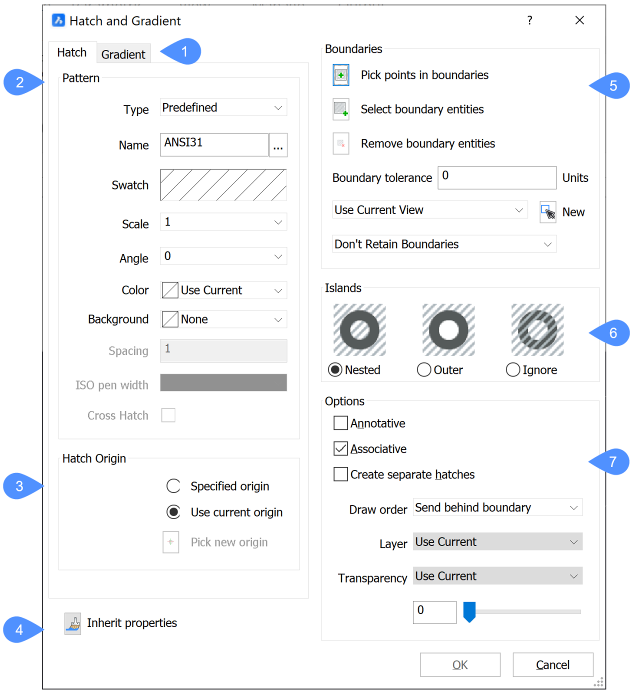
34.8 H

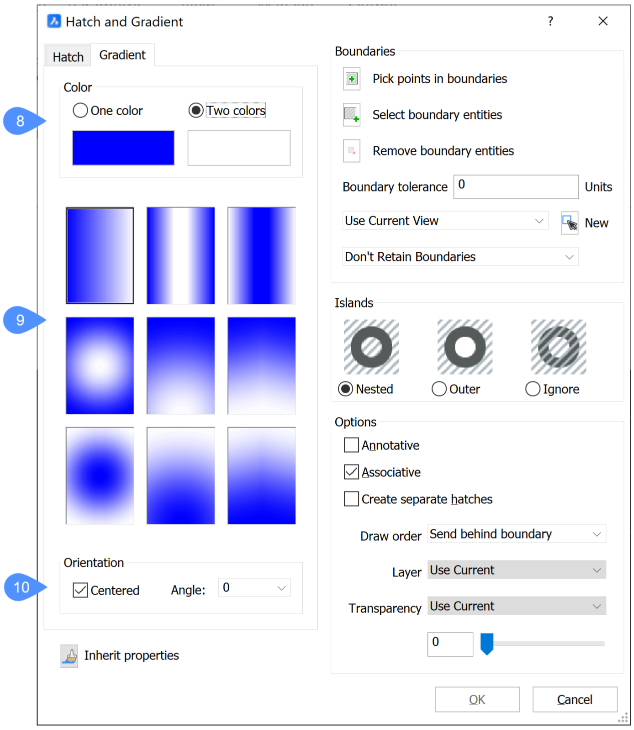
34.8.1 Hatch and Gradient dialog box

Opens via:

- GRADIENT command
- HATCH command

The **Hatch and Gradient** dialog box allows you to fill closed 2D areas with repeating patterns or solid colors.





1 Hatch and Gradient tabs

2 Hatch Pattern

3 Hatch origin

4 Inherit properties

5 Boundaries

6 Islands

7 Options

8 Gradient Color

9 Gradient pattern

10 Gradient Orientation

34.8.1.1 Pattern

Specifies the hatch pattern properties.

Type

Specifies the type of hatch.

User defined

Constructs the pattern from the user-defined parameters of Angle, Spacing, Color, Background and Cross Hatch.

Predefined

Uses the patterns defined in the iso.pat (metric units) or default.pat (imperial units) pattern definition files provided with the program.

Custom
Uses a pattern defined by *.pat files (limited to one pattern definition per file) created by users. The program searches for *.pat files in the folders that are defined in the SRCHPATH system variable.

Name
Specifies the name of the predefined hatch pattern by one of these methods. Click the **Browse** button to display the **Hatch Pattern Palette** dialog box, to choose a pattern.

Note: This option is unavailable for the User-defined patterns.

Swatch
Displays the **Hatch Pattern Palette** dialog box.

Note: This option is unavailable for the User-defined patterns.

Scale
Sets the scale factor of the pattern. To use an enlarged hatch pattern, enter a value bigger than 1.

Note: The scale factor of hatch patterns is generally the same as that for text and linetypes.

Angle
Sets the angle of the pattern.

Note: This angle is measured relative to the positive x-axis of the current UCS.

Color
Sets the color of the hatch pattern lines.

Note: For the **Use Current** option, the current color specified by the CECOLOR system variable is used.

Background
Sets the color of the background of the hatch.

Spacing
Specifies the distance between hatch lines.

Note: This option is available for only user defined patterns.

ISO pen width
Specifies the lineweight.

Note: This option is available only for ISO hatch patterns.

Cross Hatch
Determines whether the pattern is crossed (repeated at 90 degrees to the original).

Note: This option is available only for user defined patterns.

34.8.1.2 Hatch origin

Specifies the origin of the hatch. Either the current or a new specified origin can be chosen.

34.8.1.3 Inherit properties

Copies the properties of another existing hatch pattern to use with this pattern.

34.8.1.4 Boundaries

Specifies the boundary conditions of the hatch.

Pick points in boundaries
Specifies the closed areas (boundaries) in which to place the pattern.

Note: You cannot pick an area that already contains a hatch pattern, or areas that are not closed (whose gap is larger than the value specified by **Boundary tolerance**).

Select boundary entities
Selects the entities that make up the hatching boundary to constrain the extent of the pattern.

Note: This option allows you to hatch closed areas, including ones that already contain a hatch. It hatches open areas whose gap is less than the value specified by **Boundary tolerance**.

Remove boundary entities
Remove entities from the detected boundary set.

Boundary tolerance
Specifies the largest gap the program ignores when hatching a boundary that is not fully closed.

- 0 - (Default): the tolerance is set by the application, based on the current view size. When zoomed in closely, boundary detection will fail; when zooming out further so the contour 'looks' closed, the boundary will become detected
- Any Value : defines the maximum gap in drawing units.

Note: The value is saved in the HPGAPTOL system variable.

Use current view / boundary set
Specifies where the program should search for entities that make up the boundary.

New
Creates a new selection set of entities that make up the pattern boundary.

Note: The next step is to click the **Pick Points in boundaries** button to choose the hatch area.

Retain boundaries
Determines what happens to boundaries.

Don't Retain Boundaries
Removes the boundary after the hatch is created.

Retain Boundaries as Polylines
Retains boundaries and turns them into polylines.

Retain Boundaries as Regions
Retains boundaries and turns them into regions (Pro or Superior versions only).

34.8.1.5 Islands

Specifies how BricsCAD will respond when other boundaries are present within the closed hatch boundary.

Nested
When a closed hatch boundary contains other boundaries, BricsCAD hatches alternating areas.

Outer
When a closed hatch boundary contains other boundaries, BricsCAD hatches only the outermost area.

Ignore
When a closed hatch boundary contains other boundaries, BricsCAD hatches all interior areas, as if they were not present.

34.8.1.6 Options

Annotative
Toggles the Annotative property of the hatch. When turned On, BricsCAD applies the current annotative scale.

Note: When this option is turned on, the **Associative** option is unavailable.

Associative
Toggles the associativity of hatch patterns: when the boundary is changed, the pattern automatically updates itself.

Note: This option is not available for annotative hatches.

Create separate hatches
Toggles how multiple hatches are treated. If turned On, BricsCAD creates a separate hatch entity for each boundary in the selection set.

Draw order
Specifies where the pattern should be placed relative to overlapping entities.

Layer
Specifies the layer on which the hatch should be placed.

Transparency
Set the transparency property of the hatch.

Note: For the **Use current** option, the transparency value, defined by the CETRANSPARENCY system variable, is applied.

34.8.1.7 Gradient Color

Specifies the gradient color. The gradient can include either one color or two colors to create a solid fill hatch.

34.8.1.8 Gradient pattern

Specifies the pattern of the gradient.

34.8.1.9 Gradient Orientation

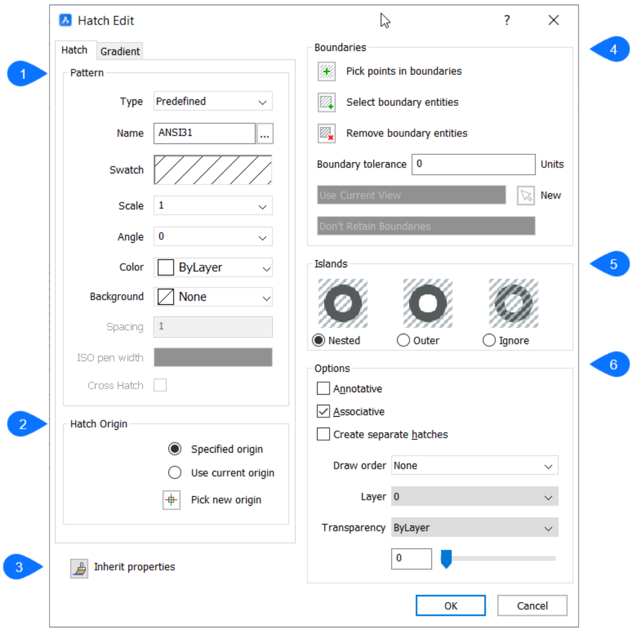
Specifies how the fill is placed in the boundary.

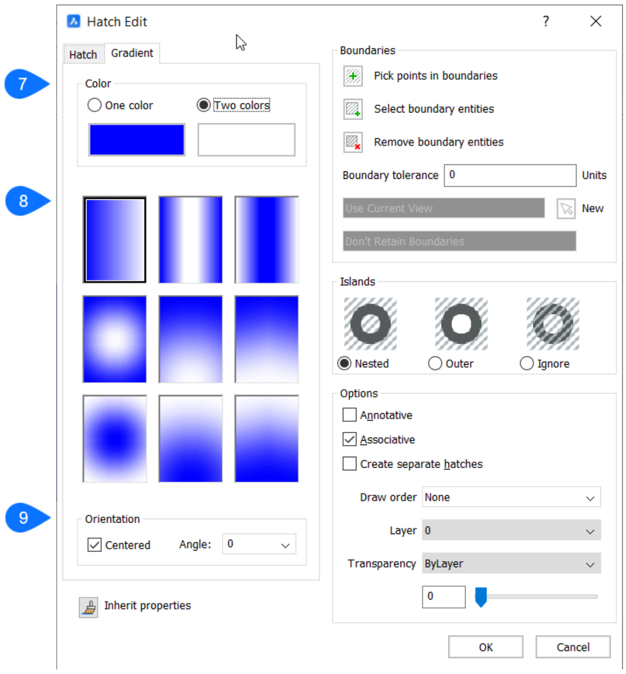
- Centered: centers gradient fill in the area to be filled; when off, starts the fill from the upper left of the boundary.
- Angle: select an angle of the gradient fill, relative to the current UCS; choose from increments on 15 degrees, or else enter any other value for the angle.

34.8.2 Hatch Edit dialog box

Opens via: HATCHEDIT command

The **Hatch Edit** dialog box allows you to edit an already existing hatch in the drawing. All options resemble the options of the **Hatch and Gradient** dialog box.





1 Pattern

2 Hatch origin

3 Inherit properties

4 Boundaries

5 Islands

6 Options

7 Color

8 Pattern

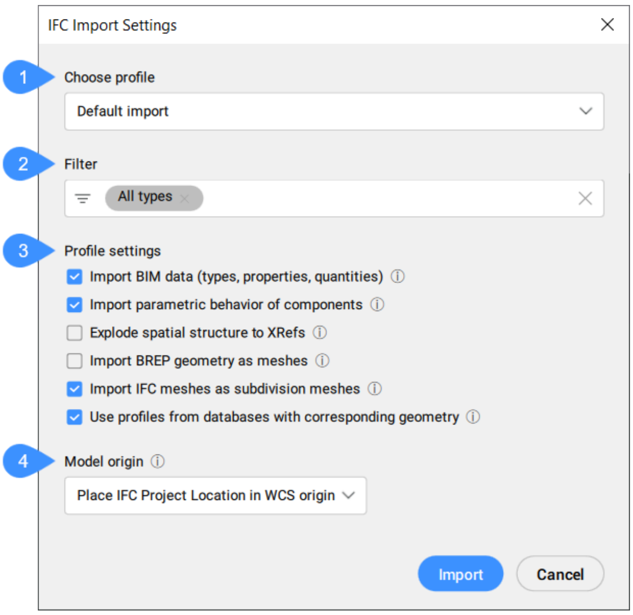
9 Orientation

34.9 I

34.9.1 IFC Import Settings dialog box

Opens via: IMPORT command

The **IFC Import Settings** dialog box lets you control how the IFC models are imported (see the **IMPORT command** article).



1 Choose profile

2 Filter

3 Profile settings

4 Model origin

34.9.1.1 Choose profile

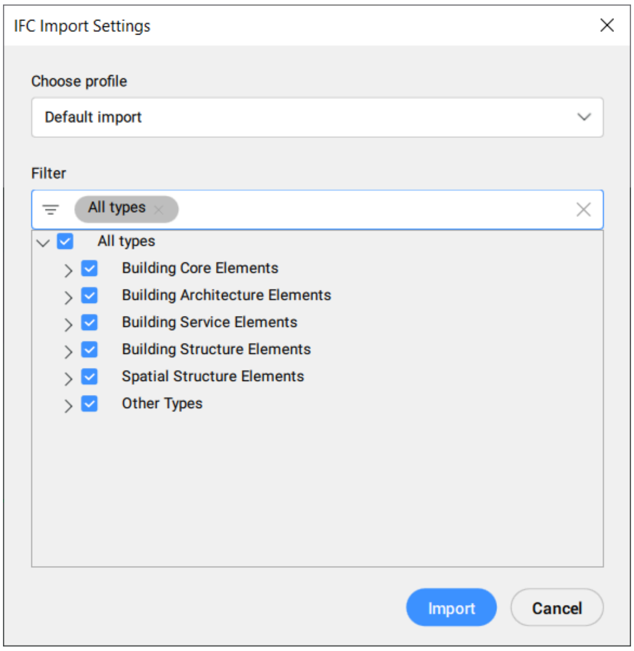
Sets a profile for importing the IFC model. The predefined profiles are:

- **Default import**.
- **Optimized for referencing (favor meshes and exclude furniture)**: imports BREP geometry as meshes during IFC import.
- **Import all**.
- **Geometry only**.

- **Architectural Elements only**.
- **Structural Elements only**.
- **MEP Elements only**.

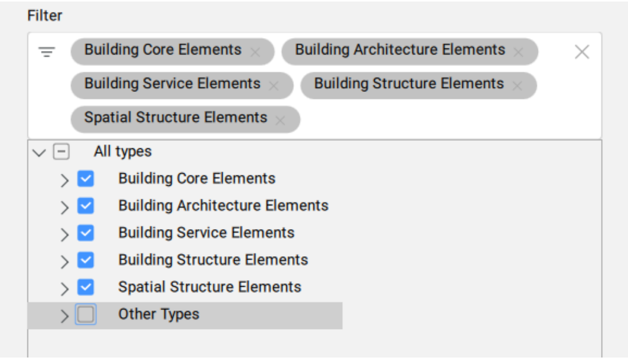
34.9.1.2 Filter

Creates a filter for importing based on element Types. Click the filter icon to toggle between the current **Profile settings** and **Filter** criteria.

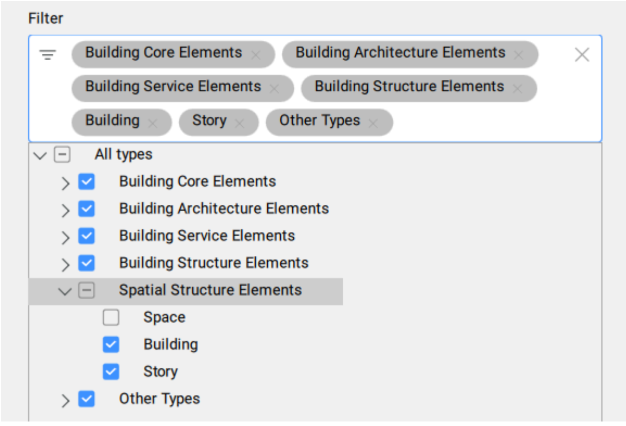


The criteria are displayed as a hierarchical structure and as tags. By default, all types of elements are selected to be imported.

After deselecting a category of element types in the tree list, the **All types** tag is replaced by the remaining categories tags.



After deselecting an element type under a category in the tree list, the category tag is replaced by the remaining entity type tags.



Note: When modifying the **Filter** criteria, the changes are saved in a temporary profile with the same name as the original profile, followed by **(Modified)**. This also applies when modifying predefined **Profile settings**. The **IFC Import Settings** dialog box loads this temporary profile for the following IFC import operation.

34.9.1.3 Profile settings

Defines the profile settings:

Import BIM data (types, properties, quantities)
Imports BIM data during IFC import (types, properties, quantities).

Import parametric behavior of components
Imports parametric windows and doors from the IFC file as parametric components in BricsCAD.

Explode spatial structure to XRefs
Imports each building and storey as a separate XRef.

Import BREP geometry as meshes
Imports BREP (Boundary Representation) geometry as meshes during IFC import. This can give a more accurate result for certain complex BREP geometry.

Import IFC meshes as subdivision meshes
Imports IFC meshes as subdivision meshes. When OFF, it uses Polyface meshes to import meshes from the IFC file.

Note:

- A polyface mesh is limited to 32767 faces or vertices.
- A subdivision has no restrictions.

Use profiles from databases with corresponding geometry
Compares profiles with the project and central database based on their geometry during IFC import. If it matches, the parameters defined in the project/central database (profile name, size, and standard) are used instead of imported parameters.

34.9.1.4 Model origin

Defines where to position the model with reference to the World Coordinate System (WCS) origin. Placing the IFC Global Coordinate System in WCS origin can result in a model positioned far from the origin.

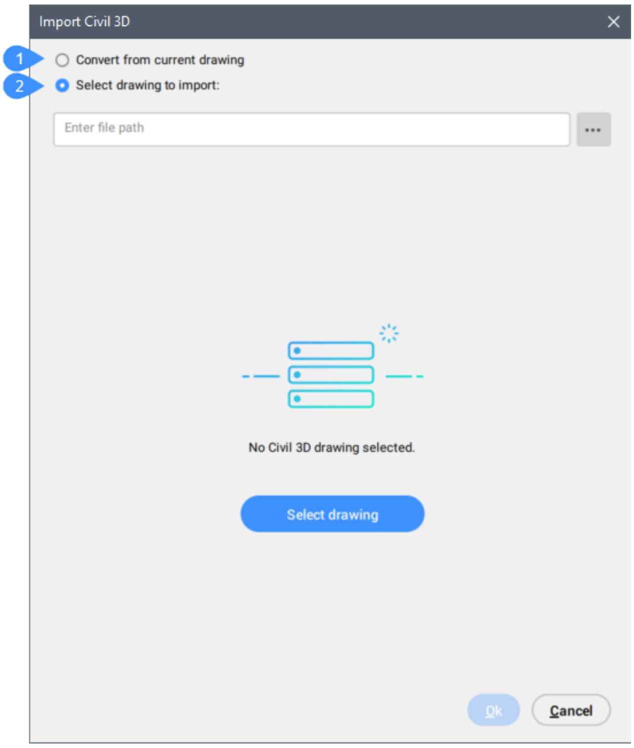
The options are:

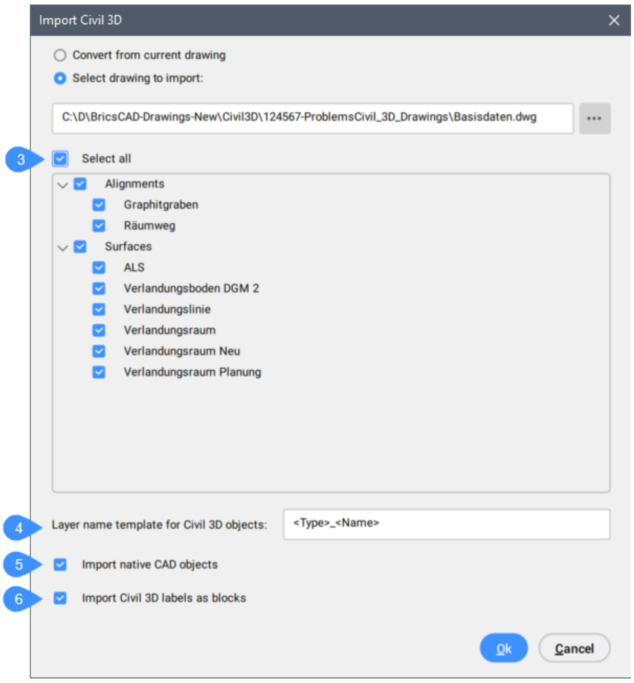
- **Place IFC Global Coordinate system in WCS origin**.
- **Place IFC Project Location in WCS origin**.
- **Place IFC Site Location in WCS origin**.

34.9.2 Import Civil 3D dialog box

Opens via: CIVIL3DIMPORT command

The **Import Civil 3D** dialog box allows you to create BricsCAD civil entities from an AutoCAD® Civil 3D drawing.





1 Convert from current drawing

2 Select drawing to import

3 Select All

4 Layer name template for Civil 3D objects

5 Import native CAD objects

6 Import Civil 3D labels as blocks

34.9.2.1 Convert from current drawing

Converts AutoCAD® Civil 3D objects in the current drawing.

34.9.2.2 Select drawing to import

Enables to enter drawing path or click the **Select drawing** button to select the drawing manually.

Note: When either the DWG file is specified or the **Convert From Current Drawing** option is selected, AutoCAD® Civil 3D objects are listed in a tree view, organized by object type.

34.9.2.3 Select All

Enables to select all AutoCAD® Civil 3D objects in the selected drawing.

34.9.2.4 Layer name template for Civil 3D objects

Enables to specify the Layer name template for imported AutoCAD® Civil 3D objects.

The following settings are available for defining the Layer name template:

- **<Type>**: includes AutoCAD® Civil 3D object type in the Layer name.

- **<Name>**: includes AutoCAD® Civil 3D object name in the Layer name.
- **<SourceLayer>**: Civil entities are created on same layer as source AutoCAD® Civil 3D objects.

If you want to draw BricsCAD Civil entities on Layers, which names will consist of three sets: the type of the source Civil 3D object, the object name and the name of the Layer of the source AutoCAD® Civil 3D object, you need to write the following text in the field:

```
<Type>_<Name>_<SourceLayer>
```

34.9.2.5 Import native CAD objects

Enables to import native CAD objects from the selected drawing in addition to AutoCAD® Civil 3D objects.

34.9.2.6 Import Civil 3D labels as blocks

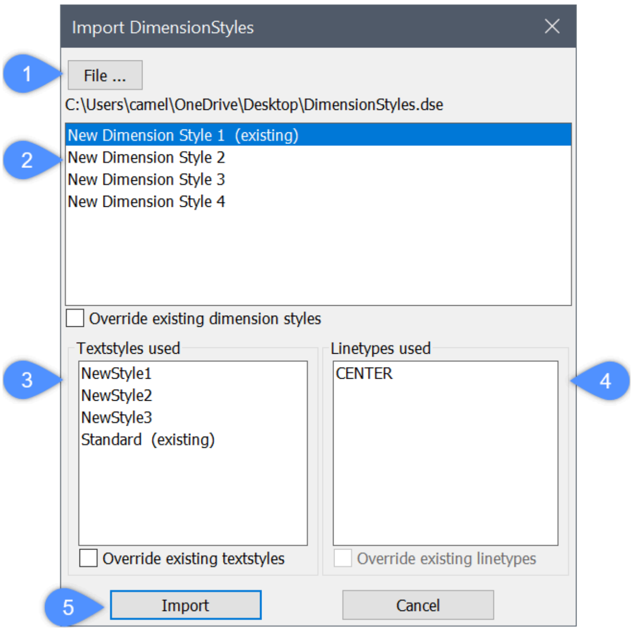
Enables to import AutoCAD® Civil 3D labels as blocks from the selected drawing.

Note: AutoCAD® Civil 3D COGO points labels are converted to BricsCAD Civil point labels.

34.9.3 Import DimensionStyles dialog box

Opens via: DIMIM command (Express Tools)

The **Import DimensionStyles** dialog box allows you to import dimension styles and their settings from an external file.



1 File...

2 Dimension styles used

3 Textstyles used

4 Linetypes used

5 Import

34.9.3.1 File...

Opens the **Select import file** dialog box that allows you to choose a DSE file to import. All dimension styles in the selected file are imported into the current drawing, including the text styles and linetypes.

Note: The DSE file must have the same format as that generated by the DIMEX command.

34.9.3.2 Dimension styles used

Displays the dimension styles from the DSE file and allows you to select the dimension styles to import into the current drawing.

Note: Tick the **Override existing dimension styles** option to overwrite the dimension styles in the current drawing and use the similarly named dimension style from the DSE file.

34.9.3.3 Textstyles used

Displays the text styles from the DSE file.

Note: Tick the **Override existing textstyles** option to overwrite the text styles in the current drawing and use the similarly named text style from the DSE file.

34.9.3.4 Linetypes used

Displays the linetypes from the DSE file.

Note: Tick the **Override existing linetypes** option to overwrite the linetypes in the current drawing and use the similarly named linetypes from the DSE file.

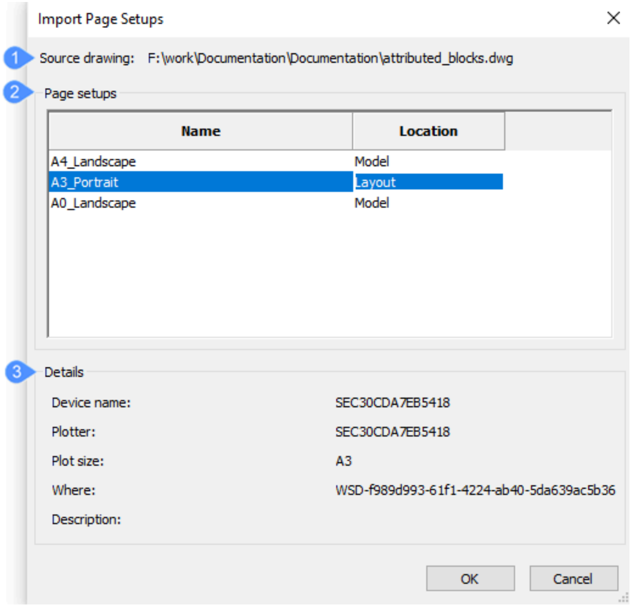
34.9.3.5 Import

Imports the selected dimension styles from the DSE file.

34.9.4 Import Page Setups dialog box

Opens via: PSETUPIN command

The **Import Page Setups** dialog box allows you to select one or more page setups (using the **Ctrl** key) from the list and import them into the current drawing.



1 Source Drawing

2 Page setups list

3 Details

34.9.4.1 Source Drawing

Displays the source of the drawing used to import the page setup(s).

34.9.4.2 Page setups list

Specifies the name and the location of each page setup from the list:

- **Name**: specifies the names of page setups in the drawing. When the drawing contains no page setups, then the dialog box is blank.
- **Location**: specifies the layout and model locations of page setups.

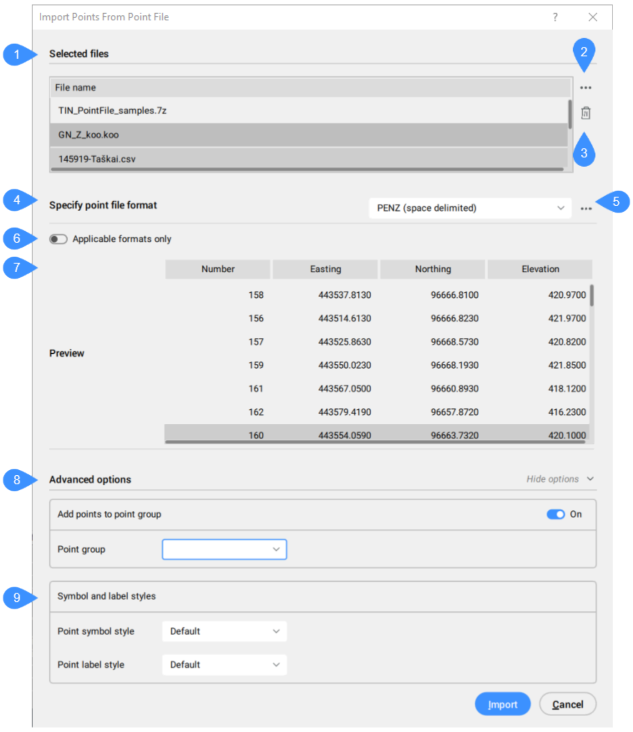
34.9.4.3 Details

Includes information about the device name, plotter, plot size, location and a description field.

34.9.5 Import Points From Point File dialog box

Opens via: CIVILPOINT command > **Import points from file** option

The **Import Points From Point File** dialog box allows you to select one or more point files to create Civil points.



1 List of selected point files

2 Open point file

3 Delete point file

4 Specify point file format

5 Manage point file formats

6 Applicable formats only

7 Preview

8 Advanced options

9 Symbol and label styles

34.9.5.1 List of selected point files

Displays the selected point files for import into the drawing.

34.9.5.2 Open file

Opens the **Open file** dialog box to select a point file. The input point file can be any ASCII file of points with any extension, any separator between columns and arbitrary arrangement of data in columns. Multiple point files could be selected.

34.9.5.3 Delete file

Removes the point file from the list.

34.9.5.4 Specify point file format

Selects a point file format from the opened drop-down list.

34.9.5.5 Manage point file formats

Opens the **Manage Point File Formats** dialog box (see the related article **Manage Point File Formats dialog box**).

34.9.5.6 Applicable formats only

If checked, only formats that are applicable to all files in the list are shown.

34.9.5.7 Preview

Displays all point's coordinate and details (such as position number, full description, or raw description).

34.9.5.8 Advanced options

Allows you to add the points into a Point group.

Add points to point group
Allows you to choose if you add the points in a new or existing point group.

Point group
Allows you to select the group's name or type a new group name.

34.9.5.9 Symbol and label styles

Allows you to select the symbol and label styles for the imported points.

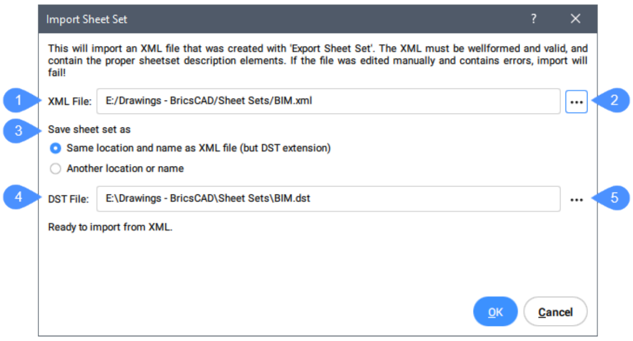
Point symbol style
Allows you to choose the point symbol style for the imported points.

Point label style
Allows you to choose the label symbol style for the imported points.

34.9.6 Import Sheet Set dialog box

Opens via: **Sheet Sets** panel

The **Import Sheet Set** dialog box allows you to import a sheet set file that has been created using the **Export Sheet Set As** dialog box.



1 XML File path

2 Select XML File

3 Save Sheet Set As

4 DST File path

5 Save Sheet Set As

34.9.6.1 XML File path

Displays the path of the *.xml file after selecting it using the **Select XML File** button.

Note: A warning message appears when trying to import a sheet layout which already exist in a sheet set.

The DST file already exists. If you click OK, it will be overwritten!

34.9.6.2 Select XML File

Opens the **Select XML File** dialog box.

34.9.6.3 Save Sheet Set As

Choose how to save the imported XML file as DST file using:

- Same location and name as XML file (but DST extension).
- Another location or name. Clicking this button, will open the **Save Sheet Set As** dialog box.

34.9.6.4 DST File path

Displays the DST file path and name.

34.9.6.5 Save Sheet Set As

Opens the **Save Sheet Set As** dialog box. This button is active only when saving the sheet set in another location or name than the XML imported file.

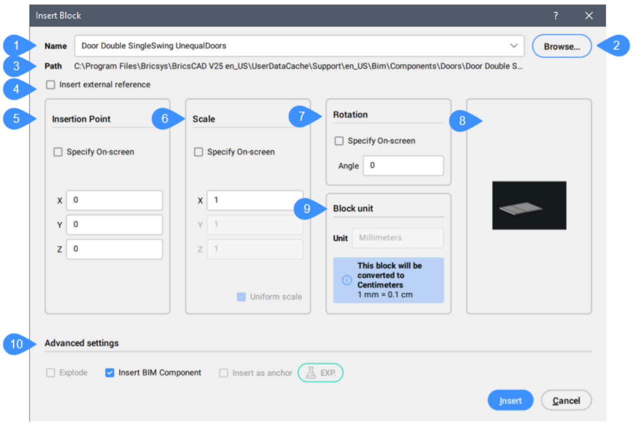
34.9.7 Insert Block dialog box

Opens via: INSERT command

The **Insert Block** dialog box allows you to insert blocks into the current drawing.

Note:

- You can insert entire DWG or DXF files as blocks into the drawing.
- You can insert parametric blocks.
- If the block contains attributes, then you are prompted to enter values for the attributes in the Command line or through **Edit Attributes** dialog box, depending on the value of the ATTDIA system variable.
- When working with drawings containing mechanical components, the dialog box displays a warning message recommending using the BMINSERT command instead.
- INSERT might not work correctly in this drawing. Please use BMINSERT instead if you notice any issues.



1 Name

2 Browse

3 Path

4 Insert external reference

5 Insertion Point

6 Scale

7 Rotation

8 Block preview

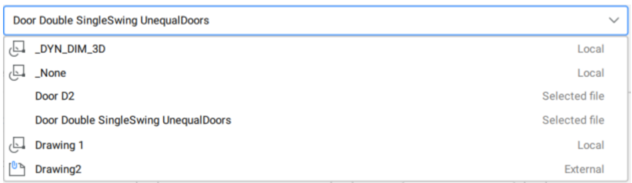
9 Block unit

10 Advanced settings

34.9.7.1 Name

Specifies the name of the block to insert, whose definition exists in the drawing, or the name of the file to insert as a block.

Note: Click the drop-down button to display the available blocks. It shows internal and external block types, with icons indicating each block's type.



34.9.7.2 Browse

Opens the **Select Drawing File** dialog box that allows you to select a DWG or DXF file from your computer or network.

34.9.7.3 Path

Displays the path to the file to insert as a block.

34.9.7.4 Insert external reference

Toggles whether the block is inserted as a local or external reference.

34.9.7.5 Insertion Point

Specifies the insertion point of the block in the drawing.

Specify On-screen
Allows you to dynamically select the insertion point of the block in the current drawing.

34.9.7.6 Scale

Scales the block.

Specify On-screen
Allows you to dynamically scale the block in the current drawing.

Input coordinates
Defines the scaling of the block.

X
Allows you to specify the scale factor along the X axis. Enter a negative value to mirror the block about the Y axis.

Y
Allows you to specify the scale factor along the Y axis. Enter a negative value to mirror the block about the X axis.

Z
Allows you to specify the scale factor along the Z axis.

Note:

- Values larger than 1 make the block larger.
- 1 inserts the block at actual size.
- Values smaller than 1 make the block smaller.
- Values less than zero flip the block, like mirroring it.

Uniform scale
Toggles whether the same scale factor is used for all axes.

Note: This option is enabled by default when inserting blocks created with the **Scale uniformity** option enabled (see the **BLOCK command** article).

34.9.7.7 Rotation

Specifies the rotation angle of the block about its insertion point, starting with the X-axis as 0 degrees.

Note: Positive angles rotate the block counter-clockwise and negative angles rotate the block clockwise.

Specify On-screen
Allows you to dynamically rotate the block in the current drawing.

34.9.7.8 Block preview

Displays a preview of the block.

34.9.7.9 Block unit

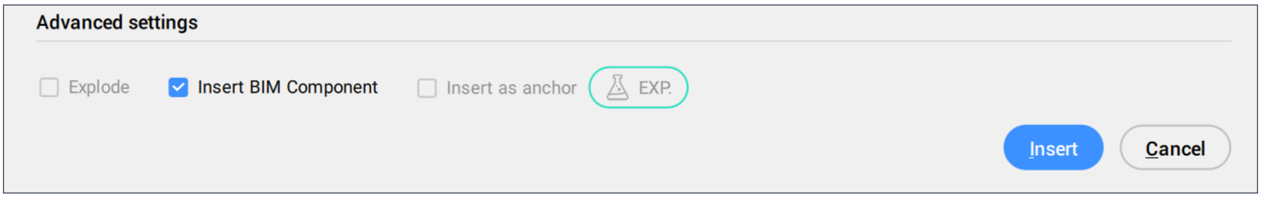
Controls the automatic scaling of the block with respect to the INSUNITS and INSUNITSSCALING system variables of the current drawing.

34.9.7.10 Advanced settings

Displays the available advanced settings, depending on the type of insert.

Explode
Toggles whether to explode the inserted block.

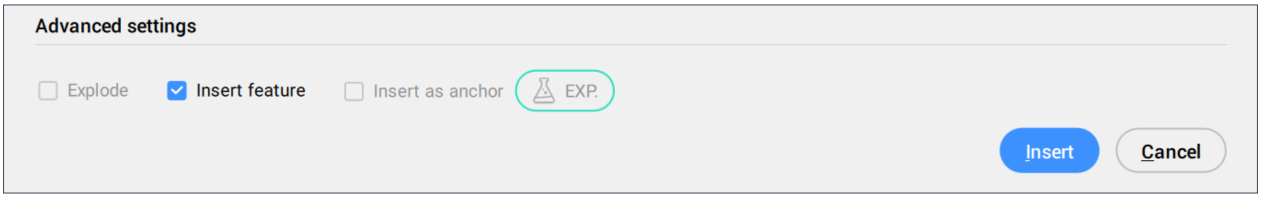
Insert BIM Component
Toggles whether to insert the block as a BIM Component.



Note:

- The **Insert BIM Component** option is available only if an external BIM component block is selected for insertion.
- This option is available for a BricsCAD BIM license. If you are using a BricsCAD Pro or a BricsCAD Mechanical license, the option becomes **Insert feature**.

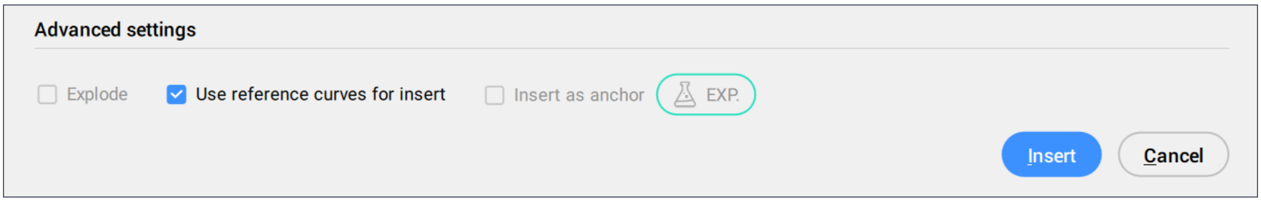
Insert feature
Toggles whether to insert the block as feature.



Note:

- The **Insert feature** option is available if an external mechanical component block is selected for insertion.
- BricsCAD Pro or higher is required for this option.

Use reference curves for insert
Toggles whether reference curves are used for insert.

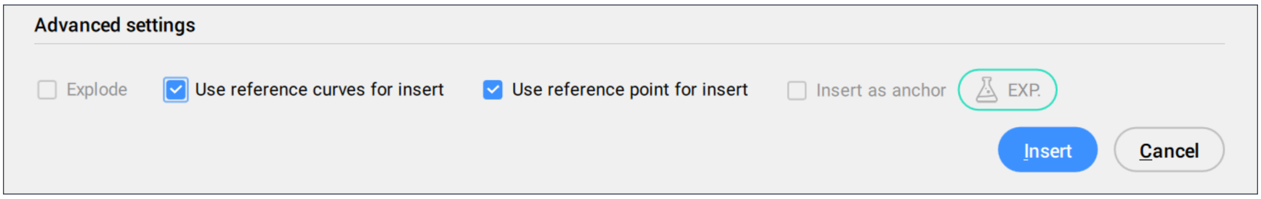


Note:

- The **Use reference curves for insert** option is only available if the selected drawing contains reference curves. See also the **Fuzzy insert guided workflow** article.
- BricsCAD Pro or higher is required for this option.

Use reference point for insert

Toggles whether a reference point is used for insert.



Note:

- The **Use reference point for insert** option is only available if the selected DWG file contains a detail with guide curves. Details with guide curves can be created using the COPYGUIDED and MOVEGUIDED commands.
- It is available only when the **Use reference curves for insert** option is enabled. Otherwise, the option becomes gray out.
- BricsCAD Pro or higher is required for this option.

Insert Sheet Metal Form Feature

Toggles whether to insert sheet metal form feature.

Note:

- The **Insert Sheet Metal Form Feature** option is available only if a sheet metal form features block is selected for insertion and the Model Space contains SM flanges.
- This option is available for a BricsCAD Pro license or higher.

Insert as anchor

Anchors a block to a face of a solid. This anchor links the block to the face. The block will stay on the face when the host solid is moved or edited.

Note: BIMANCHOR is an experimental feature. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

34.9.8 Insert Block in Cell dialog box

Opens via:

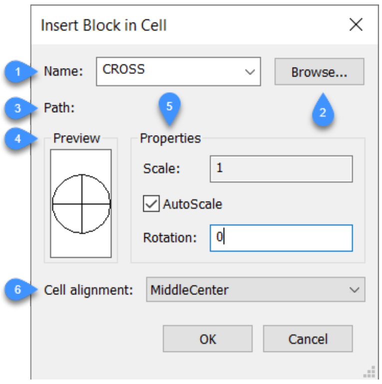
- TINSERT command

- **Table** toolbar

The **Insert Block in Cell** dialog box allows you to insert blocks into the cell of a table in the current drawing. You can also insert DWG and DXF files as blocks.

Note: You can have both text and blocks in a single cell.

Note: If the block contains attributes, then you are prompted to enter values for the attributes in the Command line or through the **Edit Attributes** dialog box, depending on the value of the ATTDIA (attribute dialog) variable.



1 Name

2 Browse

3 Path

4 Preview

5 Properties

6 Cell alignment

34.9.8.1 Name

Specifies the name of the block, whose definition exists in the drawing. It can also be a DWG or DXF file on the computer or network.

34.9.8.2 Browse

Selects a DWG or DXF file from your computer or network. Opens the **Open drawing file** dialog box.

Note: Placing a large drawing in a small cell can lead to program instability.

34.9.8.3 Path

Displays the path to the block, if the block was opened from a DWG or DXF file.

34.9.8.4 Preview

Displays a preview of the selected block definition.

Note: When the cell is small, the block or drawing may be barely perceptible. You can increase the size of the row or column to accommodate the blocks. A cell can hold text and one or more blocks. When a 3D model is pasted into the cell, the 2D plan view is displayed.

34.9.8.5 Properties

Defines additional properties of the selected block.

Scale

Specifies the scaling value of the block:

- Values larger than 1 enlarge the block.
- 1 inserts the block at actual size.
- Values smaller than 1 make the block smaller.
- Values less than zero flip the block, like mirroring it.

Note: This option is unavailable when AutoScale is turned on.

AutoScale

Sizes the block to fit the cell:

- **On**: the block is resized to fit the cell.
- **Off**: the cell is resized to fit the block.

Rotation

Rotates the block around its center point (and not its insertion point). Enter an angle:

- Positive numbers rotate the block counter clockwise.
- 0 - does not rotate the block.
- Negative numbers rotate the block clockwise.

Note: The specified rotation angle depends on the AUNITS system variable.

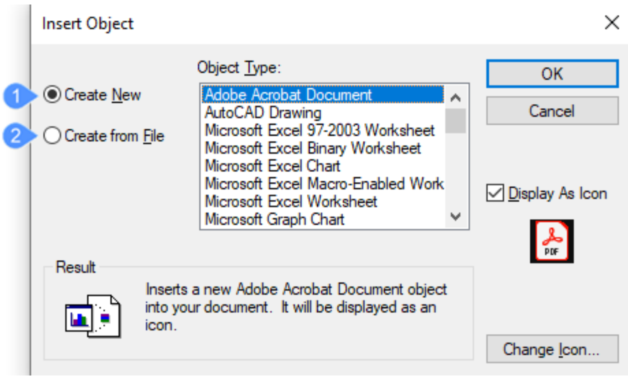
34.9.8.6 Cell alignment

Aligns the block in the cell.

34.9.9 Insert Object dialog box

Opens via: INSERTOBJ command

The **Insert Object** dialog box allows you to insert a linked or embedded document in the current drawing.

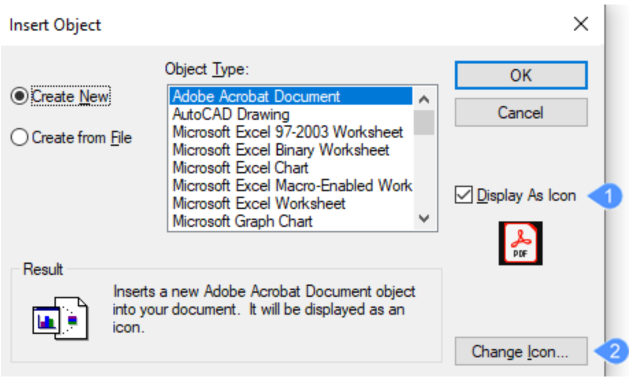


1 Create new Object

2 Create from File

Note: The object types listed varies according the software installed on your computer.

34.9.9.1 Create new Object



1 Display as icon

2 Change icon

Display as icon

Displays the object as an icon. The icon is related to the document type and is defined by source application.

Change icon

Opens the **Change Icon** dialog box. For more information, see the article **Change Icon dialog box**.

34.9.9.2 Create from File

Link

Toggles linkage to the source document:

- **On**: links the object in the drawing to the source file, so that when the source file is changed, the linked object also changes.
- **Off**: does link not the object.

34.9.10 Insert Table dialog box

Opens via: TABLE command

The **Insert Table** dialog box allows you to create a table entity in the drawing, either from scratch or from data in an external file or a data link.

34.9.10.1 From scratch

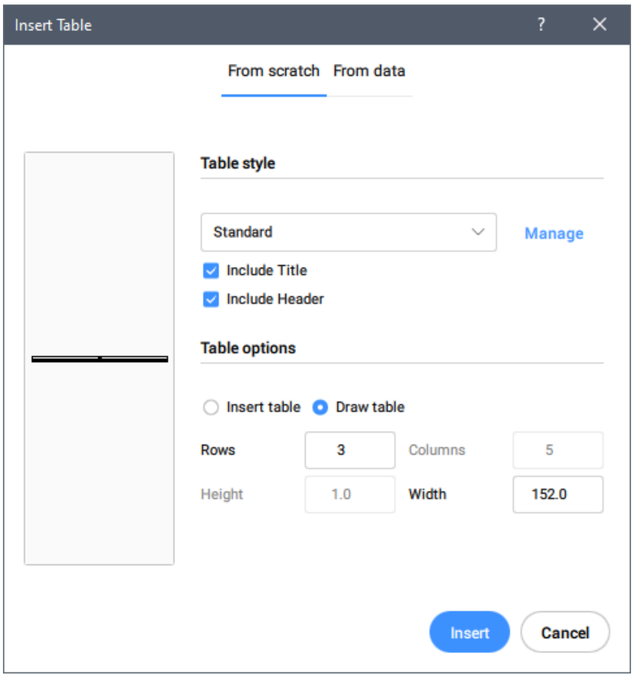


Table Style

Chooses an existing table style from the drop-down list.

Note: The **Manage** button opens **Drawing Explorer** > **Table Styles** .

Table Options

Choose to insert a table by defining its properties or to draw a table.

34.9.10.2 From data

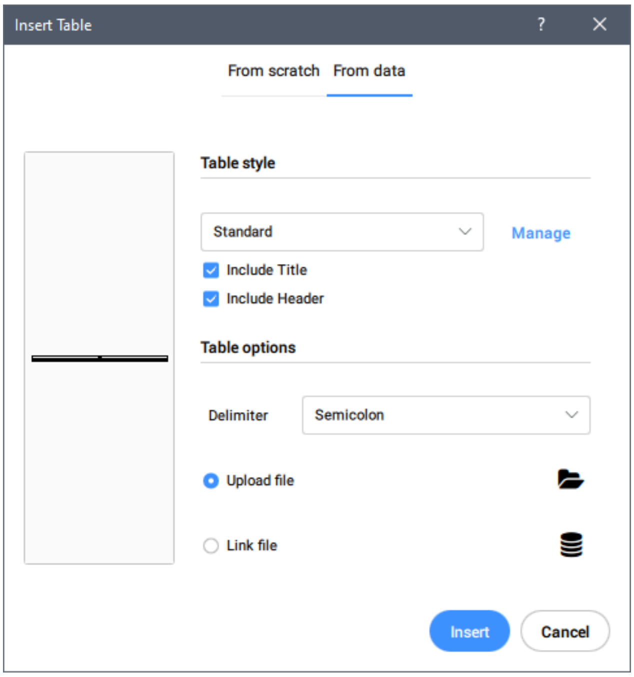


Table options

Delimiter
Specifies the delimiter character used to separate fields in CSV data files. Select the delimiter you want to use from the drop-down list.

Note: Only available for the **File** option.

Upload file
Displays the **Select source file** dialog box to choose a file for the new table.

Link file
Displays the **Datalink Manager** dialog box. See the related article **Datalink Manager dialog box**.

34.9.11 Interface Settings dialog box

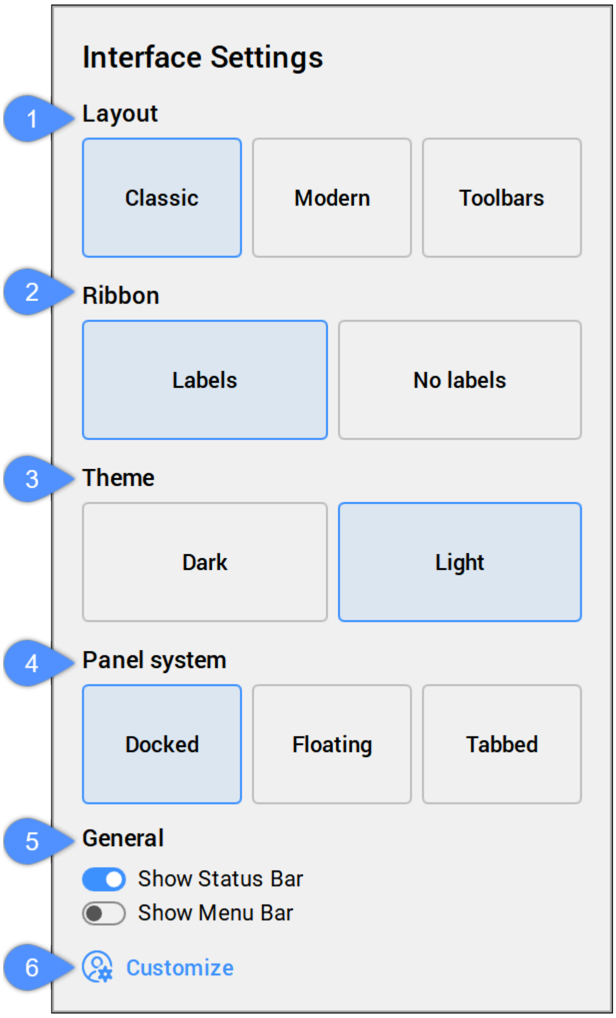
Opens via: **Ribbon** panel

The **Interface Settings** dialog box allows you to manage the interface display of your BricsCAD workspace.

Click the **Interface Settings** button to open the **Interface Settings** dialog box.

In the **Modern** and **Classic** layout, the **Interface Settings** button is located at the top-right of the Ribbon. However, the RIBBONSETTINGSENABLED system variable controls whether the button is displayed.

In the **Toolbars** layout, the **Interface settings** button is located in the **Workspaces** toolbar.



1 Layout

2 Ribbon

3 Theme

4 Panel System

5 General

6 Customize

34.9.11.1 Layout

Defines the general layout of the interface.

Classic
Sets the interface to the classic layout.

Note: The **Classic** interface is the default one.

Modern
Sets the interface to the modern layout.

Toolbars
Sets the interface to a layout using only toolbars (the Ribbon is closed).

34.9.11.2 Ribbon

Defines the displaying mode of the ribbon.

Labels
Displays ribbon labels.

No labels
Hides ribbon labels.

34.9.11.3 Theme

Defines the color theme of the interface.

Dark
Sets the interface theme to dark.

Light
Sets the interface theme to light.

34.9.11.4 Panel System

Defines how panels should be displayed when open.

Docked
Organizes the panels in a docked way, reducing the model space depending on the width of the selected panel.

Floating
Organizes the panels in a floating way. The model space is not reduced in width as panels are floating above.

Tabbed
Organizes the panels in a tabbed way on both sides of the screen.

34.9.11.5 General

Allows you to toggle on and off several interface components:

- **Show Status Bar**
- **Show Menu Bar**

34.9.11.6 Customize

Displays the **Customize** dialog box which allows you to customize the interface to own preferences in a more detailed way.

34.10 J

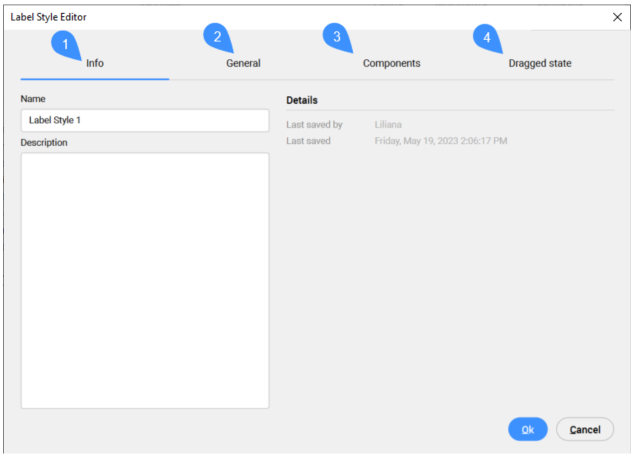
34.11 K

34.12 L

34.12.1 Label Style Editor dialog box

Opens via: **Civil Explorer** panel

The **Label Style Editor** dialog box allows you to label styles.



1 Info tab

2 General tab

3 Components tab

4 Dragged state

34.12.1.1 Info tab

Displays information about the label style.

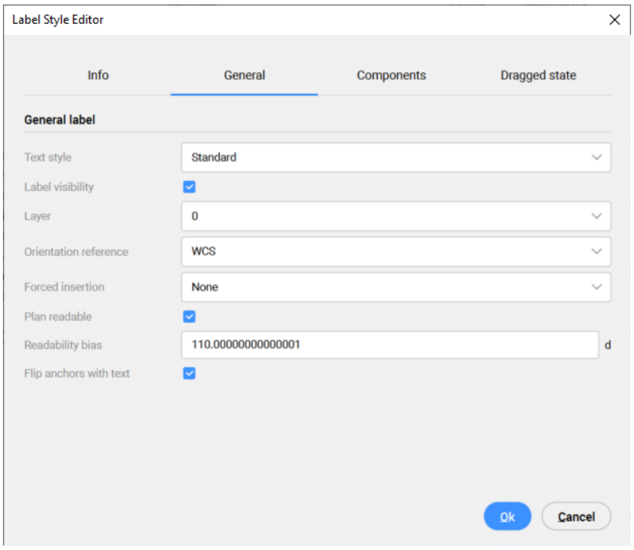
Name
Sets and displays the label style name.

Description
Sets and display the description of the label style.

Details
Displays **Last saved by** and **Last saved** date.

34.12.1.2 General tab

Defines general label settings.



Text style

Sets the text style from the drop-down list.

Note: Use the **Drawing Explorer** to manage text styles.

Label visibility

Toggles the label visibility in the drawing.

Layer

Sets the Layer for the label from the drop-down list.

Note: Use the **Layers** panel to manage layers.

Orientation reference

Sets the orientation reference of labels relative to the associated Object, the screen View orientation, or the World Coordinate System (WCS) from the drop-down list.

Forced insertion

Specifies the position of a label relative to an object:

- **None**: Maintains label position oriented relative to the object.
- **Top**: Modifies label position to be above an object.
- **Bottom**: Modifies label position to be below an object.

Plan readable

Toggles the readability flip parameter.

Readability bias

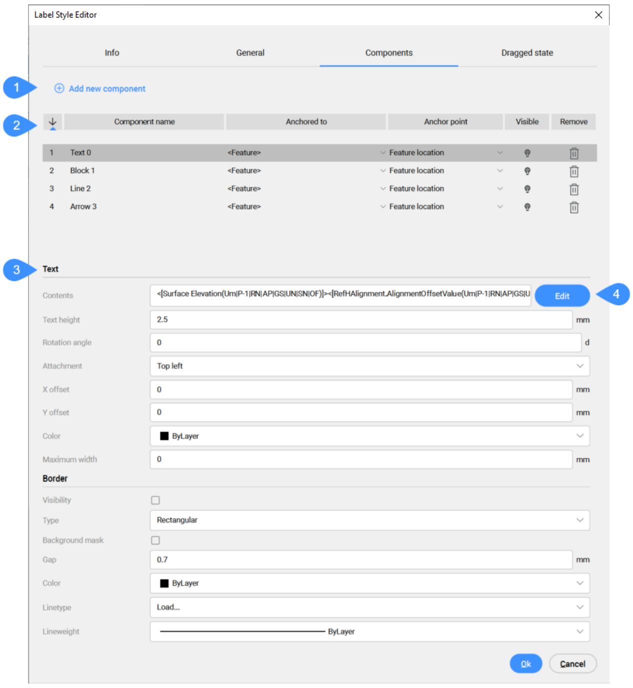
Sets the angle above which the point label text is rotated to a readable angle.

Flip anchors with text

Toggles the flip anchors with the text for the label.

34.12.1.3 Components tab

Adds and manages Label Style components.



1 Add new component

2 List of components

3 Components properties

4 Edit

Add new component

Adds new **Text**, **Block**, **Line**, and/or **Arrow** components to define Label Style.

List of components

Lists the specified components of the Label Style.

Draw order

Displays the components on the draw order. The draw order can be changed by typing the new position number for the component.

Component name

Displays the components names. The list can be sorted alphabetically.

Anchored to

Defines the dependence between Label Style components.

Anchor point

Defines the anchor point of the dependent component by choosing it from the drop-down list.

Visible

Toggles the visibility on the component.

Remove

Removes the component.

Components properties

Displays the properties of the component, which can be edited.

Data / Text component properties

Text

Contents

Displays text content of the Label Style. To add, edit or remove attributes displayed in the label, select Edit to open **Text Content Editor** dialog box.

> ***Note:*** The process of displaying a user-defined attribute on selected Civil points in a drawing is described in the article **Creating user-defined attributes for Civil points**. It is also possible to use properties from another object (Surface, Horizontal Alignment and Vertical Alignment).

Text Height

Sets the height of data/text components presented in the label.

Rotation angle

Sets the rotation angle of data/text components presented in the label. The default rotation angle is 0.

Attachment

Sets the position of data/text components relative to the point. Attachment can be set to Top left, Top middle, Top right, Middle left, Middle center, Middle right, Bottom left, Bottom center or Bottom right.

> ***Note:*** Settings Middle left means that the text is positioned so that the symbol is middle-left of the text.

X offset

Specifies the offset of the data/text component from the symbol in the X direction.

Y offset

Specifies the offset of the data/text component from the symbol in the Y direction.

Color

Sets the color of data/text component.

Maximum width

Defines the maximum width of data/text component. The maximum width defines the width of border (if gap is set to 0) and can be seen when using background mask.

Border

Visibility

Toggles the border visibility of the label.

Type

Defines the border shape of the label. The border can be set to Rectangular, Rounded Rectangular or Circular type.

Background mask

Adds a white background.

Gap

Specifies the gap between the text and the border line.

Color

Defines color the border line.

Linetype
Defines the linetype of the border line.

Lineweight
Defines the lineweight of the border line.

Block component properties

Block

Block name
Option to select a block from a list of available blocks in the drawing.

Block height
Sets the height of block component in the drawing.

Rotation
Sets the rotation angle of the block component presented in the label. The default rotation angle is 0.

Attachment
Sets the attachment point of the block component in the label. Attachment can be set to Top left, Top middle, Top right, Middle left, Middle center, Middle right, Bottom left, Bottom center, Bottom right or Insertion Point.

Note: Settings Middle left means that the text is positioned so that the symbol is middle-left of the text.

X offset
Specifies the offset of the block component from the insertion point in the X direction.

Y offset
Specifies the offset of the block component from the insertion point in the Y direction.

Color
Defines color the block component.

Linetype
Defines the linetype of the block.

Lineweight
Defines the lineweight of the block.

Line component properties

Line

Use end point anchor
Toggles the connection between the line and the anchor point.

End point anchor component
If **Use end point anchor** is checked, it is possible to select End point anchor component.

The anchor component is the item that the line is positioned relative to. End point anchor component can be set to <Feature> or Data. When the end point anchor component is set to <Feature>, the line is attached to the label component's attachment point. When it is set to Data, it can be one of several locations on the label component (defined in End point anchor point).

End point anchor point
End point anchor point can be defined only when End point anchor component is set to Data. End point anchor point can be set to Top left, Top center, Top right, Middle left, Middle center, Middle right, Bottom left, Bottom center or Bottom right.

Length type
The Line length type can set to either fixed length (defined in Fixed length) or a percentage (defined in Percentage length).

Fixed length
When the Length type is set to Fixed length, the total line length can be defined.

Percentage length
When the Length type is set to Percentage length, the total line length can be defined.

Rotation angle
When Use end point anchor is unchecked, the rotation angle of the line component can be set. The default rotation angle is 0.

Start point X offset
Specifies the offset of line's start point in the X direction.

Start point Y offset
Specifies the offset of line's start point in the Y direction.

End point X offset
When Use end point anchor is checked, the line's end point offset can be defined. End point X offset specifies the offset of line's end point in the X direction.

End point Y offset
When Use end point anchor is checked, the line's end point offset can be defined. End point Y offset specifies the offset of line's end point in the Y direction.

Color
Defines color the line component.

Linetype
Defines the linetype of the line.

Lineweight
Defines the lineweight of the line.

Arrow component properties

Arrow

Arrow head style
Specifies the arrow head style from the list of options.

Arrow head size
Defines arrow head size.

Rotation
Sets the rotation angle of the arrow component presented in the label. The default rotation angle is 0.

Fixed length
Toggles the option to set fixed length of the arrow.

Length
Defines the length of the arrow.

X offset
Specifies the offset of the midpoint of the arrow from the symbol in the x direction.

Y offset
Specifies the offset of the arrow component from the symbol in the Y direction.

Color
Defines color the arrow component.

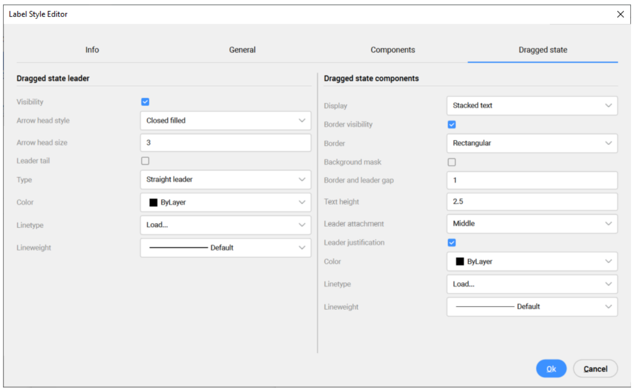
Linetype
Defines the linetype of the arrow.

Lineweight
Defines the lineweight of the arrow.

Edit
Opens the **Text Content Editor** dialog box.

34.12.1.4 Dragged state tab

Defines leader style and components style in dragged state.



Dragged state leader
Displays leader style properties in dragged state which can be edited.

Visibility
Toggles the leader visibility in the drawing.

Arrow head style
Defines arrow head style by selecting it from the drop-down list.

Arrow head size
Defines arrow head size.

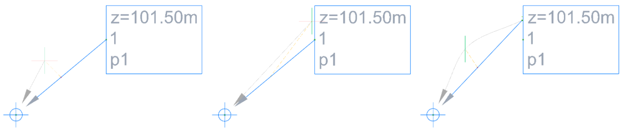
Leader tail
Toggles the leader tail in the drawing.

Note: When the leader tail is turned on, its length is equal to the arrow head size.

Type

Sets the leader type as straight line or spline. To edit spline curve, drag the “+” sign from the middle of the leader to a desired position.

Note: Selecting and dragging the '**+**' sections of the leader will create curves, while selecting '**-**' will remove sections of the spline curve.



Color

Specifies the color of leader in dragged state.

Linetype

Defines linetype of leader in dragged state.

Lineweight

Defines lineweight of leader in dragged state.

Dragged state Components

Displays components properties style in dragged state which can be edited.

Display

Displays components in dragged state as:

- **Stacked text**: means that their style will be defined under the **Dragged State** tab.
- **As composed**: means that the style is the same as it is defined under the **Components** tab, in the **Text Content Editor** dialog box.

Note: If you want the label style to be different in dragged state, select Display option Stacked text and define the rest of its properties.

Border visibility

Toggles the border visibility in dragged state.

Border

Defines border shape in dragged state as: rectangular, rounded rectangular or circular.

Background mask

Toggles the background mask of components in dragged state.

Border and leader gap

Defines border and leader gap in dragged state.

Text height

Defines text height in dragged state.

Leader attachment

Specifies the position of the leader attachment point, picked from the drop-down list.

Leader justification

Toggles leader justification. Text components change its justification with the movement of the leader.

Color
Specifies the color of components in dragged state.

Linetype
Defines linetype of components border in dragged state.

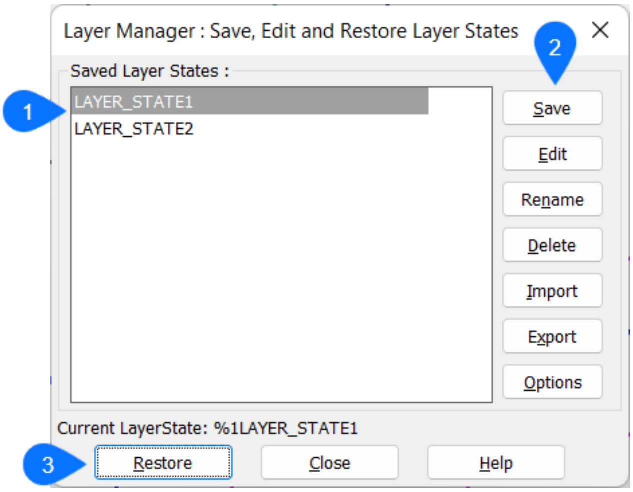
Lineweight
Defines lineweight of components border in dragged state.

Note: All the changes are dynamically displayed in the drawing.

34.12.2 Layer Manager: Save, Edit and Restore Layer States dialog box

Opens via: LMAN command (Express Tools)

The **Layer Manager: Save, Edit and Restore Layer States** dialog box allows you to save, edit and restore layer states.



1 Saved Layer States

2 Layer States options

3 Restore

34.12.2.1 Saved Layer States

Displays a list of saved layer states.

34.12.2.2 Layer States options

Save
Opens the **New LayerState Name** dialog box, which allows you to save the current layer status.

Edit
Allows you to edit the layers states by opening the **Drawing explorer** dialog box. The **New LayerState Name** dialog box opens, which allows you to save the changes.

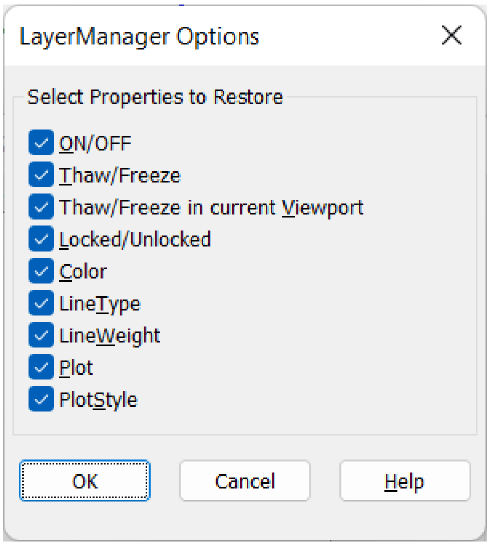
Rename
Opens the **Rename LayerState** dialog box, which allows you to rename a saved layer state.

Delete
Deletes the selected layer state. The **Warning** dialog box opens to confirm the action.

Import
Opens the **Import file name** dialog box, which allows you to choose a .lay file to open.

Export
Opens the **Export file name** dialog box, which allows you to export the selected layer state to a .lay file format.

Options
Opens the **LayerManager Options** dialog box, which allows you to set the options to be used for the **Restore** operation.



Note: All options are On by default.

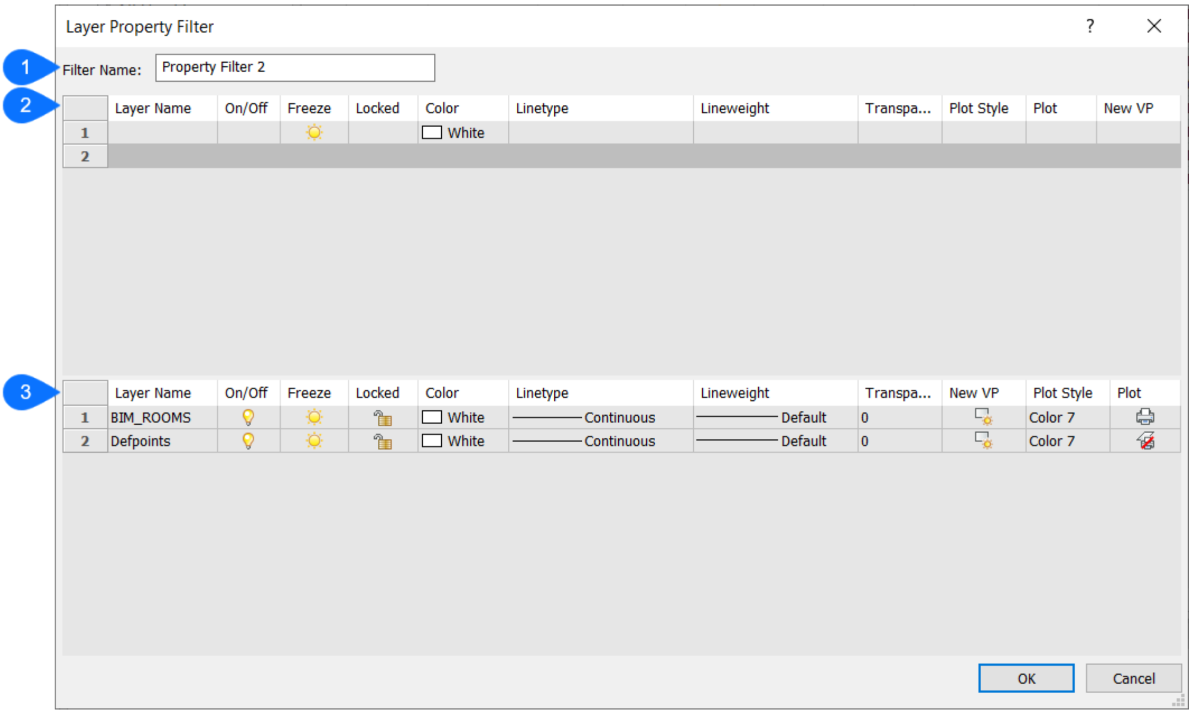
34.12.2.3 Restore

Restores the layer settings of the selected layer state.

34.12.3 Layer Property Filter dialog box

Opens via: **Drawing Explorer** dialog box (EXPLAYERS command)

The **Layer Property Filter** dialog box allows you to create a new layer filter definition, based on the properties of the layer.



1 Filter Name

2 Filter Properties

3 Filtered Layers

34.12.3.1 Filter Name

Sets the name of the filter.

34.12.3.2 Filter Properties

Sets the properties to filter the layers. Layers that fulfill all indicated properties of the filter will be added to the layer filter.

Layer Name

Filters the layers based on their name. You can use wild cards to specify more than one layer, for example Doors-* for Doors-Exterior and Doors-Interior.

- # - matches any digit. For example, ## matches 12 and 34 but not 5 or 123 (ASCII code: 35).
- @ - matches any alphabetic character, so not digits or symbols. For example, @@@@@ matches all 5-letter layer names. (ASCII code: 64).
- . - matches non-alphanumeric characters, so symbols but not letters or numbers. For example, .* matches all layer names starting with a non-alphanumeric character such as _SITEPLAN and +1_LEVEL (ASCII code: 46).
- * - matches any string, including letters, numbers, and characters. For example, A* matches all layer names starting with the letter A (ASCII code: 42).
- ? - matches any single character. For example, ?AB* matches L**AB**ORATORY and C**AB**ARET but not ST**AB**LE, because STABLE begins with two letters and ? specifies a single letter (ASCII code: 63).

- ~ - negative match, in which any string but the pattern is returned. For example, ~*A* matches all layer names not containing the letter A (ASCII code: 126).
- [ ] - Matches any one of the characters enclosed inside the square brackets. For example, LA[YB] matches LA**Y** and LA**B**(ASCII code: 91 93).
- [~] - Specifies any one of the characters not found in the square brackets. For example, LA[~YB] matches LA**D** but not LA**Y** or LA**B** (ASCII code: 91 126 93).
- [-] - Specifies a range for a single character in the layer name. For example, LAY-[A-E] matches any combination of LAY- and A through E, such as LAY-**A** ... LAY-**E** but not LAY-F (ASCII code: 91 45 93).
- ' - Matches the next string exactly to find a layer name that contains a wild-card character when it is prefixed by the reverse quote character. Use ` to search for layer names that include characters like ? and ~. For example, `~LAY* matches ~LAY-1,~LAYER-1, and ~LAY-B (ASCII code: 96).

Note: ASCII code: if a character is not available on the keyboard, press and hold the Alt key and then type the ASCII code on the numeric keypad.

Properties

Specifies which layers to include according to their properties, such as freeze and color.

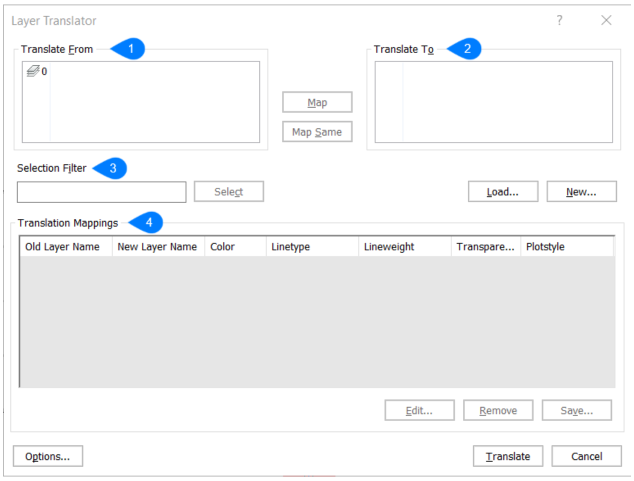
34.12.3.3 Filtered Layers

Displays the layers that fulfill the requirements of the filter.

34.12.4 Layer Translator dialog box

Opens via: LAYTRANS command

The **Layer Translator** dialog box allows you to apply layer names and properties from another drawing to the current drawing.



1 Translate from

2 Translate to

3 Selection filter

4 Translation mappings

34.12.4.1 Translate from

Lists all the layer names in the current drawing.

Note: You can specify the layers to which you want them to map by loading layer information from an existing DWG, DWS, or DWT file.

Note: New layers can be created by simply entering the layer name and properties to which you want to map existing layer.

34.12.4.2 Translate to

Specifies which layer you want the chosen layer(s) to map to.

Note: Select one or more layers from the current drawing, the list on the left, to map to a layer from the list on the right. If you choose Map Same any layer names in the current drawing with corresponding names in the list on the right will inherit the properties from the list on the right.

34.12.4.3 Translation mappings

Displays an overview of which layer and its properties are translated to another layer.

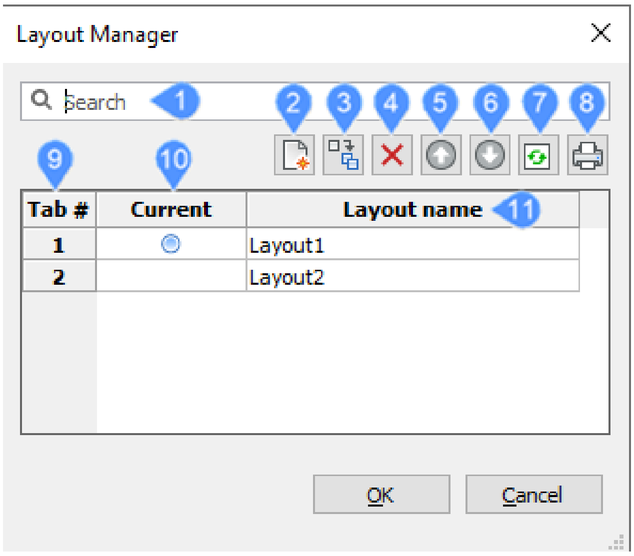
34.12.4.4 Options

Displays the **Options** dialog box, which offers additional controls for layer mapping.

34.12.5 Layout Manager dialog box

Opens via: LAYOUTMANAGER command

The **Layout Manager** dialog box allows you to create, copy and delete layouts.



1 Search

2 Add new layout

3 Copy selected layout

4 Delete

5 Move up

6 Move down

7 Clear selection

8 Publish

9 Tab

10 Current

11 Layout name

34.12.5.1 Search

Searches for a layout name and displays only the searched layout. This is useful when a drawing contains many layouts.

34.12.5.2 Add new layout

Creates a new layout.

34.12.5.3 Copy selected layout

Makes a copy of the selected layouts and adds them to the end of the list.

34.12.5.4 Delete

Deletes the selected layouts.

34.12.5.5 Move up

Moves the selected layouts up the list.

34.12.5.6 Move down

Moves the selected layouts down the list.

34.12.5.7 Clear selection

Removes highlighting from layout names, thereby unselecting them.

34.12.5.8 Publish

Opens the **Publish** dialog box.

34.12.5.9 Tab

Displays the number of the layout.

34.12.5.1 Current
0

Click in the **Current** column to make the layout current.

34.12.5.1 Layout name

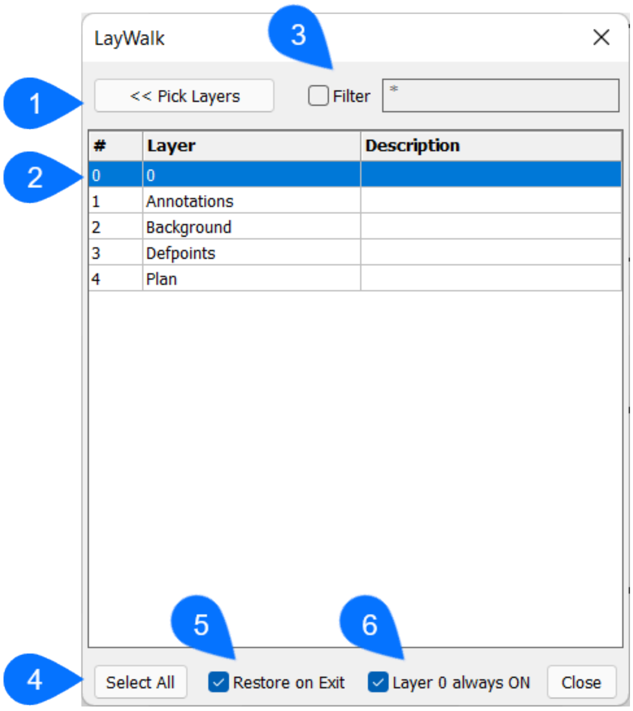
1

Displays the name of the layout.

34.12.6 LayWalk dialog box

Opens via: LAYWALK command (Express Tools)

The **LayWalk** dialog box allows you to select the layers for which entities are displayed in the drawing.



1 Pick Layers

2 Layer list

3 Filter

4 Select All

5 Restore on Exit

6 Layer 0 always ON

34.12.6.1 Pick Layers

Temporarily dismisses the **LayWalk** dialog box and allows you to select entities in the drawing, along with their layers.

34.12.6.2 Layer list

Displays a list of available layers. Select the layers for which you want to display the entities.

Note: The unselected layers are frozen.

34.12.6.3 Filter

Turns On and Off an active filter.

- Tick the checkbox to display only the layers that match the filter.
- Untick the checkbox to see the full list of layers.

34.12.6.4 Select All

Click this button to select and display all layers.

34.12.6.5 Restore on Exit

If the checkbox is ticked, it restores all layers to their previous state when closing the dialog box.

If the checkbox is unticked, the changes you made are saved.

34.12.6.6 Layer 0 always ON

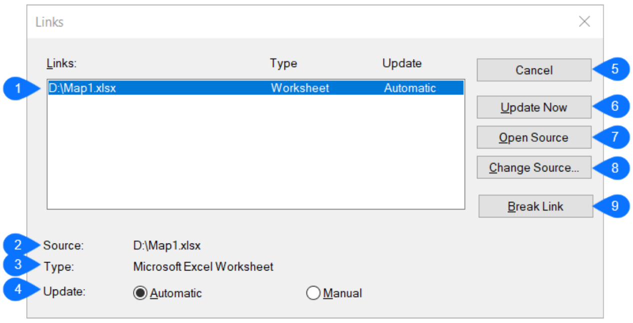
If the checkbox is ticked, the layer 0 is always displayed.

If the checkbox is unticked, only the selected layers are displayed.

34.12.7 Links dialog box

Opens via: OLELINKS command

The **Links** dialog box allows you to edit the links of OLE entities that were embedded or linked to drawings.



1 Links

2 Source

3 Type

4 Update

5 Cancel

6 Update Now

7 Open Source

8 Change Source

9 Break Link

34.12.7.1 Links

Provides a list of all links loaded in the file.

34.12.7.2 Source

Reports the path to the source file name.

34.12.7.3 Type

Reports the file type.

34.12.7.4 Update

Determines how the OLE entity in the drawing is updated when the source changes.

34.12.7.5 Cancel

Cancels the linking operation.

34.12.7.6 Update Now

Updates linked OLE entities.

34.12.7.7 Open Source

Opens the source file in the source application. This allows you to edit the source file.

34.12.7.8 Change Source

Changes the source for the linked OLE entity.

34.12.7.9 Break Link

Breaks the link with the source file.

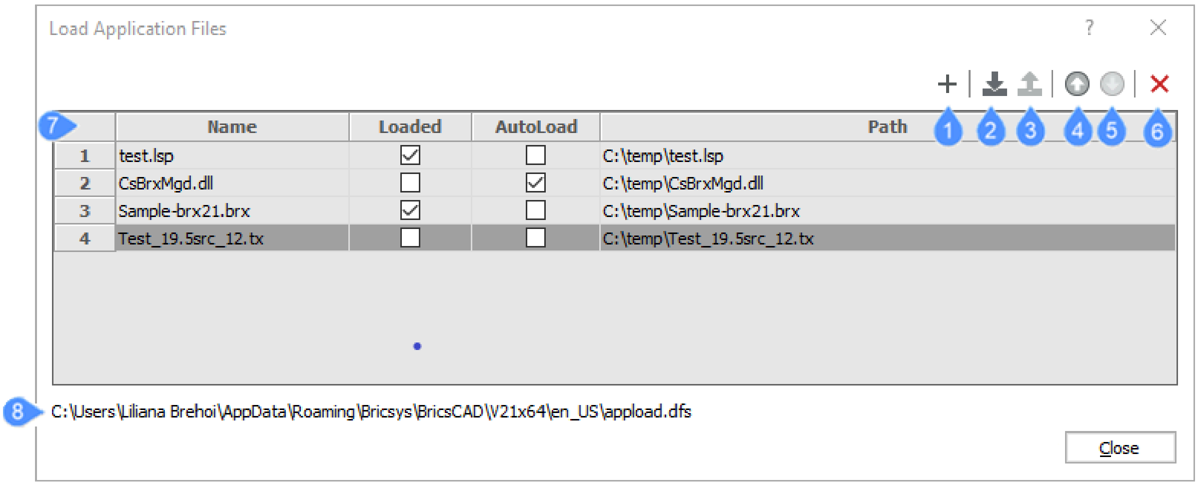
Note: When the link is broken, the OLE entity in the drawing no longer changes when the source file changes.

34.12.8 Load Application Files dialog box

Opens via: APPLOAD command

The **Load Application Files** dialog box allows you to load and unload the following applications type to run inside BricsCAD:

- LSP - LISP applications from BricsCAD and other CAD programs.
- DES - DES LISP applications.
- BRX - BricsCAD runtime extension applications.
- TX - Teigha extension applications, from Open Design Alliance.
- ARX - AutoCAD® runtime extension applications from Autodesk®.
- DBX - DBX applications.
- DRX - Drawing runtime extension applications.
- DVB - Visual Basic applications.
- MNL - LISP expressions for menu files.



1 Add application file

2 Load selected application

3 Unload selected application

4 Move application upward

5 Move application downward

6 Remove selected application

7 Application Files List

8 Application File Path

34.12.8.1 Add application file

Opens the **Select Application Files** dialog box.

34.12.8.2 Load selected application

Loads the selected application(s) into BricsCAD. Only compatible applications with BricsCAD will be loaded.

34.12.8.3 Unload selected application

Unloads the selected application(s) from BricsCAD. The LISP applications have to be removed in order to be unloaded in the current BricsCAD session.

34.12.8.4 Move application upward

Moves the selected application(s) up in the list.

34.12.8.5 Move application downward

Moves the selected application(s) down in the list.

34.12.8.6 Remove selected application

Removes the selected application(s) from the list.

34.12.8.7 Application Files List

Displays a list with application files that are available for load, unload, move up and down or remove.

- **Name**: specifies the name and the extension of the application file from the list.
- **Loaded**: specifies if a file is loaded or unloaded in the current BricsCAD session.
- **Autoload**: specifies if a file will be loaded automatically in the next BricsCAD session.
- **Path**: displays the location folder of the added files.

34.12.8.8 Application File Path

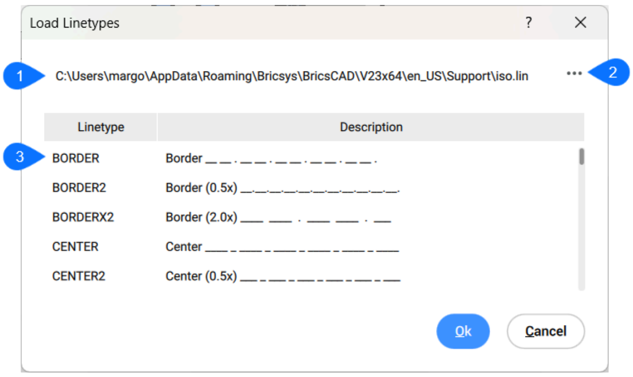
Displays the location folder for the appload.dfs file that contains all applications added, using the **Select Application Files** dialog box. You can open the file using Notepad.

Applications listed in appload.dfs are loaded on start-up.

34.12.9 Load Linetypes dialog box

Opens via: **Drawing Explorer** dialog box (LINETYPE command)

The **Load Linetypes** dialog box allows you to load additional linetype definitions.



1 File's path

2 Browse File

3 List

34.12.9.1 File's path

Displays the path of the selected *.lin (linetype) file.

34.12.9.2 Browse File

Opens the **Choose a Linetype File** dialog box.

34.12.9.3 List

Displays the available linetypes to load. Two characteristics of the linetypes are displayed:

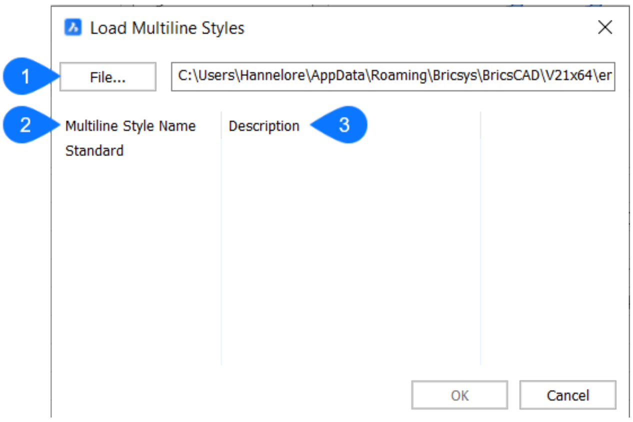
- **Linetype**
  Displays the name of the available linetypes

- **Description**
  Displays the visual representation of the linetype.

34.12.10 Load Multiline Styles dialog box

Opens via: **Drawing Explorer** dialog box (MLSTYLE command)

The **Load Multiline Style** dialog box allows you to load multiline styles from a multiline (*mln) file.



1 File

2 Multiline Style Name

3 Description

34.12.10. File 1

Displays the **Choose a Multiline Style File** dialog box where you can select a multiline style file to open.

34.12.10. Multiline Style Name 2

Displays the names of the available multiline style files in the drawing. You can select a multiline style file from this list.

34.12.10. Description 3

Displays a description of the multiline style files, if available.

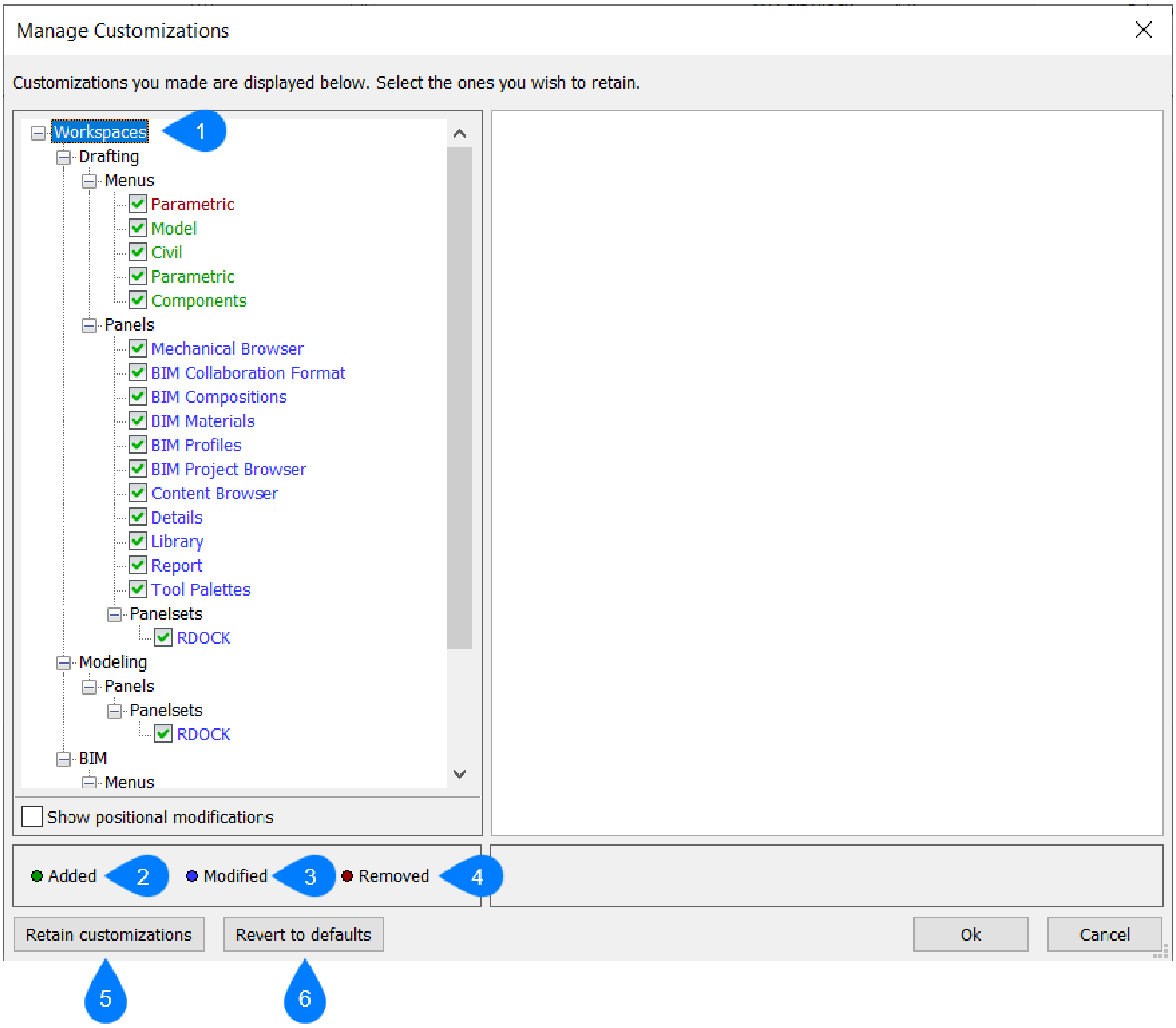
34.13 M

34.13.1 Manage Customizations dialog box

Opens via: **Customize** dialog box

The **Manage Customizations** dialog box provides a list of all customizations for confirming and reversing changes made to the user interface.

To open the dialog box, first run the CUSTOMIZE command. This opens the **Customize** dialog box. Click **Manage your customizations...** to open the **Manage Customizations** dialog box.



1 Customizations

2 Added

3 Modified

4 Removed

5 Retain customizations

6 Revert to defaults

34.13.1.1 Customizations

Lists all added, modified or removed customizations.

Note: You can check a customization to retain the modification in the CUI.

34.13.1.2 Added

Items you have created display in green.

34.13.1.3 Modified

Items you have modified display in blue.

34.13.1.4 Removed

Items you have removed display in red.

34.13.1.5 Retain customizations

Keeps your local default CUI as it is.

34.13.1.6 Revert to defaults

Reverts your local CUI to the corresponding default one installed by BricsCAD, for example:

C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support\en_US\classic-ribbon.cui

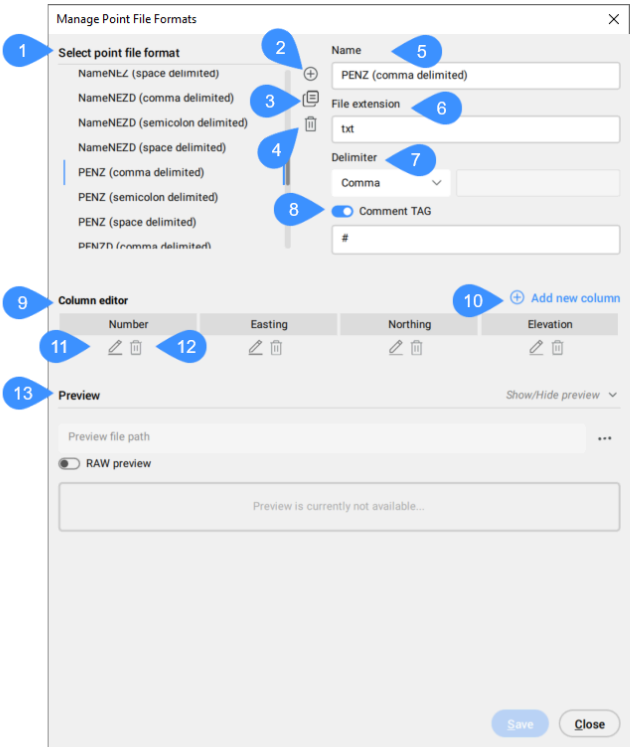
Note: BricsCAD provides three predefined default CUI files: classic-ribbon.cui, modern.cui, and classic-toolbars.cui.

34.13.2 Manage Point File Formats dialog box

Opens via:

- **Export Civil Points** dialog box
- **Import Points From Point File** dialog box

The **Manage Point File Formats** dialog box allows you to add a new point file format or to edit an existing one.



1 Select point file format

2 Add point file format

3 Duplicate point file format

4 Remove a point file format

5 Name

6 File extension

7 Delimiter

8 Comment Tag

9 Column editor

10 Add new column

11 Edit column

12 Remove column

13 Preview file path

14 Preview

34.13.2.1 Select point file format

Selects a point file format from the list.

34.13.2.2 Add point file format

Defines a new point file format by specifying its name, file extension, column delimiter, comment tag, attributes and arrangement of attributes by columns (which columns contain which point attribute in the point file).

34.13.2.3 Duplicate point file format

Creates a copy of the selected point file format.

34.13.2.4 Remove a point file format

Removes a point file format from the list.

34.13.2.5 Name

Enables to specify the point file format name. Click the field to edit it.

Note: A warning message appears when the name of the new point file format is empty or already exists.

34.13.2.6 File extension

Enables to specify the file extension. Click the field to edit it.

34.13.2.7 Delimiter

Enables to specify the column delimiter for the point file from the drop-down list.

34.13.2.8 Comment Tag

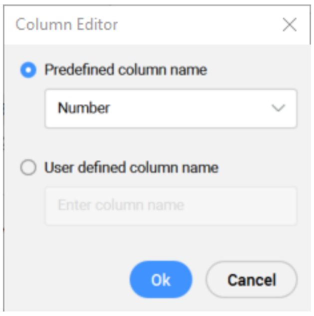
Enables or disables the comment tag. Set a comment tag in the box below.

34.13.2.9 Column editor

Shows the arrangement of point attributes into columns as written in the points file. The names, order and contents of the columns are defined by the user to match the input point files. Some of the most common point file formats are already pre-defined.

34.13.2.1 Add new column 0

Opens the **Column Editor** dialog box to add a new column.



Predefined column name

Selects a column name from the predefined drop-down list. Select the <skip> option, to ignore a specific column in the point file.

User defined column name

Allows you to specify user-defined column name.

34.13.2.1 Edit column

1

Opens the **Column Editor** dialog box to edit a column name.

34.13.2.1 Remove column

2

Removes the column from the file format.

34.13.2.1 Preview file path

3

Allows to select a point file for preview.

34.13.2.1 Preview

4

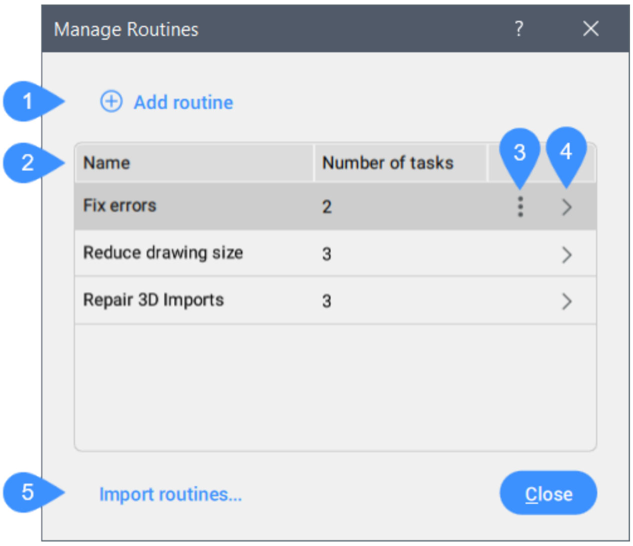
Enables preview of selected Point File based on a specified parameters in a dialog.

Note: Click the **Save** button to save the changes.

34.13.3 Manage Routines dialog box

Opens via: **Drawing Health** Command Context panel

The **Manage Routines** dialog box allows you to manage the routines.



1 Add routine
2 Routines table
3 Drop-down menu
4 Edit routine
5 Import routines...

34.13.3.1 Add routine

Adds a new empty routine under a default name (New Routine_1) that you can rename.

34.13.3.2 Routines table

Lists all saved routines and the number of tasks for each.

Right-click the table header to display a context menu that allows you to enable or disable columns, show all columns, or restore column positions.

34.13.3.3 Drop-down menu

Press the three dots button (⋮) to open a drop-down menu.

Edit
Opens the **Edit Routine** dialog box to edit the routine. See the related article **Edit Routine dialog box**.

Rename
Renames the routine.

Duplicate
Creates a duplicate of the routine.

Export
Opens the **Export Routine** dialog box to export the routine to a RTN file.

Delete
Removes the routine from the list.

34.13.3.4 Edit routine

Opens the **Edit Routine** dialog box to edit the routine. See the related article **Edit Routine dialog box**.

34.13.3.5 Import routines...

Opens the **Open** dialog box to import a RTN file.

Note: You can duplicate, delete, export, and import multiple routines at once.

34.13.4 Manage Routines (BMASSEMBLYINSPECT) dialog box

Opens via: **Assembly Inspect** Command Context panel

The **Manage Routines** dialog box allows you to manage the routines for BMASSEMBLYINSPECT commands.

1 Add routine

2 Routine list

3 Drop-down menu

4 Edit routine

5 Import routines...

34.13.4.1 Add routine

Opens the **Routine editor** dialog box to configure a new routine.

For the BMASSEMBLYINSPECT command, the default routine name is Inspection_Routine.

All fastener settings and each of the assembly criteria are stored in a routine JSON file that can be imported or exported, allowing routine files to be shared between different users.

34.13.4.2 Routine list

Lists all saved routines and the criteria defined for each.

Right-click to display a context menu that allows you to enable or disable columns, show all columns, or restore column positions.

34.13.4.3 Drop-down menu

Press the three dots button (⋮) to open a drop-down menu.

Edit
Opens the **Routine editor** dialog box to edit the routine.

Duplicate
Creates a duplicate of the routine.

Export
Opens the **Export Routines** dialog box to export the routine to a JSON file.

Delete
Removes the routine from the list.

34.13.4.4 Edit routine

Opens the **Routine editor** dialog box to edit the routine.

34.13.4.5 Import routines...

Opens the **Import Routine** dialog box to import a JSON file.

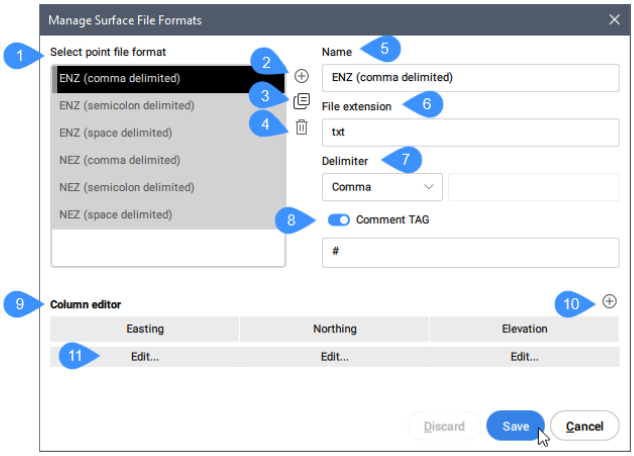
Note: For the BMASSEMBLYINSPECT command, a default master routine file (Inspection_Routine_Default.json) is included with the BricsCAD installer in the Support folder.

Note: You can duplicate, delete, export, and import multiple routines at once.

34.13.5 Manage Surface File Formats dialog box

Opens via: **Create Surface From Point File** dialog box

The **Manage Surface File Formats** dialog box allows you to add a new point file format or to edit an existing one.



1 Select point file format

2 Add point file format

3 Duplicate point file format

4 Remove a point file format

5 Name

6 File extension

7 Delimiter

8 Comment Tag

9 Column editor

10 Add column

11 Edit

34.13.5.1 Select point file format

Selects a point file format from the list. Some predefined point file formats are available. Their names indicate the coordinates (columns) order and the delimiter. For example, ENZ (comma delimited) is a point file format with Easting, Northing and Elevation coordinates with a comma delimiter.

34.13.5.2 Add point file format

Defines a new point file format by specifying comment tag, specifying point attributes by columns (which columns contain which point attribute in the point file).

Note: The newly created point file format will be preselected in the **Create Surface from Point file** dialog box.

34.13.5.3 Duplicate point file format

Creates a new point file format with the same settings as the selected one.

34.13.5.4 Remove a point file format

Removes a point file format from the list.

34.13.5.5 Name

Displays the point file format name. Click the field to edit it.

34.13.5.6 File extension

Displays the file extension. Click the field to edit it.

34.13.5.7 Delimiter

Sets a character for the column delimiter for the point file from the drop-down list.

34.13.5.8 Comment Tag

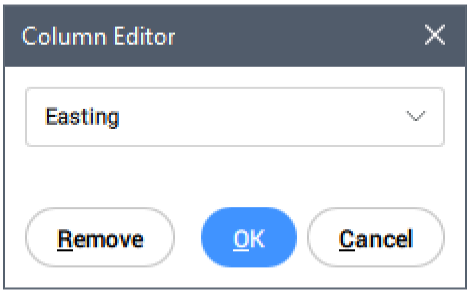
Enables or disables the comment tag.

34.13.5.9 Column editor

Displays the columns. You can change the columns order by moving it to the new location.

34.13.5.1 Add column
0

Opens **Column Editor** dialog box to add or to remove a column.



Selects a column name from the drop-down list.
You can add a <skip> column or set an existing column as <skip>, in order to create compatibility between two point format files.

34.13.5.1 Edit

1

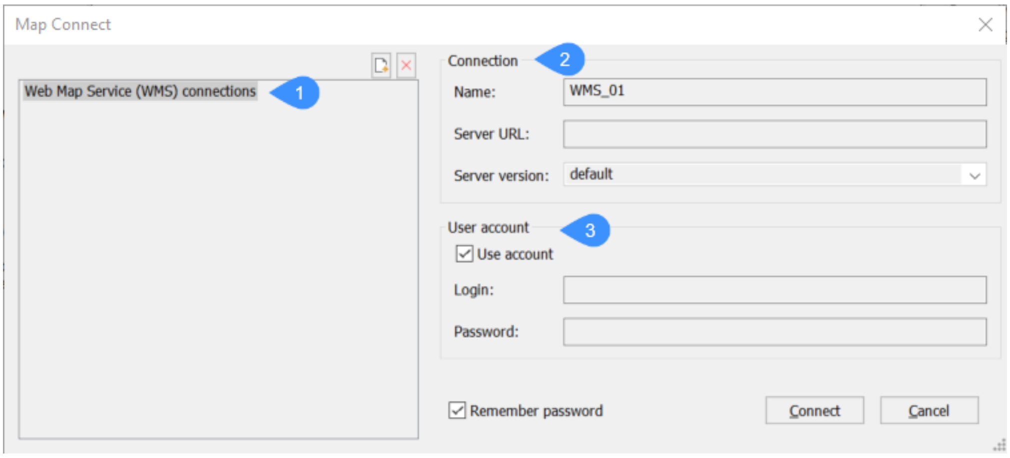
Opens the **Column Editor** dialog box to edit a column name.

34.13.6 Map Connect dialog box

Opens via: MAPCONNECT command

The **Map Connect** dialog box allows you to set up a connection with a Web Map Service.

Note: Before adding map data to a drawing, you must run the GEOGRAPHICLOCATION command to define the geographic location of the drawing.



1 WMS connections

2 Connection

3 User account

34.13.6.1 WMS connections

Displays a list of the different possible Web Map Service connections.

34.13.6.2 Connection

Allows you to create a Web Map Service (WMS) connection.

Name
Specifies the name for the new connection.

Server URL
Specifies the URL of the server you want to connect to.

Server version
Specifies the version of the server or else select one from the drop-down list.

34.13.6.3 User account

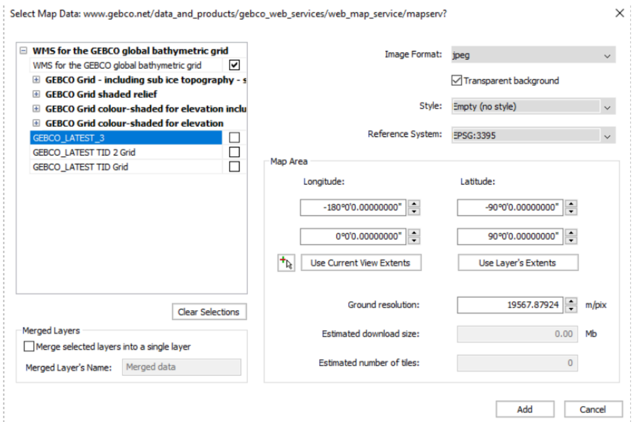
Allows you to log in to your account.

Use account
Toggles whether the login or password is used when making the connection.

Login
Specifies the login information.

Password
Specifies the password of your account.

Remember passwords
Check this box to keep being logged in. When is set to Off the program does not remember your passwords for greater security.
After connecting to a Web Map Server, the following dialog box is displayed:



Do the following:

1 Select one or more data layers.

2 Specify the desired ground resolution and the preferred maximum dimensions of a tile.

3 An estimate of the download size displays in the Estimated download size field.

4 Press the **Add** button.

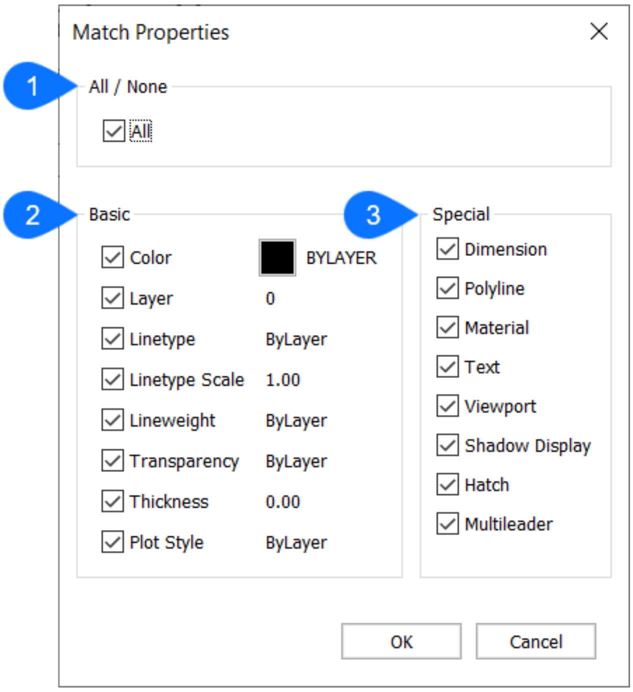
The program stores downloaded map tiles as regular image files in the folder in which the drawing is saved.
The definition of the supported Coordinate Reference Systems (CRS) is stored in the new geodatabase.xml file located in the folder pointed to by the ROAMABLEROOTPREFIX variable, such as *C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support*

The geodatabase.xml files lists cities and their WGS84 geo coordinates used by the **Choose Location** function of the **Geographic Location** dialog box.

34.13.7 Match Properties dialog box

Opens via: MATCHPROP command

The **Match Properties** dialog box allows you to select which properties you want to copy from one entity to one or more other entities.



1 All/None

2 Basic

3 Special

34.13.7.1 All/None

When checked on, all properties are automatically selected. When checked off, all entities are deselected.

34.13.7.2 Basic

Selects the basic entity properties to copy. On the right side, the entity properties of the source entity are displayed.

Color
When checked on, the color of the source entity will be copied to the selected entities. The color of the source entity is displayed on the right.

Layer
When checked on, the layer of the source entity will be copied to the selected entities. The layer of the source entity is displayed on the right.

Linetype
When checked on, the linetype of the source entity will be copied to the selected entities. The linetype of the source entity is displayed on the right.

Linetype Scale
When checked on, the linetype scale of the source entity will be copied to the selected entities. The line-type scale of the source entity is displayed on the right.

Lineweight
When checked on, the lineweight of the source entity will be copied to the selected entities. The lineweight of the source entity is displayed on the right.

Transparency
When checked on, the transparency of the source entity will be copied to the selected entities. The trans-parency of the source entity is displayed on the right.

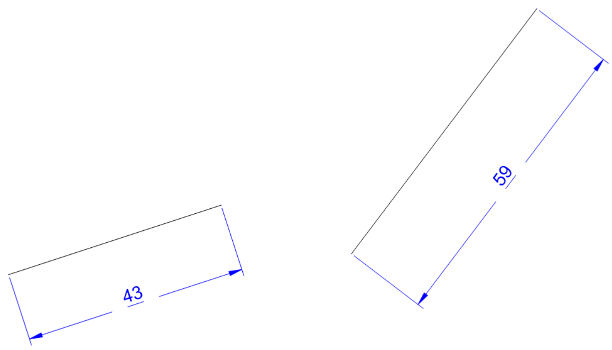
Thickness
When checked on, the thickness of the source entity will be copied to the selected entities. The thickness of the source entity is displayed on the right.

Plot Style
When checked on, the plot style of the source entity will be copied to the selected entities. The plot style of the source entity is displayed on the right.

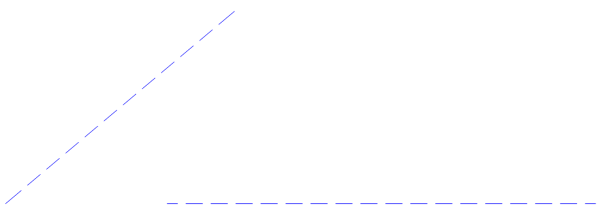
34.13.7.3 Special

Selects other less basic properties to copy.

Dimension
When checked on, the dimension style and annotative properties of dimensions, leaders and tolerances is copied.

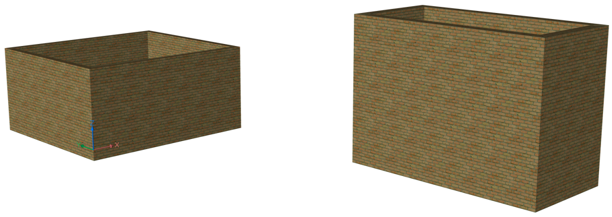


Polyline
When checked on, the width and linetype generation settings are copied. Other settings are ignored, such as taper and spline.



Material

When checked on, the material properties are copied. Adds or removes materials, depending on whether the source entities has a material definition or not.



Text

When checked on, text style and annotative property of text and mtext is copied. Other properties, like the color, are not copied.

Viewport

When checked on, the viewport-specific properties are copied, such as on or off, snap and grid status, and scale factor. Other properties, such as clipping or frozen layer states, are not copied.

Shadow Display

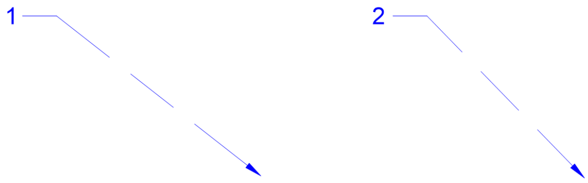
When checked on, the shadows of the source entity are copied.

Hatch

When checked on, all hatch properties and annotative scaling are copied.

Multileader

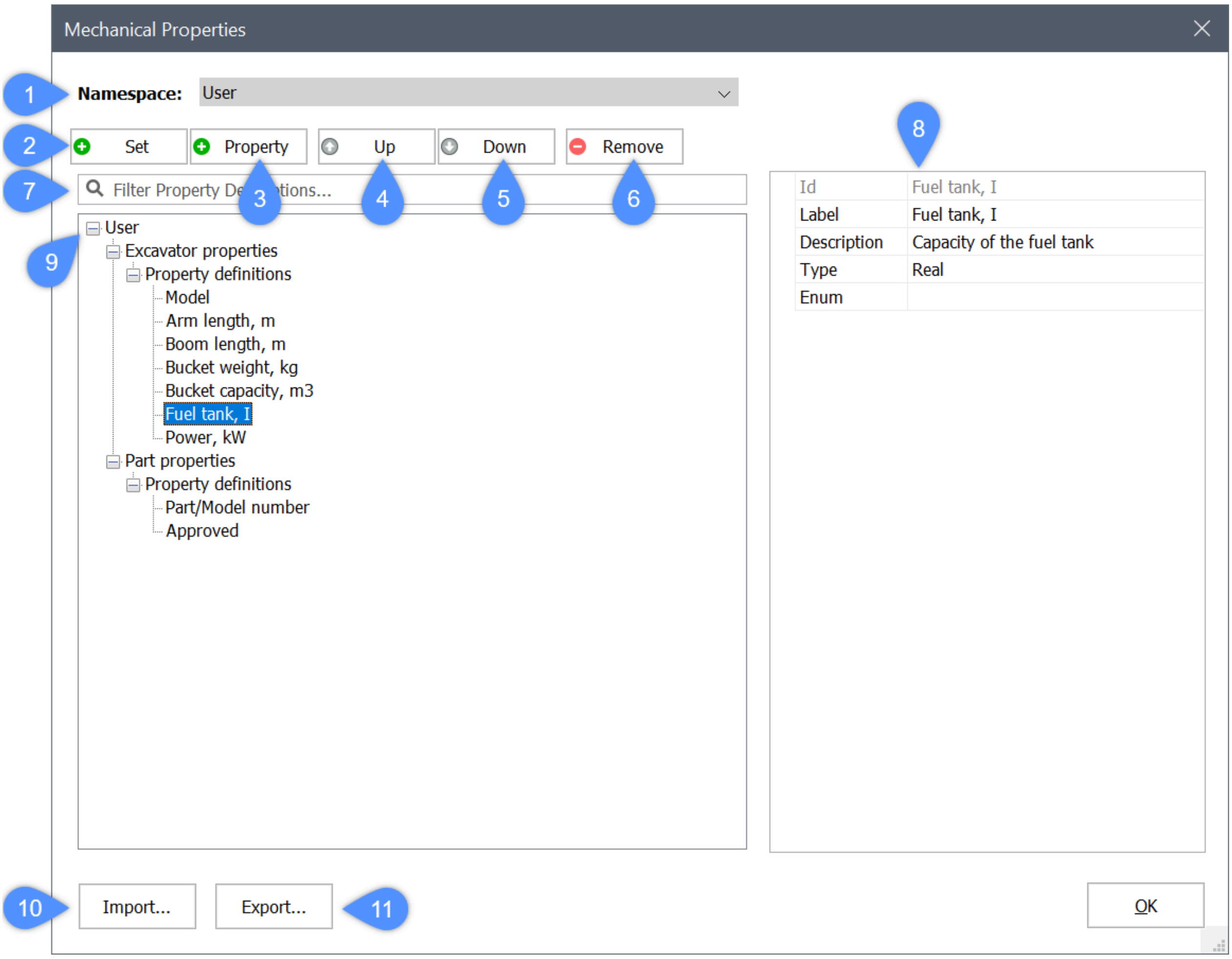
When checked on, the multileader style and annotative scaling is copied.



34.13.8 Mechanical Properties dialog box

Opens via: BMPROPERTIES command

The **Mechanical Properties** dialog box allows you to create, edit and delete property definitions and to organize properties in property sets.



1 Namespace

2 Add Set

3 Add Property

4 Move up button

5 Move down button

6 Remove

7 Filter

8 Properties

9 Properties Tree

10 Import

11 Export

34.13.8.1 Namespace

For mechanical component, currently one namespace is available:

User
Allows you to create user defined properties.

34.13.8.2 Add Set

Allows you to add a property set in the current namespace.

34.13.8.3 Add Property

Allows you to add a property to the currently selected property set.

34.13.8.4 Remove

Deletes the selected item.

34.13.8.5 Move up button

Moves up the selected property or value.

34.13.8.6 Move down button

Moves down the selected property or value.

34.13.8.7 Filter

Displays only the properties of which the **Id** and **Label** contain the search string typed. The properties tree is filtered as you type.

34.13.8.8 Properties

Displays the properties of the selected property sets, property definitions and value definitions.

34.13.8.9 Properties Tree

Displays the property sets, property definitions and value definitions.

34.13.8.1 Import 0

Opens the **Select XML file to import** dialog box.

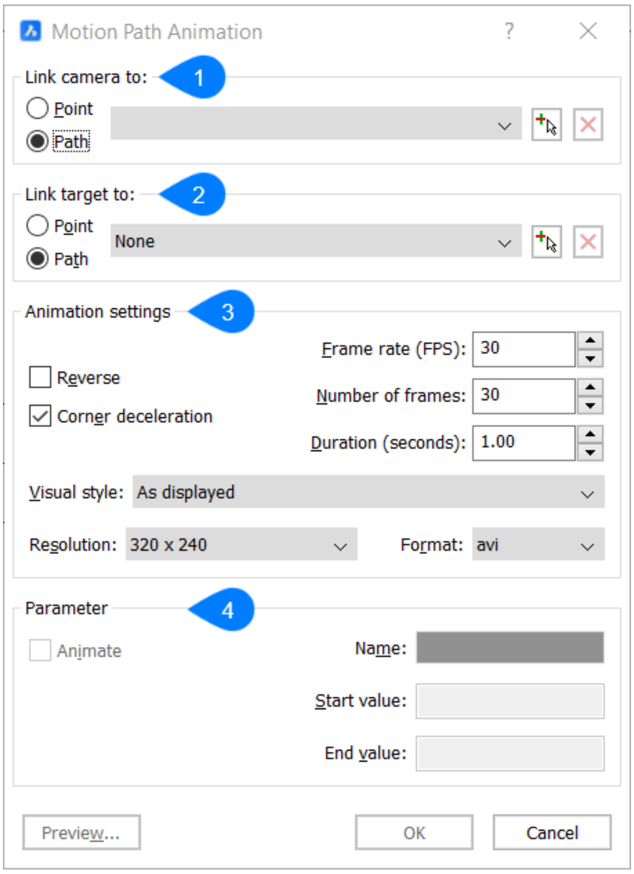
34.13.8.1 Export 1

Opens the **Select location to save xml file** dialog box.

34.13.9 Motion Path Animation dialog box

Opens via: ANIPATH command

The **Motion Path Animation** dialog box allows you to animate a camera moving along a path or panning in a 3D model and saves it to a movie file.



1 Link camera to

2 Link target to

3 Animation settings

4 Parameter

34.13.9.1 Link camera to

Defines the location of the camera by selecting a point or a path in the model space.

Note: After selecting a point or path, the option to name the entity is available.

Point

Links a camera to a static point in the drawing.

Path

Links a camera to a linear entity in the drawing.

34.13.9.2 Link target to

Defines the target point of the camera by selecting a point or path in the model space.

Note: After selecting a point or path, the option to name the entity is available.

Note: If the camera is linked to a point, the target must be linked to a path. If the camera is linked to a path, the target should be linked to either a point or a path.

Point
Links the target to a static point in the drawing.

Path
Link the target to a linear entity in the drawing.

34.13.9.3 Animation settings

Allows to control the output of the movie file.

Frame rate
Defines the number of frames per second. When the frame rate is changed, the number of frames will be adjusted according to the duration.

Number of frames
Specifies the total number of frames in the movie. When the number of frames is changed, the duration will be changed accordingly.

Duration
Specifies the duration of the movie in seconds. When the duration is changed, the number of frames will be changed accordingly.

Reverse
Reverses the sequence of the frames.

Corner deceleration
Slows down the camera at corners in the animation path.

Visual style
Defines the visual style for the movie. Click the button and select a visual style in the list. All visual styles in the current drawing are available.

Resolution
Sets the width x height resolution of the movie. Click the button to select a resolution.

Available resolutions are: 160x120, 320x240, 640x480, 800x600, 1024x768, 1280x1024, 1600x1024, 1920x1080, 1920x1200.

Format
Specifies the file format of the movie. Available formats are: avi, mpg and wmv.

34.13.9.4 Parameter

Allows to animate the change of a parameter.

Animate
Check the box to create realistic movies with moving mechanisms.

Name
Specify the name of the parameter.

Start value
Specifies a start value for the selected parameter.

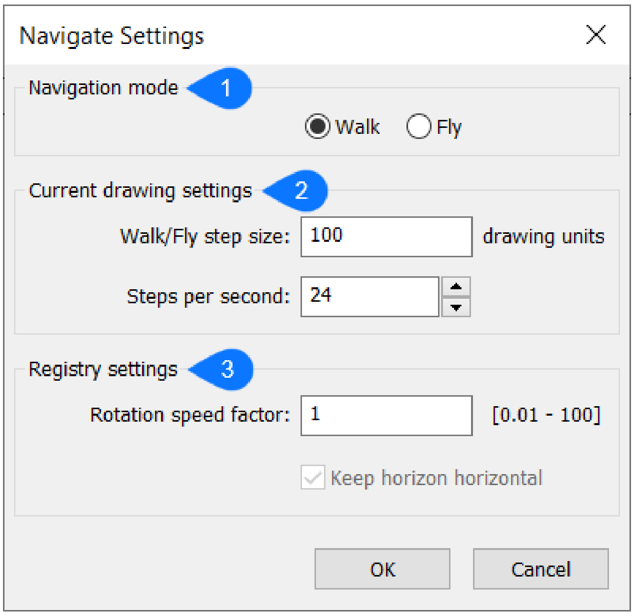
End value
The parameter step is set to the Start-End range divided by the number of frames. On each frame, the parameter value should be incremented with the step.

34.14 N

34.14.1 Navigate Settings dialog box

Opens via: NAVIGATE command

The **Navigate Settings** dialog box allows you to adjust the navigation settings in a drawing.



1 Navigation mode

2 Current drawing settings

3 Registry settings

34.14.1.1 Navigation mode

Toggles between Walk and Fly modes:

- Flight mode: permits movements in all three dimensions.
- Walk mode: restricts movement to the xy-plane.

34.14.1.2 Current drawing settings

Sets the settings in the current drawing, and so will be different in other drawings.

Walk/Fly step size
Specifies the distance the view travels with each press of the key. The distance is measured in drawing units.

Steps per second
Specifies how quickly the view changes when a travel key is held down.

34.14.1.3 Registry settings

Settings which are saved with the registry, they will be the same in other drawings.

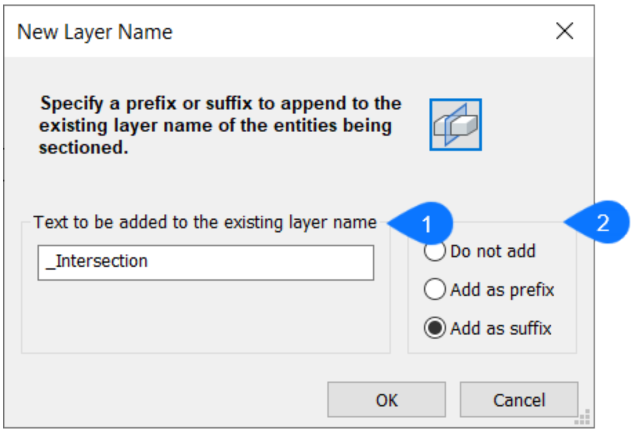
Rotation speed factor
Specifies how quickly the view rotates.

Keep horizon horizontal
When toggled on, the flight level is kept. This option is grayed-out for Walk mode.

34.14.2 New Layer Name dialog box

Opens via: **Drawing explorer** dialog box (SECTIONPLANESETTINGS command)

The **New Layer Name** dialog box allows you to specify the text which is added to the name of the layer when new layers are created as a copy from the existing layer for section planes.



1 Text to be added to the existing layer name

2 Manner to add the text

34.14.2.1 Text to be added to the existing layer name

Specifies the text that will be added to the existing layer name.

34.14.2.2 Manner to add the text

Do not add
No text will be added to name of the layer when a copy is created.

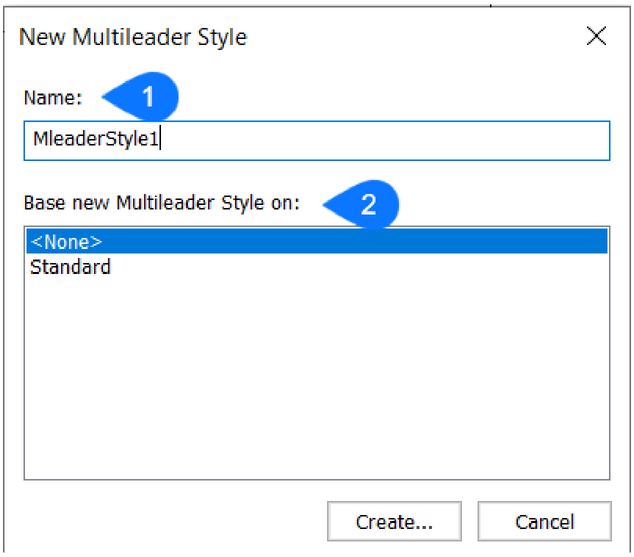
Add as prefix
The specified text will be added as a prefix to the existing name.

Add as suffix
The specified text will be added as a suffix to the existing name.

34.14.3 New Multileader Style dialog box

Opens via: **Drawing explorer** dialog box (MLSTYLE command)

The **New Multileader Style** dialog box allows you to create a new multileader style.



1 Name

2 Base new Multileader Style on

34.14.3.1 Name

Sets the name of the new multileader style.

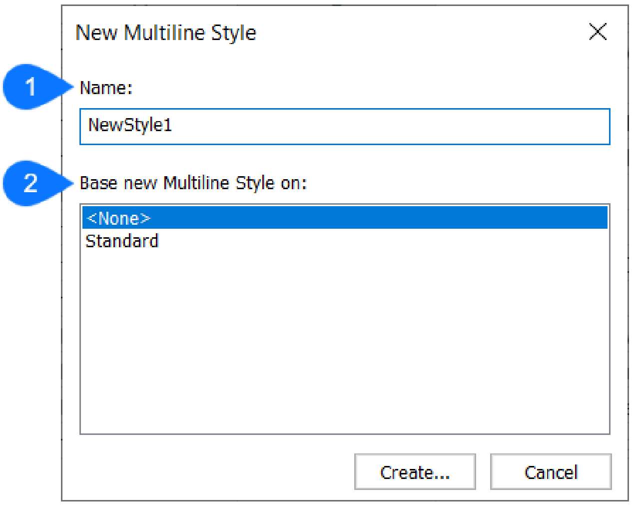
34.14.3.2 Base new Multileader Style on

Selects an existing style to create the new one as a copy of it. To start from scratch, select <None>.

34.14.4 New Multiline Style dialog box

Opens via: **Drawing explorer** dialog box (MLSTYLE command)

The **New Multiline Style** dialog box allows you to create a new multiline style definition.



1 Name

2 Base new Multiline Style on

34.14.4.1 Name

Sets the name of the new multiline style.

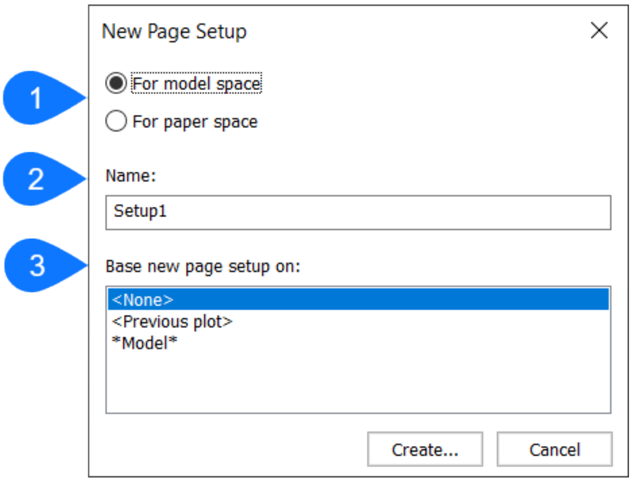
34.14.4.2 Base new Multiline Style on

Selects an existing style to create the new one as a copy of it. To start from scratch, select <None>.

34.14.5 New Page Setup dialog box

Opens via: **Drawing explorer** dialog box (PAGESETUP command)

The **New Page Setup** dialog box allows you to create a new page setup for the paper space or model space.



1 Model space or Paper space

2 Name

3 Base new page setup on

34.14.5.1 Model space or Paper space

Sets the space to create a new page setup to the model space or paper space.

34.14.5.2 Name

Sets the name of the new page setup.

34.14.5.3 Base new page setup on

Selects an existing page setup to create a new one as a copy of the selected one. To start from scratch, select <None>. To use the previous plot, select <Previous plot>.

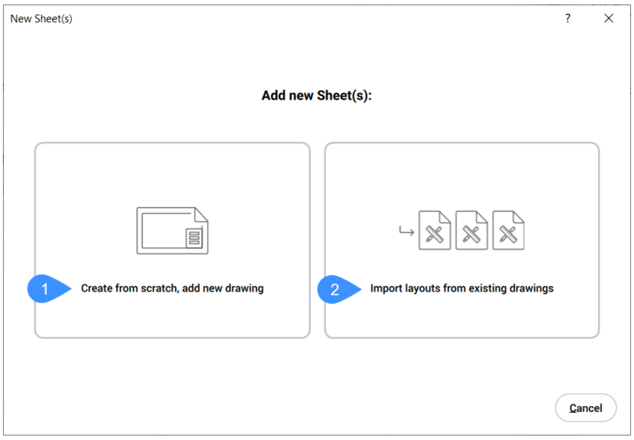
34.14.6 New Sheet(s) dialog box

Opens via:

- **Sheet Sets** panel
- **BIM Project Browser** panel

The **New Sheet(s)** dialog box allows you to create (a) new sheet(s) within an existing sheet set. You can open it by right-clicking on a sheetset, subset or sheet in the **Sheet Set Manager** panel and selecting **New Sheet(s)...** from the context menu.

There are two methods to add new sheet(s):

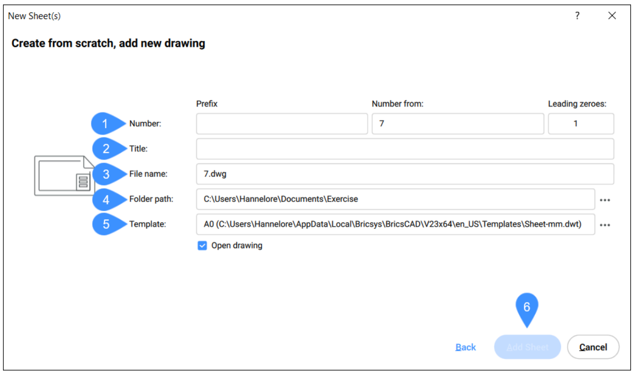


1 Create from scratch, add new drawing

2 Import layouts from existing drawings

34.14.6.1 Create from scratch, add new drawing

Allows you to create a new sheet from scratch.



1 Number

2 Title

3 File name

4 Folder path

5 Template

6 Add Sheet

Number

Defines the numbering of the created sheets.

- **Prefix**: specifies the prefix of the file name
- **Number from**: specifies the start of the numbering of the file name

- **Leading zeroes**: specifies how the Number value of new sheets is formatted by adding zeroes at the start of a sheet number.

Title
Defines the title of the new sheet(s).

File name
Defines the name of the file in which the new sheet(s) will be saved.

Folder path
Defines the location of the file.

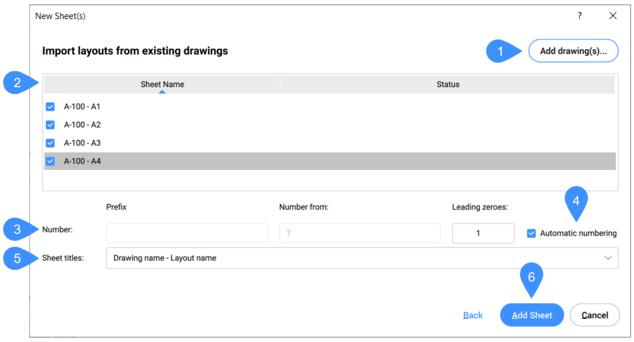
Template
Defines the template of the new sheet(s).

Add Sheet
Creates a new sheet.

Note: This option is only enabled once the title is defined.

34.14.6.2 Import layouts from existing drawings

Allows you to create a new sheet by importing layouts of existing drawings.



1 Add drawing(s)...

2 Imported layouts

3 Number

4 Automatic numbering

5 Sheet titles

6 Add Sheet

Add drawing(s)...
Selects drawings to import layouts from through the **Select Drawing(s)** dialog box.

Imported layouts
Lists the layouts that will be imported from the selected drawing. Untick the layouts that should not be imported.

Note: When importing layouts from a drawing that is not part of the project, the drawing is first copied to the project folder.

Number
Defines the numbering of the imported sheets.
- **Prefix**: specifies the prefix of the file name.
- **Number from**: specifies the start of the numbering of the file name.
- **Leading zeroes**: specifies how the Number value of new sheets is formatted by adding zeroes at the start of a sheet number.

Automatic numbering
Numbers the layouts automatically.

Sheet titles
Specifies how the selected drawing should be listed by selecting one of the options in the drop-down list.
- **Drawing name**: displays the drawing name of the selected drawings.
- **Drawing name- Layout name**: displays the drawing name and the layout name of the selected drawings.
- **Layout name**: displays the layout name of the selected drawings.

Add Sheet
The new sheet(s) are created.

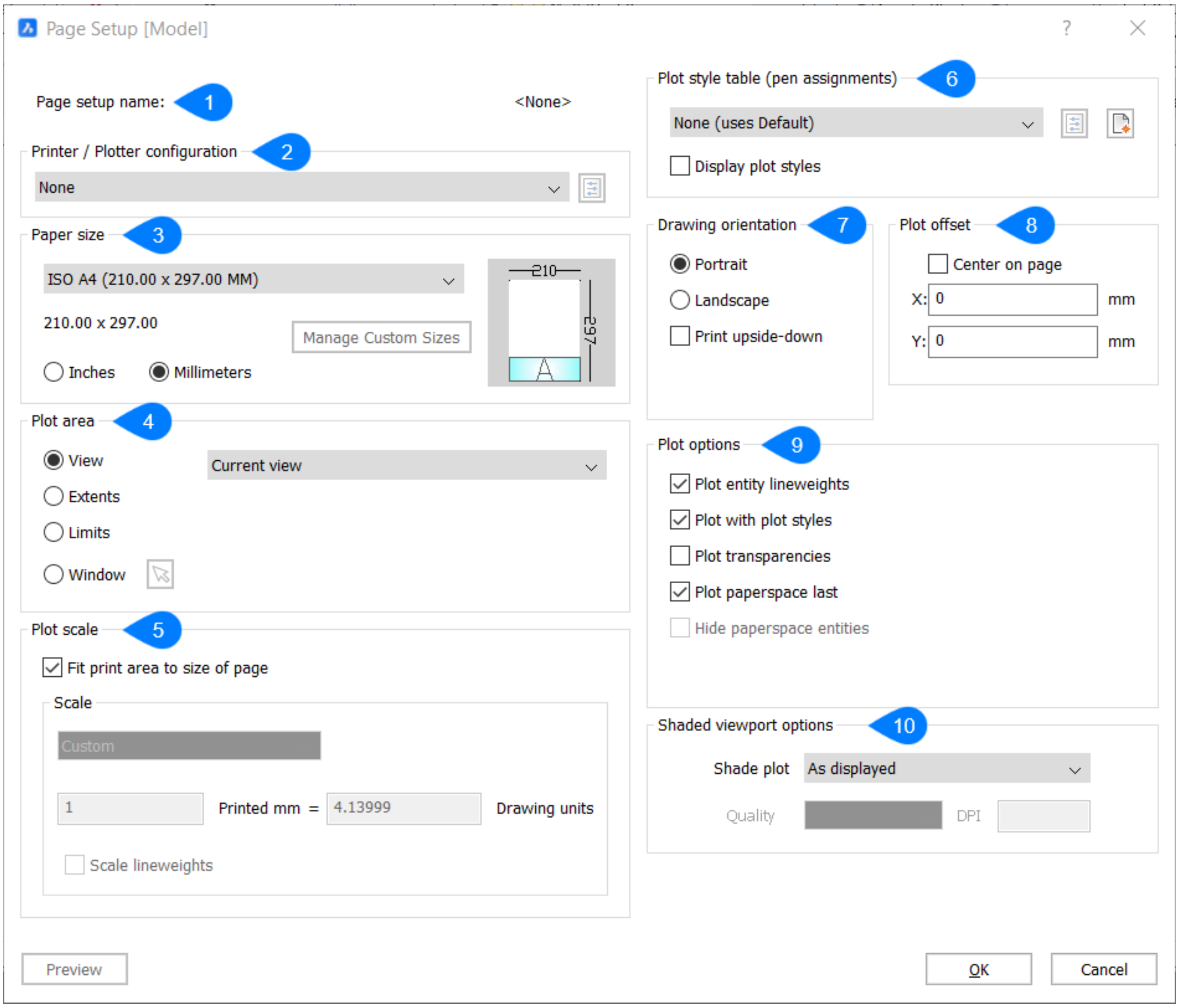
34.15 O

34.16 P

34.16.1 Page Setup dialog box

Opens via: CPAGESETUP command

The **Page Setup** dialog box allows you to edit the page setup of the current layout or model space in a dialog box.



1 Page setup name

2 Printer / Plotter configuration

3 Paper size

4 Plot area

5 Plot scale

6 Plot style table

7 Drawing orientation

8 Plot offset

9 Plot options

10 Shaded viewport options

34.16.1.1 Page setup name

Displays the name of the current setup.

34.16.1.2 Printer / Plotter configuration

Selects the printer or plotter.

34.16.1.3 Paper size

Specifies the size of paper by selecting a standard size from the drop-down list. These are sizes supported by the printer.

Note: The size can be displayed in inches or millimeters by checking one of the two options.

34.16.1.4 Plot area

Specifies which area of the drawing to print.

View
Prints the current view or a named view. Choose a view from the drop-down list.

Extents
Prints the extents of the drawing, which ensures every visible entity is printed.

Note: Entities on frozen layers are not taken into account when calculating the extents.

Limits
Prints the limits of the drawing, as specified by the Limits command.

Window
Prints a rectangular area of the drawing.

34.16.1.5 Plot scale

Scales the drawing to fit to the paper.

Fit print area to size of paper
Calculates the scale automatically, taking into account the print area of the drawing.

Scale
Specifies the scale factor to use for plotting.

Scale lineweights
Scales lineweights with respect to the plot scale.

34.16.1.6 Plot style table

Specifies the plot style table to use for the printed output, which assigns properties to "pens," colors, and entities.

34.16.1.7 Drawing orientation

Specifies the orientation of the drawing on rectangular paper.

Portrait
The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.

Landscape
The drawing or layout x-axis is aligned with the longest edge of the selected paper size.

Print upside down
Prints the drawing upside-down.

34.16.1.8 Plot offset

Specifies the offset distance for the print.

Note: To align the print in the center of the page, check the Center on page option.

Center on page
Centers the print on the page, taking into account margins.

34.16.1.9 Plot options

Plot entity lineweights
Toggles the use of lineweights.

Plot with plot styles
Toggles the use of plot styles.

Plot transparencies
Plots entities who have been assigned the transparency property, either by entity or by layer.

Plot paperspace last
Specifies the printing order.

Note: This option is only available when printing layouts.

Hide paperspace entities
Removes hidden lines from 3D entities in paper space when checked.

Note: This option is disabled when printing model space.

34.16.1.1 Shaded viewport options
0

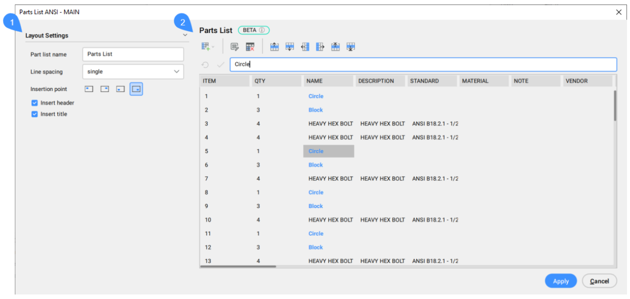
Override the visual style of the current view when printing model space.

Shade plot
Choose a shade style to plot the drawing with.

34.16.2 Parts List dialog box

Opens via: AMPARTLIST command

Defines part reference information that appears in the parts list and the layout of the parts list.

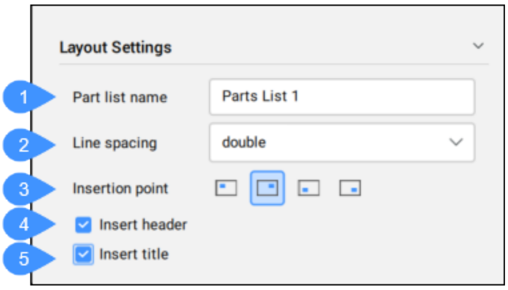


1 Layout Settings

2 Parts List

34.16.2.1 Layout Settings

Defines the layout for the parts list.



1 Part list name

2 Line spacing

3 Insertion point

4 Insert header

5 Insert title

Part list name
Defines and displays the part list name.

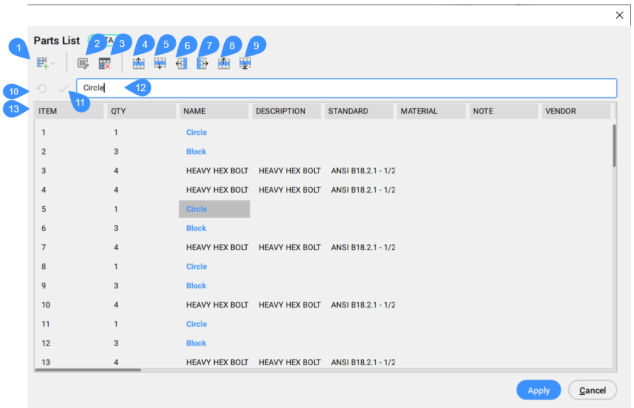
Line spacing
Defines the line spacing type by choosing one option from the drop-down list.

Insertion point
Defines the placement of the insertion point and the vertical direction of the parts list. If the insertion point is at the bottom, both the header and title points are at the bottom of the parts list, and the parts are listed from bottom to top. Similarly, if the insertion point is at the top, both the header and title appear at the top of the parts list, and the parts are listed from top to bottom.

Insert header
Defines if the list will have a header.

Insert title
Defines if the list will have a title.

34.16.2.2 Parts List



1 Manage columns

2 Set value

3 Remove overrides

4 Move up

5 Move down

6 Move left

7 Move right

8 Move to the top

9 Move to the bottom

10 Formula bar

11 Part references properties

12 List of the part references

Manage columns

Manages the columns. When you open the drop down list the available options are:

Insert column left

Inserts the chosen column to the left side of the current column. The **Select columns** dialog box opens.

Insert column right

Inserts the chosen column to the right of the current column.

Remove column

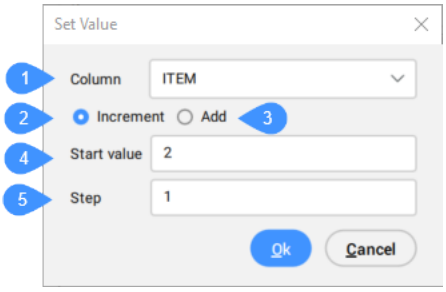
Removes the current column.

Note: The available columns are managed within the **BOM Settings** dialog box.

Note: The **Select columns** dialog box displays only those columns that have not been inserted yet.

Set value

Allows you to override values generated by formulas on the selected cells. The **Set Value** dialog box opens.



1 Column

2 Increment mode

3 Add mode

4 Start value

5 Step

Column
Allows you to select the column from the drop down list.

Increment mode
Allows you to override the value for one or more cells by setting the **Start value** and the **Step**.

Add mode
Allows you to override the value for one or more cells by setting a **Value** to add to the existing one.

Note:

- Not all columns accept overridden values.
- The overridden values are highlighted in blue color.

Start value
Defines the start value for incremental mode for overriding the cell value.

Step
Defines the step value for incremental mode for overriding the cell value.

Value
Defines the value to be added to the existing value of the cell.

Remove overrides
Removes overrides from the current selection and restores their default values as calculated by their formula.

Note: This button is available only when overrides exist within the current selection.

Move up
Moves the row of the selected cell one step up in the parts list. Multiple rows can be moved at once, too.

Move down
Moves the row of the selected cell one step down in the parts list. Multiple rows can be moved at once, too.

Move left
Moves the column of the selected cell one step to the left in the parts list. Multiple columns can be moved at once, too.

Move right
Moves the column of the selected cell one step to the right in the parts list. Multiple columns can be moved at once, too.

Move to the top
Moves the row of the selected cell to the top of the parts list. Multiple rows can be moved at once, too.

Move to the bottom
Moves the row of the selected cell to the bottom of the parts list. Multiple rows can be moved at once, too.

Formula bar
Allows you to introduce or view a formula. The new value is saved when press ENTER.

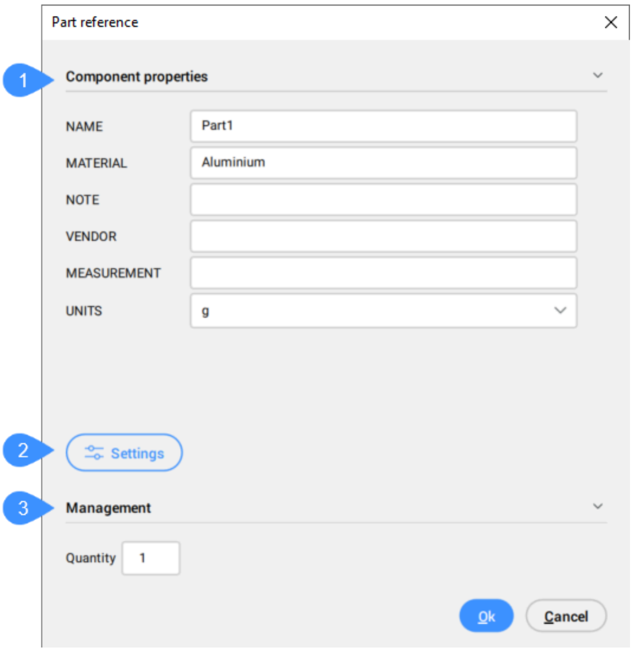
Part references properties
Displays the property names of parts references marked as visible in the **BOM Settings** dialog box that define the header of the Parts List table. The names of these properties are defined in the **Caption** field of the **BOM Settings** dialog box.

List of the part references
Lists the parts references included in the chosen BOM.

34.16.3 Part reference dialog box

Opens via: AMPARTREF command

Defines part reference information for the part list.



1 Component properties

2 Settings

3 Management

Component properties
Allows you to define some properties of the part reference. These properties are also available for editing in the **Properties** panel.

Settings
Opens the **BOM Settings** dialog box, where you can define the component properties visible in the **Part reference** dialog box.

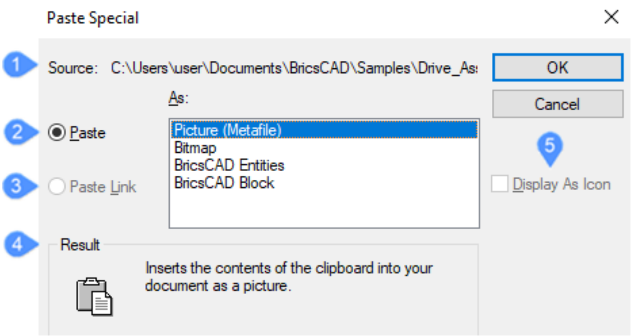
Management
Defines how part references appear in the part list.

Quantity
Allows you to override the default value of the quantity of the part references.

34.16.4 Paste Special dialog box

Opens via: PASTESPEC command

The **Paste Special** dialog box allows you the select the type of object to paste from the Clipboard into the drawing. This command is available on Windows only. Shortcut Keystroke: CTRL+ALT+V.



1 Source

2 Paste As

3 Paste Link

4 Result

5 Display As Icon

34.16.4.1 Source

Displays the path where the drawing/entity/block can be found.

34.16.4.2 Paste As

Chooses the format in which to paste the entities. This list varies, depending on the content of the Clipboard. When the Clipboard contains data copied from a BricsCAD drawing, then the following formats are available:

- **Picture Metafile**: entities are converted to WMF (Windows Metafile) format image and then pasted as an OLE (Object Linking and Embedding) image.

- **Bitmap**: Entities are converted to a BMP format image and then pasted as an OLE image.
- **BricsCAD Entities**: pastes as individual entities or table entries including layers, linetypes, etc.
- **BricsCAD Block**: inserts the contents of the clipboard into your document as a block.

Paste As BricsCAD Entities

Pastes CAD entities into the drawing as BricsCAD entities.

Select insertion point or [Rotate/Scale/Mirror]: - *(enter an option)*

1 **Select insertion point** – *(specifies a point in the drawing by picking a point or entering X, Y coordinates)*

2 **Rotate** -*prompts you:*

Specify rotation angle – *(enter an angle by which to rotate the entities)*

3 ***Scale*** -*prompts you:*

Specify scale factor for XYZ axes – *(enter a scale factor)*

4 ***Mirror*** – *prompts you:*

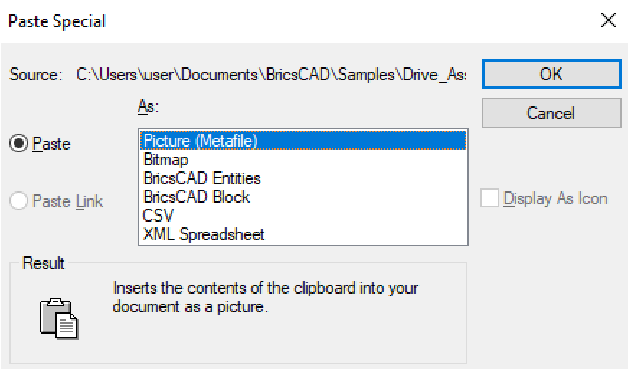
Select mirroring control point – *(Pick a second point for the mirror line)*

The insertion point is the first point for the mirror line. When you enter 0,0 for the insertion point, the entities are pasted to their original coordinates.

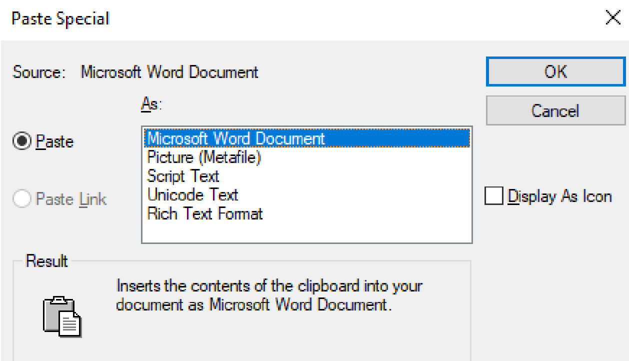
Paste As BricsCAD Block

Opens the **Insert Block** dialog box.

Here is the same dialog box, but it has some new options as the entities are copied from an Excel spreadsheet.



If you copy a text string from outside of BricsCAD, there are some new options as shown in the picture below:



34.16.4.3 Paste Link

This option is not available for CAD entities. Pastes the entities in the drawing as an OLE object and then creates a link to the originating application.

34.16.4.4 Result

Displays a short description of the selection option you want to use.

34.16.4.5 Display As Icon

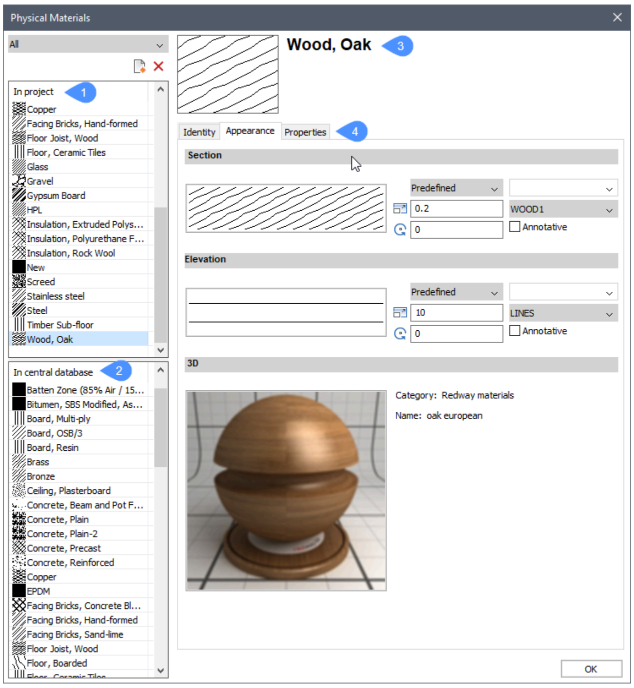
This option is not available for CAD entities. Displays the pasted entities as an icon identifying their source application.

34.16.5 Physical Materials dialog box

Opens via:

- BLMATERIALS command
- **BIM Compositions** panel

The **Physical Materials** dialog box opens the Physical Materials library in the current project and the central database.



1 In project

2 In central database

3 Material name

4 Material specification tabs

34.16.5.1 In project

Displays all the materials that are currently loaded in the project.

34.16.5.2 In central database

Displays all the materials that are loaded in the central database.

34.16.5.3 Material name

Displays the name of the selected material.

34.16.5.4 Material specification tabs

Material specification tabs

Lists the broader specifications of the selected material into three tabs.

Identity

- **Name**: specifies the name of the material.
- **Class**: specifies the material class the material belongs to.
- **Description**: displays a short description of the material.

Appearance

- **Section**: specifies the hatch pattern used to display the material in a section view.
- **Elevation**: specifies the hatch pattern used to display the material in an elevation view.

- **3D**: shows the three dimensional texture of the material that will be used in all 3D views.

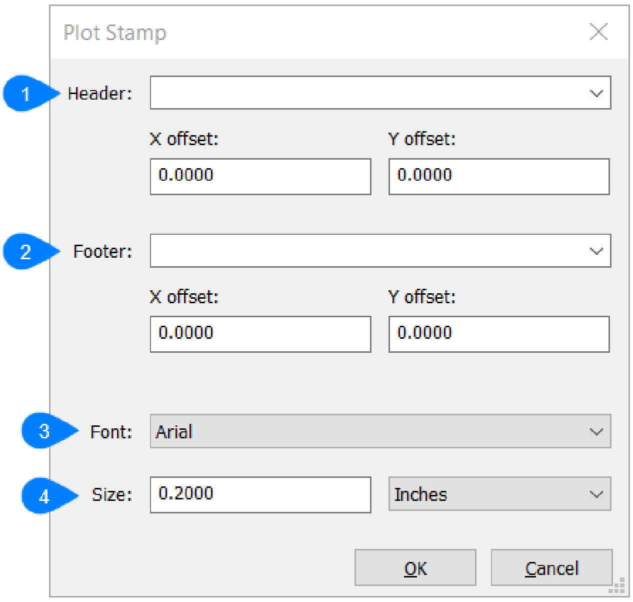
Properties

Specifies additional properties of the material such as the density.

34.16.6 Plot Stamp dialog box

Opens via: PLOTSTAMP command

The **Plot Stamp** dialog box allows you to add information about drawings and prints to the edge of the plotted drawing.



1 Header

2 Footer

3 Font

4 Size

34.16.6.1 Header

Appears at the top of the plot. A meta-phrase for the header can be chosen by clicking the drop-down list. You can also specify the X and Y offsets for the header.

34.16.6.2 Footer

Appears at the bottom of the plot. A meta-phrase for the footer can be chosen by clicking the drop-down list. You can also specify the X and Y offsets for the footer.

34.16.6.3 Font

Specifies the font used for the plotstamp.

34.16.6.4 Size

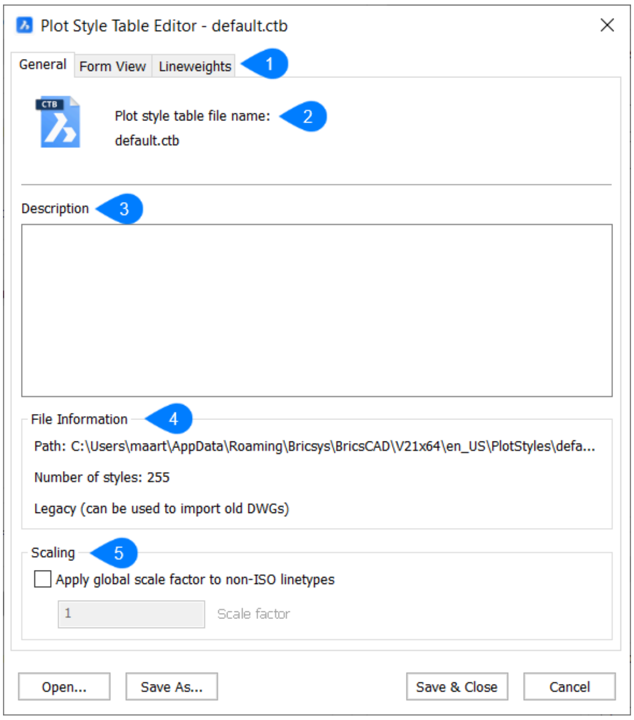
Specifies the height of the text.

- Inches: specifies the height of the text in inches.
- Millimeters: specifies the height of the text in millimeters.

34.16.7 Plot Style Table Editor dialog box

Opens via: **Current Plot Style** dialog box

The **Plot Style Table Editor** displays and allows you to edit the plot styles contained in the specified plot style table.



1 Editor tabs

2 Plot style table file name

3 Description

4 File Information

5 Scaling

34.16.7.1 Editor tabs

The Plot Style Table Editor includes the following tabs.

General tab

The **General** tab lists general information about the plot style table.

Form View tab

The **Form View** tab lists the plot styles contained in the plot style table and their settings. Plot styles are style overrides for your drawings that occur during plotting.

Lineweights tab
The **Lineweights** tab lists the available lineweights.

Note: Detailed information can be found in the **Plot style tables** article.

34.16.7.2 Plot style table file name

Displays the name of the plot style table file you are editing.

34.16.7.3 Description

Displays a description you want to include about the plot style table.

34.16.7.4 File Information

Displays information about path, number of plot styles and the version number of the Plot Style Table Editor.

34.16.7.5 Scaling

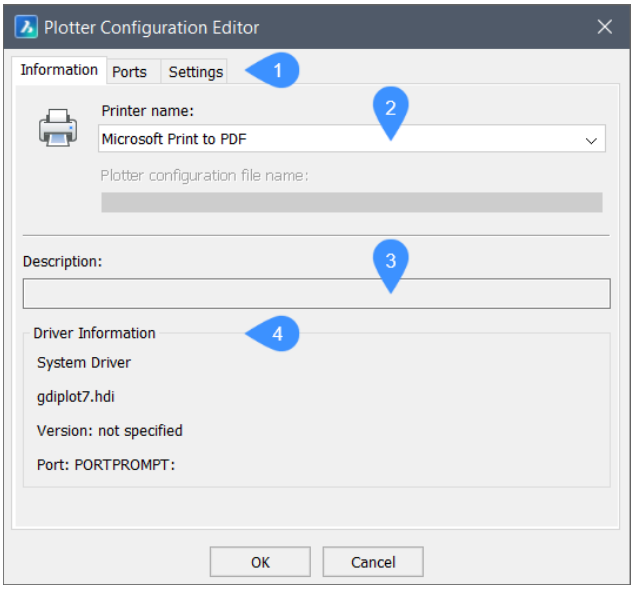
Apply global scale factor to non-ISO linetypes
Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.

Scale factor
Specifies the factor by which to scale non-ISO linetypes and fill patterns.

34.16.8 Plotter Configuration Editor dialog box

Opens via: **Print** dialog box

The **Plotter Configuration Editor** dialog box allows you to create and edit customized parameter PC3 files for printers and other output devices.



1 Editor tabs

2 Printer name

3 Description

4 Driver Information

34.16.8.1 Editor tabs

Information

Gives basic information about the plotter.

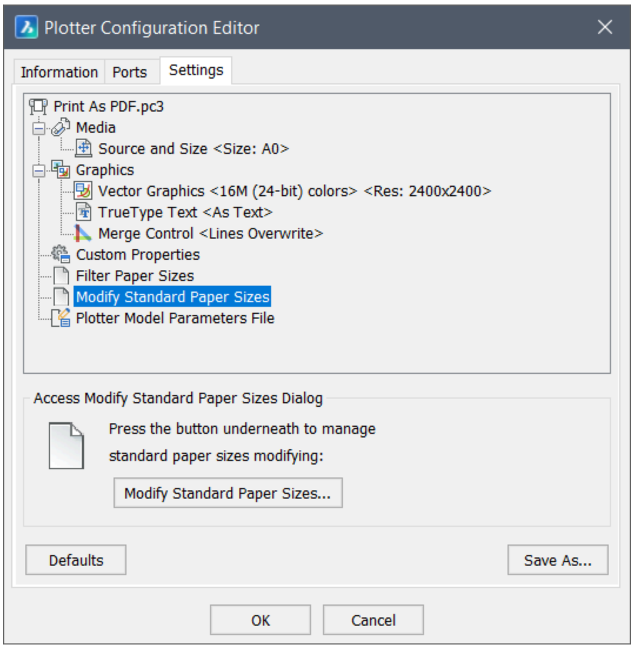
Ports

Gives options on how the file will be printed.

Settings

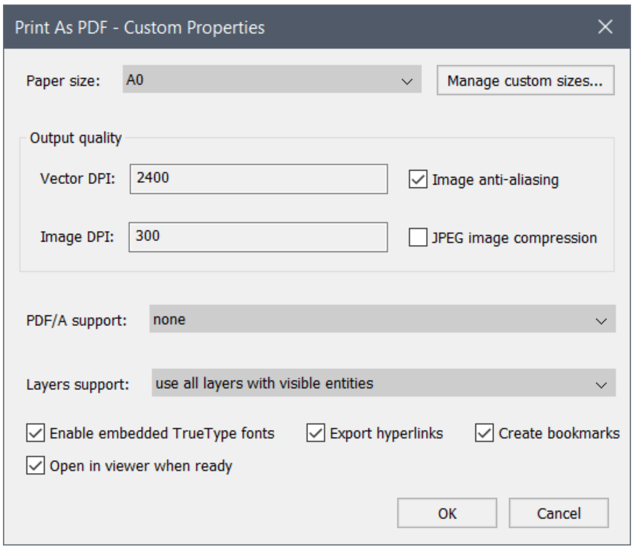
Provides additional settings to modify the available settings based on the capabilities of the selected printer.

Note: For some plotter configuration (PC3) files you can **Modify Standard Paper Sizes**:



Note: To set custom properties for the **Print as PDF.pc3** printer, click the **Custom Properties** option in the **Settings** tab, then click the **Custom Properties...** button. The **Print As PDF - Custom Properties** dialog box opens where:

- the **PDF/A support** option is available to allow you to select a PDF/A version: **version 1b**, **version 2b**, **version 3b**.
- the **Create bookmarks** option is available to automatically generate bookmarks for named view in the PDF file.
- the **Open in viewer when ready** option is available to automatically display in viewer the results of print to PDF.



34.16.8.2 Printer name

Lists the names of printers supported by the operating system.

34.16.8.3 Description

Describes the plotter configuration. It is displayed by tooltips.

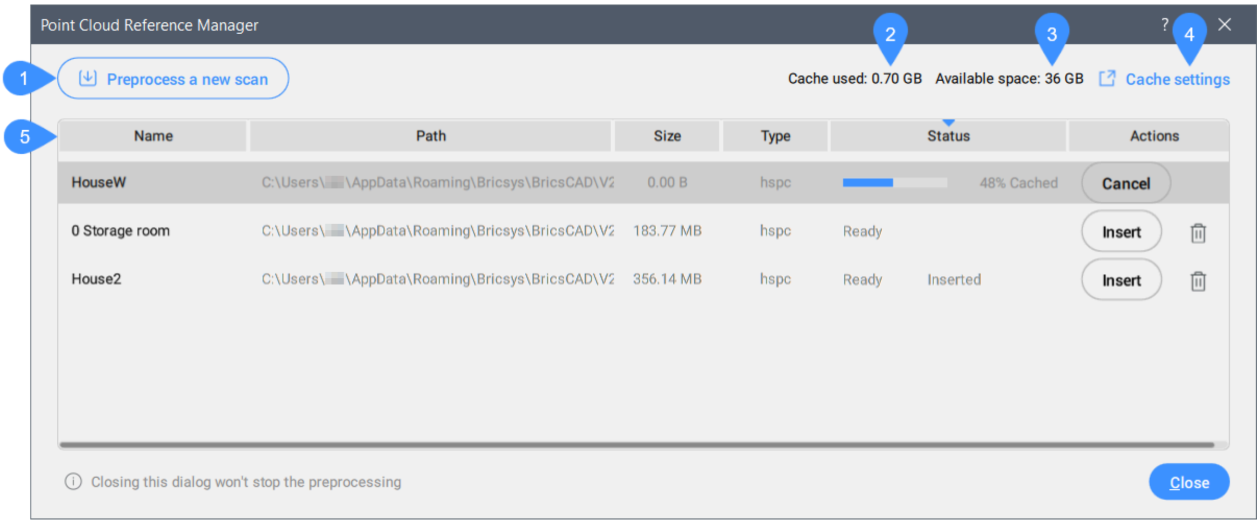
34.16.8.4 Driver Information

Displays basic information about the driver.

34.16.9 Point Cloud Reference Manager dialog box

Opens via: POINTCLOUDREFERENCE command

The **Point Cloud Reference Manager** dialog box allows you to attach and delete point cloud files that have already been processed and that are stored in the cache into the current drawing.



1 Preprocess a new scan

2 Cache used

3 Available space

4 Cache settings

5 Table of scans

34.16.9.1 Preprocess a new scan

Allows you to select a new point cloud file/folder to preprocess and opens the **Attach Point Cloud** dialog box. The **In progress bar** is displayed in the **Status** field of the **Point Cloud Reference Manager**dialog box while processing. A notification balloon displays when the point cloud is fully cached.

Note: When geographic information is available in the original data, it is stored in the point cloud's cache (only if the point cloud is preprocessed in HSPC format). You can access this information in the **Properties** panel under the **Misc** category for a selected point cloud instance. To overwrite the DWG file's geographical information with that of the point cloud instance (possible only if the CRS of the point cloud is the same as the CRS of the DWG file), turn On the **Geolocate** property of the selected point cloud instance.

34.16.9.2 Cache used

Specifies the total amount of the used cache.

34.16.9.3 Available space

Displays the total amount of available cache.

34.16.9.4 Cache folder settings

Opens the **Settings** dialog box to set the POINTCLOUDCACHEFOLDER system variable value that specifies the folder(s) where point cloud cache files are stored.

34.16.9.5 Table of scans

Lists the available preprocessed point cloud scans.

Right click a column header to display the table context menu:

- **Columns list**: Show/hide a table columns by clicking on its name in the list.
- **Show all columns**: Shows all table's columns.
- **Restore column positions**: Restores the default position of all table's columns.

Table columns
You can show or hide columns by right clicking on a column header and selecting column names in the context menu.

Name
Displays the point cloud file name.

Note:

- The point cloud cache name is a user specified name for the preprocessed point cloud. It's the default point cloud name if you do not specify a point cloud name for insertion.

- The double-byte characters are accepted for the point cloud name.

Path
Displays the point cloud file location.

Size
Displays the point cloud file size in MB.

Status
Displays the point cloud file status:

- **In progress bar**: the file is in preprocessing.
- **Ready**: available for **Insert** or **Delete**.
- **Inserted**: the file is inserted in the current drawing.

Action area

Cancel
Allows you to interrupt the preprocessing of the file. Once the point cloud file is preprocessed and ready to use, the **Cancel** button disappears.

Insert
Opens the **Attach Point Cloud** dialog box and attaches the point cloud file in the current drawing.

Note: Multiple insertions can have the same or a different point cloud name.

Delete
Removes the point cloud file from the cache.

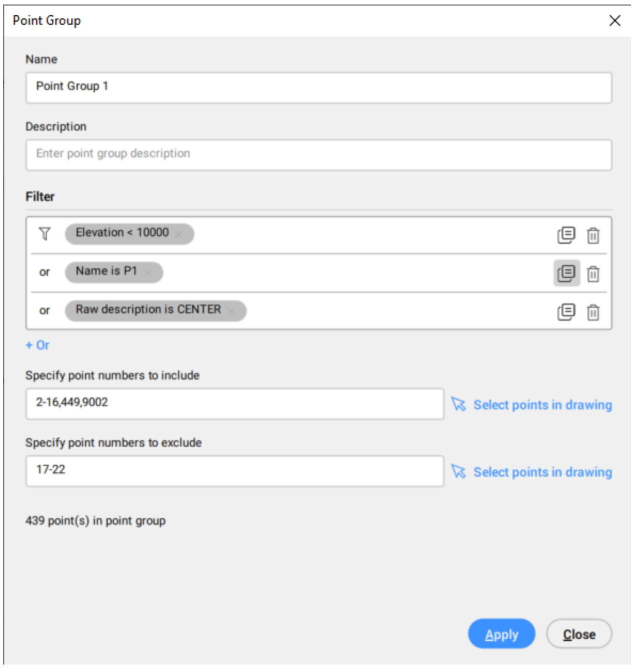
Note: It also removes all the insertions (if any) of that point cloud in the current drawing.

Type
Displays to which type the inserted file has been preprocessed. If the POINTCLOUDHSPC system variable is ON, the file is inserted as an HSPC type.

34.16.10 Point Group dialog box

Opens via: CIVILPOINTGROUP command

The **Point Group** dialog box allows you to create or edit a Civil Point Group.



Note: Click a cell to edit the content.

Name

Allows you to enter the name of a new point group for which you will create a new filter or display the name of an existing point group.

Note: If the point group name is not defined, a warning message appears.

Description

Allows you to enter a description of the point group or display the description of an existing point group.

Filter

Allows you to create a new or to edit an existing filter in which you specify the criteria for including and/or excluding points in the selected point group.

Note: You can add, duplicate or delete the filters.

Note: Once an individual filter is created in a particular drawing, it also affects all the points that are inserted later in that drawing.

The parameters used for filtering are:

- Elevation
- Name
- Raw description
- Full description
- User attribute key
- User attribute value

Specify point numbers to include

Allows you to enter the point numbers to be included in the specified filter.

Note: Points can be selected from the drawing by pressing the arrow from the right side of the field.

Specify point numbers to exclude
Allows you to enter the point numbers to be excluded from the specified filter.

Note: Points can be selected from the drawing by pressing the arrow from the right side of the field.

Note:

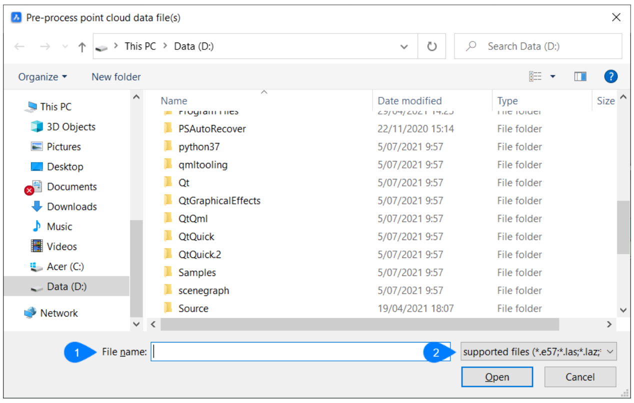
- If no filter is defined, a warning message appears.
- It is possible to create a point group even if no Civil points are present in the drawing.

34.16.11 Pre-process point cloud data file(s) dialog box

Opens via: POINTCLOUDATTACH command > **File** option

34.16.11. About
1

The **Pre-process point cloud data file(s)** dialog box allows you to attach point cloud files (that were previously converted to BricsCAD's BPT format) to the current drawing.



1 File Name

2 Supported files

34.16.11. File Name
2

Specifies the name of the point cloud data file you want to attach.

Note: You can do a manual search in the window above.

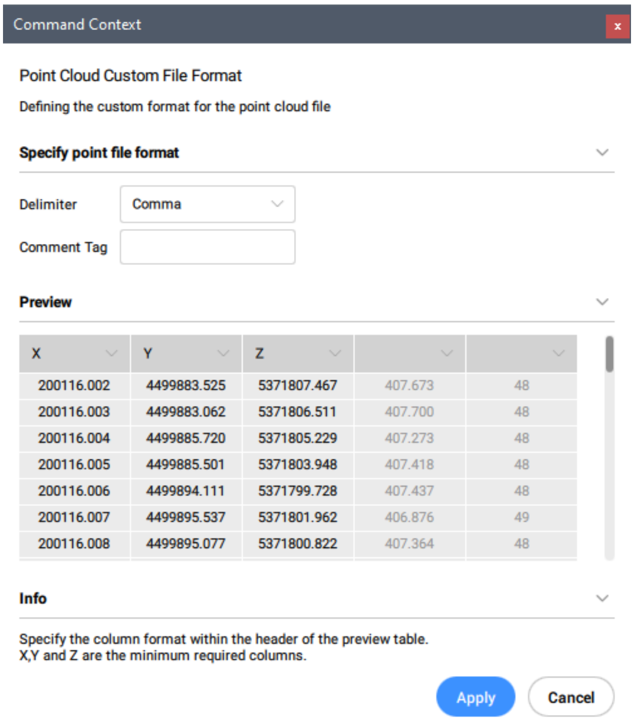
34.16.11. Supported files
3

Specifies the file types you will be able to choose from.

The supported files types are: *.e57, *.hspc, *.las, *.laz, .lgsx, *.pack, *.pts, *.ptx, *.rcp, *.rcs, *.rdbx, *.rsp, *.zfs.

If **all files (*)** is selected in the drop-down list, you can select custom point file formats for pre-processing. The **Point Cloud Custom File Format** command context opens, and you can define the parameters used within the custom file format.

You can use the custom point file format to parse point cloud files for which the data does not have a standard structure. However, it requires the data to be in a table format. By specifying the delimiter and optionally the comment tag, the first lines of the data are read and previewed in the dialog box. Next, you can specify the column headers. Note that minimum X,Y and Z need to be specified and optionally R,G,B and I. Initially, the delimiter and column format are estimated (if possible) to speed up the process. You can use the comment tag to ignore comment lines, for example # in case the comment lines start with this symbol.



34.16.12 Print dialog box

Opens via:

- PLOT command
- PRINT command

The **Print** dialog box allows you to print and preview drawings to plotters and files.

Remember:

- On macOS and Linux it is not possible to print to system printers. It is only possible to print to the **Print As PDF.pc3** printer. So, a PDF document will be generated, which needs to be sent to the printer for physical printing.
- On macOS and Linux there is a relevant PAGESETUP command.

1 Page setup name

2 Create page setup

3 Printer/Plotter configuration

4 Edit plotter configuration

5 Paper size

6 Plot area

7 Plot scale

8 Plot style table

9 Edit Plot Style

10 Create New Plot Style

11 Drawing orientation

12 Plot offset

13 Plot options

14 Shaded viewport options

15 Print to file

16 Number of copies

17 Apply

18 Preview

34.16.12. Page setup name

1

You can select plot settings used previously from the drop-down list:

- **<None>**: Uses options saved in the current layout or model space.
- **<Previous Plot>**: Uses options that were saved from the last time when this dialog box was used.
- Names of other page setups stored in the drawing.

34.16.12. Create page setup

2

Opens the **Create Page Setup** dialog box that allows you to create new page setups. See the PAGESETUP command.

34.16.12. Printer/Plotter configuration

3

Allows you to select from the drop-down list the printer or plotter.

The program works with any output device installed on the system, including networked printers, print to file, faxes, and PostScript devices, as well as predefined printer parameters stored in PC3 files.

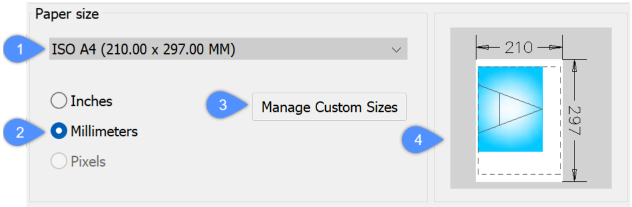
34.16.12. Edit plotter configuration

4

Opens the **Plotter Configuration Editor** dialog box for customizing the printer's parameters and creating PC3 files. See the PLOTTERMANAGER command.

34.16.12. Paper size

5



1 Standard Paper Sizes list

2 Units

3 Manage Custom Sizes

4 Preview

Standard Paper Sizes list

Specifies the size of paper. You can select standard sizes listed by the drop-down list; these are sizes supported by the printer.

While printers may appear to support many different sizes, you should select only the size of paper that is in the printer.

Units

- **Inches**: Use imperial units for measurements involving printing.
- **Millimeters**: Use metric units for measurements involving printing.

Manage Custom Sizes

Note: This option is available for the PDF, PNG, TIF, BMP, and JPG outputs.

Manages custom paper sizes for the PC3 printer selected in the **Printer/Plotter Configuration** list. The **Custom Paper Sizes** dialog box opens where you can create your own paper size.

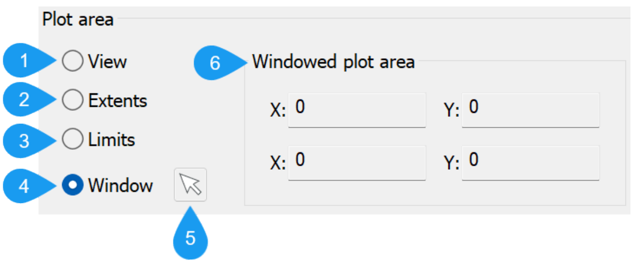
You can attach a predefined plotter model parameter (PMP) file to a plotter configuration (PC3) file.

Preview

This section indicates the size, position and orientation of the current plot area on the selected paper size.

34.16.12. Plot area

6



1 View

2 Extents

3 Layout

4 Window

5 Select area to be printed

6 Windowed plot area

View

Prints the current view or a named view. Choose a view from the drop-down list. Use the VIEW command to create named views.

Extents

Prints the extents of the drawing, which ensures every visible entity is printed. Entities on frozen layers are not considered when calculating the extents.

Layout

Prints the current layout.

Window

Prints a rectangular area of the drawing.

- You define the rectangular area by entering the x, y coordinates or by clicking the **Select Area to be Printed** button.
- The x, y coordinates resulting from the selection or from entering in the Command line are added to the **Windows Plot Area** boxes. You can edit the values in a convenient way.

34.16.12. Plot scale
7

Scales the drawing to fit it to the paper:

- **Fit print area to size of page**: Scale is calculated automatically by the program, considering the print area of the drawing. When this option is turned on, you cannot specify the scale factor:
  - **Printable area** is the size of the paper, minus the margins.
  - **Margins** are the strips along the four edges that the printer uses for handling the paper.
- **Scale**: Allows you to specify the scale factor to use for plotting; choose a scale factor from the drop-down list or choose the **Custom** option that will allow you to enter your own scale factors in **Printed inches/mm** and **Drawing units** fields. The scale factor shown can be edited with the SCALELISTEDIT command.
  - For scale factors like 1:5, the drawing is printed smaller.
  - For scale factors like 5:1, the drawing is printed larger.
- **Scale lineweights**: When on, lineweights are scaled with respect to the plot scale.

34.16.12. Plot style table
8

Specifies the plot style table to use for the printed output, which assigns properties to pens, colors, and entities.

- When the drawing does not use plot styles, then only CTB (color-based table) files are listed.
- When the drawing uses plot styles, then only STB (style-based table) files are listed.

When you switch from **None** to a named plot style, BricsCAD prompts you:

Assign plot style table to all layouts?

- **Yes**: Assigns the CTB or STB plot style file to all layouts.
- **No**: Assigns the plot style file to the current layout only.

34.16.12. Edit Plot Style
9

Opens the **Plot Style Table Editor** dialog box. See the PLOTSTYLE command. This button is available only when the drawing uses STB plot styles.

34.16.12. Create New Plot Style
10

Opens the **Add Plot Style Table** dialog box to create new plot styles. See the STYLESMANAGER command.

- When the drawing does not use named plot styles, then the wizard creates new color-dependent tables (CTB files).
- If the drawing uses named plot styles, then the wizard creates new named plot style tables (STB files).

34.16.12. Drawing orientation
11

Specifies the orientation of the drawing on rectangular paper:

- **Portrait**: The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- **Landscape**: The drawing or layout x-axis is aligned with the longest edge of the selected paper size.
- **Print upside down**: Prints the drawing upside-down. This is useful when paper with a drawing border is loaded backwards in the printer.

34.16.12. Plot offset
12

Specifies the offset distance for the print.

- **Center on Page**: Centers the print on the page, considering margins. Together with Fit Print Area to Size of Page, this option is excellent for prints where scale is unimportant.
- **X** and **Y**: Enter positive or negative distances to move the drawing in the x and/or y distances:
    - **Positive values**: Move the drawing up and to the right.
    - **Negative values**: Move the drawing down and to the left.

The lower left corner of the drawing is moved by the specified distance. This is useful when the paper has a title block area that might interfere with the drawing.

34.16.12. Plot options
13

- **Plot entity lineweights**: Toggles the use of lineweights:
    - On: Mimics the lineweights assigned to the drawing in the print.
    - Off: Ignores lineweights.
- **Plot with Plot Styles**: Toggles the use of plot styles:
    - On: Uses plot styles to determine the look of the printed drawing and overrides lineweight settings.
    - Off: Does not use plot styles.
- **Plot Transparencies**: Plots entities who have been assigned the Transparency property, either by-entity or by layer.
- **Plot Paperspace Last**: Specifies the printing order:
    - On: Prints model space entities, followed by paper space entities.
    - Off: Prints paper space entities first, followed by entities in model space.

- **Hide Paperspace Entities**: When on, removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.
- **Plot Stamp**: Toggles use of the plot stamp; click the **Edit Plot Stamp** button to change plot stamp data. See the PLOTSTAMP command. This option is unavailable when the Printer/Plotter configuration is set to **Print As PDF.pc3**.
    - On: Applies plot stamp data to the print.
    - Off: Does not apply plot stamps.
- **Save Changes to Layout**: Determines if the options you changed in this dialog box are saved; the next time you use this dialog box, you can select "Layout" from the Use Plot Settings From drop-down list.
    - On: Saves changes made to this dialog box with the layout.
    - Off: Does not save changes.

34.16.12. Shaded viewport options 14

Override the visual style of the current view when printing model space.

Choose from the drop-down list a shade style to plot the drawing with.

This option is disabled when printing a paper space layout. The **Shade Plot** mode of a paper space viewport is defined in the **Shade Plot** property of the viewport. The quality of the **Rendered** option is defined through the current render preset; see the RENDERPRESETS command.

34.16.12. Print to file 15

When checked, redirects the print output to a *.plt file, which can be processed by certain types of software.

34.16.12. Number of copies 16

Specifies the number of copies to print. Enter a number or click the buttons to change the value.

34.16.12. Apply 17

Applies the changes you made to this dialog box. The changes are remembered the next time you use this command. This is like using the PAGESETUP command.

34.16.12. Preview 18

Displays a preview of the print. In the preview window, click the **Print Settings** button to return to this dialog box. See the PREVIEW command.

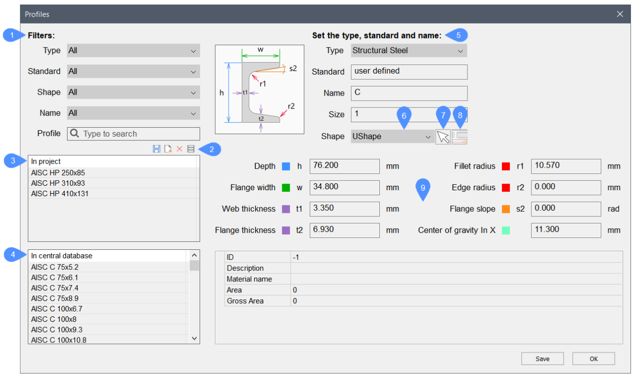
34.16.13 Profiles dialog box

Opens via:

- BIMPROFILES command

- BIMAPPLYPROFILE command
- BIMBEAM command
- BIMCOLUMN command
- BIMLINEARSOLID command

Opens the profiles library in the current project and the referenced library.



1 Filters

2 Tools

3 In Project

4 In central database

5 Custom fields

6 Preselect shape

7 Pick profile in model

8 Set profile offset

9 Properties of the profile

34.16.13. Filters

1

Allows you to filter the profiles list by type, standard, shape or name.

- **Type**: the options are **All**, **HVAC**, **Piping**, and **Structural Steel**. The **Electrical**, **Generic**, and **Structural Concrete** options become available for filtering if new profiles with these types are created.
- **Standard**: the Localized **Standard** options are **All**, **AISC** (American Institute of Steel Construction), **AS** (Australian Steel), **BS** (British Steel), **CNS** (Chinese National Standard), **EURO** (European standard steel profiles), **GOST** (Russian Steel Standard), **HVAC**, **JIS** (Japanese Industrial Standards), **Pipe**, **SAISC** (Southern African Institute of Steel Construction), and **STO ASChM** (Russian Steel Standard). The **user defined** option becomes available for filtering if new profiles are created.

- **Shape**: the options are **All**, **Circle**, **Circle Hollow**, **Custom Shape**, **I Shape**, **L Shape**, **Rectangle**, **Rectangle Hollow**, **T Shape** and **U Shape**. Depending on the selected standard, some shape options are not available. The **AsymmetricIShape**, **CraneRailAShape**, **CraneRailFShape**, **Ellipse**, **RoundedRectangle**, **Trapezium**, **ZShape** options become available for filtering if new profiles with these shapes are created.
- **Name**: filters the profiles by a selected character string.
- **Profile**: selects a character string. Only profiles that contain the selected string are listed.

34.16.13. Tools 2

- **Save Profile**: saves the currently edited profile.
- **New Profile**: creates a new profile.
- **Delete Profile**: deletes the selected profile.
- **Project and Library Information**: opens the **BIM Project Info** dialog box. Press the **Import** button to import profiles with XML/CSV files in project database (BSYSLIB).

34.16.13. In Project 3

Lists the available profiles used in the current project with respect to the above filters.

34.16.13. In Central Database 4

Lists the available profiles with respect to the above filters.

34.16.13. Custom fields 5

Lists the characteristics of the selected profile, or in case of creating a new profile, sets the type, standard and name for it.

34.16.13. Preselect shape 6

Selects a shape from the drop-down list.

34.16.13. Pick profile in model 7

Select a closed 2D entity or the boundary entities of an enclosed area.

34.16.13. Set profile offset 8

Sets the profile offset.

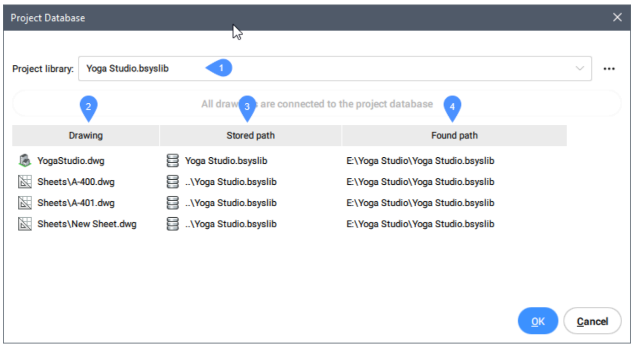
34.16.13. Properties of the profile 9

Shows the properties of the selected profile or sets the properties of a new profile.

34.16.14 Project Database dialog box

Opens via: **BIM Project Browser** panel

Shows the project database location. All drawings are connected to the project database.



1 **Project library**: lists all found bsyslib files in the current project folder. You can change the bsyslib by choosing another one from the drop-down list. By pressing the three dots a **Project library name** dialog box opens, that allows you rename the current library file.

2 **All drawings are connected to the project database**: Allows to link all drawings to the project library in one click (setting this library as their external bsyslib).

Note: This button is only enabled when not all drawings are connected to the current library file.

Note:A drawing that is not yet connected to the project database is marked with a yellow triangle and can be connected to the database by choosing the option from the context menu opened with right-click:

3 **Drawing**: displays the project drawings.

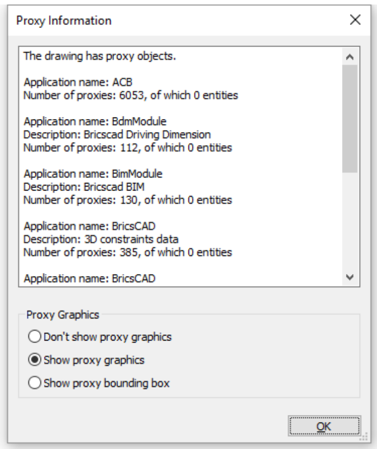
4 **Stored path**: displays each drawing's path.

5 **Found path**: displays the found path of the drawings.

34.16.15 Proxy Information dialog box

Opens via: PROXYINFO command

The **Proxy Information** dialog box displays information about proxy entities and toggles their display in the current drawing.



34.16.15. Proxy Graphics

1

Toggles the display of proxy graphics in the drawing:

- **Don't show proxy graphics**: turns off the display of proxy graphics.
- **Show proxy graphics**: turns on the display of proxy graphics
- **Show proxy bounding box**: displays a rectangle or cube instead of the proxy.

34.16.16 Publish dialog box

Opens via: PUBLISH command

The **Publish** dialog box allows you to send one or more drawings, layouts and sheets to the printer or export them in PDF format. This is useful for batch printing of books of drawings.

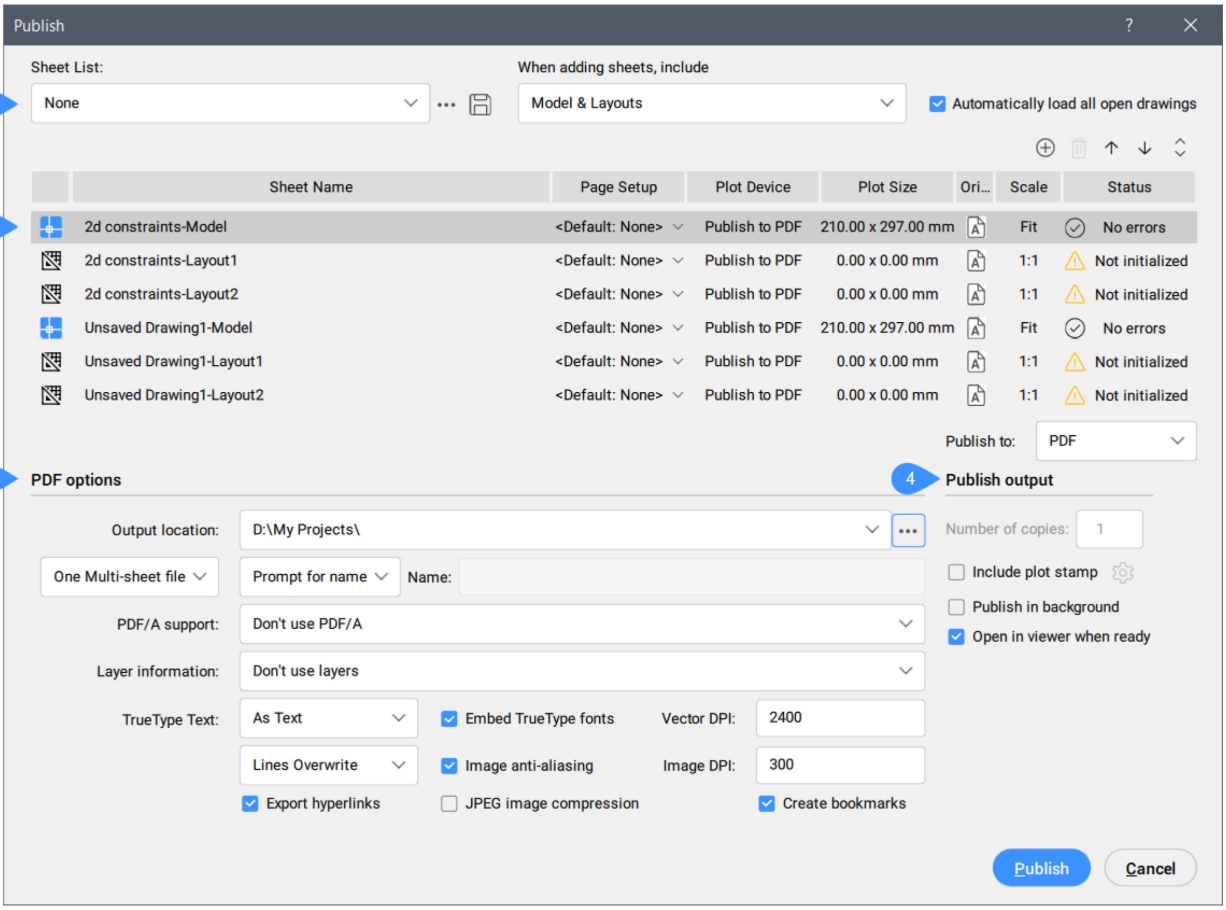
To print a batch of drawings when BricsCAD starts, write in the Terminal the following command:

```
bricscad.exe /pl <"the path of your *.dwg file"> <"the path of your *.dsd file">
```

For example:

```
bricscad.exe /pl "C:\Users\user\Desktop\doc.dwg" "C:\Users\user\Desktop\doc.dsd"
```

Note: In the **Page Setups** section of the **Drawing Explorer** dialog box, set the printer you use in the **Device name** field.



1 Sheets selection

2 Sheet list preview

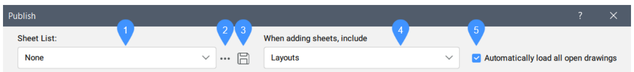
3 PDF options

4 Publish output

34.16.16. Sheets selection

1

Displays a list of sheets. If you have a *.dsd file (drawing sheets for publishing), select it and the **Load Sheet List** dialog box appears.



Sheet List

Specifies the selected sheet list. A sheet list can be selected in the drop-down menu.

Load Sheet List

Loads a saved sheet list. Opens the **Load Sheet List** dialog box. If the current list is not saved, you are prompted to save it.

If you are loading a file with an existing name, after you press the **Open** button in the **Load Sheet List** dia-log box, you are prompted to replace or append the sheets to the list.

Save Sheet List
Saves the current sheet list to a *.dsd (Drawing Set Description) file. The **Save Sheet List** dialog box opens.

When adding sheets, include
Determines which sheets are included when loading new sheets using the **Add Sheets** button. Click the drop-down button to choose between **Model**, **Layouts**, and **Model & Layouts**.

- **Model**: includes only model tabs of the selected drawings.
- **Layouts**: includes only layout tabs of the selected drawings.
- **Model & Layouts**: includes both model and layout tabs of the selected drawings.

Automatically load all open drawings
Determines how open drawings are handled:

- **On** (checked): adds all model and/or paper space layouts to the list of sheets to be plotted.
- **Off** (unchecked): adds the model and/or paper space layouts of the current drawing only.

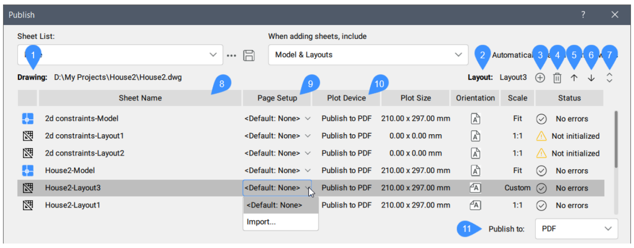
This option also sets the value of the PUBLISHALLSHEETS system variable.

Note: The PUBLISHCOLLATE system variable determines whether the plotting process of a sheet set can be interrupted by other plot jobs (for plot driver that support the multi-sheet plotting or printing option).

34.16.16. Sheet list preview

2

Displays characteristics of all selected sheets.



Drawing
Specifies the path of the drawing file selected in the list.

Layout
Specifies the layout of the drawing selected in the list.

Add Sheets
Displays the **Select Drawing(s)** dialog box to load sheets.

Remove Sheets
Removes the selected sheet from the sheet list.

Move Sheet(s) up
Moves the selected sheet one position up in the list.

Move Sheet(s) down
Moves the selected sheet one position down in the list.

Reverse Sheet Order

Reverses the order of the sheets list.

Page Setup

Specifies the Page Setup name of each sheet. Click the **Page Setup** field, then click the down arrow to select a different page setup or choose **Import...** to import page setups of another drawing. The **Import Pagesetups** dialog box opens.

Plot Device

Displays the name of the plot device, as defined in the **Page Setup** dialog box of the sheet.

Publish to:

PDF

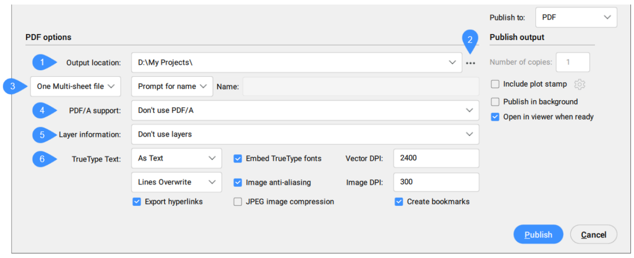
The **PDF options** section is displayed.

Plotter named in page setup

Publishes to the plotter displayed in the **Plot Device** field (set via the **Page Setup** dialog box).

34.16.16. PDF options

3



Output location

Displays the path where the PDF will be saved.

Browse button

Opens the **Choose a folder** dialog box to change the path.

Output file

Select an option:

Single–sheet files

Each layout is published to a separate PDF file. The file names are: <drawing>_<layout>.pdf.

Name options

Select an option:

- **Use sheet name**: the PDF will have the same name as the sheet file.
- **Use layout name**: the PDF will have the same name as the layout.
- **Use drawing name**: the PDF will have the same name as the DWG file.

One Multi–sheet file

Layouts are published to a single PDF file. You can specify a name or be prompted for a name when the publishing procedure starts.

Name options

Select an option:

- **Prompt for name**: after you press the **Publish** button you are prompted to insert the name you want for your PDF in the **Specify PDF file name** dialog box.
- **Specify name**: the **Name** field becomes available. Insert the PDF name.
- **Generate name**: the PDF will have the same name as the drawing.

PDF/A Support

Allows you to select an option from the drop-down list: **Don't use PDF/A** (default option), **Use version PDF/A-1b**, **Use version PDF/A-2b**, **Use version PDF/A-3b**.

Layer information

- **Don't use layers**: no layers are exported.
- **Use all layers with visible entities**: only layers which are current ON and Thawed are exported.
- **Use layers with entities, including OFF and FROZEN layers**: all layers containing entities are exported.

TrueType Text

Select an option:

- **As text**: ensures that the text in the PDF looks exactly as in the drawing.
- **As geometry**: converting to geometry increases the file size and pixelation might occur when viewed at a high zoom level.

Merge control

Determines whether overlapping lines are merged or lie on top of each other.
Select an option:

- **Lines Overwrite**: the lines on top overwrite the lines below.
- **Lines Merge**: the colors of crossing lines are blended into a new color.

Export hyperlinks

If checked, it allows you to export hyperlinks in the PDF.

Embed TrueType fonts

If the font is not embedded, a substitute font might be used by the PDF viewer.

Note: Licensed fonts cannot be embedded.

Image anti-aliasing

If the resolution of an image is too low to match the Image DPI setting, the raster image is anti-aliased (generate pixels artificially) to make the raster image look more smoothly. Without that setting an image with a too low resolution is embedded as-is.

JPEG image compression

This setting applies to all included raster images in the PDF, both raster images and rendered views. It applies a JPEG compression to these images to reduce the PDF file size. When switched off, the images are embedded as raw bitmaps. The JPEG compression is lossy compared to the raw bitmap introducing small imperfections.

Vector DPI

Controls the resolution of vector graphics and gradients.

Image DPI
Defines the resolution of the resulting image in the PDF for raster images. This setting allows you to lower the resolution of the original raster image to avoid including a gigabyte raster image in the PDF. For printing, the resolution should match the output resolution of the printer. For on-screen viewing, high resolutions (2400 DPI +) are required.

Create bookmarks
In case of choosing **One Multi-sheet file** option, it creates bookmarks for each sheet in the output file.

34.16.16. Publish output

4
Specifies to what plotter the sheets should be published to.

Note: When **Publish to** is set to **Plotter named in page setup**, the PDF options from this dialog box disappear.

Number of copies
Sets the number of copies for each sheet to be printed.

Include plot stamp
If this option is checked, it adds a plot stamp.

Edit Plot Stamp
Opens the **Plot Stamp** dialog box.

Publish in background
Determines if the sheet is published in the background:

- **Yes**: sheets are published in the background, which takes longer to produce the prints, but allows you to continue to work in the program.
- **No**: sheets are published in the foreground, which is faster, but prevents you from working with the program until the print job is completed.

The BACKGROUNDPLOT system variable controls whether documents are published in the foreground or background.

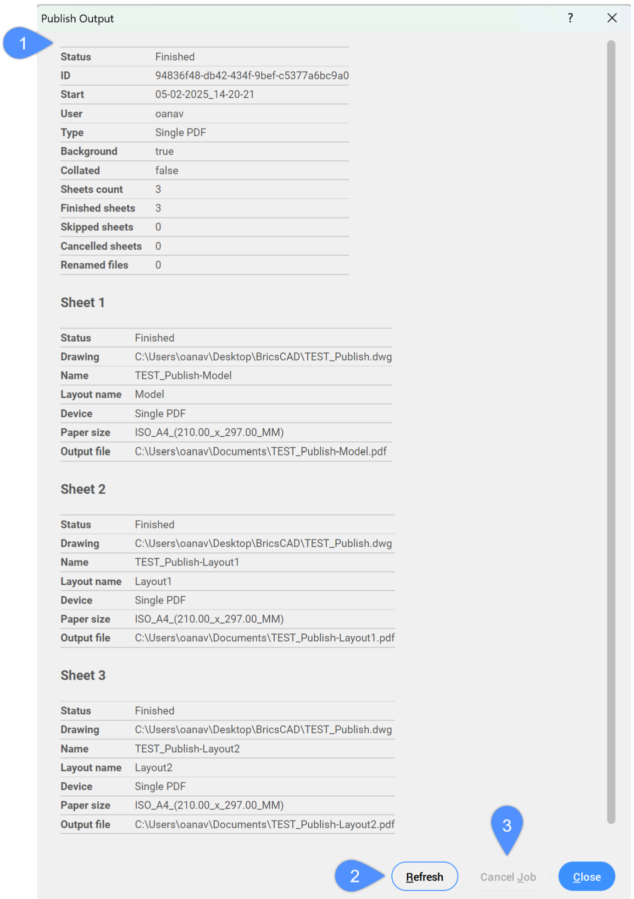
Open in viewer when ready
Determines if the sheet set is opened in the viewer when it is ready.

Note: This option is not available when **Publish to** is set to **Plotter named in page setup**, only when **Publish to** is set to **PDF**.

34.16.17 Publish Output dialog box

Opens via: VIEWPLOTDETAILS command

The **Publish Output** dialog box allows you to view the details about publish jobs, the errors that have occurred, or to cancel the ongoing publish jobs.



Publish Output Details (1)

Displays detailed information about completed publish jobs or the errors that have occurred. You can also monitor the ongoing publish jobs.

Note: Check the count to ensure that all sheets were properly processed. The following counts are displayed:

- **Sheets count**: displays the total number of sheets specified for printing.
- **Finished sheets**: displays the number of sheets that were printed.
- **Skipped sheets**: displays the number of sheets that could not be printed.
- **Cancelled sheets**: displays the number of sheets that were cancelled.
- **Renamed files**: displays the number of renamed files. This means the sheet was printed but the name was modified to avoid name collision or disallowed characters.

Refresh (2)

Refreshes the list of published jobs to see the progress before the process is finished.

Cancel Job (3)

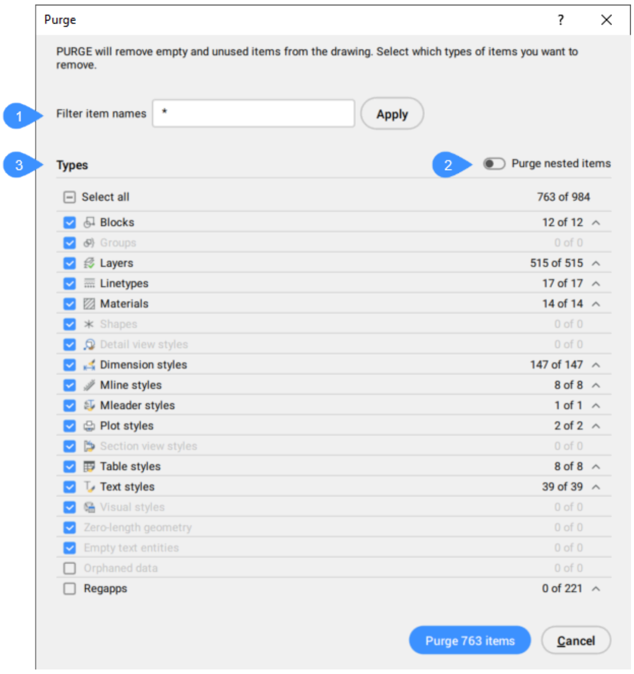
Stops the ongoing publish jobs.

Note: It is only available when the publish process is in progress. When the process is completed or cancelled, the button becomes greyed out.

34.16.18 Purge dialog box

Opens via: PURGE command

The **Purge** dialog box allows you to select which types of items and which items will be removed from the drawing.



1 Filter item names

2 Purge nested items

3 Types

34.16.18. Filter item names

1

Apply a filter to the item names in the drawing.

34.16.18. Purge nested items

2

When enabled, it purges all nested items. Therefore, it is not needed anymore to run PURGE multiple times.

34.16.18. Types

3

You can toggle on/off the different types of items that will be removed.

Blocks

Removes unused block definitions from the current drawing.

Detail view styles

Removes unused detail view styles.

Dimension styles
Removes unused dimension styles from the current drawing.

Groups
Removes unused named groups from the current drawing.

Layers
Removes unused layers from the current drawing.

Linetypes
Removes unused linetypes from the current drawing.

Materials
Removes unused materials from the current drawing.

Mline styles
Removes unused multiline styles from the current drawing.

Multileader styles
Removes unused multileader styles from the current drawing.

Plot styles
Removes unused plot styles from the current drawing.

Regapps
Removes unused regapp keys from the current drawing.

Note: Regapp (registered application) keys are used to attach additional information to entities in drawings by third-party applications.

Section view styles
Removes unused section view styles from the current drawing.

Shapes
Removes unused shapes loaded into the drawing with the LOAD command.

Table styles
Removes unused table styles from the current drawing.

Text styles
Removes unused text styles from the current drawing.

Visual styles
Removes unused user-defined visual styles from the current drawing.

Note: Only user-defined visual styles can be purged. Hard-coded visual styles cannot be purged.

Zero-length geometry
Removes geometry without length.

Empty text entities
Removes empty text entities from the current drawing.

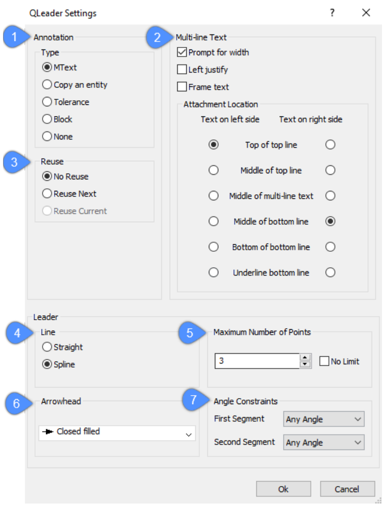
Orphaned data
Performs a drawing scan and removes unused DGN linestyles data.

34.17 Q

34.17.1 QLeader Settings dialog box

Opens via: QLEADER command

The **Qleader Settings** dialog box allows you to specify the properties of leaders.



1 Annotation

2 Multi-line Text

3 Reuse

4 Leader Line

5 Maximum Number of Points

6 Arrowhead

7 Angle Constraints

34.17.1.1 Annotation

Specifies the type of annotation:

- **Mtext**: Text is placed as the leader line's annotation; a leader can have one or more lines of text.

  Note: Although the mtext editor does not appear when entering text (as with the MLeader command), the mtext editor does appear when editing the annotation (edit the text by double-clicking it).
- **Copy of an entity**: Uses any other entity (than the leader itself) in the drawing as the annotation.
- **Tolerance**: Uses tolerance notation for the annotation; displays the **Geometric Tolerance** dialog box. See the TOLERANCE command.
- **Block**: Uses a block from the drawing or an external block as the annotation. The prompt displayed is identical to that of the -INSERT command.
- **None**: No annotation is used.

34.17.1.2 Multi-line Text

Note: This option is only available when the annotation type is MText.

Specifies some multi-line text options:

- **Prompt for width**: Toggles the use of the 'Specify text width' prompt:
  - On: Displays the 'Specify text width' prompt and so wraps the text to the specified width.
  - Off: Disables the prompt.
- **Left justify**: Determines the justification of the text:
  - On: Forces the text to be left-justified always.
  - Off: The text is left or right-justified, depending on the orientation of the landing line.
- **Frame text**: Toggles the use of a frame:
  - On: Places a rectangle around the text.
  - Off: No frame.
- **Attachment location**: Determines the relative location of where the mtext is attached to the landing line.

34.17.1.3 Reuse

Specifies how to reuse the annotation:

- **No reuse**: Does not reuse the previous annotation text; you will be prompted to enter new text each time you use this command.
- **Reuse next**: You will be prompted to enter new text, which is then reused for subsequent leader(s).
- **Reuse current**: The current text is reused for each new leader, and you will not be prompted to enter new text.

34.17.1.4 Leader Line

Specifies the style of leader line:

- **Straight**: Draws the leader lines as straight segments.
- **Spline**: Draws the leader line as a smooth continuous spline.

34.17.1.5 Maximum Number of Points

Specifies the maximum number of vertices for the leader, before it stops drawing them automatically:

- Enter a number; the minimum number is 2.
- **No limit**: Keeps drawing leader segments until you press Enter at the 'To point' prompt.

34.17.1.6 Arrowhead

Specifies the type of arrow at the start of the leader; choose from the drop-down list:

- Choose an arrowhead type.
- **None**: No arrow is drawn.

- **Select arrow**: Choose a block from those in the current drawing; displays the **Select Custom Arrow Block** dialog box. See the related article **Select Custom Arrow Block dialog box**.

To create a custom arrow, draw it to your liking, and then use the BLOCK command to turn it into a block.

34.17.1.7 Angle Constraints

Set an angle to constrain the first and second leader segments. The angles are increments; for instance, choose 15 allows you to draw the leader at 0, 15, 30, 45, and so on degrees.

Click the drop-down list, then choose an option:

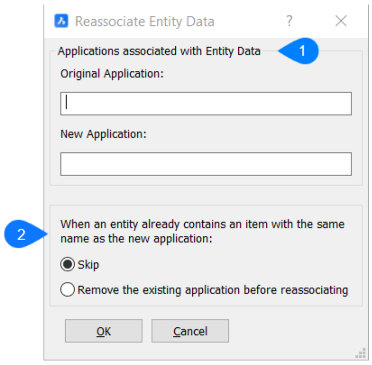
- **Any Angle**: No constraint; hold down the Shift key to draw the leader at 90 degrees.
- **Horizontal**: Forces the leader line to be drawn at 90 degrees only.
- **90°**: Draws leader lines at 90-degree increments.
- **45°**: Draws leader lines at 45-degree increments.
- **30°**: Draws leader lines at 30-degree increments.
- **15°**: Draws leader lines at 15-degree increments.

34.18 R

34.18.1 Reassociate Entity Data dialog box

Opens via: REASSOCAPP command

The **Reassociate Entity Data** dialog box allows you to associate extended entity data with a specific application.



1 Applications associated with Entity Data

2 Options

34.18.1.1 Applications associated with Entity Data

Original application
Specifies the name of the original application.

New application
Specifies the name of the new application. Entities will be reassociated with this application name.

34.18.1.2 Options

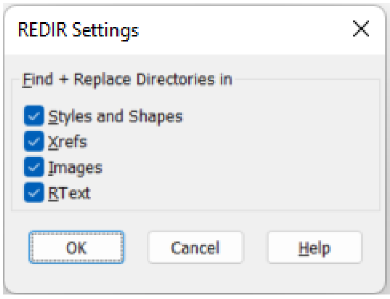
Skip
Skips entities that already contains an item with the same name as the new application.

Remove the existing application before reassociating
Removes the existing application before reassociating.

34.18.2 REDIR Settings dialog box

Opens via: REDIR command (Express Tools)

The **REDIR Settings** dialog box defines which types of entities to include for the REDIR command.

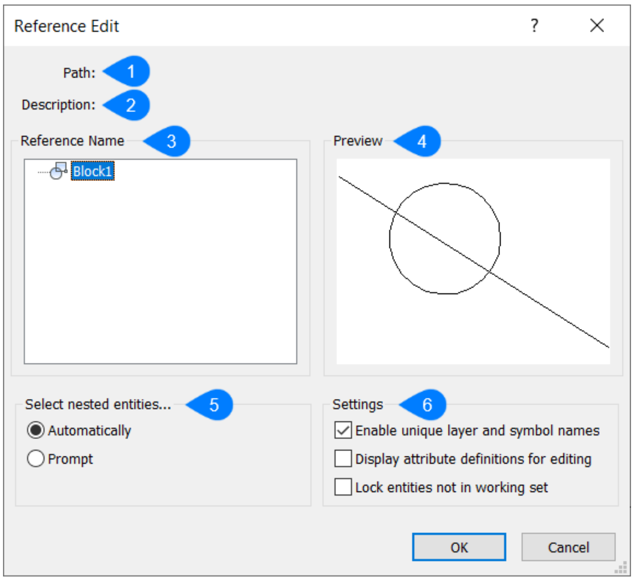


Note: At least one option must be selected.

34.18.3 Reference Edit dialog box

Opens via: REFEDIT command

The **Reference Edit** dialog box allows you to edit block references and externally-referenced drawings.



1 Path

2 Description

3 Reference Name

4 Preview

5 Select nested entities

6 Settings

34.18.3.1 Path

Displays the name of the path.

34.18.3.2 Description

Displays the description of the path.

34.18.3.3 Reference Name

Lists the name of the block and nested blocks, if any.

34.18.3.4 Preview

Displays a preview of the current reference.

34.18.3.5 Select nested entities

Determines how nested blocks/xrefs are handled.

Automatically
Includes nested entities automatically.

Prompt
Includes nested entities by selecting them after the dialog box is closed by pressing OK.

34.18.3.6 Settings

Determines how entities are treated during editing.

Enable unique layer and symbol names
Toggles the use of 0 in layer and other names.

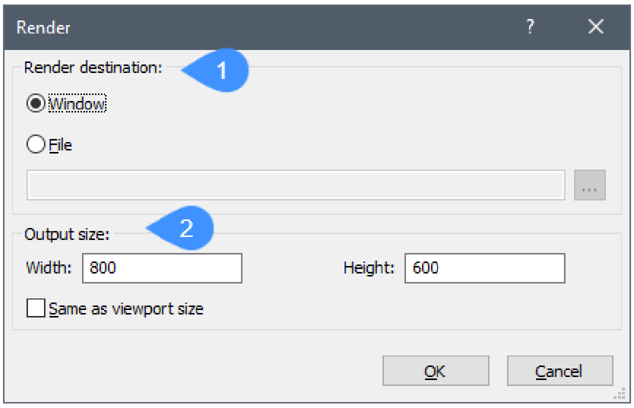
Display attribute definitions for editing
Toggles the inclusion of attributes.

Lock entities not in working set
Toggles locking of entities not being edited.

34.18.4 Render dialog box

Opens via: RENDER command

The **Render** dialog box allows you to generate a photo-realistic rendering using materials and lights.



1 Render destination

2 Output size

34.18.4.1 Render destination

Specifies where the render will appear.

Window
Generates the render in a new window.

File
Saves the render to a file. Selecting the **Browse** button opens the **Render output file** dialog box.

34.18.4.2 Output size

Specifies the size of the rendering, in pixels. Larger sizes provide more detail but take longer to render.

Width
Specifies the width of the rendering, in pixels.

Height
Specifies the height of the rendering, in pixels.

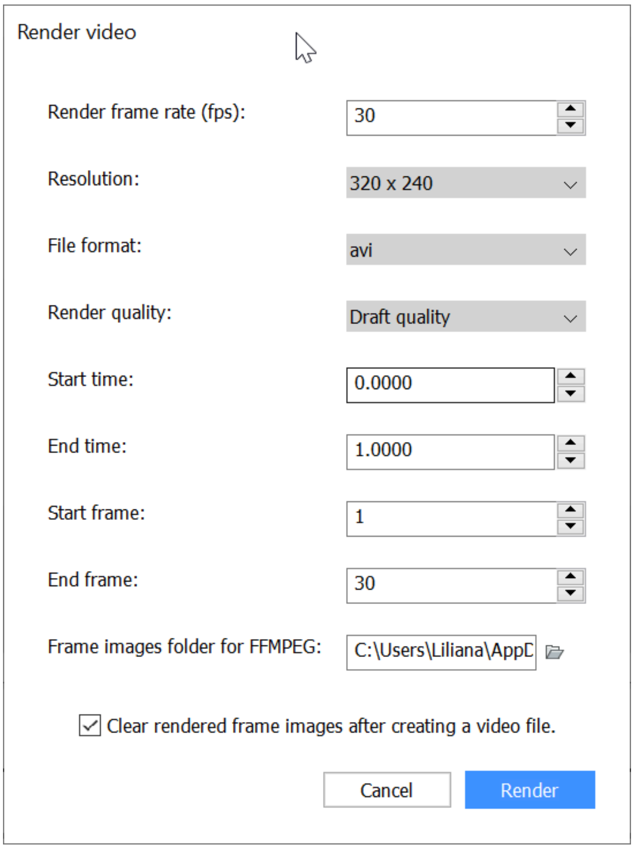
Same as viewport size
Matches the output size or the size of the output window to the current viewport size.

34.18.5 Render video dialog box

Opens via:

- **Animation Editor** panel
- -ANIMATIONEDITORPLAYBACK command

The **Render video** dialog box allows you to set the render options for your animation.



Render frame rate (fps)
Sets the frame rate of the video.

Resolution
Sets the resolution of the video by selecting it from the drop-down list.

File format
Sets the format of the output video. You can select a file format from the drop-down list:

- avi (AVI Animation)
- mpg (MPEG Animation)
- wmv (WMV Animation)

Render quality
Sets the render quality of the output video.

- **Draft quality**: for fast rendering.
- **Full quality**: the current render preset will be used to produce the same result as RENDER command.

Start time
Sets the time position of the animation to start the render.

End time
Sets the time position of the animation to end the render.

Start frame
Sets the frame position of the animation to start the render.

End frame
Sets the frame position of the animation to end the render.

Frame images folder for FFMPEG
Sets the folder where the frame images will be stored.

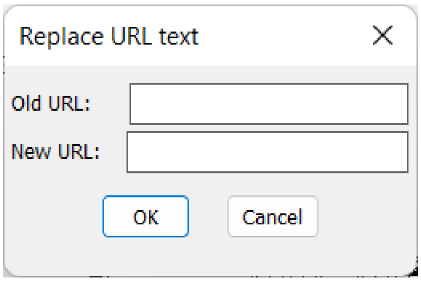
Clear rendered frame images after creating a video file
Toggles the resetting of the frame images after creating a video file. When enabled, only files created during this rendering session will be deleted.

After clicking the **Render** button, the **Save Video As** dialog box opens to allow you to save the rendered video.

34.18.6 Replace URL text dialog box

Opens via: REPURLS command (Express Tools)

The **Replace with URL text** dialog box allows you to replace a specified text string in a URL used in hyperlinks attached to the selected entities.

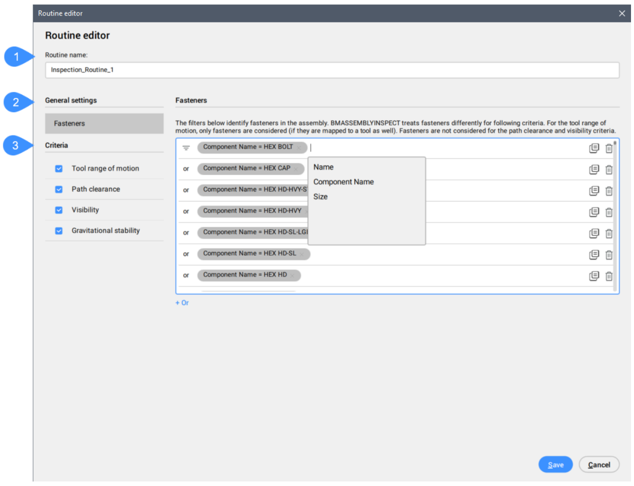


34.18.7 Routine editor dialog box

Opens via: **Manage Routines (BMASSEMBLYINSPECT)** dialog box

The **Routine editor** dialog box allows you to create criteria that will be taken in consideration in **Assembly Inspect** tool.

All fastener settings and each of the assembly criteria are stored in a routine JSON file that can be imported or exported, allowing routine files to be shared between different users.



1 Routine name

2 General settings

3 Criteria

34.18.7.1 Routine name

Specifies the profile name under which the rules are grouped.

34.18.7.2 General settings

Allows you to create a **Fasteners** pre-configuration tools.

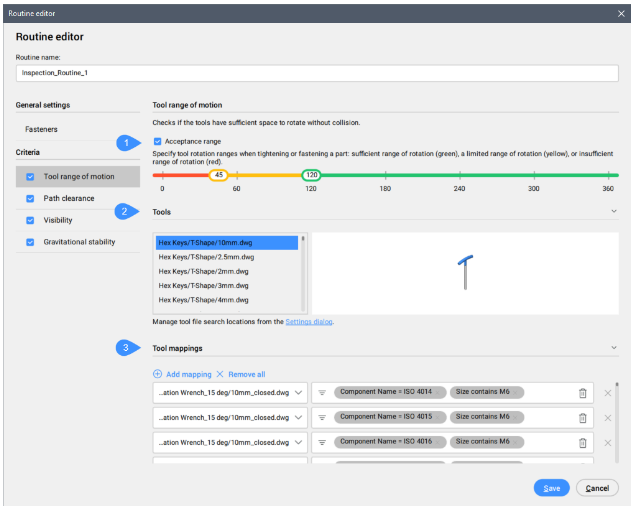
Fasteners

Defines filters for identifying fasteners in the assembly. The BMASSEMBLYINSPECT command treats fasteners differently for following criteria. For the tool range of motion, only fasteners are considered (if they are mapped to a tool as well). Fasteners are not considered for the path clearance and visibility criteria.

34.18.7.3 Criteria

There are four category of criteria which will be taken in consideration for inspection of the assembly.

Tool range of motion



1 Acceptance range

2 Tools

3 Tool mappings

Evaluates whether tools have sufficient maneuverability space. For example, whether each fasteners have enough space to be handled with the specific tool.

Acceptance range

Specifies the tool rotation ranges when tightening or fastening a part: sufficient range of rotation (green), a limited range of rotation (yellow), or insufficient range of rotation (red).

Tools

Lists the available tools (a range of standard hex keys and one screwdriver) stored in the installation folder *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support\en_US\DesignLibrary\Tools* which are loaded automatically as tools.

Note: Those hand tools can be accessed also from the **Library** panel.

The BMTOOLPATH system variable indicates the file paths used to search for tool files. You can also manage the search locations of the tool files by pressing the **Setting dialog** button.

Tool mappings

Allows you to define multiple tool-fastener mappings in order to evaluate multiple tools per fastener.

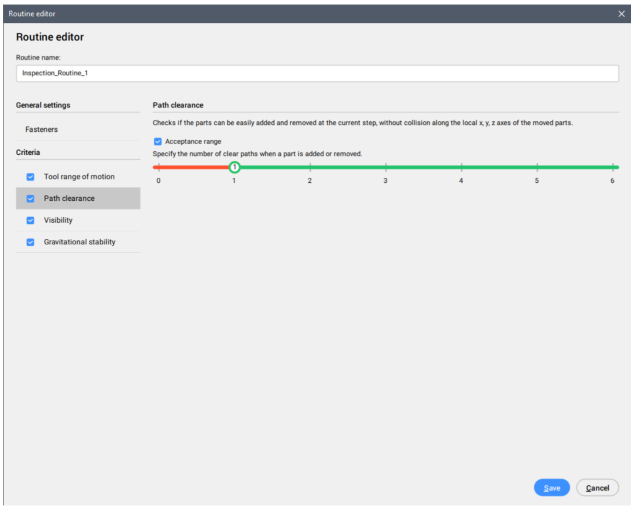
Add mappings

Allows you to add mappings.

Remove all

Removes all mappings.

Path clearance

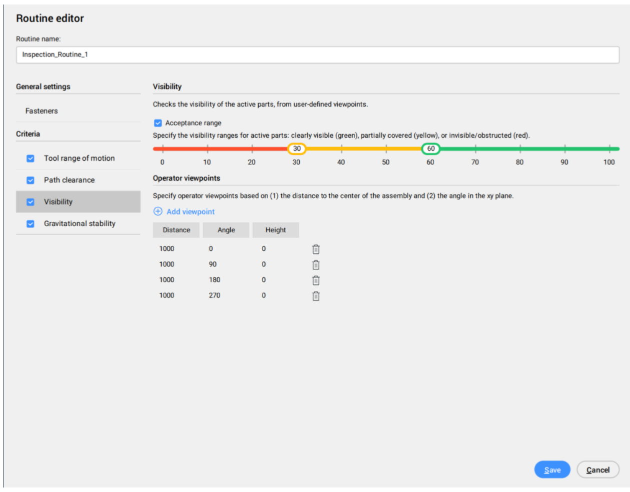


Evaluates whether each part or fastener can be easily added and removed in the current step without collision along the directions of the six local coordinate axes, called the X, Y, and Z "local axes".

Acceptance range

Specify the clearance ranges when a part is added or removed: the path is clear of obstacles (green), the path is partially restricted (yellow), or the path is blocked (red).

Visibility



Evaluates the visibility of the active parts from the user-defined viewpoints. The operator viewpoints are defined by the distance to the center of the assembly and the angle in the XY plane.

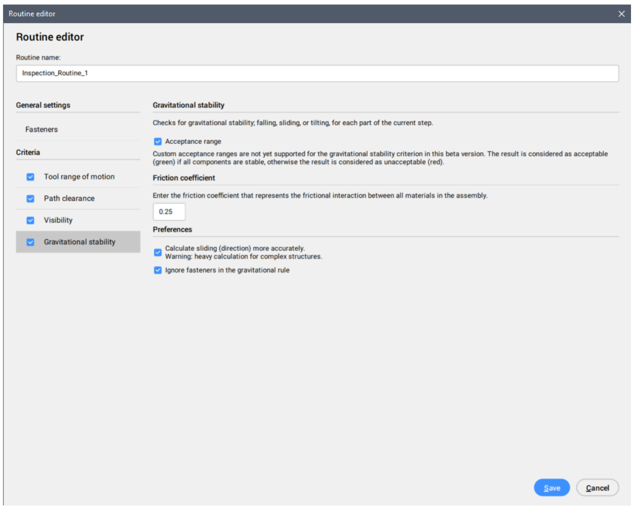
Acceptance range

Allows you to specify the visibility ranges for active parts: clear visible (green), partially covered (yellow), or invisible/obstructed (red).

Add viewpoint

Adds viewpoint for visibility check.

Gravitational stability



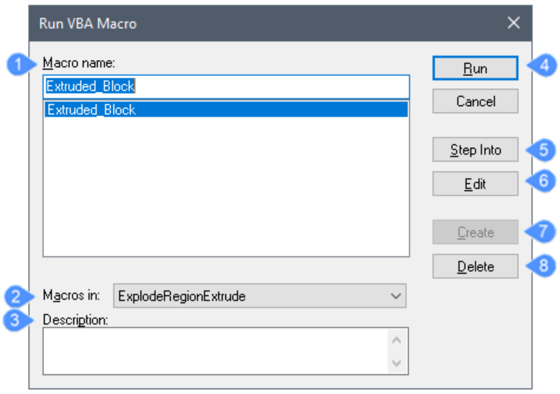
Evaluates the gravitational stability for each part of the current step in three ways: falling, sliding, and tilting. If parts are falling, sliding and tilting is not considered anymore. Partial results for the three checks are shown, as well as temporary arrows in model space. All parts of a step are considered separately, both in the calculation and in the presentation of the results.

34.18.8 Run VBA Macro dialog box

Opens via:

- VBARUN command
- **VBA Manager** dialog box

The **Run VBA Macro** dialog box runs, creates, edits, and deletes VBA macros.



1 Macro name

2 Macros in

3 Description

4 Run

5 Step Into

6 Edit

7 Create

8 Delete

34.18.8.1 Macro name

Lists the names of macros found in the project file. Choose one to run.

34.18.8.2 Macros in

Lists the names of projects loaded into the current drawing. Choose one from the drop-down list.

34.18.8.3 Description

Describes the macro. You can edit this text and save it pressing the **Close** button.

34.18.8.4 Run

Runs the selected macro.

34.18.8.5 Step Into

Opens the integrated development environment window. The macro is in debug mode, which means that it runs one line of code at a time.

See the VBAIDE command.

34.18.8.6 Edit

Opens the macro in the integrated development environment window. The macro is in editing mode, which allows you to change the code.

See the VBAIDE command.

34.18.8.7 Create

Creates new macros; follow these steps:

1. In the **Macro Name** field, enter a name for the new macro.
2. Click **Create**.
3. Notice the integrated development environment window. Enter the VBA code.

See the VBAIDE command.

34.18.8.8 Delete

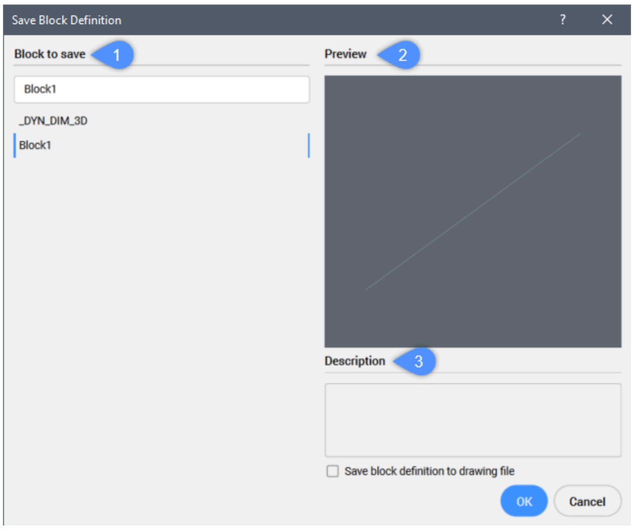
Deletes the macro. A dialog asks to confirm the action.

34.19 S

34.19.1 Save Block Definition dialog box

Opens via: BSAVEAS command

The **Save Block Definition** dialog box allows you to copy the current block definition to a new name. Opens via the BSAVEAS command.



1 Block to save

2 Preview

3 Description

34.19.1.1 Block to save

Specifies a new name for a selected block to make a copy of.

34.19.1.2 Preview

Displays a preview of the selected block.

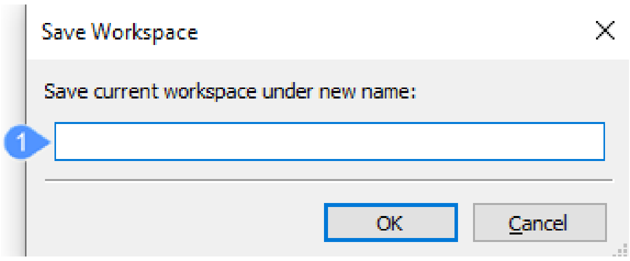
34.19.1.3 Description

Displays the description of the selected block, if it has one.

34.19.2 Save Workspace dialog box

Opens via: WSSAVE command

The **Save Workspace** dialog box allows you to save the current workspace under a different name, creating a new workspace.



1 Save current workspace under new name

34.19.2.1 Save current workspace under new name

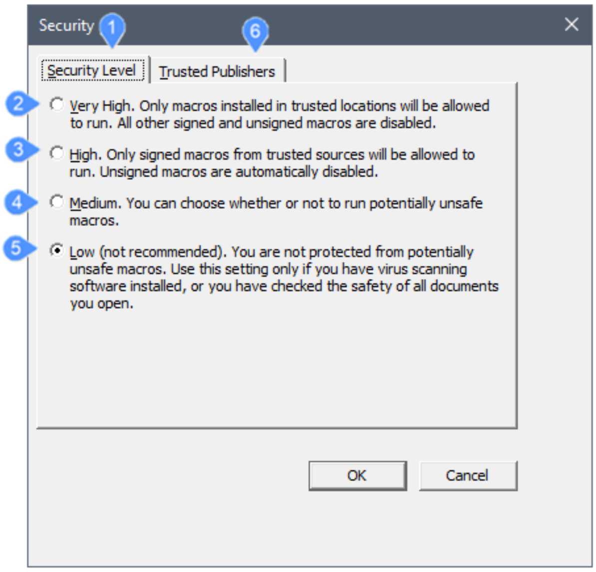
Specifies the name for the new workspace. Press **OK** to apply the changes.

You can access it later through the **Workspaces** toolbar or from the status bar.

34.19.3 Security dialog box

Opens via: SECURITY command

The **Security** dialog box sets the security level to Low, Medium, High or Very High to determine whether VBA macros are allowed to run automatically.



1 Security Level

2 Very High

3 High

4 Medium

5 Low

6 Trusted Publishers

34.19.3.1 Very High

Only VBA macros installed in trusted locations will be allowed to run. All other signed or unsigned macros are disabled.

34.19.3.2 High

Only signed macros from trusted sources will be allowed to run. Unsigned macros are automatically disabled.

34.19.3.3 Medium

BricsCAD displays a dialog box when a VBA macro runs.

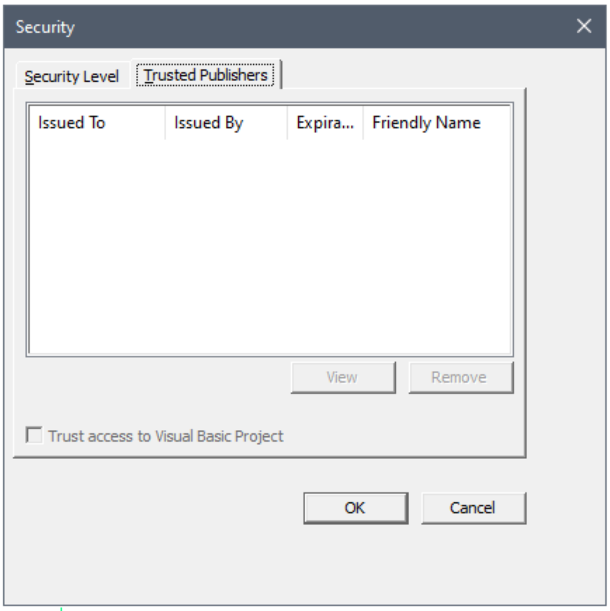
The dialog box allows you to choose whether or not to run the macro.

34.19.3.4 Low

All VBA macros are run.

34.19.3.5 Trusted Publishers

Lists the names of VBA macros that can be trusted. These are the only macros that will run when security is set to High.



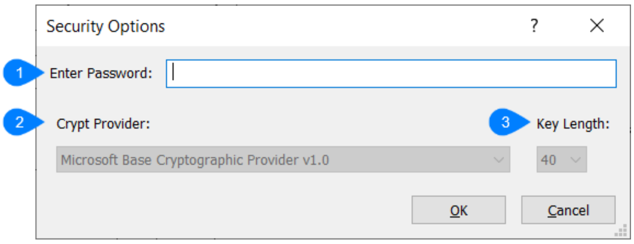
Remove
Removes the selected VBA macro from the list.

Trust access to Visual Basic Project
Allows access to the VBA Project.

34.19.4 Security Options dialog box

Opens via: SECURITYOPTIONS command

The **Security Options** dialog box allows you to set a password to protect your drawing from unauthorized access.



1 Enter Password

2 Crypt Provider

3 Key Length

34.19.4.1 Enter Password

Specify a password by entering one or more characters.

34.19.4.2 Crypt Provider

Specifies the encryption level for drawing properties. Choose an encryption provider supplied by the operating system.

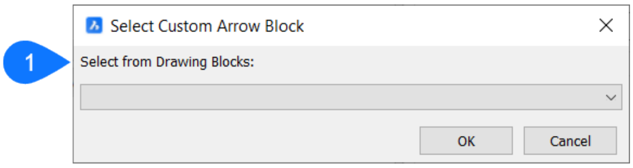
34.19.4.3 Key Length

Select a length for the encryption key. The longer the key, the higher the level of protection is, because it takes longer to break the encrypted password.

34.19.5 Select Custom Arrow Block dialog box

Opens via: **Qleader Settings** dialog box > **Select arrow** option

The **Select Custom Arrow Block** dialog box allows you to choose a block to define the shape of the arrowhead.



1 Select from Drawing Blocks

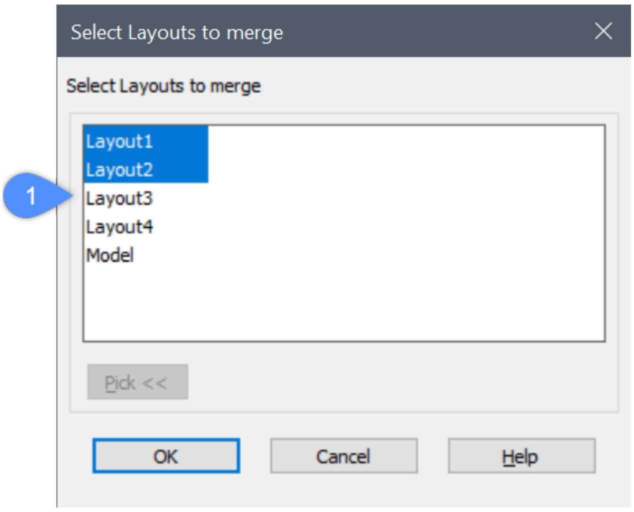
34.19.5.1 Select from Drawing Blocks

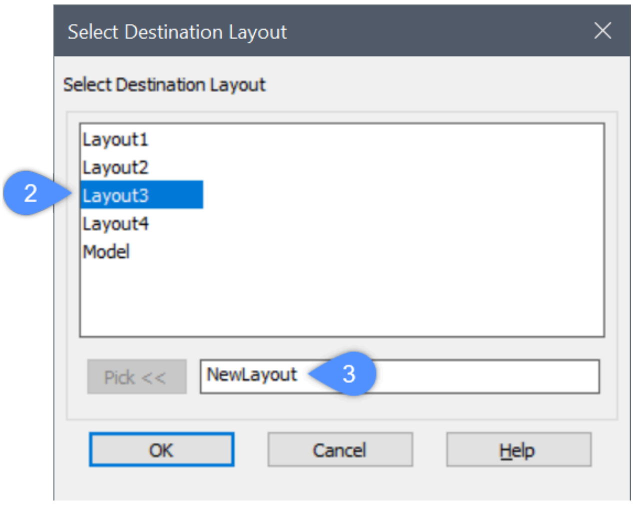
Specifies one of the blocks in the drawing to use as arrowhead.

34.19.6 Select Layouts to merge dialog box

Opens via: LAYOUTMERGE command (Express Tools)

The **Select Layouts to merge** dialog box allows you to merge entities from specified layouts into a destination layout.





1 Layouts to merge

2 Destination layout

3 Type a layout name

34.19.6.1 Layouts to merge

Allows you to choose one or more layouts to be merged.

34.19.6.2 Destination layout

Allows you select a destination layout from the list.

34.19.6.3 Type a layout name

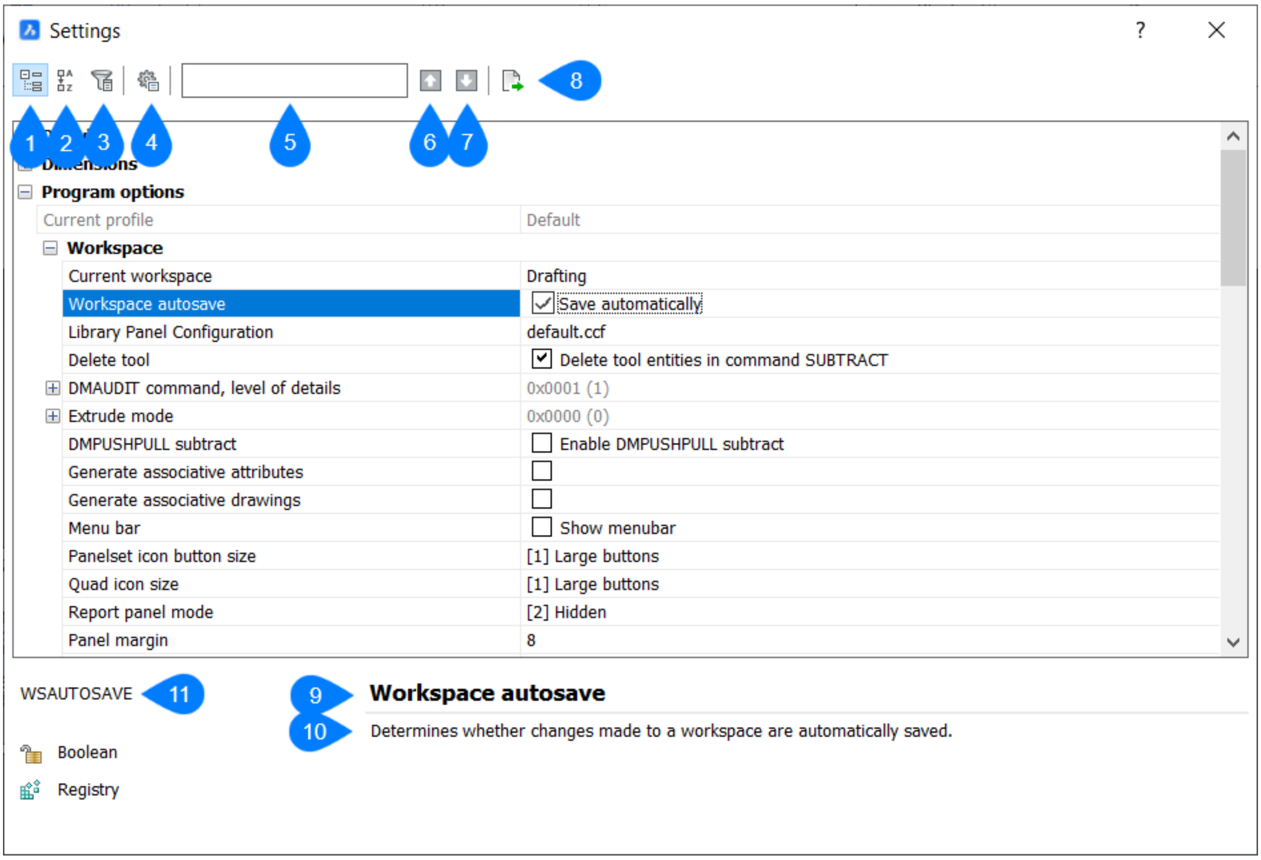
Allows you to type a layout name. If the layout does not exist, the **Create layout?** dialog box opens which asks you if you want to create the layout or not.

34.19.7 Settings dialog box

Opens via:

- SETTINGS command
- SETTINGSSEARCH command

The **Settings** dialog box allows you to change the values of system variables and user preferences.



1 Categorized

2 Alphabetic

3 Show differences

4 Dialog configurations

5 Search bar

6 Next

7 Previous

8 Export

9 Name

10 Description

11 Properties

34.19.7.1 Categorized

Lists variables in related groups.

34.19.7.2 Alphabetic

Lists the variables alphabetically.

34.19.7.3 Show differences

Lists variables whose values are changed from the initial (default) values.

34.19.7.4 Dialog configurations

Searches for variables, configures how the search is carried out, and specifies how differences are handled. The **Configure Settings** dialog box opens when clicking the icon.

34.19.7.5 Search bar

Lists variables through a real-time search field when entering the first few letters of their names or descriptions.

34.19.7.6 Next

Jumps to the next candidate that matches the search phrase.

34.19.7.7 Previous

Jumps to the previous candidate that matches the search phrase.

34.19.7.8 Export

Opens the **Export Settings** dialog box to allow you to save all settings and values to a CSV file.

34.19.7.9 Name

Displays the name of the selected system variable.

34.19.7.1 Description
0

Displays a brief description of the selected system variable.

34.19.7.1 Properties
1

Lists several basic properties/characteristics of the selected system variable.

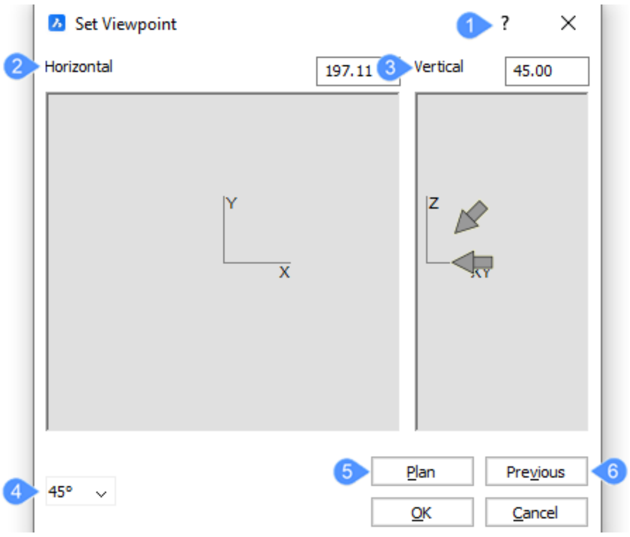
34.19.8 Set Viewpoint dialog box

Opens via:

- DDVPOINT command
- VPOINT command

The **Set Viewpoint** dialog box allows you to set the current 3D viewpoint or plan view.

Note: Turn off the perspective mode (view PERSPECTIVE command), because this command is not available in the Perspective mode.



1 Command reference

2 Horizontal

3 Vertical

4 Angle Preset

5 Plan

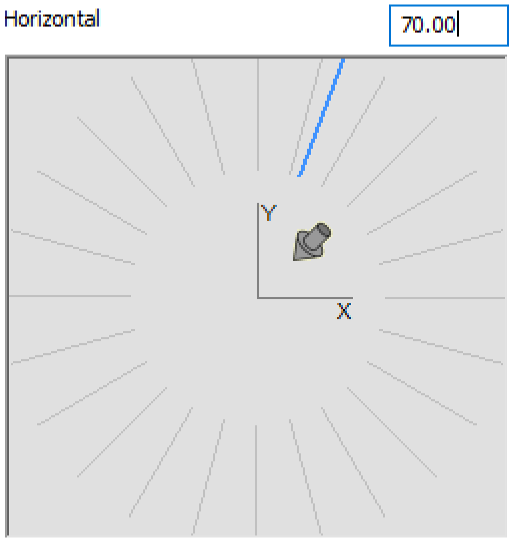
6 Previous

34.19.8.1 Command reference

Opens the Bricsys Help article about the DDVPOINT command.

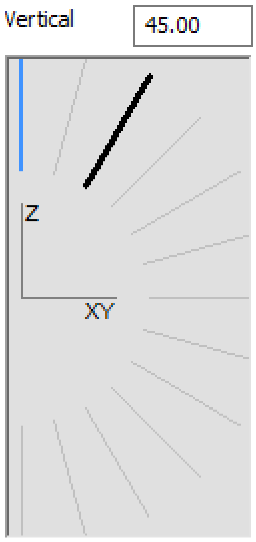
34.19.8.2 Horizontal

Specifies the horizontal angle from the X - axis. Click a preset angle or type a value in the **Horizontal** field. Click at 0°, 45°, 90°, 135°, 180°, 225°, 270° or 315° in the inner circle. The shape of the arrow indicates whether the view is horizontal or at an angle .

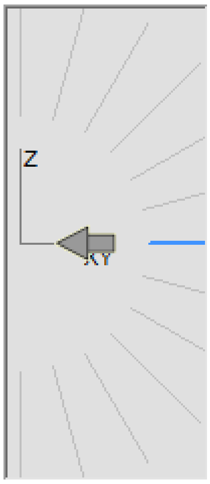


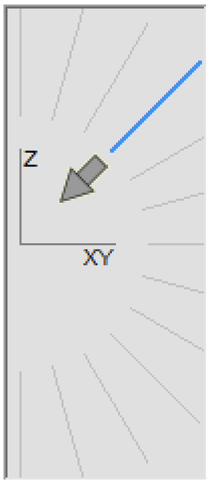
34.19.8.3 Vertical

Specifies the vertical angle from the XY plane. Click a preset angle, or else type a value in the Vertical field.

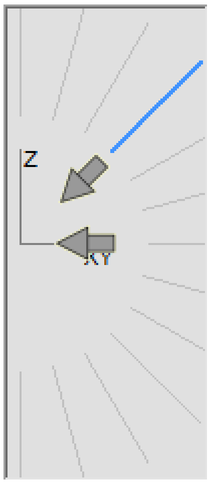


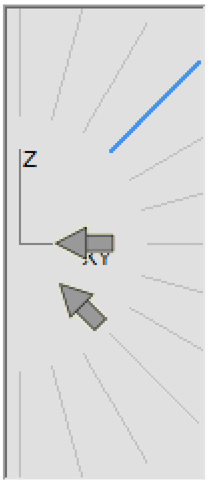
Click in the inner half circle. (1) View direction is horizontal. (2) View direction is 45° down (not available if PresetAngle = 45°).

(1) 

(2) 

View direction is 45° down, except for 0°, 90°, 180° and 270° horizontal angles, which generate orthographic views: front, back, left and right. (3) First select the horizontal arrow, then (4) click the 45° down arrow.

(3) 

(4) 

34.19.8.4 Angle Preset

Sets markers at 5, 15, or 45 degrees; default = 45.

34.19.8.5 Plan

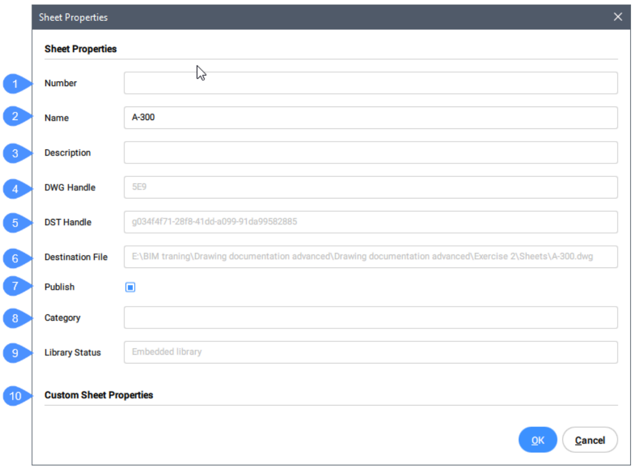
Reverts the 3D viewpoint to 2D plan view.

34.19.8.6 Previous

Reverts to the previous viewpoint.

34.19.9 Sheet Properties dialog box

The **Sheet Properties** dialog box allows you to view and edit properties of a sheet.



1 Number

2 Name

3 Description

4 DWG Handle

5 DST Handle

6 Destination File

7 Publish

8 Category

9 Library Status

10 Custom Sheet properties

34.19.9.1 Number

Specifies the number of the sheet.

34.19.9.2 Name

Specifies the name of the sheet.

34.19.9.3 Description

Specifies a short description of the sheet.

34.19.9.4 DWG Handle

Displays the DWG file handle of the layout associated with the sheet. It is read-only.

34.19.9.5 DST Handle

Displays the DST file handle of the sheet. It is read-only.

34.19.9.6 Destination File

Specifies the DWG file in which the sheet will be saved.

34.19.9.7 Publish

Toggles the option to publish the sheet.

34.19.9.8 Category

Specifies the category of the sheet.

34.19.9.9 Library Status

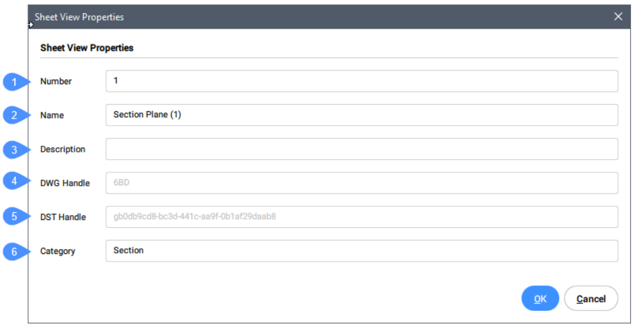
Specifies the library status.

34.19.9.1 Custom Sheet Properties 0

Lists and allows to edit the custom sheet properties, if any exists.

34.19.10 Sheet View Properties dialog box

The **Sheet View Properties** dialog box allows you to view properties of a sheet view.



1 Number

2 Name

3 Description

4 DWG Handle

5 DST Handle

6 Category

34.19.10. Number 1

Specifies the number of the sheet view.

34.19.10. Name 2

Specifies the name of the sheet view.

34.19.10. Description 3

Specifies a short description of the sheet view.

34.19.10. DWG Handle
4
Displays the .dwg file handle of the named view associated with the sheet view. It is read-only.

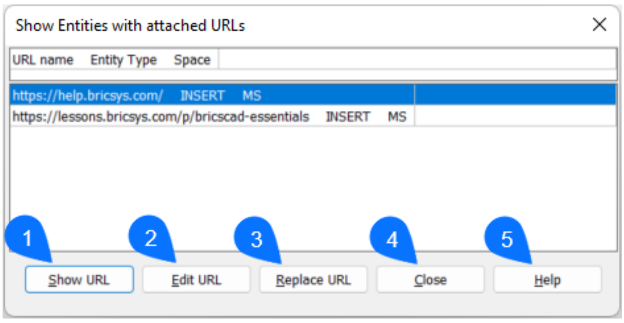
34.19.10. DST Handle
5
Displays the .dst file handle of the sheet view. It is read-only.

34.19.10. Category
6
Specifies the category of the sheet view.

34.19.11 Show Entities with attached URLs dialog box

Opens via: SHOWURLS command (Express Tools)

The **Show Entities with attached URLs** dialog box shows the attached URLs in the drawing.



1 Show URL

2 Edit URL

3 Replace URL

4 Close

5 Help

34.19.11. Show URL
1
Hides the dialog box and highlights the objects linked with the selected URL.

34.19.11. Edit
2
Allows you to edit the selected URL.

34.19.11. Replace
3
Allows you to replace the a URL by prompting the old one that has to be replaced and asking for a new one.

34.19.11. Close
4

Closes the dialog box.

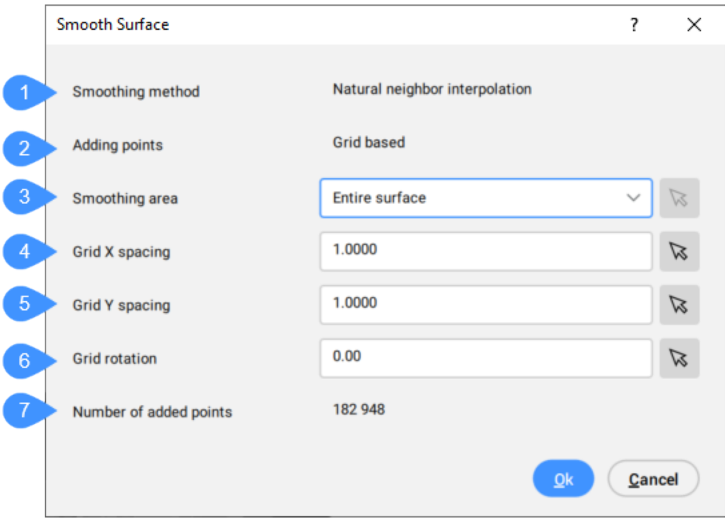
34.19.11. Help
5

Redirects you to the BricsCAD Help Center.

34.19.12 Smooth Surface dialog box

Opens via: TINEDIT command

The **Smooth Surface** dialog box allows you to smooth a TIN surface by adding additional points.



1 Smoothing method

2 Adding points

3 Smoothing area

4 Grid X spacing

5 Grid Y spacing

6 Grid rotation

7 Number of added points

Smoothing method

Displays the method for smoothing the TIN surface.
The **Natural neighbor interpolation** algorithm finds the closest subset of input samples to a query point and applies weights to them based on proportionate areas to interpolate a value.

Adding points

Displays the method for adding points to the TIN surface.

Smoothing area

Defines the area for smoothing. You can choose to smooth the **Entire surface** or a **Selected area**. To select the area, click the right side arrow to define the area in the Model space.

Grid X spacing
Sets the grid X spacing value. You can also define it by clicking the distance in the Model space.

Grid Y spacing
Sets the grid Y spacing value. You can also define it by clicking the distance in the Model space.

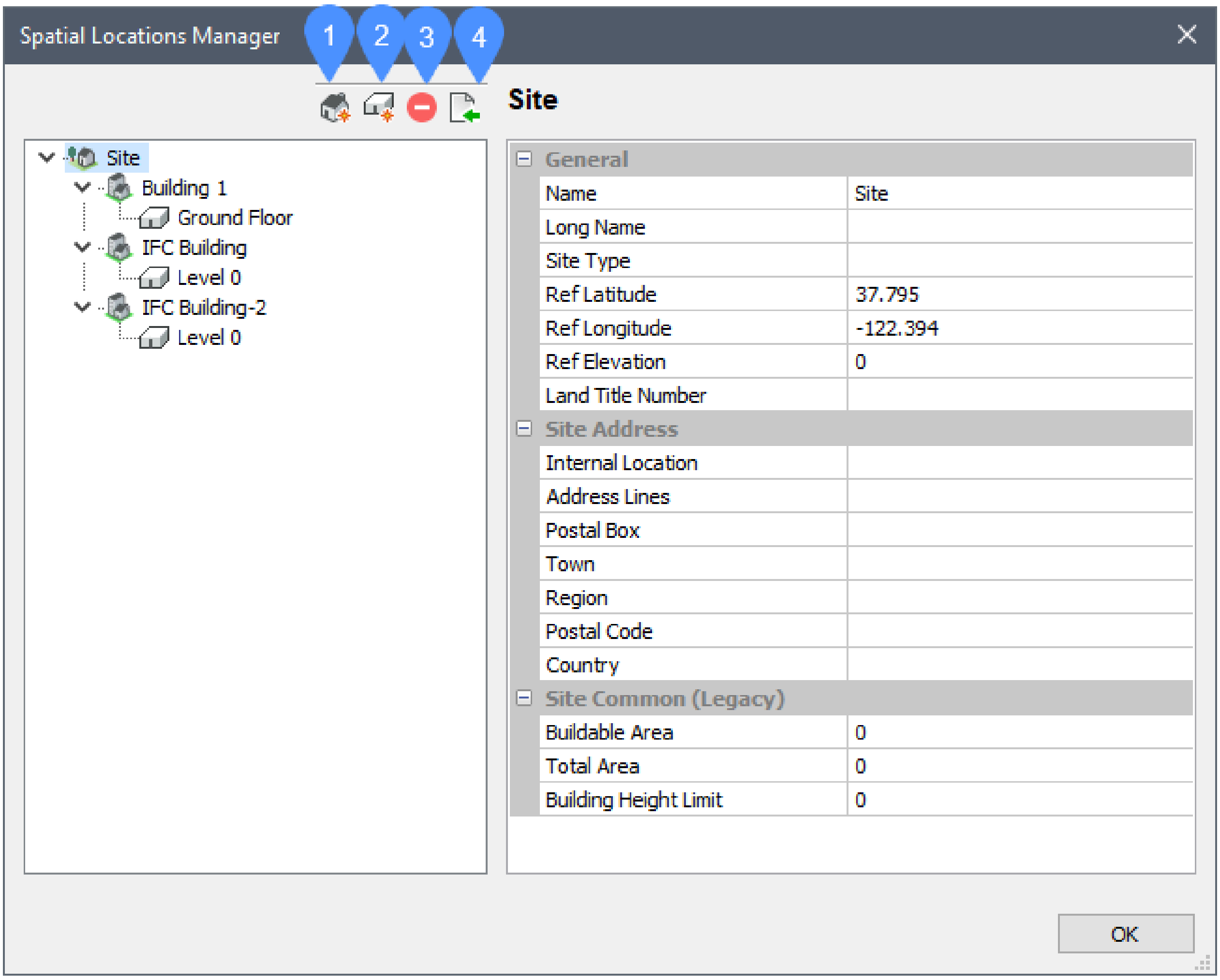
Grid rotation
Sets the rotation value. You can also define it by clicking the desired angle in the Model space.

Number of added points
Displays the number of points that will be added to the TIN surface.

34.19.13 Spatial Locations Manager dialog box

Opens via: BIMSPATIALLOCATIONS command

The **Spatial Locations Manager** dialog box allows you to create and edit sites, buildings and stories.



1 New Building

2 New Story

3 Delete selected Building or Story

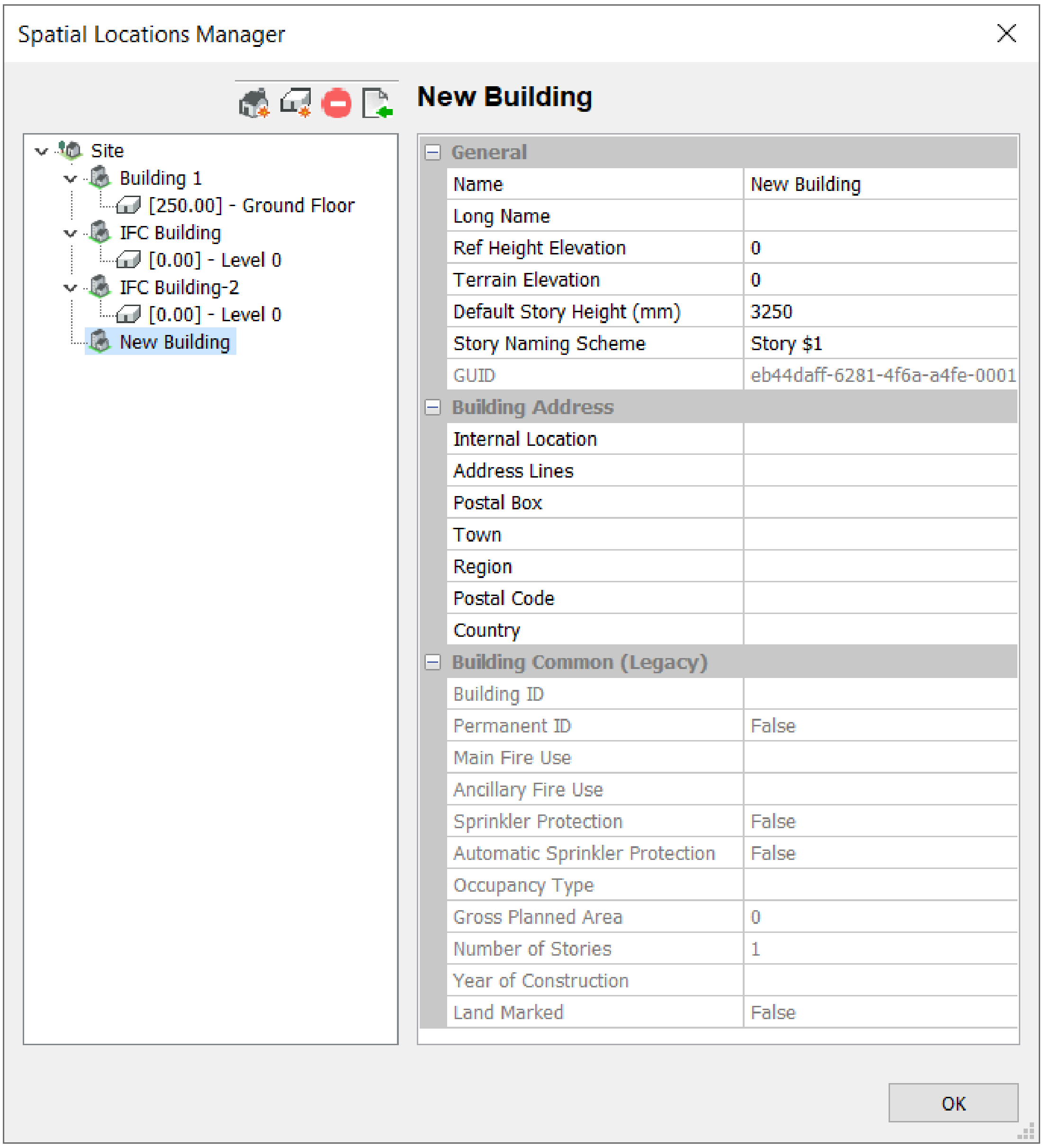
4 Import spatial locations

34.19.13. New Building

1

Adds a new building to the model.

Fill out the properties grid:



Note:

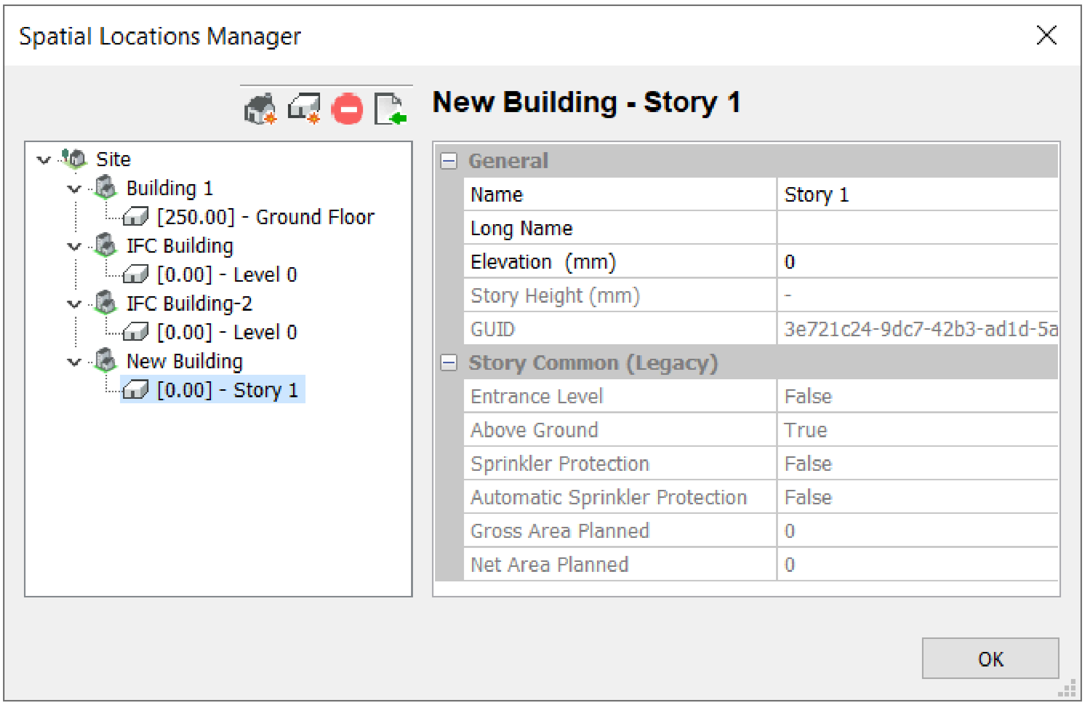
- When creating new buildings, the DEFAULTSTORYNAMINGSCHEME system variable defines the naming and numbering of the spatial elements. You can access this info in the **Spatial Locations Manager** dialog box.

- When adding new stories to an existing building, the **Story Naming Scheme** property of the building overrides the value of the DEFAULTSTORYNAMINGSCHEME system variable.
- With the **Story Bar** open (STORYBAR system variable is set to 1 or 2), you can easily switch between the defined floors in **Top View Mode** to start creating walls.

34.19.13. New Story 2

Adds a new story in the selected building.

The properties grid is completed.



34.19.13. Delete selected Building or Story 3

Deletes the selected building and story.

34.19.13. Import spatial location 4

Imports spatial locations from a text file (.txt).

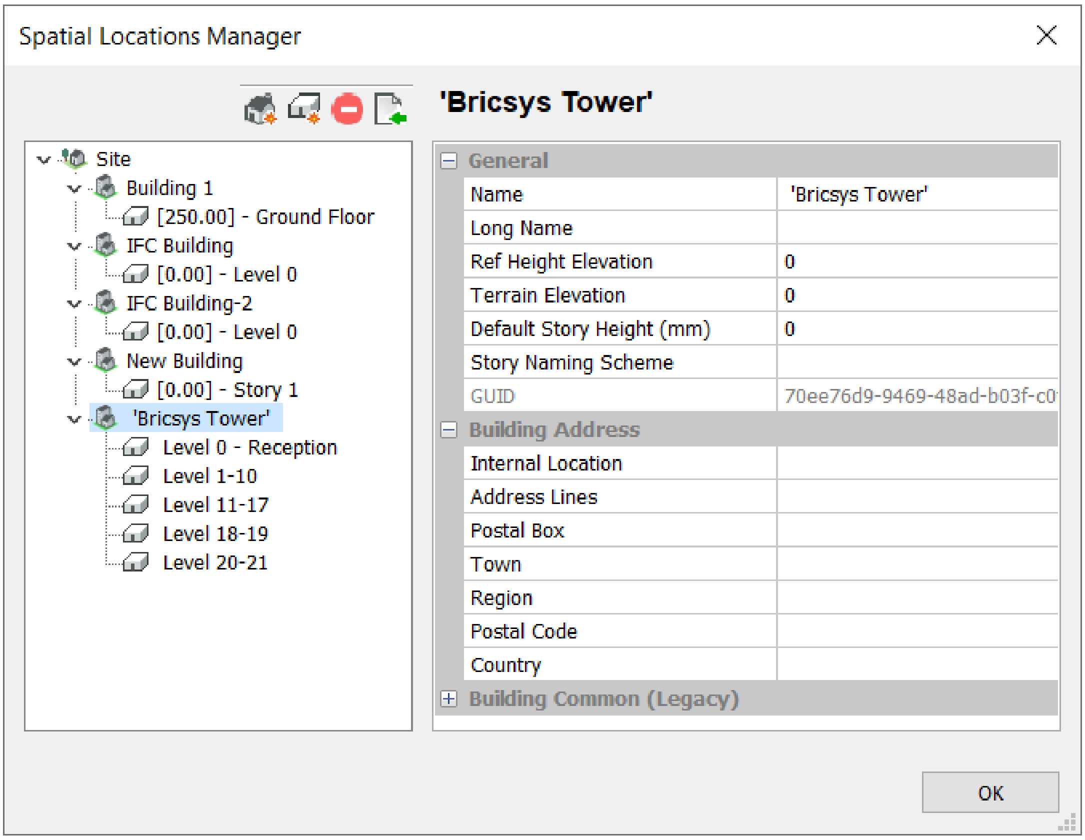
Displays a **Standard Open Files** dialog box from which will open the *.txt spatial location file.

Example of a *.txt spatial location file:

```
Space:Name= 'Bricsys Tower' ,Area=15000 Space:Name= Level 0 - Reception ,Area=6%,
 ' Floor-to-Floor Height '=5000, Count=1 Space:Name= Level 1-10 ,Area=4%, ' Floor-
to-Floor Height '=4000, Count=10 Space:Name= Level 11-17 ,Area=7%, ' Floor-to-
Floor Height '=3000, Count=7 Space:Name= Level 18-19 ,Area=5%, ' Floor-to-Floor
 Height '=4000, Count=2 Space:Name= Level 20-21 ,Area=1.5%, ' Floor-to-Floor
 Height '=4000, Count=2
```

The area and area ratios that are specified in the file are requirements.

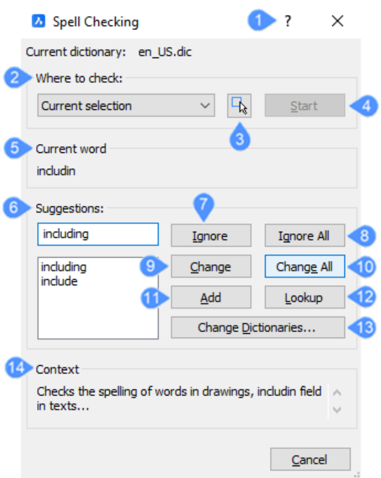
The spatial location file will be loaded as below:



34.19.14 Spell Checking dialog box

Opens via: SPELL command

The **Spell Checking** dialog box allows you to check the spelling of words in drawings, including fields in texts, multiline texts, leader, multileader, table and block attributes.



1 Command reference

2 Where to check

3 Select entities

4 Start

5 Current word

6 Suggestions

7 Ignore

8 Ignore All

9 Change

10 Change All

11 Add

12 Lookup

13 Change Dictionaries...

14 Context

34.19.14. Command reference

1

Opens the Bricsys Help article about the SPELL command.

34.19.14. Where to check

2

Checks text in:

- **Entire drawing**: Checks all text in the drawing.
- **Selected entities**: Checks text in selected entities. Click the **Select Entities** button to choose text.

34.19.14. Select entities

3

Select the text to be checked. Choose one or more pieces of text and press Enter to return to the dialog box.

34.19.14. Start

4

Starts checking the spelling of words.

34.19.14. Current word

5

Displays the misspelled word.

34.19.14. Suggestions

6

Lists the suggested replacements for misspelled words. Choose a replacement word and click **Change** or **Change All**.

34.19.14. Ignore
7

Ignores the spelling of the word.

34.19.14. Ignore All
8

Ignores all instances of the word.

34.19.14. Change
9

Changes the word to the one selected from the **Suggestion** list.

34.19.14. Change All
10

Changes all instances of the word to the one selected from the **Suggestion** list.

34.19.14. Add
11

Adds the word to the word list.

34.19.14. Lookup
12

Provides additional alternative spellings for the word selected in the **Suggestions** list.

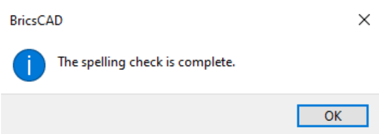
34.19.14. Change Dictionaries...
13

Changes the dictionary used for spell checking. Displays the **Change Dictionaries** dialog box. For more information, see the article **Change Dictionaries dialog box**.

34.19.14. Context
14

Displays the context in which the misspelled word appears.

When the spelling check is completed, the following message box appears:

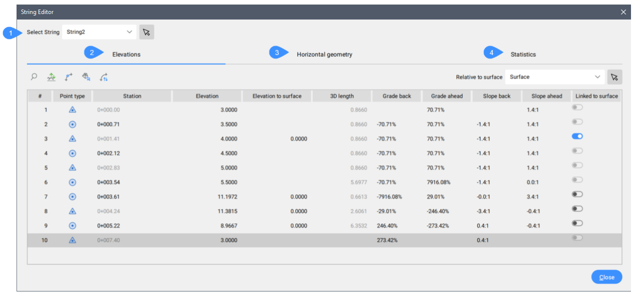


34.19.15 String Editor dialog box

Opens via:

- STRINGEDITOR command
- **Civil Explorer** panel > context menu of a **String** entity definition (**Definition** tab)

The **String Editor** dialog box allows you to edit the Civil String entities.



1 Select String

2 Elevations tab

3 Horizontal geometry tab

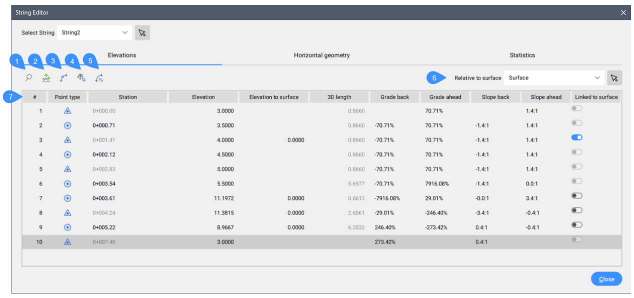
4 Statistics tab

34.19.15. Select String
1

Displays the Civil String for editing. You can select another Civil String from the drop-down list or click to pick it from the drawing area.

34.19.15. Elevations tab
2

Lists all points properties of the Civil String for editing.



1 Zoom

2 Raise/Lower

3 Add elevation point

4 Get elevation from surface

5 Reverse string

6 Relative to surface

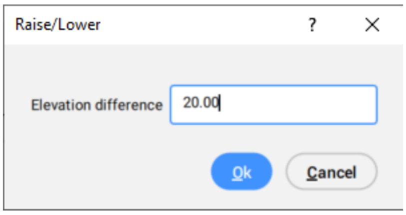
7 String points properties list

Zoom

Zooms in the drawing to the selected Civil String.

Raise / Lower

Opens the **Raise/Lower** dialog to set the elevation difference value. This value will be added to the elevations. Negative values decrease the elevation.



Add elevation point

Launches the STRINGADDELEVATIONPOINT command. Adds an elevation point to a Civil String entity.

Get elevation from surface

Opens the **Elevations From Surface** dialog box.

Reverse string

Changes the direction of the Civil String.

Relative to surface

Displays the selected TIN surface to which a Civil String elevations will be relative. You can select another TIN surface from the drop-down list or click to pick it from the drawing area.

String points properties list

Lists all String points properties.

Note: Right-click a string point properties line to open a context menu with **Zoom to**, **Pan to**, **Copy to clipboard**, and, depending on the point type, the **Delete PI** ( ) or **Delete elevation point** ( ) options.

Position number

Lists the position number of the point in the String.

Point type

Displays the type of point represented by different icons.

- Points intersection (PI)
- Point intersection (PI) read only
- Point intersection (PI) shared
- Elevation Point

Station

Displays the Station value of the point.

Note: The station value for the point intersections (PI) is read-only.

Elevation

Displays the elevation value of the point.

Elevation to surface

Displays the elevation difference between the surface and String point.

3D length
Displays the 3D length of the segment between the current point and the next point.

Grade back
Displays the grade of the current segment from its end to start.

Grade ahead
Defines the end elevation of the current segment in forward direction.

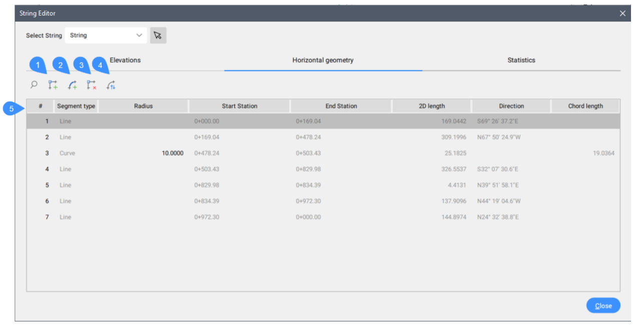
Slope back
Displays the slope of the current segment from its end to start.

Slope ahead
Defines the end elevation of the current segment in forward direction.

Link to surface
If ticked, the String is linked to TIN surface. Civil String elevations change dynamically with TIN surface elevations changes.

34.19.15. Horizontal geometry tab

3
Lists all segments properties of the Civil String for editing.



1 Add PI

2 Add curve

3 Delete PI

4 Reverse string

5 String segments properties

Add PI
Launches the STRINGADDPI command.

Add curve
Launches the STRINGADDCURVE command.

Delete PI
Launches the STRINGDELETEPI command.

Reverse string
Changes the direction of the Civil String.

String segments properties
Lists all String segments properties.

Position number
Lists the position number of the segment in the String.

Segment type
Displays the segment type.

Radius
Displays the radius value.

Start Station
Displays the start station of the segment.

End Station
Displays the end station of the segment.

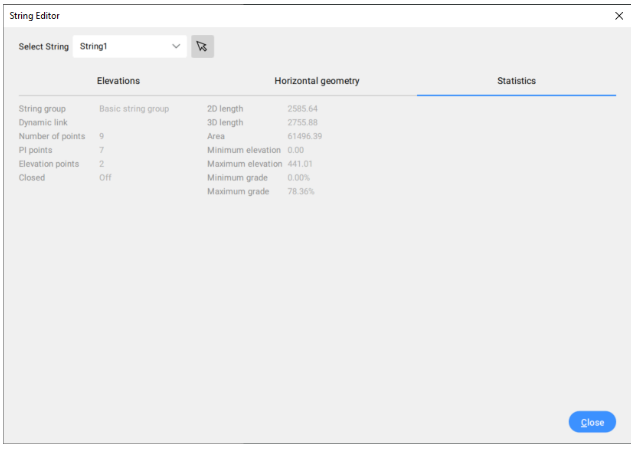
2D length
Displays the 2D length of the segment.

Direction
Displays the direction of the segment.

Chord length
Displays the chord length.

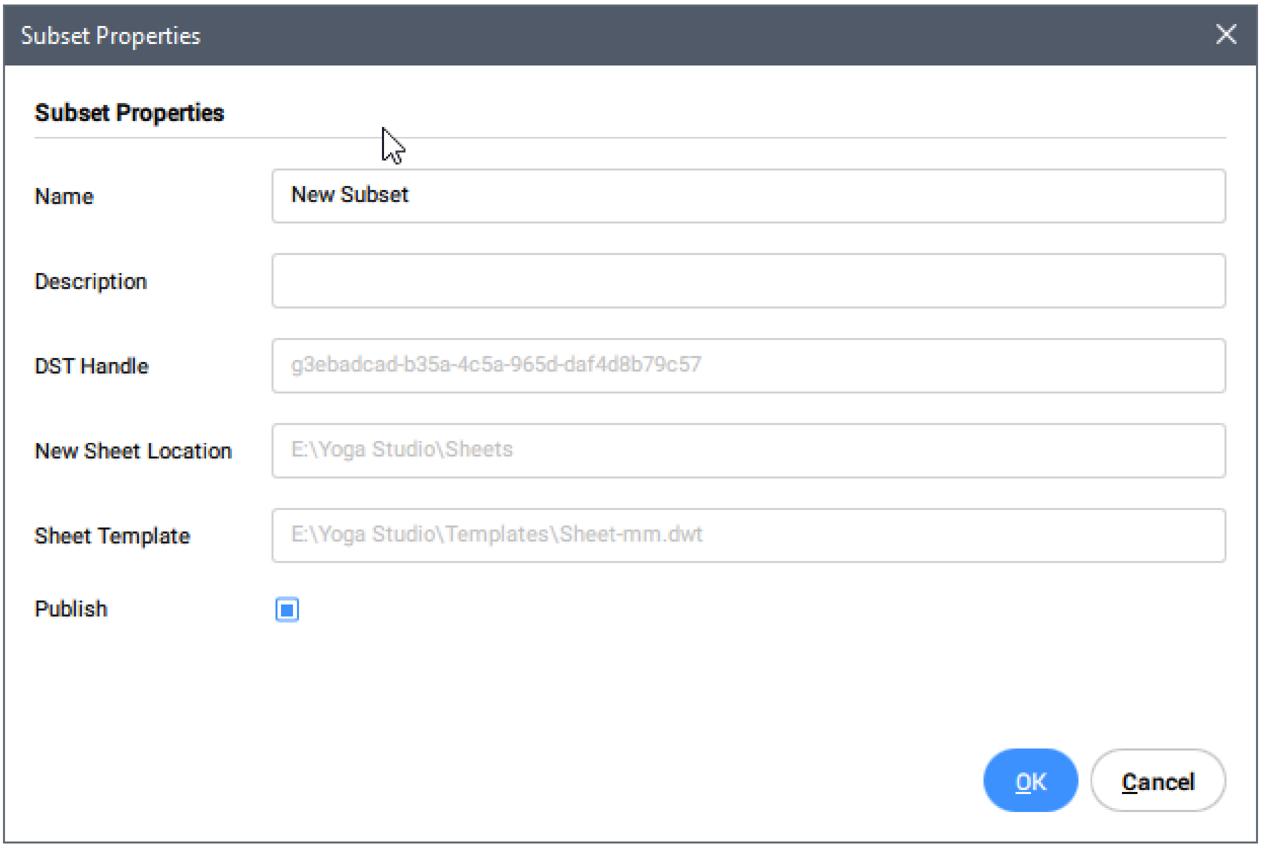
34.19.15. Statistics tab
4

Lists the general properties of the Civil String.



34.19.16 Subset Properties dialog box

Allows to edit the subset properties.



Name: names the subset.

Description: describes the subset.

DST Handle: displays the .dst file handle of the subset. It is read-only.

New Sheet Location: sets a sheet location. (click on the three points located in the right side to open **Standard File** dialog box to select the sheet location)

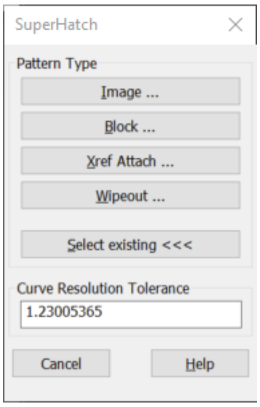
Sheet template: sets a sheet template. (click on the three points located in the right side to open **Standard File** dialog box to select the sheet template)

Publish: toggles the publishing of the subset.

34.19.17 SuperHatch dialog box

Opens via: SUPERHATCH command (Express Tools)

The **SuperHatch** dialog box allows you to hatch entities using a selected image, block, external reference or wipeout entity.



1 Image

2 Block

3 Xref Attach

4 Wipeout

5 Select existing

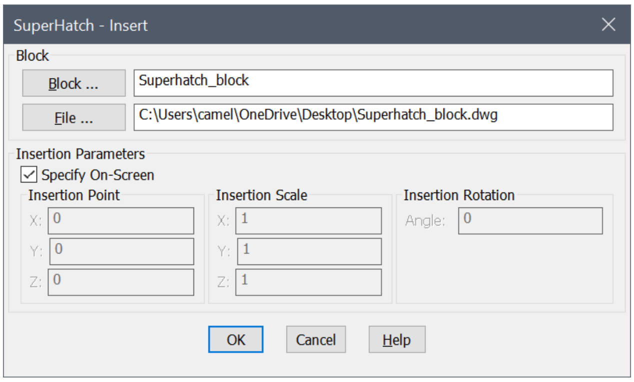
6 Curve Resolution Tolerance

34.19.17. Image
1

Opens the **Select Image File** dialog box to specify an image as superhatch pattern.

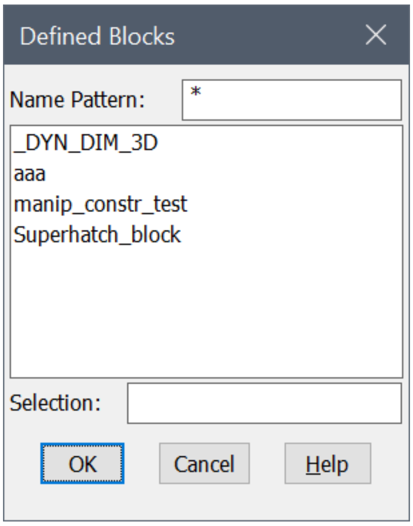
34.19.17. Block
2

Opens the **SuperHatch - Insert** dialog box that allows you to select a block reference as superhatch pattern.



Block...

Opens the **Defined Blocks** dialog box that lists all the blocks inserted in the current drawing.



File...
Opens the **Select Drawing File** dialog box that allows you to insert a new block reference.

34.19.17. Xref Attach
3
Opens the **Select Reference File** dialog box to specify an xref file as superhatch pattern.

34.19.17. Wipeout
4
Creates a wipeout inside a superhatch pattern.

34.19.17. Select Existing
5
Allows you to select an existing image, block, external reference, or wipeout entity as superhatch pattern.

34.19.17. Curve Resolution Tolerance
6
Used in case of curved boundaries, it allows you to specify the resolution tolerance for vector approximation of curved superhatch boundaries (maximum distance between the midpoint of any given hatch segment and the curved boundary).

34.19.18 Surface Texture dialog box

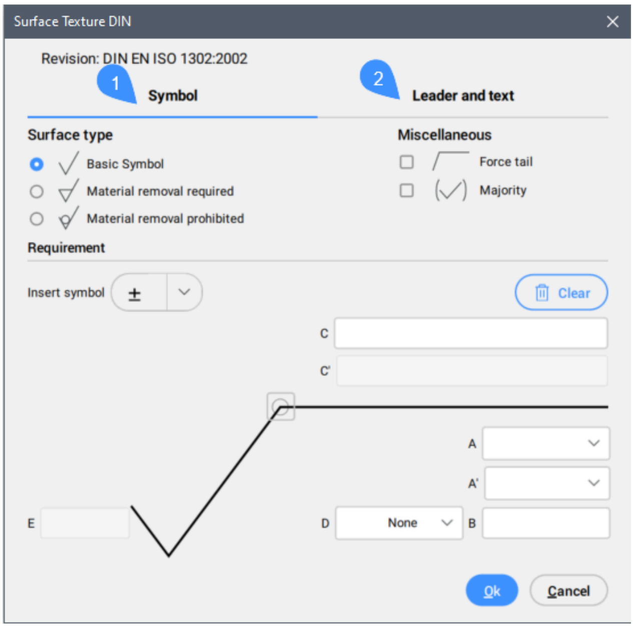
Opens via: AMSURFSYM command

The **Surface Texture** dialog box allows you to set the default surface texture symbols for the current drafting standard.

The parameters entered for the surface texture will be stored temporarily and used as default values for the next surface texture setting.

The most recent revision of the standard to which this symbol conforms is displayed at the top of the dialog box.

The dialog box contains two main tabs, **Symbol** and **Leader and text**.



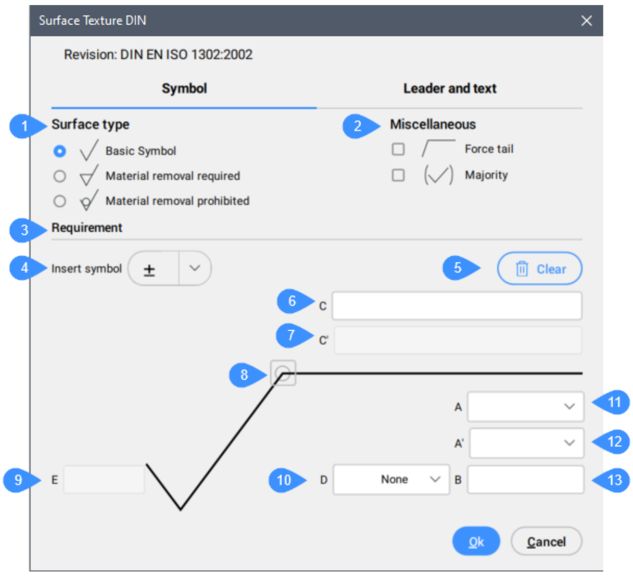
1 Symbol

2 Leader and text

34.19.18. Symbol

1

Specifies the surface texture characteristics. The available characteristics depend on the chosen drafting standard, ISO, DIN, ANSI, or JIS.



1 Surface type

2 Miscellaneous

3 Requirement

Surface type
Allows you to choose one of the following surface types: **Basic symbol**, **Material removal required**, or **Material removal prohibited**.

Miscellaneous
Indicates which surface texture symbols are available for using.

- **Force tail**: shows the state of other surface texture symbols elsewhere in the drawing in parenthesis at the right side of the collective indication.
- **Majority**: shows the collective indication (that is a symbol common to all surfaces textures). This option is not available for the ANSI drafting standard.

Requirement
Defines the parameters of the surfaces textures.

4. Insert symbol
Inserts a symbol from the drop down list into the selected parameter field.

5. Clear
Clears the last inserted parameters. The cells will become empty.

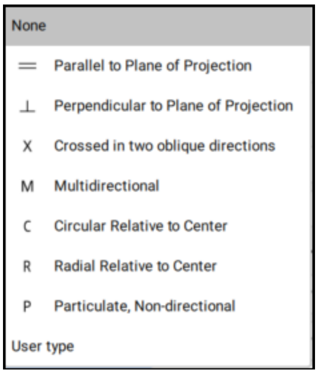
6. Process note 1
Defines the process requirements for the surface.

7. Process note 2
Defines the process requirements for the surface. This option is not available for the drafting JIS and ANSI standards.

8. All around
Toggles the visibility of the All Around symbol.

9. Machine allowance
Specifies the machining allowance for the surface.

10. Direction of lay
Sets the direction of lay for the surface.



11. First requirement
Defines the first requirement for the surface.

12. Second requirement
Defines the second requirement for the surface.

13. Third requirement
Defines the third requirement for the surface.

Some more requirements are available only for ANSI drafting standard.

Additional Requirements:

14. Roughness average max
Defines the maximum roughness average for the surface.

15. Roughness average min
Defines the minimum roughness average for the surface.

16. Cutoff
Defines the cutoff value for the surface.

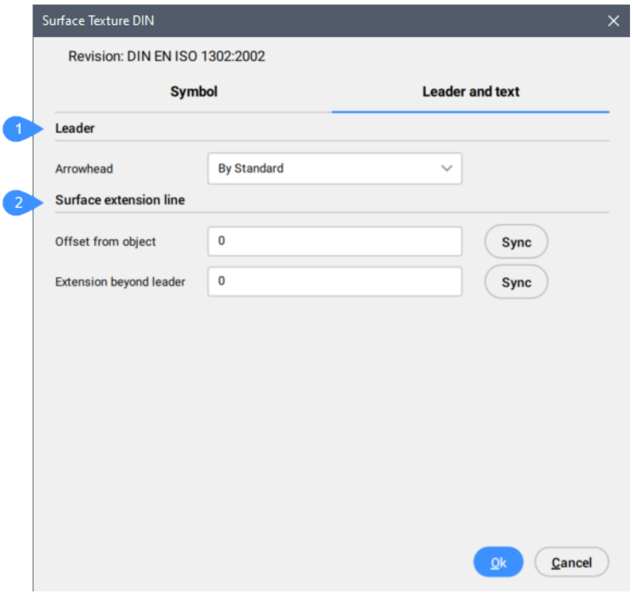
17. Sampling length
Defines the sampling length required for the surface.

18. Other roughness max
Defines the maximum other roughness for the surface.

19. Other roughness min
Defines the minimum other roughness for the surface.

34.19.18. Leader and text

2
Specifies the Leader and text characteristics.



1 Leader

2 Surface extension line

Leader
Sets the leader characteristics.

Arrowhead
Defines the default leader arrowhead type from the drop down list.

Surface extension line
Sets the surface extension line characteristics.

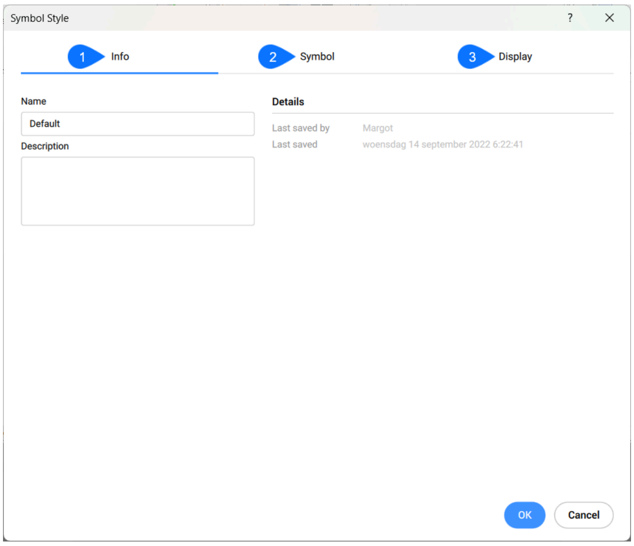
Offset from object
Defines the distance between the extension line's start point and the attached object.

Extension beyond leader
Defines the distance between the symbol's start point and the surface extension line's end point.

34.19.19 Symbol Style dialog box

Opens via: **Civil Explorer** panel > **Settings** tab

The **Symbol Style** dialog box allows you to define a new symbol style.



1 Info

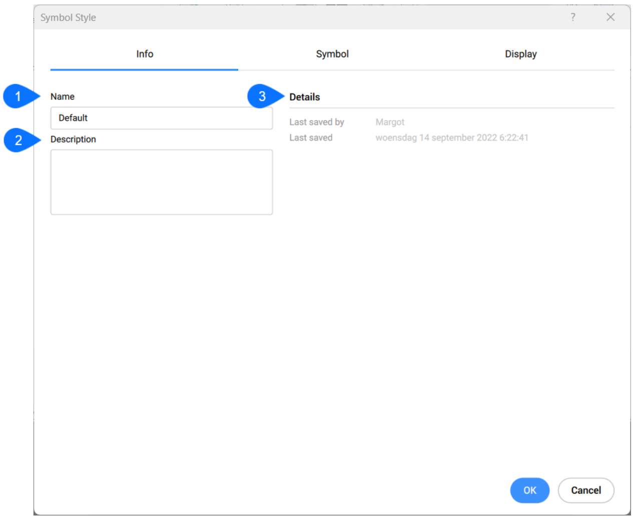
2 Symbol

3 Display

34.19.19. Info

1

Specifies general information of the selected symbol style.



1 Name

2 Description

3 Details

Name

Sets and displays of the symbol style name.

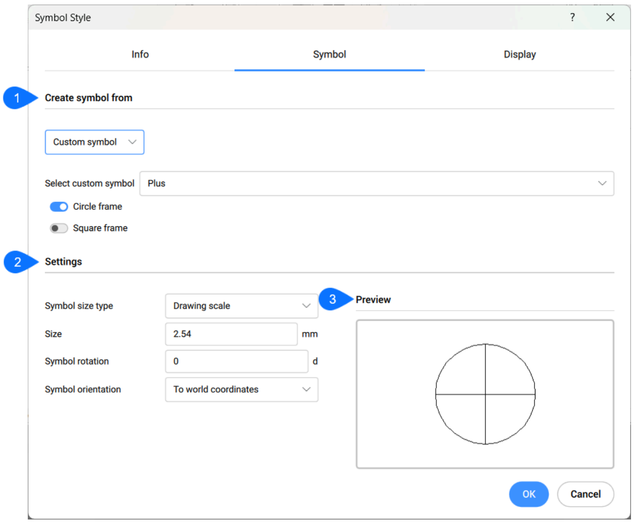
Description

Sets and displays the description of the symbol style.

Details
Displays Last saved by and Last saved date.

34.19.19. Symbol 2

The **Symbol** tab allows you to set the symbol style.



1 Create symbol from

2 Setting

3 Preview

Create symbol from
Specifies which symbol type is used to create the Symbol style from. You can select several options from the drop-down menu.

- **Point object**
  Uses a point object as basis for the Symbol style.
- **Custom symbol**
  Allows you to create a symbol style starting from one of the custom symbols below:
  - None
  - Dot
  - Plus
  - X Mark
  - Line

  Two additional options can either be toggled On or Off:

  Circle frame
  Adds a circle frame to the symbol style.

  Square frame
  Adds a square frame to the symbol style.

- **Block**
  Allows you to create a symbol style from a block in the drawing.

Settings

Allows you to specify specific characteristics depending on the selected symbol type.

Custom symbol settings

Symbol size type

Defines the size type of the symbol. Size can be set to:

- **Drawing scale**
- **Fixed scale**
- **Absolute units**
- **Relative to screen**

Depending on the selected symbol size type, there are additional settings to determine size, scale (in X,Y and Z direction), rotation or orientation.

Size

Defines the size of the symbol.

Maximum Size

Controls the maximum size of the symbol.

Scale X

Defines the fixed scale in X direction. The actual Symbol size multiplied by the value of Scale X.

Scale Y

Defines the fixed scale in Y direction. The actual Symbol size multiplied by the value of Scale Y.

Scale Z

Defines the fixed scale in Z direction. The actual Symbol size multiplied by the value of Scale Z.

Symbol Rotation

Controls the maximum size of the symbol.

Symbol Orientation

Controls the orientation of the symbol on user-defined coordinate system (based on Object, View, or To World Coordinates).

Note: The options **Symbol Size type** and **Size** are not available when starting from a Point object.

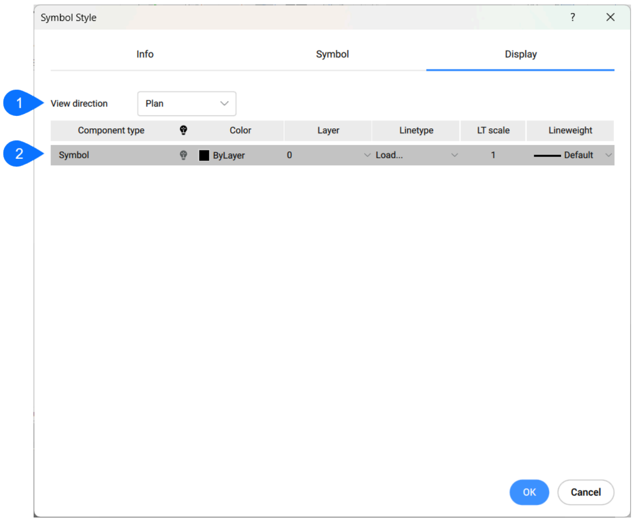
Preview

Displays a preview of the specified symbol style.

34.19.19. Display tab

3

Allows you to specify the display of all symbol components.



1 View direction

2 Component type characteristics

View direction

Specifies for which view direction the display characteristics are modified. Two options are available:

- Plan
- Model

Component type characteristics

Specifies the display characteristics of the symbol component.

Color

Specifies the color of the symbol component.

Layer

Specifies the layer of the symbol component.

Linetype

Specifies the linetype of the symbol component.

Linetype Scale

Specifies the linetype scale of the symbol component.

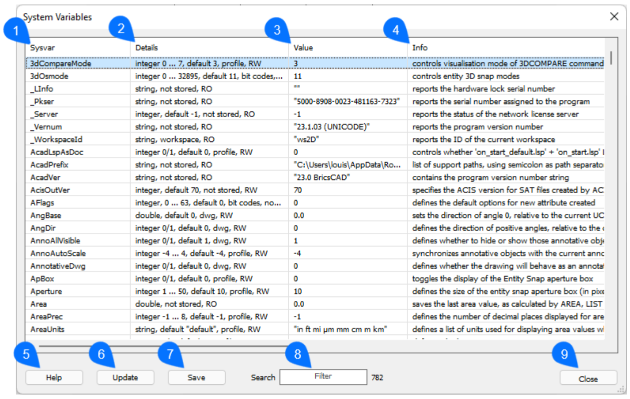
Lineweight

Specifies the lineweight of the symbol component.

34.19.20 System Variables dialog box

Opens via: SYSVDLG command (Express Tools)

The **System Variables** dialog box allows you to view, restore, edit and save the system variable settings.



1 System variables

2 Details

3 Value

4 Info

5 Help

6 Update

7 Save

8 Search

9 Close

34.19.20. System variables

1

Lists the system variables.

34.19.20. Details

2

Lists the details of the system variables. This includes: variable type, minimum and maximum value, default value...

34.19.20. Value

3

Shows the current value, double click on this value redirects you to the correct dialog box to change it. Lists the details of the system variables.

34.19.20. Info

4

Lists additional info about the system variable.

34.19.20. Help

5

Redirects you to the BricsCAD helpdesk.

34.19.20. Update
6

Refreshes the dialog box.

34.19.20. Save
7

Saves the system variable settings to a .txt file.

34.19.20. Search
8

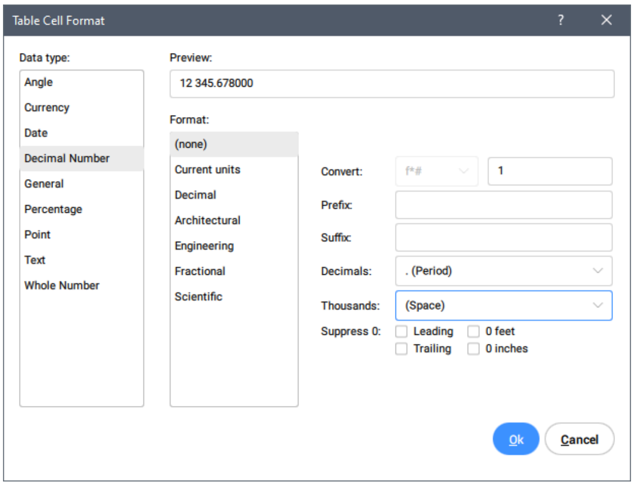
Allows you to search/filter the system variables.

34.20 T

34.20.1 Table Cell Format dialog box

Opens via: TABLEMOD command > button

Sets the cell format of the selected cell.



Data type

Sets the cell data type as:

1 Angle

2 Currency

3 Date

4 Decimal number

5 General

6 Percentage

7 Point

8 Text

9 Whole Numbers

Preview
Displays the preview of the cell according to the selected options.

Format
Sets the units for angles, numbers and points or the text format.

Convert
Sets a value for conversion. The displayed value in the cell will be multiplied with this field.

Prefix
Adds a prefix.

Suffix
Adds a suffix.

Decimals
Sets the decimal separator.

Thousands
Sets the thousands separator.

Suppress 0
Sets the behavior of leading and trailing 0 suppress.

34.20.1.1 Currency

Symbol
Defines a symbol for the currency.

Append symbol
Toggles the display of the symbol as prefix or suffix.

Negative numbers
Defines how the negative numbers will be displayed.

Precision
Sets the precision of the currency.

34.20.1.2 Date

Date
Displays a preview of the date or time.

Date format
Sets the expression of the date or time.

Examples
Lists some predefined data and time formats. You can choose one of these formats.

34.20.1.3 Point

Point
Sets the coordinate separator.

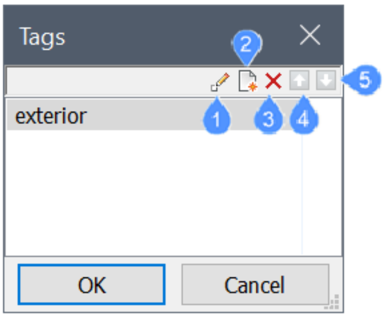
X / Y / Z

Toggles the display of each coordinate.

34.20.2 Tags dialog box

Opens via: **Compositions** dialog box

The **Tags** dialog box allows you to add or edit compositions and physical materials tags.



1 Edit

2 New

3 Delete

4 Move up

5 Move down

34.20.2.1 Edit

Edits the name of the selected tag.

34.20.2.2 New

Adds a new tag at the bottom of the list.

34.20.2.3 Delete

Deletes the selected tag.

34.20.2.4 Move up

Moves up the selected tag.

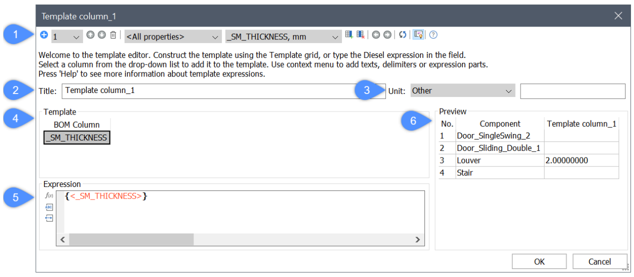
34.20.2.5 Move down

Moves down the selected tag.

34.20.3 Template column dialog box

Opens via: **BOM Manager** panel

The **Template column** dialog box allows you to configure title, expression and, if necessary, unit for your template column. It also displays a preview of the results.



1 Template column tools

2 Title

3 Unit

4 Template

5 Expression

6 Preview

34.20.3.1 Template column tools

Add template ( )

Adds a new child template.

Select

Allows to select the child template to edit.

Move up ( )

Moves the current child template one position up.

Move down ( )

Moves the current child template one position down.

Remove ( )

Removes the current child template.

Category selector (<All properties>)

Lists all categories of columns.

Column reference

Displays the available columns to use.

Add column( )

Adds the selected column to the expression.

Add column

Adds a **Column** reference.

> ***Note:*** It will be automatically added after the last selected column, or, if no column is selected, after the last column. If necessary, a **Delimiter** will be automatically added in front of the **Column** reference.

Add delimiter

Adds a **Delimiter** column.

Add text
Adds custom **Text** column.

Add expression
Adds an **Expression** column.

Delete column ()
Deletes the selected columns from the expression.

Move left ()
Moves selected columns to the left.

Move right ()
Moves selected columns to the right.

Refresh ()
Refreshes the preview.

Toggle ( )
Toggles the template grid.

34.20.3.2 Title

Displays the name of the new column.

34.20.3.3 Unit

Lists all the measurements and units.

34.20.3.4 Template

Allows you to easily insert, move and delete properties, as well as delimiters and custom texts.

Right click displays a context menu with additional options:

Set custom format...
Opens the **Format** dialog box that allows you to specify the format of text strings in the specified column.

Use original format
Restores the original format.

Set default value...
Opens the **Enter the default value** dialog box that allows you to set the default value.

Target unit
Allow to set the target unit for variables, if the variable supports a unit conversion.
It allows to write unit-dependent formula (such as density computation) without bothering about the initial unit of the variable.

34.20.3.5 Expression

Function (fx)
Inserts the selected function at the cursor position.

Variable (ABC **)**

Inserts the selected variable at the cursor position.

Delimiter (**)**

Inserts the selected delimiter at the cursor position.

Note: The **Expression** field supports autocomplete.

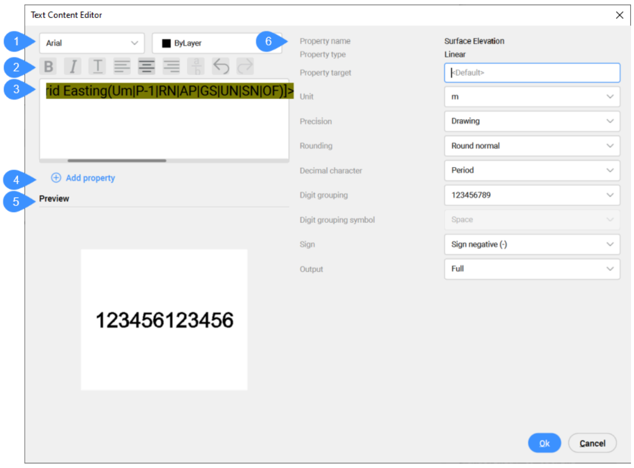
34.20.3.6 Preview

Displays a preview of the template results.

34.20.4 Text Content Editor dialog box

Opens via: **Label Style Editor** dialog box

The **Text Content Editor** dialog box allows you to add, edit or delete text components displayed in the Label Style.



1 Select font and colour

2 Select formatting of the text

3 Text editor field

4 Add property

5 Preview

6 Text component properties

34.20.4.1 Select font and colour

Selects font and colour of text components. Select the components in the text editor field you want to edit and assign them a font and a colour.

34.20.4.2 Select formatting of the text

Defines the style of the text (bold, italic, underline), formatting (align left, centre, align right) and fractional notation. Each property can have a different font, colour, style. Position of text is determined for the entire text box. At the end there are also the undo and redo buttons.

Note: To write the number in a fraction format, use / as a separator. For Example, type 3/4, mark it and select $\frac{a}{b}$ button.

34.20.4.3 Text editor field

Shows a list of properties that will be displayed in the Label Style. It is possible to add prefix and suffix to the components or insert text. Components can be rearranged by using Cut (Ctrl + X) and Paste (Ctl + V) or deleted by selecting them and pressing Delete button on your keyboard.

The properties of attributes can be edited on the right side of **Text Content Editor** dialog box, or by editing abbreviations in the brackets. The changes made in attribute settings are automatically updated in the text editor and vice versa. More information can be found the article **Understanding Civil point Symbols and Labels.**

34.20.4.4 Add property

Allows you to add new property to be displayed in the Label Style. In the **Add Property** dialog box you can select Civil Point properties as well as use properties from another object (Surface, Horizontal Alignment and Vertical Alignment).

34.20.4.5 Preview

Shows a preview of the components and any additional text (including prefixes and suffixes) that will be displayed in the label, all with an assigned style.

34.20.4.6 Text component properties

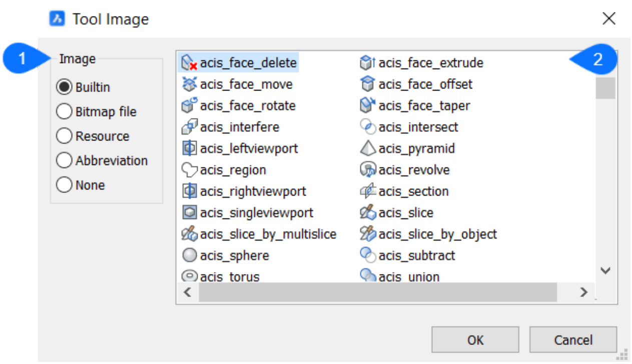
By clicking on the individual property in the text editor field, you can specify additional settings for that property on the right side of the dialog box.

Note: Click the **Ok** button to save the changes.

34.20.5 Tool Image dialog box

Open via: **Customize** dialog box

The **Tool Image** dialog box allows you to specify the image of tools.



1 Image

2 List of tool images

34.20.5.1 Image

Specifies the way to set the tool images.

Builtin
Specifies the tool image from one of the builtin tool images from the list.

Bitmap File
Specifies the tool image from a bitmap file.

Resource
Specifies the resource of the tool image.

Abbreviation
Specifies an abbreviation for the tool instead of an image.

None
No image or abbreviation is used for the tool.

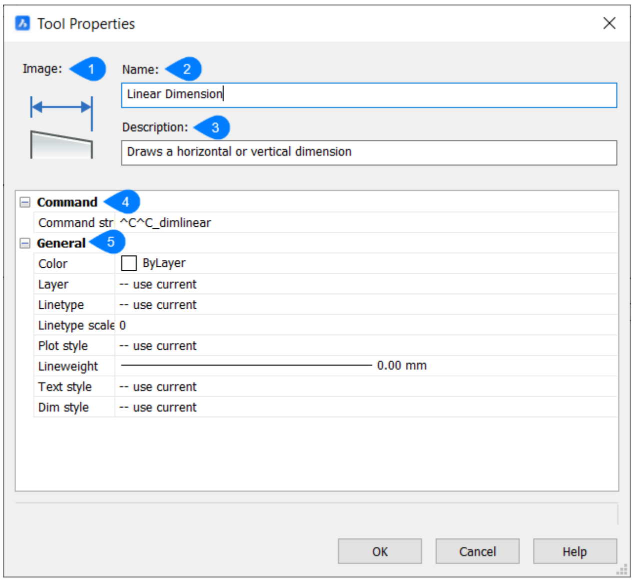
34.20.5.2 List of tools

Displays a list of built in tool images when the **Builtin** option is selected.

34.20.6 Tool Properties dialog box

Opens via: **Tool Palettes** panel

The **Tool Properties** dialog box allows you to modify the properties of a tool.



1 Image

2 Name

3 Description

4 Command String

5 General

34.20.6.1 Image

Displays an image of the selected tool.

34.20.6.2 Name

Specifies the name of the selected tool.

34.20.6.3 Description

Specifies the description of the selected tool.

34.20.6.4 Command String

Specifies the macro to run when you click the icon; you can use any macro characters used for menus and toolbar buttons.

34.20.6.5 General

Specifies the properties of the entity; the properties vary, depending on the tool.

34.21 U

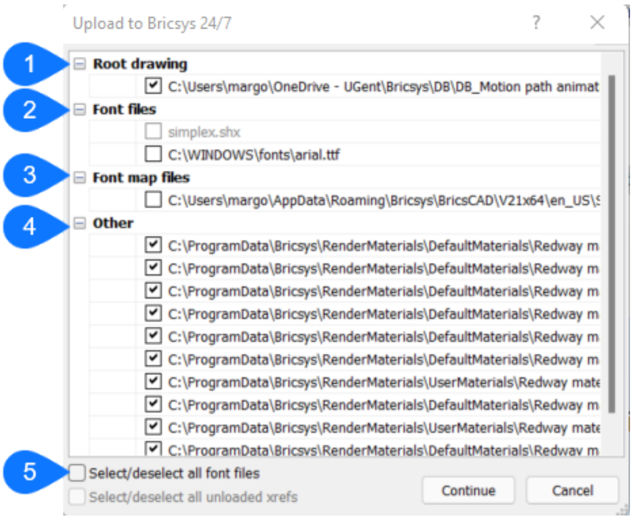
34.21.1 Upload to Bricsys 24/7 dialog box

Opens via: **Drawing Explorer** dialog box (ETRANSMIT command)

The **Upload to Bricsys 24/7** dialog box allows you to upload packages of drawing files and it dependencies to Bricsys 24/7.

A **report.txt** file is created and uploaded to Bricsys 24/7 server. The report contains a complete list of all files included and notes for distribution.

Note: The project needs to be saved before uploading it to Bricsys 24/7.



1 Root drawing

2 Font files

3 Font map files

4 Other

5 Select/deselect all font files

34.21.1.1 Root drawing

Displays a list of the current drawing files.

34.21.1.2 Font files

Displays a list of the currently used font files.

34.21.1.3 Font map files

Displays a list of the currently used font map files.

34.21.1.4 Other

Displays a list of other used files such as render material images.

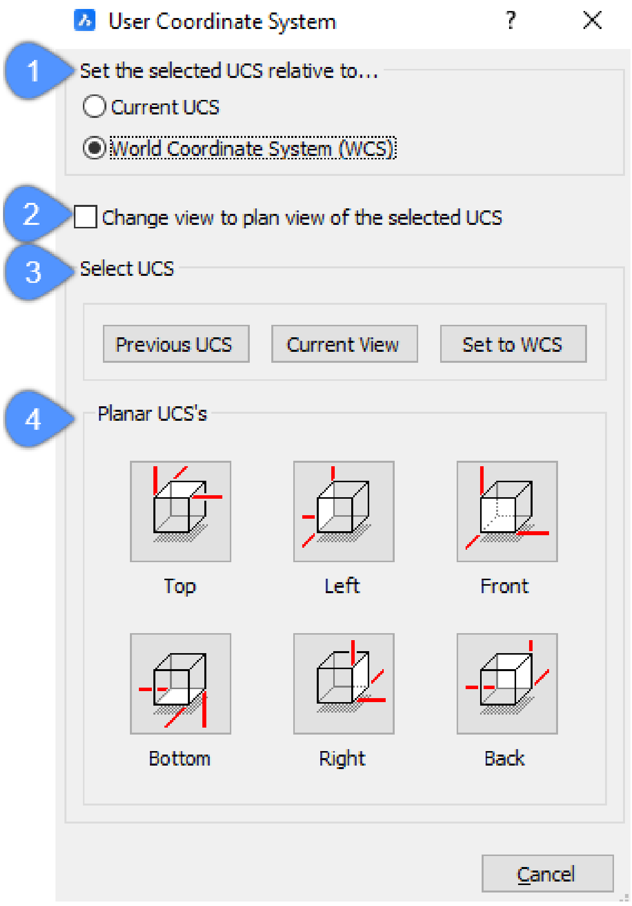
34.21.1.5 Select/deselect all font files

When checked, all font files and font map files are checked if this was not the case before. When unchecked, all font files are unchecked if there were any checked before. Pressing the **Continue** button will ask you to sign in Bricsys 24/7 to complete the uploading process.

34.21.2 User Coordinate System dialog box

Opens via: SETUCS command

The **User Coordinate System** dialog box allows you to set the UCS to a specified viewpoint.



1 Set the selected UCS relative to...

2 Change view to plan view of the selected UCS

3 Select UCS

4 Planar UCS's

34.21.2.1 Set the selected UCS relative to...

Specifies how the new UCS is set:

- **Current UCS**: Sets the new UCS relative to the current UCS.
- **World Coordinate System**: Sets the new UCS relative to the WCS. World Coordinate System is the default coordinate system when no UCS is set and it cannot be changed.

34.21.2.2 Change view to plan view of the selected UCS

Toggles whether the plan view is displayed when switching UCSES:

- **Yes**: plan view of the new UCS is displayed.

- **No**: the viewpoint does not change.

34.21.2.3 Select UCS

Sets the UCS to one of the following coordinate system. Clicking one of these buttons sets the UCS and then immediately exits the dialog box.

- **Previous UCS**: Changes the UCS to the previous UCS.
- **Current View**: Changes the UCS to match the current viewpoint.
- **Set to WCS**: Changes the UCS to match the WCS.

34.21.2.4 Planar UCS's

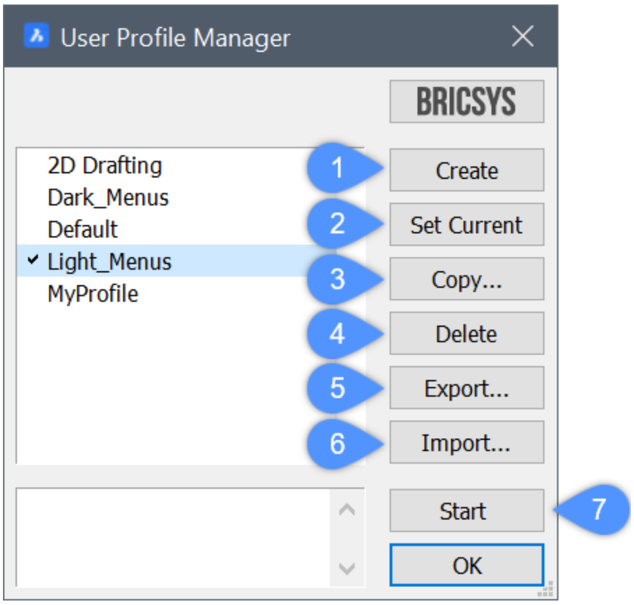
Sets the UCS to one of the following standard orthographic views. After you click one of the options, the program sets the UCS and then exits the dialog box.

- **Top**: Changes the UCS to match the top view.
- **Left**: Changes the UCS to match the left view.
- **Front**: Changes the UCS to match the front view.
- **Bottom**: Changes the UCS to match the bottom view.
- **Right**: Changes the UCS to match the right view.
- **Back**: Changes the UCS to match the back view.

34.21.3 User Profile Manager dialog box

Opens via: PROFILEMANAGER command

The **User Profile Manager** dialog box allows you to set the current user profile, to create, copy, delete, import and export profiles.



1 Create

2 Set Current

3 Copy...

4 Delete

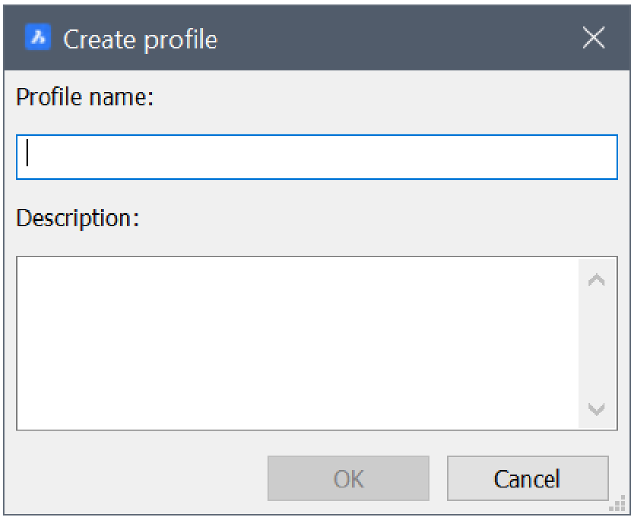
5 Export...

6 Import...

7 Start

34.21.3.1 Create

Allows you to create a new profile, using built-in default settings. The **Create profile** dialog box displays:



Profile name
Allows you to type a name for the new profile.

Description
(Optional) Allows you to type a description for the new profile.

34.21.3.2 Set Current

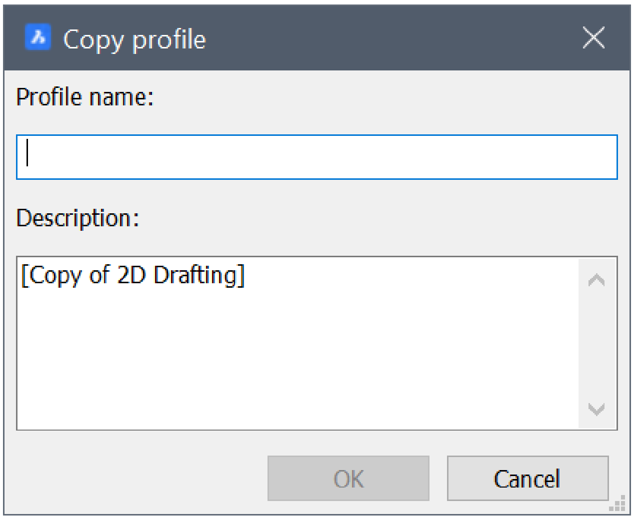
Sets the selected profile as the current profile.

Note:

- You can also double click a profile to set it current.
- The current profile is marked in the list with a check mark.

34.21.3.3 Copy

Creates a new profile as a copy of an existing profile. The **Copy profile** dialog box displays:



34.21.3.4 Delete

Deletes the selected profile.

Note: Deleting the current profile is not possible. When trying to do so, a dialog box displays that allows to reset all settings to factory defaults.

34.21.3.5 Export...

Exports the selected profile to an .arg (profile file) or .reg (registry file) to a specified folder.

Note: Exporting and importing a profile allows you to migrate user preferences from one computer to another.

34.21.3.6 Import...

Imports saved profiles.

34.21.3.7 Start

Starts a BricsCAD session using the selected profile.

Note:

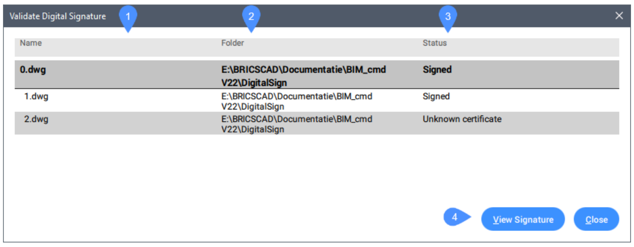
- This option is only available when the command is launched when executing **profilemanager_app.exe** in the BricsCAD installation folder. By default, the path is: *C:\Program Files\Bricsys\BricsCAD V25 en_US*.
- If the SINGLETONMODE preference variable is set to On, it is not possible to have multiple BricsCAD sessions open simultaneously.

34.22 V

34.22.1 Validate Digital Signature dialog box

Opens via: SIGVALIDATE command

Displays the status of all digital signatures for a drawing and its Xrefs.



Note: The first drawing in the list, which is written in a bold large font, is the host drawing.

Name
Displays the drawing file name.

Folder
Displays the drawing folder.

Status
Displays if the drawing is signed with a valid digital signature or not.

Signed
The drawing is signed with a valid digital signature.

Not signed
The drawing is not signed with any digital signature or the modification of the drawing was saved without applying the digital signature again.

Unknown certificate
The digital certificate is not located in the proper location or the digital certificate was revoked by the certificate authority.

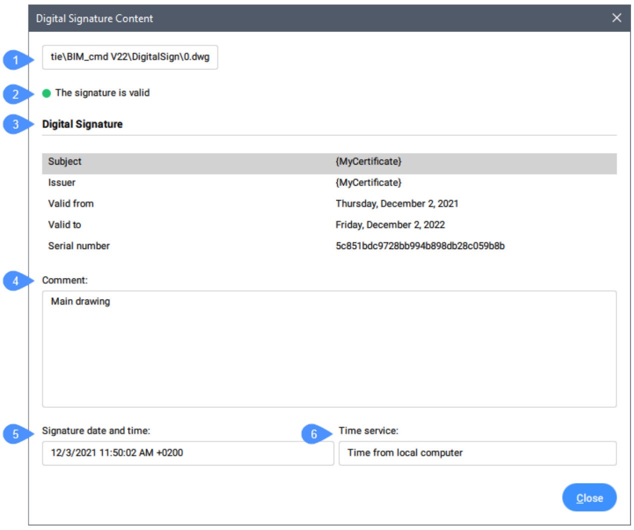
Not found
The drawing lost its initial location.

Invalid signature
The signed drawing was modified in an old BricsCAD version that doesn't support digital signature (V21).

Unloaded
The referenced file is not loaded.

Error
An error occurred during signature validation.

View Signature
Opens the **Digital Signature Content** dialog box:



Drawing identifying
Displays the path of the drawing.

Status of the signature

- Marked in green in case of a valid signature.
- Marked in red in case of an invalid signature.

Digital Signature
Subject: displays the certificate name.
Issuer: displays the author name.
Valid from:displays the date from which the certificate is valid.
Valid to: displays the date until which certificate is valid.
Serial number: displays the certificate serial number.

Comment
Displays the signature comment.

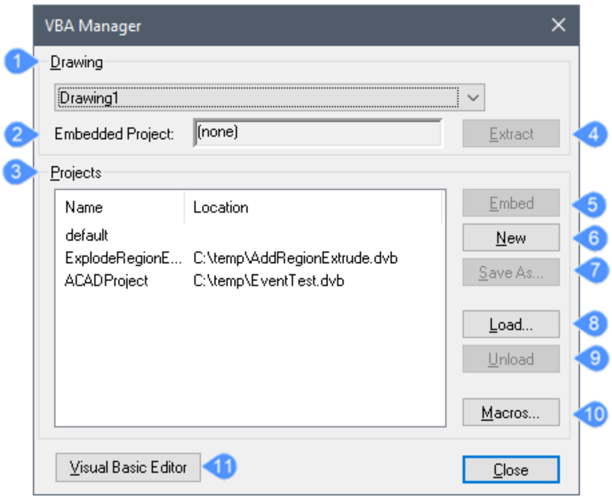
Signature date and time
Displays the date and time when the drawing was signed.

Time service
Displays the chosen time server.

34.22.2 VBA Manager dialog box

Opens via: VBAMAN command

The **VBA Manager** dialog box allows you to manage VBA projects.



1 Drawing

2 Embedded Project

3 Projects

4 Extract

5 Embed

6 New

7 Save As

8 Load

9 Unload

10 Macros

11 Visual Basic Editor

34.22.2.1 Drawing

Displays the names of drawings currently open in BricsCAD:

Choose a drawing name from the drop-down list; the embedded project is listed in the Embedded Project text box, if any.

34.22.2.2 Embedded Project

Displays the name of the embedded project, if any.

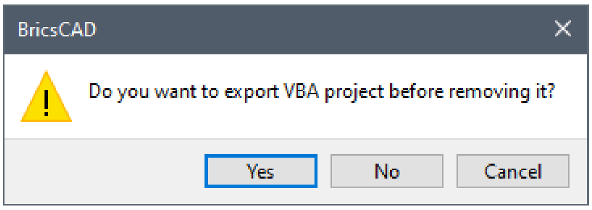
34.22.2.3 Projects

Name
Displays the names of projects loaded into BricsCAD at this time.

Location
Displays the drive and folder names of DVB and VBI files.

34.22.2.4 Extract

Removes the project from the current drawing; displays dialog box:



Yes: displays the **Save As** dialog box:

Name the project and click **Save**.

No: removes the project from the drawing without saving it as a DVB file.

Cancel: cancels the command; the project remains in the drawing.

34.22.2.5 Embed

Embeds a project in the current drawing. If the project is a new one, and has not yet been saved, this option displays the *Save As* dialog box.

Each drawing can contain just one embedded project; after one project is embedded, the Embed button becomes unavailable. To change embedded projects, use the Extract option to remove the current project.

34.22.2.6 New

Creates a new VBA project; gives the generic name VbaProject . Use the Save As option to rename and save the DVB or VBI project file.

34.22.2.7 Save As

Saves the selected project by another name. Displays the *Save As* dialog box.

34.22.2.8 Load

Loads DVB and VBI project files; displays the *Open* dialog box.

34.22.2.9 Unload

Unloads the selected project without a warning.

34.22.2.1 Macros
0

Displays the *Run BricsCAD VBA Macro* dialog box (see **Run BricsCAD VBA Macro dialog box** related article).

34.22.2.1 Visual Basic Editor
1

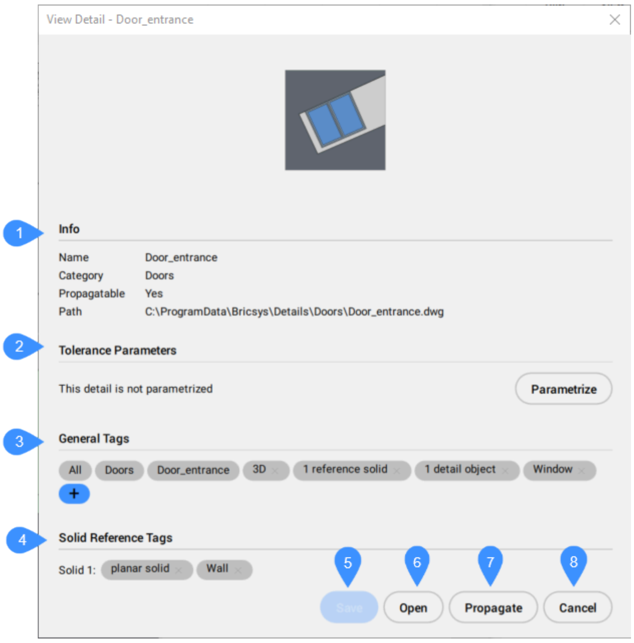
Opens the VBA integrated development environment.

34.22.3 View Detail dialog box

Opens via: **Details** panel

The **View Detail** dialog box displays information about the selected BIM detail.

View Detail for advanced (propagatable) details:



1 Info

2 Tolerance Parameters

3 General Tags

4 Solid Reference Tags

5 Save

6 Open

7 Propagate

8 Cancel

34.22.3.1 Info

Displays general information about the detail.

Name
Displays the name of the detail.

Category
Displays the category name of the detail.

Propagatable
Specifies if the detail is propagatable.

Path
Displays the path of the detail file.

34.22.3.2 Tolerance Parameters

Displays the parameters of the detail, or the option to parametrize the detail.

Parametrize

Opens the DWG detail file and executes the BIMPARAMETRIZEDETAIL command.

Note: If the details is already parametrized, this option is turned off, and the parameters are displayed.

34.22.3.3 General Tags

Displays the tags that generally identify the detail.

- Click the cross button to delete tags.
- Click the plus button to add a tag. The new added tags will be in an orange box.

34.22.3.4 Solid Reference Tags

Displays the tags that identify an individual solid reference from the detail. You cannot add reference solid tags here, you can only delete the existing tags.

Note: Tags are used to search for a specific detail in the library.

Note: Tags are automatically generated for the detail during the BIMCREATEDETAIL command.

34.22.3.5 Save

Saves the changes and closes the **View Detail** dialog box.

34.22.3.6 Open

Opens the DWG file of the detail.

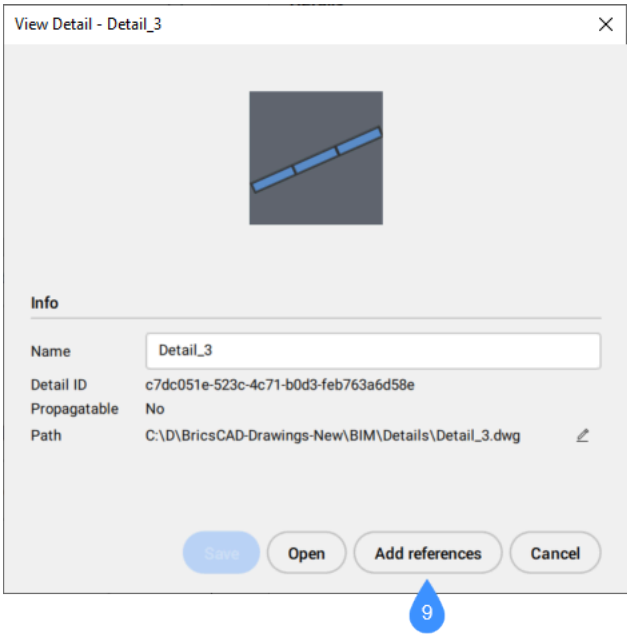
34.22.3.7 Propagate

Propagates the detail. See the BIMPROPAGATE command.

34.22.3.8 Cancel

Closes the **View Detail** dialog box without saving.

View Detail for basic details:



9. Add references

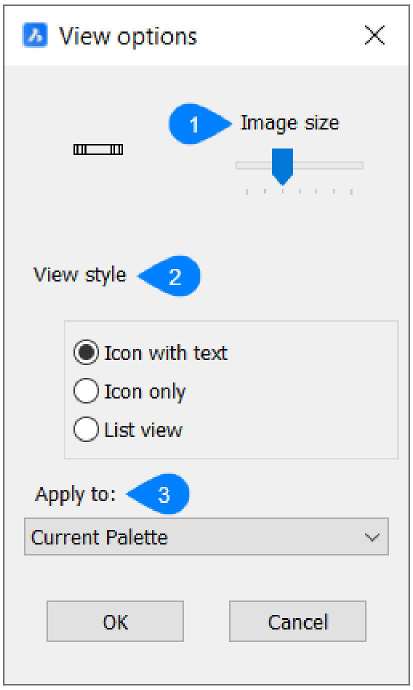
34.22.3.9 Add references

Launches the BIMADDDETAILREFS command.

34.22.4 View options dialog box

Opens via: **Tool Palettes** panel

The **View options** dialog box allows you to customize how the tools in the tool palettes panel are shown.



1 Image size

2 View style

3 Apply to

34.22.4.1 Image size

Sets the size of the displayed icons by sliding the indication arrow.

34.22.4.2 View style

Specifies how the tools are shown in the palette.

Icon with text
Displays the tools in a grid view with their icon and name.

Icon only
Displays the tools in a grid view with their icon only.

List view
Displays the tools in a list view with their icon and name next to it.

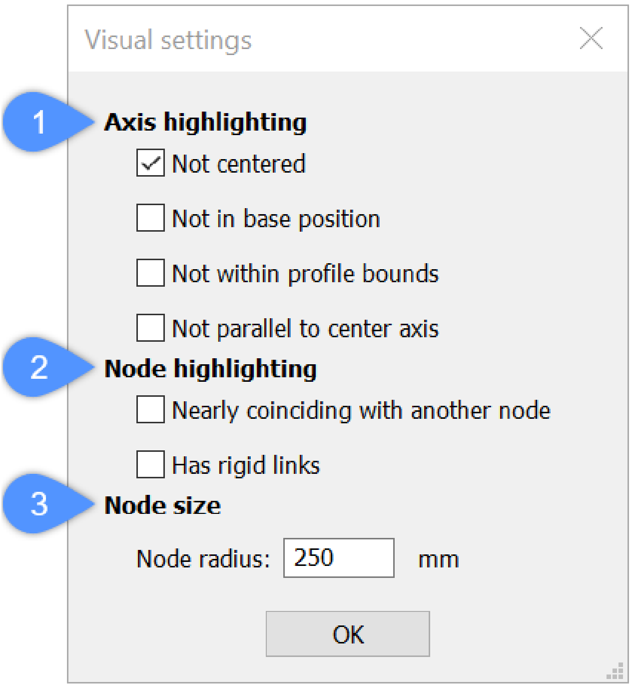
34.22.4.3 Apply to

The changes made can either be applied to the current palette only, or to all palettes.

34.22.5 Visual settings dialog box

Opens via: BIMANALYTICALMODEL command

The **Visual settings** dialog box allows you to tick and untick different features of axes and nodes to highlight (in yellow) the axes and nodes that match these features. You can also change the size of the disks representing the nodes.



1 Axis highlighting

2 Node highlighting

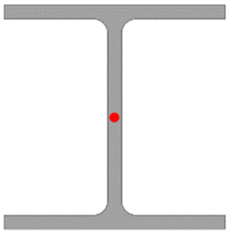
3 Node size

34.22.5.1 Axis highlighting

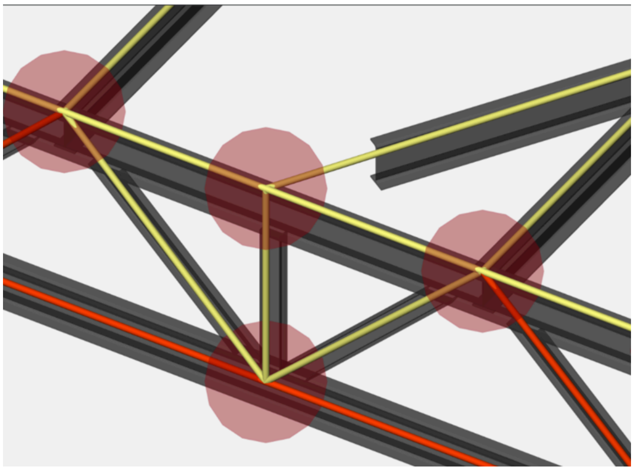
Specifies when/how the axis is highlighted.

Not centered
Highlights any axis that is not in the center of the linear solid.

- The red dot represents the center of the profile:

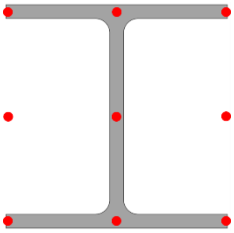


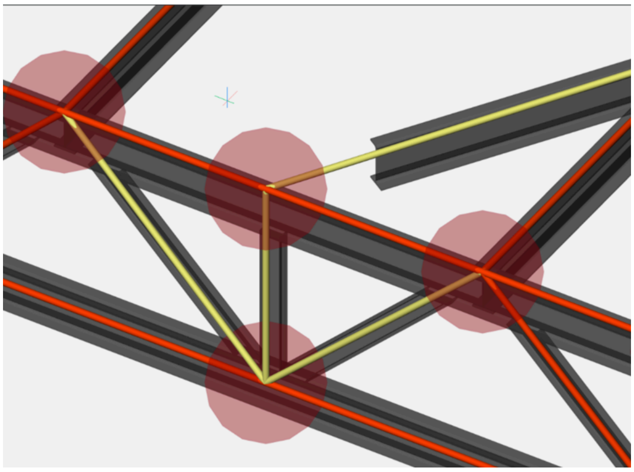
- The axes of the beams are highlighted in yellow because the axes of the top beams are not in the center of the beam but on the top instead.



Not in base position
Highlights any axis that is not in one of the base positions of the linear solid.

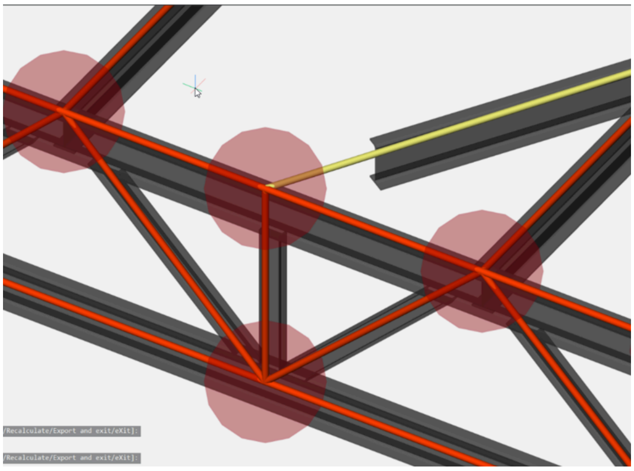
- The red dots represent the base positions of the profile:





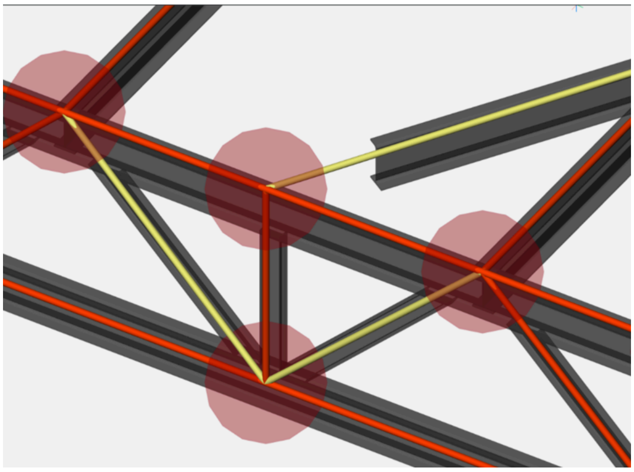
Not within profile bounds

Highlights any axis that is outside the profile boundaries.



Not parallel to center axis

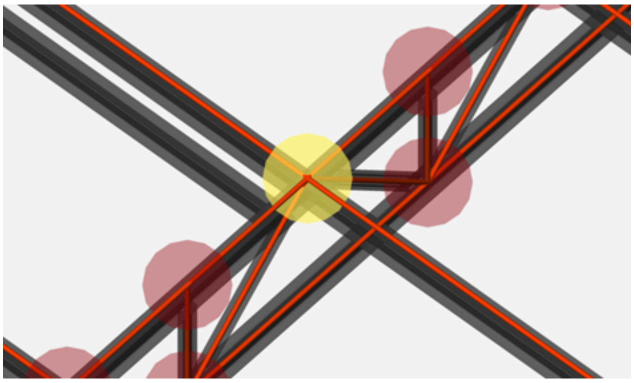
Highlights any axis that is not parallel to the center axis.



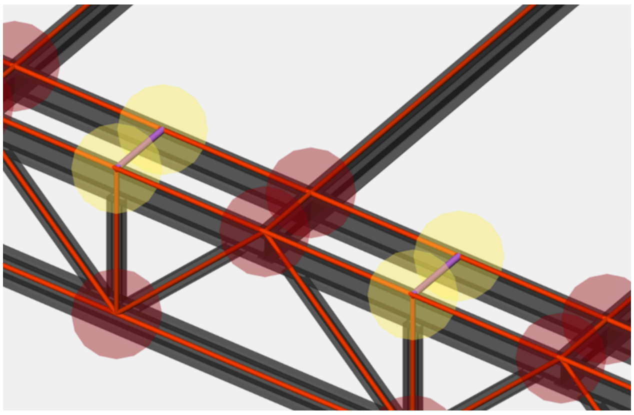
34.22.5.2 Node highlighting

Specifies when/how nodes are highlighted.

Nearly coinciding with another node
Highlights any node that is nearly coinciding with another node.



Has rigid links
Highlights any node that has rigid links. See the purple links in the image:



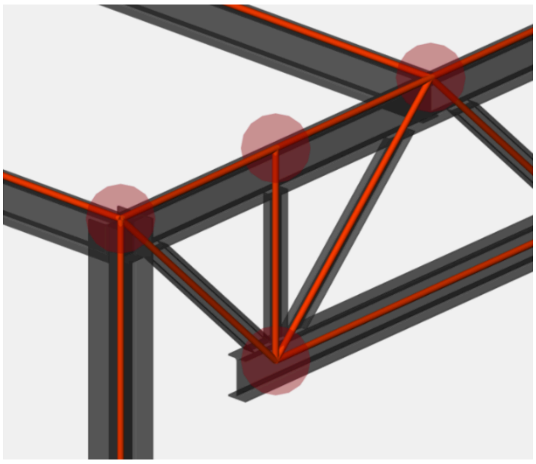
34.22.5.3 Node size

Specifies node size characteristics.

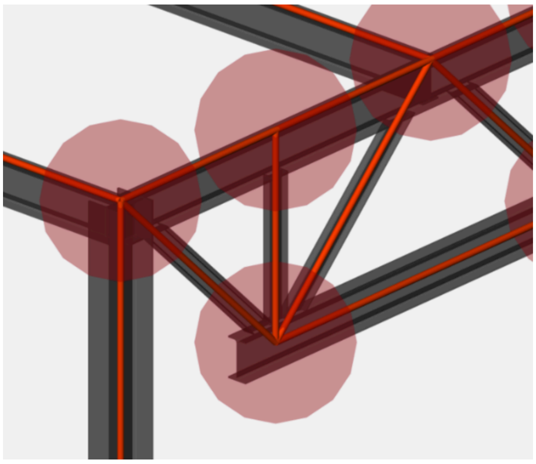
Node radius

The nodes in the analytical model are highlighted using a filled circle. Specify the size of nodes by entering the node radius.

- Node radius: 150mm



- Node radius: 350mm



34.23 W

34.23.1 Weld Symbol dialog box

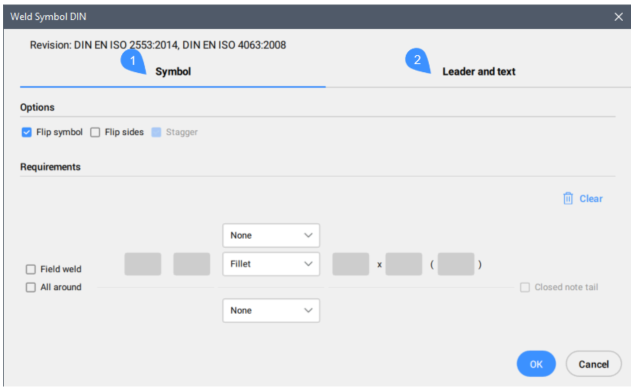
Opens via: AMWELDSYM command

The **Weld Symbol** dialog box allows you set the default weld symbols for the current drafting standard.

The parameters entered for the weld symbols will be stored temporarily and used as default values for the next weld symbols setting.

The most recent revision of the standard to which this symbol conforms is displayed at the top of the dialog box.

The dialog box contains 2 main tabs, **Symbol** and **Leader and text**.

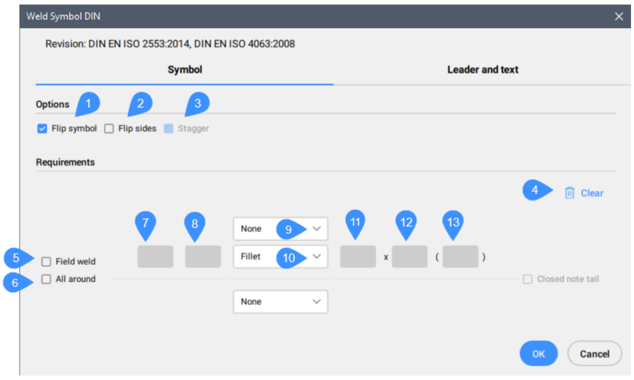


1 Symbol

2 Leader and text

34.23.1.1 Symbol

Specifies the weld symbol characteristics.



1 Flip symbol

2 Flip sides

3 Stagger

4 Clear

5 Field weld

6 All Around

7 Depth

8 Size

9 Weld type

10 Contour

11 Number

12 Length

13 Spacing

14 Closed note tail

Options

Flip symbol (1)

Specifies whether the symbol is aligned to the right or left.

- **Not ticked**: aligned to the right.
- **Ticked**: aligned to the left.

Flip sides (2)

Swaps the symbol and its parameters on the arrow side with the symbols on the other side.

Note: The **Flip sides** setting is taken into account next time you apply a weld symbol.

Stagger (3)

Specifies whether the weld is a staggered intermittent weld.

Requirements

Clear (4)

Clears the last inserted parameters. The cells will become empty.

Field weld (5)

Toggles the visibility of the Field weld symbol in the welding symbol.

All Around (6)

Toggles the visibility of the All Around symbol in the welding symbol.

Depth (7)

Defines the groove depth of the weld.

Size (8)

Defines the size of the weld.

Weld type (9)

Defines the weld type from the below options:

- Fillet
- Plug in circular holes
- Fusion spot (projection weld in system B)
- Resistance spot (projection weld in system A)
- Fusion seam
- Resistance seam
- Backing Run; Back or Backing Weld / USA
- Flanged butt/corner
- Square butt
- V Butt
- Bevel butt
- U Butt
- J Butt
- Edge

- Fold Joint
- Inclined Joint
- Surface Joint
- Steep flanked V butt
- Steep flanked bevel butt
- V butt with broad root face
- Bevel butt with broad root face

Note: Each weld type has its associated icon.

Contour (10)

Defines the shape of the weld from the below list:

- Concave Contour
- Convex Contour
- Flat/Flush Contour
- Flush Finished Contour
- Toes Shall Be Blended Smoothly
- None

Number (11)

Defines the number of welds.

Length (12)

Defines the length of the weld.

Spacing (13)

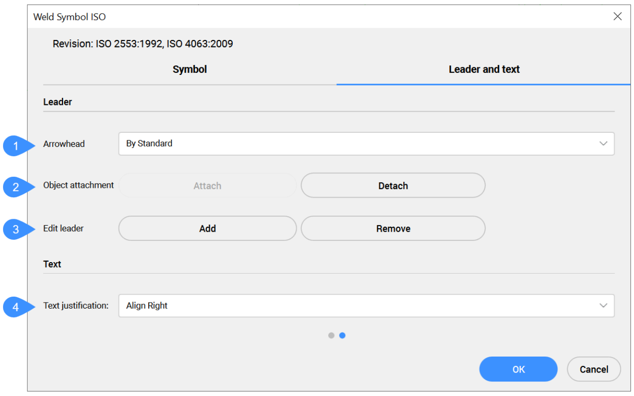
Defines the distance between welds.

Closed note tail (14)

Displays a closed state of the note tail.

34.23.1.2 Leader and text

Specifies the Leader and text characteristics.



1 Arrowhead

2 Object attachment

3 Edit leader

4 Text justification

Leader

Arrowhead (1)
Defines the default arrowhead type for welding symbols.

Object attachment (2)
Defines if the symbol leader is attached or detached to the object.

Attach
Attaches the symbol to the selected object.

Detach
Detaches the symbol from the object.

Edit leader (3)
Edits leader segments.

Add
Adds new leader segments or leader nodes.

Remove
Removes leader segments or leader nodes.

Text

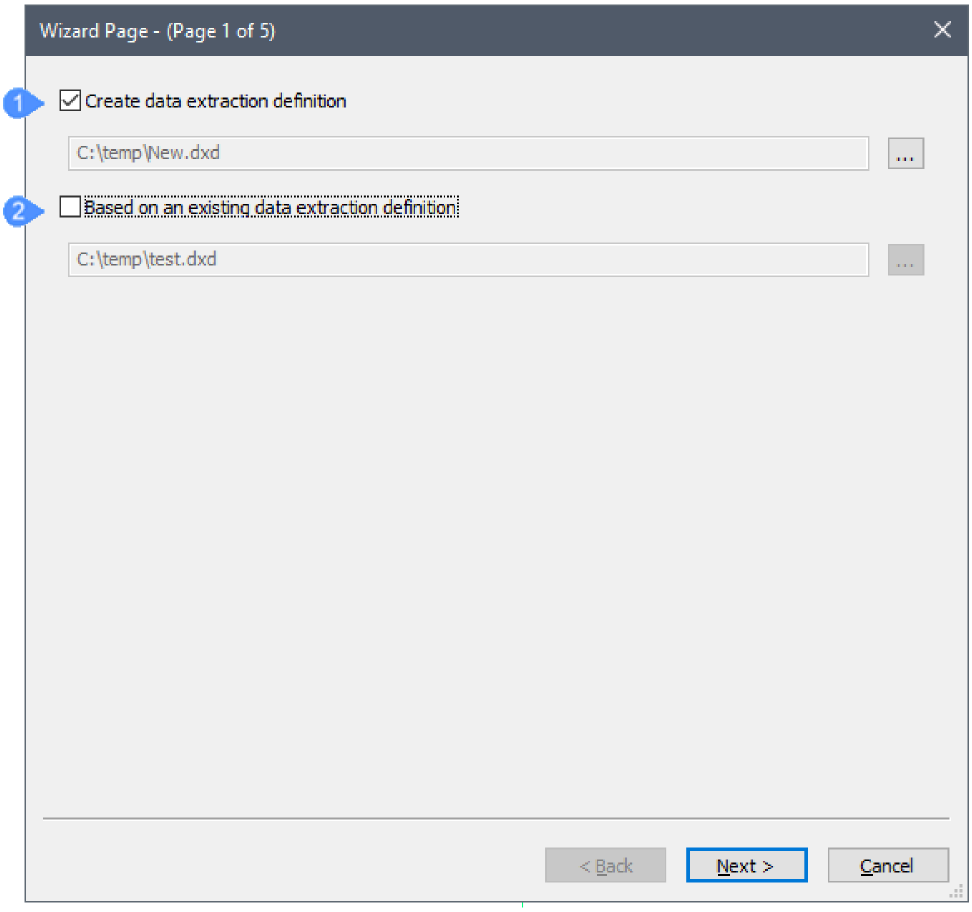
Text justification (4)
Specifies whether the symbol text is aligned to the right, center or left.

34.23.2 Wizard Page dialog box

Opens via: DATAEXTRACTION command

The **Wizard Page** dialog box guides you to export entity properties, block attributes, and drawing information to a table or to an external data file.

34.23.2.1 Wizard Page 1 Dialog Box



1 Create data extraction definition

2 Based on an existing data extraction definition

Create data extraction definition

Creates a new .dxd (Data eXtraction Definition) file:

1 Tick the checkbox.

2 Click **Browse**.

3 Select a folder in the **Save Data Extraction** file dialog box.

4 Enter a name in the **File name** field.

5 Click **Save**.

Based on an existing data extraction definition

Uses an existing .dxd file as the template for the new one to perform the same type of extraction in a different drawing, or to edit the data extraction definition:

1 Tick the checkbox.

2 Click **Browse**.

3 In the **Open Data Extraction Definition** file dialog box, select a .dxd file.

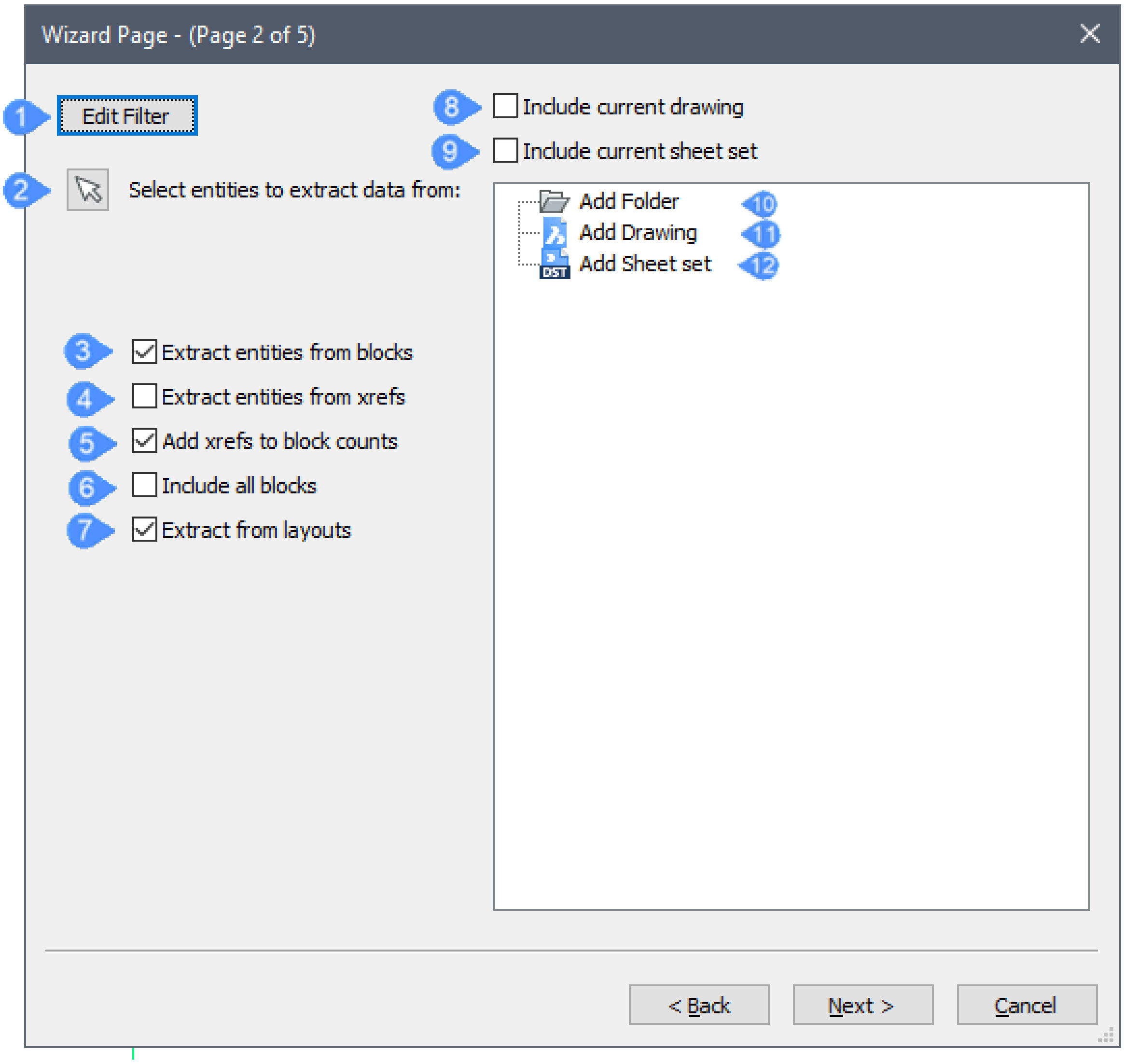
4 Click **Open**.

Newly created custom *.dxd files can be used to create schedules in the Project Browser.

.dxd files can be edited in a text editor such as MS Notepad.

Click **Next** for the next page.

34.23.2.2 Wizard Page 2 Dialog Box



1 Edit filter

2 Select entities to extract data from

3 Extract entities from blocks

4 Extract entities from Xrefs

5 Add Xrefs to block counts

6 Include all blocks

7 Extract from layouts

8 Include current drawing

9 Include current sheet set

10 Add Folder

11 Add Drawing

12 Add Sheet Set

Guides you to select entities for Data Extraction and apply filters for them before it is extracted in order to have less information to sort later.

The DXEVAL system variable controls when the update notification occurs in the drawings where changes affect the extracted data.

Edit filter
Creates or edits a Filter.

Select entities to extract data from
Dismisses the dialog box temporarily so you can select entities in the drawing. Enter **Ctrl+A** to select all entities in the drawing. Press **Enter** or right-click to stop selecting entities.

Extract entities from blocks
Includes entities nested in blocks.

Extract entities from Xrefs
Includes entities in externally referenced drawings (Xref files).

Add Xrefs to block counts
Counts selected Xrefs as blocks.

Include all blocks
Includes all block entities.

Extract from layouts
Includes entities from paper space of the drawing.

Include current drawing
Adds the current drawing from which to extract data.

Include current sheet set
Adds the current sheet set from which to extract data.

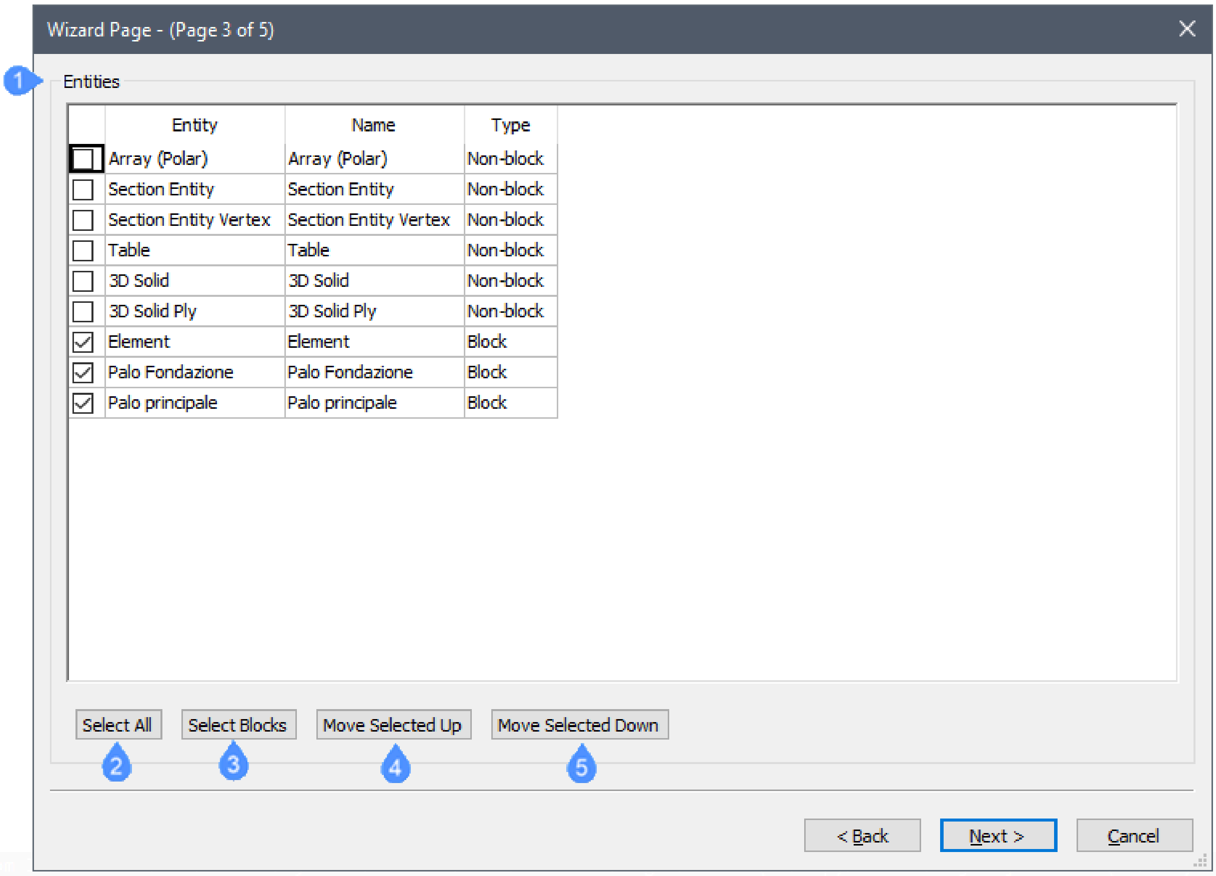
Add Folder
Adds a folder with the drawing from which to extract data.
Double-click to open the *Folder Options* dialog box.

- **Folder**: click **Browse**, then select a folder in *Choose a Folder* dialog box.
- **Options**
    - **Include subfolders**: tick the checkbox to include subfolders.
    - **Use a wildcard to filter drawings to be included**: tick the checkbox, then type a text string in the filter field. For example, House*.dwg includes all files starting with 'House'.

Add Drawing
Adds a drawing from which to extract data with a double click.

Add Sheet Set
Adds a sheet set from which to extract data with a double click.
Click **Next** for the next page.

34.23.2.3 Wizard Page 3 Dialog Box



1 Entities

2 Select All

3 Select Blocks

4 Move Selected Up

5 Move Selected Down

Controls which entity types are included or not in the data extraction procedure.

Select multiple items:

- To select more than one item in a row: select an item, then hold down the **Shift** key while clicking additional items.
- To select multiple items, hold the **Ctrl** key while selecting multiple entities.
- To deselect items, click them again while holding the **Ctrl** key.

To sort the list, click the title in the column heading. Click again to sort in reverse order.

Entities

- **Entity**: lists the entity types and blocks in the selection, according to the options set in the *Data Extraction* dialog box.
- **Name**: entity names. Accept the default names or type a customized name. The entity type name displays in the default **Name** field in the data extraction export.
- **Type**: reports the type of the selected entity: Non-Block, Block or Block with attributes.

Select All
Selects all items for data extraction.

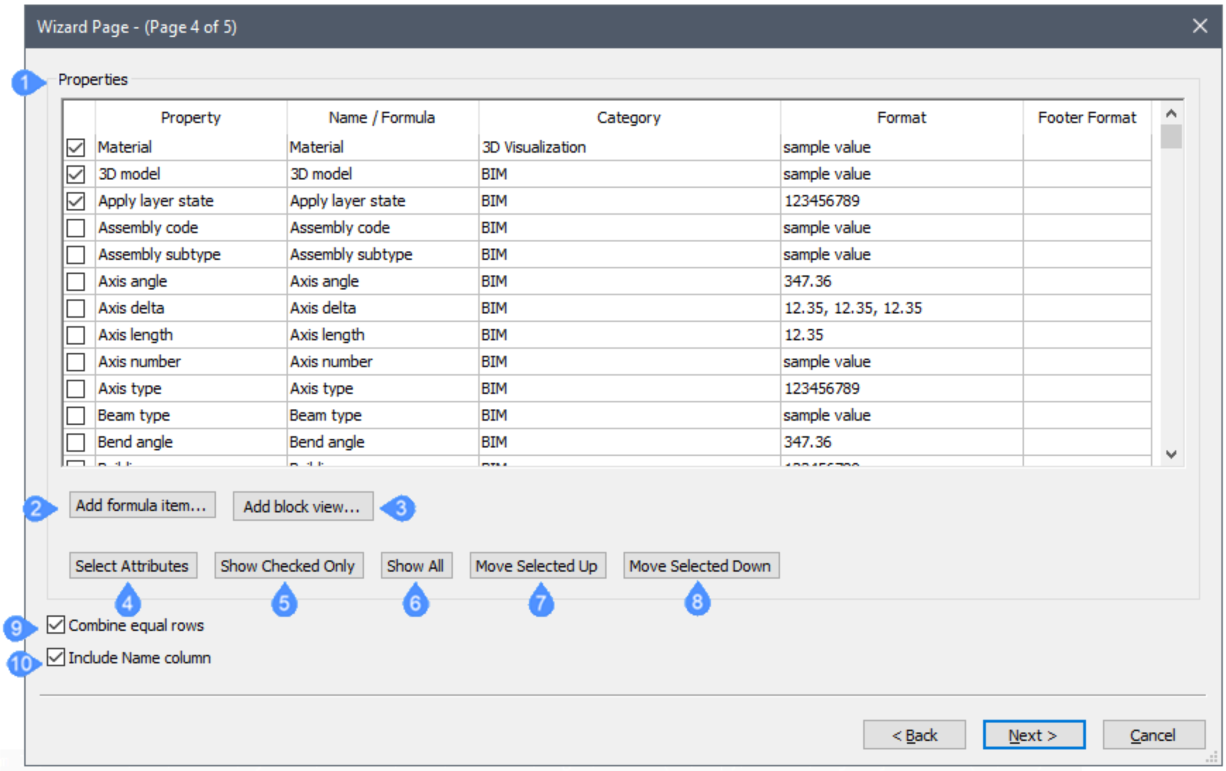
Select Blocks
Selects only blocks in the list, because they usually contain attributes.

Move Selected Up
Move the selected item(s) up the list.

Move Selected Down
Move the selected item(s) down the list.

Click **Next** for the next page.

34.23.2.4 Wizard Page 4 Dialog Box



1 Properties

2 Add formula item

3 Add block view

4 Select Attributes

5 Show checked only

6 Show All

7 Move Selected Up

8 Move Selected Down

9 Combine equal rows

10 Include Name column

Select the properties to be included in the data extraction file: tick the checkbox to check/uncheck a property. Use the same techniques to select more than one item as described in **Wizard Page 3** Dialog Box.

To sort the list, click the title in the column heading. Click again to sort in reverse order.

Properties

- **Property**: lists all properties of the selected entities.
- **Name /Formula**: lists the name of the selected entities or the formula. Accept the default names or type a customized name. The property names display in the column headings in the data extraction export.
- **Category**: reports the category of the selected entities.
- **Format**: formats the item in the export file. Click to display the dialog box. The layout of the dialog box depends on the selected item, just like the Format section of the **Field** dialog box.
- **Footer Format**: selects from the drop-down list opened with right mouse button the Footer settings:

Add formula item
Adds a formula item.

Add block view
Adds BlockViewProperty as Block view category.

Select Attributes
Select only items that contain attributes.

Show checked only
Show only selected items, hiding the ones that are not selected.

Show All
Shows all items.

Move Selected Up
Move the selected item(s) up the list.

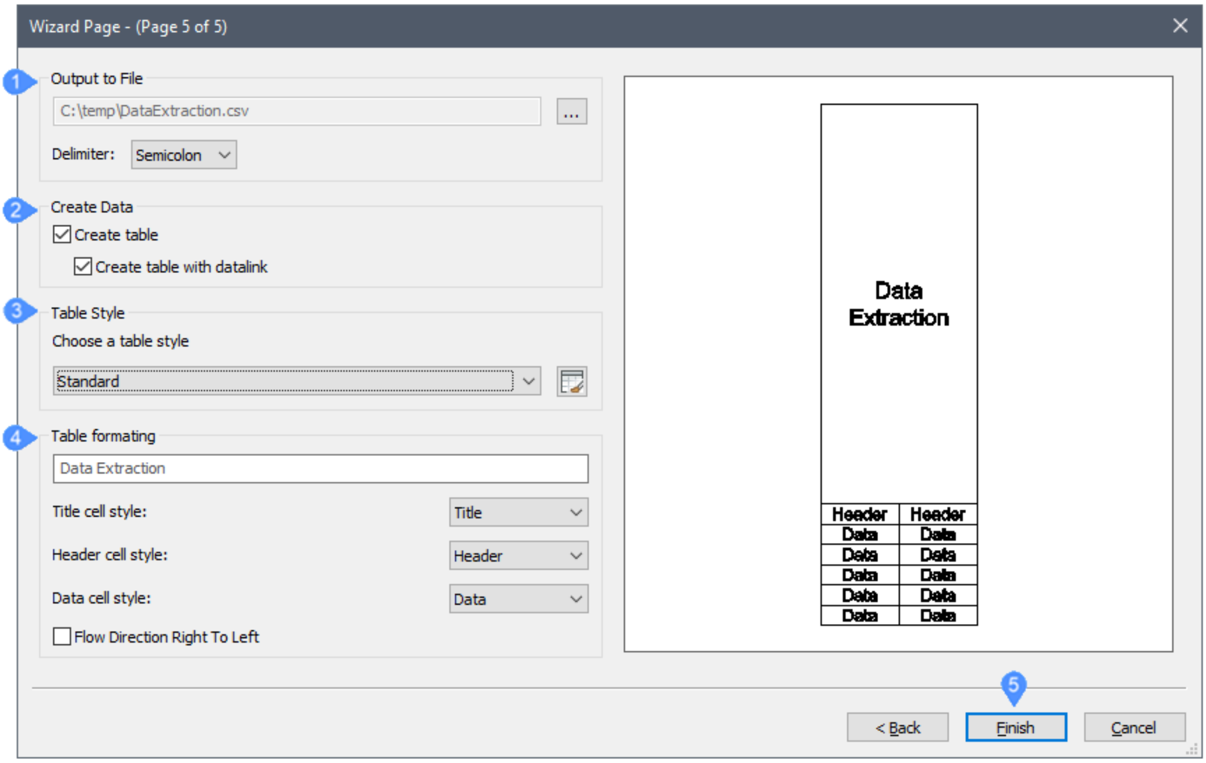
Move Selected Down
Move the selected item(s) down the list.

Combine equal rows
Combine rows that contain the same entities.

Include Name column
Toggles the display of the **Name** column in the exported data.

Click **Next** for the next page.

34.23.2.5 Wizard Page 5 Dialog Box



1 Output to File

2 Create Data

3 Table Style

4 Table formatting

5 Finish

Output to File
Exports the data in a delimiter separated format (.csv):
Click **Browse**, enter a name for the data file, and then click **Save**. If you select an existing .csv file, you will be prompted to overwrite it.
Delimiter: specifies the separator between fields of data:

- Semicolon
- Comma

  Note: Multiply composition names contain a comma, therefore do not select comma as the delimiter character in case multiply compositions are processed.

- Space
- Tab

Create Data
Exports the data in a Table:

- Create table: creates a static table.
- Create table with datalink: creates a dynamic table. Executing the DataLinkUpdate command synchronizes the data in the table with the model. Save the model first. You will be prompted to save the *.dxd definition file with the table.

Table Style
Select a table style from the drop-down list.
Click the **Table Style** icon; edit an existing table style or create a new one.

Table formatting
Type a title in the Title field.

- Title cell style: sets the title cell style.
- Header cell style: sets the Header cell style.
- Data cell style: sets the Data cell style.
- Flow Direction Right to Left: indicates the flow direction from right to left.

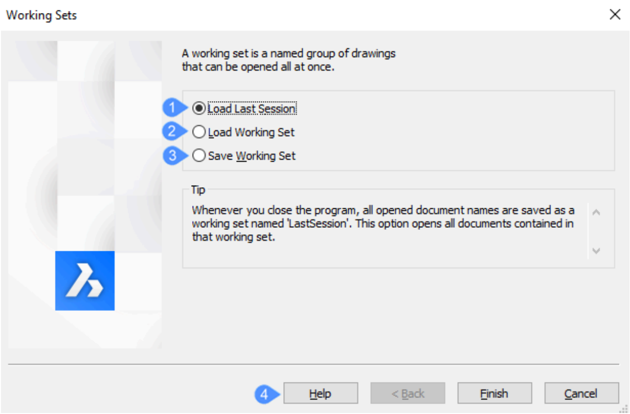
Finish
Depending on the selected output:

- Extracts the data and saves them to the .csv file. The file can be opened in a spreadsheet or database program.
- Prompts you to specify an insertion point for the table(s).

34.23.3 Working Sets dialog box

Opens via: WORKSETS command

The **Working Sets** dialog box allows you to create and load sets of drawing files, making it convenient to load and entire group of drawings at once.



1 Load Last Session

2 Load Working Set

3 Save Working Set

4 Help

34.23.3.1 Load Last Session

Opens all drawings that where open when the last BricsCAD session was closed.

After you click the **Finish** button, BricsCAD opens all the drawings that were open the last time you closed the program.

34.23.3.2 Load Working Set

Click the **Next** button and choose the name of a workset from the **Select a Workset** list. Click the **Finish** button. The program opens all the drawings that are part of the workset.

34.23.3.3 Save Working Set

Saves drawings currently open by name as a working set. Click the **Next** button.

When you click on the **Save** button, the **Save workset** dialog box opens. Click the **Finish** button to close the dialog box.

Note: Working sets are saved in the registry and so they are defined differently for each user profile. See the **PROFILEMANAGER command** article to learn more about user profiles.

Note: Delete removes a named working set. Redefine (overwrites) an existing named working set.

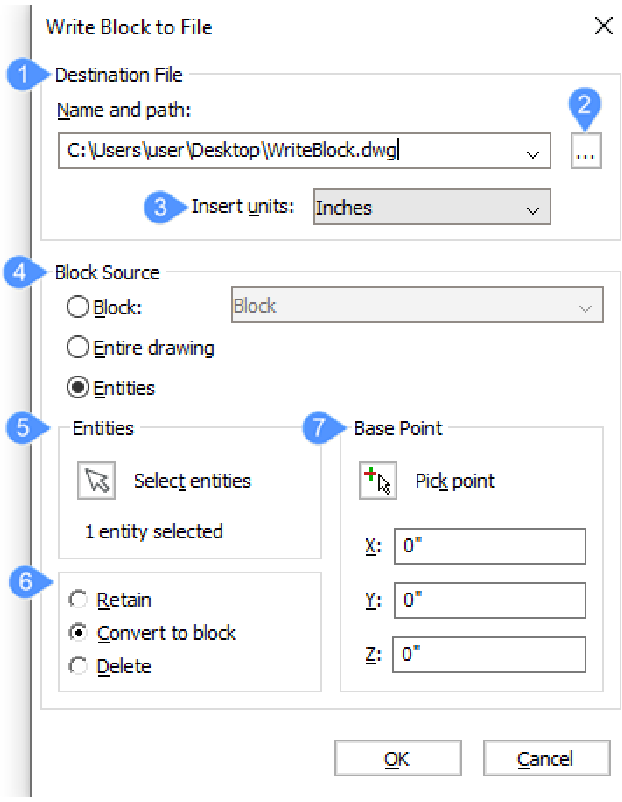
34.23.3.4 Help

Opens the Bricsys Help article about WORKSETS command.

34.23.4 Write Block to File dialog box

Opens via: WBLOCK command

The **Write Block to File** dialog box allows you to write blocks and other drawing parts to a separate drawing file.



1 Destination file

2 Save Block

3 Insert units

4 Block Source

5 Entities

6 Options

7 Base Point

34.23.4.1 Destination file

Specifies what the file should be named and where it should be saved.

- **Name and path**: Specify the default path and file name (new block.dwg).

34.23.4.2 Save Block

Changes the path and file name. Opens the **Save Block** dialog box.

34.23.4.3 Insert units

Specifies the units to be used by the exported entities when they are later inserted into other drawings.

You can choose the measurement unit from the drop-down list.

Units are saved in the INSUNITS variable.

34.23.4.4 Block Source

Selects the part of the drawing to save to a new block:

- **Block**: Saves a block from the drawing. This option is unavailable when no blocks exist in the drawing. When this option is selected, the other options become unavailable. You can choose from the drop-down list the name of a block you want to save.
- **Entire drawing**: Saves the entire drawing to the file. When this option is selected, all other options become unavailable.
- **Entities**: Saves entities that you select to the file.

34.23.4.5 Entities

When the **Block Source** is set to **Entities**, you are prompted at the Command line to choose one or more entities using any selection method.

34.23.4.6 Options

Specifies what to do with the selected entities after you click **OK** to close the dialog box:

- **Retain**: Keeps the selected entities as they are; this is like copying the selection to a new drawing.
- **Convert to block**: Converts the selected entities to a block in the current drawing.
- **Delete**: Deletes the entities you had selected in the drawing; this is like moving the selection to a new drawing.

  Note: If the **Delete** option was set, the OOPS command brings back the erased block entities.

34.23.4.7 Base Point

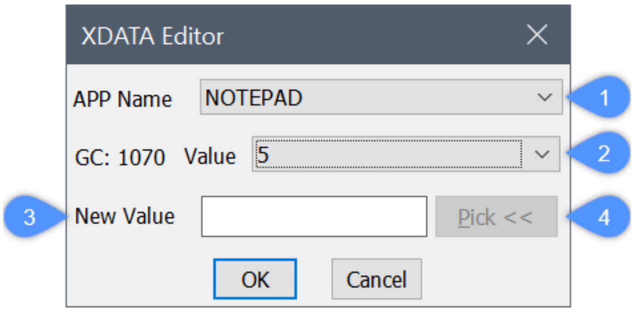
Specifies the origin point of the saved entities in the new file. You can pick a point in the drawing or specify the X, Y, Z coordinates.

34.24 X

34.24.1 XDATA Editor dialog box

Opens via: XDEDIT command (Express Tools)

The **XDATA Editor** dialog box allows you to edit the extended entity data (xdata) associated with an entity.



1 APP Name

2 DXF Code

3 DXF Value

4 Pick <<

34.24.1.1 APP Name

Allows you to choose an application name from the list.

34.24.1.2 DXF Code

Allows you to change the XData DXF key code.

34.24.1.3 DXF Value

Allows you to specify a new DXF value.

34.24.1.4 Pick<<

Temporarily disables the **XDATA Editor** dialog box and allows you to pick a new value.

34.25 Y

34.26 Z

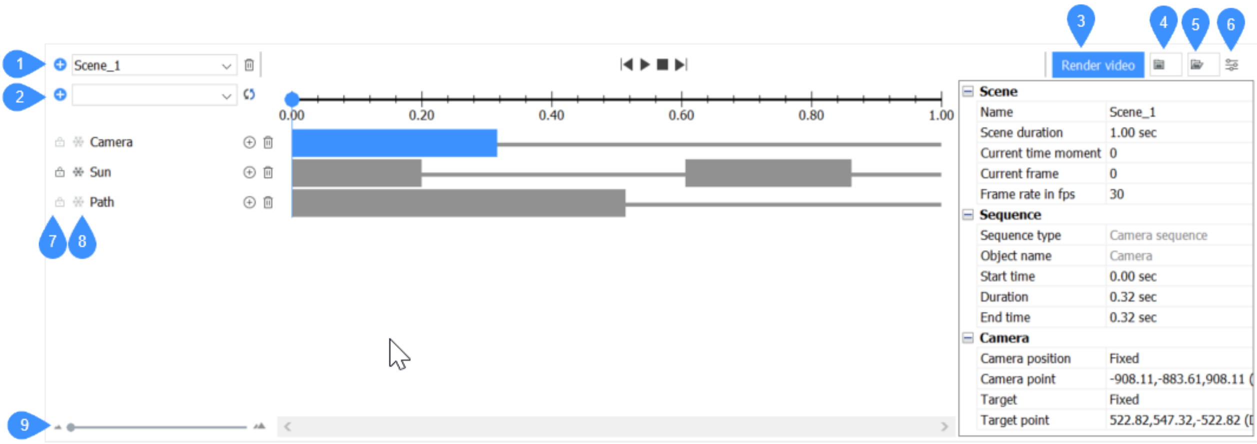
35. Panels

35.1 Animation Editor panel

To open the **Animation Editor** panel, run the ANIMATIONEDITOROPEN command or right-click a ribbon panel/toolbar and choose **Animation Editor** from the **Panels** list.

The **Animation Editor** panel allows you to create schedules for multi-parametric animations with complex camera movements and render them as 2D videos.

Note: The **Animation Editor** panel is disabled on macOS and Linux.



1 Add scene

2 Add timeline

3 Render video

4 Record graphics for the current scene

5 Load animation graphics

6 Configurator

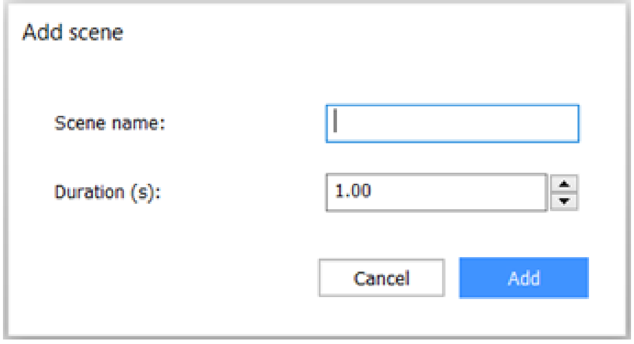
7 Lock/unlock timeline button

8 Disable/enable timeline effects button

9 Slider

35.1.1 Add scene

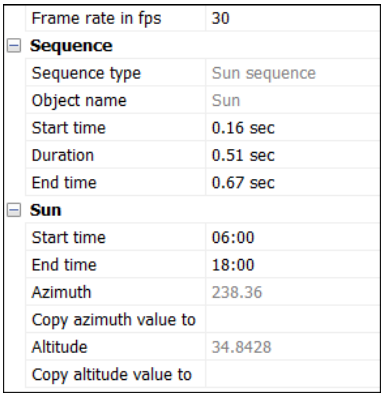
Opens the **Add scene** dialog box to specify the scene name and its duration (s):



35.1.2 Add timeline

Adds the selected parameter to the timeline.

One of the parameters that can be added is the sun effect. Once the sun animation sequence is added to the timeline, several properties are available in the properties configurator of the **Animation Editor** panel.



Sequence properties

Defines the properties of the sun sequences on the timeline.

Sun properties

Defines the properties of the sun itself regarding its position and timing.

35.1.3 Render video

Opens the **Render video** dialog box to specify the render options.

35.1.4 Record graphics for the current scene

Records animation graphics and saves it to a zip-archive or to a folder of your choice.

35.1.5 Load animation graphics

Loads animation graphics data from a zip archive or a folder and opens the special mode for playing these graphics.

35.1.6 Configurator

Toggles the **settings** panel.

35.1.7 Lock/unlock timeline button

Toggles the timeline to lock any timeline changes or vice versa, in order to enable changes.

35.1.8 Disable/enable timeline effects button

Toggles the timeline effects between enabled and disabled. Disabled timeline effects are excluded from the animation.

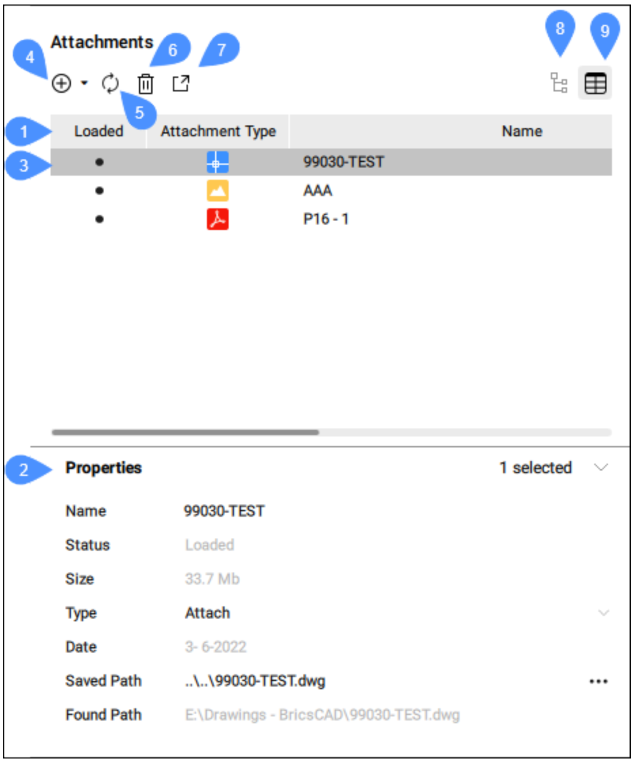
35.1.9 Slider

Allows you to move the slider to zoom in/out on the timelines.

35.2 Attachments panel

To open the **Attachments** panel, run the ATTACHMENTSPANELOPEN command or right-click a ribbon panel/toolbar and choose **Attachments** from the **Panels** list.

The **Attachments** panel offers a central location for viewing and managing attached drawings, images, PDF files and point clouds.



1 Attachments properties list header

2 Properties

3 Attached files list

4 Attach a file

5 Refresh

6 Detach

7 Open

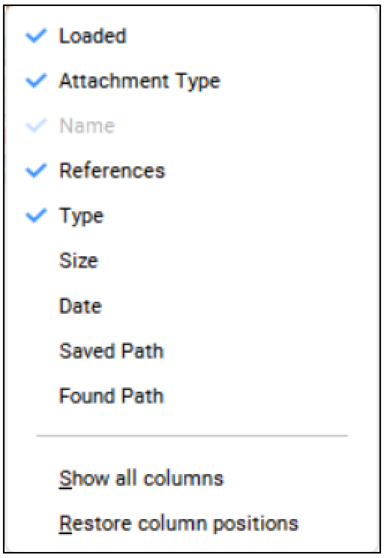
8 Tree view

9 Grid view

35.2.1 Attachments properties list header

Column headers describe theproperties of each attachment. You can sort theattached files list in ascending or descending order foranyproperty by clicking once or twice on thecolumn header. A right-click menuon the column headers enables you to customize them:

Note: Some columns are hidden by default.



Loaded

Specifies the load status of the attachments:

- **Load**: the attachments are loaded and displayed in the drawing editor.
- **Unload**: the attachments are not loaded or displayed in the drawing editor, but maintain their attachment information, such as scale and location.
- **Not found**: the path is not found. The attachments have to be re-attached.

Attachment Type

Displays the attached file type (DWG, PDF, image or point cloud).

Name

Specifies the attachment name. By default, the attachment name is the same as the file name. To rename an attachment, double-click the name and enter the new name. Renaming an attachment does not change the file name.

References

Specifies how many times the attachments are referenced in the current drawing.

Note: Only visible block reference inserts on all nested levels are counted.

Type

Specifies the attachment type. Click the attachment type to switch between **Attach** and **Overlay**.

Date
Specifies the date on which the attached files were last saved.

Saved Path
Specifies the path, including the location and file name, of the attachments. To change the saved path, click it and enter the new path or select the **Browse** button to display the **Select reference file** dialog box. The **Browse** button is only displayed after you click on the saved path.

Found Path
Specifies the path, including the location and file name, of the attachments. The Found path may or may not be the same as the Saved path.

35.2.2 Properties

Displays the properties of the selected attachment in a row format. These are the same properties displayed in the Attachments properties list header with a column format.

35.2.3 Document name

Displays the active document's name.

35.2.4 Attached files list

Displays a list of attached filesand their properties.

Note: When you select an XREF in the drawing area, it is highlighted in the **Attachment** panel and vice versa.

35.2.5 Attach button

Attach DWG
Displays the **Select Reference File** dialog box like the XATTACH command.

Attach PDF
Displays the **Select PDF Underlay File** dialog box like the PDFATTACH command.

Attach Image
Displays the **Select image file** dialog box like the IMAGEATTACH command.

Attach Point Cloud
Displays the **Point Cloud Reference Manager** dialog box like the POINTCLOUDREFERENCE command.

35.2.6 Refresh

Updates the Grid or Tree Views.

35.2.7 Detach

Detaches the selected attachments from the drawing.

35.2.8 Open

Opens the selected attachments.

35.2.9 Tree View

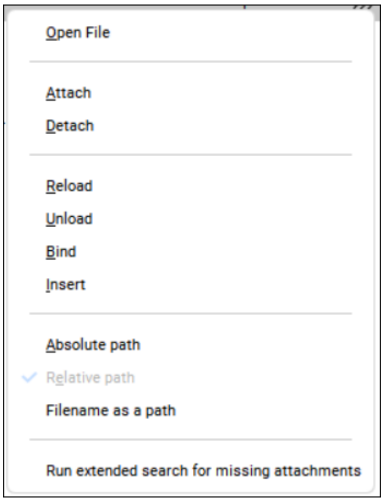
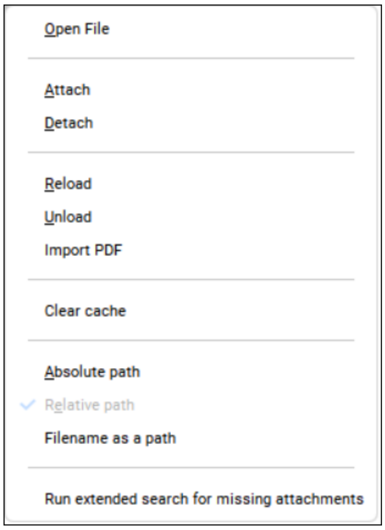
Displays the attachments in a structured tree view.

35.2.10 Grid View

Displays the attachments in a detailed grid view.

35.2.11 Context menu

A right-click menu on the selected attachments offers additional menu options depending on the file type:

DWG file	PDF file
	

Open file
Opens the DWG attachment file.

Attach
Attaches a new reference of the attachment file.

Detach
Detaches all references of the attachment.

Reload
Reloads all references of the attachment, and all its inserts, including parametric ones, are rebuilt.

Unload
Unloads all references of the attachment.

Bind
Copies the contents of the attached drawing to the current drawing as a block reference (-XREF: _B command), according to the value of the BINDTYPE system variable.
Parametric external references are converted to local parametric blocks.

Insert
Inserts all instances of the attachment in the drawing as block references (-XREF: _I command).

Parametric external references are converted to local parametric blocks.

Run extended search for missing attachments
Searches for the attached file in an upper level folder if the folder is added in SRCHPATH system variable.

Import PDF
Launches the -PDFIMPORT command to import the attachment.

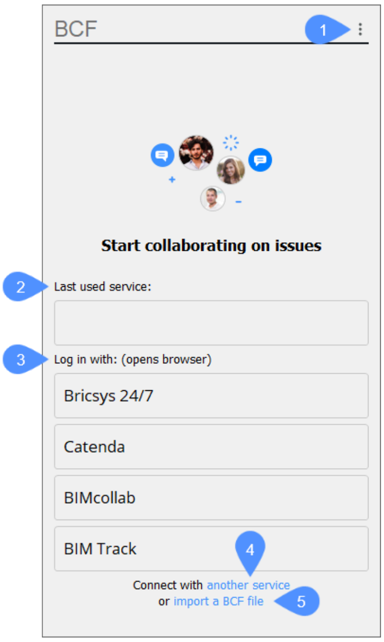
Clear cache
Removes the files from the related disk cache folder.

35.3 BIM BCF panel

To open the **BIM BCF** panel, right-click a ribbon panel/toolbar and choose **BIM BCF** from the **Panels** list.

The **BIM BCF** panel allows you to link information, such as a comment, a screenshot, a list of involved objects and a camera viewpoint to an issue and add new issues.

35.3.1 Login tab



1 Menu

2 Last used service

3 Log in with

4 Connect with another service

5 Import a BCF file

Menu
Opens the main menu of the **BIM BCF** panel.

Import BCF file
Allows you to import BIM Collaboration Format Files (*.bcf,*.bcfzip).

Log in with custom service
Allows you to connect to a cloud service to start collaborating on issues in the drawings.

Refresh
Makes sure you see the latest updates in case you are working on the cloud.

Clear sources
Disconnects you with the cloud service, in case you are online, or closes the BCF file, in case you are working on an imported BCF file.

Last used service
Displays the last used collaboration service whose address is stored by the BCFSOURCEURL system variable.

Log in with
Displays the **Bricsys 247** service and the three most popular collaboration services. Click the desired button to directly connect with that service.

Note:

- The **Bricsys 247** button connects you to the service whose address is stored by the CLOUDSERVER system variable (https://my.bricsys247.com by default).
- The Source URL for BIMcollab includes the name of the space you bought. If using a free account, use join.bimcollab.com.

Connect with another service
Opens the **Custom log in** dialog box (see **BIM BCF panel - Custom log in dialog box** article).

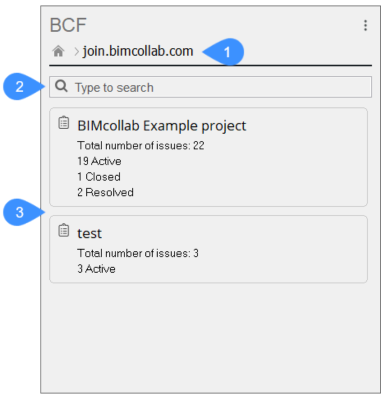
Import a BCF file
Displays the **Select a BIM Collaboration Format file to import** dialog box.

Note: This option only allows to read an imported BCF file. Edits cannot be saved.

35.3.2 Cloud service tab

Once you enter the desired service, you are connected to the selected cloud service. Insert the e-mail address and the password to connect.

After logging in, close the browser window and start collaboration on issues in BricsCAD.



1 Source URL

2 Search

3 Project list

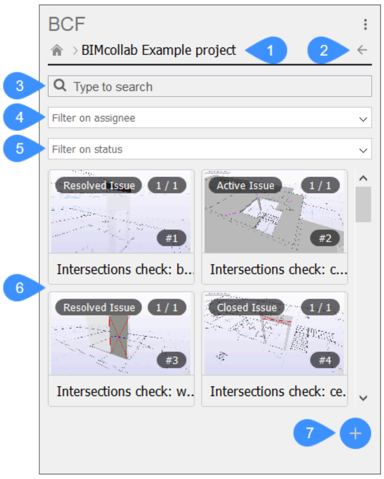
Source URL
Displays the source URL of the BCF collaboration service.

Search
Displays only the projects filtered by the word you type in the search bar.

Project list
Displays the list of projects that you are involved in.

35.3.3 Issue Overview tab

Displays the issues available for the selected project.



1 Project name

2 Project list

3 Search

4 Filter on assignee

5 Filter on status

6 Search report

7 Add issue

Project name
Displays the name of the opened project.

Project list
Returns to the project list.

Search
Allows you to specify a search in the issues list.

Filter on assignee
Allows you to filter issues on the assignee. All assignees in the project are listed in a drop-down menu.

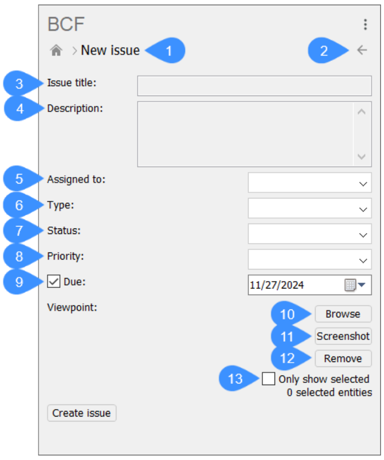
Filter on status
Allows you to filter issues on the status. Different statuses can be accessed through a drop-down menu.

Search report
Lists all issues that comply with the entered search.

Add issue
Allows to add and define a new issue.

Note: This option is only available when logged in with a BIM collaboration cloud service.

35.3.4 New issue tab



1 Issue name

2 Issue list

3 Issue title

4 Description

5 Assign to

6 Type

7 Status

8 Priority

9 Due

10 Browse

11 Screenshot

12 Remove

13 Only show selected

Issue name
Displays the name of the new issue.

Issue list
Returns to the issues list of the project.

Issue title
Defines the title of the issue to add.

Description
Defines a specific description of the issue.

Assign to
Defines the assignee of the issue.

Type
Defines the type of the issue. Several options can be accessed through a drop-down menu.

Status
Defines the status of the issue. Several options can be accessed through a drop-down menu.

Priority
Defines the importance of the issue. Several options can be accessed through a drop-down menu.

Due
Defines whether this issue has a due date to solve or not. If checked, a date can be specified.

Browse
Opens the **Select a viewpoint for this comment** dialog box to add a PNG file as viewpoint.

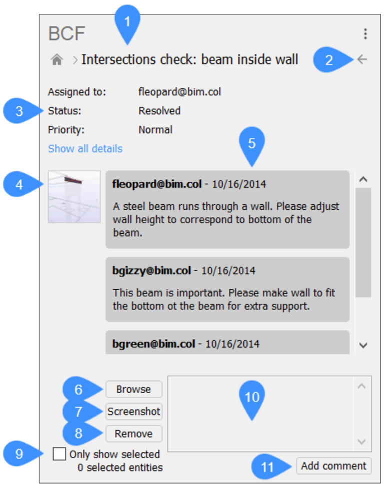
Screenshot
Insert a screenshot from model space as viewpoint. The screenshot image is based on the current visibility in model space. The option also takes into account the camera position and active section planes. These are reapplied later on when the viewpoint is activated.

Remove
Removes the added viewpoint.

Only show selected
When unchecked, entities that are selected during the screenshot creation are highlighted when activating the viewpoint later on.
When checked, only the selected entities are set to be visible in the viewpoint of the issue comment, allowing you to isolate the selected entities when activating the viewpoint.

35.3.5 Issue information

When you click an issue, the issue information is displayed.



1 Issue name

2 Issue overview

3 Details

4 Thumbnail image

5 Comments

6 Browse

7 Screenshot

8 Remove

9 Only show selected

10 Comment

11 Add comment

Issue name
Displays the name of the opened issue.

Issue overview
Returns you to the issue overview.

Details
Displays detailed information about the issue, such as status, author and creation date.

Thumbnail image
If the comment has an associated camera position, clicking the thumbnail image brings the camera in the current drawing to this position.

Comments
Displays the comments on the issue.

Browse
Opens the **Select a viewpoint for this comment** dialog box to add a PNG file as viewpoint.

Screenshot
Insert a screenshot from model space as viewpoint. The screenshot image is based on the current visibility in model space. The option also takes into account the camera position and active section planes. These are reapplied later on when the viewpoint is activated.

Remove
Removes the added viewpoint.

Only show selected
When unchecked, entities that are selected during the screenshot creation are highlighted when activating the viewpoint later on.
When checked, only the selected entities are set to be visible in the viewpoint of the issue comment, allowing you to isolate the selected entities when activating the viewpoint.

Comment
Allows you to insert a comment for the selected issue.

Add comment
Adds the comment (and viewpoint) you have inserted to the selected issue.

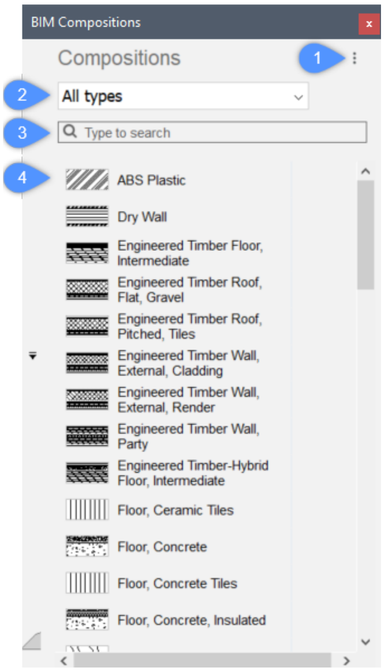
Note:

- Comments can only be added if a BCF cloud service was selected. They cannot be saved to BCF files since this options allows you to read a BCF file.
- BCF 2.1 and BCF 3.0 are supported when connecting to a BCF service through the BCF API. When importing BCF files, only BCF 2.1 is supported.

35.4 BIM Compositions panel

To open the **BIM Composition** panel, right-click a ribbon panel/toolbar and choose **BIM Composition** from the **Panels** list.

The **BIM Composition** panel allows you to assign compositions to any solid in a BIM model.



1 Menu

2 Compositions types list

3 Search

4 Available compositions

35.4.1 Menu

Open material dialog...
Opens the **Physical Material** dialog box that allows you to create, modify and delete physical materials.

Open composition dialog...
Opens the **Compositions** dialog box that allows you to create, modify and delete compositions.

Open project dialog...
Opens the **BIM Project Info** dialog box that allows you to control the location, properties, and content of the Project and Central databases.

Show only compositions in project
Displays only the compositions that are used in the project.

35.4.2 Compositions types list

Displays the compositions of the selected category (all types, generic, roof, slab, wall) in the Library database.

35.4.3 Search

Filters the list of compositions. Only compositions that contain the current search term are displayed.

35.4.4 Available compositions

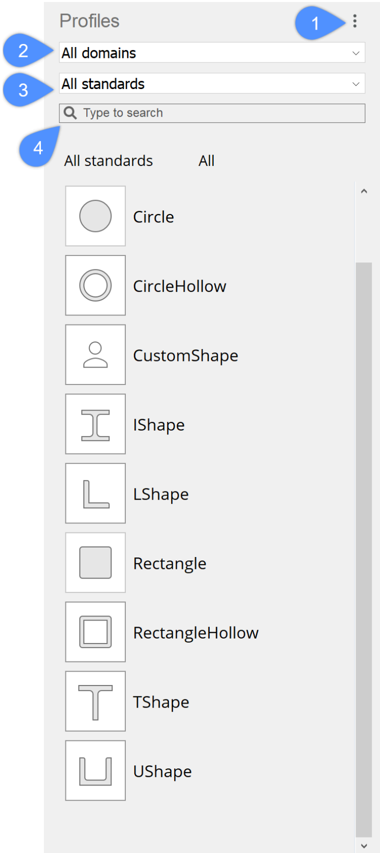
Displays all the available compositions.

When you double-click on a composition, the **Compositions** dialog box will open.

35.5 BIM Profiles panel

To open the **BIM Profiles** panel, right-click a ribbon panel/toolbar and choose **BIM Profiles** from the **Panels** list.

The **BIM Profiles** panel offers a central location to access standard and custom profiles.



1 Menu

2 Domains

3 Standards

4 Search bar

35.5.1 Menu

Open material dialog...
Opens the **Physical Material** dialog box that allows you to create, modify and delete physical materials.

Open profile dialog...
Opens the **Profiles** dialog box that allows you to create, modify and delete profiles

Open project dialog...
Opens the **BIM Project Info** dialog box that allows you to import profiles with XML/CSV files in the project database, control the location, properties, and content of the Project and Central databases.

Show only profiles in project
Displays only the profiles that are used in the project.

35.5.2 Domains

Filters profiles based on the domain.

35.5.3 Standards

Filters profiles based on the standard.

Note: You can set which standards will be visible in the **BIM Profiles** panel in the Settings (Profile's standards). By default all standards are visible.

35.5.4 Search bar

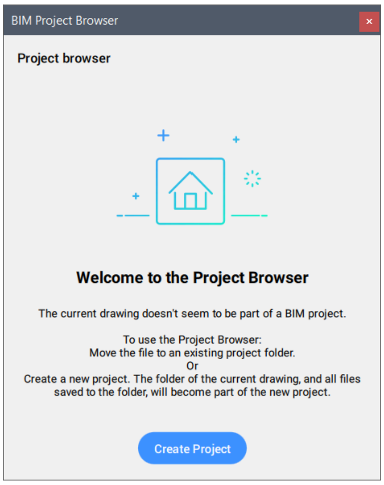
Search for specific profiles using the search bar.

35.6 BIM Project Browser panel

To open the **BIM Project Browser** panel, right-click a ribbon panel/toolbar and choose **BIM Project Browser** from the **Panels** list.

The **BIM Project Browser** panel offers a central location to manage all files and views in your BIM project.

Once a new drawing is saved, you can press the **Create Project** button.



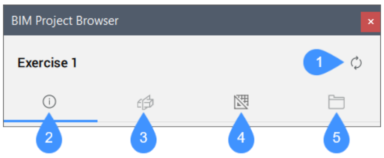
Note: When starting BricsCAD BIM, you can also create a new project from the **Start** tab by pressing the **New project** button.

When you save the new drawing in an existing project folder, the **BIM Project Browser** displays the **Add to Project** button. Click it to add the drawing and all it's necessary files that don't yet exist to the project folder.

During project creation, some files are automatically generated in the project folder, such as a sheetset file (DST), a database file (BSYSLIB) and a default sheet template (DWT). A new subfolder for the sheet templates is generated as well. Other files can be added manually to the project folder or can be generated later on.

Note: The folder in which the active drawing is stored becomes the project folder. All other files and subfolders of this folder will be considered part of this project.

The **BIM Project Browser** panel consists of four tabs and a refresh button:



1 Refresh project

2 Info tab

3 Sections and views tab

4 Sheets tab

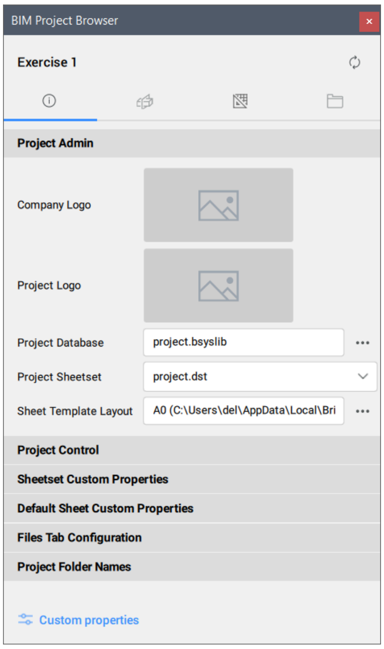
5 Files tab

35.6.1 Refresh project

Refreshes the **BIM Project Browser** panel to match changes in all BIM project (for example, Xref in the master drawings, sheets, manually added files).

35.6.2 Info tab

Offers a complete overview about the current project.



Project Admin
Shows and sets the:

- **Company Logo**: click the thumbnail to select a PNG file. A copy will be made in the project folder and will be renamed 'project.png'. For example, this can be used in a title block.

  Note: The company logo will automatically update in the sheet views of the currently open sheet drawings after saving the open sheet drawings.

- **Project Logo**: click the thumbnail to select a PNG file. A copy will be made in the project folder and will be renamed 'project.png'. For example, this can be used in a title block.
- **Project Database**: click the browse button (...) to open the **Project Database** dialog box that gives a complete overview of the project's drawings and their stored and found database paths. The dialog box allows you to select another project database or to rename the current one. It also supports connecting the listed drawings to the project database.
- **Project Sheetset**: displays the current sheet set. Click the drop-down arrow to select an available sheet set from the list.
- **Sheet Template Layout**: click the browse button (...) to open the **Select Layout** dialog box where you can select the template drawing file path and a layout from the template file.

Project Control
Sets and shows the sheet set properties:

- **Project Name**
- **Project Number**
- **Project Phase**
- **Project Milestone**

Sheetset Custom Properties
Sets and shows the sheet set properties defined by the user (for example, **Client City**, **Client Name**, **Client State**, **Client Street**, **Project City**, **Project State**, **Project Street**).

Default Sheet Custom Properties
Sets and shows the sheet properties defined by the user (for example, **Checked By**, **Drawn By**).

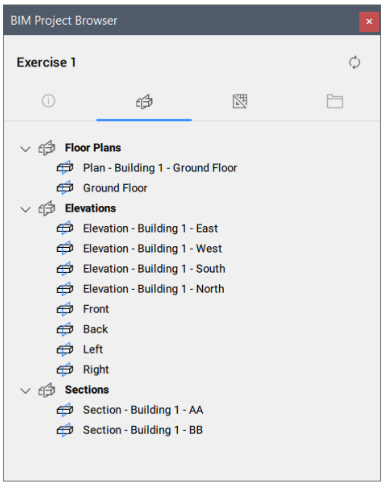
Files Tab Configuration
Defines the folders and extensions to ignore. It allows you to specify what part of the project you want to see in the **Files** tab.

Project Folder Names
Sets the names of the project folders for components, details, images and schedules.

Custom properties
Opens the **Custom Properties** dialog box where Sheet Set and Sheets custom properties can be added, edited or deleted.

35.6.3 Sections and views tab

Displays a list of project sections organized by type. You can expand a section type to see its sections.



Right-click a **Section** to display the context menu:

- **Select Section Entity**: selects the section entity in the project model drawing.

  Note: This can also be done by double clicking the section entity.

- **Rename**: renames the section entity.
- **Display Section Result**: displays the section result in the corresponding sheet and zooms in on its viewport. If not already open, the drawing is opened.
- **Create Sheets**: opens the **Create Sheets** dialog box that allows you to set the viewports and settings of the new sheet.

  Note:

  - If a sheet is already created for the selected section, a notification pop-up will appear indicating the name of the sheet. This sheet can be edited in the panel's **Sheets** tab. Double click it or select the **Edit Sheets** option from the sheet's context menu.
  - BIM project data is now initialized with an XML file that can be edited by users.

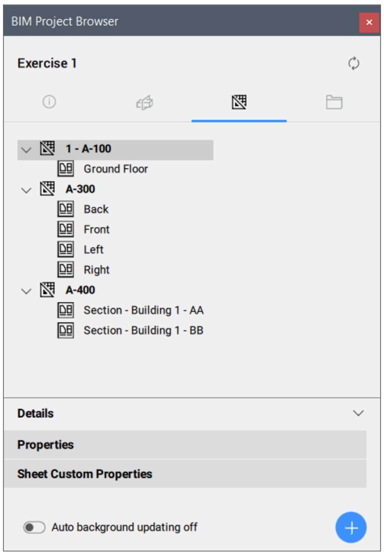
- **Update Now**: regenerates the section result.
- **Update in Background**: regenerates the section result in background, without interrupting the user workflow.

Note: You can place sections (single/multiple items selection) or 3D views (single item selection) on an active sheet's Paperspace by dragging items from the **Sections and views** tab tree and dropping them to that sheet's Paperspace:

- For single item selection, specify the insertion point. When placing a 3D view, right click before specifying the insertion point to display a context menu that allows you to select the viewport scale.
- For multiple items selection, specify first the insertion point, then the second point of a rectangle that defines how the items are arranged (how many rows/columns).

35.6.4 Sheets tab

Displays the contents of the project's sheetset. You can expand a sheet name to see its sheet views.



Note:

- Double clicking a sheet will open the sheet drawing.
- You can move a sheet view from one sheet to another by dragging and dropping it above the sheet's name. You can hold down the **Ctrl** key to drag a copy of that sheet view.

Details

Click the section name to expand it. The section will display the relevant properties for the selected items in the tab. If not grayed out, you can edit these properties.

Auto background updating

Updates the section results automatically, without interrupting the your workflow. It toggles the ENABLE-BIMBKUPDATE system variable.

Add button

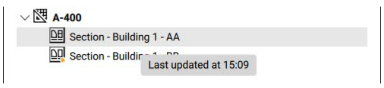
By pressing the **Add** button the below options are available:

- **New Sheet**: opens the **New Sheet(s)** dialog box which allows you to add a new sheet to the **Sheets** tab or to the currently selected subset, if any.
- **New Subset**: adds a new subset to the **Sheets** tab or to the currently selected subset, if any.

Status of the sheet views:

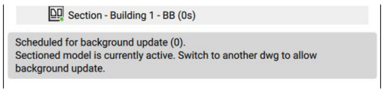
- : the sheet view is updated.

  Note: When you hover over the sheet view, a tooltip appears explaining the status and indicating the time the section was last updated:

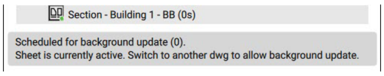
  

- : the sheet view is background updated.

  Note: When you hover over the sheet view, a tooltip explaining the status appears:

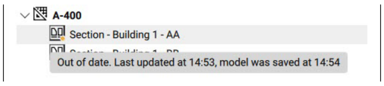
- When a 3D model is active:
- 

Note: **(n)** shows the order in which background update will operate (is the index of the view in the queue).

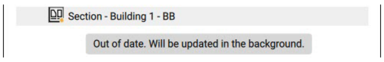
- When a sheet is active:
- 

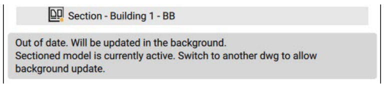
- : the sheet view is out of date.

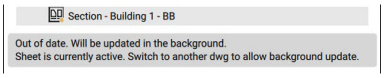
  Note: When you hover over the sheet view, a tooltip explaining the status appears. It also indicates the time the section result block was last updated as well as the time the model was saved. In this way you can check whether the save time of the model is more recent than the update time of the sheet view:

  when background update is on:

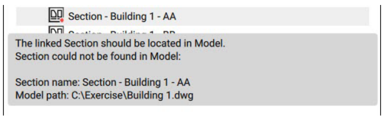
  - When a 3D model is active:
  - 

  - When a sheet is active:
  - 

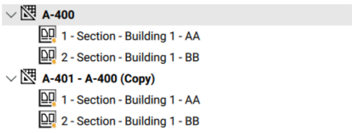
- : the sheet view has an error.

  Note: When you hover over the sheet view, a tooltip explaining the status appears:

Right-click a **Sheet** to display the context menu:

- **New Sheet(s)...**: opens the **New Sheet(s)** dialog box that allows you to add a new sheet to the **Sheets** tab.
- **Open Sheet**: opens the sheet drawing. You can also double click the sheet to open it.
- **Copy Sheet**: makes a copy of the sheet dwg and the sheet entry in the sheetset (under the same subset). All blocks, viewports, named views and sheet views will be copied as well. The naming and numbering of the copied sheet will be handled automatically.



- **Edit Sheet**: opens the **Edit Sheets** dialog box that allows you to edit the properties of the selected viewports.

  Note: There is always an entire sheet opened in the dialog box. If you select a single sheet view, all the viewports of that sheet are listed and when a subset is selected, all sheets in the subset are shown.
- **Remove Sheet**: removes the sheet.
- **Collect Details**: collects all tagged details from the selected sheets and inserts the layouts of these details in the current sheet.
- **Update Now**: cancels the queue that is currently running to update sheets and schedules the selected set of sheets as a new queue. The background updating continues once this queue finished updating.
- **Update in Background**: updates the sheets view in background.

Right-click a **Sheet view** to display the context menu:

- **Display View**: displays the corresponding result and zooms in on its viewport.
- **Edit Sheet**: opens the **Edit Sheets** dialog box that allows you to edit the properties of the selected viewports.
- **Remove View/Result**: removes the sheet view and corresponding result (viewport, viewlabel, sheetview, and result block, but only when no other viewport views it).
- **Update Now**: updates the sheet view.
- **Update in Background**: updates the sheet view in background.
- **Place view label**: insert a view label in the current sheet and establish the link between view label and viewport. The new view label replaces the old one if the view label already exists.

  Note: This action is available only when the current dwg tab is an opened sheet document.

You can easily place a sheet view to another active sheet's paperspace by dragging it from the **Sheets** tab and dropping it on the paperspace.

Right-click a **Subset** to display the context menu:

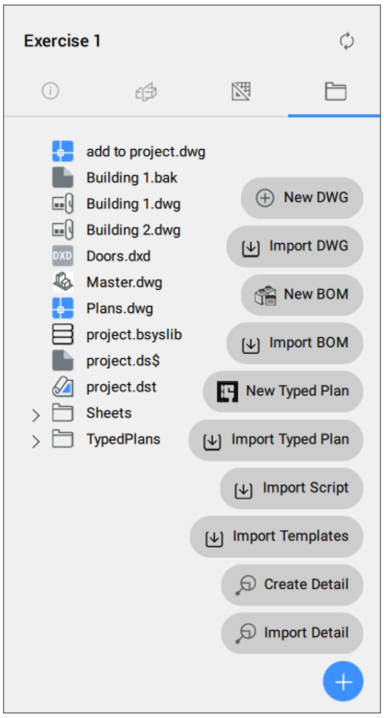
- **Remove Subset**: removes the subset (only possible when it doesn't contain any sheets/subsets).

35.6.5 Files tab

Shows the contents of the project folder and allows you to place all supported files (by drag and drop) on an active drawing:

- Drawings (models, sheets, ...)
- Schedules, scripts, ...
- Images:
  - Company logos, googlemap images, site photos, etc.
  - PNG created by script based on named views.

- Renders created in Bricscad or Enscape, Twinmotion.



Note: Some of the supported files for import can only be placed in the active drawing's Paperspace.

Add button (+) options:

New DWG
Opens the **Choose name** dialog box to add a new DWG drawing to the file.

Import DWG
Opens the **Select dwg to import** dialog box to import an existing DWG into the files.

New BOM
Opens the **Where should the file be created?** dialog box to create a new BOM table template.

Import BOM
Opens the **Select BOMs to import** dialog box that allows you to import the BOMs.

New Typed Plan
Opens the **BIM Typed Plan Editor** dialog box that allows you to create a drawing customization.

Import Typed Plan
Opens the **Select a Typed Plan to import** dialog box to import an already defined drawing customization.

Import script
Opens the **Select Scripts to import** dialog box from which can import a Python, Lisp or Grasshopper script.

Import Templates
Opens the **Select templates to import** dialog box from which you can choose a template to import.

Create Detail
Launches the BIMCREATEDETAIL command.

Import Detail
Opens the **Select DWG to import as detail** dialog box from which you can choose a drawing to import as detail.

Right-click a **Detail** to display the context menu:

- **Check details**: launches the BIMCHECKDETAILS command.
- **Open**: opens the detail file.
- **View detail**: opens the **View Detail** dialog box.
- **Rename**: allows you to rename the detail file.
- **Unmark as detail**: unmarks the file as a detail.
- **Add references**: launches the BIMADDDETAILREFS command to create a link between the master model and the detail file (option available for basic details).
- **Propagate**: launches the -BIMPROPAGATEFROMFILE command (option available for advanced details).
- **Duplicate**: creates a duplicate file of the detail.
- **Delete**: deletes the detail file.
- **Rename**: renames the detail file.

Right-click a **Sheet** to display the context menu:

- **Duplicate**: creates a duplicate of the sheet.
- **Delete**: deletes the selected sheet.
- **Rename file**: opens the **Rename Sheet** dialog box that allows you to rename the sheet.

  Note: If the linked sheets contain the name of the file, the linked sheets' number and title are automatically renamed. You can override the automatic renaming before closing the dialog box.

Right-click a **DWG** to display the context menu:

- **Mark as master model**: makes the drawing a master drawing. This means any sections or 3D views defined in the drawing or any of its Xrefs will be listed in the **Sections and views** tab.

  Note: It is allowed to have multiple master drawings.

- **Duplicate**: creates a duplicate of the drawing.
- **Delete**: removes the drawing from the files explorer.
- **Rename file**: opens the **Rename drawing** dialog box where you can enter a new name for the drawing.

Note: You can drag and drop a DWG or a DXF file from the **Files** tab into the current drawing. You can then choose the insertion method (copy the file content or insert it as a block or xref).

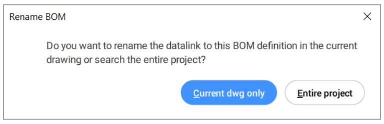
Right-click a **Master model** to display the context menu:

- **Unmark as master model**: the selected model is no longer a Master model.
- **Duplicate**: creates a duplicate of the Master model.
- **Delete**: deletes the selected Master model.

- **Rename file**: opens the **Rename drawing** dialog box that allows you to rename the Master model. All related are automatically renamed as well.

Right-click a **BOM file** in the **Schedule** folder to display context menu:

- **Open file**: opens the Schedule file on a BOM reader on your computer where you can edit and save the file.
- **Generate csv**: generates a CSV file from the BOM file. The CSV file is located in the same schedules folder as the original BOM file.
- **Generate xlsx**: generates an XLSX file from the BOM file. The XLSX file is located in the same schedules folder as the original BOM file.
- **Duplicate**: creates a duplicate of the BOM file.
- **Delete**: removes the BOM file.
- **Rename file**: opens the **Rename BOM file** dialog box where you can enter a new name for the file. Click **Rename** to open the **Rename BOM** dialog box. Here you can choose to rename the file in the **Current dwg only** or in the **Entire project**.



Note: Renaming the datalink to the BOM definition in the entire project can take longer than renaming it in the current drawing.

Note: The **BIM Project Browser** panel supports the old schedules workflow. You can add DXD files manually to the **Schedules** folder. When a DXD file is present in the **Schedules** folder, its context menu displays the same options as for a BOM file.

- Right-click a CSV or XLSX file to display the context menu:
  - **Create table**: inserts the table in the current sheet using a Datalink.
  - **Duplicate**: creates a duplicate of the CSV or XLSX file.
  - **Delete**: deletes the selected CSV or XLSX file.

    Note: when you double click the CSV or XLSX file the file opens in an application set by the system.
  - **Rename file**: opens the **Rename file** dialog box where you can enter the new name for the CSV or XLSX file.

Note: You can place a BOM schedule on a currently opened sheet’s Paperspace by dragging the BOM file from the **Schedules** folder in the **Files** tab tree and dropping it onto that sheet's Paperspace. This generates a table with a datalink to the BOM file. You can update the BOM schedule later on using the DATALINKUPDATE command.

Right-click a **XREF file** to display the context menu:

- **Duplicate**: makes a copy of the selected XREF file in the project folder.
- **Delete**: removes the XREF file from the project folder.

Right-click a **Script file** to display the context menu:

- **Python files:**
  - **Run script**: executes the python script.
  - **Open with default program**: opens the python file (PY) with default program.
  - **Duplicate**: creates a duplicate of the PY file.
  - **Delete**: removes the python file.
  - **Rename file**: opens the **Rename file** dialog box where you can enter the new name for the file.
- **LISP files:**
  - **Load script**: loads and executes the lisp script.
  - **Duplicate**: creates a duplicate of the LISP file.
  - **Delete**: removes the lisp (LSP) file.
  - **Rename file**: opens the **Rename file** dialog box where you can enter the new name for the LISP file.
- **Any other file type**: double click to open the file with an application set by the system.

Right-click a **Typed plan file** to display the context menu:

- **Open folder**: opens the folder on your computer where the typed plan is located.
- **Duplicate**: creates a duplicate of the typed plan.
- **Delete**: removes the typed plan file.

  Note: Double click a typed plan to open the **BIM Typed Plan Editor** dialog box that allows you to edit it.

- **Rename file**: opens the **Rename typed plan** dialog box where you can enter a new name for the file.

35.7 BOM Manager panel

To open the **BOM Manager** panel, right-click a ribbon panel/toolbar and choose **BOM Manager** from the **Panels** list.

The **BOM Manager** panel allows you to browse, create and edit bills of materials (BOMs) in the current drawing.

The configuration of a bill of materials can be saved as a template file and reused later in other drawings.

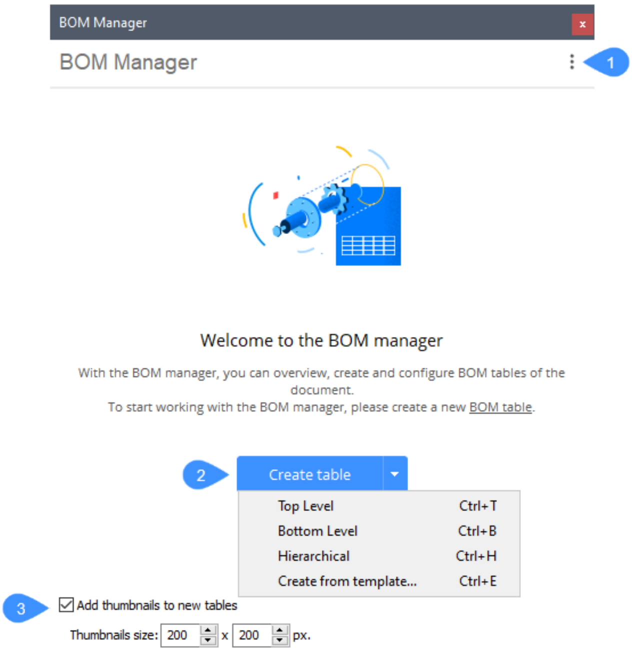
With the **Bill of Materials Manager**, you can:

- Browse existing bills of materials in the drawing.
- Create a new bill of materials using either a standard type (such as top-level, bottom-level, hierarchical) or an existing BOM template.
- Edit bills of materials.
- Add or remove columns (includes formula columns).
- Move selected columns to new positions.
- Rename the columns.
- Configure how the BOM table is sorted.
- Configure the footer value for each individual column.

- Configure the units and how they are displayed for each column.
- Configure the width of each column with column properties.
- Configure the visibility of each individual column.
- Filter the components with a filter expression.
- Change the title of both the title row and the footer row.
- Search for components in the table.
- Highlight and select component instances.
- Save the BOM configuration as a template file.
- Load a BOM table configuration from a template file.
- Place the BOM table in a drawing.
- Delete BOM tables.
- Configure which kind of objects should be present in the BOM table with BOM filtering mode.
- BOM can include parts from other documents to the selection set.
- BOM can exclude the current document to the selection set.
- ***Note:*** When a BOM table is created in a file that is a part of a BIM project, the filter settings, grouping mode and property set will be automatically extended to include non-mechanical objects, even if there are mechanical components in the model. In addition, the current project will be included in the BOM, and the current document will be excluded (as it will be added as a part of BIM project).

A Mechanical or Ultimate license is required to modify or create bills of materials. However, you can launch the **BOM Manager** using BMBOMTEMPLATEEDIT command, that requires either BIM, Mechanical or Ultimate license.

Note: BOM manager allows you to create a BOM table in any document, not necessary containing a mechanical assembly or a BIM model. In that case, the filter settings will be automatically adjusted to include all block references, xrefs and solids in the model.



1 Menu

2 Create a table

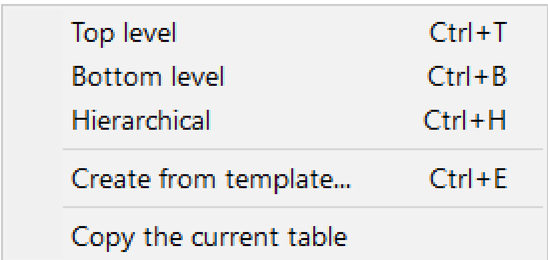
3 Add thumbnails to new tables

35.7.1 Menu

Opens the main menu.

New BOM

Opens a submenu.



Note: The BOMTEMPLATE system variable defines the path to the file with default BOM template which will be used to create new BOM tables.

Note: If the BOMTEMPLATE system variable is empty or the path is not valid, the below default BOM configuration will be used.

- **Top level**: Creates a new, top-level components only, BOM.
- **Bottom level**: Creates a new, bottom-level components only, BOM.
- **Hierarchical**: Creates a new hierarchical BOM, which contains all components with their subcomponents listed directly below.

- **Create from template**: Opens a dialog box to select a BOM configuration template file.
- **Copy the current table**: Adds a new BOM table using the template of the current table.

BOM tables in document
Opens a submenu.

- **Go to first table**: Switches to the first BOM table in the document.
- **Go to previous table**: Switches to the previous BOM table in the document.
- **Go to next table**: Switches to the next BOM table in the document.
- **Go to last table**: Switches to the last BOM table in the document.
- **<Name of the BOM>**: Switches to the corresponding BOM table in the document.

Place BOM table
Starts the interactive placement of the table in the drawing. Multiple instances of a BOM table can be inserted into the document.

Export to CSV
Saves the content of the current BOM table to a CSV file.

Export to Excel
Saves the content of the current BOM table to a XLSX file.

Replace from template…
Replaces the current BOM configuration with the one from the template.

Save as template…
Specify the location of the BOM template file.

Save as default template
Specify the location of the default BOM template file, stored in BOMTEMPLATE system variable, which will be used to create new BOM tables.

Delete current BOM
Deletes the current BOM table from the document.

Delete all BOMs
Removes all BOM tables.

Set filter
Opens the **Filter Configurator** dialog box, which allows you to specify the filter configuration. For more information, see the related article **Filter Configurator dialog box**.

Selection set
Opens a submenu.

- **Entire model**: The BOM table will be created for the entire model.
- **Subassembly**: The BOM table will be created for a selected instance of a subassembly.
- **Drawing view**: The BOM table will be created for the parts selected for the corresponding drawing view, regardless of their visibility in the viewport.
- **Layout viewport**: The BOM table will be created for the selected layout viewport.
- **Custom selection**: The BOM table will be created only for the selected parts.
- **Exclude current document**: The BOM table will not include objects from the currents document.

- **Show selection**: Shows the current selection for the BOM table.
- **Additional documents**: The BOM table will include parts from other documents.
    - **Add drawing or sheet set**: The BOM table will include objects from the selected drawing or sheet set.
    - **Add folder**: The BOM table will include objects from drawings or sheets set which are present in a folder.

      Note: The user may include only the documents that match specified wildcards, entire BIM project or entire sheet set.
- **Edit**: Allows you to edit the selection.

Refresh
Updates the current BOM and the list of all BOM tables in the drawing.

Update all
Updates all BOM tables.

Insert column after
Inserts a column from the **Column selector** after the selected column.

Insert column before
Inserts a column from the **Column selector** before the selected column.

Note: If no column is selected, a new column is added after the last column in the table.

Insert formula column
Opens a formula editor, which allows you to configure a formula column (see **Formula Column dialog box** article).

Insert template column
Opens a template editor, which allows you to configure a template column (see **Template Column dialog box** article).

Insert user fillable column
Inserts an user fillable column which can be used in formula and template columns.

Note: You can double-click any cell in a user-fillable column, a quantity column, or any other column excepts the **Number**, **Thumbnails**, and **Component** columns to insert or edit unitless values (numbers or text). The overridden values are displayed with a different font (Italic font and brand-blue colour) in the **BOM Manager**, and the BOM columns have a pen symbol in front of their title.

Edit evaluated column
Allows you to edit the selected formula or template column.

Insert thumbnails column
If a column is selected: Inserts a thumbnail column with default parameters after the selected column in the table.
If no column is selected: A new thumbnail column is added after the last column in the table.

Clear unused thumbnails
Removes blocks and attachments with thumbnail images that are no longer used.

Clear unused thumbnails with files
Removes blocks, attachments and files with thumbnail images that are no longer used.

Delete column
Removes the selected column from the table.

Move column left
Moves the selected column(s) one column to the left.

Move column right
Moves the selected column(s) one column to the right.

Make column visible
Shows the selected column in the associated table in the drawing.

Make column invisible
Removes the selected column from the associated table. Invisible columns remain visible in the BOM manager but are marked with a diagonal hatch pattern in the background.

Associated table

Synchronize associated tables (Alt + Ctrl + R)
Synchronizes the associated tables.

Hovering over a table name opens a submenu.

- **Fixed corner**: Sets the fixed corner table to Top left, Top right, Bottom right or Bottom left.
- **Zoom to**: Zooms to the table. If necessary, sets the table layout as the current one.
- **Select**: Selects the table. If necessary, sets the table layout as the current one.
- **Delete**: Deletes the table from the document.
- **Detach**: Detaches the table. The table in the document is no longer updated when the BOM table is modified.
- **Fit row height**: Sets the row height that fit the text.

Footer
Opens a submenu, for each of the selected columns.

- **No footer**: Removes the footer function.
- **Sum**: Displays the sum in the footer.
- **Average value**: Displays the average in the footer.
- **Minimal value:**: Displays the minimum value in the footer.
- **Maximal value:**: Displays the maximum value in the footer.

Note: Only user-fillable columns and the quantity column with defined footers accept user values.

Note: It is possible to add footers to formula or template columns as well.

Sorting
Opens a submenu.

- **Natural order**: Sort the table by the natural order of component instances in the document.
- **Automatic order**: Sorts the table automatically.

- **Custom order**: Sorts the selected columns in a manually order defined by the user. When parts manually arranged, their numbers are automatically regenerated. The numbers of child parts are updated accordingly.
- **Fixed order**: The positions of parts in the BOM are remembered, and their numbers stay the same after update, regardless of added or deleted parts.

  Note: When the sorting mode is changed back to **Auto**, **Natural** or **Custom**, numbers are regenerated automatically.

Aggregate function
Opens a submenu.

- **No function**: Removes the aggregate function.
- **Sum**: Displays the sum of all values in the group.
- **Average value**: Displays the average of all values in the group.
- **Minimal value**: Displays the minimum of all values in the group.
- **Maximal value**: Displays the maximum of all values in the group.
- **Total count**: Displays the total number of encounters with the same value among the groups.
- **Average count**: Displays the average number of encounters with the same value among the groups.
- **Minimal count**: Displays the minimum number of encounters with the same value among the groups.
- **Maximal count**: Displays the maximum number of encounters with the same value among the groups.
- **Least frequent value**: Displays the value with least corresponding counts. If there are more than one value with the minimal count, then all those values are displayed.
- **Most frequent value**: Displays the value with most corresponding counts. If there are more than one value with the maximal count, then all those values are displayed.
- **Concatenation**: Displays a concatenation of all values in the group in ascending order, separated by a specified delimiter.
- **Concatenation with count**: Displays a concatenation of all values in the group with their corresponding number of encounters in ascending order. Value and count pairs are separated by a specified delimiter.

Add thumbnails to new tables
Changes the setting to add thumbnail column to any new BOM table.

Display warnings
Controls the display of warning messages during the creation or updating of BOM tables.

Set default template
Sets a default template file for the BOM table.

35.7.2 Create a table

Note: The BOMTEMPLATE system variable defines the path to the file with default BOM template which will be used to create new BOM tables.

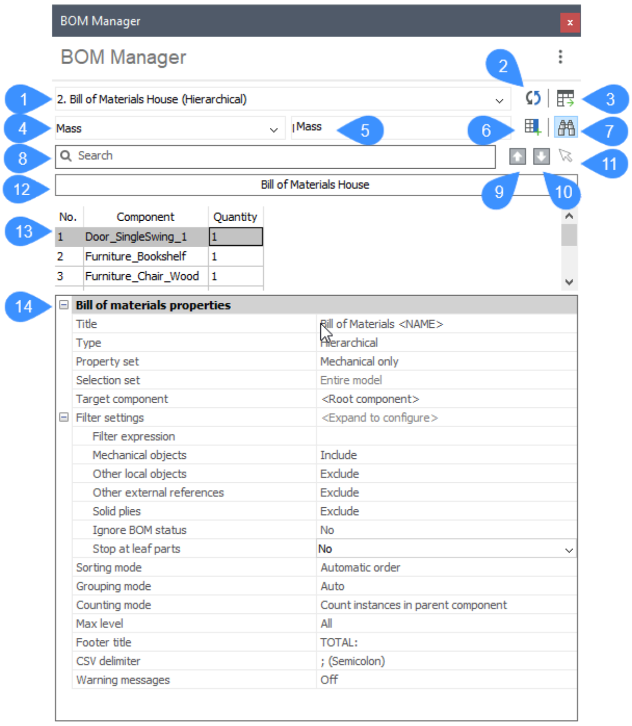
Note: If the BOMTEMPLATE system variable is empty or the path is not valid, the below default BOM configuration will be used.

Top level
Creates a new Top level components only BOM.

Bottom level
Creates a new Bottom level only components BOM.

Hierarchical
Creates a new Hierarchical BOM, which contains all components with their subcomponents listed directly below.

Create from template
Opens a dialog box, which allows you to select a BOM Table template (*.BOM) file.



1 BOM Selector

2 Refresh

3 Place BOM table

4 Category selector

5 Column selector

6 Insert column after

7 Find

8 Search

9 Find previous

10 Find next

11 Select all found

12 BOM title

13 BOM table

14 Bill of materials properties

BOM Selector
Selects the current BOM table. The selection is stored in the drawing. When the drawing is closed and reopened, the selected BOM is restored in the manager. When selected, the table is updated in the current state of the drawing.

Refresh
Updates both the current BOM and the list of all BOM tables in the drawing.

Place BOM table
Starts the interactive placement of the table in the drawing. Multiple instances of a BOM table can be inserted into the document.

Category selector
Select the category of the properties that are displayed in the column selector. The category can be added to the current BOM pressing button no. 9.

Column selector
Select the column to add to the current BOM.

You can type here the formula for a formula field. To add a formula, type an expression starting with ”=”. To add a template expression type “$”, to refer columns, use <>. For example, to refer to the Mass column you will type: “=<Mass>”.

When you add a formula field and press button no. 9 the Formula column_1 dialog box will appear.

Insert column after
If a column is selected, it will insert the column from the Column selector after the selected column in the table.

If no columns are selected, the new column will be inserted after the last column in the table.

Find
Toggles the search field on and off. When you toggle on this button, buttons no. 11, 12, 13 and 14 will appear.

If you click on the **BOM Manager** panel, you can toggle the search field using **CTRL+F**.

Search
Type a string to search and press Enter; wildcards are supported.

Find previous
Selects the previous match on the BOM table.

You can also use **Shift+F3**.

Find next
Selects the next match in the BOM table.

You can also use **F3**.

Select all found
Selects all the items searched in the search field.

BOM title

Displays the title of BOM table. You can type the name you want instead of <NAME>. Press Enter or exit the field to apply the changes.

BOM table

Displays the content of the BOM table.
Right-click a column label offers additional tools.

- **Add template column from selected**: Opens a template editor to configure a new template column. The selected columns will be automatically added to the template expression.
- **Delete**: Deletes the column.
- **Footer**: Opens a submenu.
  - **No footer**: Removes the footer function.
  - **Sum**: Displays the sum in the footer.
  - **Average value**: Displays the average in the footer.
  - **Minimal value:**: Displays the minimum value in the footer.
  - **Maximal value:**: Displays the maximum value in the footer.
- **Sorting**: Opens a submenu.
  - **Automatic order**: Sorts the table automatically.
  - **Natural order**: Sort the table by the natural order of component instances in the document.
  - **Ascending order**: Sorts the selected columns in ascending order.
  - **Descending order**: Sorts the selected columns in descending order.
- **Aggregate function**: Opens a submenu.
  - **No function**: Removes the aggregate function.
  - **Sum**: Displays the sum of all values in the group.
  - **Average value**: Displays the average of all values in the group.
  - **Minimal value**: Displays the minimum of all values in the group.
  - **Maximal value**: Displays the maximum of all values in the group.
  - **Concatenation**: Displays a concatenation of all values in the group in ascending order, separated by a specified delimiter.
  - **Concatenation with count**: Displays a concatenation of all values in the group with their corresponding number of encounters in ascending order. Value and count pairs are separated by a specified delimiter.
- **Column role**: Opens a submenu.
  - **Number**: The column will be used as a source of numbers by the BMBALLOON command.
  - **Name**: The column will be used as a source of names by the BMBALLOON command.
  - **Quantity**: The column will be used as a source of quantities by the BMBALLOON command.
  - **Regular**: The column will have no specific role.
- **Format**: Opens the **Format** dialog box, which allows you to specify the format for the selected column(s). For more information, see the related article **Format dialog box**.

- **Set column filter**: Opens the **Column Filter** dialog box, which allows you to configure the column filter. For more information, see the related article **Column Filter dialog box**.
- **Make column visible**: Shows the selected column in the associated table in the drawing.
- **Make column invisible**: Removes the selected column from the associated table. Invisible columns remain visible in the BOM manager but are marked with a diagonal hatch pattern in the background.
- **Fit column(s)**: Fits the width of the selected column(s) to their content.
- **Fit all columns**: Fits the width of all columns to their content.

  Note: The column width automatically fits the content unless the width has been changed manually.

Right-click on cells offers additional tools:

- **Zoom to**: Zooms to the select part(s) in the model.
- **Select**: Selects the part(s). Replaces an existing selection.
- **Add to Selection**: Adds part(s) to the selection.
- **Remove from Selection**: Removes the parts from the selection.
- **Remove highlighting**: Removes highlighting.

Bill of materials properties

Displays the BOM and columns Properties. The columns properties will appear after you insert a column on the BOM table.

Note: To remove the title or header row, the BOM title or all column headers must be empty. Also, titles entered from a command line will be trimmed to remove all leading or trailing spaces.

35.7.3 Add thumbnails to new tables

Changes the setting to add thumbnail column to any new BOM table.

You can set the width and height of the thumbnails in the **Thumbnails size** section.

35.8 Bricsys 24/7 panel

To open the **Bricsys 24/7** panel, right-click a ribbon panel/toolbar and choose **Bricsys 24/7** from the **Panels** list.

The **Bricsys 24/7** panel helps team members to find and share their drawings.

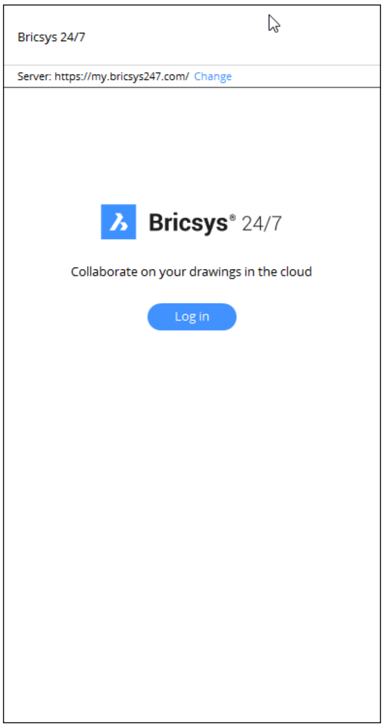
Bricsys 24/7 enables project management, a collaboration between project teams, cloud computing security and task automation. The big advantage of this cloud-based service is that BricsCAD is directly linked to it by the **Bricsys 24/7** panel.

On certain environments, the **Bricsys 24/7** panel supports Single Sign On (SSO).

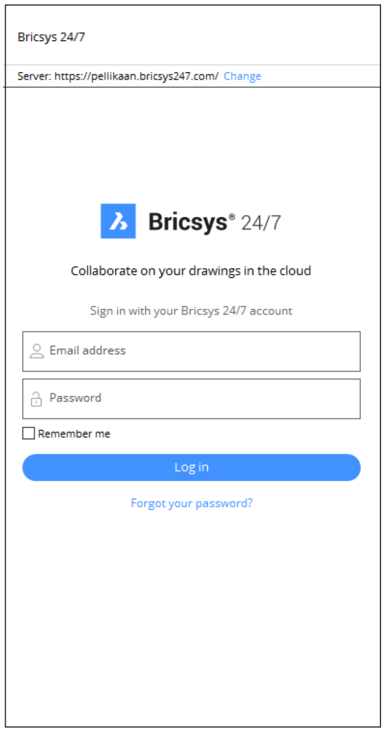
The CLOUDSSOCLIENTID system variable stores the client_id to be used when connecting to SSO service.

The CLOUDSSOSCOPE system variable stores the scopes or permissions to be used when connecting to SSO service. It depends on the authorization server for a given Bricsys 24/7 service (what scopes are configured/allowed for the client). For example, Azure AD requires an explicit client id on the scope parameter, like **<client id>/access_as_user openid**.

On servers like *https:/my.bricsys247.com/*, with default scope settings, the log in appears as described below.



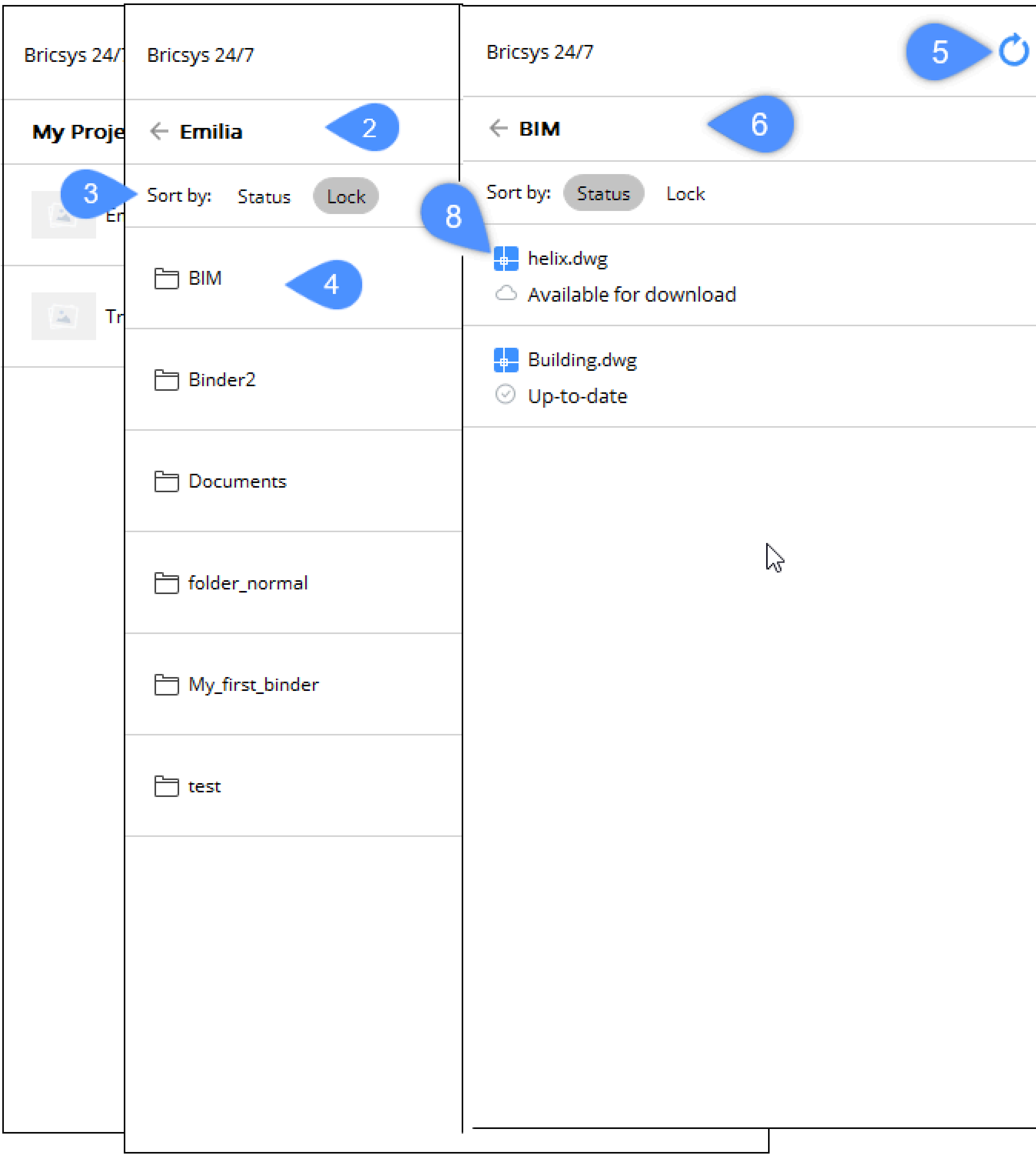
By clicking **Change**, the **Settings** dialog box is opened at the CLOUDSERVER system variable to specify another address of the Bricsys 24/7 server. For other certain environments, like *https://pellikaan.bricsys247.com/*, that does not support SSO at this moment, the log in will appear as described below.



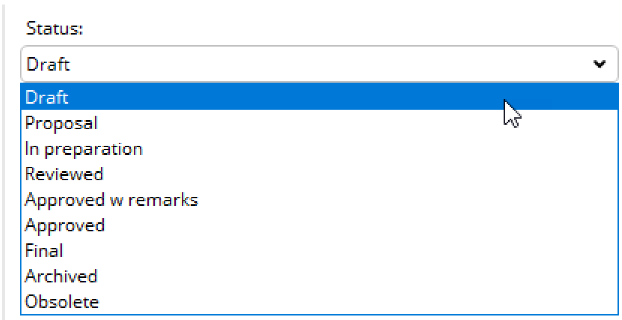
1 Enter your Email address and Password and press the **Log in** button.

2 (Optional) Tick the **Remember me** checkbox to store the username and password.

3 Once logged in, you will see an overview of your **in the cloud** Projects, Folders and Documents.



(1)	**Project**	When logged in, the first thing you see on the **Bricsys 24/7** panel is all the projects you are assigned to.

(2)	**Project name**	At the top of the panel you see the project name. You can go back to the overview of projects by clicking on the back arrow.
(3)	**Sort by**	Displays the folders/documents sorted by status of the file or lock attribute.
(4)	**Folder**	When you open a project, a list of folders is shown.
(5)	**Refresh**	Manually refreshes the **Bricsys 24/7** panel. The panel refreshes automatically when you switch between folders.
(6)	**Folder name**	At the top of the panel you see the folder you are in. You can go back to the overview of by clicking on the back arrow.
(7)	**Add a new file**	Opens the **Add new document** dialog box to add a new file to the folder. Choose a file status from the drop-down list and type a description.  ***Note:*** If the folder where you upload the file contains metadata, a form appears and you can fill it out.

(8)	**Documents**	In a folder you find the documents. Some details about the document are displayed as icons: : The profile picture or the initials from the author of the document. Available for download : The document is in the cloud, available for download. Up-to-date : The document is up to date. Locally modified : The document was locally modified. Newer version available : A new version of the document is available in the cloud. File is open/read-only : Someone is currently working on the document. Document is locked : The document that you try to modify is locked by someone else. Conflicted : Appears when you try to save changes in a read-only file or save changes in an old file version. Document already exists : Appears when you try to upload a file that already exists. Uploading... : The document is currently uploading. Downloading... : The document is currently downloading. : The document is locked by someone, another person cannot change it. ***Tip:*** If you hover the cursor over a document, a tooltip displays and you can see which is the latest version uploaded to the Bricsys 24/7 server.

You can download a BIM model from your Bricsys 24/7 project to a local folder by right-clicking the folder. The local folder for download is defined with the CLOUDDOWNLOADPATH system variable.

Download

4 Right-click a document to open a menu with possible document actions:

5 

(9)	Locks the document so no changes can be made.
(10)	Unlocks the document so changes can be made.
(11)	Downloads the latest revision from the cloud.
(12)	Uploads your drawing as a revision of the original drawing. See the CLOUDUPLOADDEPENDENCIES system variable options.

35.8.1 Uploading and downloading drawings with Xrefs

Uploading drawings with Xrefs

If you add a drawing that contains external references (Xrefs), make sure you also add the reference files to the Bricsys 24/7 server.

To do that, you can:

1. Upload the files one by one using the **Bricsys 24/7** panel.
2. Use the **Upload to Bricsys 24/7** option in the **Drawing Explorer** > **Dependencies** dialog box. Make sure you include the root drawing and the external references.

   Note: The **report.txt** file is uploaded as well. The report contains a complete list of all files included and notes for distribution.

Downloading drawings with Xrefs

If you download a drawing that contains Xrefs, the reference files are automatically downloaded.

35.9 Calculator panel

To open the **Calculator** panel, run the QUICKCALC command or right-click a ribbon panel/toolbar and choose **Calculator** from the **Panels** list.

The **Calculator** panel provides a way to perform calculation inside BricsCAD. It has geometric functions, unit conversions, variables, and the basic features of a scientific calculator.

1 Get coordinates

2 Distance between two points

3 Angle of line defined by two points

4 Intersection of two lines defined by four points

5 Clear history

6 History panel

7 Input area

8 Paste value to command line

9 Scientific

10 Units conversion

11 Variables

35.9.1 Get coordinates

Retrieves coordinates of a picked point.

35.9.2 Distance between two points

Retrieves the distance between two picked points.

35.9.3 Angle of line defined by two points

Retrieves the angle of line defined by two picked points.

35.9.4 Intersection of two lines defined by four points

Retrieves the coordinates of the intersection of two lines defined by four picked points.

35.9.5 Clear history

Clears the history panel.

35.9.6 History panel

Displays all calculations.

35.9.7 Input area

Displays the calculations input and the retrieved geometrical functions.

35.9.8 Paste value to command line

Copies the value from the input area and pastes it to the Command line.

35.9.9 Scientific

Displays the scientific calculator functions.

35.9.10 Units conversion

Performs units conversions. You can choose from the drop-down lists the parameters.

1 Return conversion to input area

Return conversion to input area
Copies the converted value and pastes it to the input area.

35.9.11 Variables

Supports variables that can either be constants (coordinates/vectors, real numbers, and integers) or functions.

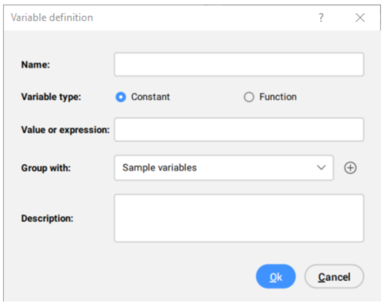
1 New variable

2 Edit variable

3 Delete

4 Return variable to input area

New variable...
Opens the **Variable definition** dialog box that allows you to define a new variable.



Note: User variables are stored in the calvar.xml file. The default folder for this file is *C:\Users\%username %\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support*

Edit variable...
Edits the selected variable.

Delete
Removes the selected variable.

Return variable to input area
Copies the variable and pastes it to the input area.

Note: Double-click a variable to retrieve it to the input area.

Right-click a variable or a category to display a context menu with additional options:

New Category...
Opens the **Category definition** dialog box that allows you to define a new category.

35.9.12 Custom functions that can be used in Calculator expression

cur
Obtains a point by cursor. The **cur** function sets the value of the LASTPOINT system variable.
Sample:

- `cur()`
- `cur+2*[3,4]`

end, mid, cen, nod, qua, ins, per, nea
These snap modes are used as parts of arithmetic expressions. BricsCAD prompts you to select an entity and returns the coordinate of the appropriate snap point. Snap modes set the value of the LASTPOINT system variable.
Sample:

- `(end+mid)/2`

dist
Obtains the distance between two points.
Sample:

- `dist(cur,end)`

vec
Obtains the vector from two points.
Sample:

- `vec(end,mid)`

vec1
Obtains the unit vector from two points.
Sample:

- `vec1(cen,end)`

dee
Obtains the distance between two points. It is a shortcut for "dist(end,end)".
Sample:

- `dee`
- `dee()`

ill
Obtains the intersection of two lines defined by four endpoints.
Sample:

- `ill(end,end,cen,mid`

ille
It is a shortcut for "ill(ene,end,ene,end)".
Sample:

- `ille`
- `ille()`

mee
Obtains the midpoint between two endpoints. It is a shortcut for "(end+end)/2".
Sample:

- `mee`
- `mee()`

rad
Obtains the radius of a selected circle, arc, or polyline arc.
Sample:

- `rad`
- `rad()`

nor - without arguments
Obtains the unit normal vector of the selected object in UCS. Only a circle, arc, or polyline is accepted.
Sample:

- `nor`
- `nor()`

nor - with one argument
Obtains the unit normal to a 2D vector. The argument is treated as a 2D vector.
Sample:

- `nor(cur)`

- `nor([1,1])`

nor - with two arguments

Obtains the unit normal vector to two 2D endpoints. The Z-coordinate of the points is simply ignored.

Sample:

- `nor(cur,cur)`

nor - with three arguments

Obtains the unit normal vector by three 3D points.

Sample:

- `nor(cur,cur,cur)`

nee

It is a shortcut for "nor(end,end)".

vee

It is a shortcut for "vec(end,end)".

vee1

It is a shortcut for "vec1(end,end)".

getvar

Returns the value of the BricsCAD system variables. Only real, integer or point system variables accepted as argument.

Sample:

- `getvar(UCSXDIR)`
- `getvar(textsize)`

35.10 Civil Explorer panel

To open the **Civil Explorer** panel, run the CIVILEXPLOREROPEN command or right-click a ribbon panel/ toolbar and choose **Civil Explorer** from the **Panels** list.

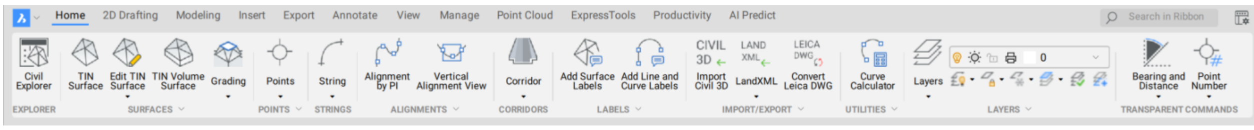
Civil Explorer is the main panel to manage and access Civil entities. Entities are arranged in a tree view and grouped by entity type.

Through this panel, you can access settings and properties of Civil entities and allows you to edit existing and add new entities and their components.

The panel is divided into two parts. The upper part with list of entities in a tree view, and the bottom part with additional properties, depending on the selection in the tree view.

The **Civil Explorer** panel is the core panel for Civil design. In the Civil Workspace, click the **Civil Explorer** icon in the Ribbon under the **Home** tab to toggle the panel on/off.

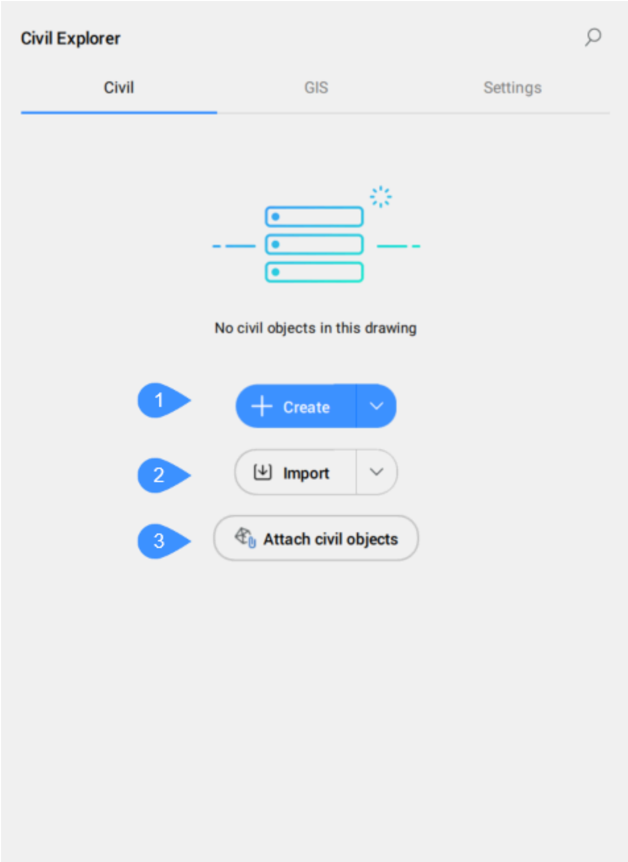
Note: Instead of individual Civil points, only point groups are listed on the **Civil Explorer**. Grading elements are listed in the tree-view, but it is not possible to edit their properties via **Civil Explorer**.



The **Civil Explorer** panel contains three tabs: **Civil**, **GIS** and **Settings**.

35.10.1 Civil Tab

If there are no Civil objects in the drawing, three buttons are available within the **Civil** tab.

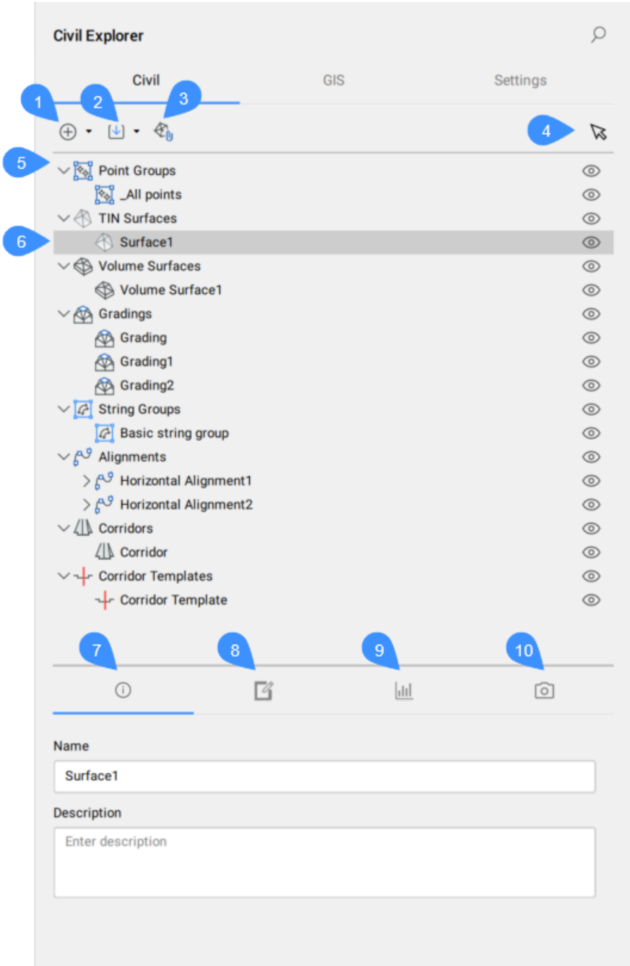


1 Create

2 Import

3 Attach civil objects

The **Civil** tab enables you to manage and access Civil entities in the current drawing.



1 Create

2 Import

3 Attach Civil objects

4 Pick a Civil object

5 Civil object types

6 Selected Civil object

7 Info

8 Definitions

9 Statistics

10 Visual Styles

Create

Allows you to create all types of the Civil objects. You have to select from the drop down list the Civil object type you want to create.

TIN Surface

Launches the TIN command.

Volume Surface
Launches the TINVOLUME command.

Grading
Launches the GRADING command.

Civil point
Launches the CIVILPOINT command.

Point Group
Launches the CIVILPOINTGROUP command.

String
Launches the STRING command.

Alignments
Launches the ALIGNMENT command.

Corridors
Creates corridors.

Corridor
Launches the CORRIDOR command.

Corridor Template
Launches the CORRIDORTEMPLATE command.

Template Element
Launches the CORRIDORTEMPLATEELEMENT command.

Import
Imports civil drawing files.

Import Civil 3D...
Launches the CIVIL3DIMPORT command.

Import LandXML...
Launches the LANDXMLIMPORT command.

Convert Leica DWG
Launches the LEICACONVERT command.

Attach civil objects
Launches the ATTACHCIVILOBJECT command.

Pick a Civil object
Pick a Civil object in the drawing to highlight it in the **Civil Explorer** panel.

Civil object types
Civil objects are arranged in a tree view. At the top of the tree structure, the types of objects that already exist in the drawing are listed. At the lower level, under each type of objects, the objects that already exist in the drawing are collected.

Right-click the Civil object type to open a context menu that contain the **Create** option, which launches specific commands to create a selected Civil object.

The right-click menu also contains a **Hide** option that can toggle the visibility for each type of Civil object.

Selected Civil object

Highlights the selected Civil object in a tree-view.

Right-click the entity to open a context menu that contains entity-specific **Edit** options, as well as options common to all entities, such as **Zoom to**, **Select**, **Delete**, **Pan to**, and **Hide**.

The available edit options depend on the entity type:

Points

Existing points are not listed on the Civil tab in the **Civil Explorer**. You can create new Civil Points by clicking the Points icon in the Ribbon under the Home tab.

Point Groups

Select the **Create** option from the context menu to open the **Point Group** dialog box.

The context menu for already defined Point Groups contains the **Civil Point Editor...**, **Edit point group...**, **Set styles...**, **Export points...**, **Zoom to**, **Select**, **Pan to**, **Delete**, and **Hide** options.

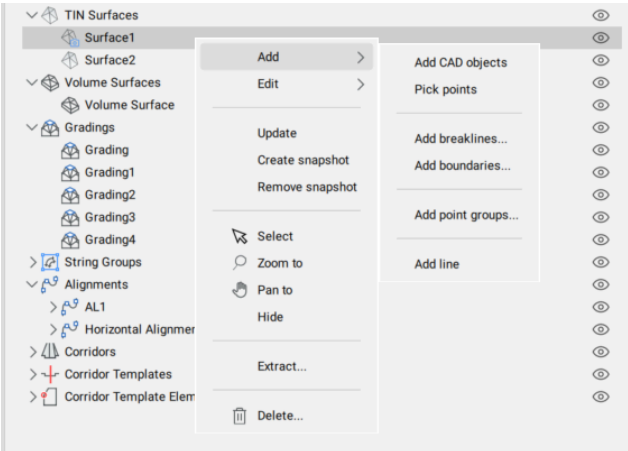
Note:

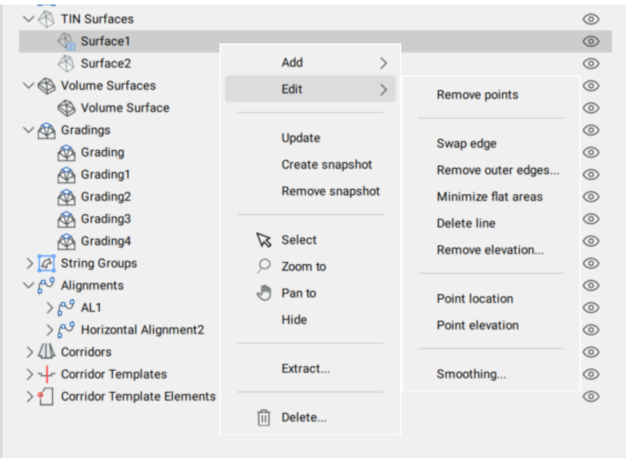
- Double-click a **Point Groups** node to open the **Civil Point Editor** dialog box.
- **Edit point group...** option opens the **Point Group** dialog box.
- **Set styles...** option opens the **Set Styles** dialog box to set the Symbol and Label styles for the point group.
- **Export points...** option opens the **Export Civil Points** dialog box.

TIN Surfaces

Using the **Civil Explorer**, you can **Edit** existing and **Add** new elements to existing TIN Surfaces.

Note: All the options under **Add** and **Edit** in the context menu are also available in the TIN and TINEDIT commands.





In addition to these options, you can **Update**, **Create snapshot**, **Remove snapshot**, or **Extract**.

Note: The **Extract...** option opens the **Extract from** dialog box.

Use the **Settings** tab in **Civil Explorer** panel to edit Surface label styles.
Also, TIN surfaces attached to the drawing with the ATTACHCIVILOBJECT command are listed.

Volume Surfaces

Update, **Select**, **Zoom to**, **Pan to**, **Hide**, **Extract**, and **Delete** options are available for Volume Surfaces.

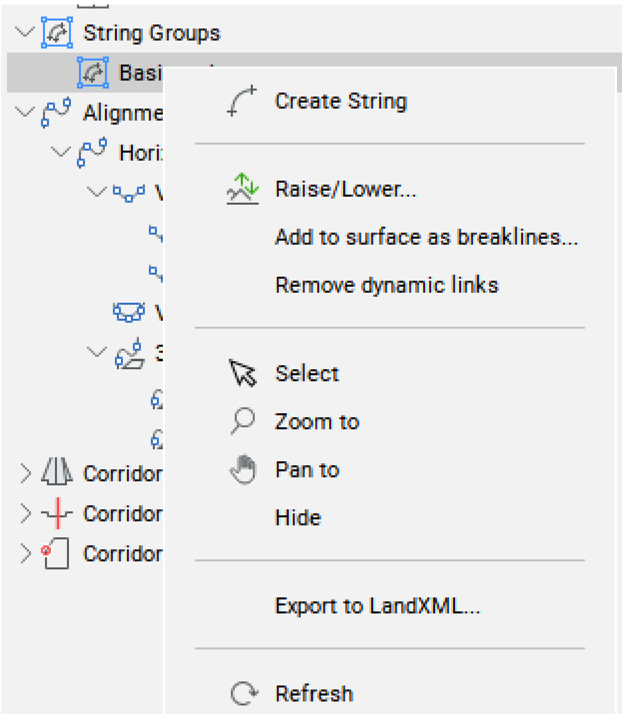
Gradings

In addition to the common options, there are **Split** and **Merge** options available for Gradings.

Note: These options are also available in the GRADINGEDIT command.

String Groups

Lists all String groups created. The context menu contains the option **Create string group**.
In addition to the common options, the context menu of a string group has the options:



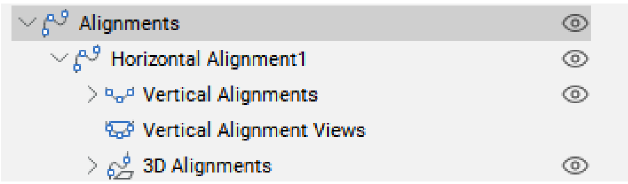
Create String
Launches the STRING command.

Raise/Lower
Allows you to raise or lower the elevation of Strings in selected String Group.

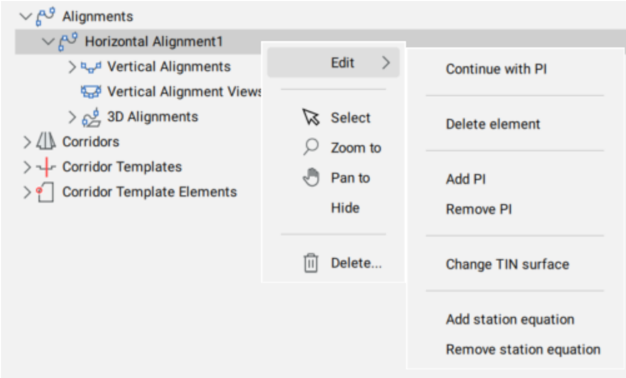
Add to surface as breaklines
Allows you to select the surface from the drop down menu, then opens **Add Breaklines** dialog box.

Export to LandXML
Allows you to export Strings in LandXML file.

Alignments
The Alignments entity type collects existing Horizontal Alignments and associated Vertical Alignments, 3D Alignments and Vertical Alignment Views.



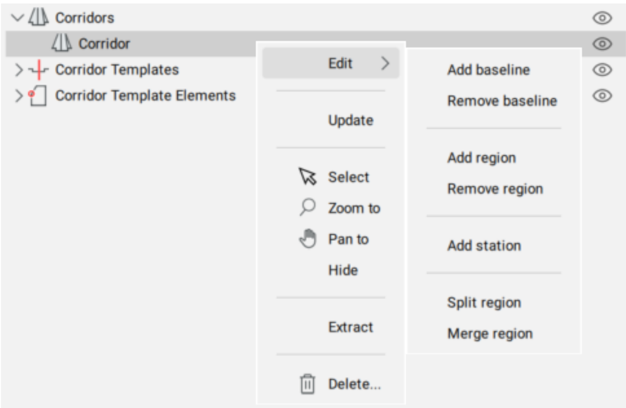
Right-click the Horizontal Alignment provides all the **Edit** options available in the ALIGNMENTEDIT command.



While for Vertical Alignments, Vertical Alignment views and 3D Alignments, the **Select**, **Zoom to**, **Pan to**, **Hide**, and **Delete** options are available.

Corridors

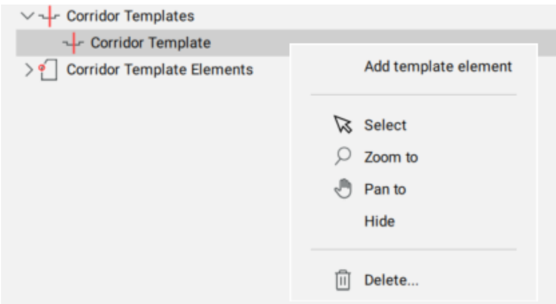
Right-click the existing Corridor provides all the **Edit** options available in the CORRIDOREDIT command. Besides **Edit** options, also the **Update, Select**, **Zoom to**, **Pan to**, **Hide**, **Extract**, and **Delete** options are available.



Corridor Templates

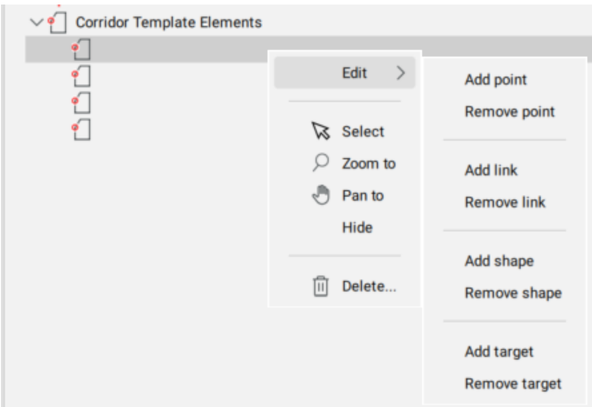
You can choose between **Add template element**, **Select**, **Zoom to**, **Pan to**, **Hide**, and **Delete** options.

Note: The **Add template element** option is also available in the CORRIDORTEMPLATE command.



Corridor Template Elements

Right-click the Corridor Template Elements provides all the edit options available in the CORRIDORTEMPLATEELEMENTEDIT command. Besides **Edit** options, also the common options, such as **Select**, **Zoom to**, **Pan to**, **Hide**, and **Delete** are available.



Info

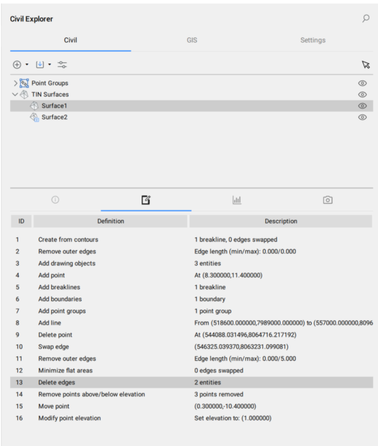
Displays information about the selected Civil object (Name and Description). Both can be edited.

Note: For String Groups, 'Interact with other strings in the group' switch is available.

Definitions

It shows the components (source data) from which the TIN surface was built, the editing operations used, and the chronological order in which definitions were added.

The surface build process for a TIN surface is incremental. When you add data to a surface or edit the surface, the surface is updated. When data is removed, the surface is rebuilt. The definitions list supports the incremental build process and sequentially shows all operations performed on the surface in its current state.

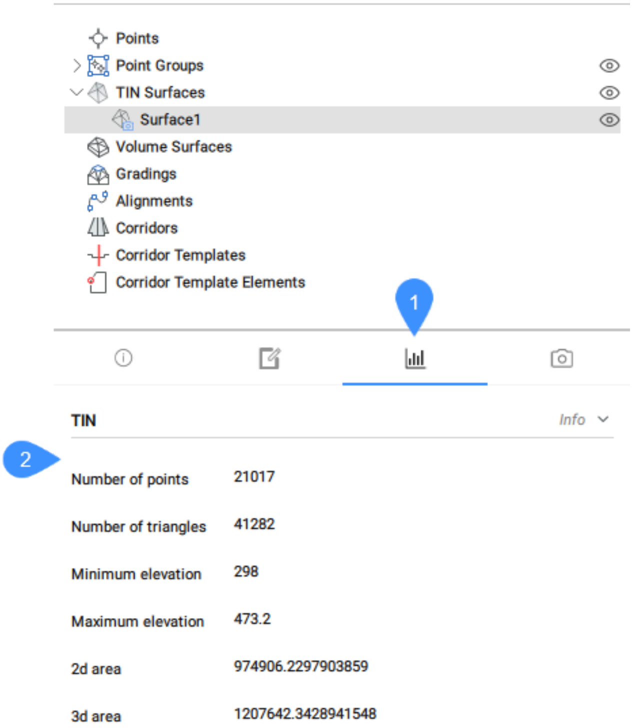


The ID field indicates in which step each component was added to the TIN Surface.

The context menu offers the possibility to change the order of definitions, to **Delete** or **Disable** them. Additionally, for most TIN Surface definitions, there are the options **Select** and **Zoom to**, which makes finding them in the drawing very easy. Double click a definition or press the **Edit** option from the context menu to open the **Surface: Surface name, Definition step ID** dialog box, which displays all details for the selected definition. You can edit the definition parameters.

Statistics

The **Statistics** tab displays the statistics for the selected TIN surface (number of points and triangles, minimum and maximum elevation, 2D and 3D area).



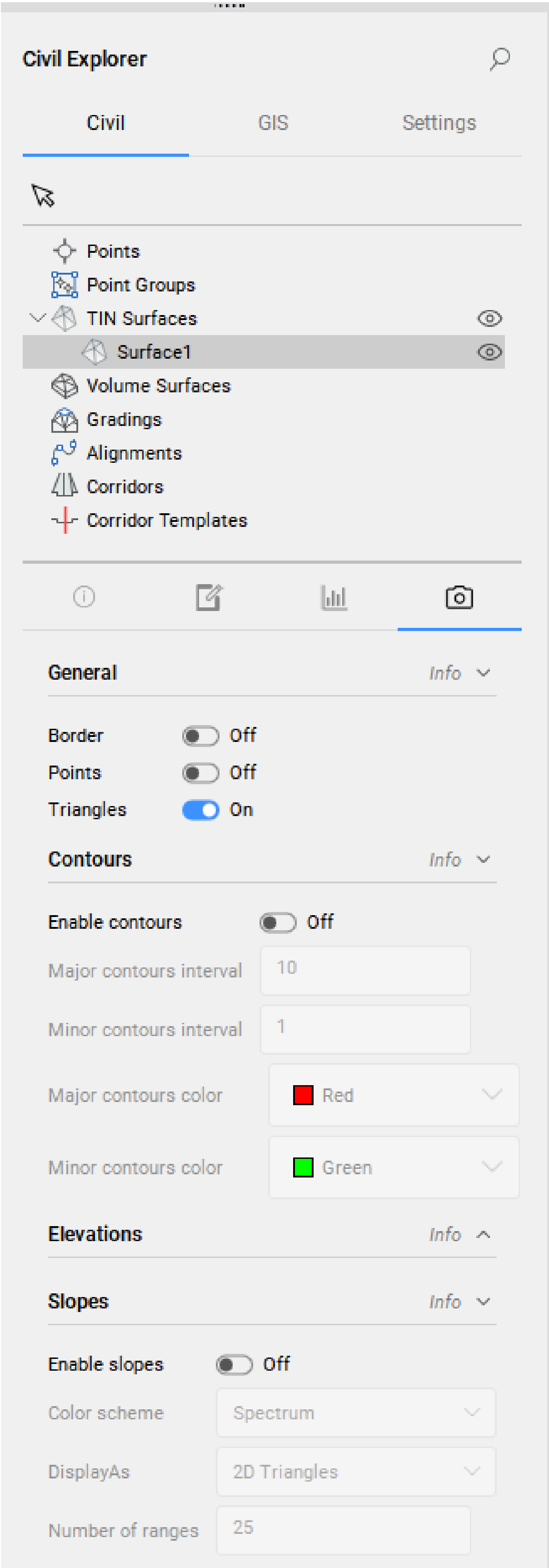
1 Statistics tab

2 Statistics data

Visual styles

The **Visual styles** tab allows you to set the visual styles for the TIN surface:

- general visual style – choose which components of the TIN surface to display (Border, Points, Triangles).
- visual style for contours.
- visual style for the elevation analysis.
- visual style for the slope analysis.


Civil Explorer
Civil
GIS
Settings
Points
Point Groups
TIN Surfaces
Surface1
Volume Surfaces
Gradings
Alignments
Corridors
Corridor Templates
General
Info
Border
Off
Points
Off
Triangles
On
Contours
Info
Enable contours
Off
Major contours interval
10
Minor contours interval
1
Major contours color
Red
Minor contours color
Green
Elevations
Info
Slopes
Info
Enable slopes
Off
Color scheme
Spectrum
DisplayAs
2D Triangles
Number of ranges
25

General
Enables/disables the display of certain TIN surface components.

Border
Toggles the display of the Boundary line of the TIN surface.

Points
Toggles the display of the Points of the TIN surface.

Triangles
Toggles the display of the Triangles of the TIN surface.

Contours
Specifies parameters for displaying contours on a TIN surface.

Enable Contours
Toggles the display of the Contours of the TIN surface. The visual style of the contours can be adjusted.

Major contours interval
Allows you to enter the interval for the major contours.

Minor contours interval
Allows you to enter the interval for the minor contours.

Major contours color
Allows you to enter the color for the major contours.

Minor contours color
Allows you to enter the color for the minor contours.

Elevations
Specifies parameters for creating TIN Elevations Analysis.

Enable Elevations
Toggles the display of the Elevations of the TIN surface. The visual style of the elevations can be adjusted.

Create ranges by
Allows you to choose between the methods of creating Elevation Analysis:

- Number of ranges
- Range interval
- Range interval with datum

Color scheme
Allows you to select the color scheme for the Elevation Analysis.
Choose between the available options:

- **Spectrum**
- **Earth**
- **Hydro**
- **Gray**
- **Reds**
- **Greens**
- **Blues**

Above datum color scheme
Allows you to select the Above datum color scheme.

Below datum color scheme
Select the Below datum color scheme.

DisplayAs
Select the TIN Surface elements used to create the Elevation Analysis.
Choose between the available options:

- **Triangles**
- **Contours**
- **Points**
- **2D Triangles**
- **2D Contours**

Range interval
Specifies the Range interval for the Elevation Analysis.

Number of ranges
Specifies the number of ranges for the Elevation Analysis.

Datum
Specifies the datum elevation.

Slopes
Specifies parameters for creating TIN Slopes Analysis.

Enable Slopes
Toggles the display of the Slopes of the TIN surface. The visual style of the slopes can be adjusted.

Color Scheme
Allows you to select the color scheme for the Slopes Analysis.
Choose between the available options:

- **Spectrum**
- **Earth**
- **Hydro**
- **Gray**
- **Reds**
- **Greens**

DisplayAs
Select the TIN Surface elements used to create the Slopes Analysis.
Choose between the available options:

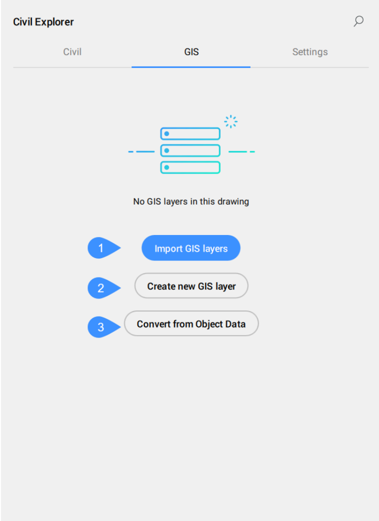
- **Triangles**
- **2D Triangles**

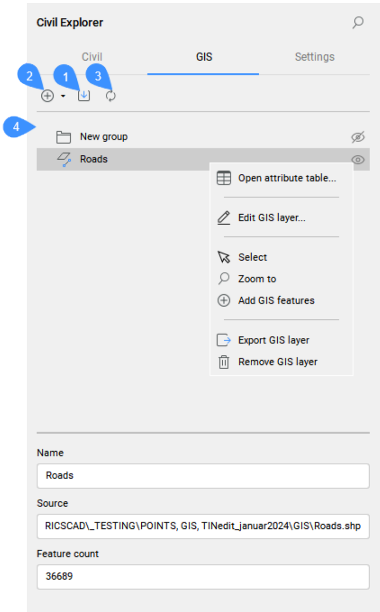
35.10.2 GIS Tab

GIS Tab contains a list of all the GIS layers in the drawing and helps you manage their visibility.

If there is no GIS layer in the drawing, there are three options on the GIS tab:

- Import GIS Layer
- Create GIS Layer
- Convert from Object Data





1 Import GIS layers

2 Create group and Create GIS layer

3 Convert from Object Data

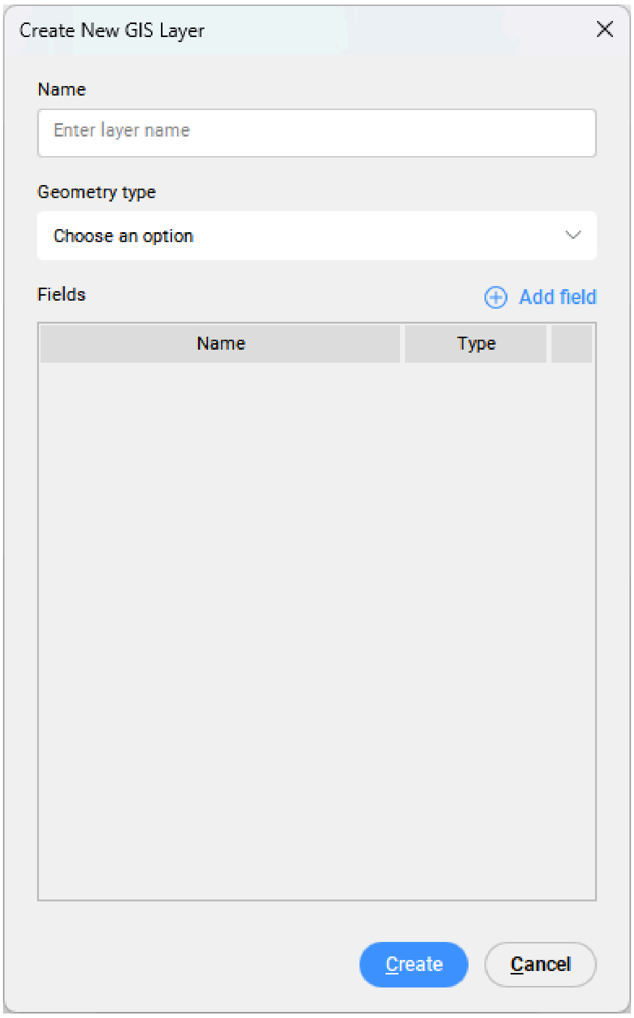
4 GIS tree view

Import GIS layers

Allows you to import ESRI SHP or ESRI Geodatabase files. Check the GISIMPORT command for the detailed description of **GIS Import** dialog box.

Create GIS layer

Allows you to create a New GIS Layer. The **Create New GIS Layer** dialog opens when the command is launched:



Name

Allows you to enter the new Layer name.

Geometry type

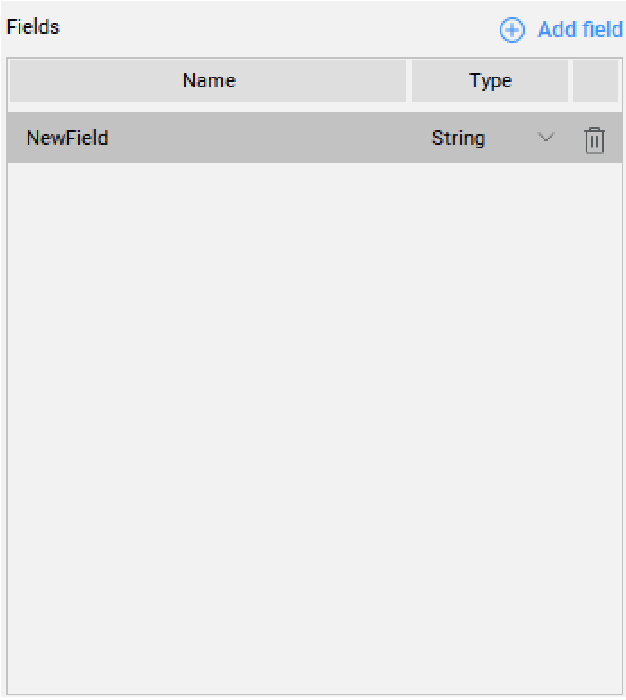
Allows you choose between the following geometry types:

- **Point**
- **Line**
- **Polygon**

Add Field

Adds a new field.

When clicking the **Add field** button, a new field is added to Fields list, with the default name **NewField** and default type **String**. You can edit the **Name** and **Type** fields.

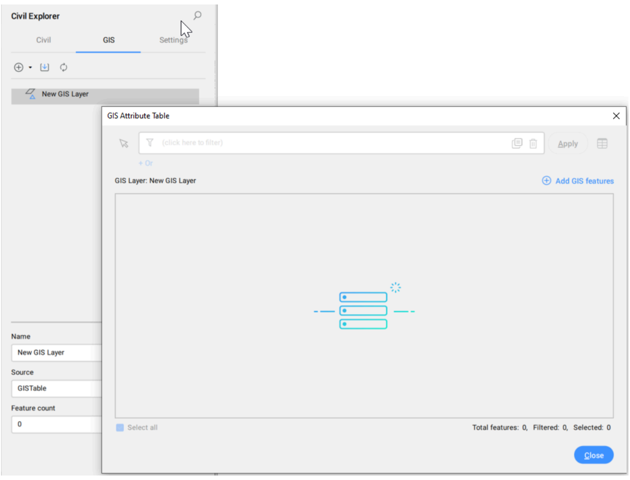


The following field Types are available:

- **String**
- **Real**
- **Integer**

Click the trash icon in the table to delete fields.

When you first open the table of a newly created GIS Layer, the table is empty. To add GIS features to the GIS Layer, click the button **Add GIS features**. Select linear entities in the drawing and press Enter. The **GIS Attribute table** dialog box contains selected GIS features with appropriate GIS data.



Convert from Object Data

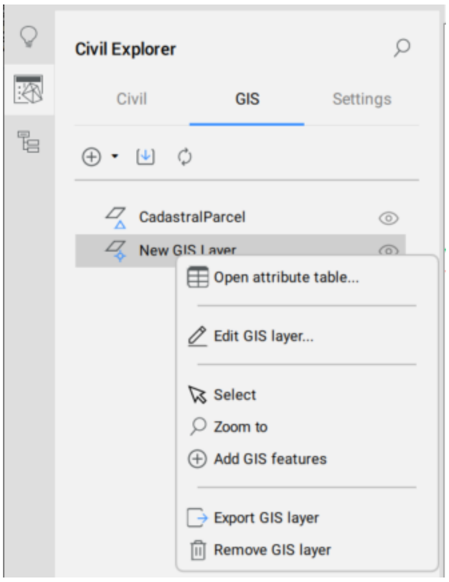
Converts AutoCAD® Map 3D and Autodesk® Civil 3D GIS features from current drawing to BricsCAD GIS data.

Note: The AutoCAD® Map 3D and Autodesk® Civil 3D Object Data can be viewed and edited directly in the **Properties** panel.

GIS tree view
Lists all GIS layers and groups.

Note: You can rearrange groups and GIS layers by using drag and drop.

The following context menu opens when you right-click a GIS Layer:



Open attribute table...
Opens the **GIS Attributes Table** dialog box.

Edit GIS layer
Opens **Edit GIS Layer** dialog box.

Select
Selects the GIS Layer.

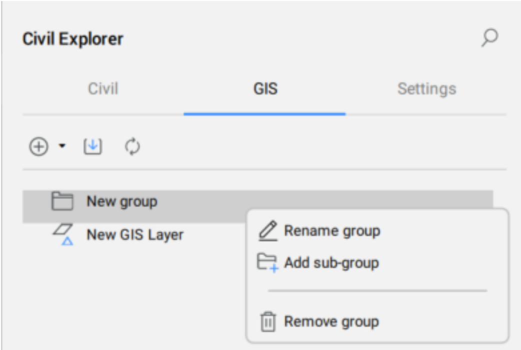
Zoom to
Zooms to the GIS Layer.

Add GIS features
Adds entities to a GIS Layer. A command prompt for selecting linear entities in the drawing is shown. The **GIS Attributes Table** dialog box opens where the new GIS features are added at the end of the table.

Export GIS layer
Exports the GIS layer.

Remove GIS layer
Removes the GIS layer.

The following context menu opens when you right-click a Group:



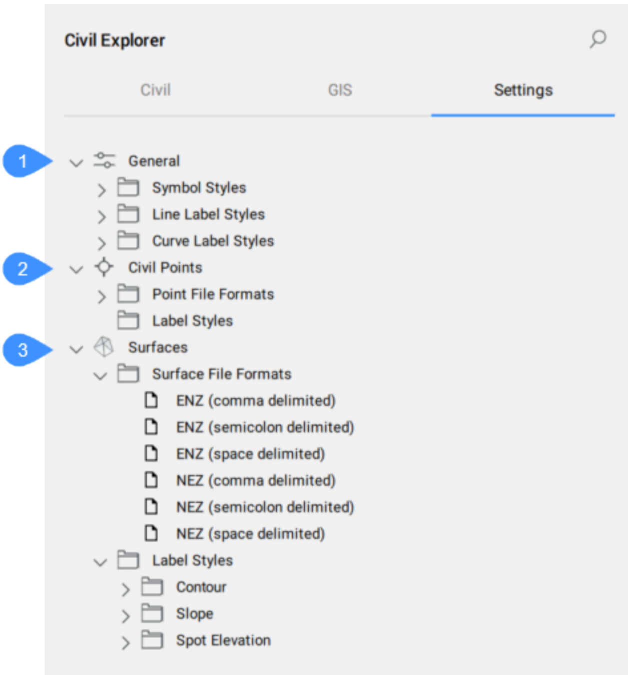
Rename group
Renames the group.

Add sub-group
Adds a sub-group.

Remove group
Removes the group.

35.10.3 Settings Tab

The **Settings** tab manages the civil entity styles and formats.



1 General

2 Civil Points

3 Surfaces

General
Defines the general symbol and label styles for civil entities.

Symbol styles

Lists predefined and custom-created symbol styles. You can **Create**, **Edit**, **Copy**, or **Delete** a symbol style. Click the corresponding option from the context menu to open the **Symbol Style** dialog box.

Line Label Styles

Lists predefined and custom-created line label styles.

Curve label Styles

Lists predefined and custom-created curve label styles.

The context menu contains the **Create**, **Edit**, **Copy**, or **Delete** options. Click the corresponding option to open the **Label Style Editor** dialog box.

Civil Points

Defines civil points file formats and label styles for civil points.

Point File Formats

Lists predefined and custom-created point file formats. The list can be managed by double-clicking or pressing the **Edit** option from the context menu which opens the **Manage Point File Formats** dialog box.

Predefined point file formats:

- NameENZ (comma delimited)
- NameENZ (semicolon delimited)
- NameENZ (space delimited)
- NameENZD (comma delimited)
- NameENZD (semicolon delimited)
- NameENZD (space delimited)
- NameNEZ (comma delimited)
- NameNEZ (semicolon delimited)
- NameNEZ (space delimited)
- NameNEZD (comma delimited)
- NameNEZD (semicolon delimited)
- NameNEZD (space delimited)
- PENZ (comma delimited)
- PENZ (semicolon delimited)
- PENZ (space delimited)
- PENZD (comma delimited)
- PENZD (semicolon delimited)
- PENZD (space delimited)
- PNEZ (comma delimited)
- PNEZ (semicolon delimited)
- PNEZ (space delimited)
- PNEZD (comma delimited)
- PNEZD (semicolon delimited)

- PNEZD (space delimited)

where:

- P is Point Number
- E is Easting
- N is Northing
- Z is Elevation
- D is Raw Description
- Name is for points that contain alphanumeric values for the point number. Also, the point name supports alphanumeric values.

Label Styles
Lists the available label styles for Civil Points. The context menu allows you to **Create**, **Edit**, **Copy**, or **Delete** a label style. **Create**, **Edit** and **Copy** options open the **Label Style Editor** dialog box which allows you to define a new label style.

Surfaces
Defines surface file formats and label styles for Contour, Slope and Spot elevation.

Surface File Formats
Lists predefined and custom-created surface file formats. The list can be managed by double-clicking or pressing the **Edit** option from the context menu that opens the **Manage Point File Formats** dialog box.

Label styles
Lists predefined and custom-created styles for Contour, Slope and Spot elevation. The context menu allows you to **Edit**, **Copy**, or **Delete** a label style. The **Edit** and **Copy** options open the **Label Style Editor** dialog box which allows you to define a new label style.

35.11 Cloud panel (Experimental)

35.11.1 Disclaimer

EXP.

The **Cloud** panel is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

35.11.2 Description

When the **Experimental mode** is enabled, the **Cloud** panel and the **Cloud** tab on the **Start** page are active.

To open the **Cloud** panel, right-click a ribbon panel/toolbar and choose **Cloud** from the **Panels** list.

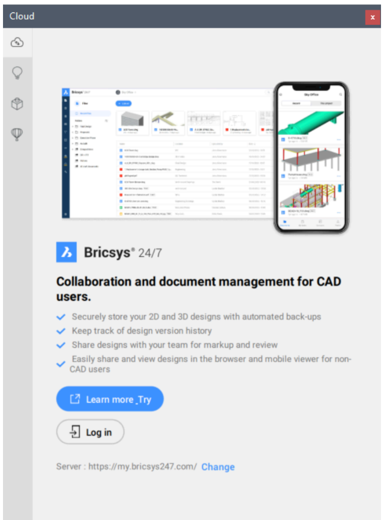
The **Cloud** panel helps team members to find and share their drawings.

Cloud panel enables project management, a collaboration between project teams and cloud computing security. The big advantage of this cloud-based service is that BricsCAD is directly linked to it by the **Cloud** panel.

The CLOUDDOWNLOADPATH system variable stores the local cloud cache folder for creating, importing, or downloading files from the Bricsys24/7 server.

Any drawing you access through the **Cloud** panel to create, import, or edit will first be synced to your local cache folder and later can be uploaded back to the Bricsys 24/7 server.

35.11.3 Log in



The CLOUDSSOCLIENTID system variable stores the client_id to be used when connecting to SSO service.

The CLOUDSSOSCOPE system variable stores the scopes or permissions to be used when connecting to SSO service. It depends on the authorization server for a given Bricsys 24/7 service (what scopes are configured/allowed for the client). For example, Azure AD requires an explicit client id on the scope parameter, like **<client id>/access_as_user openid**.

The default server used is **https:/my.bricsys247.com/**, but it can be changed by clicking **Change** button, which opens the **Settings** dialog box for the CLOUDSERVER system variable, which allows you to specify a different address for the Bricsys 24/7 server.

For other certain environments, like *https://pellikaan.bricsys247.com/*, that does not support SSO at this moment, the log in dialog box can be different.

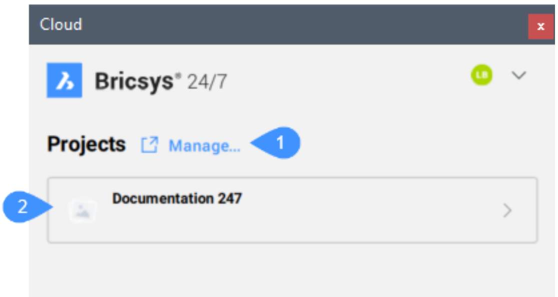
For **Log in** follow the steps:

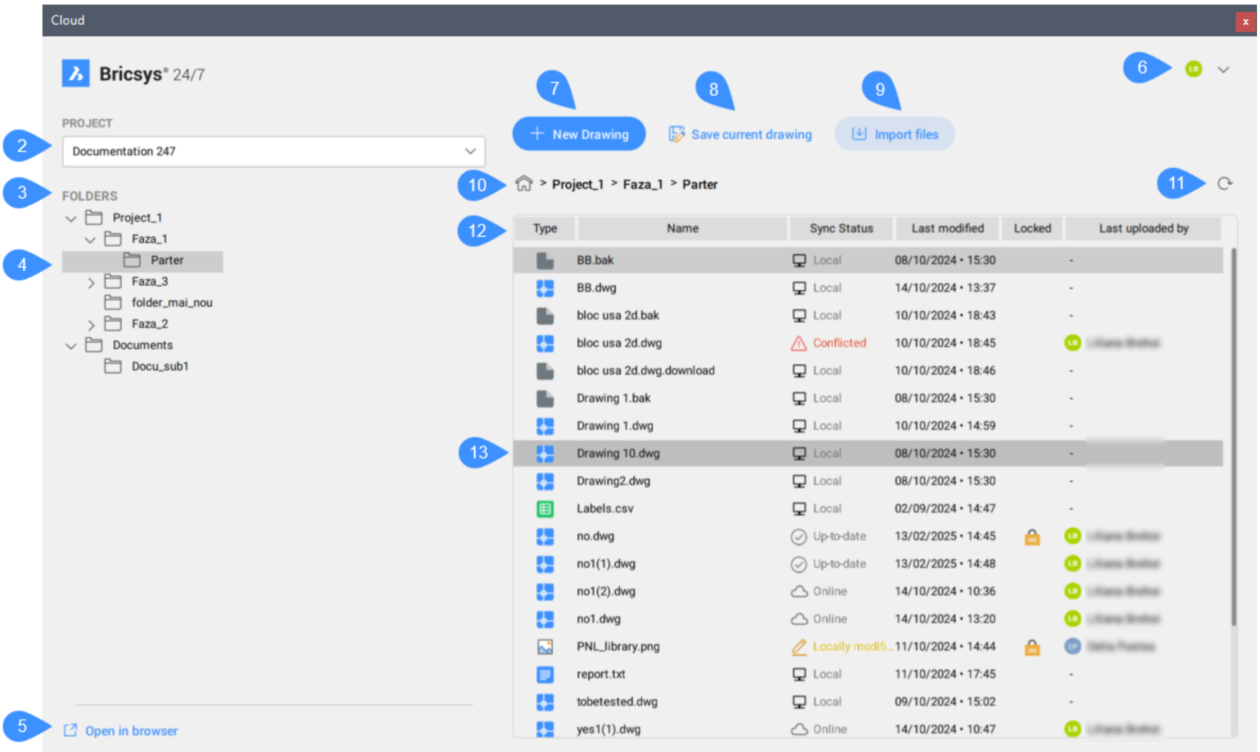
1 Enter your Email address and Password and press the **Log in** button.

2 (Optional) Tick the **Remember me** checkbox to store the username and password.

3 Once logged in, you will see an overview of your **in the cloud** Projects, Folders and Documents.

35.11.4 The Cloud panel content





1 Manage

2 Project name

3 Tree structure of the project

4 Active Folder

5 Open in Browser button

6 Pop-up menu

7 New drawing button

8 Save current drawing button

9 Import files button

10 Home button

11 Refresh button

12 Characteristics of the files

13 Files list

Manage
Opens the Bricsys 24/7 application in browser https://my.bricsys247.com/app/.

Project
When logged in, in the **Cloud** panel you will see all the projects you are assigned to. Choose a project and open it by pressing the right-side arrow.

Tree structure of the project
Lists all the folders is in the project.

Active Folder
Marks the active folder name. In the right side of the panel there are all files included in this folder.

Open in Browser button
Opens the Bricsys 24/7 application in browser https://my.bricsys247.com/app/.

Pop-up menu
Provides the user name and email address information, and buttons to access the Bricsys 24/7 browser, Bricsys Help Center, and Log out.

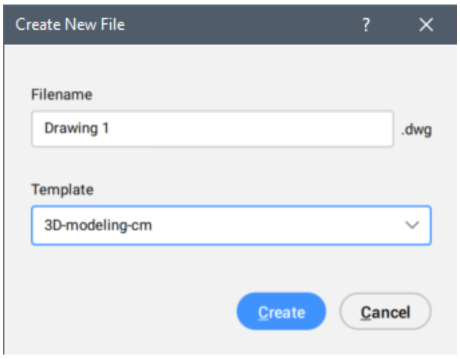
User name
Displays the user name and email address used for log in.

Bricsys 24/7 web app
Opens the Bricsys 24/7 application in browser in the active project and folder https://my.bricsys247.com/app/.

Online help
Opens the help article in the Briscys Help Center browser.

Log out
Log out from application.

New drawing button
Opens the **Create New File** dialog box, a new file will be created in the local cache cloud folder. The file can then be uploaded to the correct project and folder with a single click.



Filename
Defines a file name for the new drawing.

Template
Allows you to choose a template for the new drawing.

Save current drawing button

Saves the current active drawing to the local Bricsys247 folder.

Note:

- The button is not visible when the top-level folder is active.
- This button becomes active only when there is an active drawing.

Import files button

Opens the **Copy files to current folder** dialog box to choose a file to import. The selected folder will be copied to the local cache cloud folder.
The file can then be uploaded to the correct project and folder with a single click.

Home button

Goes back to your main project folders.

Refresh button

Manually refreshes the **Cloud** panel.
The panel refreshes automatically when you switch between folders.

Characteristics of the files

Displays the characteristics of the files.
You can sort the list of files ascending or descending by pressing the up or down arrow for each field.

Type

Displays the type of the file.

Name

Displays the name of the file.

Sync Status

Displays the sync status of the file.

- Local : the file is stored on the local computer and not on the cloud.
- Online : the file is stored in the cloud.
- Up-to-date : the local and online files are at the same version
- Locally modified : the file is available online and on the local computer, the local version has a more recent save date.

Last modified

Displays the last date and time the file was modified.

Locked

Displays if a file is locked ( ) or not locked by a user. When a file is locked by someone logged in to Bricsys 24/7, another user cannot upload a new version to Bricsys 24/7.

Last updated by

Displays the name of the user who most recently updated the file.

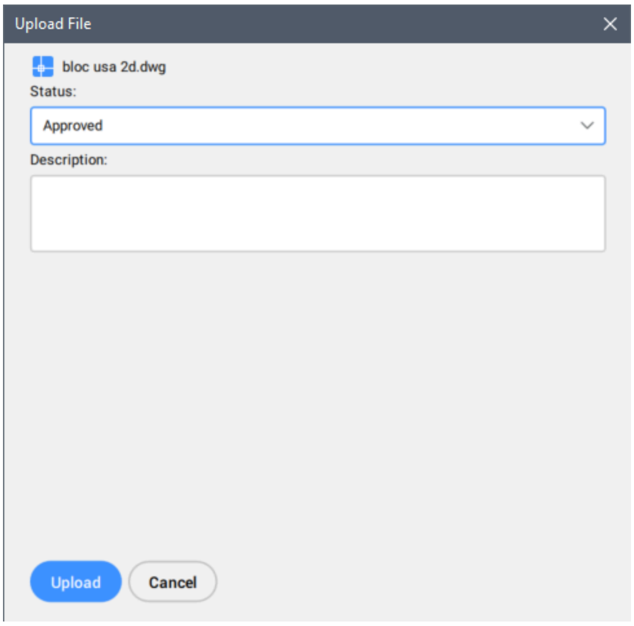
Files list

Lists all files in the folder.
Depending on the **Sync Status** of the selected file, the context menu contains the below options.

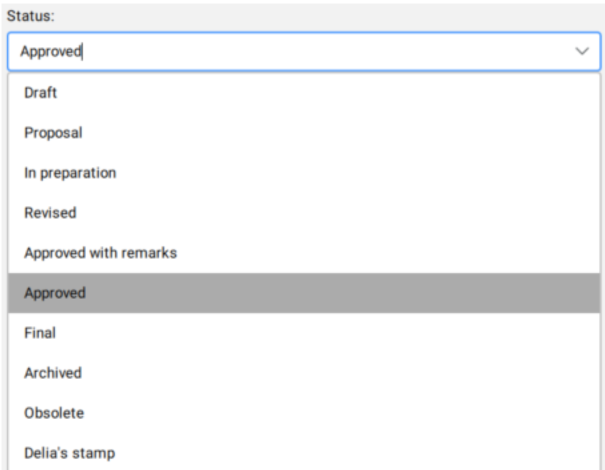
- **Open**: opens a drawing (DWG) file in BricsCAD.

  Note: If the file is other than the DWG file, a warning message will notify you that it cannot be opened.

- **Upload as revision**: opens the **Upload File** dialog box to upload your file as a revision of the original file. For drawing files, see the CLOUDUPLOADDEPENDENCIES system variable options.



  a **Status**: allows you to choose the status of the new file from the drop-down list and a description.



  b **Description**: allows you to write a description for the revision.

  c **Upload**: uploads the file.

- **Reveal in file explorer**: opens the File Explorer to reveal the file location.
- **Reveal in browser**: opens the Bricsys 24/7 browser to reveal the location of the file.
- **Download**: downloads the latest revision of the file from the cloud. When a local version of the file has been modified, you will be asked if you want to overwrite the local changes.

Note: The local folder for download is defined with the CLOUDDOWNLOADPATH system variable.

- **Download and open**: downloads and opens the file.

  Note: If the file is other than the DWG file, a warning message will notify you that it cannot be opened.

- **Lock**: locks the document so no changes can be made.
- **Unlock**: unlocks the document so changes can be made.

35.11.5 Uploading and downloading drawings with Xrefs

Uploading drawings with Xrefs

If you add a drawing that contains external references (Xrefs), make sure you also add the reference files to the Bricsys 24/7 server.

To do that, you can:

1. Upload the files one by one using the **Cloud** panel.
2. Use the **Upload to Bricsys 24/7** option in the **Drawing Explorer** > **Dependencies** dialog box. Make sure you include the root drawing and the external references.

   Note: The **report.txt** file is uploaded as well. The report contains a complete list of all files included and notes for distribution.

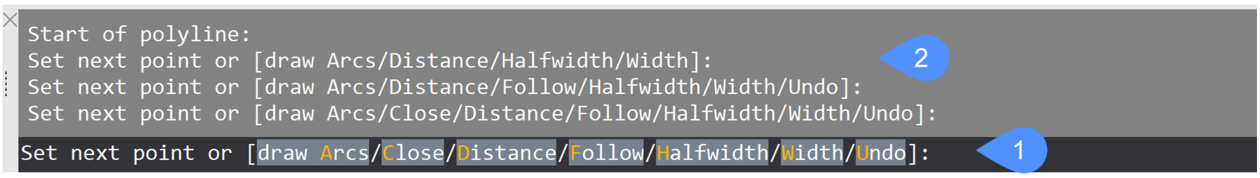
Downloading drawings with Xrefs

If you download a drawing that contains Xrefs, the reference files are automatically downloaded.

35.12 Command line panel

To open the **Command line** panel, run the COMMANDLINE command, right-click a ribbon panel/toolbar and choose **Command Line** from the **Panels** list, or use the **Ctrl+9** keyboard shortcut.

The **Command line** panel allows you to launch commands or modify system variables by typing the command or variable name.



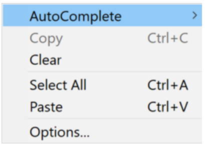
1 Command line

2 Command history

35.12.1 Command line

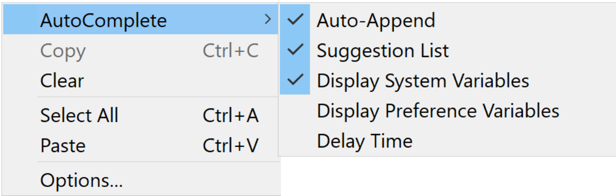
Displays the current command prompt. If no command is active, it prompts you to Enter Command. If a command is active, it displays prompts and options relevant to the current command. Prompt shortcuts are highlighted. You may enter the shortcut characters in the Command line or click on them with the left mouse button.

A right-click menu offers relevant tools.



AutoComplete

Hover over the **AutoComplete** option. Another context menu displays:



- If **Auto-Append** is checked, the entry is automatically completed if only one possible command is left.
- If **Suggestion List** is checked, the list of possible commands is displayed when typing in the command bar.
- If **Display System Variables** is checked, the system variables are included in the suggestion list.
- The **Delay Time** option sets the time in seconds before automated keyboard features display.

Copy

This option copies the selected text to the clipboard.

Clear

This option clears complete prompt history.

Select All

This option selects all the completed content of the prompt history.

Paste

This option pastes the text from the clipboard to the Command line.

Options

This option displays the **Settings** dialog box at the Command line section.

35.12.2 Command History

Displays the history of the Command line input for the current BricsCAD session.

35.12.3 Accessing the input history

Use the Up Arrow and Down Arrow keys at the Command prompt or at an input prompt to access and reuse the history of recent commands input.

Note: Markers for recent input of point locations are displayed in the drawing area. The markers are controlled by SNAPMARKERSIZE, SNAPMARKERCOLOR, and SNAPMARKERTHICKNESS system variables.

Remember: The input history can include the last 12 items from the current command (from the current and previous calls) and the last 12 items from other previous commands.

35.13 Command panel

A command panel offers intermediate user interaction when executing a command. The advantage of a panel compared to a dialog box is that the content of the drawing remains visible, instead of being partially blocked.

The following commands use a floating command panel:

- BIMDIMENSION
- BMBOMTEMPLATEEDIT opens **BOM Manager** panel
- FITARC
- FITLINE

The following commands use a **Command Context** panel, which is a dockable panel. It appears at the same size and location that it did before it was closed or collapsed, and like any other dockable panel, the **Command Context** panel can be either floating, docked or stacked.

- ARRANGE
- BIMIFY
- BIMINVERTSPACES
- BIMQUICKBUILDING
- BIMQUICKDRAW
- BIMTAG
- BLOCKIFY (Find Collections option)
- CAEANALYZE2D command
- DMAUDIT
- DMSIMPLIFY
- DMSIMPLIFYALL
- DWGHEALTH
- FINDOUTLIERS
- NUMBER
- OPTIMIZE
- POINTCLOUDCOLORMAP
- POINTCLOUDDEVIATION
- POINTCLOUDFITPLANAR
- POINTCLOUDPROJECTSECTION
- SIMPLIFY

The content of a command panel depends on the active command. It can be categorized into three domains:

1 Necessary steps, for example:

1 - Entity selection

- (Base) point selection
- Naming
- Choosing alternatives

2 Settings, for example:

3 - On/off options
- Default values
- Tolerances

4 Command progress, for example:

5 - Number of handled elements versus total number of elements to process.
- Progress meter.

The command panel opens when the command starts and closes when the command ends.

35.13.1 Command Context panels icons

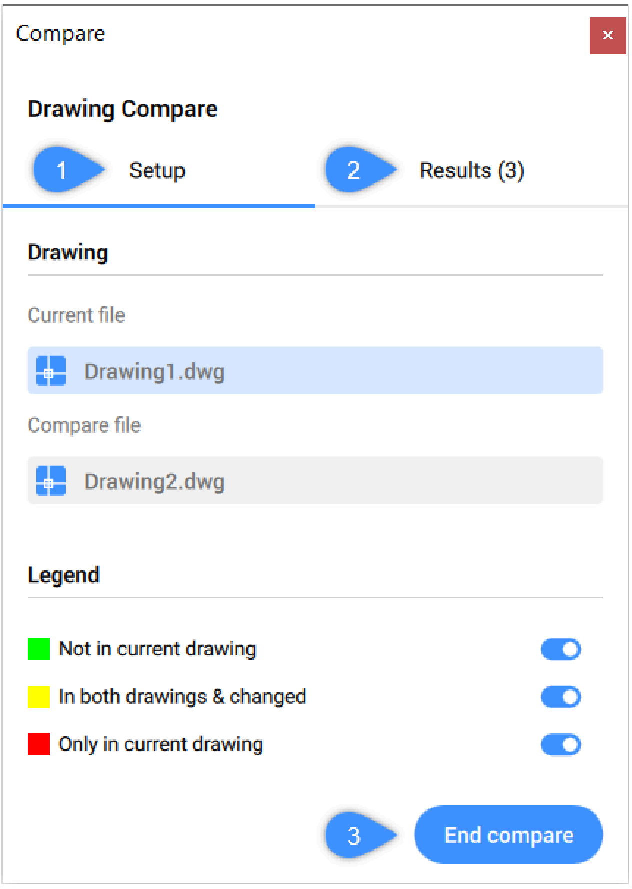
Icon	Command
	
	
	
	
	
	
	
	
	
	

Icon	Command

35.14 Compare panel

To open the **Compare** panel, run the DWGCOMPAREOPEN command or right-click a ribbon panel/toolbar and choose **Compare** from the **Panels** list.

The **Compare** panel allows you to compare the current drawing with a selected drawing.

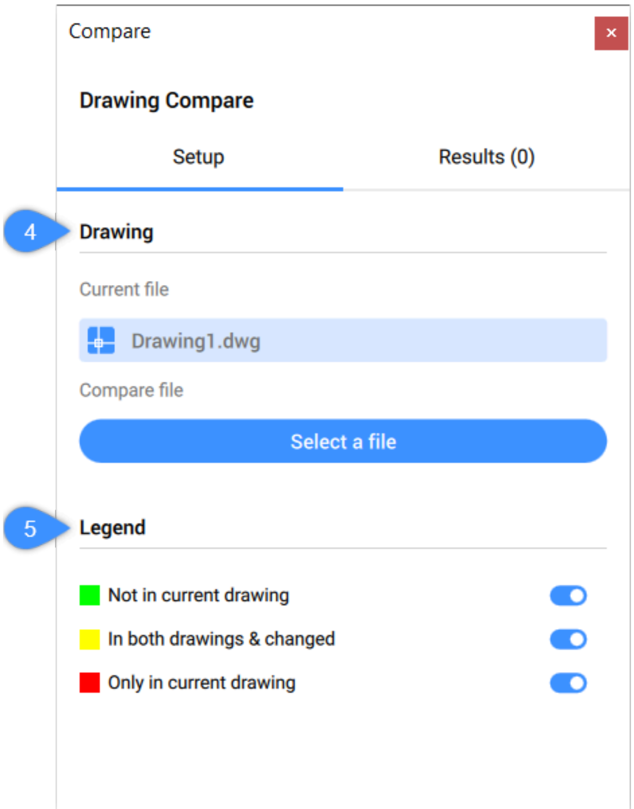


1 Setup

2 Results

3 End compare

35.14.1 Setup



4. Drawing

5. Legend

Drawing

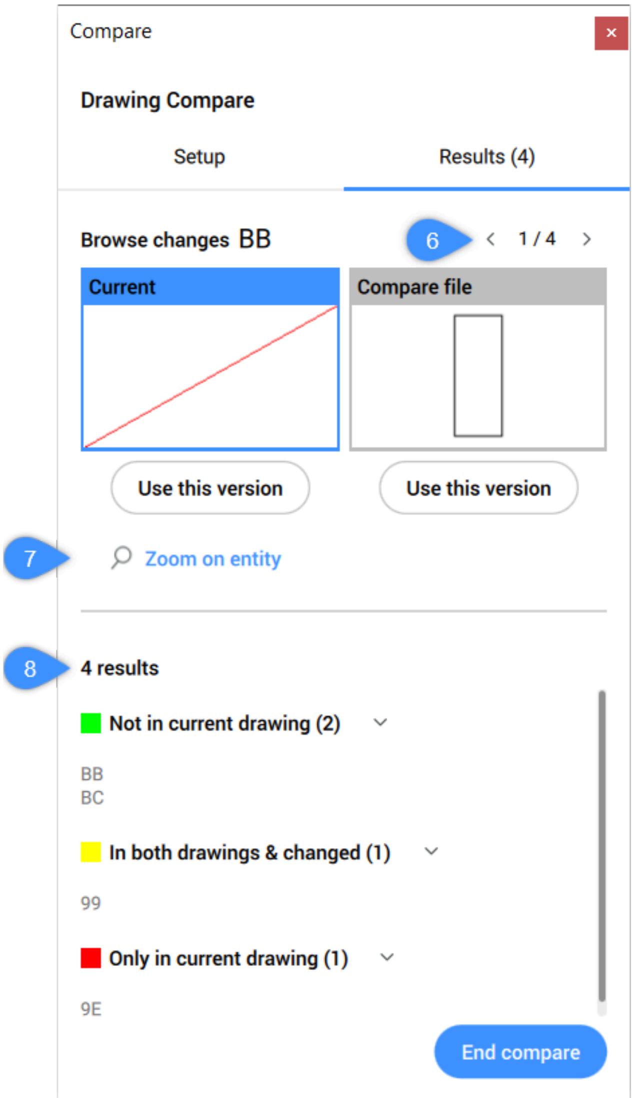
Displays the name of the current file and the selected file to compare the current file against. If no compare file is selected yet, click the **Select a file** button. The **Please pick a file to compare against.** dialog box opens to let you select a file.

Legend

Displays the colors in which the entities are displayed when the program finds differences. You can toggle the different categories on and off to determine whether the corresponding entities should be shown or not.

Note: You can click the colors to open the **Color** dialog box where you can pick another color for the selected category.

35.14.2 Results



6. Browse changes
7. Zoom on entity
8. Results list

Browse changes
Allows you to browse through the changes that are found between the two drawings. A preview of the entities is shown.

- Click on the arrows to browse through the changes
- Click on **Use this version** to select from which drawing you want to keep the entity

Zoom on entity
Zooms on to the entity that is shown in the preview

Results
Displays the handles of the entities that are different from the compare drawing.

- Not in current drawing: the entities only exist in the compare drawing
- In both drawings & changed: the entities exist in both drawings, but are changed

- Only in current drawing: the entities only exist in the current drawing

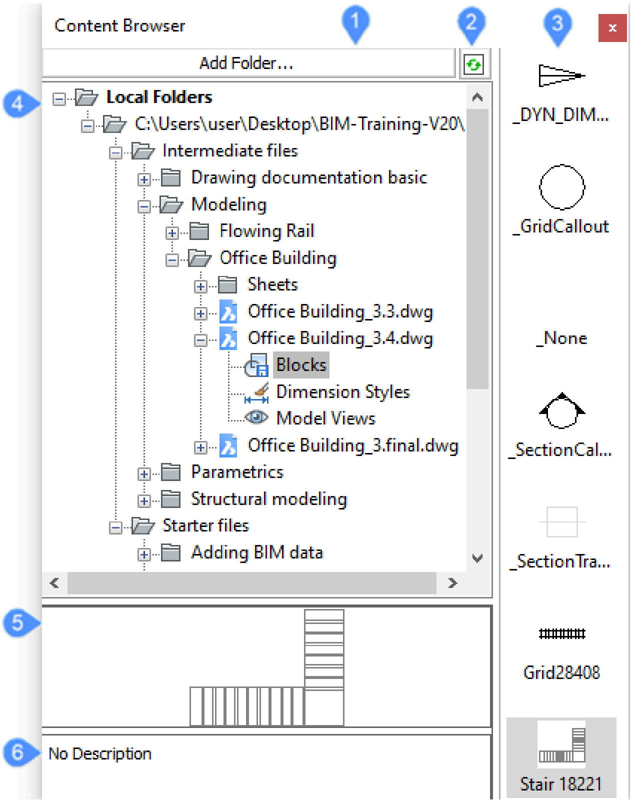
35.14.3 End compare

Ends the comparison and unloads the compare file.

35.15 Content Browser panel

To open the **Content Browser** panel, run the CONTENTBROWSEROPEN command or right-click a ribbon panel/toolbar and choose **Content Browser** from the **Panels** list.

The **Content Browser** panel offers you a quick and convenient way to open drawings, manage your block libraries, copy dimension styles, place views for your entire DWG files stored on your computer.



1 Add Folder...

2 Refresh

3 Definitions in the selected drawing

4 Local Folders tree

5 Definitions' preview

6 Description

35.15.1 Add folder...

Displays the **Choose a folder** dialog box.

35.15.2 Refresh

Select a drawing and press the Refresh button to expand its content (Blocks, Dimension Styles, Model Views).

35.15.3 Definitions in the selected drawing

Select Blocks/Dimension styles/Model views to see the definitions used within the drawing.

You can drag & drop a block onto the current drawing and an **Insert Block** dialog box will appear.

You can drag & drop a dimension style into the current drawing. If a dimension style with the same name already exists, a **Copy / Paste** dialog box will appear.

You can drag & drop a model view into the current drawing. This option is available only in Paper Space.

35.15.4 Local Folders tree

Displays the content of a drawing/project. Right-click menus enable you to add or remove folders from within the Folders tree.

35.15.5 Definitions' preview

Shows a preview of your drawing/block/dimension style.

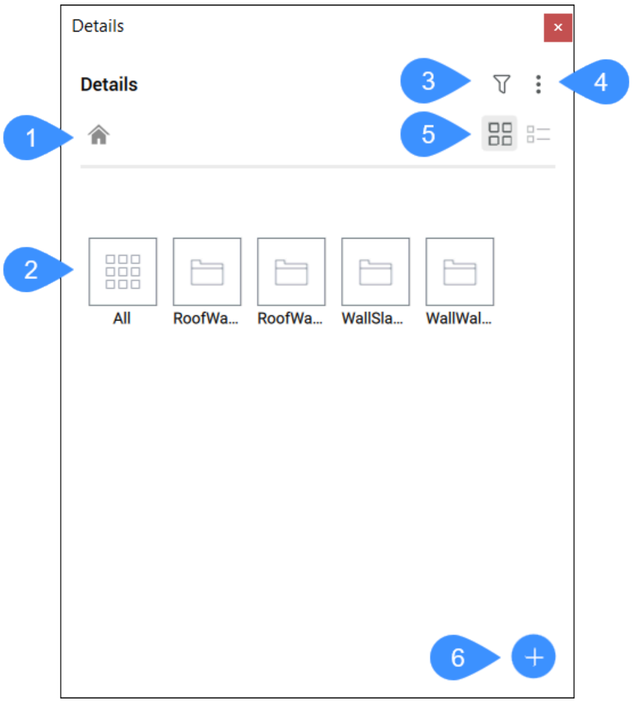
35.15.6 Description

Displays a description of your selected definition.

35.16 Details panel

To open the **Details** panel, run the DETAILSPANELOPEN command or right-click a ribbon panel/toolbar and choose **Details** from the **Panels** list.

The **Details** panel allows you to view details, add tags to details, propagate and adjust details.



1 Home

2 Details library

3 Filter

4 Menu

5 List view / Grid view

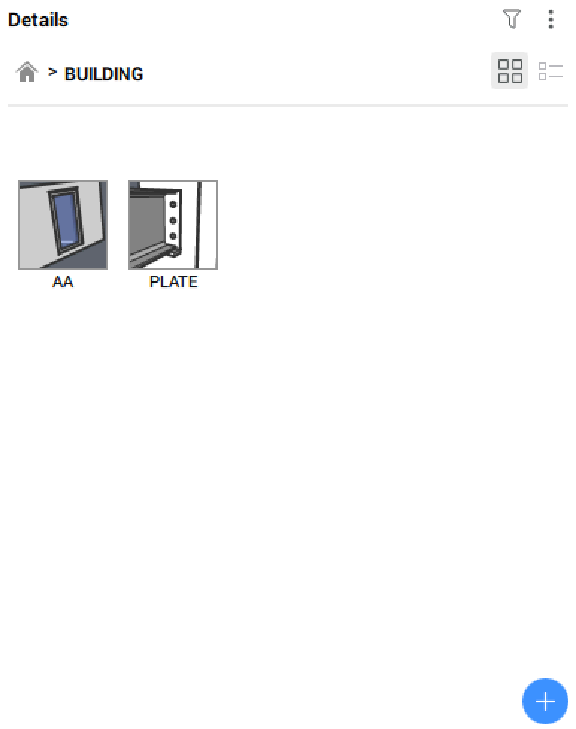
6 Create detail

35.16.1 Home

Click this button to go back to the details library overview.

35.16.2 Details library

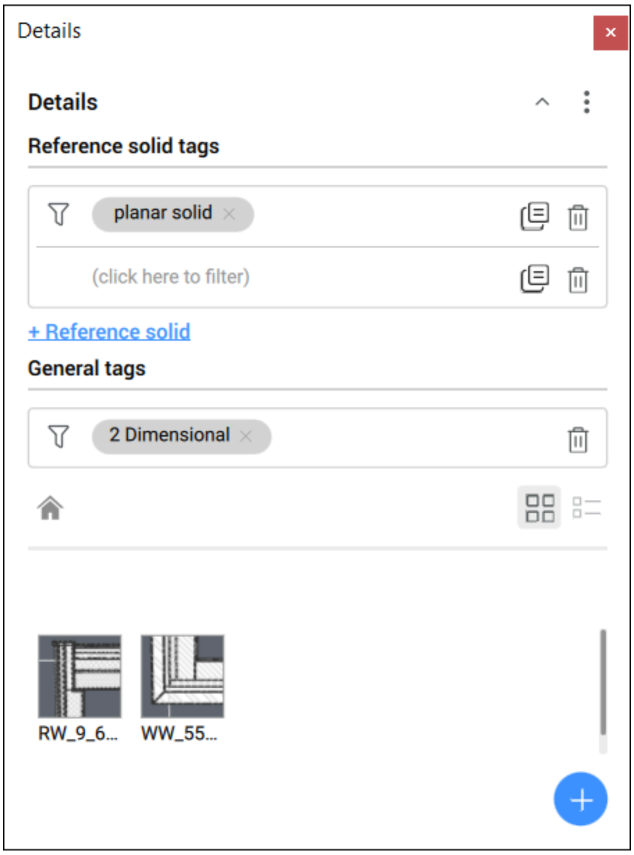
You can click on the category of the detail you want to view. The details assigned to that category will be displayed.



When you click on a detail, the **View Detail** dialog box will appear (see **View Detail dialog box** article).

35.16.3 Filter

Click the filter icon to open the filter options. Here you can search the library for specific details using tags.



Reference solid tags
Filters the details based on Reference solid tags. Click **+ Reference solid** to add a filter and then select a filter from the drop-down list. You can copy or delete filters by clicking on the symbols on the right.

General tags
Filters the details based on General tags. Click on **(click here to filter)** to add filters. Select a tag from the drop-down list. Click the cross next to a filter to delete a filter, or click the delete icon to the left to delete all the added filters at once.

35.16.4 Menu

Generate thumbnails
Loads the new details in your **Library** panel.

Manage libraries
Opens the **Settings** dialog box to change the details directory path.

35.16.5 List view / Grid view

Toggles between the list view and the grid view of the details by clicking the icons.

35.16.6 Create detail

Launches the BIMCREATEDETAIL command.

35.17 Drawing Recovery Manager panel

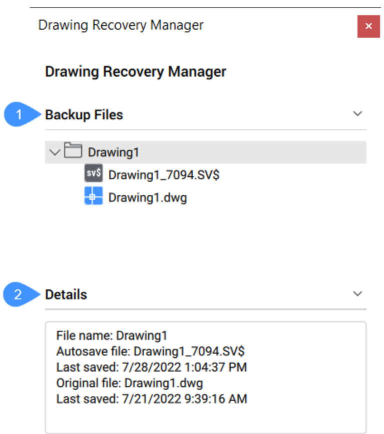
To open the **Drawing Recovery Manager** panel, run the DRAWINGRECOVERY command or right-click a ribbon panel/toolbar and choose **Drawing Recovery Manager** from the **Panels** list.

The **Drawing Recovery Manager** panel displays a list of drawing files that can be recovered after a program or system failure.

After a program or system failure, the **Drawing Recovery Manager** panel appears automatically when needed and displays a list of all drawing files that were open, including DWG, DWT and DWS files. A new unsaved drawing can be found in the **Drawing Recovery Manager** panel if the corresponding autosave file (.sv$) was created while editing the new drawing.

Note: You can disable the balloon notification display by toggling off one of the TRAYICONS and TRAYNOTIFY system variables. In this case a drawing recovery message dialog box appears on the Start page instead.

Note: The DRAWINGRECOVERY and DRAWINGRECOVERYHIDE commands can be used to show/hide this panel.



1 Backup Files

2 Details

35.17.1 Backup Files

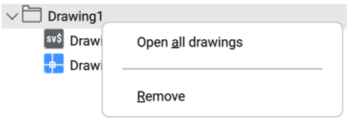
Displays a list of folders containing the drawing files that can be recovered. For each drawing, you choose from the following file types, if they exist:

- DrawingFileName_recover.dwg
- DrawingFileName_a_b_nnnn.sv$
- DrawingFileName.dwg
- DrawingFileName.bak

Recovered drawings are added to the **Recent Files** list and removed from the **Drawing Recovery Manager** panel the next time the program is launched.

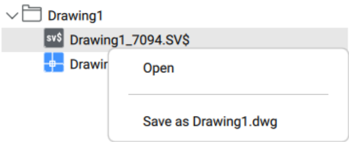
Right-click any item to display the context menu to access different actions depending on the selected item.

Folder:



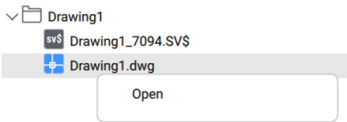
- **Open all drawings**: opens all files from that folder.
- **Remove**: removes the folder from the panel without recovering the files.

Sv$ file:



- **Open**: opens the file.
- **Save as .dwg**: saves the file as a dwg file.

Dwg file:



- **Open**: opens the dwg file.

35.17.2 Details

Displays a few details for the selected file, such as the name, location and size of the file and the time it was last saved.

35.18 Interference Manager panel (Experimental)

35.18.1 Disclaimer

EXP.

The **Interference Manager** panel is an experimental feature which may not yet be stable, and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

35.18.2 Description

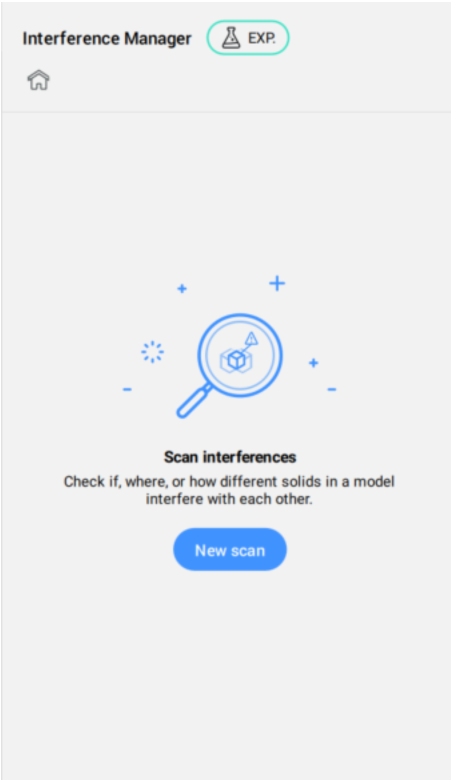
The **Interference Manager** panel allows you to identify and manage interference between 3D solids or block references in complex 3D models.

When the **Experimental mode** is enabled, the **Interference Manager** panel is active.

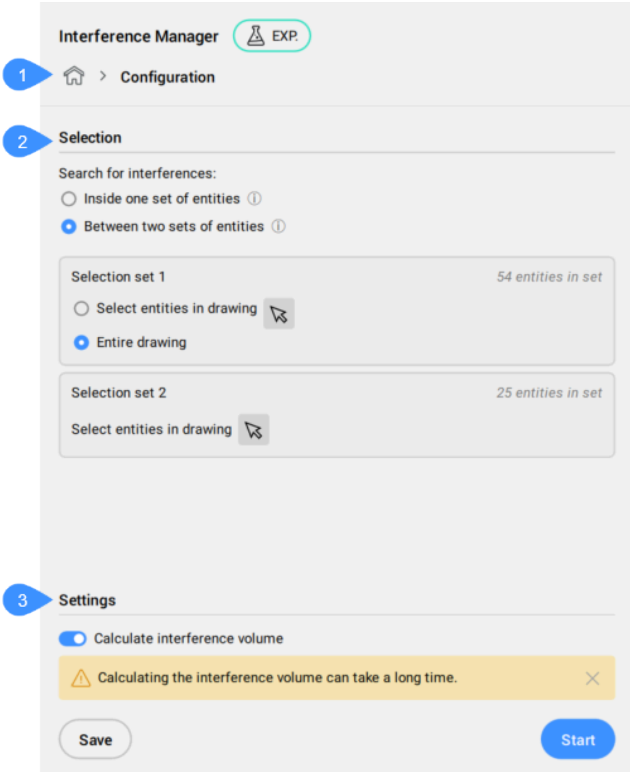
To open the **Interference Manager** panel:

- Launch the INTERFERENCEMANAGERPANELOPEN command.
- Right-click a ribbon panel/toolbar and choose **Interference Manager** from the **Panels** list.
- Launch the SCANINTERFERENCES command.

To close the **Interference Manager** panel, launch the INTERFERENCEMANAGERPANELCLOSE command.



Press the **New scan** button to open the **Configuration** window that allows you to start scanning interferences.



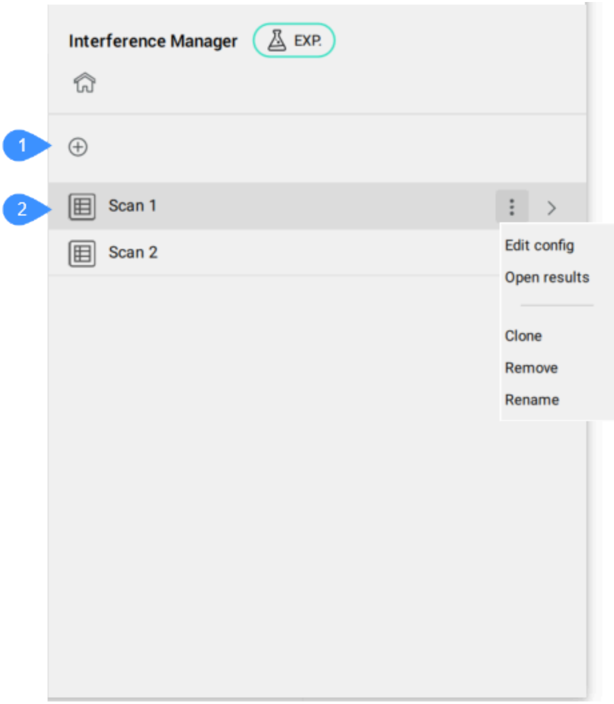
1 Home

2 Selection

3 Settings

35.18.3 Home

Restarts the interference scanning or opens the **Home** window that contains the saved scans.



1 New scan

2 List of saved scans

New scan
Opens the **Configuration** window to start a new scan.

List of saved scans
Lists the saved scans.
Right-click a saved scan to open a context menu with the following options.

Edit config
Opens the **Configuration** window to edit the scan.

Open results
Opens the **Result** window for the scan.

Clone
Creates a clone of the scan.

Remove
Removes the scan from the list.

Rename
Renames the scan.

35.18.4 Selection

Defines the entities that will be searched for interferences.

Inside one set of entities
Searches for interference in a specific set of entities.

Between two sets of entities
Searches for interference between two distinct sets of entities.

Selection set
Defines the entity selection set.

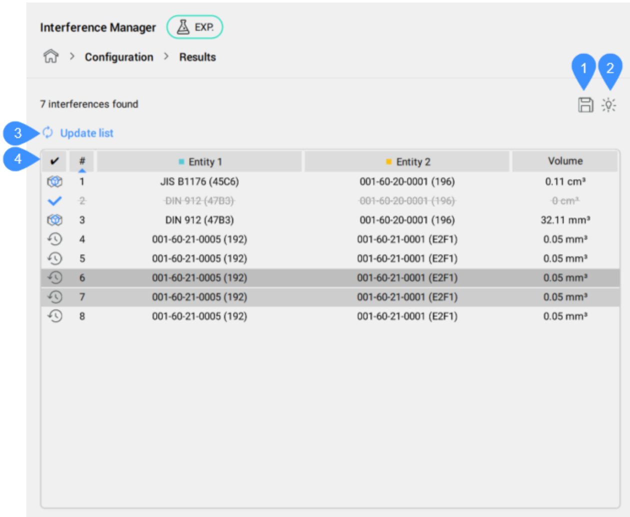
Select entities in the drawing
Allows you to select the entities directly from the drawing area.

Entire drawing
Selects all entities in the drawing.

35.18.5 Settings

Calculate interference volume
Toggles the option to calculate the volume of interference.

35.18.6 Results



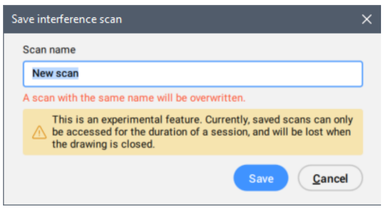
1 Save

2 Highlight the interfering entities

3 Update list

4 List of the entities that interfere

Save
Saves the configuration and results of this scan. The **Save interference scan** dialog box opens.



Note: Saved scans can only be accessed during the current session and will be lost when the drawing is closed.

Highlight the interfering entities
Toggles the display of the interfering entities and the interference volumes in distinct colors, while the rest of the entities are transparent. Interfering entities are colored in cyan and yellow, while the interference volume is red.

Update list
Re-evaluates all outdated interference pairs in the result lists.

List of the entities that interfere
Lists all interference pairs found between entities and displays the interference volumes.

The list is sortable by any of its columns. Easily navigate through interference pairs using the up and down keys. Select one or multiple rows to visualize their interferences simultaneously on the model.

While keeping the **Results** window open as a checklist, you can execute commands, such as DMPUSHPULL or MANIPULATE to resolve geometric interference.

Status
Displays the status of the interfering entities.

Interference
Entities are interfering.

Outdated
One of the interfering entities might be updated.

Resolved
Interference between entities has been resolved.

Cannot be found
One of the interfering entities cannot be found. It might be deleted or unloaded.

Position
Displays the position number of the pair of interfering entities.

Entity 1
Displays the first entity in the pair of interfering entities. It is colored cyan in the model.

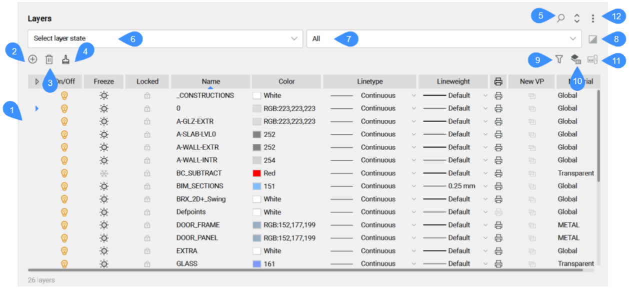
Entity 2
Displays the second entity in the pair of interfering entities. It is colored yellow in the model.

Volume
Displays the value of the interference volume. The interference volume is colored red in the model.

35.19 Layers panel

To open the **Layers** panel, run the LAYERSPANELOPEN command or right-click a ribbon panel/toolbar and choose **Layers** from the **Panels** list.

The **Layers** panel allows you to edit layer names and properties in the current drawing, and to create, delete, edit, switch layers, layer states and layer filters.



1 Layer list
2 Add layer
3 Delete
4 Purge
5 Search
6 Layer State Control
7 Filter used
8 Invert filter
9 Layer Explorer
10 Layer States
11 Hide xref symbols on/off
12 Menu

35.19.1 Layer list

Displays a list of layers and their properties. The list can include all layers in the current drawing or a subset of those layers based on selected layer states or filters. Column headers describe the properties of each layer. You can sort the layer list in order or reverse order for any of the properties by clicking once or twice on the column header.

A context menu displays when you right click menu on a layer name.

New
Creates a new layer with the generic name **NewLayer1**. The new layer that is created copies the properties of the currently selected layer.

Set Current
Specifies the current working layer

Rename
Renames the selected layer

Delete
Deletes the selected layer

Purge
Purges layers from the current drawing. You cannot purge the following layers:

- The layer 0.

- The current layer.
- Any layer with entities.

Merge to
Merges the selected layers to a target layer.

Note: Merged layers will be purged from the drawing.

Bind
Binds the selected layer(s) from Xrefs to the drawing.

Select All
Selects all available layers. You can also use the keyboard shortcut Ctrl+A.

Clear All
Deselects all selected layers.

Select All but Current
Selects all available layers except the current layer.

Invert Selection
Inverts the selected layers.

Remove Viewport Overrides
Removes overrides from properties that were set differently for viewports.

- **From Selected Layers**: removes the overrides from the selected layers.
- **From All Layers**: removes the overrides from all layers.
- **For Current Viewport**: removes overrides from the current viewport.
- **For All Viewports**: removes overrides from all viewports in the drawing

Isolate Selected Layers
Isolates the selected layers. All other layers are either locked or turned off.

Indicate Layers in Use
If the option is active, it displays a check mark in the **Current** column:

- **Used**: the layer is used in the drawing.
- **Unused**: the layer is not used in the drawing.

Right-click on the column headers to customize them. You can enable or disable the columns and restore the column positions.

Index
Specifies the layer's index.

Current

Specifies the current working layer. A check mark ( ) appears for the current layer.

Note: Only one layer can be current at a time.

On/Off

Specifies the On/Off status of the layers. Entities on layers are turned off are not visible in the drawing. However, they are processed during operations that require a regeneration of the drawing. Turning a layer on and off does not require the drawing to regenerate. Therefore, this toggle is most useful when you want entities on a layer to be hidden for only a short time while you edit the drawing. To change the status, select one or more layers and then click a light bulb icon for one of the selected layers. The status is applied to all selected layers.

- **On** ( ): displays entities that are on that layer.
- **Off** ( ): hides entities that are on that layer.

Freeze

Specifies the Thaw/Freeze status of the layers. Entities on frozen layers are not visible in the drawing and are not processed during operations that require a regeneration of the drawing. This can save processing time if the layer includes many entities. However, the process of freezing and thawing a layer requires the drawing to regenerate. Therefore, this toggle is most useful when you want entities on a layer to be hidden for an extended time while you edit the drawing. To change the status, select one or more layers, and then click a freeze icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw** ( ): displays entities that are on that layer.
- **Freeze** ( ): hides entities that are on that layer.

Note: Freezing and thawing layers is similar to turning them On and Off. However, when working with drawings with many layers, freezing unnecessary layers helps speed up display and regeneration. Entities on a frozen layer are not taken into account during a zoom to extents.

Locked

Specifies the lock status of the layers. Entities on locked layers cannot be edited. They remain visible. However, they are dimmed based on the LAYLOCKFADECTL System Variable. To change the status, select one or more layers, and then click a padlock icon for one of the selected layers. The status is applied to all selected layers.

- **Unlocked** ( ): allows entities on that layer to be edited.
- **Locked** ( ): prevents entities on that layer from being edited.

Name

Specifies the layer name. To rename a layer, double-click the layer name or select **Rename** from the right-click menu.

Description

(Optional) Specifies the content or purpose of the layer. To create or edit a description, select one or more layers and then double-click the description field for one of the selected layers. The text you enter is applied to all selected layers.

Color

Specifies the color of the layers. To change the color, select one or more layers and then click the current color for one of the selected layers. The **Color** dialog box is displayed. The color you select is applied to all selected layers.

Linetype

Specifies the linetype of the layers. To change the linetype, select one or more layers and then double-click the linetype for one of the selected layers. If the linetype you want is not displayed in the drop-down list, choose **Load** to display the **Load Linetypes** dialog box. The linetype you select is applied to all selected layers.

Lineweight

Specifies the Lineweight of the layers. To change the lineweight, select one or more layers and then double-click the lineweight for one of the selected layers. The lineweight you select is applied to all selected layers.

Transparency

Specifies the transparency of the layers. To change the amount of transparency, select one or more layers and then double-click the transparency for one of the selected layers. You can enter a value between 0 (fully opaque) and 90 (almost fully transparent) or select a predefined value from the drop-down list. The value you specify is applied to all selected layers.

Note: The TRANSPARENCYDISPLAY system variable must be turned on to experience transparency.

Plot style

Specifies the plot style of the layers. If plot style mode (PSTYLEMODE system variable) for the current drawing is set to color dependent plot styles, the layers' plot styles use the layers' colors and are read-only. If plot style mode for the current drawing is set to named plot styles, you can change the plot style. To specify a different named plot style, select one or more layers and then click the current plot style for one of the selected layers. The **Select Plot Style** dialog box is displayed. The plot style you select is applied to all selected layers.

Plot

Specifies the print status of the layers. Entities on **No Print** layers remain visible in the drawing but are not printed. To change the status, select one or more layers, and then click the printer icon for one of the selected layers. The status is applied to all selected layers.

- **Print** (the printer icon is highlighted): Prints the layer.
- **No Print** (the printer icon is not highlighted): Does not print the layer.

Some layers created by BricsCAD, such as Defpoints, are set to **No Print** by default and cannot be changed.

New VP

Specifies the default Thaw/Freeze status of layers for new layout viewports. To change the status, select one or more layers and then click the VP icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw** (the VP icon is not highlighted): displays entities that are on that layer when a new viewport is created.
- **Freeze** (the VP icon is highlighted): hides entities that are on that layer when a new viewport is created.

Material

Specifies the material of the layers. To change the material, select one or more layers and then double-click the material for one of the selected layers. The material you select is applied to all selected layers. If the material you want is not displayed in the drop-down list, you can add it to the current drawing using the MATERIALS command.

VP Freeze

Specifies the Thaw/Freeze status of layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space Thaw/Freeze status. To change the status, select one or more layers and then click the VP Freeze icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw** (): displays entities that are on that layer in the current layout or viewport.
- **Freeze** (): hides entities that are on that layer in the current layout or viewport.

VP Color

Specifies the color of layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space color.
To change the color, select one or more layers and then click the current vp color for one of the selected layers. The **Color** dialog box is displayed. The color you select is applied to all selected layers.

VP Linetype

Specifies the linetype of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space linetype.
To change the linetype, select one or more layers and then double-click the vp linetype for one of the selected layers. If the linetype you want is not displayed in the drop-down list, choose Load to display the **Load Linetypes** dialog box. The linetype you select is applied to all selected layers.

VP Lineweight

Specifies the Lineweight of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space lineweight.
To change the lineweight, select one or more layers and then double-click the vp lineweight for one of the selected layers. The lineweight you select is applied to all selected layers.

VP Transparency

Specifies the transparency of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space transparency.
To change the amount of transparency, select one or more layers and then double-click the transparency for one of the selected layers. You can enter a value between 0 (fully opaque) and 90 (almost fully transparent) or select a predefined value from the drop-down list. The value you specify is applied to all selected layers.

Note: The TRANSPARENCYDISPLAY system variable must be turned on to experience transparency.

VP Plot Style

Specifies the plot style of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space plot style. If plot style mode (PSTYLEMODE system variable) for the current drawing is set to color dependent plot styles, the layers' plot styles use the layers' colors and are read-only.
If plot style mode for the current drawing is set to named plot styles, you can change the plot style. To specify a different named plot style, select one or more layers and then click the current plot style for one of the selected layers. The **Select Plot Style** dialog box is displayed. The plot style you select is applied to all selected layers.

35.19.2 Add layer

Creates new layer with the generic name **NewLayer1**. The new layer that is created copies the properties of the currently selected layer.

35.19.3 Delete

Deletes layers, except for the layer with the name **0** and any layer with entities, which cannot be deleted.

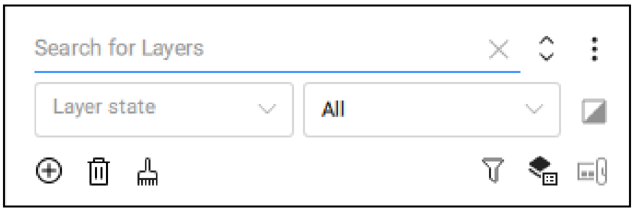
35.19.4 Purge

Purges layers from the current drawing, except for the following ones, which cannot be purged:

- The layer 0.
- The current layer.
- Any layer with entities.

35.19.5 Search

Allows you to search for specific layer names. The list is shortened to display only the layers that contain the entered sequence of characters, regardless of where they are located in the layer name.



Note: You can remove the active filter by pressing the close button on the right side of the search box.

35.19.6 Layer State Control

Displays the current layer state if there are any defined in the drawing. Click the down-arrow button to choose a different layer state.

35.19.7 Filter used

Displays the current layer filter if there are any defined in the drawing. Click the down-arrow button to choose a different layer filter. Some filters are automatically created by BricsCAD: All, All Used Layers.

35.19.8 Invert filter

Inverts the content of the current filter so that the panel displays all layers not included in the filter.

35.19.9 Layers Explorer

Displays the **Drawing Explorer** dialog box with the **Layers** category selected.

35.19.10 Layer States Explorer

Displays the **Drawing Explorer** dialog box with the **Layer States** category selected.

35.19.11 Hide xref symbols on/off

Allows you to toggle On/Off the display of xref layers.

35.19.12 Menu

Hide xref layers
When on, hides layers that came from externally referenced drawings.

Apply layer filter to layer toolbar
When on, shortens the list of layers displayed, according to the conditions of the filter. When off, displays all layer names.

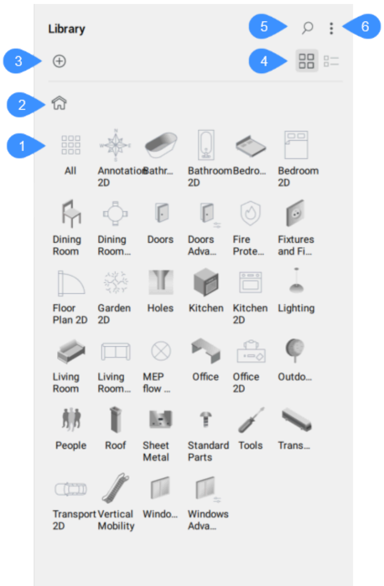
Indicate layers in use
When on, displays icons in the **Current** column that indicate whether the layer has entities assigned to it. Unused layers can be purged.

It is possible to change the row height of the table view, by switching between **Compact View**, **Default View** and **Comfortable View**.

35.20 Library panel

To open the **Library** panel, run the LIBRARYPANELOPEN command or right-click a ribbon panel/toolbar and choose **Library** from the **Panels** list.

The **Library** panel offers a central location to access 2D and 3D block libraries.



1 Block library

2 Home

3 Add block to library

4 View

5 Search

6 Menu

35.20.1 Block library

The block library is organized by categories. Select a category to view its blocks. Categories and blocks may be user-defined or pre-defined by BricsCAD. You can scroll through the categories and blocks using the mouse wheel or the scroll bar. The availability of pre-defined blocks depends on your license level.

35.20.2 Home

Returns the block library to the home screen.

35.20.3 Add block to library

Opens the **Add block to library** dialog box.

Note: For information on how to use the **Library** panel, see the article **Using the library panel**.

35.20.4 View

Switches between grid and list views of the block content.

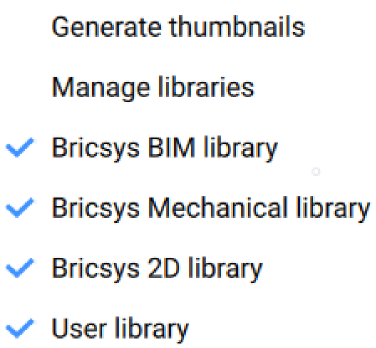
35.20.5 Search

Searches the library for words that you enter in the search box. Only displays the components that contain the entered sequence of characters.

Note: You can remove the active filter by pressing the close button on the right side of the search box.

35.20.6 Menu

The **Library** panel menu enables you to control what is displayed in the block library.



Generate thumbnails
Generates or updates the thumbnail images of all blocks.

Manage libraries
Opens the **Settings** dialog box to change the library directory path.

Bricsys BIM library
When ticked, the panel displays BIM components. The default path for BIM components is *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support\en_US\Bim\Components*.

Note: The parametric components under **Windows Advanced** and **Doors Advanced** folders allow for greater customization and contain hinge markers to be displayed on 2D sheets.

Bricsys Mechanical library
When ticked, the panel displays Mechanical components. The default path for Mechanical components is *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support\en_US\DesignLibrary*.

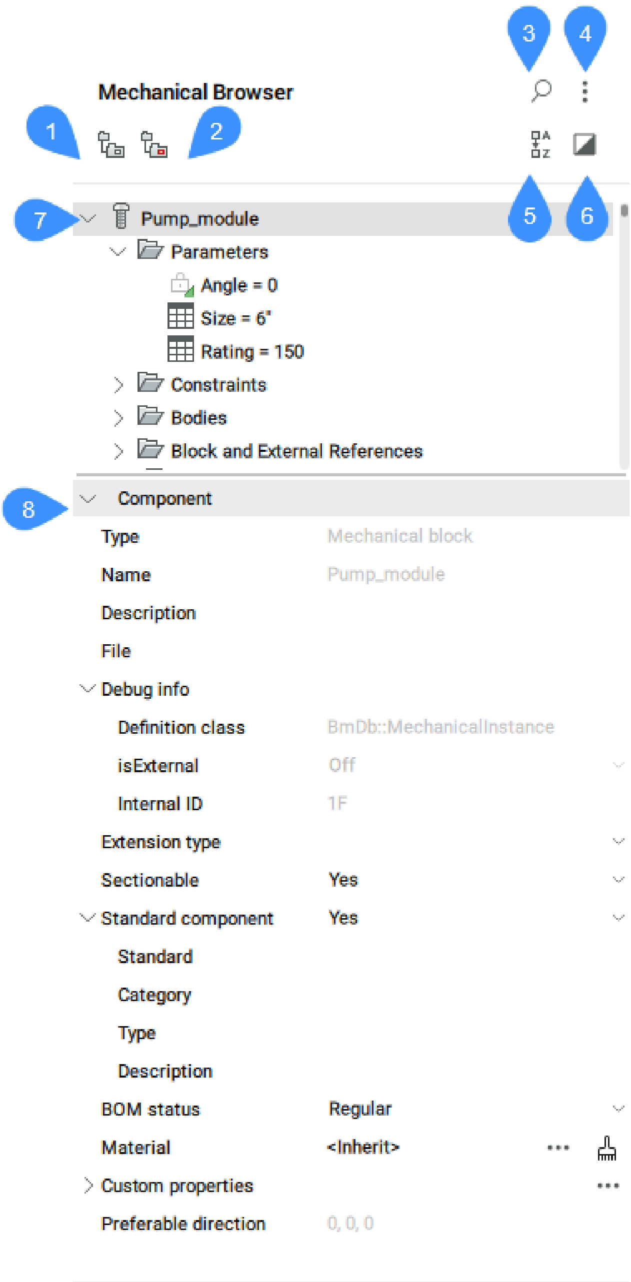
Bricsys 2D library
When ticked, the panel displays the content of the 2D library. The default path for the 2D library is *C:\Program Files\Bricsys\BricsCAD V25 en_US\UserDataCache\Support\en_US\Blocks2D*.

User library
When ticked, the panel displays the user defined components. The path for user library is defined by the COMPONENTSPATH system variable, which is by default *C:\ProgramData\Bricsys\Components*.
Edit the COMPONENTSPATH system variable in the **Settings** dialog box to add more paths. User library folders are added in the **Library** panel menu, where you can control their availability in the **Library** panel. When the path list is empty, the default path is added automatically.

35.21 Mechanical Browser panel

To open the **Mechanical Browser** panel, run the MECHANICALBROWSEROPEN command or right-click a ribbon panel/toolbar and choose **Mechanical Browser** from the **Panels** list.

The **Mechanical Browser** panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.



1 Group by entity

2 Group by type

3 Show search

4 Settings

5 Sort Alphabetically

6 Show selected

7 Mechanical entities list

8 Properties

35.21.1 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

35.21.2 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

35.21.3 Show search

Opens the search field and the associated tools.

Search field
Searches the mechanical browser tree for the character string you enter. The search acts as a filter, so that the browser shows all components whose names or component names (or part of the names or component names) match the character string. Press **X** on the right of the search field to close the field.

35.21.4 Settings

Expressions of constraints
Controls whether the numeric value or the assigned parameter name displays.

Components parameters
Controls the visibility of sub-component parameters.

Expressions of components parameters
Controls the visual representation of sub-component parameters: numeric value or assigned parameter name.

Sub-components of standard parts
Controls the visibility of sub-components of standard parts which are mechanical assemblies.

Arrays of entities
Toggles the display of Array nodes in the browser tree, including Components arrays.

Block and External references
Toggles the display of Blocks and External References nodes in the browser tree. Also, toggles node building in the **Bodies** folder for Block entities.

Always synchronize selection
Toggles the synchronization of the selection nodes in the **Mechanical Browser** with the selection of the corresponding objects in the model space. When an object is removed from a selection, only nodes corresponding to the objects remaining in the selection remain selected. If no objects are selected, the root node will be selected (similarly to when a document is open).

Keep values list order
Toggles the sorting of the values list for parameters inside the drop-down list.

Highlight selections
Toggles the highlighting of the selected entities in the drawing.

Asynchronous properties load
Controls if the **Mechanical Browser** collects properties asynchronously if more than one node is selected in the Mechanical Browser tree.

Expose all solids
Toggles the display of solids for mechanical blocks and solids not related to constraints under the **Bodies** folder.

Note: **Mechanical Browser** settings are controlled also by the MECHANICALBROWSERSETTINGS system variable.

35.21.5 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

35.21.6 Show selected

When enabled, only nodes corresponding to selected objects or their parents are displayed.

When enabled and no objects are selected, or selection is cleared, the full model is shown in the browser.

The state of the **Show selected** mode is remembered between sessions, and is the same for all opened documents.

35.21.7 Mechanical entities list

Displays a hierarchical tree list of mechanical entities, such as user parameters, constraints, parts, arrays and features. Top-level nodes can have sub-nodes. For example, constraint node have sub-node(s) for its argument(s). An array node has sub-nodes for its arguments, etc.

Double click a node to expand or collapse it, double click (click-pause-click) a node to edit its label if applicable. Not all node names can be edited.

Each node in the list has associated context menu. Right-click a node to invoke the menu. The content of the menu can be different for each kind of node. You can also select a group of nodes and invoke a context menu for them. The menu will contain only those elements which are applicable to each selected node.

Context menu for the root component contains the following items:

Update
Updates the hierarchy of mechanical components for the current drawing in case the referenced drawing files of sub-components have been modified.

Visual style > All by Viewport
Applies the current visual style to all components in the assembly.

Switch all to local
Converts all external components in the model to local components.

Switch all to external…
Switches all internal components to external components.

Add new parameter
Creates a new parameter in the assembly.

Custom properties
Opens the **Mechanical Properties** dialog box to define custom properties for the drawing.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same name and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlight all components with the same name and the same parameter values.

Create sequence
Creates a block with a representation of the current assembly.

Renumerate child nodes
Renumerates all child nodes according to their types.

Renumerate child nodes continuously
Renumerates all child nodes with continuous numeration when applicable.

Collapse all
Collapses the main component and all components and subcomponents.

Expand all
Expands the main component and all components and subcomponents.

Context menu for child (non-root) components, internal or external, also offers:

Open
Opens the referenced drawing (see the BMOPEN command).

Open a copy
Opens a copy of a component insert as a new drawing (see the BMOPENCOPY command).

Update
Reloads all referenced components from external files and updates BOM tables (see the BMUPDATE command).

Replace…
Replaces a component insert (see the BMREPLACE command).

Note: Replacing a local insert turns it into an external insert.

Replace all inserts...
Replaces all inserts that refer to the same source (see the BMREPLACE command).

Switch to local
Switches an external component to an internal component (see the BMLOCALIZE command).

Switch to external
Switches an internal component to an external component (see the BMEXTERNALIZE command).

Dissolve
The selected feature is removed from the part, but it will keep its geometry. However, the design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

Custom properties
Opens the **Mechanical Properties** dialog box to define custom properties for the drawing.

Reset instance custom properties
Resets instance custom properties to defaults inherited from a corresponding block.

Hide / Show
Hides / Shows the component.

Visual Style
Sets a visual style for the component.

Zoom to
Zooms to the selection set.

Select
Selects the component.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same definition and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlights all components with the same definition and the same parameter values.

Delete
Analogue to the SMDELETE command. In this case, the feature is removed from the browser and the geometry is changed depending on the type of the feature.

Set material to component
Opens the **Physical Materials** dialog box, which allows you to assign a physical material to the component.

Remove material from component...
Removes the physical material definition from a local component.

Make component standard / Make component non-standard
Changes the type of the component.

Include all inserts to section
Sets the **Sectionable** property of all similar inserts to **YES**. Defines whether an insert is affected by the VIEWSECTION command.

Exclude all inserts to section
Sets the **Sectionable** property of all similar inserts to **NO**. Defines whether an insert is affected by the VIEWSECTION command.

Renumerate similar nodes
Renumerates all nodes of the same type and the same level.

Component's BOM status
Controls the appearance of the component in BOM tables.

Context menu for Parameters contains additional tools:

Geometry-driven
If ticked, it makes the parameter geometry-driven.

Note: If the parameter is assigned to a geometry-driven 3D constraint, its boundaries are taken into account while moving the constrained geometry.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates models by means of parameters.

Context menu for Component parameters contains additional tools:

Link to parameter
Links subcomponent parameter to the main level parameter.

Context menu for Constraints contains additional tools:

Enabled
Toggles the enabling of the constraint.

Flip side
Flips the side of the constraint (not available for fix constraints).

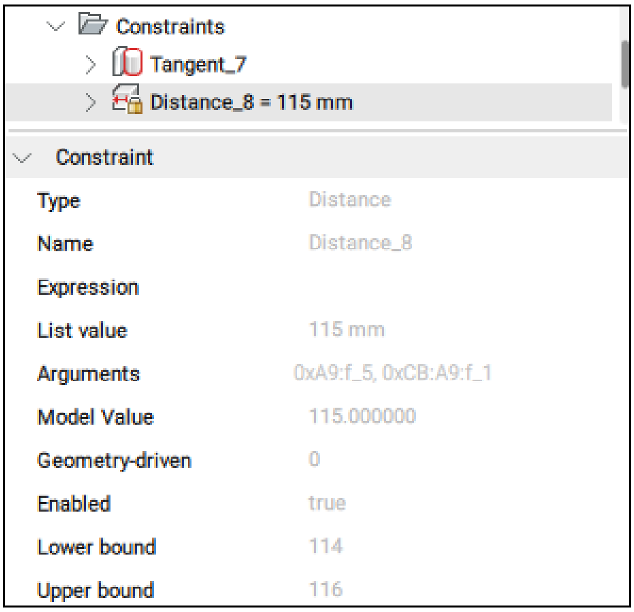
Select geometry
Selects the entities involved in the constraint.

Context menu for dimensional Constraints contains additional tools:

Geometry-driven
If ticked, it makes the constraint geometry-driven.

Note: If the parameter is assigned to a geometry-driven 3D constraint, its boundaries are taken into account while moving the constrained geometry.

Note: You can set upper and lower bounds in the **Mechanical Browser** panel, which are taken into account in dynamic operations, like DMMOVE and ROTATE3D.



Create design table
Creates a design table to drive parametric block parameters (see the **Parameters Manager** panel).

Context menu for constraint arguments contains additional tools:

Flip
Reverses direction.

Replace...
Replaces the current argument with a another one (see the DMCONSTRAINT3D command).

Exclude
Excludes the current argument.

35.21.8 Properties

Displays the properties of the selected item(s). If several nodes are selected in the tree, the Properties section displays the properties of all the selected nodes. Changing a property modifies all the selected nodes.

Change a property by clicking a numeric field to edit its value or by selecting another option from the drop-down list, which is opened when you press the down arrow from the right side of the field.

The most important properties of a mechanical block are:

- **Extension type**: defines the extension type for the mechanical block.
- **Sectionable**: defines if the mechanical block is sectionable.
- **Standard component**: defines if the mechanical block is a standard component.
- **BOM status**: defines the BOM status.
- **Material**: defines the material.

  Note: Click the browse button (...) to open the **Physical Materials** dialog box.

- **Custom properties**: displays the custom properties.

Note:

- Click the browse button (…) to open the **Mechanical Properties** dialog box.
- Click the **Clear material** button to remove the material.

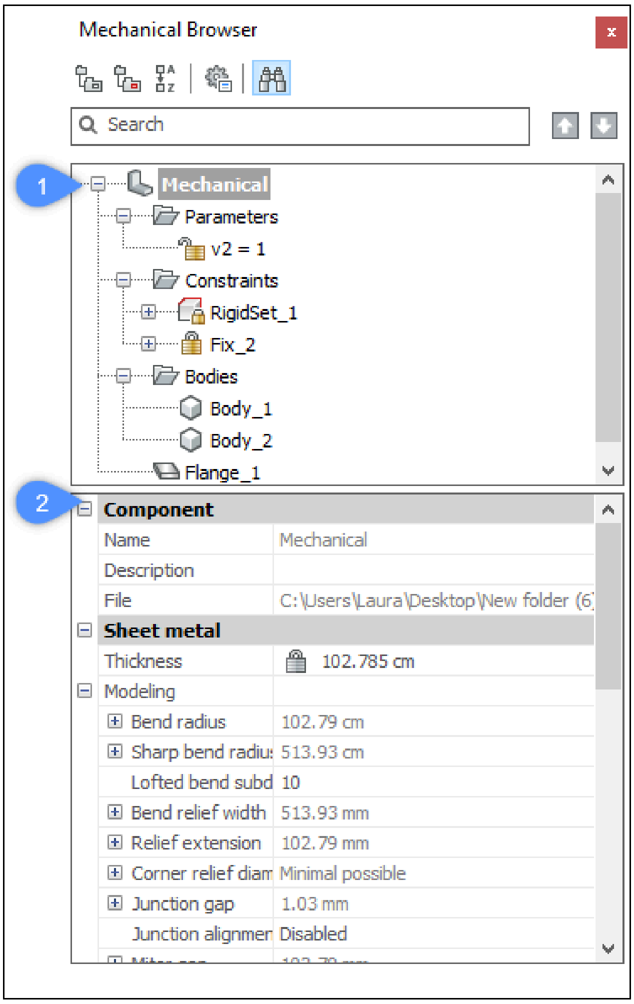
- **Preferable direction**: indicates the direction for exploded views, which is set with the BMEXPLODECONFIG command.

Note: Select a Sheet Metal Body to display the **Measurements** properties in the **Mechanical Browser**.

Measurements	
Bend length	239.91 mm
Cut length	153.64 mm
Junction length	153.64 mm
Miter length	0.00 mm

35.22 Mechanical Browser (legacy) panel

The **Mechanical Browser (legacy)** panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.



1 Parameter list

2 Properties

35.22.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters, 2D dimensional constraints, 3D dimensional constraints, 3D geometric constraints, block parameters and array parameters.

A context menu displays when right-click the main component name:

Update
Updates the hierarchy of mechanical components for the current drawing in case referenced drawing files of sub-components have been modified.

Visual style > All by Viewport
Applies the current visual style to all components in the assembly.

Switch all to local
Converts all external components in the model to local components.

Switch all to external…
Switches all internal components to external components.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same name and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlight all components with the same name and the same parameter values.

Create exploded view
Creates a block with an exploded representation of the current assembly.

Dissolve
The selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

Delete
This is the analogue of the SMDELETE command. In this case the feature is removed from the browser and geometry is changed depending upon the type of the feature.

Disable
The feature will remain in the **Mechanical Browser**, but the part will behave like the feature was dissolved. It is beneficial in contrast with dissolve, that feature still updates on geometry changes and you can turn it on when you want, avoiding recreating it or recognizing by SMCONVERT.

Right-click menus offer additional tools for components:

Open
Opens the referenced drawing.

Open a copy
Opens a copy of a component insert as a new drawing.

Update
Reloads all referenced components from external files and updates BOM tables.

Replace…
Replaces a component insert.

Note: Replacing a local insert turns it into an external insert.

Set material to component
Opens the Physical Materials dialog box, which allows you to assign a physical material to the component.

BOM status
Controls the appearance of the component in BOM tables.

Right-click menus offer additional tools for parameters:

Geometry-driven
If checked, it makes the parameter geometry-driven.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates models by means of parameters.

Link to parameter
Links subcomponent parameter to the main level parameter.

> ***Note:*** Right-click on a 3D constraint argument and choose the Replace option to replace the current argument with a another one.

35.22.2 Properties

Specifies the properties of the selected item.

> ***Note:*** Select a Sheet Metal Body to display the **Measurements** properties in the **Mechanical Browser** panel.

Measurements	
Bend length	239.91 mm
Cut length	153.64 mm
Junction length	153.64 mm
Miter length	0.00 mm

35.22.3 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

35.22.4 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

35.22.5 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

35.22.6 Settings

Expressions of constraints
Controls whether the numeric value or the assigned parameter name displays.

Components parameters
Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.

Expressions of components parameters
Sub-component parameters control the visibility of sub-component parameters.

Sub-components of standard parts
Parameters at properties add a parameters section for the selected instance to the Mechanical browser properties

35.22.7 Show search

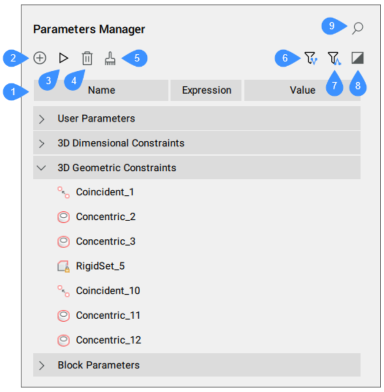
Toggles the search field and associated tools on and off. You can also press CTRL+F to toggle Search.

Search field
Searches the parameter list for the character string you enter. The arrow tools to the right of the search field enable you find the next or previous occurrence of the text string.

35.23 Parameters Manager panel

To open the **Parameters Manager** panel, run the PARAMETERSPANELOPEN command or right-click a ribbon panel/toolbar and choose **Parameters Manager** from the **Panels** list.

The **Parameters Manager** panel is the main tool to view and modify parameters and constraints in the current drawing.



1 Parameters list

2 New parameter

3 Animate

4 Delete

5 Clean unused variables

6 Show dependent parameters

7 Show defining parameters

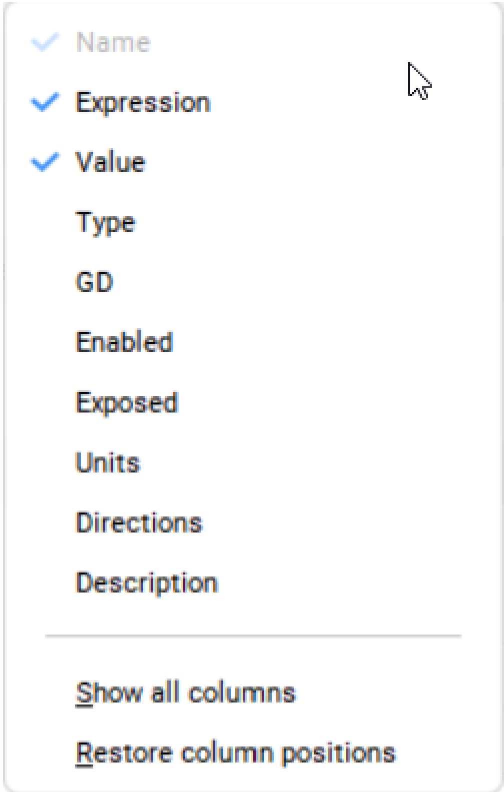
8 Show selected

9 Search

35.23.1 Parameter list

Displays a list of all the parameters in the drawing. The parameters are organized in expandable categories which can include user parameters (including the ones which are linked with Design Tables), 2D dimensional constraints, 3D dimensional constraints, reference parameters, 2D geometric constraints, 3D geometric constraints, block (components) parameters, and array parameters.

The parameter list header contains columns which represent the properties of the parameters. Columns can be moved to a new position, resized or hidden. You can arrange parameters in ascending or descending order by pressing a specific column head. The parameter list header has three columns by default: **Name**, **Expression** and **Value**. To add more columns, right click the columns header and select the desired columns from the list:



Note: Some properties are read-only (for example **Type**), others can be modified by the user (for example **Name**).

Name

Displays the name of the parameter.

Note: The name can be edited. When you rename an item, all instances of the item are updated in the drawing and in the **Parameters Manager** parameters list.

Expression

Specifies the mathematical expression of the parameter. You can edit the expression by clicking on the cell.

Value

Displays the value resulting from the parameter’s expression. The value is a read-only real number, unless the parameter is linked to a Design table. In that case, the parameter has a list of possible values represented as a combo-box. You can change the parameter's value by picking a new value from the list. The combo-box is searchable by typing a sub-string of the value to quickly find it. This is useful for large lists of values.

Type

Displays the parameter's type:

- Constraint type for constraints (Distance, Concentric etc.).
- Parameter's type for user and block parameters (Real, String).
- Parameter's type for array parameters (Real, Boolean, Enumeration).

GD (geometry-driven)

If ON, the parameter is geometry-driven.

Lower bound

Defines the lower value for the parameter.

Upper bound

Defines the upper value for the parameter.

Note: The upper and lower bounds are taken into account in dynamic operations, like DMMOVE and ROTATE3D.

Enabled

Defines whether a geometric constraint parameter is taken into consideration.

Exposed

Defines whether the parameter is visible and can be modified when the component is inserted in an assembly.

Units

Sets the dimension of the parameter. You can choose between: none, linear (mm), square (mm^2) and cubic (mm^3).

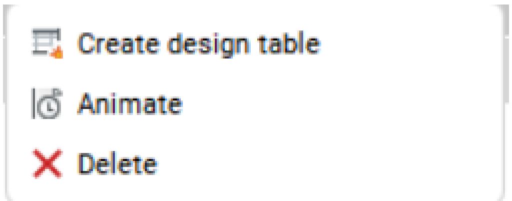
Directions

Defines the direction for a geometric constraint parameter (any, same, opposite or reset).

Description

Allows you to type a description for the user parameters.

Right-clicking on a selected parameter opens a specific context-menu that contains only those options, which apply to that item. These options can also be found in the context menu for the same item in the **Mechanical Browser** panel.



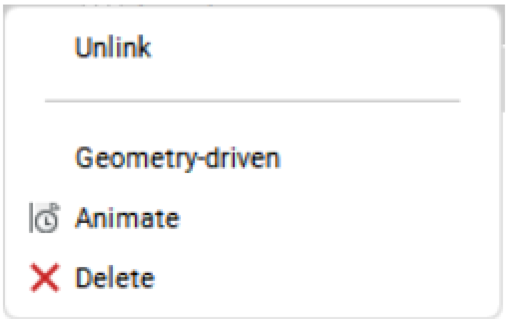
Create design table

Creates a design table to drive parametric block parameters.

Animate

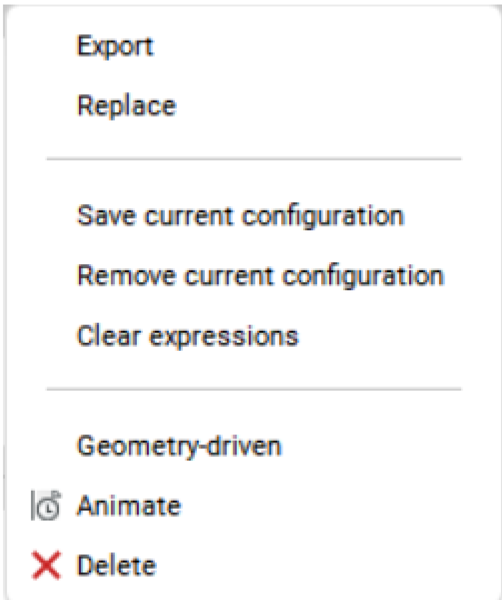
Animates the parameter value within some range.

Link to parameter
Links the subcomponent parameter to the main level parameter.



Unlink
Breaks the links between the subcomponent parameter and the main level parameter.

Geometry-driven
If ticked, the parameter is geometry-driven.



Note: If the parameter is assigned to a geometry-driven 3D constraint, its boundaries will be taken into account while moving the constrained geometry.

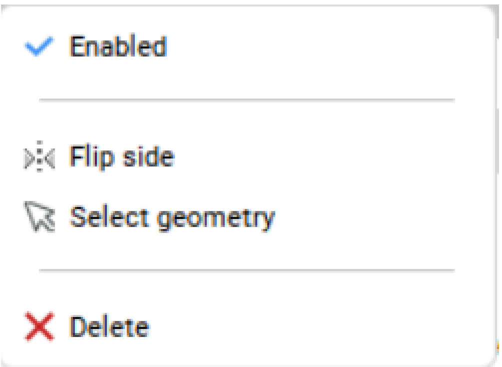
Export
Opens the **Select design table file** dialog box to save the design table.

Replace
Opens the **Select design table file** dialog box to choose a design table to replace the existing one.

Save current configuration
Saves the current design table configuration.

Remove current configuration
Removes the current design table configuration.

Clear expression
Removes the expression formula.



Enabled
Displays and toggles the enabled status of the constraint.

Flip side
Flips the side of the constraint.

Select geometry
Selects the entities involved in the selected constraint in the model space.

35.23.2 New parameter

Inserts a new user parameter with default values for **Expression** and **Value**.

35.23.3 Animate

Animates the selected parameter value within a range.

35.23.4 Delete

Removes selected parameters and constraints from the drawing.

35.23.5 Clean unused variables

Erases unused variables.

35.23.6 Show dependent parameters

When enabled, the panel shows parameters and constraints, depending on the selected parameter(s).

35.23.7 Show defining parameters

When enabled, the panel shows parameters which define the selected parameter(s).

35.23.8 Show selected

When enabled, the panel shows parameters and constraints for the objects currently selected in the model space, or all objects if nothing is selected.

35.23.9 Search

Filters the parameter list to display only the parameter names that match the character string you enter in the search box.

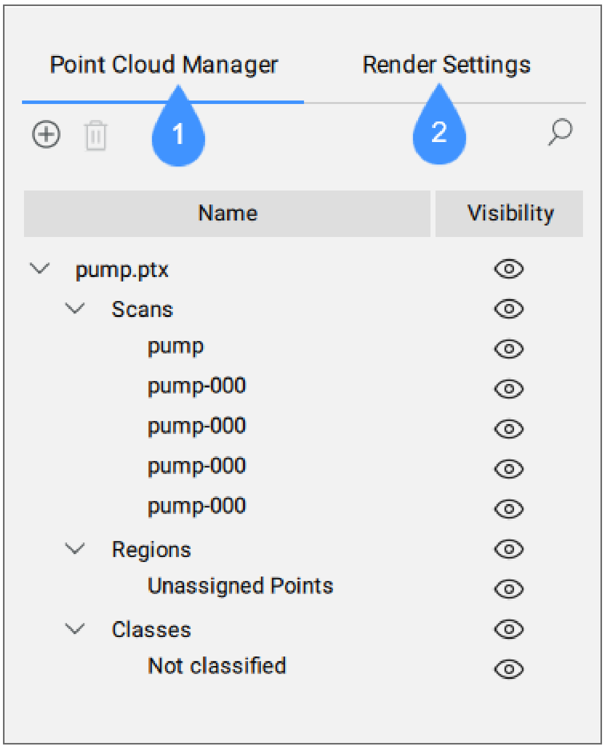
Note: You can remove the active filter by pressing the Close (X) button from the right side of the search box.

35.24 Point cloud manager panel

To open the **Point Cloud Manager** panel, run the POINTCLOUDMANAGERPANELOPEN command or right-click a ribbon panel/toolbar and choose **Point Cloud Manager** from the **Panels** list.

The **Point Cloud Manager** panel creates and manages regions in a point cloud and controls the display of point clouds based on scans/waypoints, classification and regions.

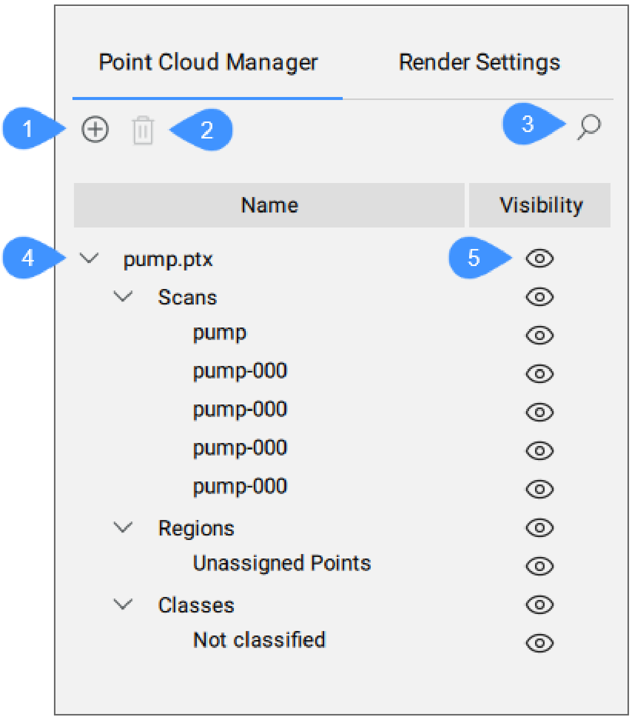
Note: Scan information will only be available if the point cloud is preprocessed in HSPC format. Only structured point clouds contain scan information. Classification information will be available if the point cloud data source contains classification information (some *.LAS/*.LAZ files) and it is preprocessed in HSPC, or if the point cloud is classified using the Hexagon point cloud classifier (POINTCLOUDCLASSIFY command).



1 Point Cloud Manager tab

2 Render Settings tab

35.24.1 Point Cloud Manager tab



1 New region

2 Remove region

3 Search

4 List of point clouds and their scans, regions, classes, and building information

5 Visibility

New region
Allows you to create a region by running the POINTCLOUDREGION command:

Draw region
Runs the **Draw** option of the POINTCLOUDREGION command.

Convert crop solids to region
Runs the **Convert** option of the POINTCLOUDREGION command.

Remove region
Removes the selected region.

Note: The **Unassigned Points** region cannot be removed.

Search
Allows you to search for a specific scan, class or region name. The list is shortened and only shows the items that contain the entered sequence of characters, regardless of where they are located in the item's name.

List of point clouds and their scans, regions, classes, and building information
Displays a tree view list of point clouds with their scans, regions, and classes. Building information (building/floor/room) is also displayed after the POINTCLOUDDETECTFLOORS and POINTCLOUDDETECTROOMS commands (BricsCAD BIM only) have been used.
Right-click an item in the list to open a context menu:

Note:

- The **Toggle Visibility** option, available in each item's context menu, allows you to switch On/Off the visibility of the selected item.
- Where available, the **Delete** option deletes the selected item/items from the list and removes the specific classification of the corresponding points.

Note: Deleting the **Buildings** category from the **Point Cloud Manager** panel removes all building-related information (building, floor, room) for the items listed in this category.

Scans

Rename

Allows you to type in a new name.

Note: Double click an item to activate the rename mode.

Open scan in bubble viewer

Opens the **Bubble viewer** displaying the selected scan.
Nearby scans are indicated in the viewer by red bubbles. Hover the cursor over a bubble to display the scan name. Double-click a bubble to open the scan in the viewer.

Note: Hover over scan items in the list to show a preview of the scan.

Regions

Rename

Allows you to type in a new name.

Note: Double click an item to activate the rename mode.

Classes

Add new class

Allows you to add a new class to the **Classes** list by selecting one from the predefined lists: **Lidar**, **Indoor**, **Construction**, or add a custom class using the **Custom...** option.

Note: Use the POINTCLOUDCLASSIFY command to automatically classify points.

Add points

Runs the POINTCLOUDMODIFYCLASS command.
Allows you to manually classify points under the selected class by selecting points in the point cloud. Two selection methods are available: **Rectangular** and **Polygonal**.

Select

Allows you to add more points to the selection set.

Finish

Finishes the selection process and classifies the points.

Note: Use the POINTCLOUDCOLORMAP command and select **Classification** for **Color Stylization** to better visualize how the points are classified.

Buildings

Add Building

Runs the **Add** option of the POINTCLOUDMODIFYBUILDING command.

Allows you to specify the area of points to be classified under a new building.

Building
Right-click a building item to display a context menu:

Modify
Runs the **Edit** option of the POINTCLOUDMODIFYBUILDING command.
Allows you to modify the volume of the building (points assigned) by moving the faces of the temporary crop solid.

Add Floor
Runs the **Add** option of the POINTCLOUDMODIFYFLOOR command.
Allows you to specify the elevation and height of the new floor.

Building floors
Right-click a floor item to display a context menu:

Modify
Runs the **Edit** option of the POINTCLOUDMODIFYFLOOR command.
Allows you to modify the volume of the floor (points assigned) by moving the faces of the temporary crop solid.

Detect Rooms
Automatically detects rooms in the selected floor (see POINTCLOUDDETECTROOMS command).

Add Room
Allows you to manually add a new room by selecting a point cloud area.
Enables the **Top** view and lets you define a closed polyline.

Building rooms
Right-click a room item to display a context menu:

Edit
Allows you to edit the contour of the selected room.
Enables the **Top** view and lets you reposition the vertices of the closed polyline.
To exit the editing mode, right-click the room item again and select **Accept** or **Decline**.

Rename
Allows you to type in a new room name.

Note:

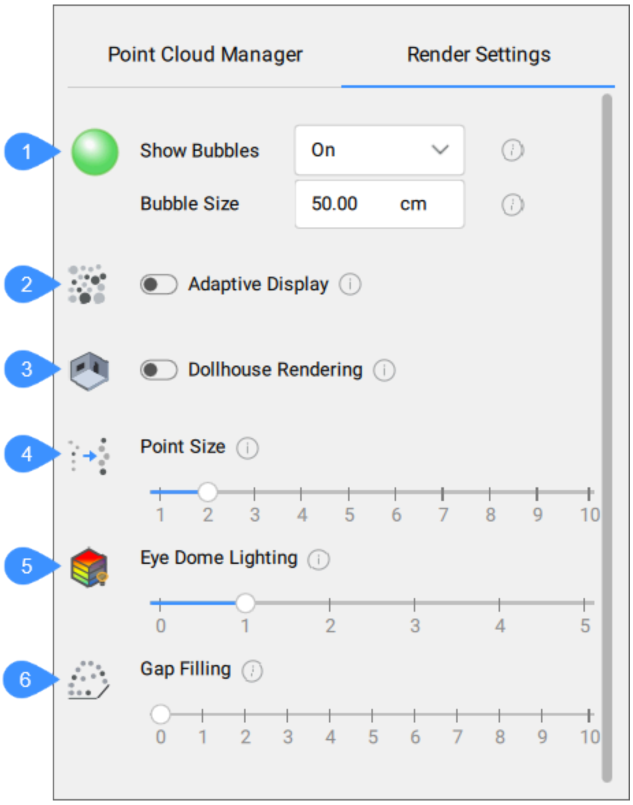
- This option can also be accessed by double-clicking a selected room's name.
- The room keeps the number assigned during the detection process.

Multiple building rooms
Select multiple room items and right-click to display a context menu:

Merge rooms
The points corresponding to the selected rooms are classified under a single room.

Visibility
Displays the current visibility state of listed items. Click the icon to switch On/Off the visibility of the selected items.

35.24.2 Render Settings tab



1 Bubbles settings

2 Adaptive Display

3 Dollhouse Rendering

4 Point Size

5 Eye Dome Lighting

6 Gap Filling

Bubbles settings

Show Bubbles ( )
Allows you to set the visibility of scan bubbles in Model space and **Bubble viewer**:
Off: hides all bubbles in the point cloud.
On: displays all bubbles in the point cloud.
Visible Only: displays non-cropped bubbles only.

Note: This setting applies to all point clouds inserted in the drawing.

Bubbles Size
Allows you to set the size of scan bubbles in Model space.

Adaptive Display ( )
Toggles the adaptive display of point sizes.

Note:

- Available for Windows and Linux, only for structured point clouds.
- Sets the value of POINTCLOUDADAPTIVEDISPLAY system variable.

When enabled, point sizes are adjusted according to local point density and distance from the camera to the position of the scanned points.

Dollhouse Rendering ( )
Toggles the display of points with normals in the view direction.
When enabled, points with normals pointing away from the camera are not shown.

Note: Only works when normals are available.

Point Size ( )
Sets the size of points displayed.

Eye Dome Lighting ( )
Allows you to set the non-photo-realistic shading to improve depth perception.

Gap Filling ( )
Fills gaps between points on surfaces.

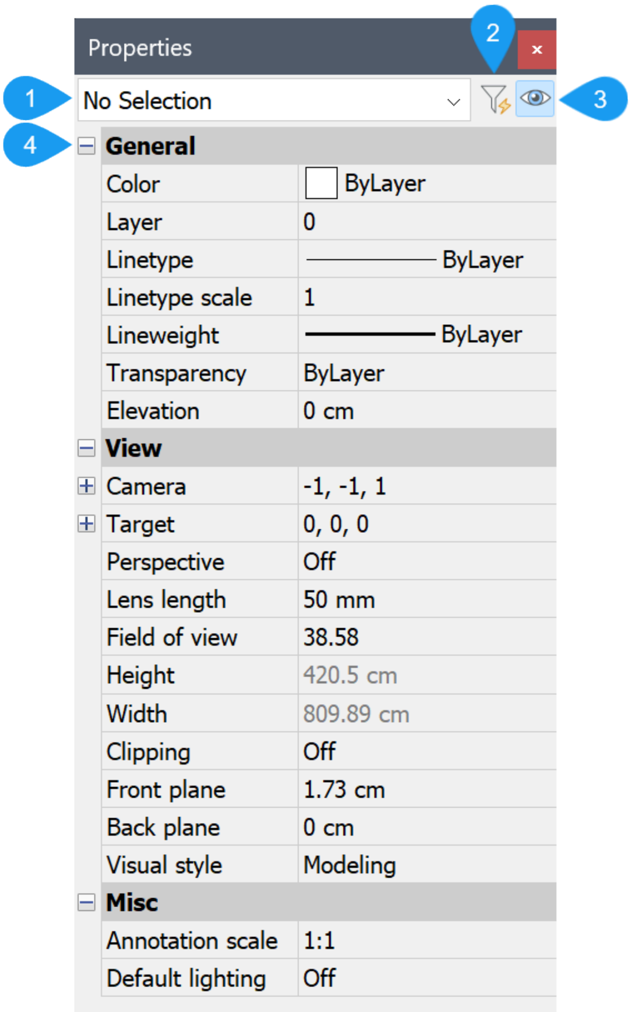
Note: Only works in perspective view, not in orthogonal view.

35.25 Properties panel

To open the **Properties** panel, run the PROPERTIES command or right-click a ribbon panel/toolbar and choose **Properties** from the **Panels** list.

The **Properties** panel offers a central location to access drawing, viewport, and entity properties.

It can be used in either **Properties** mode or **Quick Select** mode.



1 Entities list

2 Properties/Quick Select toggle

3 Property Preview toggle

4 Properties list

Entities list

Displays the type and number of selected entities.

Note: The PROPOBJLIMIT system variable defines the limit of entities displayed in the **Properties** panel to improve performance. If the number of selected entities exceeds the limit, a warning message appears, and you are allowed to choose one of the options: **Ignore limit** or **Change limit**.

Properties/Quick Select toggle

Switches between **Properties** and **Quick Select** mode.

Property Preview toggle

Turns the PROPERTYPREVIEW setting On or Off.

Properties list

Displays relevant property categories based on the current mode and selection set. You can expand and collapse categories by clicking on the **+** and **−** icons.

35.25.1 Properties mode

In **Properties** mode, the **Properties** panel enables you to view and modify drawing, viewport, and entity properties. Drawing and viewport properties are accessible when the current selection set is empty. Entity properties are accessible when the current selection set has at least one entity.

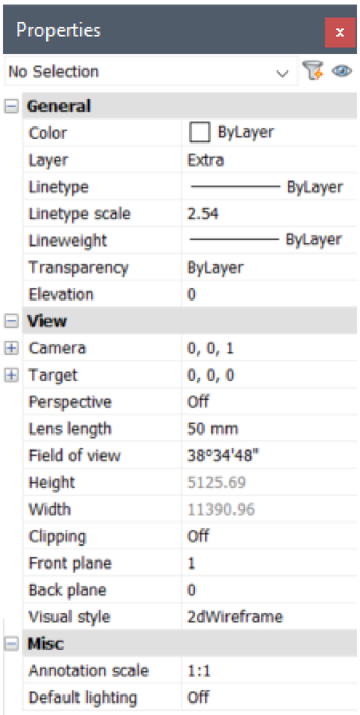
Entities list

When no entities are selected in **Properties** mode, the entities list shows No Selection and the properties list displays drawing and viewport properties.

When entities are selected in **Properties** mode, the entities list describes the type and number of entities in the selection set. You can filter the selection set by choosing a type of entity from the drop-down list. The types of entities in the selection set determine which properties are accessible.

Properties list (no selection set)

The drawing and viewport properties list, available when no entities are selected, displays **General**, **View**, and **Misc** properties.



General

In **Properties** mode, when no entities are selected, **General** properties describe the current entity settings. View and modify current entity settings to specify the properties that apply to new entities as they are created. Changing the current entity settings in the Properties list has the same effect as changing the relevant system variables.

Color

Describes the current color (CECOLOR system variable). You can specify a new color from the drop-down list or choose Select color to access additional colors from the **Color** dialog box.

Layer
Describes the current layer (CLAYER system variable). You can specify a new layer from the drop-down list.

Linetype
Describes the current linetype (CELTYPE system variable). You can specify a new linetype from the drop-down list or choose Load to access additional linetypes from the **Load Linetypes** dialog box.

Linetype scale
Describes the current linetype scale (CELTSCALE system variable). You can enter a new linetype scale.

Lineweight
Describes the current lineweight (CELWEIGHT system variable). You can specify a new lineweight from the drop-down list.

Transparency
Describes the current transparency level (CETRANSPARENCY system variable). You can specify a new transparency level from the drop-down list or enter a value.

Elevation
Describes the elevation relevant to the current UCS (ELEVATION system variable). You can enter a new elevation.

View
When no entities are selected, **View** properties describe the current view and relevant view settings.

Camera
Describes the absolute coordinates of the camera in the current UCS, based on the cartesian coordinate system (X,Y,Z). These values automatically update when you orbit or change the viewpoint in other ways.

Target
Describes the absolute coordinates of the camera target in the current UCS, based on the cartesian coordinate system (X,Y,Z). These values automatically update when you orbit or change the viewpoint in other ways.

Perspective
Describes the current perspective setting (PERSPECTIVE system variable). You can turn Perspective on or off from the drop-down list.

Lens length
Describes the current lens length (LENSLENGTH system variable). You can enter a new lens length.

Field of view
Describes the current angle for the field of view. You can enter a new angle for the field of view.

Height (read only)
Describes the height of the current view.

Width (read only)
Describes the width of the current view.

Clipping
Describes the clipping mode for the current view. You can specify a new clipping mode from the drop-down list. When clipping is turned on, parts of the view before the front clipping plane and/or behind the back clipping plane are hidden. This property is read only for layouts.

Front plane
Describes the location of the front clipping plane. This property is only available when the **Clipping** property is turned on. This property is read only for layouts.

Back plane
Describes the location of the back clipping plane. This property is only available when the **Clipping** property is turned on. This property is read only for layouts.

Visual style
Describes the current visual style of the drawing. You can specify a new visual style from the drop-down list. This property is read only for layouts.

Misc
When no entities are selected, **Misc** properties describe the current annotation scale and default lighting status.

Annotation scale
Describes the current annotation scale (CANNOSCALE system variable). You can specify a new annotation scale from the drop-down list.

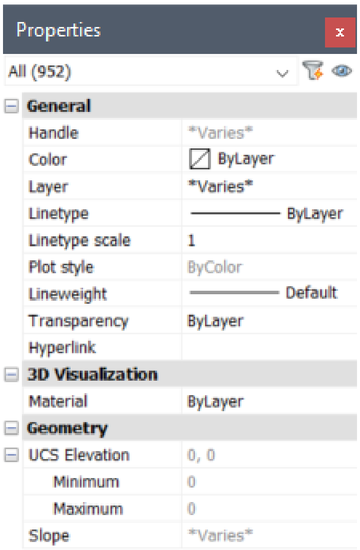
Default lighting
Describes the default lighting status (DEFAULTLIGHTING system variable). You can turn **Default lighting** On or Off from the drop-down list.

Properties list (current selection set)
The Properties list only displays properties that are valid for all the selected or filtered entities. The more types of entities you select, the fewer properties are available on the Properties list. Filtering the entity list enables you to view and modify properties for multiple entities of the same type with minimal effort. When two or more entities are selected, the properties show their common values. Properties that differ between the selected entities are shown as *varies*.

Note:

- When area calculations take longer than three seconds, temporary yellowish node labeled "Computing..." is inserted in **Properties** panel. Even if you interrupt the calculation by pressing ESC in the **Properties** panel, the entities remain highlighted and their properties are available for the user.
- The mass properties of 3D solids and surfaces are visible in the **Properties** panel when multiple entities are selected.

All entities display **General**, **3D Visualization** (Model space only), and **Geometry** properties. Additional properties may display for certain types of entities.



General Properties

In **Properties** mode, when entities are selected, **General** properties describe properties that are applied to the selected entities and are valid for almost any type of entity. Besides the properties that are available when no entity is selected, there are:

Handle (read only)

Describes the unique hexadecimal value by which the selected entity is identified in the DWG database.

Plot style

Describes the plot style of the selected entity. You can specify a new plot style from the drop-down list. Note that this property is only available when plot styles are enabled. Also see the PLOTSTYLE command

Hyperlink

Describes the URL of the selected entity. You can enter a new URL or choose the Browse button to access the **Edit Hyperlink** dialog box.

History

Enables you to undo editing operations for a selected entity. You can specify a previous step from the drop-down list to undo the entity back to one of the previous steps without undoing all the other commands and view operations that have been executed since. The **History** property only displays when the selection set includes only one entity.

Thickness

Describes the thickness of the selected entity. You can specify a new thickness by entering a value or selecting the picker icon and picking two points in the drawing.

3D Visualization Properties

Describes the materials assigned to the selected entities.

Material

Describes the current material of the selected entities. You can specify a new material from the drop-list.

Geometry Properties

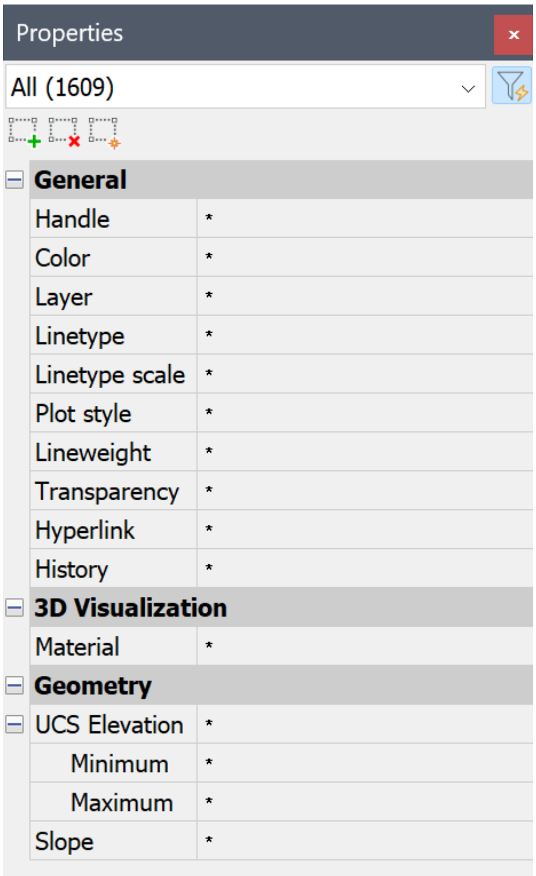
Describes the geometric characteristics of the selected entities. These properties can vary for different types of entities.

Note: When you click a numeric field, a button that opens the **Calculator** panel will appear on the right side.

Click the Apply button from the Calculator panel to paste the calculated value in the numeric field from the Properties panel.

35.25.2 Quick Select mode

In **Quick Select** mode, the **Properties** panel enables you to create a selection set based on entity properties. It includes selection tools and operators.



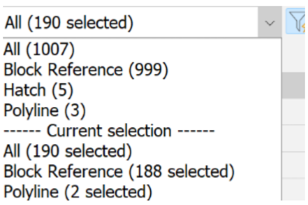
Selection tools

Creates a selection set based on entity properties.

Add to current selection set

Adds entities to the current selection set based on the entity types and property values you specify in the Entities and Properties lists.

Note: Click the drop-down button in the Entities list to see the current selection.



Remove from current selection set

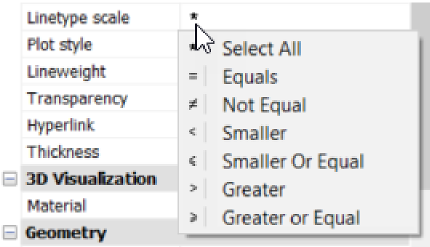
Removes entities from the current selection set based on the entity types and property values you specify in the Entities and Properties lists.

Clear current selection and create to new selection

Creates a new selection set based on the entity types and property values you specify in the Entities and Properties lists.

Operators

A list of operators enables you to filter properties based on values. You can click the current operator to display the list and change it to a different operator.



Select All

Does not filter the entity list based on the property.

Equals

Filters the entity list to include only those entities whose property equals the specified value.

Not Equal

Filters the entity list to include only those entities whose property does not equal the specified value.

Smaller

Filters the entity list to include only those entities whose property is less than the specified value.

Smaller Or Equal

Filters the entity list to include only those entities whose property is less than or equal to the specified value.

Greater

Filters the entity list to include only those entities whose property is greater than the specified value.

Greater or Equal

Filters the entity list to include only those entities whose property is greater than or equal to the specified value.

Properties list

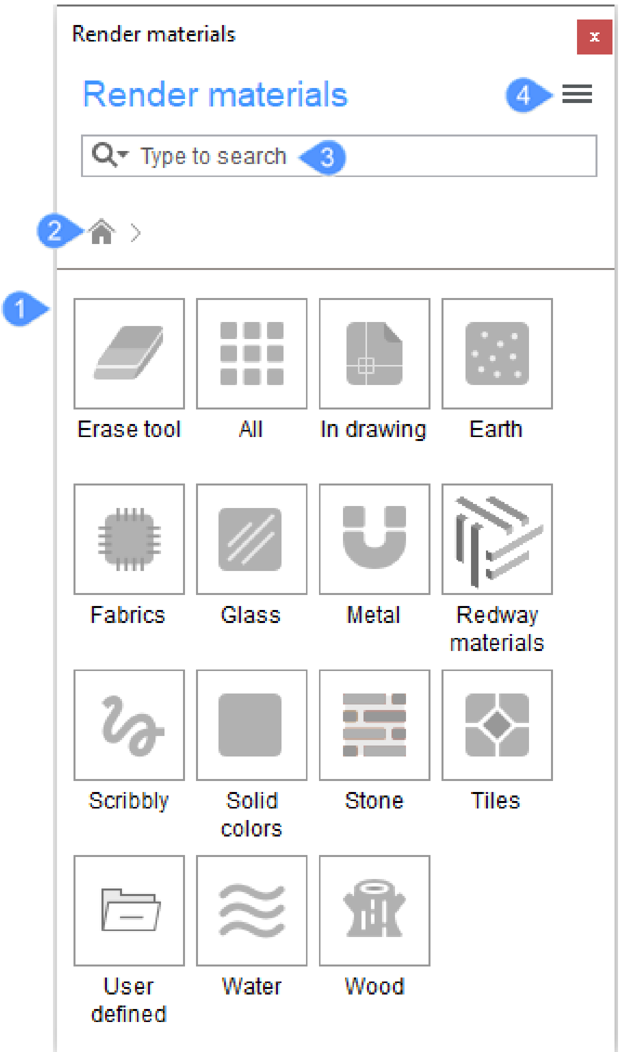
Displays entity properties on which you can filter. The available properties depend on the types of entities in the drawing and whether the option to **Apply** to current selection set is enabled. **General**, **3D Visualization** (Model space only), and **Geometry** properties are available for all entities. Additional properties may display for certain types of entities.

35.26 Render materials panel

To open the **Render materials** panel, run the MATBROWSEROPEN command or right-click a ribbon panel/toolbar and choose **Render materials** from the **Panels** list.

The **Render materials** panel offers access to materials used for renderings.

Material definitions are saved in the drawing. Each drawing contains the **Global** material. It is not possible to delete or rename the **Global** material. However you can edit its properties.



1 Material library

2 Home

3 Search

4 Menu

35.26.1 Material library

The material library is organized by categories. You can select a material from a category and then apply it to an entity using the brush that appears on the screen. You can scroll through the materials using the mouse wheel or the scroll bar.

Note: Right-click a material to add it to library, to remove it from the drawing or to add it to the drawing.

35.26.2 Home

Returns the material library to the home screen.

35.26.3 Search

Searches the library for words you enter in the search box.

35.26.4 Menu

Grid View
Displays the materials in a grid view.

List View
Displays the materials in a list view.

Manage libraries
Opens the **Settings** dialog box to change the render materials directory path.

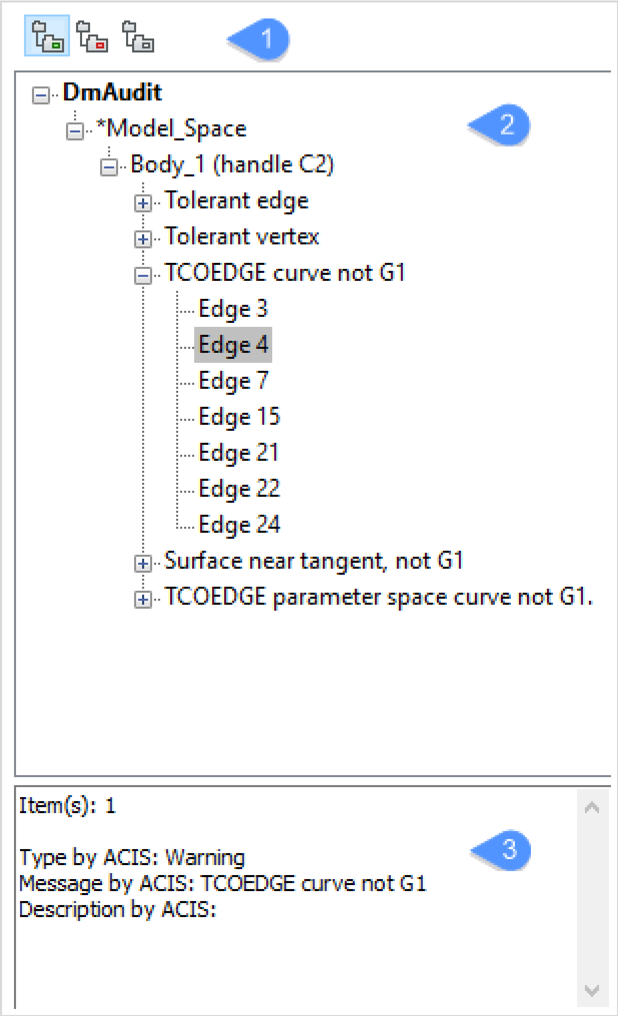
Get more materials
Opens the **Adding the Redway material library** documentation in a browser, where you can download the complete **Redway Material Library**.

35.27 Report panel

To open the **Report** panel, run the REPORTPANELOPEN command or right-click a ribbon panel/toolbar and choose **Report** from the **Panels** list.

The **Report** panel offers feedback for relevant commands including DMAUDIT, DMSIMPLIFY, DMSIMPLIFYALL, DMSTITCH, and all sheet metal (SM*) commands.

The appearance of the **Report** panel is determined by the REPORTPANELMODE system variable.



1 Display grouping

2 Report output

3 Detailed items output

35.27.1 Display grouping

Specifies how information is grouped for the report.

- Sort by Structure/Message/Subentity
- Group by Message/Structure/Subentity
- Group by Structure/Subentity/Message

35.27.2 Report output

Displays command output.

35.27.3 Detailed items output

Shows detailed command output for selected item in the report output.

35.28 Sequence Manager panel (Experimental)

35.28.1 Disclaimer

EXP.

Sequence Manager is an experimental feature that may not yet be stable and could be removed in the future. Use the MANAGEEXPERIMENTALFEATURES command to enable or disable experimental features.

The Experimental mode is disabled by default. Enabling or disabling the **Experimental mode** requires a restart of BricsCAD.

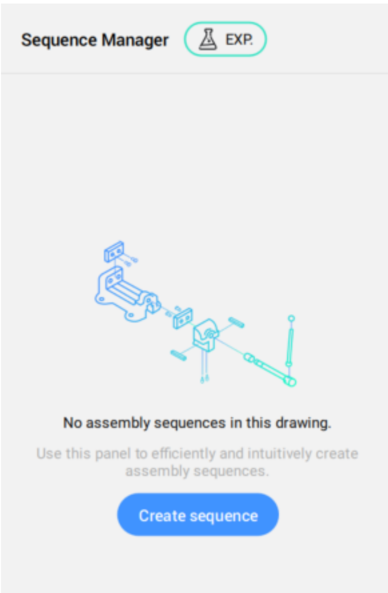
35.28.2 About

Launch he BMSEQUENCE command to open the **Sequence Manager** panel in the block-edit mode of the block in which the sequence is stored.

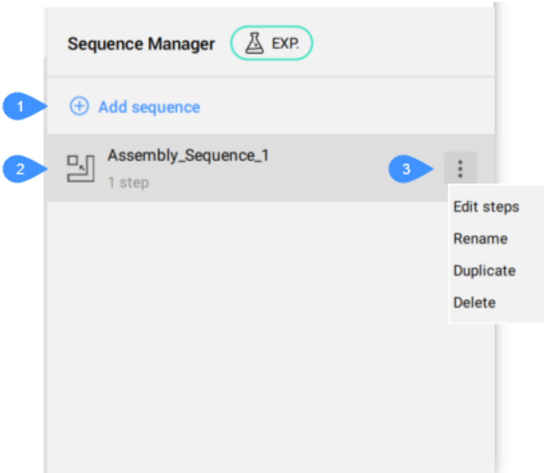
To open the **Sequence Manager** panel, run the BMSEQUENCEPANELOPEN. Run BMSEQUENCEPANELCLOSE command to close the panel.

The **Sequence Manager** panel allows you to manage the sequences and exploded views of a mechanical assembly.

If there are no assembly sequences in the drawing, there is only **Create sequence** option in the panel.



You can create the first sequence in the drawing by pressing the **Create sequence button**. The default sequence name is **Assembly_Sequence_1**.



1 Add sequence

2 Assembly_Sequence created

3 Options button

Add sequence
Adds a Assembly_Sequence in the list.

Assembly_Sequence created
Displays the created Assembly_Sequence.

Options button
Displays a list with editing options for the Assembly_Sequence.

Edit steps
Opens the **Sequence Manager** panel in the block-edit mode of the block in which the Assembly_Sequence is stored.

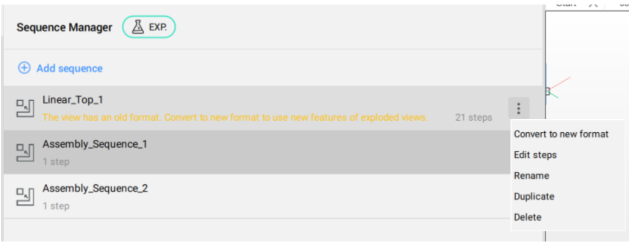
Note: Also, double click an Assembly_Sequence opens the **Sequence Manager** panel in the block-edit mode.

Rename
Displays the **Rename sequence** dialog box to introduce the new name for the Assembly_Sequence.

Duplicate
Creates a duplicate for the Assembly_Sequence.

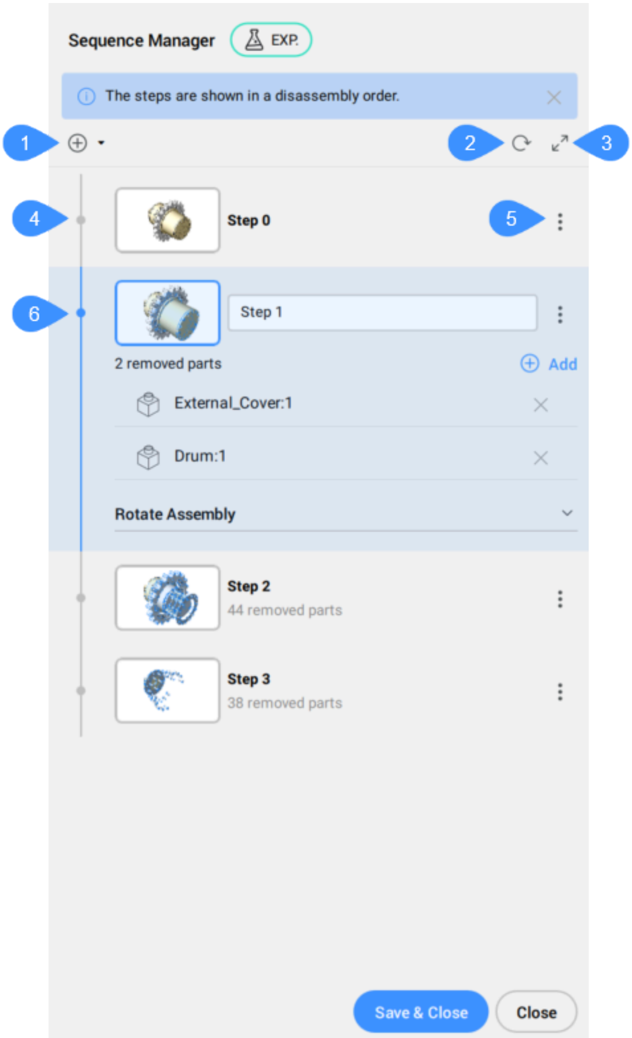
Delete
Deletes the Assembly_Sequence.

Exploded views in the old format can be converted to the new format by choosing the **Convert to new format** option from the **Options button**.



When open the **Sequence Manager** panel in the block-edit mode of the block in which the sequence is stored, you are able to manage the sequence steps.

The **Sequence Manager** panel in this mode shows a list of steps in the sequence in disassembly order, with a thumbnail for each step and a list of all parts in that step.



1 Add new step

2 Update thumbnails

3 Switch to large thumbnails

4 List of steps

5 Options button

6 Current/active step

Add new step
Adds a new step after the last step.
The drop down list contains several options for adding the step:

After last step
Adds a step after the last step.

After current step
Adds a step after the current step.

Before current step
Adds a step before the current step.

Before first step
Adds a step before the first step.

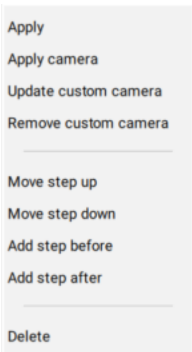
Update thumbnails
Updates the thumbnails after editing step(s).

Switch to large thumbnails
Enables to display large thumbnails.

List of steps
Displays the steps of the sequence.

Note: Set the step as the current by clicking on its thumbnail.

Options
Lists the options applicable to the step.



Apply
Sets the step as current.

Apply camera
Applies the camera position for the step.

Update custom camera
Updates the custom camera position.

Remove custom camera
Removes the custom camera position.

Move step up
Moves the step up.

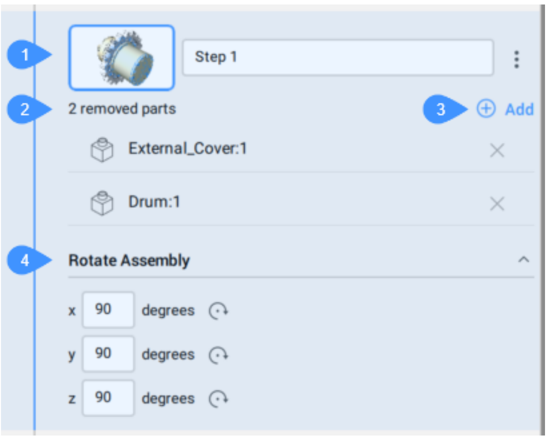
Move step down
Moves the step down.

Add step before
Adds a step before this one.

Add step after
Adds a step after this one.

Delete
Deletes the step.

Current/active step
Allows you to edit the current step.



1 Thumbnail

2 Removed parts list

3 Add removed part

4 Rotate Assembly

Thumbnail
Displays the thumbnail of the step.

Note: By clicking on the thumbnail you will launch the **Apply camera** option, to apply the custom camera.

Removed parts list
Lists the entities (mechanical components) that were disassembled in this step.

Note: There are **Show more** and **Show less** buttons for large lists to manage the display of components.

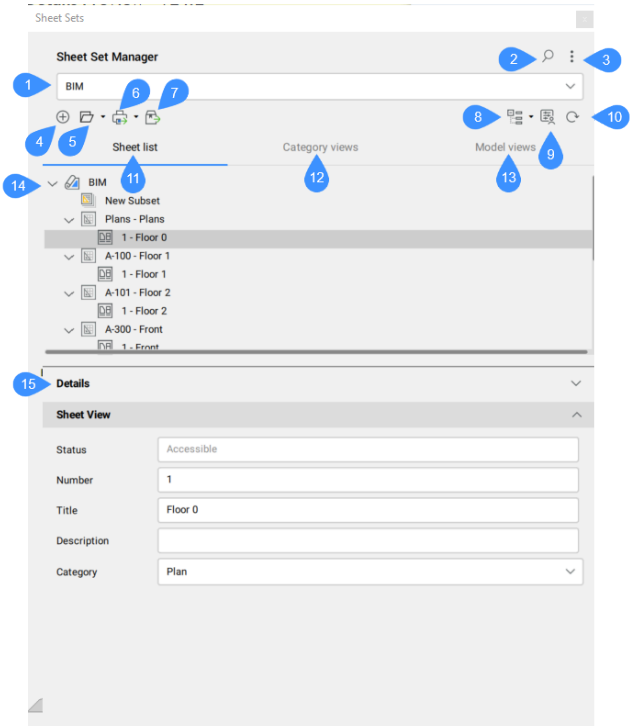
Add removed part
Adds one or more mechanical components to this step.

Rotate Assembly
Rotates the assembly around the X, Y, or Z axes. You can click on the thumbnail to keep the camera position (**Apply camera**).

35.29 Sheet Sets panel

To open the **Sheet Sets** panel, run the SHEETSET command or right-click a ribbon panel/toolbar and choose **Sheet Sets** from the **Panels** list.

The **Sheet Sets** panel offers a central location to create and to manage sheet sets.



1 Current sheet set name
2 Search
3 Menu
4 Create sheet set
5 Open sheet set
6 Publish
7 eTransmit
8 Sheet selection sets
9 Custom properties
10 Refresh
11 Sheet list
12 Category views
13 Model views
14 Sheet Sets list
15 Details

35.29.1 Current sheet set name

Displays the name of the current sheet set. From the drop-down list you can change the current sheet set.

35.29.2 Search

Opens a search field that allows you to search for specific sheet set, subset or sheet names.

35.29.3 Menu

Opens a menu with a number of options depending on the sheet set panel state:

Import from XML
Allows you to import an XML file that was created in BricsCAD with **Export to XML...**. Opens the **Import Sheet Set** dialog box.

Export to XML
Exports a sheet set in XML format. Opens the **Export Sheet Set As** dialog box.

Sheet Set Options
Opens the **Sheet Controls** options in the **Settings** dialog box.

Custom Properties
Allows you to add custom properties for sheet sets and sheets. Opens the **Custom Properties** dialog box.

35.29.4 Create sheet set

Opens the **Create New Sheet Set** dialog box.

35.29.5 Open sheet set

Opens the **Select a Sheet Set file** dialog box to select the DST file to be loaded. Click the right-side arrow to open a drop-down menu which contains a list with the recently opened DST files. Click one of these files to load the sheet set into the panel.

Open
Opens the selected DST file from the **Select a Sheet Set file** dialog box.

35.29.6 Publish

Publishes the currently selected sheet set(s), subset(s) or sheet(s) using the **Publish** dialog box. Click the right-side arrow to open a drop-down menu:

Publish
Publishes the currently selected sheet set(s), subset(s) or sheet(s) using the **Publish** dialog box.

Publish to PDF
Publishes the currently selected sheet set(s), subset(s) or sheet(s) to PDF using the last used presets.

Publish to Plotter
Publishes the currently selected sheet set(s), subset(s) or sheet(s) using a plotter based on the last used presets.

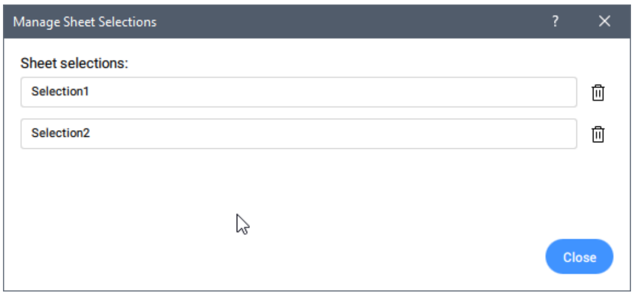
35.29.7 eTransmit

Launches the ETRANSMIT command. The currently selected sheet set(s), subset(s) or sheet(s) are included in the eTransmit procedure. Opens the **eTransmit** dialog box.

35.29.8 Sheet selection sets

Opens the **Manage Sheet Selections** dialog box. Click the right-side arrow to open a drop-down menu:

Manage...
Opens the **Manage Sheet Selections** dialog box where you can rename or delete Sheet selections.



Create
Saves the Sheet selection with a specified name.

List of existing sheet selection sets
Lists the existing sheet selection sets of the active (selected) sheet set in the panel. Click one of these items to activate the selection set.

35.29.9 Custom properties

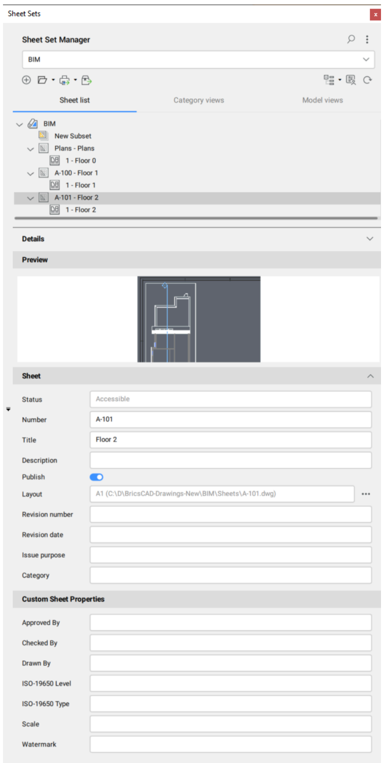
Allows you to add custom properties for sheet sets and sheets. Opens the **Custom Properties** dialog box.

35.29.10 Refresh

Updates all information stored in the sheet set data (DST) file, checks all folders in every resource location for new or removed drawing files, and all expanded drawing files for new or removed model space views.

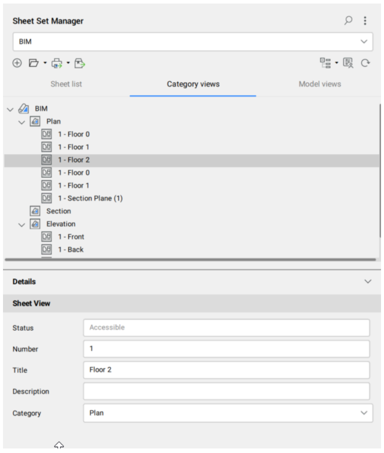
35.29.11 Sheet list

Displays the Sheet Sets in the **Sheet list** mode.

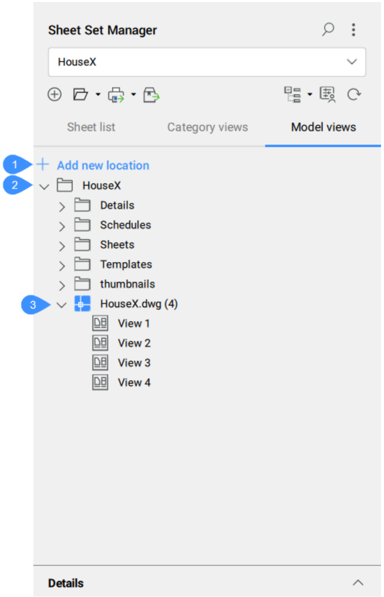


35.29.12 Category view

Displays the Sheet Sets grouped by category.



35.29.13 Model views



1 Add new location

2 Folders

3 Model views

Add new location

Opens the **Browse for Folder** dialog box to choose a folder to be added. Only unique folders can be added.

Folders

Lists all added folders that contain the drawings and their model views. The **Model Views** functionality allows you to browse and select model views (created with the VIEW command) from external drawings and insert them into the Paper Space layout of the current drawing, without opening the original file.

Note: The context menu opened by right-clicking a folder offers the option to **Remove** it from the **Model views** tab.

Model views

Model views (created with the VIEW command) from external drawings can be dragged and dropped into a Paper Space layout of the current drawing.

Note: Double-click a drawing/drawing Model view in the **Model views** tab to open it in the Model Space. Access the **Open**/**Open Read-Only** options from the right-click context menus.

35.29.14 Sheet Sets list

Displays the current sheet set.

When you change the project name in the **BIM Project Browser** panel, the Sheet Set name is updated correspondingly.

A Sheet Set can contain Sheets, Subsets, and Sheet Views.

Right-clickmenusoffer additional tools:

Sheet Set

Close Sheet Set
Removes the Sheet set from the panel.

Export as XML
Opens the **Export Sheet Set As** dialog box.

Resave All Sheets
Saves all sheets.

New Sheet(s)
Displays the **New Sheet(s)** dialog box, which allows you to add new sheets to the current sheet set.

New Subset
Adds a new subset to the current sheet set.

Selection Sets
Opens the **Manage Sheet Selections** dialog box.

Custom Properties
Opens the **Custom Properties** dialog box.

Renumber
Opens the **Renumber Sheet Set** dialog box, which allows you to renumber all sheets in the sheet set.

Plot
Plots the currently selected sheet set(s), subset(s) or sheet(s).

Publish
Opens the **Publish** dialog box.

Export Package (eTransmit)
Opens the **eTransmit** dialog box.

Insert Sheet List Table
Inserts the sheet list table into the current drawing.

Sheet

Open
Opens the drawing with the layout of the selected sheet.

Open read-only
Opens the drawing with the layout of the selected sheet in read-only mode to protect it.

Remove Sheet
Removes the selected sheet.

Insert Sheet List Table
Inserts a Sheet List Table in the drawing area.

Subset

Rename Subset
Allows you to rename the selected subset.

Remove Subset
Removes the selected subset.

Sheet View

Display View
Opens the drawing file related to the selected Sheet View.

Place Callout Block > Number Bubble
Places a callout block of the corresponding Sheet View.

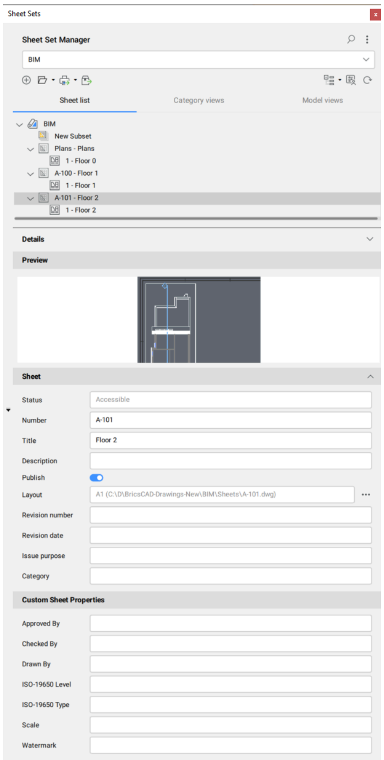
Place View Label Block
Places a view label block of the corresponding Sheet View.

Note:

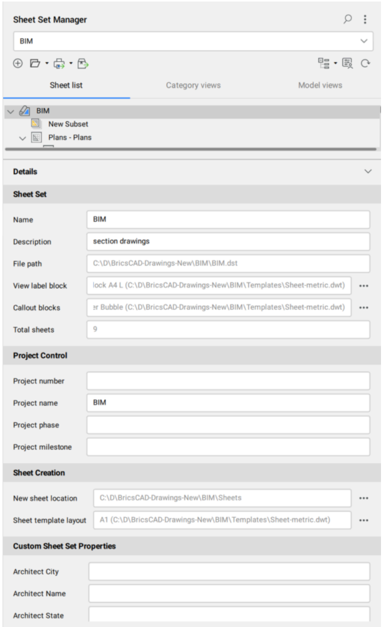
- Double-click a sheet set or a subset to expand/collapse. Double-click a Sheet or a Sheet View to open it.
- Sheet set tree nodes can be reordered using drag and drop.
- The properties and variables of a selected Sheet Set, Subset, Sheet or Sheet View are displayed in the **Details** section.

35.29.15 Details

Displays the details of the selected Sheet Set, Subset, Sheet, or Sheet View. A preview of the selected sheets or subset can be shown or hidden.



In the **Details** > **Sheet Set**, the **File path**, **View label block**, **Callout blocks** and **Total sheets** options are read only, but selectable.

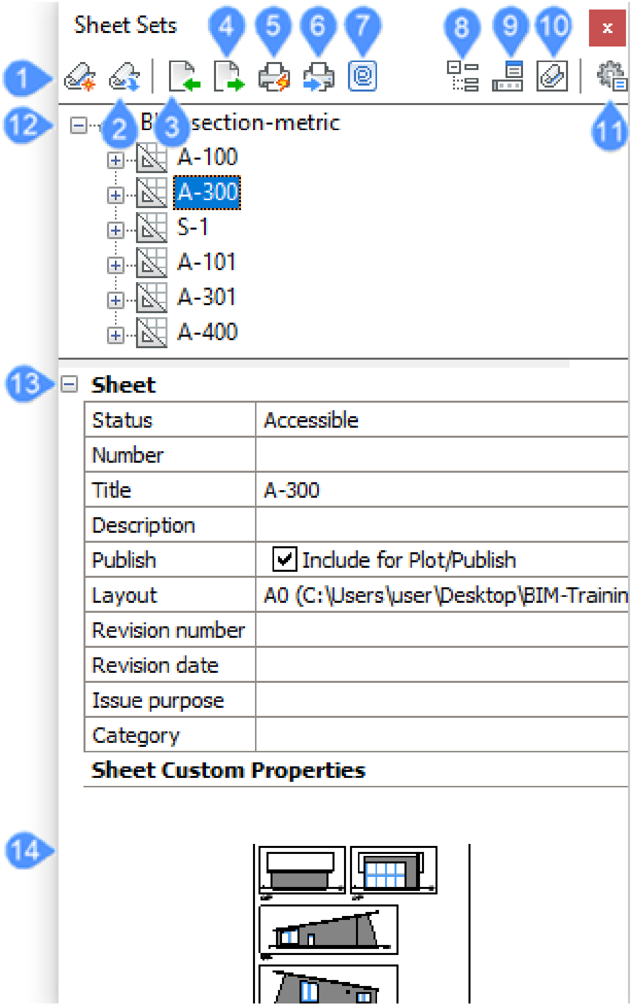


Note:

- Click the **Browse** button (...) next to **View label block** option to open the **Select Block** dialog box.
- Click the **Browse** button (...) next to **Callout blocks** option to open the **Blocks List** dialog box.

35.30 Sheet Sets (legacy) panel

The **Sheet Sets (legacy)** panel offers a central location to create and manage sheet sets.



1 New sheet set...

2 Open sheet set...

3 Import from XML...

4 Export to XML...

5 Plot

6 Publish...

7 eTransmit...

8 Sheet selections sets...

9 Custom properties...

10 View categories

11 Sheet set options...

12 Sheets

13 Sheet properties

14 Sheet preview

35.30.1 New sheet set...

Opens the **Create New Sheet Set** dialog box.

35.30.2 Open sheet set...

Opens the **Select a Sheet Set file** dialog box.

35.30.3 Import from XML...

Allows you to import an XML file that was created in BricsCAD with **Export to XML...**. Opens the **Import Sheet Set** dialog box.

35.30.4 Export to XML...

Exports a sheet set in XML format. Opens the **Export Sheet Set As** dialog box.

35.30.5 Plot

Plots the currently selected sheet set(s), subset(s) or sheet(s).

35.30.6 Publish...

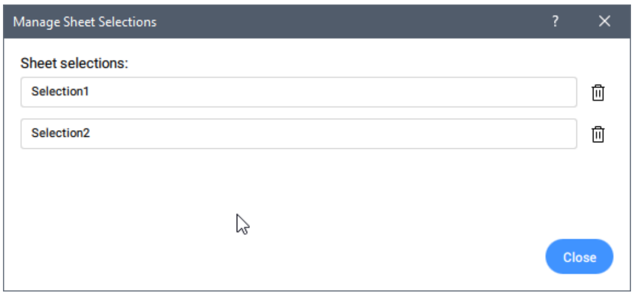
Publishes the currently selected sheet set(s), subset(s) or sheet(s). Opens the **Publish** dialog box.

35.30.7 eTransmit...

Opens the **eTransmit** dialog box. The currently selected sheet set(s), subset(s) or sheet(s) are included in the eTransmit procedure.

35.30.8 Sheet selections sets...

Opens the **Manage Sheet Selections** dialog box where you can rename or delete Sheet selections.



35.30.9 Custom properties...

Allows you to add custom properties for sheet sets and sheets. Opens the **Custom Properties** dialog box.

35.30.10 View categories

Displays the sheet sets as Sheet list, or Category view.

35.30.11 Sheet set options...

Opens the **Sheet sets** options in the **Settings** dialog box.

35.30.12 Sheets

Displays the opened and created sheet sets.

Right-clickmenusoffer additional tools:

Renumber...
Opens the **Renumber Sheet Set** dialog box, which allows you to renumber all sheets in the sheet set.

New Sheet(s)...
Displays the **New Sheet(s)** dialog box, which allows to add new sheets to the current sheet set.

New Subset
Adds a new subset to the current sheet set.

Insert Sheet List Table
Inserts the sheet list table into the current drawing.

Open
Opens the drawing with the layout of the selected sheet.

Open read-only
Opens the drawing with the layout of the selected sheet in read-only mode to protect it.

Note: You can double-click a sheet set or a subset to expand/collapse. Double-click a sheet to open it.

35.30.13 Sheet properties

Displays the properties and variables of the selected sheet set, subset or sheet.

35.30.14 Sheet preview

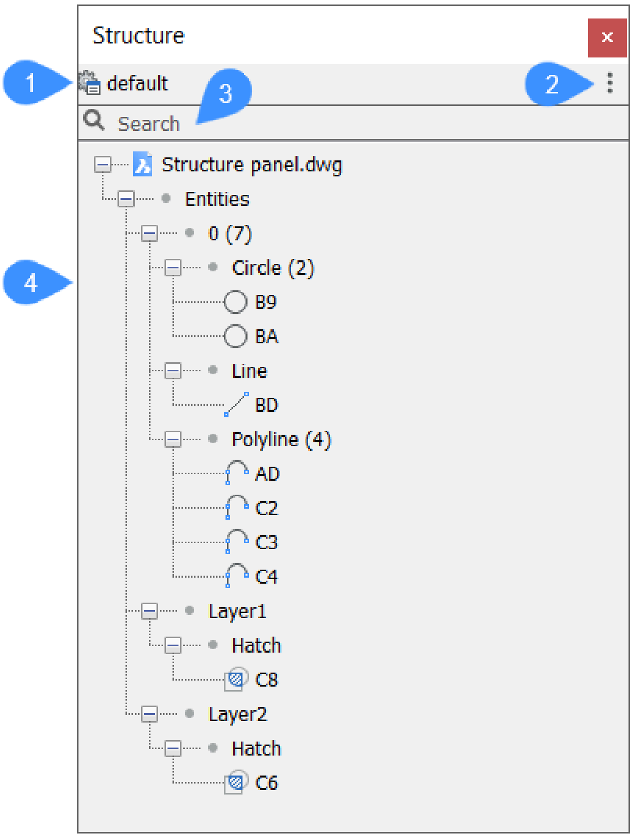
Displays a thumbnail of the currently selected sheet set without opening it.

35.31 Structure panel

To open the **Structure** panel, run the STRUCTUREPANEL command or right-click a ribbon panel/toolbar and choose **Structure** from the **Panels** list.

The **Structure** panel enables you to:

- View the entire content of a drawing listed as a structure tree configured by predefined rules.
- Decide how the entities are displayed in the structure tree by selecting a **Structure Tree Configuration** or creating a new one via the **Configure Structure Tree** dialog box.
- Select specific entities from the structure tree to identify them in the model space (and vice versa) and to modify their properties.
- ***Note:*** BricsCAD does not support modifying sub-entities of blocks using the **Structure** panel.



1 Structure tree configuration

2 Configurations menu

3 Search bar

4 Structure tree

35.31.1 Structure tree configuration

Displays the name of the structure tree's active configuration. To edit the active configuration or create a new one, open the **Configure Structure Tree** dialog box by clicking on the configuration's name. Here you can create rules by selecting entity properties as filter, group and sorting criteria (see **Configure Structure Tree dialog box** article).

35.31.2 Configurations menu

This drop-down menu lists the configuration (CST) files stored in the **Support Folder**. Select one to set it as the active Structure Tree Configuration.

default
Displays entities sorted and grouped by layer-based structure.

mechanical
Displays entities sorted and grouped by based structure.

bim
Displays BIM entities sorted and grouped by based structure.

bim - Composition
Displays BIM entities sorted and grouped by composition. For example, you can see/select all concrete walls on all floors at once, or see at a glance which compositions are used.

bim - Profiles
Displays BIM entities sorted and grouped by profile.

bim - Type
Displays BIM entities sorted and grouped by BIM-type. For example, you can see/select all walls on all floors at once.

Configure structure tree
Opens the **Configure Structure Tree** dialog box to edit the existing configurations or to add your own structure tree configurations (see **Configure Structure Tree dialog box** section below).

Select Configuration...
Allows you to load **Structure Tree Configuration** files (*.cst).

Note: The default path were predefined CST files are stored is: *C:\Program Files\Bricsys\V25x64\UserDataCache\Support\en_US*.

35.31.3 Search field

Type in a string to search the structure tree for character strings that match what you typed. The structure tree adapts accordingly, displaying only the result entities.

Note: The search field allows the use of case-sensitive query language. For example, type '*$EntityType=Line and (Length>400 and Length<1000)*' to find lines with lengths between 400 and 1000 units.

35.31.4 Structure tree

Displays a structured tree view of the drawing content. The configuration of the structure tree is defined in a CST file and specified by the STRUCTURETREECONFIG system variable. The location of CST files is specified by the SRCHPATH system variable. You can create custom CST files to meet your needs.

General usage information

Click the drop-down menu (⋮) to select a definition.
In the **Structure** panel you can:

- Click the expand (**+**) icon to expand a node.
- Click the collapse icon (**-**) to collapse a node.
- Click a node or entity in the **Structure** panel to select it in the drawing.

  Note: This depends on the settings in the **Structure tree**. To configure the Structure tree, right-click in **Structure panel** > **Configure** > **Options**.

  Hold down the **Ctrl** key to compose a selection set.

Note: Select the drawing name. Next, right-click and choose **Show** in the context menu to display the entire drawing, or choose **Show Entities** in the **General** tab of the No Selection Quad.

A right-click menu on a tree node or on an entity offers relevant tools:

Show
Shows the selection set, when it was previously hidden.

Hide
Hides the selection set, and keeps all other drawing entities visible.

Isolate
Hides all other entities in the drawing, except the selection set.

Zoom
Zooms the view to the extents of the selected entities.

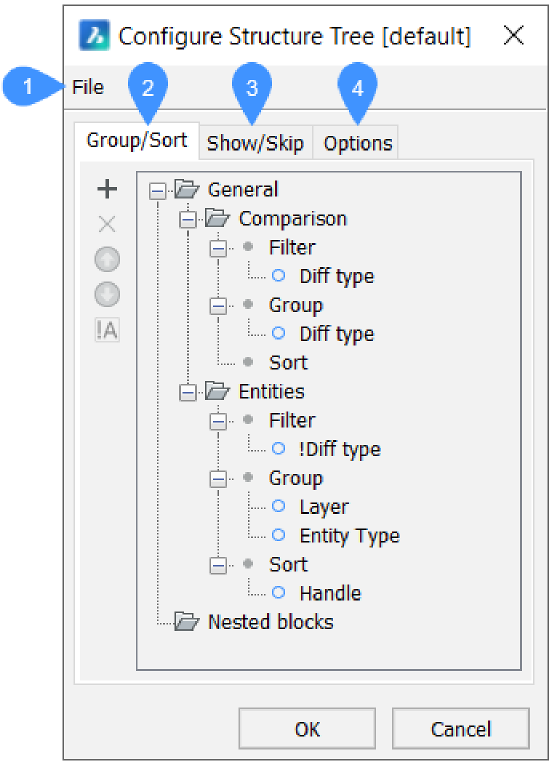
Expand All
Expands all nodes in the structure tree.
Select the drawing name and use this option to expand all nodes in the entire tree.

Collapse All
Collapses all nodes in the structure tree.
Select the drawing name and use this option to collapse the entire tree.

Configure...
Opens the **Configure Structure Tree** dialog box to edit the existing configurations or to add your own structure tree configurations.

35.31.5 Configure Structure Tree dialog box

The **Configure Structure Tree** dialog box allows you to define the **Structure Tree Configuration** of the **Structure** panel.



1 File

2 Group/Sort tab

3 Show/Skip tab

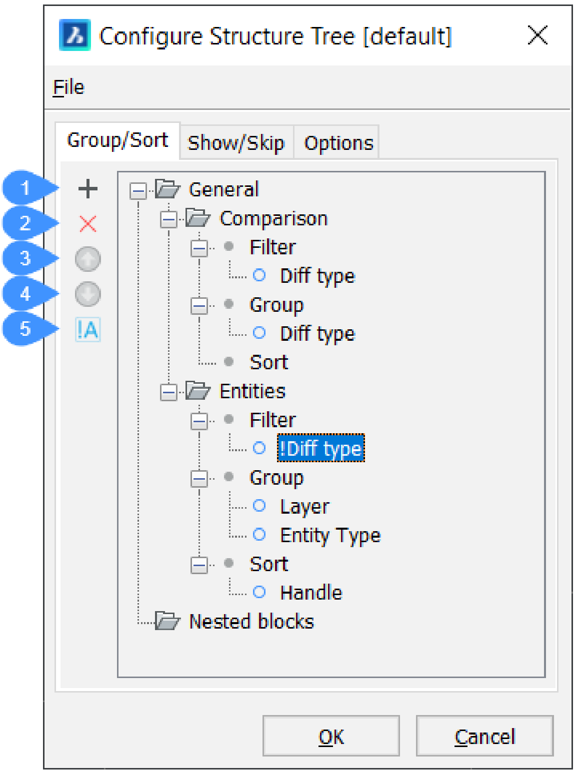
4 Options tab

35.31.6 File

Click the **File** menu in the dialog box and choose one of the available options in the list:

- **Open...**: opens the **Select Structure Tree Configuration File** dialog box to select another CST configuration file.
- **Save**: saves the current configuration.
- **Save As...**: saves the current configuration under a different name.

35.31.7 Group/Sort Tab



1 Add rule or property

2 Delete rule or property

3 Move property up

4 Move property down

5 Toggle negation (filter properties only)

Add rule or property (+)

Adds a new rule or a new property for **Filter**, **Group**, or **Sort**.

Note: The position of the rules and properties is essential, as the rule structure is hierarchical.

- Select an existing rule, then click the **Add** icon (+) or right-click and choose **Add Rule** in the context menu.

  The new rule is added below the selected rule (or at the end of the rule hierarchy when no rule is selected).

  Note: To rename the rule, select it and click its name or right-click and choose **Rename** in the context menu.

- Select the **Filter** node of the rule, then click the **Add** icon or right-click and choose **Add filter property** in the context menu.
- Select the **Group** node of the rule, then click the **Add** icon or right-click and choose **Add grouping property** in the context menu.
- Select the **Sort** node of the rule, then click the **Add** icon or right-click and choose **Add sorting property** in the context menu.

The **Select Property** dialog box displays to let you choose a property for filtering, grouping, or sorting.

- Select a property, then double-click or click **OK**.

  Note: Properties and value definitions that are defined with the BIMPROPERTIES command can be selected to configure the structure tree.

- Optionally, type a search string in the **Search** field, then click the **Find previous** (**Shift+F3**) ( ) or **Find next** (**F3**) ( ) arrow buttons to look up the properties containing the search string.
- Repeat the previous step to add more filter/ grouping/ sorting properties.

Delete rule or property ( )

Deletes the selected rule or property. You can also use the **Delete rule**/**Delete property** option from the context menu of the selected rule/property.

Move property up ( )

Moves up the selected rule or property.

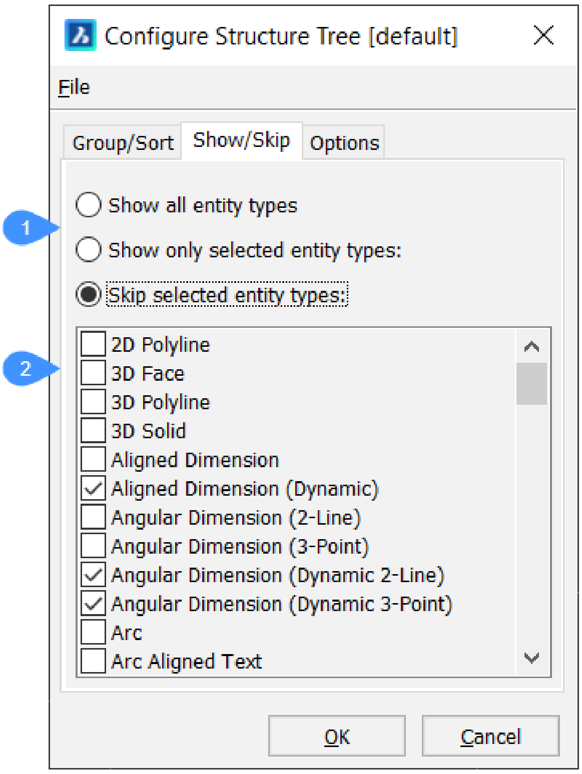
Move property down ( )

Moves down the selected rule or property.

Toggle negation (filter properties only)

Toggles negation only for filter properties.

35.31.8 Show/Skip Tab



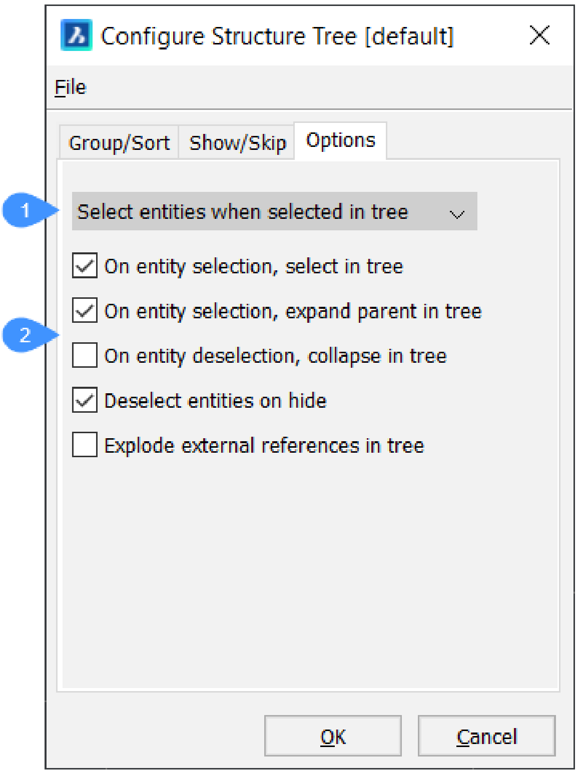
1 Show/skip options

2 Selection list

Show/skip options
Specifies whether selected entities should be shown or not.

Selection list
Allows you to create a selection of entity types to show or exclude.

35.31.9 Options Tab



1 Selection options

2 Additional options

Selection options

Defines what happens in the model when entities are selected in the structure tree. Several options are available:

- **Ignore tree selection**: entities are not selected in the drawing.
- **Highlight entities when selected in tree**: entities in the drawing are highlighted, but not selected.
- **Select entities when selected in tree**: matching entities are selected in the drawing.

Additional options

Defines what happens in the structure tree when an entity is selected/deselected in the model.

On entity selection, select in tree

When on, selects matching entity(ies) in the structure tree when you select them in the drawing.

On entity selection, expand parent in tree

When on, expands the node related to the entity(ies) selected in the drawing.

On entity deselection, collapse in tree

When on, collapses the node related to the entity(ies) selected in the drawing.

Deselect entities on hide

When on, deselects hidden entities.

Explode external references in tree

When on, lists the contents of externally-referenced drawings in the structure tree.

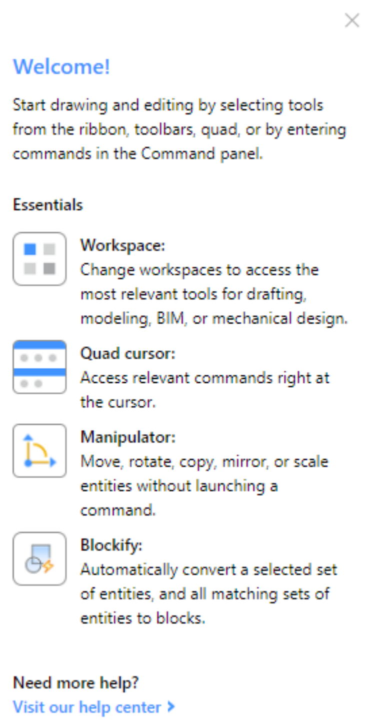
See also the **Working with the Structure panel** guide linked in the **Related information** section which provides a step by step example on how to use the **Structure** panel.

35.32 Tips panel

To open the **Tips** panel, right-click a ribbon panel/toolbar and choose **Tips** from the **Panels** list.

The **Tips** panel offers relevant advice based on the context in which you are working.

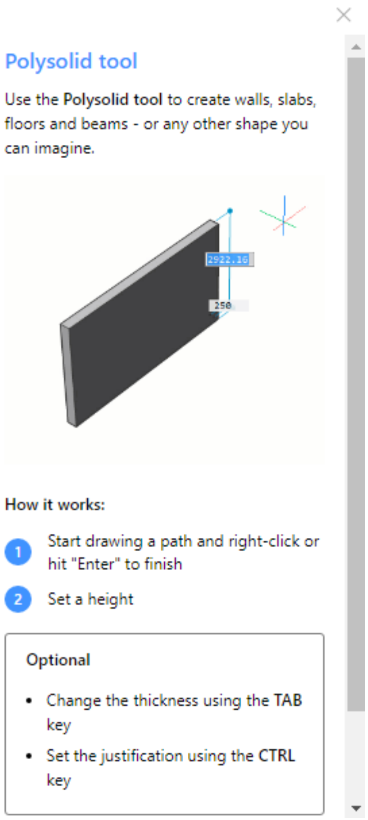
Note: The **Tips** panel content is hosted online. You have to check your internet connection.



Visit our help center
Opens the [Bricsys Help Center website page](https://help.bricsys.com).

Note: The panel will be closed by pressing the cross button on the upper right side.

When you start certain commands, the panel displays a short animation and additional information about the command.

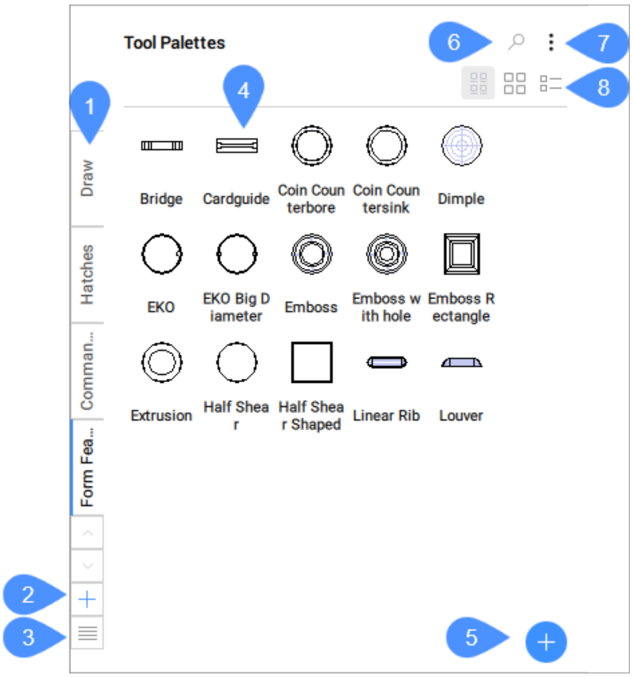


35.33 Tool Palettes panel

To open the **Tool Palettes** panel, run the TOOLPALETTES command or right-click a ribbon panel/toolbar and choose **Tool Palettes** from the **Panels** list.

The **Tool Palettes** panel offers a central location to access blocks, hatches, and command tools.

It includes default tabs with sample tools. You can easily create and customize your own tabs and tools using the right-click menu.



1 Palettes Tabs

2 Add tool palette button

3 Tool Palettes list button

4 Tools

5 Add button

6 Search

7 Tool palettes context menu

8 View modes

35.33.1 Palettes Tabs

Displays a list of available palettes.

Note: The current palette is highlighted with a blue line.

Note: The tab items have a fixed height, so not always the name is legible. The entire name of the palette will be displayed when hovering over the tab. A palette can be renamed by double clicking on it, or from the context menu which is displayed by right clicking the panel tab.

Note: Using the up and down arrow button the user can see the hidden tabs.

- Tab labels should be "Elided"

Note: The Tab bar changes his position depending on the docking of the **Tool Palettes** panel.

35.33.2 Add tool palette button

Adds a new tool palette.

35.33.3 Tool Palettes list button

Lists all available tool palettes. User may choose the current palette.

35.33.4 Tools

Displays a list of available tools in the current palette.

Right-click menus offer additional functionality for tools:

Note: Rectangle selection is available to select multiple tools.

Cut
Removes the selected tools from the palette.

Copy
Copies the selected tools to the clipboard.

Delete
Removes the selected tools from the palette.

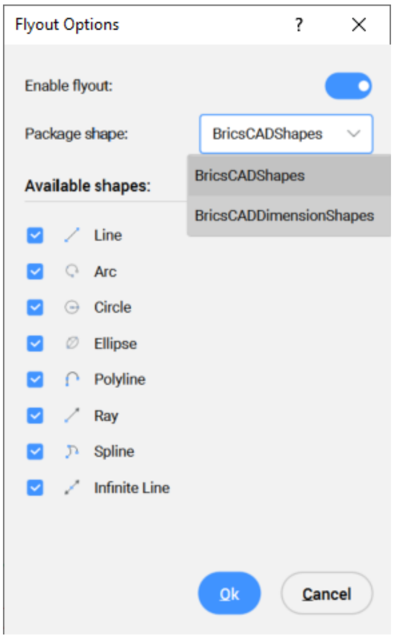
Rename
Renames a tool.

Specify Image…
Allows you to specify another image for the tool.

Note: To restore the original tool image, right-click the tool and choose **Remove Image...** in the context menu.

Manage Flyout options...
Opens the **Flyout Options** dialog box which allows you to enable tool flyouts.

Note: Tool flyouts via XTP import are supported.



The predefined flyouts Package Shape are available in: C:\Users\%username%\AppData\Roaming\Bricsys\BricsCAD\V25x64\en_US\Support\shapes.btc file.

Properties…
Opens the **Tool Properties** dialog box, which allows you to edit a tool (see **Tool Properties dialog box** article).

35.33.5 Add button

Allows you to add new **tools**, **texts** and **separators**.

Note:

- Texts and separators can be cut, copy and pasted from the context menu.
- Texts and separators can be drag & dropped.

35.33.6 Search

Allows you to search for specific tools in the current palette. The list is shortened to display only the tools that contain the entered sequence of characters, regardless of where they are located in the tool's name.

35.33.7 Tool Palettes context menu

Opens the tool palettes context menu.

View Options…
Opens the **View options** dialog box, which allows you to modify the display of the tool icons (see **View options dialog box** article).

Add text
Adds a text into the current palette.

Add Separator
Adds a separator into the current palette.

Update Palette
Updates the palettes.

New Palette
Allows you to add a new palette.

Delete Palette
Allows you to delete the current palette.

Rename Palette
Allows you to rename the current palette.

Customize Palettes…
Opens the **Customize** (Tool Palettes) dialog box, which allows you to customize the palettes. You can import and export palettes or create, import and export palettes groups.

Add Tool...
Opens the **Customize** (Tool Palettes) dialog box, which allows you to add a new tool (see **Customize (Tool Palettes) dialog box** article).

35.33.8 View modes

Icon with text
Displays tools in a grid view in icon and text mode.

Icon only
Displays tools in a grid view in icon mode only.

List view
Displays the tools in a list view in the current palette.

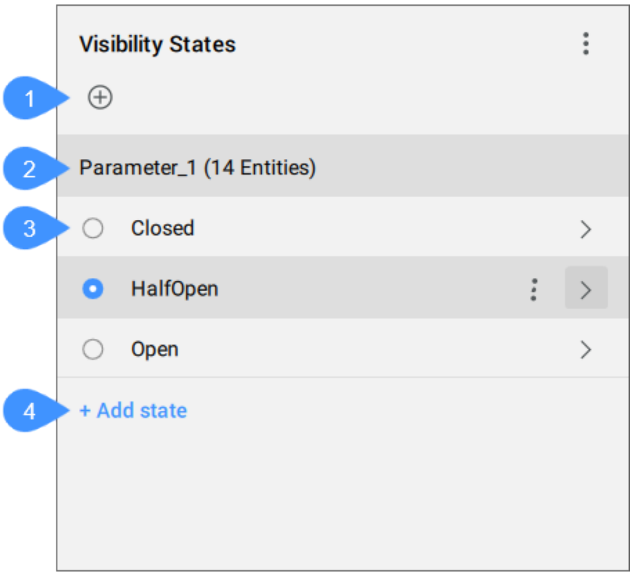
35.34 Visibility States panel

To open the **Visibility States** panel, run the VISIBILITYSTATESPANELOPEN command or right-click a ribbon panel/toolbar and choose **Visibility States** from the **Panels** list.

Visibility states parameters allow you to create blocks where just selecting one or more parameters change the appearance of the block by showing or hiding various entities inside the block. Each visibility states parameter can have one or more states.

Note: There is no need to be in the **Block editor** to use Visibility States.

The **Visibility States** panel displays the visibility parameters found in the drawing and their corresponding visibility states.



1 Add parameter

2 Parameter's name

3 List of parameter's visibility states

4 Add state

Add parameter
Adds a new parameter to the block.

Note: The new created parameters appear in the **Parameters** section in the **Properties** panel.

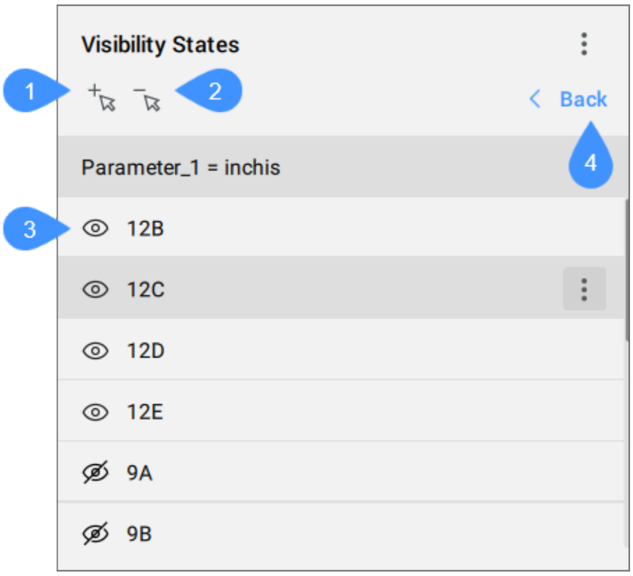
Parameter's name
Double-click to edit the parameter's name.

List of parameter visibility states
Lists all the parameter's visibility states. Double-click a state name or click the right-side arrow to access its entities.

Add state
Adds a new visibility state to the parameter. If the parameter already has states with entities assigned, all these entities will, by default, be invisible for the new state.

35.34.1 State entities

After selecting a visibility state, the panel shows the list of entities assigned to that state.



1 Add entities

2 Remove entities

3 List of entities

4 Back button

Add entities
Allows you to select entities in the drawing and add them to this parameter.
Entities that are not in the parameter are added and made visible only for the selected state, and invisible for all other states. Entities that are already in the parameter are made not affected.

Remove entities
Allows you to select entities in the drawing and remove them from this parameter and all its states.

Note: If entities are already selected before adding or removing them, the actions apply on the already selected entities.

List of entities
Lists all the entities that belong to the selected visibility state, along with an icon that indicates the visibility of each of these entities.

Note: One entity can be assigned to multiple parameters at the same time. In this case, the entities will only be visible if they are set to be visible for the current states of both parameters.

If you select one or more entities, then click one entity's 'eye' icon, its visibility is switched (from visible to invisible or vice versa). The new visibility also applies to all other selected entities.

Note: If multiple entities were selected and the eye icon was clicked for one entity that is currently invisible, all selected entities will become visible.

Whenever there are entities that are (in)visible for all states of a parameter, a message label will appear at the bottom of the main view of the panel, together with two link buttons. **Show** will highlight these entities in red and, if they are invisible, make them visible and semi-transparent. **Remove from Parameter** will remove these entities from the appropriate parameters, making them visible but no longer linked to the parameter.
The selection of entities in the panel changes the selection in the drawing, and vice versa. This allows you to easily check which entities are visible or invisible in the current state.

Back button
Takes you back to the main view of the panel.

35.34.2 Context menu options

Show invisible
This option can be toggled on and off. When on, any entities that are currently made invisible by the selected state are highlighted in blue and are semi-transparent.

Note: This enables you to perform any actions for which you might need to select both visible and invisible entities at the same time (move, align, remove from parameter).

Note: This option stays active until you close the panel, switch to a new document or enter block edit mode.

Options for Parameter

Remove
Removes the parameter and its states. Any entities that were made invisible by this parameter will become visible again.

Rename
Allows you to type a new name to replace the existing one.

Note: As for parameters and constraints, there are limitations on which characters are allowed in a parameter.

Remove entities from parameter
Removes the selected entities from the parameter.

Options for State

Clone
Creates a copy of the state.

Remove

Removes the state from the parameter.

Note: This option does not affect the other states of the parameters.

Rename

Type a new name to replace the existing one.

Note: There are no limitations on which characters are allowed.

Make entities visible

Makes the selected entities visible.

Note: Entities that are not in the parameter are added and made visible only for the selected states. Entities that are already in the parameter are made visible for the selected state, without affecting the other states.

Make entities invisible

Makes the selected entities invisible.

Note: Entities that are not in the parameter are added and made invisible only for the selected states and visible for the others. Entities that are already in the parameter are made invisible for the selected state, without affecting the other states.