Bricsys®

Справочник команд V22

Документация по продукту



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1. Command reference

1.1 Executing Commands

In BricsCAD, you can launch commands in the following ways:

- Command Bar: by typing the CommandName
- Menu Bar: searching in the menu bar the CommandName, e.g. Draw | Line
- Ribbon: searching in the ribbon the CommandName, e.g. Home | Draw | Line
- Toolbar: searching in the toolbar the CommandName, e.g. Draw |

  Note: To open a specific toolbar, place the cursor on a toolbar, right-click and open **Toolbars** > **BricsCAD** and choose the toolbar that you want to be visible on the display.
- Quad tab: searching in the quad tab the CommandName, e.g. Draw |

1.2 Command options

Command options appear in the status bar or command bar and also in a prompt menu, depending on the PROMPTMENU system variable setting.

The PROMPTOPTIONFORMAT system variable controls how command options display in the command line prompts.

1.3 Command Prefixes

The following prefixes can be used with the BricsCAD commands:

Prefix	Type	Result
' (single quote or apostrophe)	Transparency	Allows a command to be executed inside another command. E.g. If you are drawing a line, type **'zoom** at the command prompt to temporarily interrupt the LINE command while executing the ZOOM command.
- (dash)	Command line	Launches a non-dialog version of a command. All command options are available in the Command line. E.g. Type **-layer** in the command prompt to suppress the Drawing explorer / Layers dialog box. All command options are available in the Command line. This is especially useful if you need to execute such commands in a script.
_ (underscore)	Global	Executes the original English command (global command) in translated version of BricsCAD. It is recommended to use the global command prefix in scripts, or when defining keyboard shortcuts or customized tools.

Prefix	Type	Result
. (period)	Non-redefined	Executes commands that have been temporarily removed from BricsCAD using the **UNDEFINE** command.

Command prefixes can be combined. E.g. if the ZOOM command is temporarily undefined '.zoom executes the ZOOM command transparently.

2. +

2.1 +STRUCTUREPANEL команда

Открывает диалоговое окно "Файл конфигурации дерева структуры".

Shape Lite Pro Mechanical BIM

Значок:

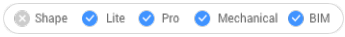
2.1.1 Description (Описание)

Открывает диалоговое окно "Файл конфигурации дерева структуры" для выбора cst файла для включения в панель Структура. После выбора **Открыть** в диалоговом окне, отобразится панель Структура.

3. ?

3.1 ? команда

Открывает Справочный центр BricsCAD.



3.1.1 Description (Описание)

Открывает описание Справочного центра BricsCAD, чтобы узнать больше о командах BricsCAD, системных переменных и рабочих процессах. Он открывается в окне внешнего приложения, используя веб-браузер по умолчанию, что позволяет ему оставаться открытым, пока вы работаете над своими чертежами в BricsCAD. Вы можете перемещать и изменять его размер с помощью стандартных элементов управления окном приложения.

4. 2

4.1 2DINTERSECTION команда

Переключает объектную привязку Пересечение.

Shape Lite Pro Mechanical BIM

4.1.1 Description (Описание)

Объектная привязка Мнимое пересечение позволяет курсору привязываться к пересечению объектов, даже если они только выглядят пересекающимися, но не пересекаются физически в 3D пространстве.

Note: Эта команда может быть введена прозрачно во время выполнения команд, но должна быть введена без префикса апострофа прозрачности (').

5. 3

5.1 3D команда

Рисует основные трехмерные сетчатые объекты.

Shape Lite Pro Mechanical BIM

5.1.1 Description (Описание)

Рисует основные трехмерные сетчатые объекты, включая параллелепипеды, конусы, цилиндры, чаши, купола, сетки, пирамиды, сферы, торы и клинья.

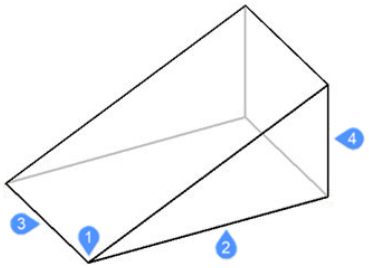
5.1.2 Methods (Методы)

Существует десять методов для базовых сетчатых 3D сущностей:

- Клин
- Пирамида
- Параллелепипед
- Конус
- Цилиндр
- Сфера
- Чаша
- Купол
- Тор
- Сетка

Клин

Рисует 3D клинья (см. команду AI_WEDGE).



1 Угол клина (начальная точка)

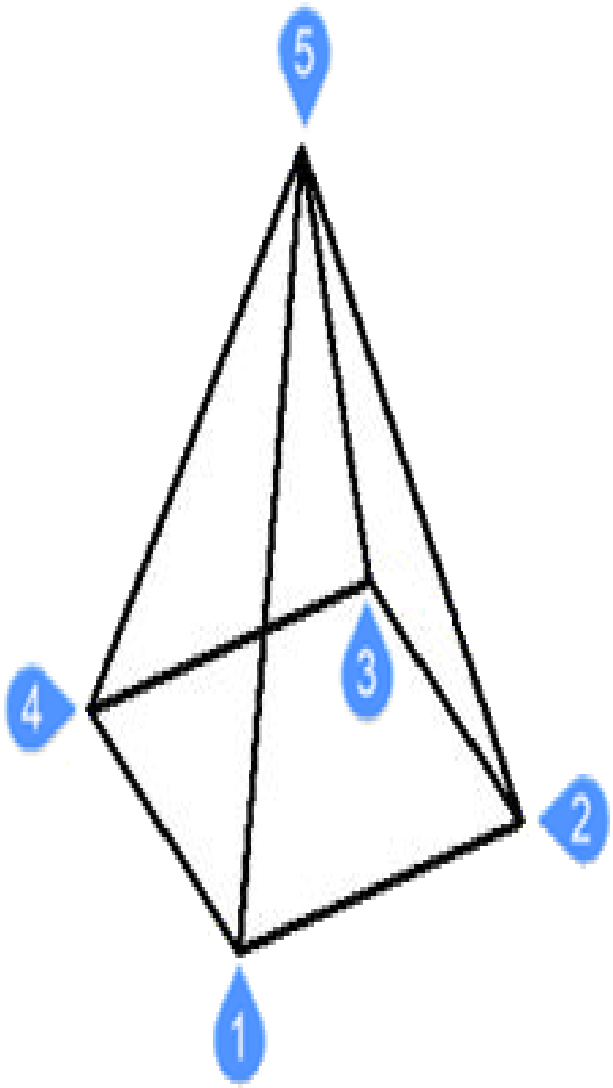
2 Длина клина

3 Ширина клина

4 Высота клина

Пирамида

Рисует 3D пирамиды (см. команду AI_PYRAMID).



1 Первая точка основания (Начальная точка)

2 Вторая точка

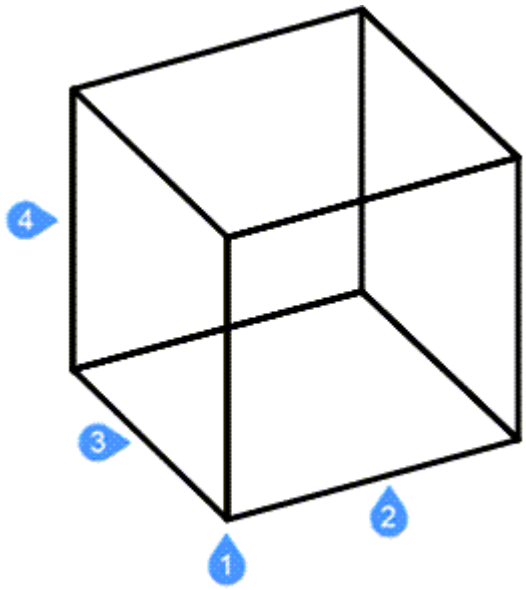
3 Третья точка

4 Последняя точка основания

5 Укажите вершину пирамиды

Параллелепипед

Рисует 3D параллелепипеды из многогранных сеток (см. команду AI_BOX).



1 Угол параллелепипеда (начальная точка)

2 Длина стороны параллелепипеда

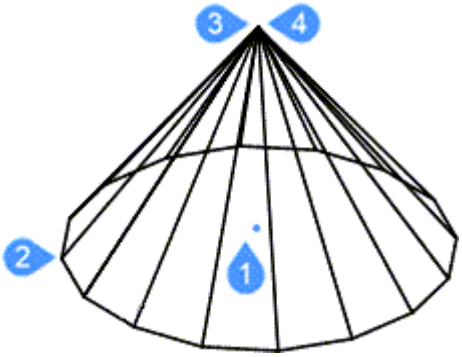
3 Ширина параллелепипеда

4 Высота параллелепипеда

(Завершите работу, применив угол поворота параллелепипеда)

Конус

Рисует 3D конусы (см. команду AI_CONE).



1 Центр основания конуса (начальная точка)

2 Радиус основания конуса или Диаметр

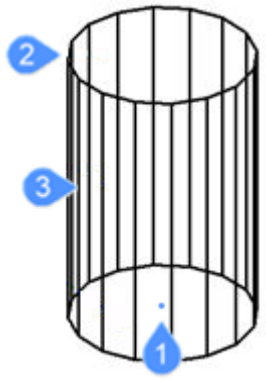
3 Радиус верха конуса или Диаметр

4 Высота конуса

(Закончите нанесением количества сегментов в конусе)

Цилиндр

Рисует 3D цилиндры (см. команду AI_CYLIDER).



1 Укажите центр основания цилиндра (начальная точка)

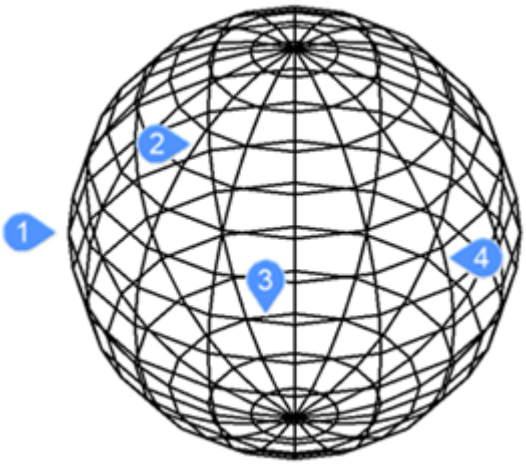
2 Радиус основания цилиндра или Диаметр

3 Высота цилиндра

(Закончите нанесением количества сегментов в цилиндре)

Сфера

Рисует 3D сферы (см. команду AI_SPHERE).



1 Укажите центр сферы (начальная точка)

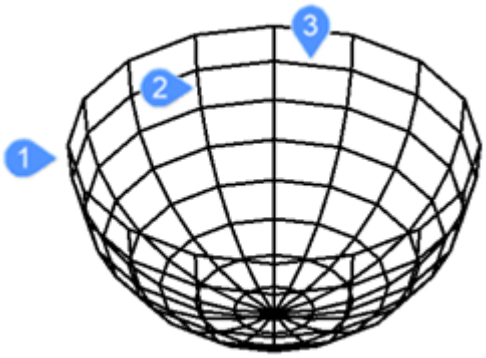
2 Радиус сферы (Диаметр)

3 Количество долготных сегментов

4 Количество широтных сегментов

Чаша

Рисует 3D чаши (нижняя половина сфер) (см. команду AI_DISH).



Укажите центр чаши (начальная точка)

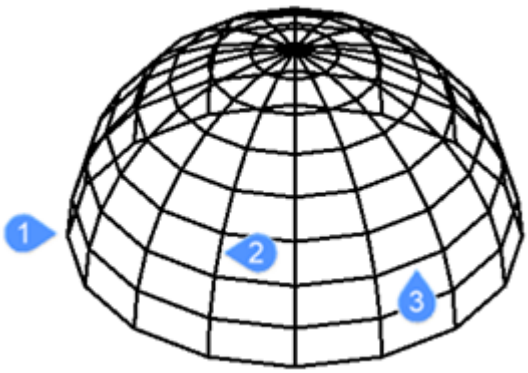
1 Радиус чаши (Диаметр)

2 Количество сегментов вокруг чаши

3 Количество сегментов сверху вниз

Купол

Рисует 3D купола (верхние половины сфер) (см. команду AI_DOME).



Укажите центр купола (начальная точка)

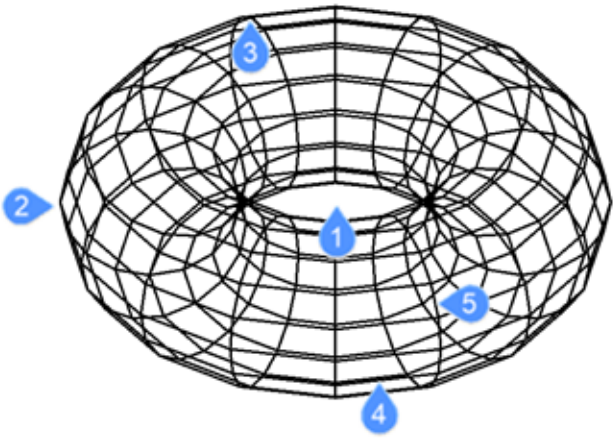
1 Радиус купола (Диаметр)

2 Количество сегментов вокруг купола

3 Количество сегментов сверху вниз

Тор

Рисует 3D торы (пончики) (см. команду AI_TORUS).



1 Укажите центр всего тора (начальная точка)

2 Укажите радиус тора

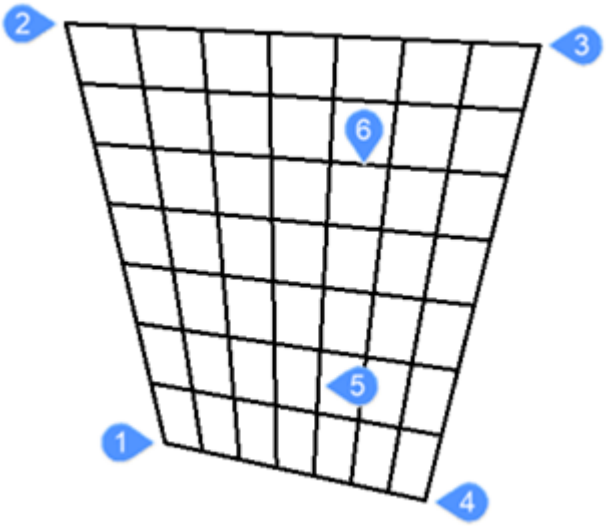
3 Радиус тела тора (Диаметр)

4 Количество сегментов вокруг тела тора

5 Количество сегментов вокруг всего тора

Сетка

Рисует 3D сетки (см. команду 3DMESH).



1 Первый из четырех углов многоугольной сетки (начальная точка)

2 Второй угол

3 Третий угол

4 Последний угол сетки

5 Количество сегментов - первичная ось

6 Количество сегментов - вторичная ось

5.2 3DARRAY команда

Конструирует статичные прямоугольные и полярные массивы в строках, столбцах и уровнях.

Shape Lite Pro Mechanical BIM

Значок: 

Псевдоним: 3A, ARRAY3D

5.2.1 Description (Описание)

Создает 3D прямоугольные массивы с использованием строк, столбцов и уровней и 3D полярные массивы с использованием центральной оси/центральной точки и угла поворота.

5.2.2 Способы редактирования

Существует два метода создания 3D массивов:

- Полярный
- Прямоугольный

5.2.3 Опции команды

Полярный
Создает 3D полярные массивы.

Прямоугольный
Создает 3D прямоугольные массивы.

5.3 3DCOMPARE команда

Открывает Диалоговое окно Сравнение двух моделей

Shape Lite Pro Mechanical BIM

Значок: 

5.3.1 Description (Описание)

Открывает диалоговое окно Сравнение двух моделей для сравнения геометрии твердых тел и поверхностей в двух файлах чертежах.

5.4 3DCONVERT команда

Конвертирует твердые тела в многогранные сетки.

Shape Lite Pro Mechanical BIM

5.4.1 Description (Описание)

Конвертирует выбранные 3D объекты ACIS в 3D многогранные сетки.

5.5 3DDWF команда

Открывает экспортируемый набор листов в виде диалогового окна.

Shape Lite Pro Mechanical BIM

5.5.1 Description (Описание)

Открывает диалоговое окно Экспорт 3D DWF для сохранения данных из текущего чертежа в файл 3D dwf или dwfx. Формат файла по умолчанию задается системной переменной DWFFORMAT.

5.6 3DFACE команда

Рисует трехмерные четырехсторонние грани.

Shape Lite Pro Mechanical BIM

Значок: 

Псевдоним: 3F, FACE

5.6.1 Description (Описание)

Рисует трехмерные четырехсторонние грани с дополнительными невидимыми ребрами путем ввода значений в командной строке или выбора точек в пространстве чертежа.

5.6.2 Опции команды

Укажите невидимые ребра
Делает следующее ребро невидимым.

Note: В визуальном стиле Каркас это ребро не видно.

5.7 3DINTERSECTION команда

Переключает объектную привязку Пересечение.

Shape Lite Pro Mechanical BIM

Значок: 

5.7.1 Description (Описание)

Эта команда может быть введена прозрачно во время выполнения команд, но должна быть введена без префикса апострофа прозрачности (').

Note: Эта объектная привязка позволяет курсору привязываться к объектам, которые физически пересекаются друг с другом. Для привязки к видимым пересечениям используйте объектную привязку 2DINTERSECTION.

5.8 3DMESH команда

Создает трехмерные сетки поверхностей.

Shape Lite Pro Mechanical BIM

Значок: 

Псевдоним: MESH

5.8.1 Description (Описание)

Создает трехмерные сетки поверхностей, путем ввода количества вершин и их расположений.

5.8.2 Опции команды

Количество вершин в направлении M (от 2 до 256)
Указывает количество вершин в первичном направлении.

Количество вершин в направлении N (от 2 до 256)
Указывает количество вершин во вторичном направлении.

Расположение вершины (M,N)
Укажите расположение вершины.

Note: Трехмерные сетки можно редактировать непосредственно через узловые точки.

5.9 3DOSNAP команда

Открывает диалоговое окно "Параметры" с раскрытой категорией **Режим 3D-привязки**.

Shape Lite Pro Mechanical BIM

Значки: 

5.9.1 Description (Описание)

Открывает диалоговое окно "Параметры" с развернутой категорией **Режим 3D-привязки** для просмотра и изменения соответствующих системных переменных.

5.10 -3DOSNAP команда

Переключает режимы объектной привязки для 3D объектов.

Shape Lite Pro Mechanical BIM

5.10.1 Description (Описание)

Переключает режимы 3D-привязки объектов для рисования и редактирования объектов в трехмерном пространстве в командной строке (сокращение от "3D object snap").

5.10.2 Опции команды

ближайшая точка (ZNEA)
Привязка к ближайшей к курсору точке на грани трехмерного объекта, эквивалентна команде ZNEAREST.

вершина (ZVER)
Привязка к ближайшей вершине трехмерного объекта, эквивалентна команде ZVERTEX.

средняя точка (ZMID)
Привязка к средней точке края грани, эквивалентна команде ZMIDPOINT.

центр (ZCEN)
Привязка к центру плоской или изогнутой 3D грани, эквивалентна команде ZCENTER.

перпендикуляр (ZPER)
Привязка к точке, перпендикулярной грани, эквивалентна команде ZPERPENDICULAR.

узел (ZKNO)
Привязка к узлу на сплайне, эквивалентна команде ZKNOT.

фактическое пересечение (ZINT)
Привязка к пересечению линейных объектов, ребер и линий трассировки полярных или объектных привязок с гранями.

облако точек ближайшая (PCN)
Привязка к ближайшей точке облака точек.

обнулить привязки (NON)
Выключает все режимы 3D привязки, эквивалентно команде ZNONE.

включить (ON)
Включает все установленные в данный момент режимы 3D привязки.

выключить (OFF)
Отключает все установленные в данный момент режимы 3D привязки.

5.11 3DPOLY команда

Создает трехмерную полилинию

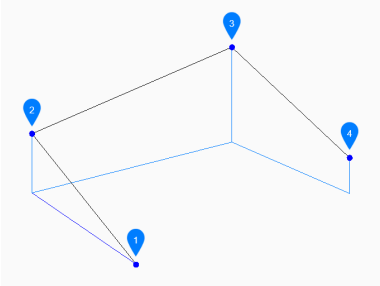
Shape Lite Pro Mechanical BIM

Значок: 

Псевдоним: 3P

5.11.1 Description (Описание)

Создает трехмерную полилинию как единый объект с несколькими линейными сегментами, путем указания начальной и конечной точек каждого сегмента. Опции позволяют отменить и замкнуть геометрию.



1 Начало

2 Найти следующий

3 Найти следующий

4 Конец

5.11.2 Методы создания трехмерной полилинии

Эта команда имеет 3 способа начать создание трехмерной полилинии:

- Начало полилинии
- Последняя точка
- Продлить

Вы можете продолжать добавлять неограниченное количество сегментов, пока не нажмете Enter для завершения команды.

Начало полилинии
Начните создание трехмерной полилинии, указав начальную точку:

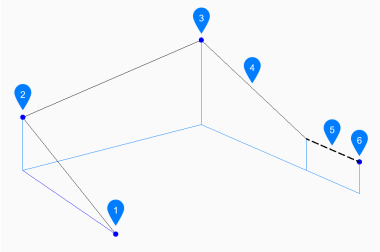
Укажите следующую точку или
Укажите следующую вершину трехмерной полилинии.
Дополнительные опции: [замкнуть (C)/на шаг назад (U)]:

Последняя точка
Начните создание трехмерной полилинии от последней выбранной точки:

Следующая точка
Укажите следующую вершину трехмерной полилинии.
Дополнительные опции: [замкнуть (C)/на шаг назад (U)]:

Продлить
Начните создание трехмерной полилинии с последнего нарисованного дугового или линейного сегмента, следуя его углу, затем:

Следующая точка
Укажите длину сегмента. Выбор точки определяет только длину, так как угол следует за последним отрезком.
Дополнительные опции: [замкнуть (C)/на шаг назад (U)]:



1 Начало

2 Найти следующий

3 Найти следующий

4 Последний нарисованный сегмент

5 Продлить

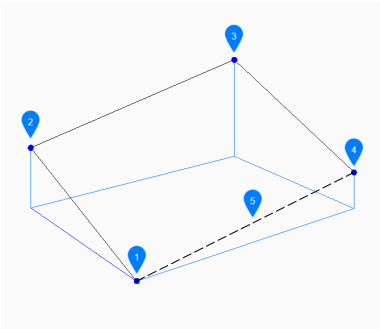
6 Конец

5.11.3 Опции команды

После начала создания трехмерной полилинии могут быть доступны следующие опции:

На шаг назад
Отмените последний сегмент трехмерной полилинии и продолжите рисование с предыдущей начальной точки.

Закрыть
Автоматическое построение сегмента трехмерной полилинии от конечной точки последнего сегмента до начальной точки первого сегмента. На этом команда завершается.



1 Начало/конец

2 Найти следующий

3 Найти следующий

4 Найти следующий

5 Последний нарисованный сегмент

6. A

6.1 ABOUT команда

Открывает диалоговое окно О BricsCAD.

Shape Lite Pro Mechanical BIM

Значок: 

6.1.1 Description (Описание)

Открывает диалоговое окно О BricsCAD для просмотра информации о версии BricsCAD и авторских правах.

6.2 ACISIN command

Opens the Open an ACIS file dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

6.2.1 Description

Opens the Open an ACIS file dialog box to select an sat or sab file to import into the current drawing.

6.3 ACISOUT command

Exports solid and surface models in ACIS format for use by other solid modeling programs.

Shape Lite Pro Mechanical BIM

Icon: 

6.3.1 Description

Opens the Create an ACIS file dialog box to export 3D solids and surface entities in ACIS format, either in human-readable ASCII .sat or compact binary .sab format. Exported files can be shared with other 3D CAD programs capable of solid modeling or analyzing solid models.

6.4 ACTIVATESECTIONVIEW команда

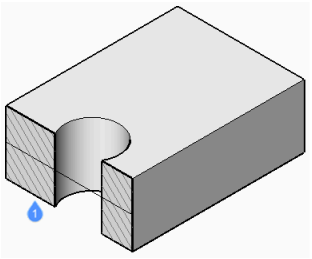
Устанавливает точку обзора по плоскости сечения.

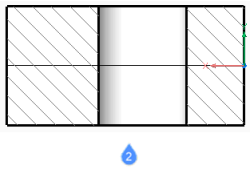
Shape Lite Pro Mechanical BIM

6.4.1 Description (Описание)

Команда переориентирует точку обзора так, чтобы она была направлена на выбранную плоскость сечения.

Для этого требуется наличие на чертеже хотя бы одной плоскости сечения, выполненной с помощью команды SECTIONPLANE.





1 Секущая плоскость

2 Вид, ориентированный в соответствии с плоскостью сечения

6.5 ADDINMAN команда

Открывает диалоговое окно Диспетчер дополнений (Add-in manager).

Shape Lite Pro Mechanical BIM

6.5.1 Description (Описание)

Открывает диалоговое окно Диспетчер дополнений (Add-in manager) для указания способа загрузки модулей VBA.

6.6 ADDSELECTED команда

Создает объект, аналогичный существующему.

Shape Lite Pro Mechanical BIM

Значок:

6.6.1 Способы редактирования

Начните добавление нового объекта на чертеж, выбрав существующий объект с тем же типом и свойствами, которые вы хотите добавить. BricsCAD автоматически запускает соответствующую команду и применяет свойства для создания соответствующего объекта. Например, выберите сплайн, и программа сразу же запустит команду Spline. Некоторые типы объекта пока не поддерживаются.

6.7 AI_BOX команда

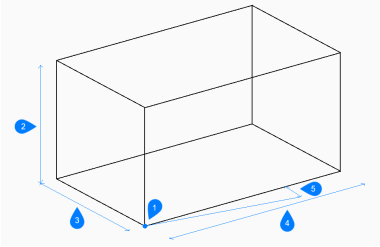
Создает трехмерную многогранную сетку в форме параллелепипеда.

Shape Lite Pro Mechanical BIM

Значок:

6.7.1 Description (Описание)

Создает трехмерную многогранную сетку в форме прямоугольного или квадратного параллелепипеда. Выберите один из вариантов, включая вершину, длину, ширину, высоту, куб и угол поворота.



1 Вершина параллелепипеда

2 Высота

3 Ширина

4 Длина

5 Угол поворота

6.7.2 Методы создания параллелепипеда

Существует один способ начать создание параллелепипеда:

- Вершина параллелепипеда

Вершина параллелепипеда
Начните создание параллелепипеда, указав вершину основания коробки:

Длина стороны параллелепипеда
Укажите длину параллелепипеда.

Укажите ширину параллелепипеда
Укажите ширину параллелепипеда.
Дополнительная опция: [Куб]

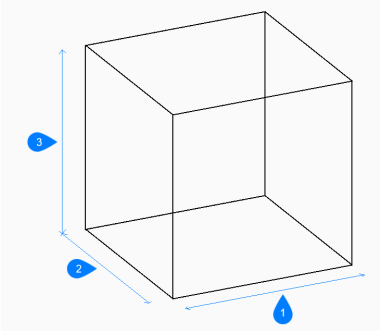
Высота параллелепипеда
Укажите высоту параллелепипеда.

Угол поворота параллелепипеда
Укажите угол поворота основания параллелепипеда в плоскости xy. По умолчанию - 0.

6.7.3 Опции команды

После начала создания параллелепипеда могут быть доступны следующие опции:

Куб
Укажите одно расстояние, которое будет использоваться для длины, ширины и высоты параллелепипеда.



1 Длина

2 Ширина

3 Высота

6.8 AI_CONE команда

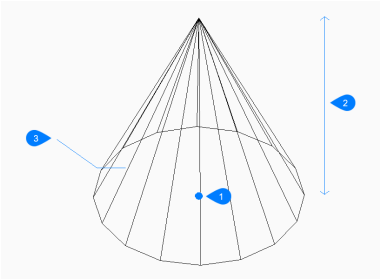
Создает трехмерную полигональную сетку в форме конуса.

Shape Lite Pro Mechanical BIM

Значок:

6.8.1 Description (Описание)

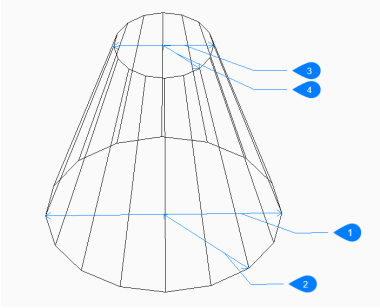
Создает трехмерную полигональную сетку в форме круглого или эллиптического конуса. Выберите один из вариантов, включая центр, радиус, диаметр и количество сегментов.



1 Центр грани

2 Высота

3 Сегменты

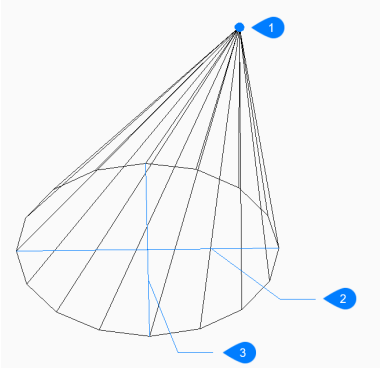


1 Диаметр основания

2 Радиус основания

3 Верхний диаметр

4 Радиус верхнего основания



1 Вершина

2 Ось эллипса

3 Вторая ось

6.8.2 Методы создания конуса

Эта команда имеет 2 способа начать создание конуса:

- Центр основания конуса
- Эллиптический

Центр основания конуса
Начните создание круглого конуса с указания центра основания:

Радиус основания
Укажите радиус основания конуса.
Дополнительная опция: [Диаметр]

Высота конуса
Укажите высоту конуса.

Число сегментов
Укажите количество изолиний, используемых для моделирования изогнутой поверхности конуса. По умолчанию - 16.

Эллиптический
Начните создание эллиптического конуса, указав первый конец оси эллипса, затем:
Дополнительная опция: [Центр]

Конечная точка оси эллипса
Укажите второй конец оси эллипса.

Вторая ось
Укажите радиус второй оси.

Высота конуса
Укажите высоту конуса.
Дополнительная опция: [Вершина]

Число сегментов
Укажите количество изолиний, используемых для моделирования изогнутой поверхности конуса. По умолчанию - 16.

6.8.3 Опции команды

После начала создания конуса могут быть доступны следующие опции:

Диаметр
Укажите диаметр основания или вершины конуса.

Центр грани
Укажите центр эллиптического основания.

Вершина
Укажите местоположение вершины конуса.

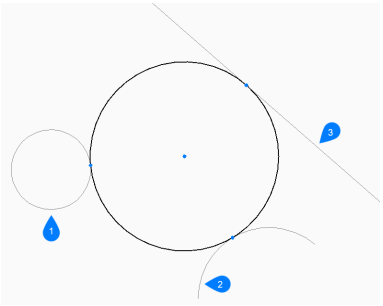
6.9 AI_CIRCTAN command

Creates a tangent circle.

Shape Lite Pro Mechanical BIM

6.9.1 Description

Creates a circle tangent to three entities that can include lines, polyline segments, polyarcs, circles, arcs, xlines or rays.



1 First tangent spec

2 Second tangent spec

3 Third tangent spec

6.9.2 Methods to create a tangent circle

There is one method to begin creating a circle with three tangents:

- Enter tangent spec

Enter Tangent spec
Begin creating a circle by selecting a tangent point on the first entity then:

Enter second Tangent spec
Select a tangent point on the second entity.

Enter third Tangent spec
Select a tangent point on the third entity.

6.10 AI_CYLINDER command

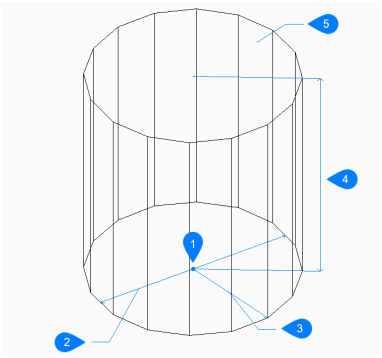
Creates a 3D polygon mesh in the shape of cylinder.

Shape Lite Pro Mechanical BIM

Icon:

6.10.1 Description

Creates a 3D polygon mesh in the shape of a circular or elliptical cylinder. Choose from a combination of options including center, radius, diameter, height, axis endpoints, and number of segments.



1 Center

2 Diameter

3 Radius

4 Height

5 Segment

6.10.2 Methods tocreatea cylinder

This command has 2 methods to begin creating a cylinder:

- Center of base of cylinder
- Elliptical

Center of base of cylinder
Begin creating a circular cylinder by specifying the center of the base then:

Radius of base of cylinder
Specify the radius of the cylinder.
Additional option: [Diameter]

Height of cylinder
Specify the height of the cylinder.

Number of segments
Specify the number of isolines used to simulate the cylinder's curved surface. The default is 16.

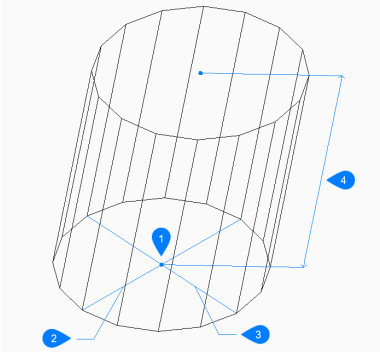
Elliptical
Begin creating an elliptical cylinder by specifying the first end of the ellipse axis then:
Alternate option: [Center]

Second end of ellipse axis
Specify the second end of the ellipse axis.

Other axis
Specify the radius of the other axis.

Height of cylinder
Specify the height of the cylinder.
Additional option: [Center of second end]

Number of segments
Specify the number of isolines used to simulate the cylinder's curved surface. The default is 16.



1 Center

2 First axis

3 Other axis

4 Height

6.10.3 Optionswithin the AI_CYLINDER command

After you begin creating a cylinder, the following options may be available:

Diameter
Specify the diameter of the cylinder.

Center of second end
Specify the center for the other end of the axis to create an oblique cylinder.

6.11 AI_DISH команда

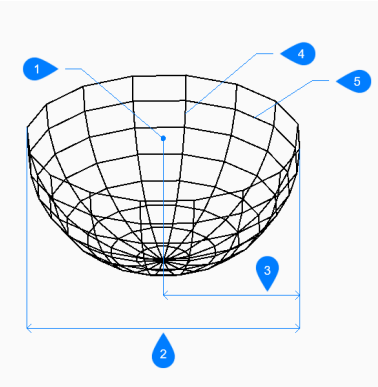
Создает трехмерную полигональную сетку в форме нижней половины сферы.

Shape Lite Pro Mechanical BIM

Значок:

6.11.1 Description (Описание)

Создает трехмерную полигональную сетку в форме нижней половины сферы. Выберите один из вариантов, включая центр, радиус, диаметр и количество сегментов.



1 Центр грани

2 Диаметр

3 Радиус

4 Сегменты сверху вниз

5 Сегменты вокруг чаши

6.11.2 Методы создания чаши

Существует один способ начать создание чаши:

- Центр чаши

Центр чаши
Начните создание чаши, указав центр открытого конца:

Укажите радиус чаши
Укажите радиус чаши.
Дополнительная опция: [Диаметр]

Количество сегментов вокруг чаши
Укажите количество изолиний, перпендикулярных плоскости xy, используемых для моделирования изогнутой поверхности чаши. По умолчанию - 16.

Количество сегментов сверху вниз
Укажите количество изолиний, параллельных плоскости xy, используемых для моделирования изогнутой поверхности чаши. По умолчанию - 8.

6.11.3 Опции команды

После начала создания трехмерной полилинии могут быть доступны следующие опции:

Диаметр
Укажите диаметр чаши.

6.12 AI_DOME команда

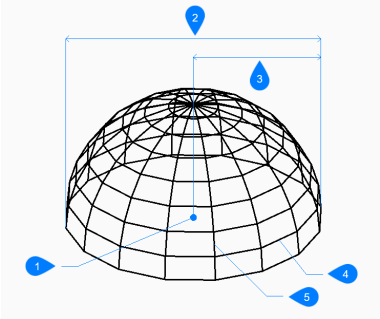
Создает трехмерную полигональную сетку в форме верхней половины сферы.

Shape Lite Pro Mechanical BIM

Значок:

6.12.1 Description (Описание)

Создает трехмерную полигональную сетку в форме верхней половины сферы. Выберите один из вариантов, включая центр, радиус, диаметр и количество сегментов.



1 Центр грани

2 Диаметр

3 Радиус

4 Сегменты сверху вниз

5 Сегменты вокруг купола

6.12.2 Методы создания купола

Существует один способ начать создание купола:

- Центр купола

Центр купола
Начните создание купола, указав центр открытого конца:

Радиус купола
Укажите радиус купола.
Дополнительная опция: [Диаметр]

Количество сегментов вокруг купола

Укажите количество изолиний, перпендикулярных плоскости xy, используемых для моделирования изогнутой поверхности купола. По умолчанию - 16.

Количество сегментов сверху вниз

Укажите количество изолиний, параллельных плоскости xy, используемых для моделирования изогнутой поверхности купола. По умолчанию - 8.

6.12.3 Опции команды

После начала создания купола могут быть доступны следующие опции:

Диаметр

Укажите диаметр купола.

6.13 AI_EDGESURF команда

Эта команда устарела. Она по-прежнему существует только для обратной совместимости.

Shape Lite Pro Mechanical BIM

Вместо этого используйте команду EDGESURF.

6.14 AI_PYRAMID команда

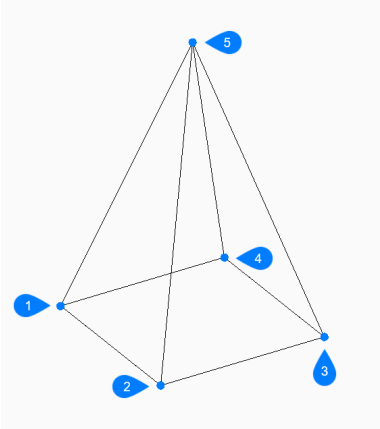
Создает трехмерную многогранную сетку в форме пирамиды.

Shape Lite Pro Mechanical BIM

Значок: 

6.14.1 Description (Описание)

Создает трехмерную многогранную сетку в форме пирамиды с тремя или четырьмя гранями. Стороны основания могут иметь неодинаковую длину. Опции позволяют задать заостренную, ребристую или плоскую вершину.



1 Первая точка

2 Вторая точка

3 Третья точка

4 Последняя точка

5 Вершина

6.14.2 Методы создания пирамиды

Существует один способ начать создание пирамиды

- Первая точка основания пирамиды:

Первая точка основания пирамиды
Начните создание пирамиды, указав угол для основания пирамиды, затем:

Вторая точка
Укажите вторую точку основания.

Третья точка
Укажите третью точку основания.

Укажите последнюю точку основания
Укажите четвертую точку основания.
Дополнительная опция: [Тетраэдр].

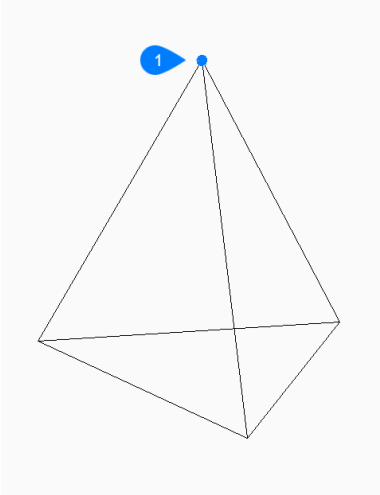
Укажите вершину пирамиды
Укажите расположение вершины пирамиды.
Дополнительные опции: [ребро/верхнее основание]

6.14.3 Опции команды

После начала создания пирамиды могут быть доступны следующие опции:

Тетраэдр
Выберите эту опцию, чтобы закончить создание основания пирамиды после указания третьей точки, чтобы затем создать трехгранную пирамиду (тетраэдр):

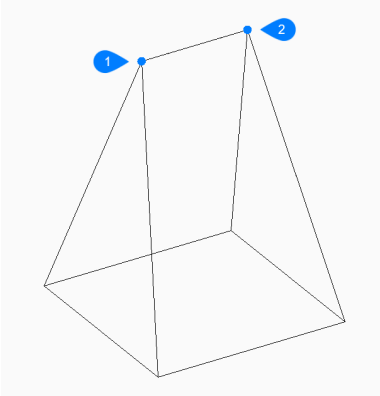
Укажите вершину тетраэдра
Укажите вершину тетраэдра.
Дополнительная опция: [Верхнее основание]



1 Вершина

Ребро

Укажите начальную и конечную точки для создания ребра на вершине пирамиды.

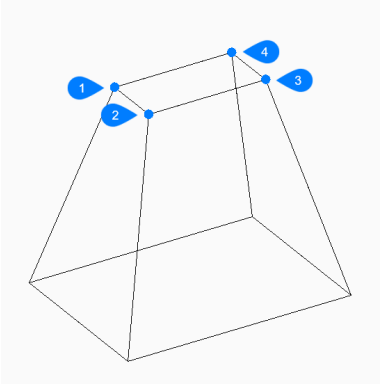


1 Начальная точка ребра

2 Конечная точка ребра

Верхнее основание

Укажите четыре точки для создания грани на вершине пирамиды.



1 Первая точка

2 Вторая точка

3 Третья точка

4 Последняя точка

6.15 AI_REVSURF команда

Эта команда устарела. Она по-прежнему существует только для обратной совместимости.

Shape Lite Pro Mechanical BIM

Вместо этого используйте команду REVSURF.

6.16 AI_RULESURF команда

Эта команда устарела. Она по-прежнему существует только для обратной совместимости.

Shape Lite Pro Mechanical BIM

Вместо этого используйте команду RULESURF.

6.17 AI_SPHERE команда

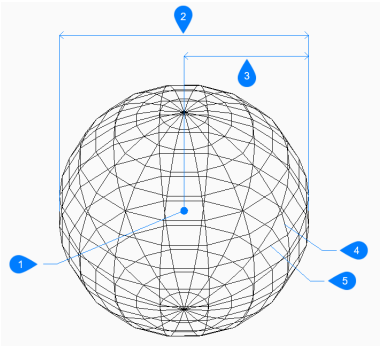
Создает трехмерную полигональную сетку в форме сферы.

Shape Lite Pro Mechanical BIM

Значок:

6.17.1 Description (Описание)

Создает трехмерную полигональную сетку в форме сферы. Выберите один из вариантов, включая центр, радиус, диаметр и количество сегментов.



1 Центр грани

2 Диаметр

3 Радиус

4 Широтные сегменты

5 Продольные сегменты

6.17.2 Методы создания сферы

Существует один способ начать создание сферы:

- Центр сферы

Центр сферы
Начните создание сферы с указания центра:

Укажите радиус сферы
Укажите радиус сферы.
Дополнительная опция: [Диаметр]

Количество долготных сегментов
Укажите количество изолиний, перпендикулярных плоскости xy, используемых для моделирования поверхности сферы. По умолчанию - 16.

Количество широтных сегментов
Укажите количество изолиний, параллельных плоскости xy, используемых для моделирования поверхности сферы. По умолчанию - 16.

6.17.3 Опции команды

После начала создания сферы может появиться следующая опция:

Диаметр
Укажите диаметр сферы.

6.18 AI_TABSURF команда

Эта команда устарела. Она по-прежнему существует только для обратной совместимости.

Shape Lite Pro Mechanical BIM

Вместо этого используйте команду TABSURF.

6.19 AI_TORUS команда

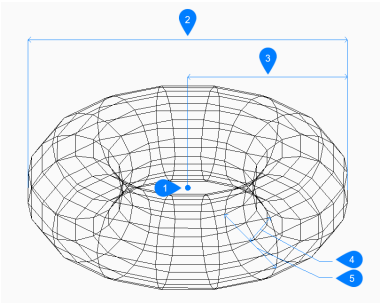
Создает трехмерную полигональную сетку в форме тора.

Shape Lite Pro Mechanical BIM

Значок:

6.19.1 Description (Описание)

Создает трехмерную полигональную сетку в форме тора. Выберите один из вариантов, включая центр и радиус или диаметр как всего тора, так и его тела.



1 Центр грани

2 Диаметр всего тора

3 Радиус всего тора

4 Радиус тела тора

5 Диаметр тела тора

6.19.2 Методы создания тора

Существует один способ начать создание тора:

- Центр всего тора

Центр всего тора

Начните создание тора с указания его центра:

Укажите радиус всего тора

Укажите общий радиус тора. Радиус измеряется от центра всего тора до внешнего края тела (трубки) тора.

Дополнительная опция: [Диаметр]

Укажите радиус тела тора

Укажите радиус тела (трубы) тора.

Дополнительная опция: [Диаметр]

Количество сегментов вокруг тела тора

Укажите количество изолиний, перпендикулярных плоскости xy, используемых для моделирования изогнутой поверхности тора. По умолчанию - 16.

Количество сегментов вокруг всего тора

Укажите количество изолиний, параллельных плоскости xy, используемых для моделирования изогнутой поверхности тора. По умолчанию - 16.

6.19.3 Опции команды

После начала создания тора может появиться следующая опция:

Диаметр (всего тора)
Укажите диаметр всего тора. Диаметр - это удвоенное расстояние от центра всего тора до внешнего края тела (трубки) тора.

Диаметр (тела тора)
Укажите диаметр тела тора.

6.20 AI_WEDGE команда

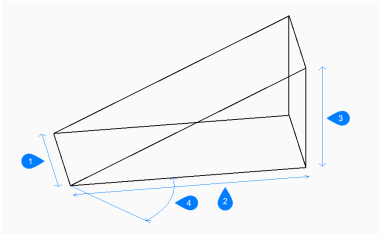
Создает трехмерную многогранную сетку в форме клина.

Shape Lite Pro Mechanical BIM

Значок:

6.20.1 Description (Описание)

Создает трехмерную многогранную сетку в форме клина путем указания угла, длины, ширины, высоты и угла поворота.



1 Ширина

2 Высота

3 Длина

4 Угол поворота

6.20.2 Методы создания клина

Существует один способ начать создание клина:

- Угол клина

Угол клина
Начните создание клина с указания угла для основания клина:

Длина клина
Укажите длину клина.

Ширина клина
Укажите ширину клина.

Высота клина
Укажите высоту клина.

Угол поворота клина
Укажите угол поворота основания клина в плоскости xy.

6.21 AIMLEADEREDITADD команда

Добавляет и удаляет выноски из мультивыносок.

Shape Lite Pro Mechanical BIM

Значок: 

6.21.1 Description (Описание)

Удаляет и добавляет одну или несколько выносных линий в многострочные выноски и из них.

6.21.2 Способы редактирования

Существует два метода добавления и удаления выносок:

- Добавить выноски к существующей мультивыноске.
- Удалить выноски из мультивыноски.

6.21.3 Опции команды

Добавить выноски
Новые выноски добавляются к существующему объекту-мультивыноске.

Удалить выноски
Один или несколько выносок удаляются из существующего объекта-мультивыноски.
Эта команда может удалить все выноски из объекта-мультивыноски, оставив только текст.

6.22 AIMLEADEREDITREMOVE команда

Удаляет и добавляет выноски в мультивыноски.

Shape Lite Pro Mechanical BIM

Значок: 

6.22.1 Description (Описание)

Удаляет и добавляет одну или несколько выносных линий в многострочные выноски и из них.

6.22.2 Способы редактирования

Существует два метода удаления и добавления выносок:

- Удалить выноски из мультивыноски.
- Добавить выноски к существующей мультивыноске.

6.22.3 Опции команды

Удалить выноски
Один или несколько выносок удаляются из существующего объекта-мультивыноски.
Эта команда может удалить все выноски из объекта-мультивыноски, оставив только текст.

Добавить выноски
Новые выноски добавляются к существующему объекту-мультивыноске.

6.23 AIDIMFLIPARROW команда

Переворачивает размерную стрелку.

Shape Lite Pro Mechanical BIM

Значок:

6.23.1 Description (Описание)

Переворачивает размерные стрелки так, чтобы они указывали в противоположном направлении.

Несмотря на то, что в подсказке говорится о множественных сущностях, команда переворачивает одну стрелку за раз, ближайшую к выбранной точке вдоль размерной или выносной линии.

6.24 AIDIMPREC команда

Изменяет точность отображения размерного текста.

Shape Lite Pro Mechanical BIM

Значок: 

6.24.1 Description (Описание)

Увеличивает и уменьшает точность отображения значений размерности задним числом, изменяя либо количество десятичных знаков, либо знаменатель дробей. Измеренная точность остается неизменной.

Эта команда округляет значения размерности в большую или меньшую сторону, поэтому она может отображать ложные значения, которые кажутся слишком большими или слишком малыми. Например, установка AIDIMPREC на 0 для размера с длиной 3,525" округляет отображение до 4", но измеренная длина остается 3,525".

6.25 AIDIMSTYLE команда

Применяет и сохраняет размерные стили.

Shape Lite Pro Mechanical BIM

Значок: 

6.25.1 Description (Описание)

Сохраняет текущие настройки размеров в виде именованного стиля и применяет заданный стиль к выбранным размерам на текущем чертеже.

6.25.2 Способы редактирования

Существует два способа начать работу с размерными стилями:

- Применить сохраненный размерный стиль к размерам на чертеже.
- Сохранить настройки размеров на текущем чертеже в виде именованного стиля.

6.25.3 Опции команды

Применить
Применить существующий размерный стиль к одному или нескольким размерам.

Note: Используйте команду '-DIMSTYLE прозрачно во время этой команды, чтобы перечислить имена размерных стилей в текущем чертеже.

Это отменяет изменения, внесенные в размеры с помощью команд AIDIMFLIPARROW и AIDIM-PREC.

Сохранить
Сохранить свойства стиля существующего размера в именованном стиле.
Свойства сохраненного стиля можно изучить с помощью команды DIMSTYLE.

6.26 ALIGN command

Moves, rotates, and scales entities to align with specified points.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: AL

6.26.1 Description

Moves, rotates and scales entities in 2D or 3D space by specifying a set of alignment points.

6.26.2 Options

Select entities
Selects entities that are to be aligned.

Note: Do not select destination entities, otherwise they are also moved.

Specify first source point
Specifies the source alignment point.

Specify first destination point
Specifies the destination alignment point.

Note: Specify a second pair of source and destination points to move and rotate entities.

Specify second source point
Specifies the second source alignment point.

Note: If you press Enter at this prompt, the source object moves.

Specify second destination point
Specifies the matching destination point.

Note: Specify a third pair of source and destination points to move and rotate 3D entities.

Specify third source point
Specifies the source point for 3D alignment.

Note: Press Enter to move and rotate 2D entities.

Specify third destination point
Specifies the matching destination point.

Scale objects based on alignment points
Optionally scales the source object:

Yes
Scales objects relative to the picked points

No
Does not scale objects

6.27 ALIGNMENT command

Creates Horizontal and 3D alignments.

Shape Lite Pro Mechanical BIM

Icon: 

6.27.1 Description

Creates Horizontal and 3D alignments to represent a reference system used to position linear objects in space.

Note: You can offset Horizontal Alignments. The result of the OFFSET command on Horizontal Alignments is a non-associative polyline.

Horizontal Alignment is created by specifying points of intersection (PI) of the tangent polygon. Curves are drawn automatically between the tangents.

Note: The DEFAULTCURVETYPEHA system variable specifies the curve type to be used when create a new horizontal alignment or adding new PI. The default value is **Auto Spiral-Curve-Spiral**.

6.27.2 Options within the command

Select TIN surface
Allows you to select a TIN surface used to drape Horizontal alignment and to create 3D alignment.

Note: Two 3D alignments are created: one represents the projection of the Horizontal Alignment to the TIN Surface and second represents the Vertical Alignment, which is calculated based on the **Vertical tolerance** parameter.

Pick pi point
Allows you to pick the PI points for your Horizontal alignment.

Note: If **Select TIN surface** option was previously selected, the 3D alignment is automatically generated.

Undo
Undoes the last PI point.

Specify start station
Allows you to define the start station when start creating a new alignment.

6.28 ALIGNMENT3D command

Creates a **3D Alignment** from a specified vertical alignment.

Shape Lite Pro Mechanical BIM

Icon: 

6.28.1 Description

Creates a **3D Alignment** entity by selecting a vertical alignment.

6.29 ALIGNMENTCURVE command

Creates unconstrained curve element on existing or on a new **Horizontal Alignment**.

Shape Lite Pro Mechanical BIM

Icon: 

6.29.1 Method

There are two methods:

- Create a new curve element on existing horizontal alignment.
- Create a new alignment by drawing a curve element.

The unconstrained curve element can be created on **existing** or on a **new Horizontal Alignment** in two ways. In both ways an existing horizontal alignment needs to be selected first. The user can then either specify a curve centre point in the drawing or select **Points** option from the command line. If the user has specified a centre point, he determines the orientation of the curve element in the next step, and at the end specify its diameter. Otherwise, if option **Points** has been selected in the command line, the user specifies three points that define the new curve element. User can then continue drawing new curve elements on existing horizontal alignment until presses the Enter key.

6.29.2 Options within the command

Select existing horizontal alignment
Creates a new curve element on existing alignment.

Create new alignment
Creates new curve element on existing **Horizontal Alignment**..

Specify curve direction
Toggles between clockwise and counterclockwise direction.

Points
Creates new curve element of the **Horizontal Alignment** by specifying its start point, second point and end point.

6.30 ALIGNMENTEDIT command

Edits an alignment.

Shape Lite Pro Mechanical BIM

Icon: 

6.30.1 Method

Select a Horizontal or Vertical alignment to edit.

6.30.2 Options within the command for Horizontal alignment

Continue with Pi
Allows you to continue the Horizontal alignment.

Note: This option works only if the last element is fixed line.

Delete Element
Allows you to delete alignment elements.

Add Pi
Adds a Pi Point to the alignment.

Remove Pi
Removes a Pi point.

changes Tin surface
Changes the Tin surface of the Horizontal alignment by adding additional TIN surface.

Add Station equation
Allows you to add station equation.

Remove Station equation
Allows you to remove station equation by specifying the index.

6.30.3 Options within the command for Vertical alignment

Add Pvi
Adds a Pvi point to the alignment.

Remove Pvi
Removes a PVI point.

change Tin surface
Changes the Tin surface of the Vertical alignment by adding additional TIN surface.

6.31 ALIGNMENTELEMENTBETWEEN command

Creates a new element between two selected alignment elements.

Shape Lite Pro Mechanical BIM

Icon: 

6.31.1 Description

The new element is drawn tangential to both attachment elements.

Note: Therefore, we say that such an element is constrained by both selected elements.

Depending on the selected type of the new element, some parameters of the element are maintained when attachment elements are edited.

The following types of new elements can be created between two existing elements: lines, curves, spirals, spiral-curve-spiral combinations, spiral-spiral combinations and the spiral-line-spiral combinations.

6.31.2 Options within the command

Line

Creates new line element between two existing curve elements. The new line is drawn tangential to both attachment elements. The tangency is maintained when attachment elements are edited.

The line can be added between the following element combinations:

- Between two existing unconstrained circles.
- Between two existing constrained circles as a part of combinations, such as a Curve with Spiral and a Curve-Spiral-Spiral combination.
- Between an existing unconstrained and constrained circle.

Note: The way the line is drawn between the selected elements depends on their directions.

Curve

Creates a new curve element between two existing alignment elements. The new curve is drawn tangential to both attachment elements. The tangency and the curve radius are maintained when attachment elements are edited.

The position of points on (attachment) elements, between which new curve is created and the curve length are adjusted according to the editing of attachment elements.

The curve can be added between all element combinations except between the first selected element and the constrained spiral.

Note: The curve is defined by the type of attachment elements, specified curve radius and solution angle (< or >180°). The way the circle is drawn between the selected elements depends on the attachment elements direction.

Spiral

Creates new spiral element between two existing alignment elements. The new spiral is drawn tangential to both attachment elements. The tangency is maintained when attachment elements are edited.

Note: A free spiral is defined by the type and the direction of selected attachment elements. Since the spiral has only one geometric solution, the user cannot specify parameters, such as parameter A or the spiral length.

Spiral-Curve-Spiral

Creates new spiral-curve-spiral combination between two existing alignment elements. The new combination is drawn tangential to both attachment elements. The tangency, curve radius and both spiral lengths are maintained, when attachment elements are edited.

The position of points on (attachment) elements, between which new combination is created and the curve length are adjusted according to the editing of attachment elements.

Select attachment elements, specify curve radius and spiral lengths to add spiral-curve-spiral combination between selected alignment elements.

Spiral-Spiral

Creates new spiral-spiral combination between two existing curve elements with the same orientation and possibly different radii. The new combination is drawn tangential to both attachment elements. The tangency is maintained when attachment elements are edited.

Spiral parameters (length and parameter A) and the position of points on (attachment) elements, between which new combination is created, are calculated according to the specified A1/A2 (or L1/L2) ratio.

Note: The position of points on (attachment) elements, between which new combination is created and the curve length are adjusted according to the editing of attachment elements.

Spiral-Line-Spiral

Creates new spiral-line-spiral combination, with a specified spiral lengths, between two existing curve elements. The new combination is drawn tangential to both attachment elements.

Note: The position of points on (attachment) elements, between which new combination is created, and spiral parameters are adjusted according to the editing of attachment elements.

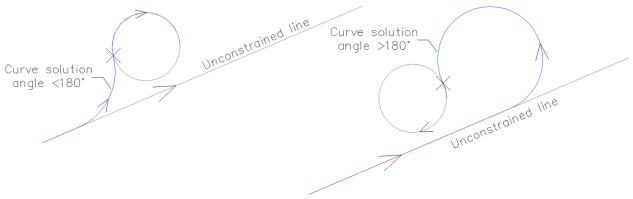
Specify first / second entity

Allows you to select the first and second alignment element, between which a new element is created.

Is curve solution angle

Toggles between **Greaterthan180** and **Lessthan180**.

The figure below shows two possible solutions for a curve with an angle < 180° (left) and > 180° (right) added between an unconstrained line and an arc element.



spiral parameter A

Allows you to specify the A1/A2 ratio.

Spiral length

Allows you to specify the L1/L2 ratio.

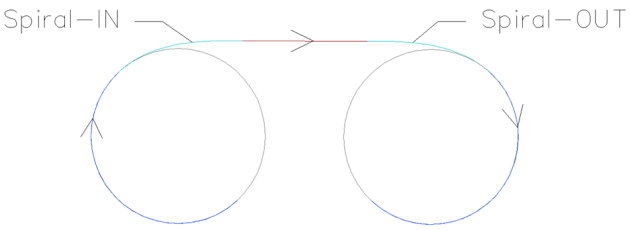
Spiral in-length

Allows you to specify the spiral-in length.

Spiral out-length

Allows you to specify the spiral-out length.

The figure below shows the spiral-line-spiral combination, created between two unconstrained curves, where the user specifies the spiral-IN and spiral-OUT lengths.



Specify radius
Allows you to specify the radius of the new spiral and curve element.

6.32 ALIGNMENTELEMENTFROM command

Attaches new alignment element before (to) or after (from) the existing alignment element.

Shape Lite Pro Mechanical BIM

Icon:

6.32.1 Description

The following types of new elements can be attached to existing elements: lines, curves, spirals, lines with spirals, curves with spirals and curve-spiral-spiral combination.

Note: New element is always drawn tangential to the specified point on the selected attachment element.

6.32.2 Options within the command

Line
Attach a new line element before (to) or after (from) the selected alignment element.

Note: Line length can be either specified in the drawing or its value can be entered in the command line.

Curve
Attach a new curve element before (to) or after (from) the selected alignment element.
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.

Note: The curve geometry is defined by the curve radius, curve angle solution (greater or less than 180°) and the pass-through point.

Spiral
Attach a new spiral element before (to) or after (from) the selected alignment element.
To attach a spiral before the existing element, select it closer to the element start point. Otherwise click closer to the attachment element end point, to attach curve after (starting from) the element.

Note: The spiral geometry is defined by the radius, length and the direction (clockwise or counter-clockwise).

Line with Spiral
Attach a **Line-Spiral** combination before (to) or after (from) the selected alignment element. As the name implies, a combination consists of two elements, a spiral and a line.

Note: The sequence of created elements depends on whether the combination is attached before or after the selected element. To create a combination before the selected element, select attachment element closer to its start point. In this way we get the **Line-Spiral** combination. Otherwise, if existing element is selected closer to its end point, then we get the **Spiral-Line** combination after the attachment element. Such combination starts by spiral transition to a line, which passes through a specified pass-through point (line end point).

Curve with Spiral

Attach a **Curve-Spiral** combination before (to) or after (from) the selected alignment element. As the name implies, a combination consists of two elements, a spiral and a curve.

Note: The sequence of created elements depends on whether the combination is attached before or after the selected element. To create a combination before the selected element, select existing element closer to its start point. In this way we get the **Curve-Spiral** combination. Otherwise, if attachment point is selected closer to element end point, then we get the **Spiral-Curve** combination. Such combination starts by spiral transition to a curve, which passes through a specified pass-through point.

First select the attachment element, then specify spiral radius and length, then specify the curve solution angle (greater or less than 180°) and last specify the curve pass-through point to draw the **Curve-Spiral** combination.

Curve-Spiral-Spiral

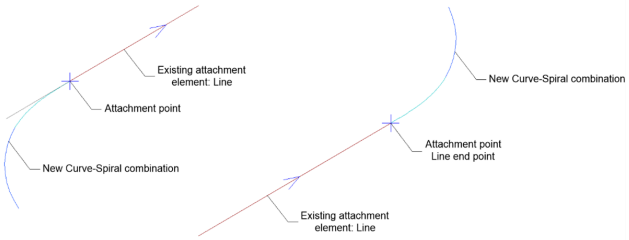
Attach a **Curve-Spiral-Spiral** combination before (to) or after (from) the selected alignment element. As the name implies, a combination consists of two oppositely directed spirals and a curve that passes through a specified point.

Note: The sequence of created elements depends on whether the combination is attached before or after the selected element. To create a combination before the selected element, select attachment element closer to its start point. In this way we get the **Curve-Spiral-Spiral** combination. Otherwise, if existing element is selected closer to its end point, then we get the **Spiral-Spiral-Curve** combination.

First select the attachment element, then specify spiral-in length and radius, then specify spiral-out length and specify curve pass-through point in the last step to draw **Curve-Spiral-Spiral** combination.

Element

Select this option to attach new element (or combination of elements) to the specified point on the attachment element. Select attachment element closer to its start point to attach new element before (to) the attachment element. Or select attachment element closer to its end point to attach new element after (from) attachment element.



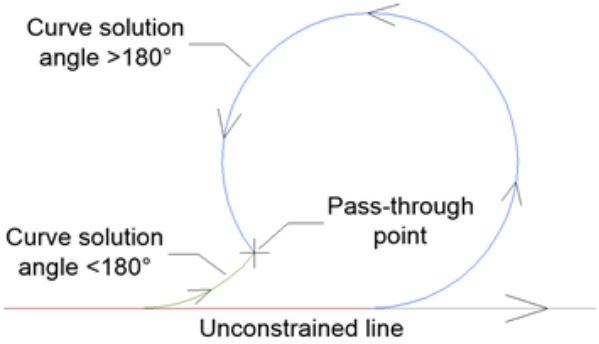
element END

Select this option to attach new element (or combination of elements) to the start/end point of the attachment element. Select attachment element closer to its start point to attach new element before (to) the attachment element. Or select attachment element closer to its end point to attach new element after (from) attachment element.

Is curve solution angle

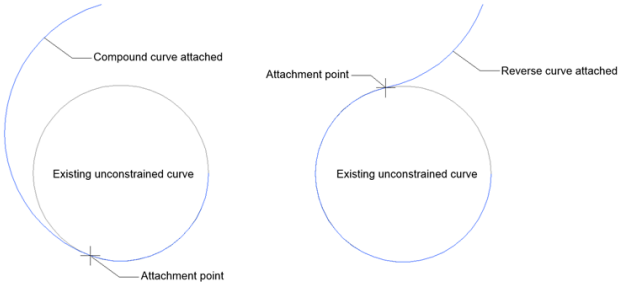
Toggles between **Greaterthan180** and **Lessthan180**.

The figure below shows two possible solutions for the curve passes through the same specified point, depends on the selected option for the curve solution angle.



Is curve compound or reverse?

Toggles between **Compound** and **Reverse** curve attached to existing unconstrained curve element, as illustrated in the figure below:



Length

Allows you to specify the new line length.

Specify spiral length

Allows you to specify the new spiral length.

Specify radius

Allows you to specify the radius of the new spiral and curve element.

Specify curve direction

Specifies the new element orientation. Toggles between clockwise and counter-clockwise orientation.

6.33 ALIGNMENTLINE command

Creates unconstrained line element between two specified points on existing or on a new **Horizontal Alignment**.

Shape Lite Pro Mechanical BIM

Icon:

6.33.1 Method

There are two options:

- Create a new line element between two specified points on existing horizontal alignment.
- Create a new alignment by drawing a line element between two specified points.

6.33.2 Options within the command

Select existing horizontal alignment
The new unconstrained line element is created on selected existing Horizontal Alignment, between two specified points.

Note: First select the existing horizontal alignment in the drawing, and then specify the first and second point, between which a new line element is created. You can then continue drawing new line elements on existing horizontal alignment until you press the Enter key.

Create new alignment
Creates new **Horizontal Alignment** by drawing a line element between two specified points.

Note: Specify the first and second point in the drawing, between which a new line element is created. You can continue drawing new line elements until press the Enter key.

6.34 ALIGNMENTVIEW command

Creates a vertical alignment view for the selected horizontal alignment.

Shape Lite Pro Mechanical BIM

Icon: 

6.34.1 Method

Select a horizontal alignment, previously created with ALIGNMENT command, and a point as the origin for vertical alignment view.

6.35 ALIGNMENTVERTICAL command

Creates **Vertical Alignment**.

Shape Lite Pro Mechanical BIM

Icon: 

6.35.1 Description

Creates **Vertical Alignment** on a selected **Vertical Alignment View**. It also automatically draws the **3D Alignment** of the corresponding alignment.

6.35.2 Method

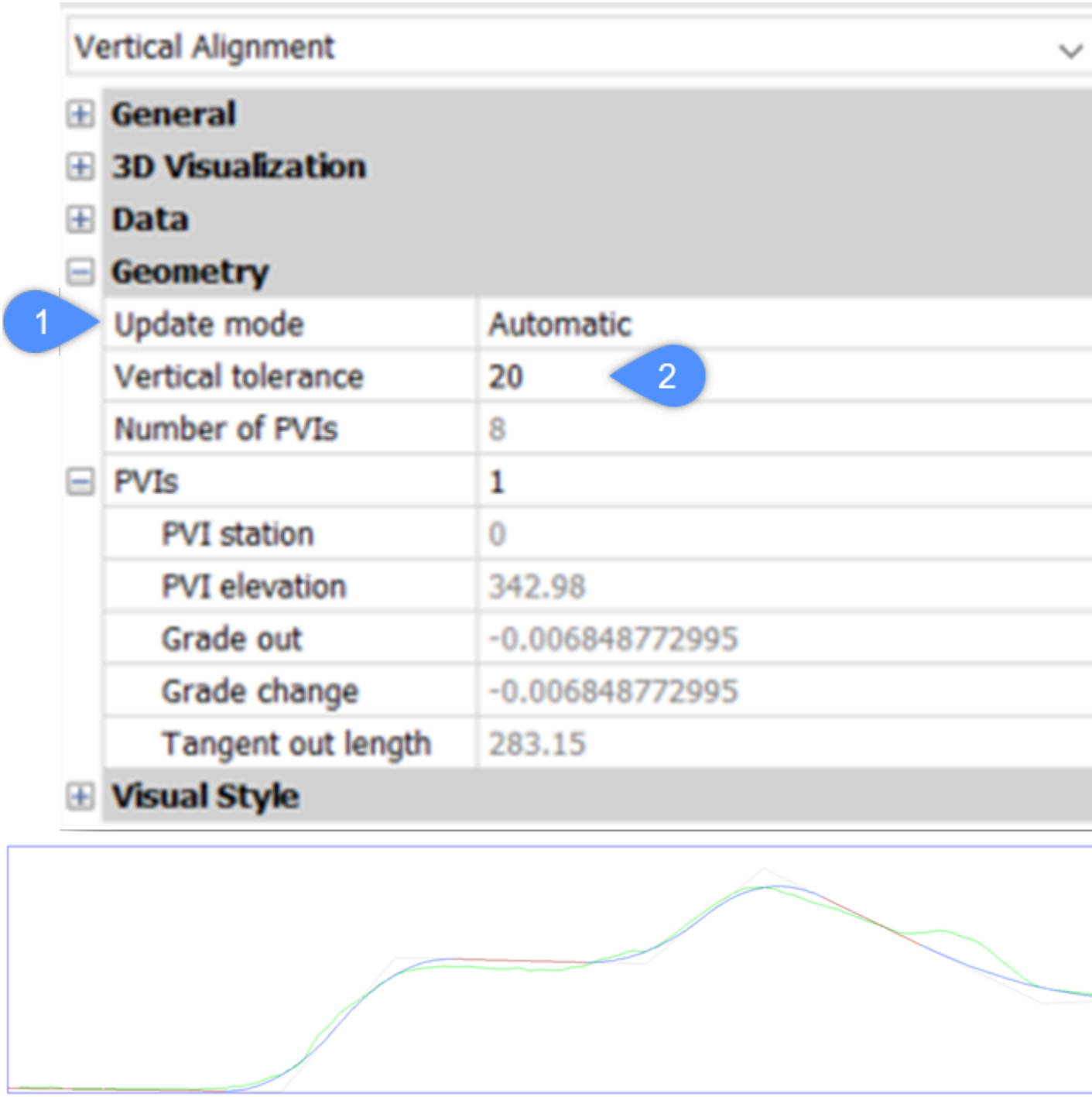
There are two methods:

- Creates **Vertical Alignment** by picking PVI points.
- Creates **Vertical Alignment** automatically.

6.35.3 Options within the command

Undo
Undoes the last specified vertical alignment PVI point.

Automatic 3d alignment
Automatically creates **Vertical Alignment** and **3D Alignment**.



The Automatic Vertical Alignment is calculated so that the elevation difference to the terrain line does not exceed the value of the **Vertical tolerance** parameter, which is set to 0.1 by default.
Once the **Vertical Alignment** is created, you can edit the value of the **Vertical tolerance** (2) parameter in the **Properties** panel.
Additionally, the value of the **Update mode** (1) parameter is set to **Automatic**. This means that you cannot edit the geometry of the Automatic Vertical Alignment. To edit it, set the **Update mode** parameter to **Manual**.

6.36 ALIGNSPACE команда

Регулирует угол обзора видового окна, коэффициент масштабирования и положение панорамирования по отношению к точкам выравнивания, указанным в пространстве модели и пространстве листа.

Shape Lite Pro Mechanical BIM

Note: Эта команда может быть введена прозрачно во время выполнения команд ('alignspace).

Note: Команда работает только в пространстве листа.

Note: Видовые экраны пространства модели должны иметь выключенный режим Перспектива.

6.36.1 Description (Описание)

Регулирует угол обзора, коэффициент масштабирования и положение панорамирования путем выбора точек в видовом окне пространства модели и пространстве листа.

6.37 AMSURFSYM command

Creates a surface symbol.

Shape Lite Pro Mechanical BIM

Icon: 

6.37.1 Methods

Select the object to attach the surface symbol and specify the points for its location.

There are two use cases:

1 **When creating a new drawing that will contain mechanical entities:**
   a Set the LOADMECHANICAL2D system variable to ON (1).
   b Start a new drawing using a Mechanical2d template.

2 **When opening a drawing that contains mechanical entities**
   a Set the LOADMECHANICAL2D system variable to ON (1).
   b Open an existent ACM drawing and start special symbols creation.

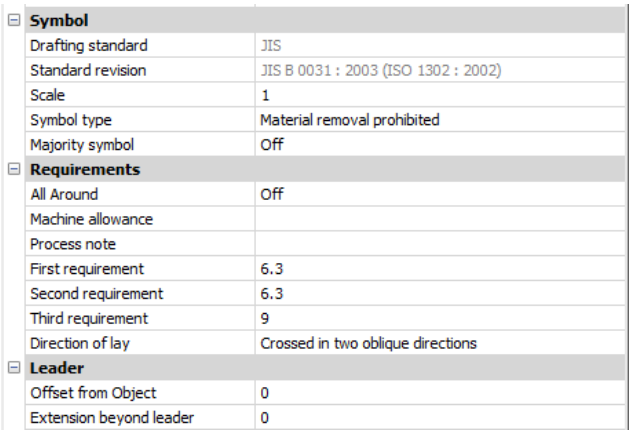
Note: These symbols are compatible with the legacy AutoCAD Mechanical application.

Note: The symbol will be added to the AM_5 layer.

Note: After opening a drawing that contains mechanical entities, filling other drawings with mechanical-related data will be done on-demand in contrast to previous versions. It will be possible when a user

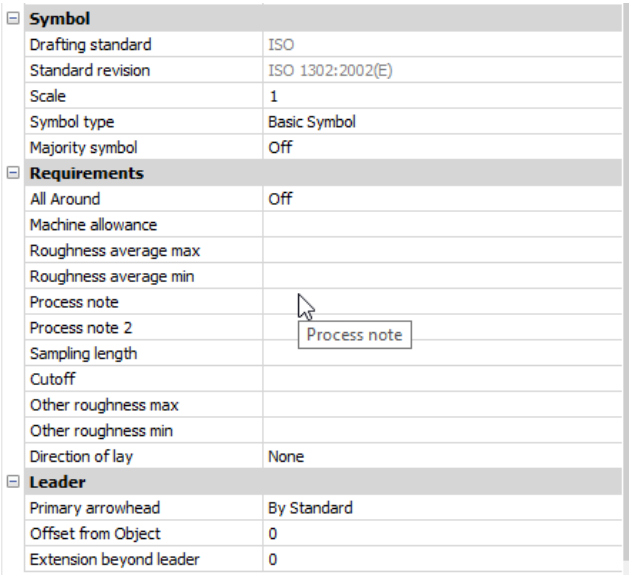
copies the mechanical-related entities to the vanilla drawing. In the case of copying, entities that are not related to the mechanical data, a vanilla drawing will not be filled with mechanical data.

After creating the symbol prototype with the AMSURFSYM command, the user can configure the exact surface symbol's properties in the **Properties** panel:



Symbol	
Drafting standard	JIS
Standard revision	JIS B 0031 : 2003 (ISO 1302 : 2002)
Scale	1
Symbol type	Material removal prohibited
Majority symbol	Off
Requirements	
All Around	Off
Machine allowance	
Process note	
First requirement	6.3
Second requirement	6.3
Third requirement	9
Direction of lay	Crossed in two oblique directions
Leader	
Offset from Object	0
Extension beyond leader	0

Note: The surface symbol properties depend on the used standard.



Symbol	
Drafting standard	ISO
Standard revision	ISO 1302:2002(E)
Scale	1
Symbol type	Basic Symbol
Majority symbol	Off
Requirements	
All Around	Off
Machine allowance	
Roughness average max	
Roughness average min	
Process note	
Process note 2	
Sampling length	
Cutoff	
Other roughness max	
Other roughness min	
Direction of lay	None
Leader	
Primary arrowhead	By Standard
Offset from Object	0
Extension beyond leader	0

6.37.2 Symbol

Symbol type
Choose one of the:

- Basic symbol ✓
- Material removal required ✓
- Material removal prohibited ✓

Majority symbol
Indicates the common state to all surfaces using a single collective indication symbol.

OFF: shows only the collective indication.
ON: shows the state of other surface texture symbols elsewhere in the drawing in parenthesis at the right side of the collective indication.

6.37.3 Requirements

All Around
Toggles the visibility of the All Around surface texture in the symbol.

Machine allowance
Specifies the machining allowance for the surface.

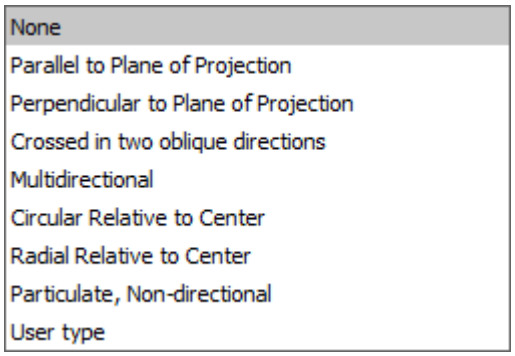
Process note
Defines the process requirements for the surface.

First requirement
Defines the first requirement for the surface.

Second requirement
Defines the second requirement for the surface.

Third requirement
Defines the third requirement for the surface.

Direction of lay
Sets the direction of lay for the surface.



Roughness average max
Defines the maximum roughness average for the surface.

Roughness average min
Defines the minimum roughness average for the surface.

Process note2
Defines the process requirements for the surface.

Sampling length
Defines the sampling length required for the surface.

Other roughness max
Defines the maximum other roughness for the surface.

Other roughness min
Defines the minimum other roughness for the surface.

6.37.4 Leader

Offset from Object
Specifies the distance between the extension line's start point and the attached object.

Extension beyond leader
Specifies the distance between the symbol's start point and the surface extension line's end point.

6.38 AMWELDSYM command

Creates a welding symbols.

Shape Lite Pro Mechanical BIM

Icon: 

6.38.1 Methods

Select the object to attach the welding symbol and specify the points for its location.

There are two use cases:

1 **When creating a new drawing that will contain mechanical entities:**
   a Set the LOADMECHANICAL2D system variable to ON (1).
   b Start a new drawing using a Mechanical2d template.

2 **When opening a drawing that contains mechanical entities**
   a Set the LOADMECHANICAL2D system variable to ON (1).
   b Open an existent ACM drawing and you can start special symbols creation.

Note: These symbols are compatible with the legacy AutoCAD Mechanical application.

Note: The symbol will be added to the AM_5 layer.

Note: After opening a drawing that contains mechanical entities, filling other drawings with mechanical-related data will be done on-demand in contrast to previous versions. It will be possible when a user copies the mechanical-related entities to the vanilla drawing. In the case of copying, entities that are not related to the mechanical data, a vanilla drawing will not be filled with mechanical data.

After creating the symbol prototype with the AMWELDSYM command, you can configure the exact welding symbol's properties from the **Properties** panel:

Symbol	
Drafting standard	ANSI
Standard revision	ANSI/AWS A2.4 - 98
Scale	1
All Around	Off
Field weld	Off
Note tail	Off
Process notes	
Stagger	None
Flip symbol	Right aligned
Requirements - Arrow Side	
Weld type	Fillet Weld
Small leg	x
Size	
Length	
Pitch	-
Contour	None
Requirements - Other Side	
Weld type	None
Contour	Concave Contour
Leader	
Arrowhead	By Standard

6.38.2 Symbol

All Around
Toggles the visibility of the All Around symbol in the welding symbol.

Field weld
Toggle the visibility of the Field weld symbol in the welding symbol.

Note tail
Displays the state of the note tail.

Process notes
Defines the process notes.

Stagger
Chooses between move and mirrored stagger for intermittent welds on both sides.

Flip symbol
Chooses between left and right aligned for weld symbol.

6.38.3 Requirements - Arrow side

Weld type
Defines the weld type from the below options:



Small leg
Defines the small leg dimension of the weld.

Size
Defines the size of the weld.

Length
Defines the length of the weld.

Pitch
Defines the pitch dimension for intermittent weld.

Contour
Defines the shape of the weld.

6.38.4 Requirements - other side

Defines the type and dimensions of the weld on the other side.

6.38.5 Leader

Arrowhead
Defines the default arrowhead type for welding symbols.

6.39 ANIMATIONEDITORCLOSE command

Closes the Animation editor panel.

Shape Lite Pro Mechanical BIM

6.39.1 Description

Closes the Animation editor panel to hide it from the current workspace. If the Animation editor panel is stacked when you close it, the Animation editor tab or icon is removed from the stack.

6.40 ANIMATIONEDITOROPEN command

Opens the Animation editor panel.

Shape Lite Pro Mechanical BIM

Icon:

6.40.1 Description

Opens the Animation editor panel to display it in the current workspace. The Animation editor panel appears at the same size and on the same location as it did before it was closed or collapsed. Like any other dockable panel, the Animation editor panel can be either floating, docked or stacked.

6.41 ANIMATIONEDITORPLAYBACK command

Loads animation graphics.

Shape Lite Pro Mechanical BIM

6.41.1 Description

Loads animation graphics data from a zip-archive or a folder.

Note: This command is only executed if the **Animation Editor** panel is active. After the graphics are loaded, **Animation Editor** is switched to a special animation graphics playback mode.

6.42 ANIMATIONEDITORRECORD command

Records animation graphics.

Shape Lite Pro Mechanical BIM

6.42.1 Description

Records animation graphics and saves it to a zip-archive or to a folder of your choice. After selecting the needed save option, the recording process will start. Depending on the complexity of the model, this can take a long time.

6.42.2 Options within the command

Zip-file

Opens the **Select a zip-file name to save animation graphics** dialog box that allows you to save the animation graphics to a zip-archive.

Folder

Opens the **Select a folder to save animation graphics** dialog box that allows you to save the animation graphics to a folder of your choice.

6.43 ANIPATH command

Records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.

Shape Lite Pro Mechanical BIM

Icon:

6.43.1 Description

Displays the Motion Path Animation dialog box.

6.44 ANNORESET command

Resets moved annotative scale representations.

Shape Lite Pro Mechanical BIM

Icon: 

6.44.1 Description

The locations of scale representations linked with annotative entities can be moved with grips. This command returns selected scale representations to their default positions.

6.45 ANNOUPDATE command

Updates the selected annotative entities to match their current annotative style.

Shape Lite Pro Mechanical BIM

Icon: 

6.45.1 Method

Select one or more annotative entities, such as text, dimensions, hatches, blocks to update them to their current annotative styles. If all the entities in the drawing are selected, the program will ignore non-annotative entities.

6.46 APERTURE command

6.47 APPARENT command

Toggles the Apparent intersection entity snap.

Shape Lite Pro Mechanical BIM

Alias: PLANVIEWINT

6.47.1 Description

Toggles the Apparent intersection entity snap to enable or disable snapping to apparent intersection. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

6.48 APPLOAD command

Opens the Load application files dialog box.

Shape Lite Pro Mechanical BIM

Icon:

6.48.1 Description

Opens the Load application files dialog box to load application files to run inside BricsCAD.

6.49 ARC command

Creates an arc.

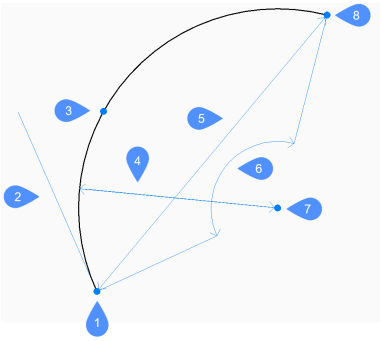
Shape Lite Pro Mechanical BIM

Icon:

Alias: A

6.49.1 Description

Creates an arc from a combination of options including the start point, second point, end point, center, included angle, direction, and length of chord.



1 Start
2 Direction
3 Second
4 Radius
5 Length of cord
6 Angle
7 Center
8 End

6.49.2 Methods to create an arc

This command has 3 methods to begin creating an arc:

- Start of arc
- Center
- Follow last point

You can press the Ctrl key while drawing an arc to change its direction from counterclockwise to clockwise.

Start of arc
Begin creating an arc by specifying a start point then:

Second point
Specify a point along the arc's circumference.
Additional options: [Angle/Center/Direction/End/Radius]

End
Specify the end point of the arc.

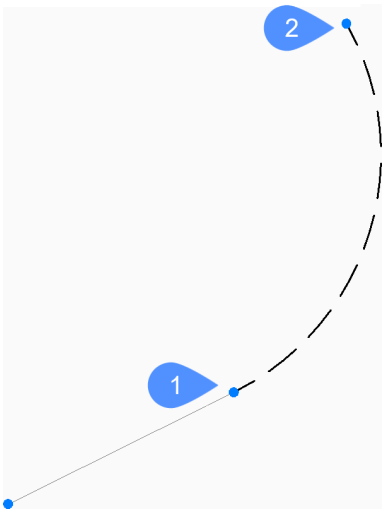
Center
Begin creating an arc by specifying a center point then:

Start
Specify the start point of the arc.

End
Specify the end point of the arc.
Additional options: [Angle/Length of chord]

Follow last point
Begin creating an arc from the last drawn arc or line segment, following its angle, then:

End
Specify the end point of the arc. The arc is drawn tangent to the previous segment.



1 Last point

2 End of arc

6.49.3 Additional options within the ARC command

After you begin creating an arc, the following options may be available:

Angle
Specify the included angle of the arc.

Direction
Specify the direction of the arc.

Radius
Specify the radius of the arc.

6.50 AREA command

Finds the area and perimeter of 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: AA

6.50.1 Method

Specify the points that define the area for which BricsCAD will report the area and perimeter.

6.50.2 Options

find area of one Entity
Provides the area and/or perimeter of the selected entity.

Note: The information reported by this command varies according to the entity selected:

- Line and sketch - length
- Arc and elliptical arc – area and length
- Circle and ellipse - area and circumference/perimeter
- Closed polyline and closed spline - area and perimeter.
- Open polyline and spline - area and length.
- 3D objects - surface area

Add areas together
Allows to add up the areas of two or more entities.

Subtract areas
Allows to remove areas of entities from the total area and perimeter.

Note: This option can only be used after you have found the area of at least one entity.

6.51 ARRANGE command

Spatially organizes a set of entities by aligning and/or distributing them along the axes of the current UCS.

Shape Lite Pro Mechanical BIM

6.51.1 Methods

The ARRANGE command spatially organizes a set of entities (2D and/or 3D) by aligning and/or distributing them along the axes of the current UCS.

This command creates a new arrangement of the selected entities. The arrangement of the entities is defined based on the bounding boxes of the selected entities (i.e., the smallest box which contains the

entity), with respect to the selection bounding box (i.e., the smallest box which contains all selected entities).

To make a new arrangement with the selected entities, the ARRANGE command uses the following elements:

- A selection of entities (these can be both 2D and 3D).
- A direction for the arrangement. This can be a main axis (X, Y, or Z) of the current UCS.
- The bounding box of the selection.
- The bounding boxes of each entity in the selection.
- The arrangement option selected by the user.

The workflow of the ARRANGE command includes the following steps:

1 Launch the ARRANGE command.

2 Select the entities to arrange. The entities can be both 2D and 3D.

3 Choose a direction.

Choose the UCS axis (X, Y, or Z) along which you want to arrange the entities. Entities will only be moved along this direction.

1 You can perform as many arrange operation as you want on the original set of entities with a single run of the command.

2 Choose one of the 8 arrangement options of the command (see below).

3 Return to step 3.

4 (Optionally) **Undo** the arrangements step by step.

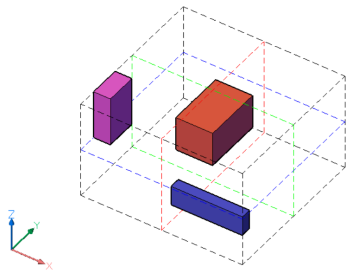
Note: The UNDO option is available after an alignment option is applied.

5 Select **Finish** or press **Enter** to finalize the command.

The arrangement of the selected entities is done with the following approach:

To create a new configuration of the selected entities, the ARRANGE command uses some bounds. First, there are the main bounds, that are attached to the selection bounding box. Then, there are the secondary bounds, that are attached to the bounding boxes of each entity in the selection.

This is a simple example of a selection bounding box and three other bounding boxes for the entities in the selection:



The selection bounding box is represented by black dashed lines. The bounds of the selection bounding box are defined by its six faces, two for each axis of the current UCS. There are also three center bounds, one for each axis, located in the center of the selection bounding box. The center bounds are represented by colored dashed lines, red for X axis, green for Y axis and blue for Z axis.

The colored boxes are the bounding boxes of the selected entities. For each entity in the selection, there are six outside bounds, defined by the faces, and three center bounds. The bounds of an entity are defined similarly to the ones of the selection bounding box.

In summary:

- The selection of entities has a total of nine bounds. There are three bounds for each of the axes of the UCS. The notation of these bounds are **1** (Center), **2** (Lower), and **3** (Upper), for each of the axes.
- Also, each entity has nine bounds, similar to the ones of the selection.

For each axis, the bounds are defined as follows:

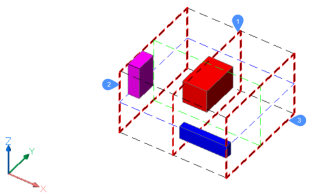
2 - the Lower bound is defined by the face of the bounding box placed at the smallest coordinate along the specified axis.

3 - the Upper bound is defined by the face of the bounding box placed at the largest coordinate along the specified axis.

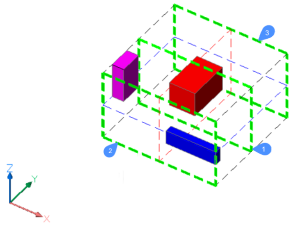
1 - the Center bound is located between the **2** (Lower) and **3** (Upper) bounds at the same distance from each of them.

In the pictures below, the bounds of the selection bounding box are highlighted with thick dashed lines.

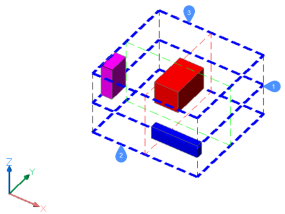
For the X axis:



For the Y axis:



For the Z axis:

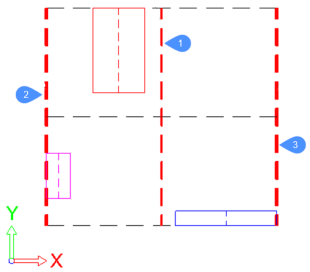


To define the new position of an entity in the selection, the ARRANGE command uses the bounds of the selection, the bounds of the entity, and the method selected by the user.

6.51.2 Options within the command

In this example, the arrangement will be made by using **X** as direction.

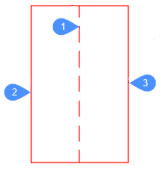
Here is a simplified 2D view, in XY plane, of the situation described above:



The main bounds, attached to the selection bounding box, are represented by thick dashed red lines.

For each entity in the selection, its bounding box is represented by continuous lines, each with a different color. In this case, for each entity, the bounds, **2 (Lower)**, **1 (Center)** and **3 (Upper)**, will be the left side, center line and right side of its bounding rectangle, respectively.

For the red entity, the bounds will be like this:



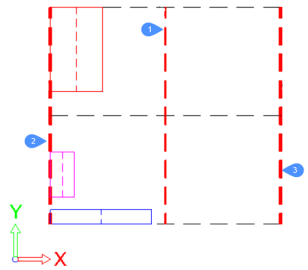
For the other entities, the bounds are defined in a similar way.

None

The entities will not be arranged along the specified direction. If an arrangement was already set along the specified direction during the execution of the command, the entity will be moved to its original position on the specified axis.

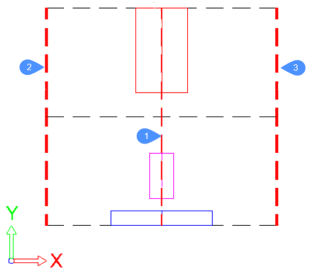
Lower Align

The lower bound of each entity will be aligned with the lower bound of the selection bounding box in the specified direction.



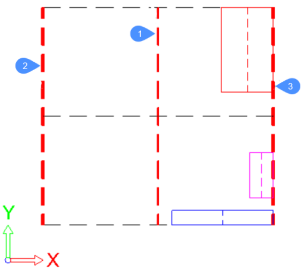
Center Align

The center bound of each entity will be aligned with the center bound of the selection bounding box in the specified direction.



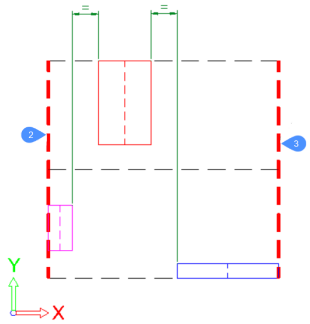
Upper Align

The lower bound of each entity will be aligned with the lower bound of the selection bounding box in the specified direction.



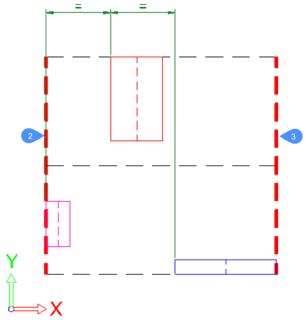
Equally spaced

The gaps between the entities will have the same size along the specified direction.



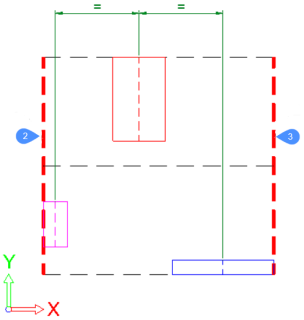
Lower Equally spaced

The distance between the lower bounds of consecutive entities will be the same along the specified direction.



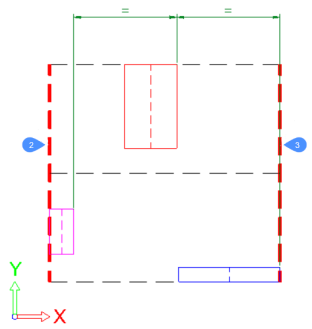
Center Equally spaced

The distance between the center bounds of consecutive entities will be the same along the specified direction.



Upper Equally spaced

The distance between the lower bounds of consecutive entities will be the same along the specified direction.



6.52 ARRAY command

Creates an array of entities.

Shape Lite Pro Mechanical BIM

Icon:

Alias: AR

6.52.1 Description

Creates a polar or rectangular array, or along a path ofentitiesthrough the command line.

Note: Arrays can be created using 2D or 3D entities.

6.52.2 Methods

There are three methods to create an array of entities:

- Rectangular
- Path
- Polar

Note: The default array type is saved by the ARRAYTYPE variable.

6.52.3 Options

Rectangular
Distributes entity copies into any number of rows, columns and levels (on Z direction).

PAth
Distributes entity copies evenly along a path into multiple rows and levels.

POlar
Distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows and levels.

6.53 -ARRAY command

Creates an array of entities.

Shape Lite Pro Mechanical BIM

6.53.1 Description

Creates a static polar or rectangular array of entities through the Command line.

Note: Arrays can be created using 2D or 3D entities.

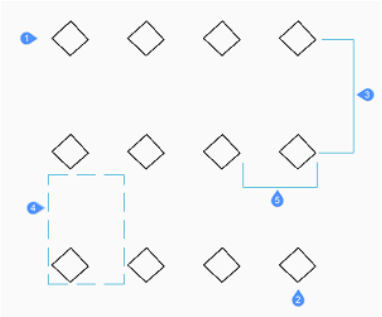
6.53.2 Methods

There are two methods to create an array of entities:

- Polar
- Rectangular

6.53.3 Options

Rectangular
Creates linear, rectangular, or square arrays.



Number of rows in the array
Specifies the number of rows. (1)

Note: Enter 1 for a linear array and a negative number to draw the array downwards.

Number of columns
Specifies the number of columns. (2)

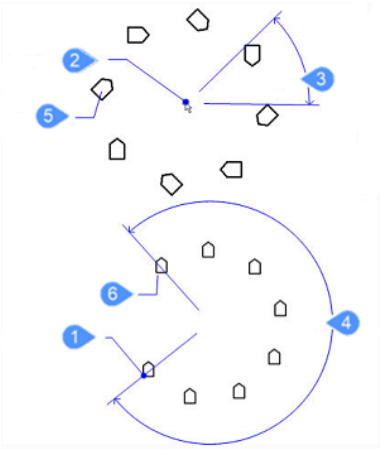
Note: Enter a number value to draw the array to the left.

Vertical distance between rows of entities
Specifies the distance between rows of entities. (3)

Spacing rectangle
Specifies two points that indicate the spacing between rows and columns. (4)

Horizontal distance between columns
Specifies the distance between columns of entities. (5)

Polar
creates polar (circular) arrays.



Base
Relocates the base point of the selected entities. (1)

Center of polar array
Specifies the center point. (2)

Number of items to array
Specifies the number of items in the array.

Note: Enter a number larger than 1.

enter to specify Angle between items
Specifies the angle between each copy. (3)

Angle to fill
Determines the extent and direction of the array. (4)

Note: Enter 360 for a full circle of copies or a smaller number to draw a partial polar array. Enter a positive angle to draw the array counterclockwise or a negative number to draw the array clockwise.

Rotate entities around the array?
Determines whether copies are rotated when they are arrayed.
Yes (5)
No (6)

6.54 ARRAYCLASSIC command

Opens the Array dialog box.

Shape Lite Pro Mechanical BIM

6.54.1 Description

Opens the Array dialog box to create a non-associative 2D rectangular or polar array.

6.55 ARRAYCLOSE command

Exits the associative array editing state.

Shape Lite Pro Mechanical BIM

Icon: 

6.55.1 Description

Exits the associative array editing state, displaying the Array Close dialog box to save or discard the changes made to the array's source entities.

Note: This command is available only after an associative array is checked out with the ARRAYEDIT command, Source option.

6.55.2 Options

Yes
Saves the changes made to the source entities of the array and then exits the array editing state.

No
Exits the array editing state and discards all changes, thus returning the array to its original state.

Cancel
Closes the dialog box and stays in the array editing state.

6.56 -ARRAYCLOSE command

Exits the associative array editing state.

Shape Lite Pro Mechanical BIM

6.56.1 Description

Exits the associative array editing state, prompting at the Command line to save or discard the changes made to the array's source entities.

Note: This command is available only after an associative array is checked out with ARRAYEDIT command, Source option.

6.56.2 Options

Yes
Saves the changes made to the source entities of the array and then exits the array editing state.

No
Exits the array editing state and discards all changes, thus returning the array to its original state.

6.57 ARRAYEDIT command

Edits associative arrays.

Shape Lite Pro Mechanical BIM

Icon: 

6.57.1 Method

Select the entities that make up associative arrays.

6.57.2 Options

Source
Edits the source entities of the array.

Note: Displays Array Editing State dialog box. Only the selected entity is displayed in Associative Array Edit environment so it can be easy to modify, add and delete entities. The changes are applied to all the entities once the array editing state is closed.

REPlace
Replace some or all the entities in the array.

RESet
Restores erased entities and removes any item overrides.

Note: Holt the Ctrl key to select more than one item in the associative array, then tap the Delete key to erase the selection.

eXit
Accept the changes and conclude the command.

6.58 ARRAYEDITEXT command

Provides additional editing of associative arrays.

Shape Lite Pro Mechanical BIM

Icon: 

6.58.1 Description

Provides additional editing for all elements of an associative array or for an individual element in the associative array.

Note: Arrays edited by this command must have been made associative by the ARRAY command. It does not work with arrays created by the ARRAYCLASSIC command or -ARRAY command.

6.58.2 Method

There are two methods when editing an associative array using ARRAYEDITEXT command:

- Edit all elements of the array: interactively adjusts the spacing between them and changes the number of elements.
- Edit an individual item of the array: moves (offset), scales, rotates or deletes it.

6.58.3 Options

SPacing
Adjusts the spacing of an entire associative array interactively.

REsizing
Resizes the entire array by removing entities interactively.

Offset items
Moves the entities of an associative array to a different location.

Note: Pick points or enter x, y coordinates.

Scale item
Resize an entity of an associative array.

Note: A scale factor larger than 1 makes the entity larger and smaller than 1 makes is smaller.

Rotate item
Rotates an entity of an associative array.

Note: : Positive numbers, for the rotation angle, rotate counter clockwise and negative numbers rotate clockwise.

Delete item
Deletes an entity of an associative array.

6.59 ARRAYPATH command

Creates an array along a path.

Shape Lite Pro Mechanical BIM

Icon:

6.59.1 Description

Associatively distributes entity copies evenly along a path into multiple rows and levels.

6.59.2 Options

ASsociative
Determines whether an array entity is created or a series of copies of the selected entities.

Note: The default setting is saved by the ARRAYASSOCIATIVITY system variable.

Note: Parameters for associative arrays can be specified with parametric expressions set in the Array properties, found in Mechanical browser, Properties panel or through the ARRAYEDIT command.

Method
Specifies how to distribute entities along the path.

Divide
Distributes a specific number of items evenly spaced along the path.

Note: Distributes a specific number of items evenly spaced along the path.

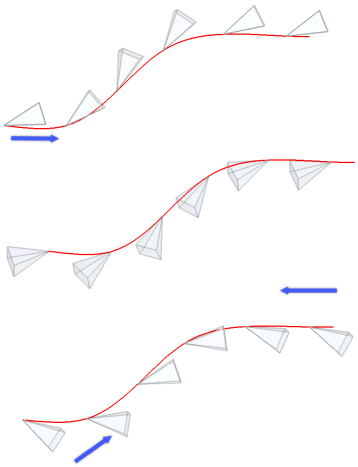
Measure
Places a sufficient number of items along the path, separated by the specified interval.

Base point
Sets the base point of the array. Items are placed relative to the base point.

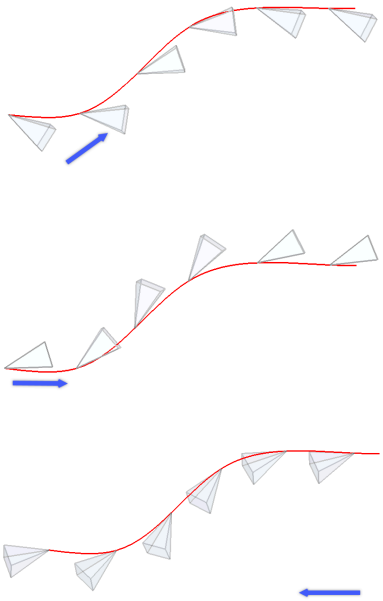
Tangent direction
Determines how the items are aligned relative to the starting direction of the path.

2 Points

Specify two points that define the tangency of the first item relative to the starting direction of the path. Depending on the Align Items setting, the other items are placed parallel to the first item or at the specified tangent direction relative to the direction of the path. In the image **Tangent direction - 2 points**, the blue arrow indicated the tangency direction.



Tangent direction - 2 points



Normal

Orients the Z direction of the item with the starting direction of the path.

Items

Specifies the number of items or the distance between items, depending on the Method setting.

Enter number of items along path

Places items along the entire path at the specified distance.

Note: Placing items along a portion of the path is possible, by specifying a smaller number.

Enter the distance between items along path
Specifies the distance between items by entering a value or picking two points.

Expression
A mathematical formula or equation can be used to derive the value.

Fill entire path
Fills the entire path with items at given spacing.

Note: This option operates like grips editing. If the number of items changes, the length of the path should also change. When the path is modified using grips editing, the array entity is recreated along the modifies path.

Rows
Specifies the number of rows in the array, the distance between them and the incremental elevation.

Distance between rows
Specify the distance between subsequent rows by entering a value or picking two point.

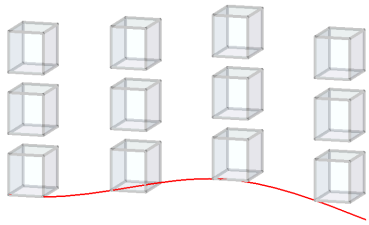
Total
Specify the distance between the first and the last row.

Incrementing elevation between rows
Defines the increasing or decreasing elevation for each subsequent row.

Levels
Creates 3D arrays by specifying the number and spacing of levels.

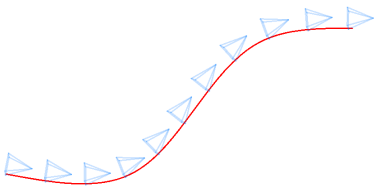
Distance
Specifies the distance between the bottom and the top level.

Total
Specifies the distance between the bottom and the top level.

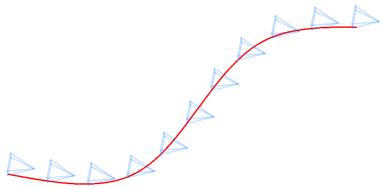


Align items
Specifies whether to align each item tangent to the path's direction. Alignment is relative to the first item's orientation.

Yes
Each item is placed tangent to the path direction.

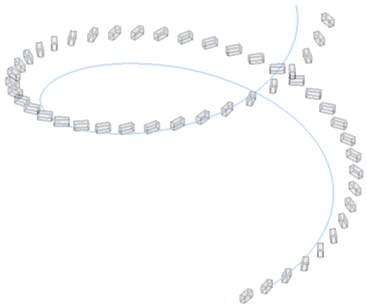


No
Each item keeps the orientation of the first item.

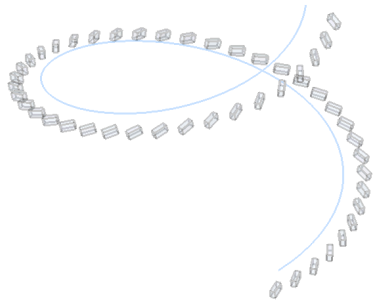


Z direction
Determines whether to maintain the items' original Z direction or to bank the items naturally along a 3D path.

Yes
Z direction (vertical) is maintained.



No
Items banked along the helix.



eXit
Accept the current settings and conclude the command.

6.60 ARRAYPOLAR command

Creates an array of entities, in a circular pattern.

Shape Lite Pro Mechanical BIM

Icon:

6.60.1 Description

Associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows and levels.

6.60.2 Options

ASsociative
Determines whether an array entity is created or a series of copies of the selected entities.

Note: The default setting is saved by the ARRAYASSOCIATIVITY system variable.

Note: Parameters for associative arrays can be specified with parametric expressions set in the Array properties, found in Mechanical browser, Properties panel or through the ARRAYEDIT command.

Base point
Sets the base point of the array. Items are placed relative to the base point.

Items
Specifies the number of items in the array.

Angle between
Specifies the angle between two subsequent items.

Expression
A mathematical formula or equation can be used to derive the value.

Fill angle
Specifies the angle between the first and last item.

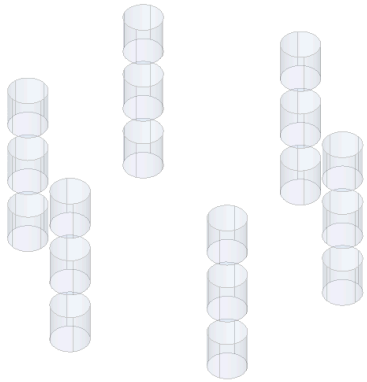
ROWs
Specifies the number of rows, the spacing between subsequent rows and the incremental elevation.

Distance
Specifies the distance between subsequent rows.

Total
Specifies the distance between the first and the last row.

Incremental elevation between rows
Specifies the increasing or decreasing elevation for each subsequent row.

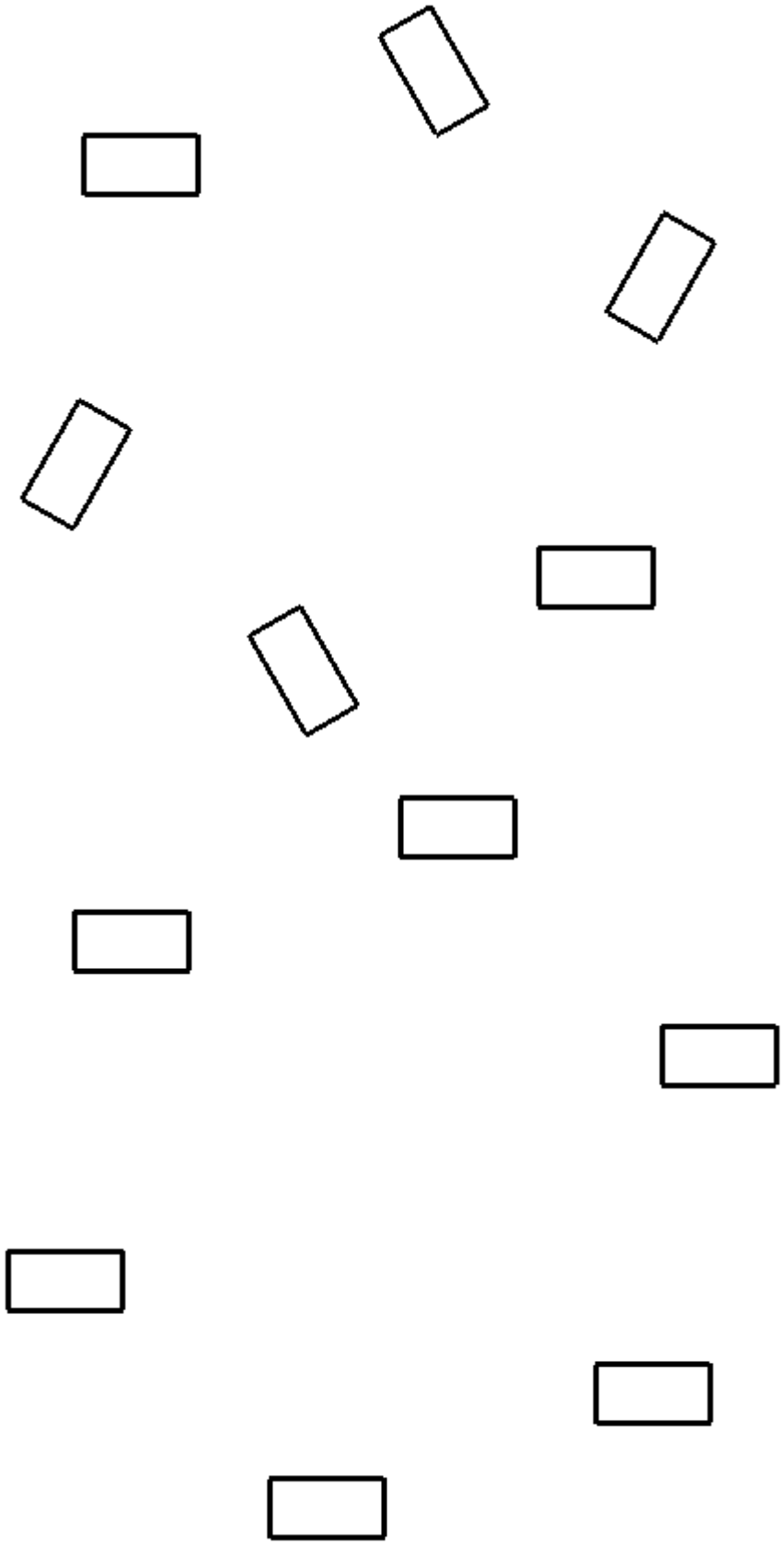
Levels
Creates 3D arrays by specifying the number and spacing of levels.



Distance
Specifies the distance between subsequent levels by typing a value or picking two points.

Total
Specifies the distance between the bottom and the top level.

ROTate
Determines whether items are rotated around the circular path or they preserve the orientation of the source entities.



eXit
Accept the current settings and conclude the command.

6.61 ARRAYRECT command

Creates a rectangular array of entities.

Shape Lite Pro Mechanical BIM

Icon:

6.61.1 Description

Associatively distributes copies of entities into any number of rows, columns and levels.

6.61.2 Options

ASsociative
Determines whether an array entity is created or a series of copies of the selected entities.

Note: The default setting is saved by the ARRAYASSOCIATIVITY system variable.

Note: Parameters for associative arrays can be specified with parametric expressions set in the Array properties, found in Mechanical browser, Properties panel or through the ARRAYEDIT command.

Base point
Sets the base point of the array. Items are placed relative to the base point.

COUnt
Specifies the number of rows and columns.

Spacing
Specifies the distance between columns and/or rows.

Unit cell
A rectangle is displayed dynamically. Specify a point or enter the desired column and row spacing in the dynamic entry fields. To switch back and forth between the entry fields, press Tab key.

Note: If dynamic input is on, then the width and height of the spacing rectangle are displayed.

COLumns
Specifies the number of columns and the spacing between subsequent columns.

Expression
A mathematical formula or equation can be used to derive the value.

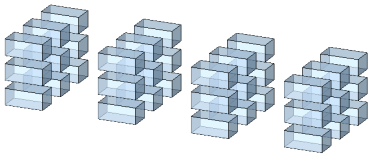
Rows
Specifies the number of rows, the spacing between subsequent rows and the incremental elevation.

Distance
Specifies the distance between subsequent columns/rows.

Total
Specifies the distance between the first and the last column/row.

Incrementing elevation between rows
Specifies the increasing or decreasing elevation for each subsequent row.

Levels
Creates 3D arrays by specifying the number and spacing of levels.



Distance
Specifies the distance between subsequent levels by typing a value or picking two points.

Total
Specifies the distance between the bottom and the top level.

eXit
Accept the current settings and conclude the command.

6.62 ATTACHMENTSPANELCLOSE command

Closes the Attachments panel.

Shape Lite Pro Mechanical BIM

6.62.1 Description

Closes the Attachments panel to hide it from the current workspace. If the Attachments panel is stacked when you close it, the Attachments tab or icon is removed from the stack.

6.63 ATTACHMENTSPANELOPEN command

Opens the Attachments panel.

Shape Lite Pro Mechanical BIM

6.63.1 Description

Opens the Attachments panel to display it in the current workspace. The Attachments panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Attachments panel can be either floating, docked or stacked.

6.64 ATTDEF command

Opens the Define attribute dialog box.

Shape Lite Pro Mechanical BIM

Icon:

Alias: AT, DDATTDEF

6.64.1 Description

Opens the Define attribute dialog box to add an attribute to a block definition.

6.65 -ATTDEF command

Defines attribute data.

Shape Lite Pro Mechanical BIM

Alias: -AT

Note: This command is meant for use by macros, scripts, and LISP routines. To specify attributes with a dialog box, use the ATTDEF command.

6.65.1 Description

Defines attribute data (used by blocks) at the Command line.

6.65.2 Options

Define an attribute by toggling attribute modes.

Toggle attribute modes: Constant = Off Invisible = Off Preset = Off Verify = Off Lock position = Off Annotative = Off Multiple Lines = Off

Constant – type **C**

- Off - users can change the value of the attribute.
- On - users cannot change the value.

Invisible – type **I**

- Off - displays the attribute in the drawing.
- On - hides the attribute

Note: Invisible attributes are not displayed nor printed, but can be displayed with the ATTDISP command.

Preset – type **P**

- Off - prompts the user to enter values.
- On - inserts attributes automatically, without prompting the user.

Note: The value of attributes can be changed later with the ATTEDIT command.

Verify -type **V**

- Off - user enters the attribute value once.
- On- forces the user to enter the value a second time to ensure the value is entered correctly.

Lock position – type **L**

- Off - users can move attributes using grip editing.
- On - locks the position of the attribute within the block reference.

Annotative – type **A**

- Off – creates attributes using a non-annotative style.
- On - creates attributes using an annotative style.

Multiple lines - type **M**

- Off - forces the use only of single-line text.

- On - allows the use uses multiline text.

Tag name of attribute
Specifies the tag of the attribute; this is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers and punctuation.

Prompt
Specifies 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 uses the tag as the prompt during attribute insertion.

Default text
Specifies the default value and set the style and justification of the text, as with TEXT command. If the Multiple lines mode is on, you may introduce more text lines.

Text start point
Specify the start point of the text.

user defined Style
Specifies the name of the text style.

Align on line
Aligns the text between two points. The text height is adjusted to maintain its aspect ratio.

Fit between points
Aligns the text between two points.

Center horizontally
Centers the text's baseline on the pick point.

Middle (hor/vert)
Centers the text on the pick point.

Justification options...
Accesses additional justification options.

Height of text
Specifies the height of the attribute text, unless overridden by the text style or justification.

Rotation angle of text
Specifies the rotation angle of the rotation text.

6.66 ATTDISP command

Sets the display mode of attribute text in the drawing.

Shape Lite Pro Mechanical BIM

Icon:

Alias: AD

6.66.1 Description

Toggles the display of attribute text in the drawing, between turn all on, turn all off, or turn on only non-visible ones.

When attributes are off or invisible, they are not displayed in the drawing and are not plotted.

6.66.2 Options within the command

Attribute display mode
Changes the display of attribute values:

- display attribute (**ON**): displays all attributes, including those set to invisible mode by the ATTDEF command.
- hide attributes (**OF**): hides all attributes.
- display **N**ormally: displays attributes, except those set to invisible mode.

6.67 ATTEDIT command

Edits the values and properties of attributes.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: ATE

6.67.1 Description

Edits the values and properties of attributes. This command is meant for editing attributes globally with macros, scripts, and LISP routines. For editing individual attributes, it is easier to use the dialog box of the EATTEDIT command.

6.67.2 Methods

There are two methods:

- **Y**es – edit one at a time: edits attributes one at a time; allows changes to values and properties.
- **N**o – edit multiple: edits attributes globally; allows changes to values only.

6.67.3 Options within the command

Edit attributes of which blocks
Specifies the names of blocks; you can use wildcards to specify similarly named blocks: * refers to all characters in the block's name.

Edit attributes with these names
Specifies the names of attributes. You can use wildcards to specify similarly named tags.

Edit attributes with this text
Specifies the values of attributes. You can use wildcards to specify similarly named values.

Select attributes
Chooses the attributes to edit.

Position
Moves the attribute's insertion point to the new specified point.

Angle
Changes the rotation angle of the attribute text to the specified angle.

Text
Changes the value of the attribute.

- **Change**: searches and replaces text.
- **Replace**: new attribute text replaces the text of the currently selected attribute.

Style
Changes the text style of the attribute; the style name must exist in the drawing. (Use the STYLE command to create or edit text styles.)
Enter a distinctive style name, or press Enter to keep the style name.

Color
Changes the color of the attribute text by entering a different color name or number.

Height
Change the height of the attribute text by entering a different text height, pick two points in the drawing, or else press Enter to keep the height.

Layer
Changes the layer of the attribute; the layer name must exist in the drawing. (Use the Layer command to create layers.)
Enter a different layer name, or press Enter to keep the layer name.

Next
Goes to the next attribute; if there are no more attributes to edit, exits the command.

pRevious
Goes to the previous attribute, if any.

Quit
Exits the command.

Edit only attributes visible on screen
This option appears when you choose to edit multiple attributes at a time. It determines which group of attributes to edit:

- **Yes – visible only**: edits only attributes visible in the current viewport.
- **No – edit all**: edits all attributes in the drawing.

Text to change
Specifies the attribute value to change.

New text
Specifies the replacement value.

6.68 ATTEXT command

Opens the Extract attributes dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: AX, DDATTEXT

6.68.1 Description

Opens the Extract attributes dialog box to export data from block attributes to a text file.

6.69 -ATTEXT command

Copies data from attributes to a text file through the Command line.

Shape Lite Pro Mechanical BIM

Alias: -AX

Note: The template .txt file must exist before this command is used. See the ATTEXT command for the format of this file.

6.69.1 Methods

There are two methods to use –ATTEXT command:

- Select entities: the Attribute Extract Template File dialog box displays.
- Extract as Dxf/Cdf/Sdf: the Attribute Extract Output File dialog box displays.

6.69.2 Options

select Entities

Specifies the entities from which will extract attribute value.

Note: If you select blocks without attributes and non-block entities, BricsCAD ignores them. If you ignore this option by entering DXF, SDF, or CDF directly, then BricsCAD selects all attributes in the drawing.

extract attributes as Dxf

Exports the attribute values in a partial DXF format.

Note: This format is meant for use with other computer programs that can read .dxx files (DXf eXtraction).

extract attributes as Cdf

Exports the attributes with commas separating values.

Note: : This format is meant for use with spreadsheets and databases.

extract attributes as Sdf

Exports the attributes in fixed-width format.

Note: This format is meant for use in word processors.

6.70 ATTREDEF command

Redefines a block and updates associated attributes.

Shape Lite Pro Mechanical BIM

6.70.1 Description

Redefines an existing block reference, and updates any associated attributes, from new or existing entities and attributes available in the drawing.

The following rules apply to existing block references:

- New attributes use default values.
- Values of old attributes that are included in the new block definition are preserved.
- Old attributes that are not included in the new block definition are deleted.
- Format or property changes made with the ATTEDIT or EATTEDIT commands are removed.
- Extended data associated with the block are deleted.

6.71 ATTSYNC command

Synchronizes block attributes.

Shape Lite Pro Mechanical BIM

6.71.1 Description

Synchronizes attribute definitions in all block references of a specified block definition.

6.71.2 Options within the command

Select
Selects the block whose attributes should be used as the template for synchronizing other blocks.

Name
Selects the block definition to be synchronized by name or list of names.

ATTSYNC block?
Synchronizes attributes in the specified blocks.

Yes
Synchronizes attributes in the selected block.

No – skip block
Does not synchronize the block and skips to next block.

6.72 AUDIT command

Analyzes the integrity of the current drawing and fixes errors.

Shape Lite Pro Mechanical BIM

Icon: 

Note: Set the AUDITCTL system variable to 1, to let Audit create an ASCII file describing problems and the action taken. This report, with the file extension .adt, is placed in the same directory as the current drawing.

6.72.1 Description

Checks the current drawing for errors and optionally repairs the drawing. It removes duplicate ACIS attributes.

6.72.2 Options within the command

Yes

Fixes errors, if BricsCAD® finds any.

No

Searches and reports errors in the drawing database but does not repair them.

6.73 AUTOCOMPLETE command

Specifies how the auto-complete function works in the BricsCAD command bar.

Shape Lite Pro Mechanical BIM

6.73.1 Description

This command sets the behavior of the auto-complete function In the BricsCAD command line. This is done by defining how to append and list which commands and system variables are displayed.

Note: There are six settings, which can also be set by right-clicking in the Command line, using the AutoComplete item in the context menu.



6.73.2 Method

There are six settings available when using the AUTOCOMPLETE command:

- Append: automatically appends suggestions as you type.
- List: displays a list of suggestions.
- System Variables: includes system variables in the above suggestions.
- Delay: specifies a delay time before the suggestion list appears.
- ON: switches on the auto-complete mode.
- Off: switches off the auto-complete mode.

6.73.3 Options

Append

Toggles the automatically append suggestion as you type.

List

Toggles weather the list of suggestions is displayed or not.

System Variables

Toggles weather the system variables are included in the list of suggestions or not.

Delay

Specifies the delay time before the list of suggestions displays.

ON

Switches on auto-complete mode.

OFf
Switches off auto-complete mode.

6.73.4 AI-based AutoComplete suggestions

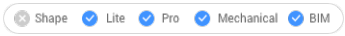
This functionality is an extension to the list of auto-completed commands. This feature adds three more suggestions on top of the ones that are completed in an alphabetical order. These three suggestions are predicted by a Machine Learning model, which is trained based on the commands the user has used in the past.

This feature is available only for the users that choose to be a part of the **Diagnostics & Usage Data Program**.

Note: This feature is set OFF by default. It can be enabled from Settings.

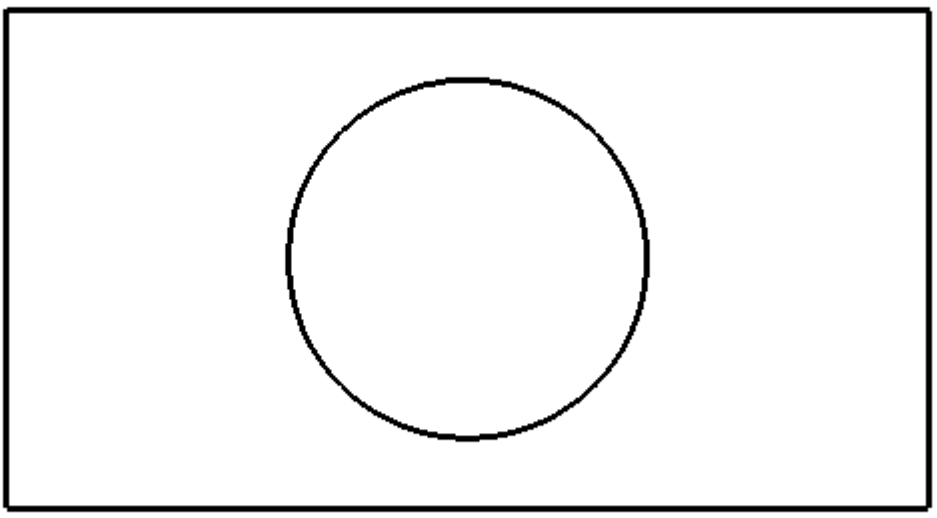
6.74 AUTOCONSTRAIN command

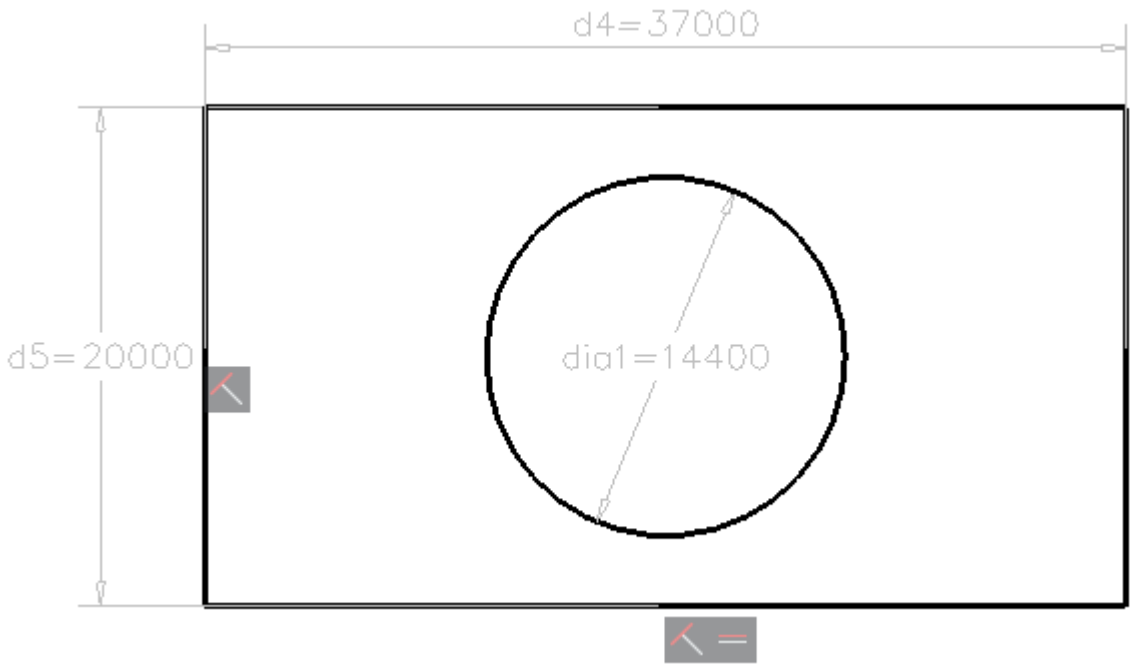
Automatically constrains 2D geometry.



6.74.1 Description

Automatically constrains 2D entities in a drawing, providing both 2D geometric constraints and 2D dimensional constraints.





Note: For Lite license users, constraints can be edited in the Parameters and Constraints panel.

Note: For Pro or higher license users. Constraints can be edited in the Mechanical Browser panel.

7. B

7.1 BACKGROUND command

Opens the Background dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: BACKGROUNDS

7.1.1 Description

Opens the Background dialog box to apply a background to the current viewport.

7.2 BASE command

Changes the drawing's base insertion point.

Shape Lite Pro Mechanical BIM

Icon: 

7.2.1 Description

Changes the drawing's base insertion point for when it is inserted in other drawings, usually as a block or external reference. The base point can be indicated using X, Y, Z coordinates in the current UCS or by picking a point in the drawing.

Alias: BA

7.2.2 Description

Changes the drawing's base insertion point for when it is inserted in other drawings, usually as a block or external reference. The base point can be indicated using X, Y, Z coordinates in the current UCS or by picking a point in the drawing.

7.3 BATTMAN command

Opens the Block attribute manager dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

7.3.1 Description

Opens the Block attribute manager dialog box to view and modify block attribute definitions, properties, and text.

7.4 BCLOSE command

Closes a Block Edit session and saves or discards changes.

Shape Lite Pro Mechanical BIM

7.4.1 Options within the command

Save
Save changes and close Edit Block session.

Discard block editing changes
Discard changes and close Edit Block session.

7.5 BEDIT command

Opens the Create or edit block definition dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: BE

7.5.1 Description

Opens the **Create or Edit Block Definition** dialog box to specify a block to open in the Block Editor.

Note: The HIDEOBJECTS, ISOLATEOBJECTS, and UNISOLATEOBJECTS commands are enabled.

7.6 Команда -BEDIT

Редактирует объекты, образующие блок.

Shape Lite Pro Mechanical BIM

7.6.1 Description (Описание)

Редактирует объекты блока в Редакторе блоков после введения имени блока в командную строку. При необходимости создает новый блок.

7.6.2 Способы редактирования

Существует два способа использования команды -BEDIT:

- Редактирование уже существующего блока.
- Создание нового блока.

7.6.3 Опции команды

Имя блока
Введите имя блока, чтобы открыть Редактора блоков и начать редактирование.

Note: Для создания нового блока введите неиспользуемое имя блока.

?
Отображает список существующих блоков.

7.7 BHATCH command

Opens the Hatch and gradient dialog box.

Shape Lite Pro Mechanical BIM

7.7.1 Description

Opens the Hatch and gradient dialog box to create a hatch entity in the current drawing.

7.8 -BHATCH command

Fills closed 2D areas with repeating patterns or solid colors.

Shape Lite Pro Mechanical BIM

See -HATCH command.

7.9 BIMACTIVATEPYTHON command

7.10 BIMADDECCENTRICITY command

Controls relative positions of the axis of linear solids.

Shape Lite Pro Mechanical BIM

Icon: 

7.10.1 Description

Adds eccentricity to linear solids with respect to their axis.

7.10.2 Method

Select one or multiple linear solids and choose a base location. You can use the widget or type a location in the Command line.

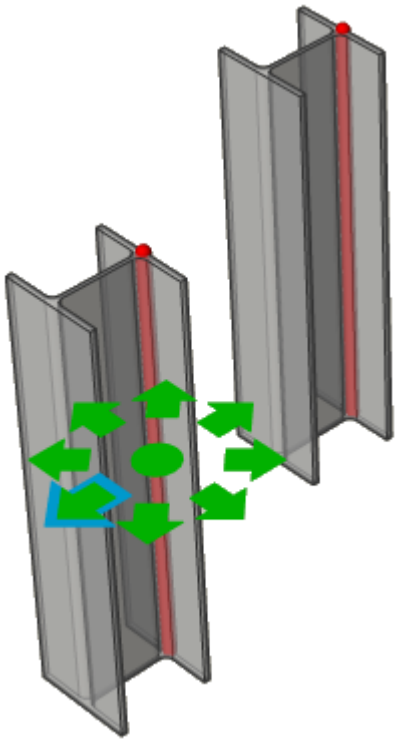
Optionally, you can insert an additional offset vector.

Note: BIMADDECCENTRICITY command only works with structural elements.

7.10.3 Options within the BIMADDECCENTRICITY command

Choose base location

- Click one of the nine locations of the widget.



- Type a location in the Command line. The options are: **TR:** Top Right, **TM**: Top Middle, **TL**: Top Left, **MR**: Middle Right, **ML**: Middle Left, **BR**: Bottom Right, **BM**: Bottom Middle, **BL**: Bottom left, **C**: Center (resets the axes of the selection set to the default location)

Extra offset

Allows to move axis to any point.

- **Reset**: resets extra offset.
- **To centroid**: moves axis to the geometrical midpoint of the profile.
- **Eccentricity**: allows to set axis to one of the nine predefined positions.

7.11 BIMALIGNSECTIONBLOCKS command

Aligns section blocks in model space.

Shape Lite Pro Mechanical BIM

7.11.1 Description

This command realigns section results based on *SECTIONRESULTINTERVAL* system variable.

7.11.2 Method

Aligns BIM section blocks in the model space of generated section blocks drawing.

Note: *BIMSECTIONUPDATE* identifies if overlap between generated section blocks happened in model space and suggests calling *BIMALIGNSECTIONBLOCKS* in this case.

7.12 BIMANALYTICALMODEL command

Creates an analytical model from a fully classified BIM model.

Shape Lite Pro Mechanical BIM

7.12.1 Description

Allows you to generate an analytical model from a full 3D model of a structure.

7.12.2 Method

The command opens the **Allowed Deviations From Center Axis** dialog box and starts with an automatically generated proposal but allows users 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.

Note: The generated proposal can be modified afterward by using the options inside the **Allowed Deviations From Center Axis** Dialog Box.

7.12.3 Options within the command

Visual settings
Opens the **Visual settings** dialog box. Allows to tick and untick different features of axes and nodes to highlight (in yellow) the axes and nodes that match these features. Also, the size of the disks representing the nodes can be changed from here.

Recalculate
Propagates the effect of a local change further, allowing modifications in the complete model. It will launch a global solution that again tries to minimize rigid links, respecting the changes made.

Export and exit
Exports the model to an IFC (*.ifc) or CIS/2 (*.stp) file .

7.12.4 Options in a node

Select connected axes
Selects the axes connected to the node.

Remove
Removes the node and the connected axes will be disconnected.

Connect to axes
Connects the node to a chosen axis (axes).

Disconnect from axes
Disconnects the node from a chosen axis (axes).

Auto split
Automatically splits the node into two nodes.

Split along axis
Manually splits the node along an axis.

Merge
Merges two or more selected nodes together.

7.12.5 Options in an axis

Select connected nodes
Selects the nodes connected to the axis.

Remove
Removes the axis.

Connect to node
Connects the axis to a chosen node.

Disconnect from node
Disconnects the axis from a chosen node.

Add restrictions
Adds a restriction on the axis.

The restriction you can add depends on the restrictions that are already imposed.

If there are no restrictions yet, you can add the following restrictions:

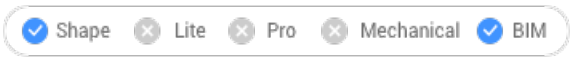
- Parallel to the centerline.
- Inside profile bounds.
- To closest midplane.
- To closest base position.
- To center.
- No length extension.

Relax restrictions
Relaxes a restriction on the axis.
The restriction you can relax depends on the restrictions that are imposed.
Mostly these will be the restrictions that you can relax:

- Inside profile bounds.
- Outside profile bounds.
- Non-parallel.
- Allow length extension.

7.13 BIMAPPLYPROFILE command

Applies a profile to linear entities and linear solids.



Icon: 

7.13.1 Description

Allows you to add profiles, to detach current profile and to convert solids to line.

Note: Linear entities accepted: lines, polylines, arcs, circles, elliptical arcs, ellipses and helixes. Open and closed splines are accepted only if they have default tangents that are not self-intersecting.

7.13.2 Methods

There are two methods to apply profiles:

- Choose a profile from the library.
- Choose a profile entity in the drawing.

7.13.3 Options within the BIMAPPLYPROFILE command

Select path
Select linear entities or linear solids.

Select profile
Select a profile entity in the drawing (a closed 2D entity, a region or an existing linear solid) or press Enter to display the Profiles dialog box.

Apply profile?

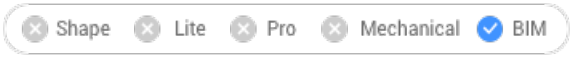
- **Quarter turn**: rotates the profile 90° counterclockwise.
- **Rotate**: rotates the profile at a user-defined angle. Positive values rotate the profile counterclockwise.
- **Delete clippings**: removes all clippings from the selected linear solid(s).

If you select as path linear solids, you have three options:

- **Library**: change the profile.
- **Convert solids to line**: converts linear solids to their axis line.
- **Detach current profile(s)**: detaches profiles only if the entities have an attached profile.

7.14 BIMATTACHCOMPOSITION command

Attaches a BIM composition to solids.



Icon: 

7.14.1 Method/s

This command offers 3 methods to attach the compositions.

- Enter the composition name
- Compositions dialog box
- Select an entity

7.14.2 Options within the command

Dialog
Opens the Compositions dialog box.
See BLCompositions for more information about the dialog.

Entity
Applies the composition of an existing solid. First you select the entities to copy the composition from, next you select one or multiple entities to attach the composition to.

deTach
Allows to detach a composition, while keeping the BIM classification, as different from the Unclassify option of the BIMCLASSIFY command, which removes all BIM data.

7.15 BIMATTACHSPATIALLOCATION command

Allows to attach or remove a spatial location to a selection of entities.

Shape Lite Pro Mechanical BIM

Icon: 

7.15.1 Method

Run the command and choose one of the location numbers or choose to attach or unattach locations.

7.15.2 Options within the command

Auto attach locations
Allows you to automatically assign a spatial location. If necessary, new building(s) and stories are created in the Building and Story Manager dialog box.

Unattach current location
Allows to remove the current spatial location from a selection set.

7.16 BIMAUTOMATCH command

Matches the composition, properties or parameters of example entities to all similar entities in your project.

Shape Lite Pro Mechanical BIM

Icon: 

7.16.1 Method

BIMAUTOMATCH finds similarities based on the type of element, interior/exterior property, orientation, spatial location, amount of windows in a wall and already defined properties. It will not overwrite already defined properties, unless no unmodified entity can be found and the user specifies a source entity.

E.g.: You have attached a specific composition to a wall and a floorslab in the project. Instead of attaching the composition to all the similar walls and floor slabs 1 by 1, you can use this command to do it for you.

Note: This command works best when the project is bimified.

7.16.2 Options

Displays Auto Match dialog box:

Note: When there are unmodified or unclassified entities in the drawing and there is only one building, nothing is prompted.

Autocomplete
Matches all chosen options for entities in the open drawing with similar entities in the open drawing.

Note: When choosing **Autocomplete** with multiple buildings in the drawing, you have to specifies the building/(s) to apply style on:

All
All the buildings will be taken into account while doing AutoMatch.

Building
AutoMatch is only applied as if **Building** was the only building in the drawing.

0 for building-2
AutoMatch is only applied as if **building-2** was the only building in the drawing.

1 for building-3
AutoMatch is only applied as if **building-3** was the only building in the drawing.

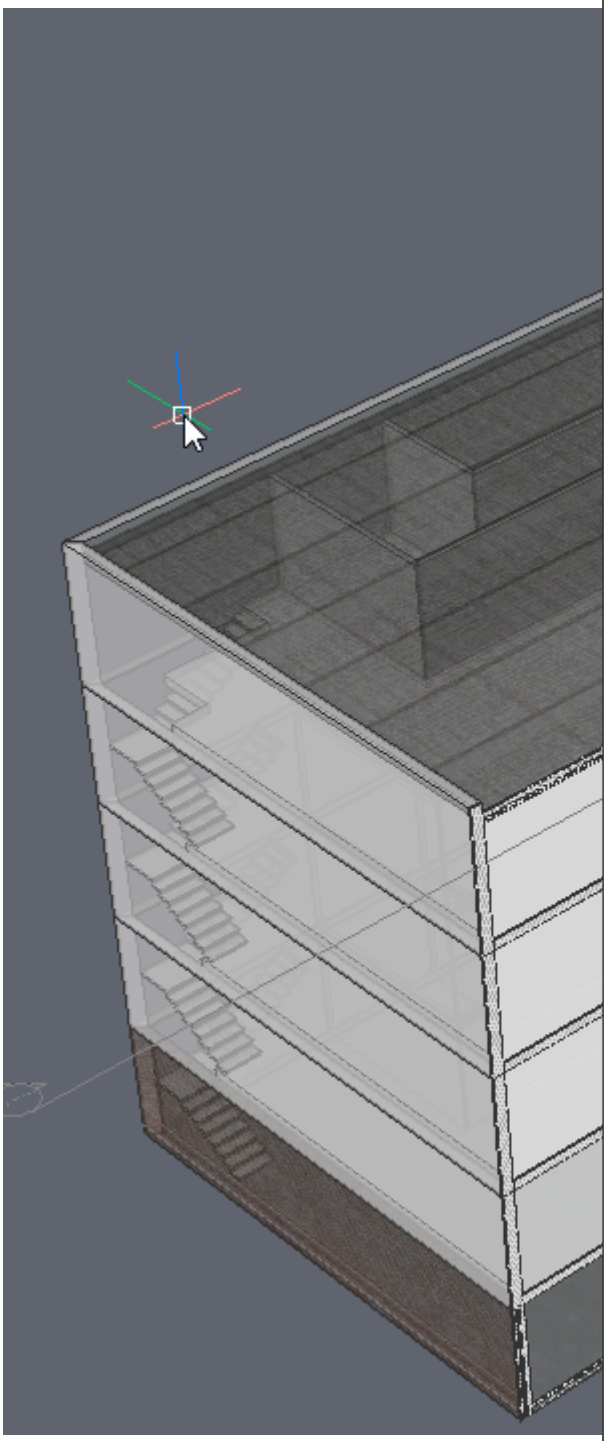
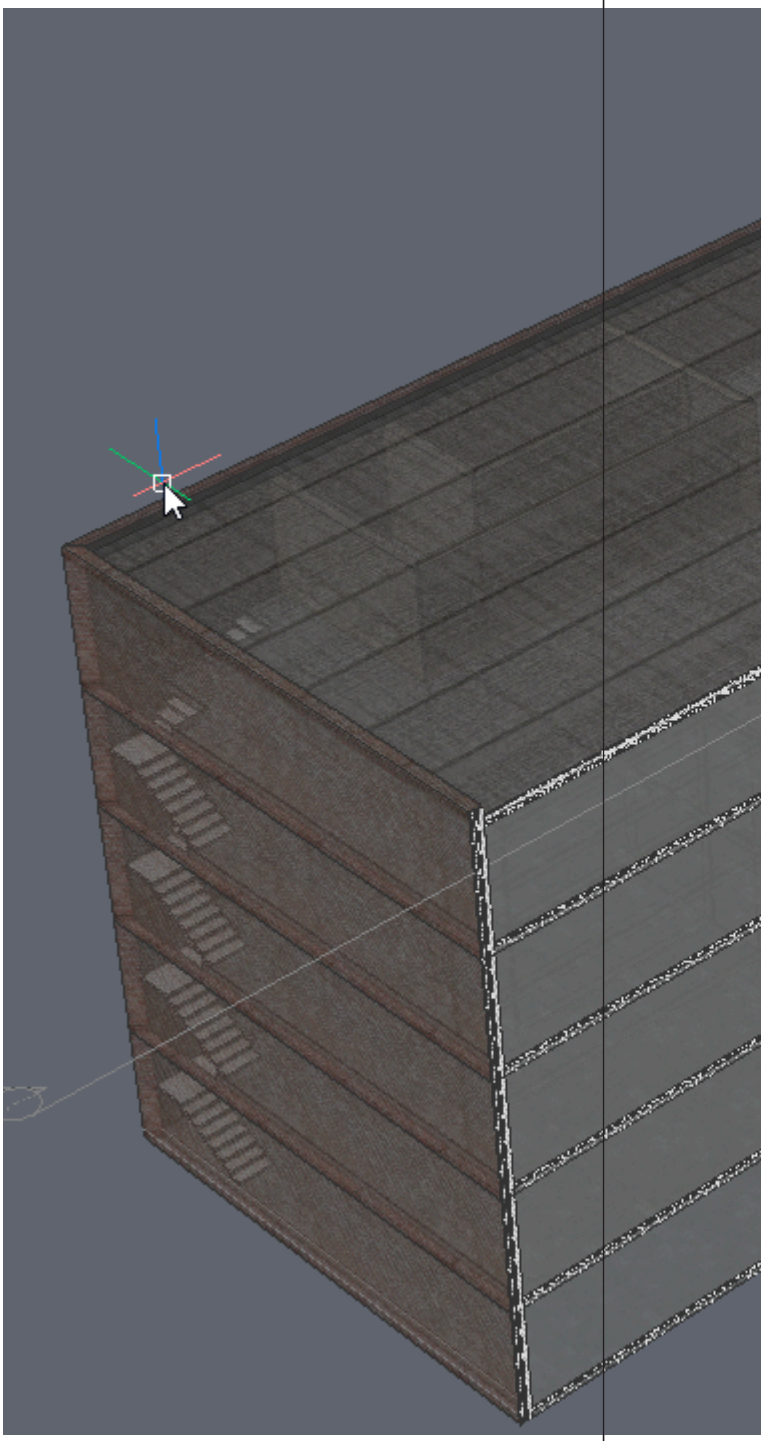
Other building name (first letter capitalized)
All building names, available in the current drawing, are listed here.

Note: AutoMatch is only applied as if **Other building name** was the only building in the drawing.

External file
Matches all chosen options for entities in the chosen drawing with similar entities in the open drawing.

Note: When choosing **External file** with multiple buildings in both drawings, you have to select a building to detect style from and all or only one building to apply style on (see above options).

Note: When all entities have compositions or a same customized property or all windows or stairs have customized parameters you will be prompted to select an entity, a window or a stair in the drawing from which you want to copy the composition or parameters to let AUTOMATCH to overwrite them.

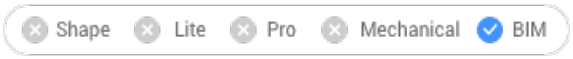
Every type of element has one entity with the correct composition.	All elements are given the correct composition.
	

7.17 BIMCLASSIFY command

Classifies an entity and gets a name and an internal unique GUID (= globally unique identifier).

Note: The GUID is maintained even when the BIM classification changes.

Any dwg entity can be given a classification.



Icons: 

7.17.1 Method

Select the entities to classify or press Enter to classify the entire drawing.

The BIM elements definition are detailed in the **Classify as BIM element** dialog box.

7.17.2 Options

Wall
Classifies the selection as walls.

Note: Attaching a Wall composition to a solid automatically classifies the solid as a wall.

Column
Classifies the selection as columns.

Slab
Classifies the selection as slabs.

Note: Attaching a Slab composition to a solid automatically classifies the solid as a slab.

Beam
Classifies the selection as beams.

wIndow
Classifies the selection as windows.

Door
Classifies the selection as doors.

sPace
Classifies 2D and 3D entities as spaces. (IFC Spatial Structure Element)
Converts BIM rooms to BIM spaces.

buiLding
Classifies 3D entities as buildings. (IFC Spatial Structure Element)

sTory
Classifies 3D entities as stories. (IFC Spatial Structure Element)

building Element
Classifies the selection as unspecified building elements.

Xref
Classifies the selection as External References.

Other
Allows to classify the selection or convert the selection to a block and classify the block reference.

Note: Displays the **Classify As Bim Element** dialog box.

Auto
Automatically classifies the selected entity or entities.

Note: This is the same as running BIMIFY command.

stRuctural
Classifies the selection as Building Structure Elements.

Unclassify
Removes all BIM data from the selected entities.

7.18 BIMCOPY command

Creates copies from the planar faces of 3D solids.

Shape Lite Pro Mechanical BIM

Icon: 

7.18.1 Description

Creates copies from the planar faces of 3D solids, and the sides and ends of linear solids.

7.18.2 Method

Specify a distance to create copy.

Note: You can specify a distance in the dynamic dimension field and right-click to create a single copy.

7.18.3 Options within the command

Copy
Creates copies.

Repeat
Repeats the copy using the same displacement. Move the cursor in the displacement direction.

Note: Move the cursor in the displacement direction. The further you move away from the source solid, the more copies are created.

Number
Creates a number of copies you want to have in your model.

Accept
Accepts the current distance you entered in the dynamic dimension field.

7.19 BIMCREATEDETAIL command

Creates a detail and saves it in the library.

Shape Lite Pro Mechanical BIM

7.19.1 Method

Select reference solids, detail entities and section in the Create detail dialog box.

7.19.2 Options within the command

Accept
Accepts the detail area. Displays the Save detail dialog box, which allows you to save the detail.

Reject
Does not accept the detail.

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

7.20 BIMCURTAINWALL command

Creates a curtain wall.

Shape Lite Pro Mechanical BIM

Icon: 

7.20.1 Description

Creates a curtain wall, as a block, from the face of a 3D Solid.

7.20.2 Method

Select a face and create a grid.

7.20.3 Options within the command

Length U panels
Sets the length of the panels in U direction (height).

Length V panels
Sets the length of the panels in V direction (width).

Number U panels
Number of panels in U direction (rows).

Number V panels
Number of panels in V direction (columns).

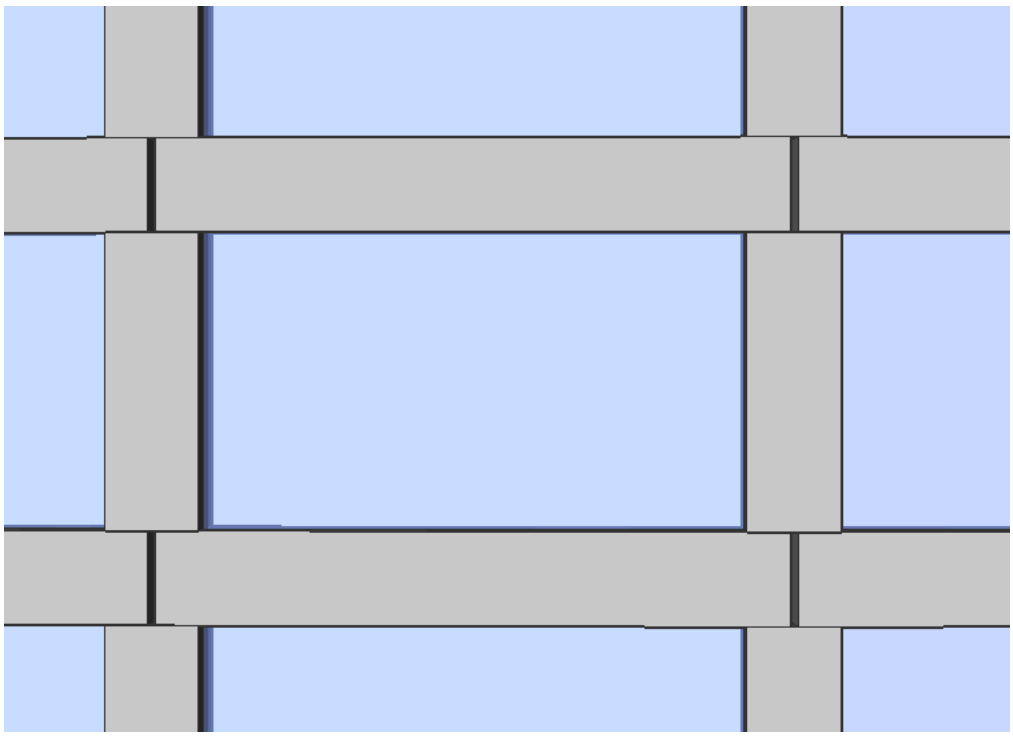
Width
Defines the width of the curtain wall frame, mullions and transoms.

Depth
Defines the depth of the curtain wall frame, mullions and transoms.

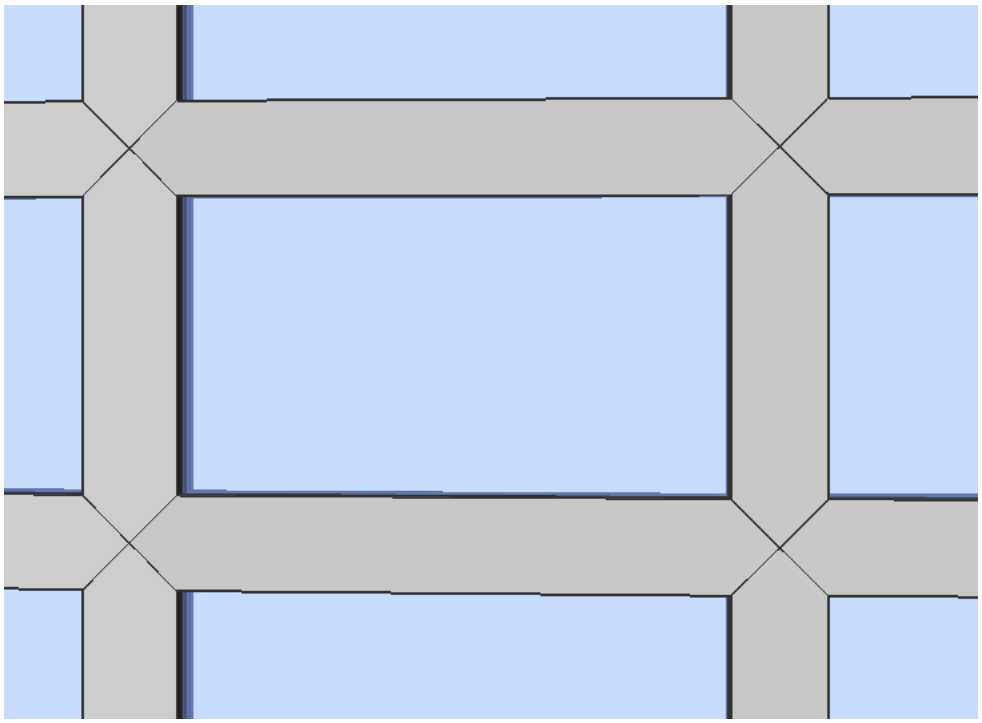
Glass thickness
Defines the thickness of the glass panels.

Connections type
Defines the type of the connections between horizontal and vertical mullions.

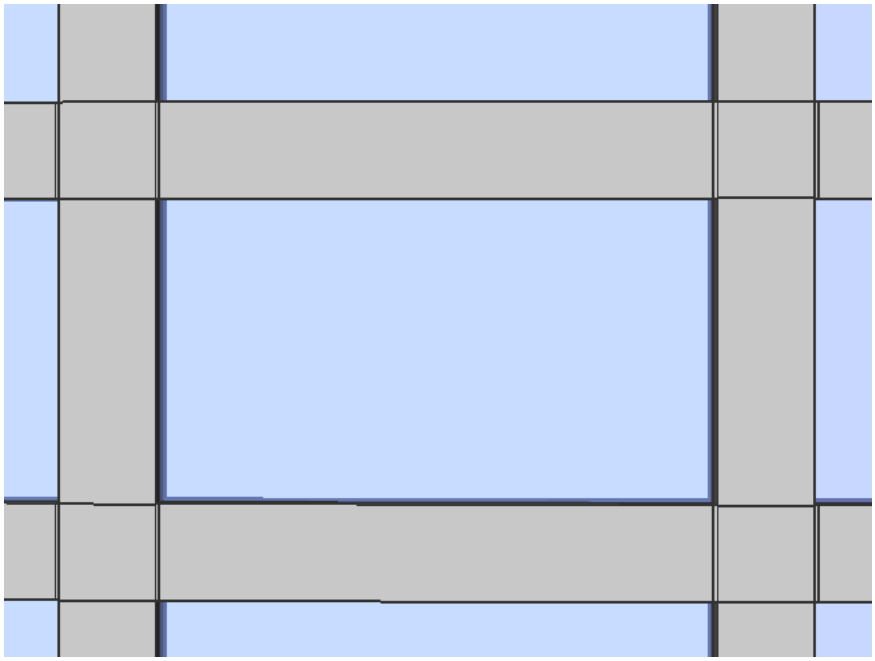
sTraight
Creates a straight connection.



Smooth
Creates a smooth connection.



Nodes
Creates a node connection.



Delete defining entities
Whether the selected source entity is retained or deleted depends on the value of the DELOBJ system variable.

7.21 BIMDECOMPOSE command

Decomposes compositions.

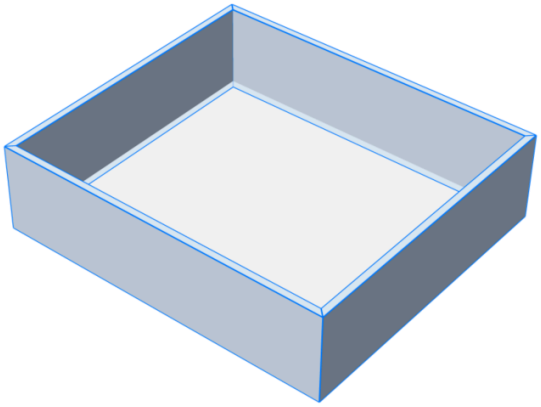
Shape Lite Pro Mechanical BIM

7.21.1 Description

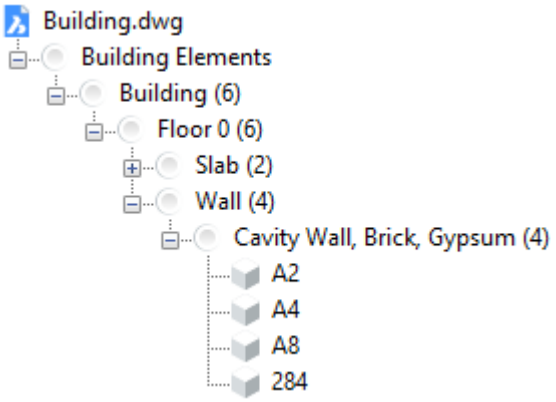
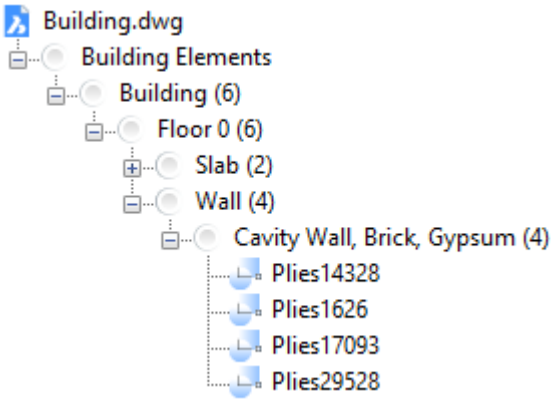
Decomposes compositions into their separate plies.

7.21.2 Method

This command offers one method to decompose compositions. After running the command, you can manually select BIM solids to decompose.



The solids are decomposed into their separate plies, which are put into one block. Each ply has inherited the BIM data of the original solid. You can see this in the Structure Browser.

Before	After
	

Note: If you now want to manipulate the plies, you can use the BEDIT command.

7.22 BIMDIMENSION command

Dimensioning semi - automatically the selected entities in the viewport.

Note: The command is available only in Paper Space.

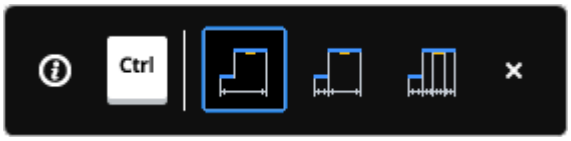
Shape Lite Pro Mechanical BIM

Icon:

This command opens the **Bim Dimension** command panel.

7.22.1 Method

Select single or multiple entities in the viewport by clicking one by one or with Box selection, apply filters available in the BIM Dimension dialog box and pick the location for dimensions. Press CTRL key to toggle dimensioning options that depend on the selected entities type.



Note: Make sure that the Hot Key Assistant (HKA field in the Status bar) is ON.

Press Enter for a new selection set.

Note: You can also make a selection by using a python script to query specific objects (see the BIMPYTHON command).

7.22.2 Options within the command

place Dimensions
Pick a point to place the dimensions.

Undo
Undo within the command.

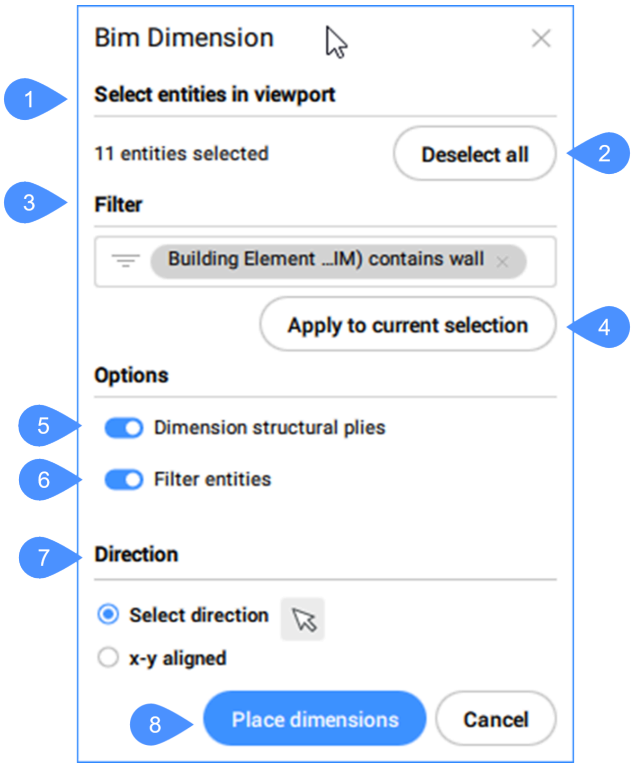
Deselect all
Removes the selection set.

Select direction
Selects the direction for the dimensions.

x-y align
Dimensions will be aligned with x, y axis (default direction).

Angle
Dimensions will be oriented with the specified angle.

7.22.3 Bim Dimension command panel



1 Select entities in viewport

2 Deselect all

3 Filter

4 Apply filter to current selection

5 Dimension structural plies

6 Filter entities

7 Direction

8 Place dimensions

Select entities in viewport
Shows the number of selected entities.

Deselect all
Deselects all entities.

Filter
Allows to filter the selected entities by adding one or more parameter filters from the drop down list.

Apply filter to current selection
Applies the filter to current selection.

Dimension structural plies
Chooses to dimension or not the plies of a wall with Function equal to Structure.

Filter entities
Toggles the display of the Filter selection section.

Direction
Indicates the direction for the dimensions.

Select direction
Allows to define a direction by pressing the arrow from the right side and specify an Angle. The default direction is x, y axis.

x-y align
Dimensions will be aligned with x, y axis (horizontal or vertical).

Place dimensions
Specifies the place for dimensions.

7.23 BIMFLIP command

Flips a starting face of a composition or mirrors/flips an insert.

Shape Lite Pro Mechanical BIM

Icon:

7.23.1 Description

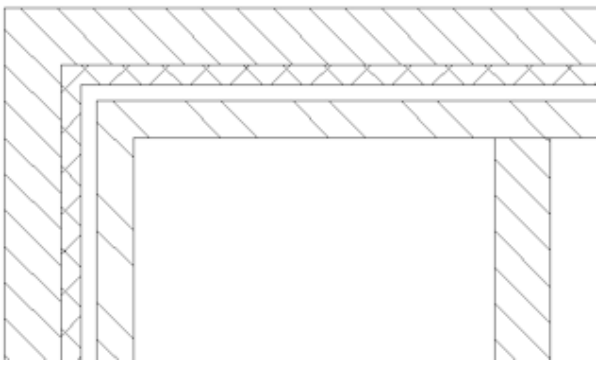
Flips the starting face from which the layers of a composition are set out, or mirrors an insert (e.g. a window or door) left/right or flips it inside/outside.

7.23.2 Method

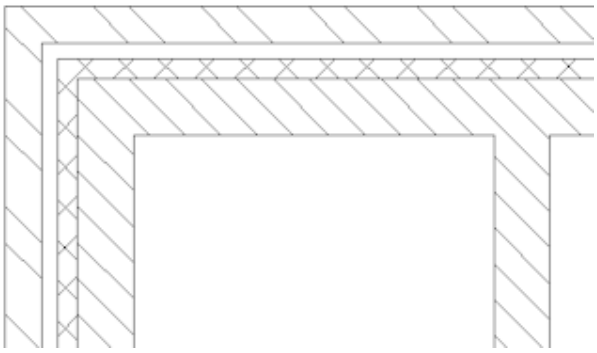
There are two methods to flip entities:

- Left- right.
- In-out

Before:



After:



Note: The result is visible in drawings generated by the BIMSECTIONUPDATE command if a multi-layered composition is attached to the sectioned solid.

7.23.3 Options

Select BIM entity to flip
Inserts, walls and slabs are accepted in the command.

Note: The command only succeeds for solids that meet the following conditions:

- have a BIM composition attached.
- contain a BIM classification
- are in a calculated BIM section.

Left-right
Mirrors the insert about a vertical axis in the face of the solid.

In-out
Flips the insert to the opposite face of the solid.

7.24 BIMFLOWCONNECT command

Creates a connection between flow segments.

Shape Lite Pro Mechanical BIM

Icon:

7.24.1 Description

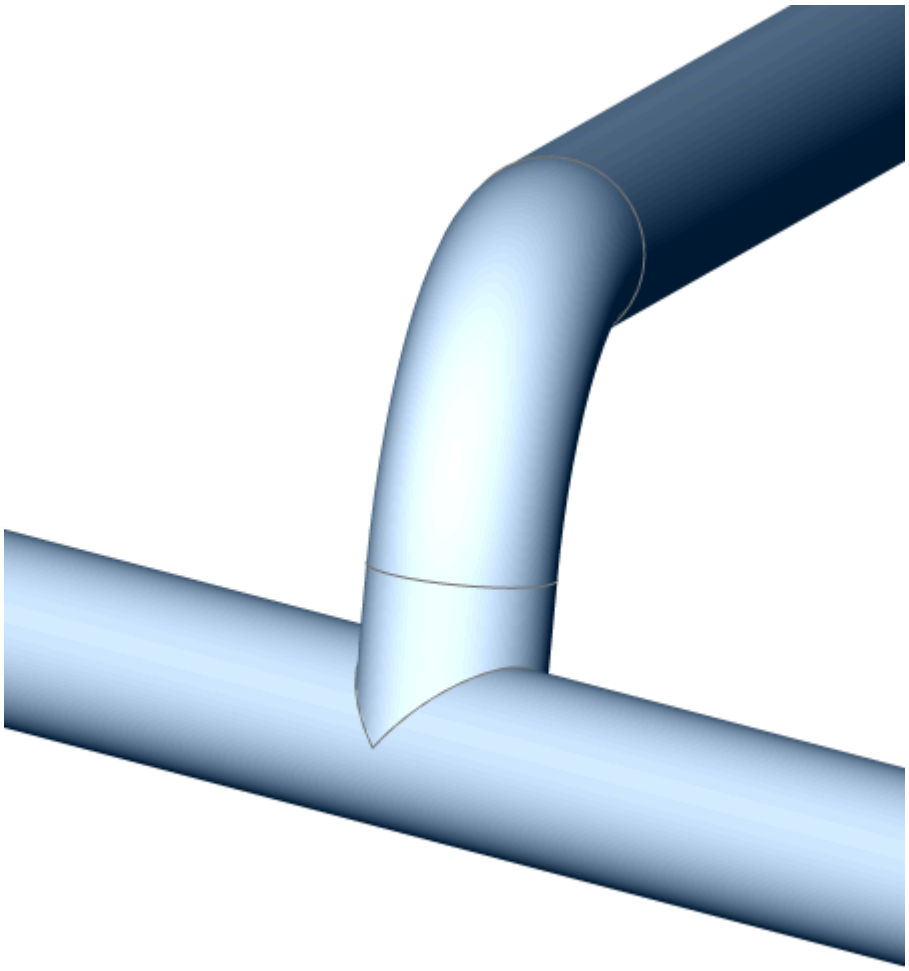
Creates a connection between flow segments, such as pipes or HVAC ducts.

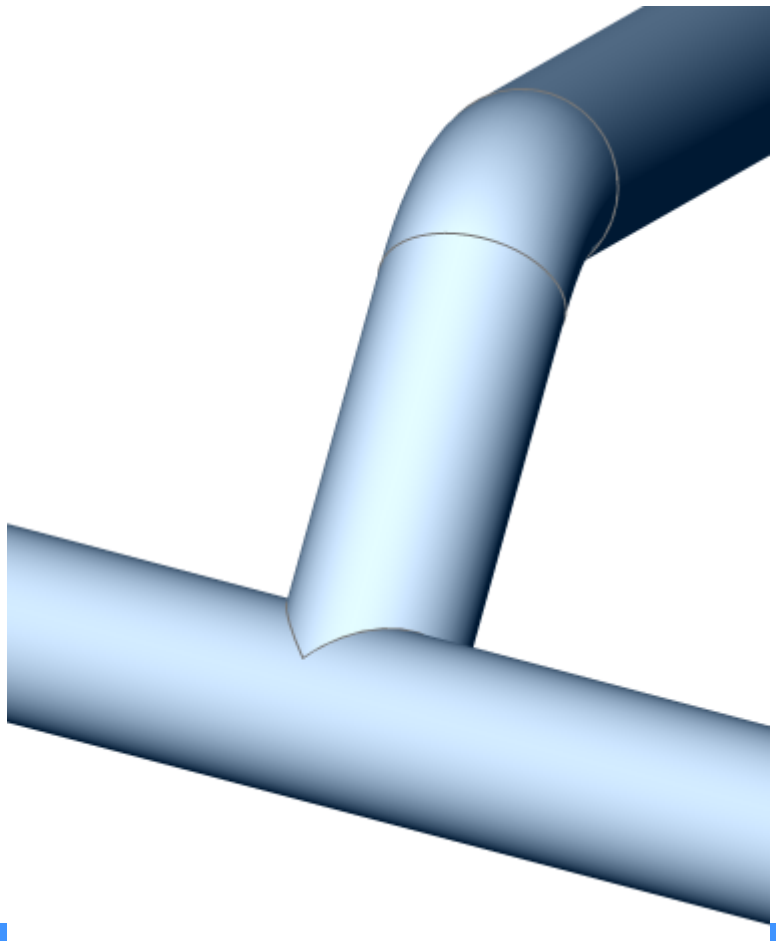
Note: When more than 2 flow segments are selected, the axes of the selected segments must be coplanar. When multiple coplanar segments are selected, the connections are created. When two non-coplanar solids are selected, an extra connection element is introduced.

7.24.2 Options

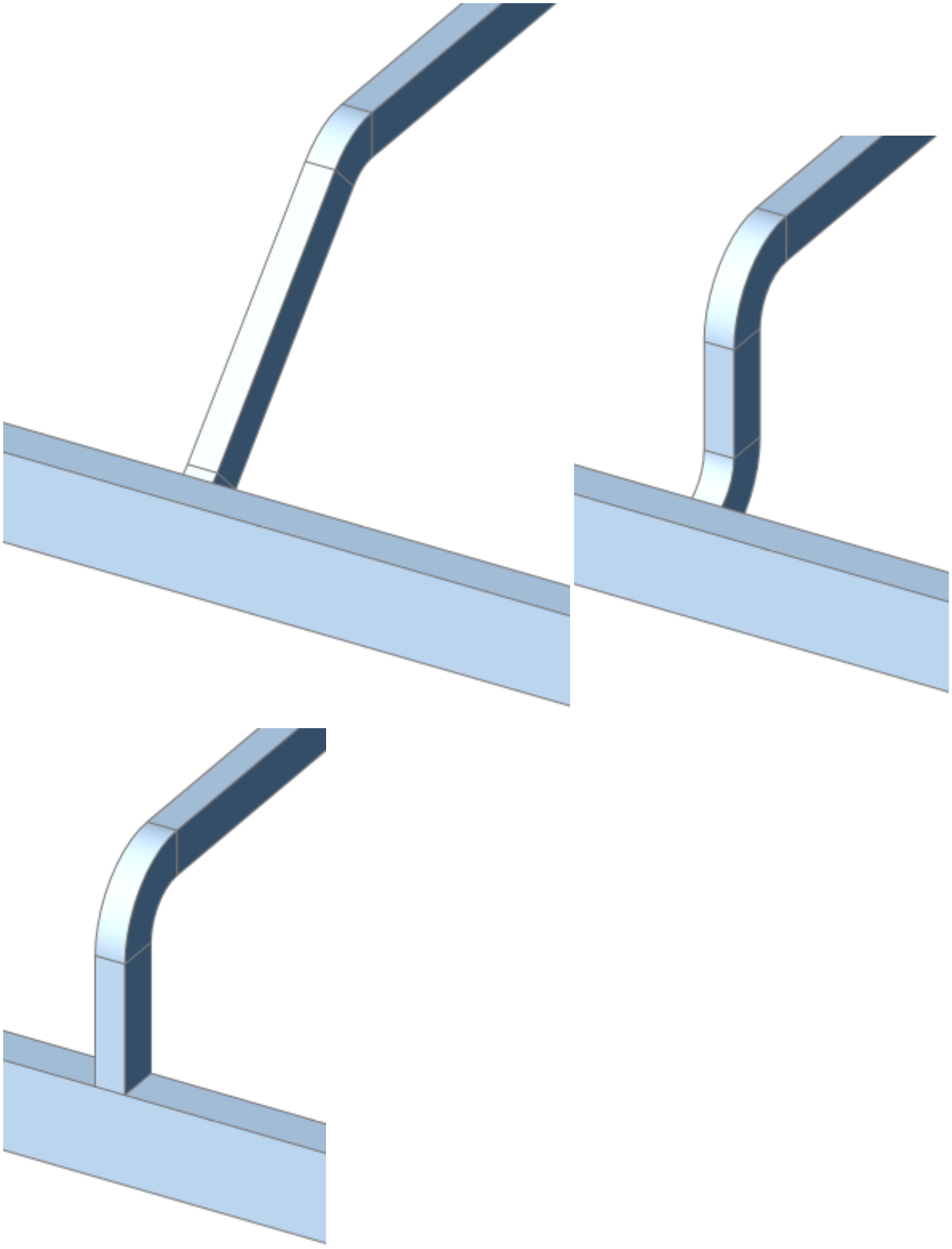
Switch
Switches between possible connections.

Alternatives for circular flow segments:



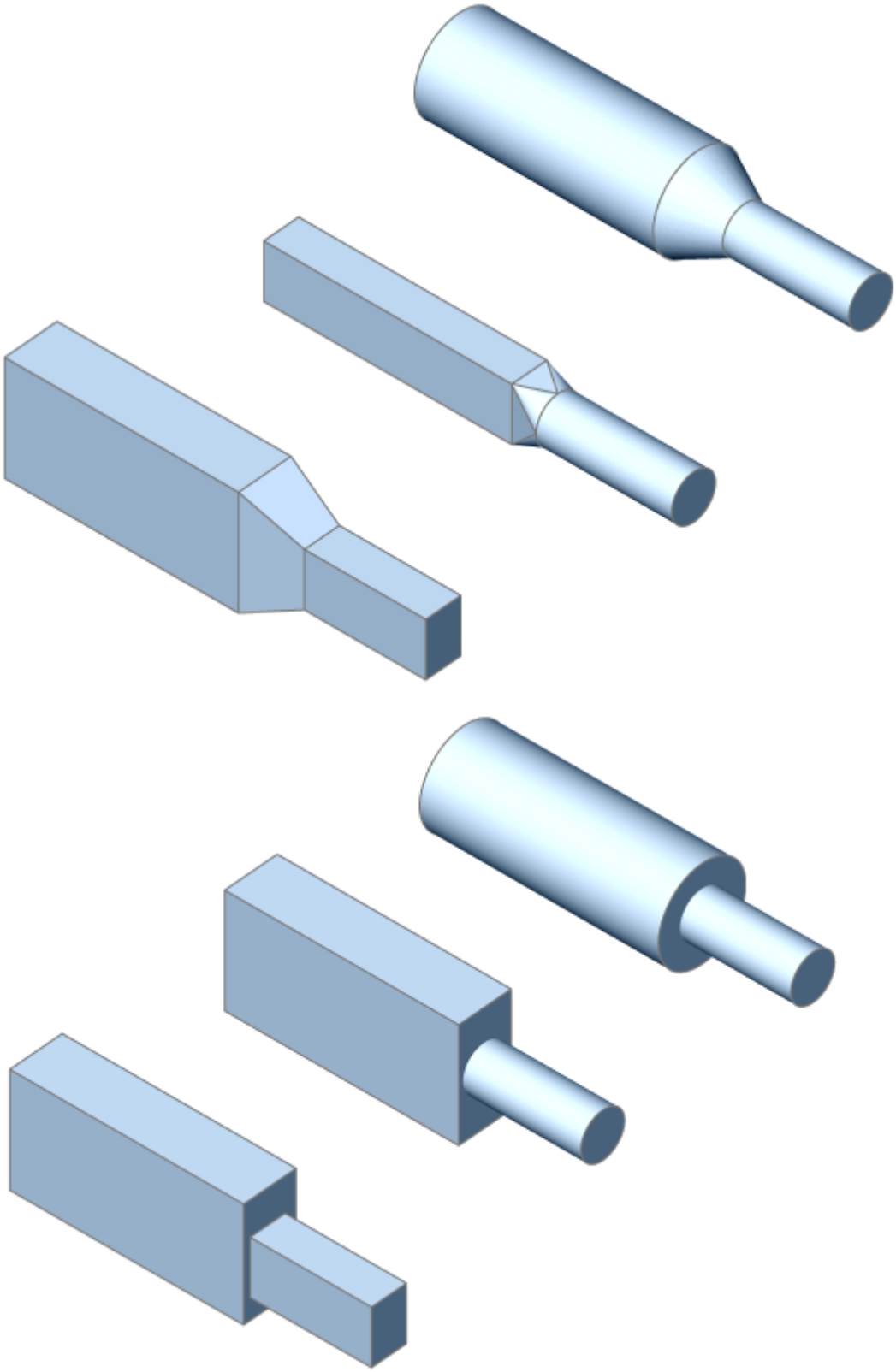


Alternatives for rectangular flow segments:



Note: Press the Ctrl key to cycle through different types of connection. Hot Key Assistant must be set ON.

When the section of the selected flows is different, reducer elements are inserted.
Alternatives for reducers:



Note: The creation of bends, tees and reducers is controlled by related system variables.

7.25 BIMGRID command

Creates 2-dimensional grids.

Shape Lite Pro Mechanical BIM

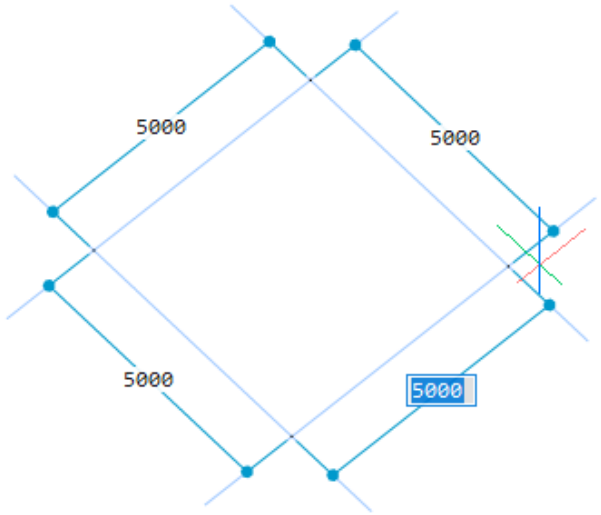
Icon:

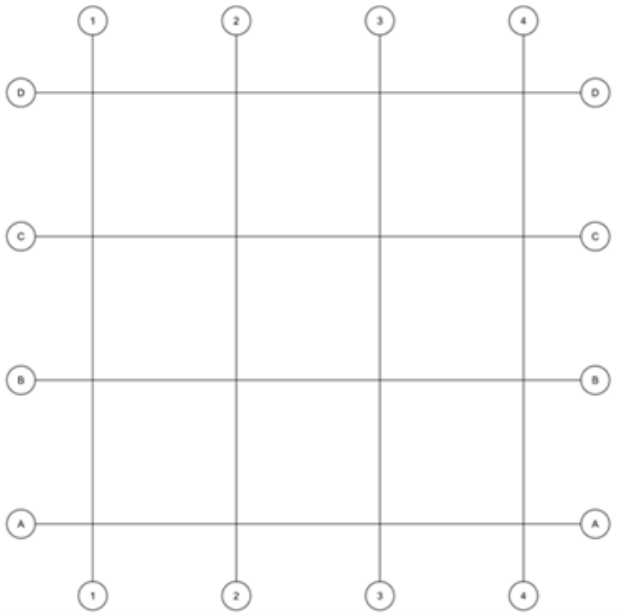
7.25.1 Description

Creates 2-dimensional rectangular and radial grids.

7.25.2 Method

The grid is created by specifying two points.





7.25.3 Options

Offset U
Sets the line offset between U lines in V direction (interdistance between the U lines).

Offset V
Sets the line offset between V lines in U direction (interdistance between the V lines).

Offset Labels
Sets the label offset to grid axes.

Radial
Creates a radial grid by specifying the center point, the start lines and the end lines.

7.26 BIMIFY command

Analyzes the model and runs automatic classification and spatial location assignment for the entire model.

Shape Lite Pro Mechanical BIM

Icon:

7.26.1 Description

Analyzes the model and runs an automatic classification and spatial location assignment for the entire model. Creates new buildings and/or stories and spaces if necessary. Optionally creates plan sections and elevations.

Note:

- Section entities are converted to BimSection entities.
- Simple block references can be classified.

7.26.2 Method

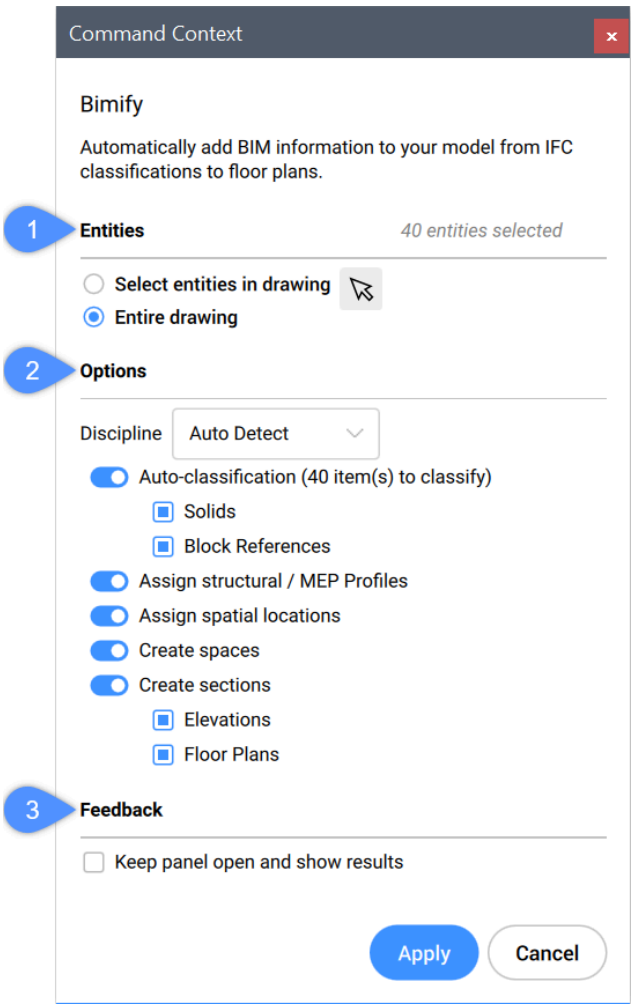
Running Bimify on the entire model will classify objects, assign spatial locations, detect spaces and create elevations and floor plan section planes in the model.

When the Bimify process is completed, an overview of the result is provided in the Command line.

This command opens the **Bimify** command panel.

7.26.3 Bimify command panel

The **Bimify** command panel allows you to analyze the model and run an automatic classification and a spatial location assignment for the entire model.



1 Entities

2 Options

3 Feedback

Entities

Select entities in drawing

The BIMIFY command runs for the selected entities in the drawing.

Entire drawing
The BIMIFY command runs for all entities in the drawing.

Options

Discipline
Determines to what discipline the model belongs in order to improve the automatic classification.

Auto Detect
An AI algorithm determines the discipline automatically.

Architectural
The model is classified as an architectural model.

Structural
The model is classified as a structural model.

MEP
The model is classified as a MEP model.

Mixed Model
The model cannot be assigned to 1 discipline and is perceived as a mixed model.

Auto-classification
Defines the classification of solids and blocks.

Note: Also see the BIMCLASSIFY command.

Solids
Makes sure that 3D solids are classified as walls, slabs, etc.

Block References
Makes sure that blocks are classified as windows, doors, etc.

Assign structural/MEP Profiles
Assigns columns, beams, members or flow segments profiles that are available in the **BIM Profiles** panel.

Note: If no match can be found for a profile element in the standard library, the BIMIFY command creates a new custom profile in the project's library.

Assign spatial locations
Assigns spatial locations (buildings and floors). See the BIMATTACHSPATIALLOCATION command.

Create spaces
Detects external walls and sets the property Wall Common/is External = ON and also detects spaces. See the BIMSPACE command.

Create sections
Creates BIM Section entities. See the BIMSECTION command.

Elevations
Creates 4 elevations (Front, Back, Left and Right).

Floor Plans
Creates a plan section for each floor.

Feedback

Keep panel open and show results
If toggled, the panel will show the progress and results while calculating.

Note: This option is off by default for consistency with previous versions.

7.27 -BIMINSERT command

Inserts sheet metal form features, and mechanical and BIM components in the command line.

Shape Lite Pro Mechanical BIM

7.27.1 Method

Specify the full path and file name of the file.

Note: Type ~ (tilde) to display the Select file to insert dialog box, which allows you to select a file to insert.

7.28 BIMINSERT command

Inserts BIM components into the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command is deprecated. Use BMINSERT instead.

7.29 BIMINVERTSPACES command

Converts a set of input 3D solid entities, which represent spaces, to a building structure which encompasses these spaces.

Shape Lite Pro Mechanical BIM

Icon: 

This command uses a set of 3D solid entities to create a building structure made of walls, slabs, and roofs.

This command is envisioned to be most useful in a Scan-2-BIM workflow. Starting from a point cloud, using the FITPLANAR command to fit planar faces to the point cloud and stitch them together to solids (representing spaces) and finally using these solids as input for the BIMINVERTSPACES command.

This command opens the **Bim Invert Spaces** command panel.

7.29.1 Methods

There are two methods of running this command.

1 Launch the command with nothing selected.

You are prompted:

Apply to entire drawing or [**S**elect solids/change **T**olerances/change **O**ptions] <Apply>:

In this case, by default, all 3D solid entities will be selected.

2 Select the entities (3D solids) to be used by the command and then launch the command.

You are prompted:

Apply to 4 solid(s) or [change **S**election/**E**ntire drawing/change **T**olerances/change **O**ptions] <Apply>:

In this case, four solids have been selected prior to launching the command.

7.29.2 Options within the command

Select solids
Allows to select some 3D solid entities from the drawing to be used by the command.

change Selection
Allows to select some 3D solid entities from the drawing to be used by the command.

Entire drawing
All the 3d solid entities in the drawing will be used by the command.

change Tolerances
This option allows to change the next settings:

Set bottom slab thickness <350.000000>:
Sets the thickness of the slabs created at the bottom of the structure.

Set top slab thickness <250.000000>:
Sets the thickness of the slabs created at the top of the structure.

Set min. inner slab thickness <150.000000>:
Sets the minimum distance required between parallel faces for creating an inner slab.

Set outer wall thickness <350.000000>:
Sets the thickness of the outside walls.

Set min. inner wall thickness <50.000000>:
Sets the thickness of walls created on the inside of the structure.

Set parallel angle tolerance <5.000000>:
Sets the angular tolerance for determining if the faces are parallel.

Set coplanar dist. tolerance <50.000000>:
Sets the allowed distance between the faces for being coplanar.

Note: The units used by these settings are the those set in the drawing.

change Options
This setting allows to make of the next options:

Erase input spaces <0>:
On the completion of the command, the input entities are erased or kept in the drawing.
When this value is 0, the input entities will be erased.
When this value is greater than 0, the input entities will be kept in the drawing.

Create individual walls per floor (0) or single wall shells per floor (1) <0>:
When this value is 0, the command creates individual outer walls.

When this value is greater than 0, the command creates the outer wall in one piece (one 3D solid entity).

Note: This option works in conjunction with the options **Slice outer wall shell per floor** and **Split outer walls in separate solids**. The configuration of the walls created by the command depends on all these settings.

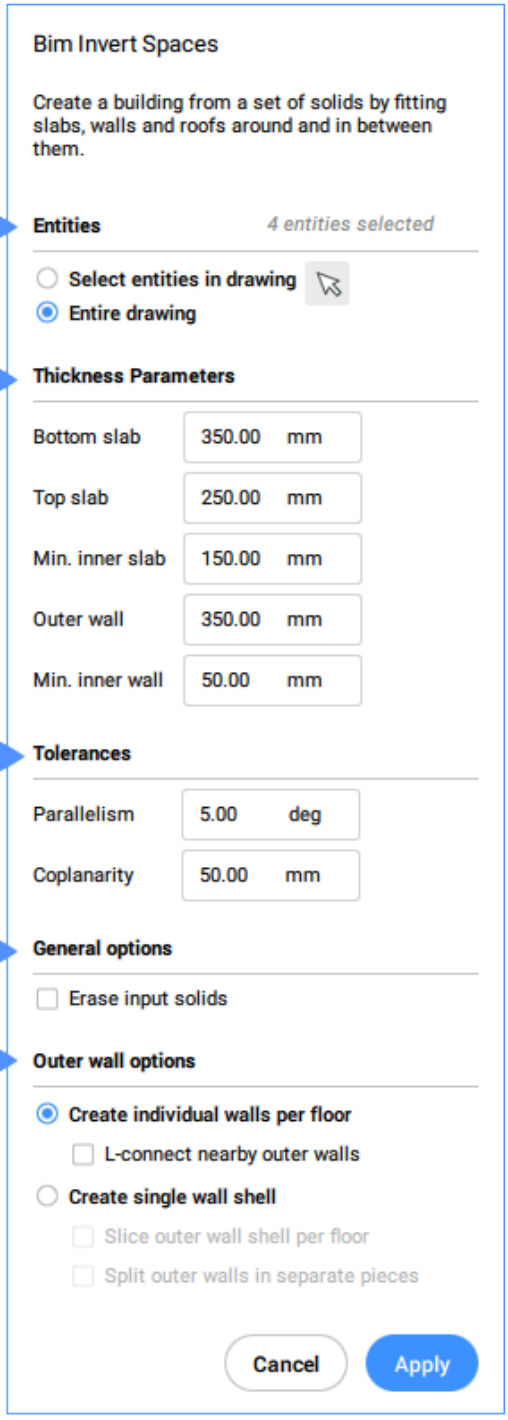
Apply L-Connect to nearby outer walls <0>:
When this value is 0, the nearby walls will not be connected at corners.
When this value is greater than 0, the command creates mitered corner connections between nearby walls.

Slice outer wall shell per floor <0>:
Creates the outer wall of the structure as many 3D solid entities. The outer walls will be sliced horizontally at each floor level.

Split outer walls in separate solids <0>:
Creates separate 3D solid entities as outer walls. The outer wall will be split into many 3D solid entities, one for each side of the structure.

7.29.3 Bim Invert Spaces command panel

This panel contains the settings for the BIMINVERTSPACES command.



1 Entities

2 Thickness Parameters

3 Tolerances

4 General options

5 Outer wall options

Entities
This section contains the options for selecting the entities to be used by the command.

Select entities in the drawing
Clicking on the arrow button allows the user to select specific 3D solid entities.

Entire drawing
All the 3D solid entities in the drawing will be used as input.

Thickness Parameters
Contains the values of the parameters used by the command to create the building structure.

Bottom slab
Sets the thickness of the slabs created at the bottom of the structure.

Top slab
Sets the thickness of the slabs created at the top of the structure.

Min. inner slab
Sets the minimum distance required between parallel faces for creating an inner slab.

Outer wall
Sets the thickness of the walls created on the outside of the structure.

Min. inner wall
Sets the minimum distance required between parallel faces for creating an inner wall.

Tolerances
Contains the tolerance values for the relative positions of the faces that belong to the 3D solid entities used as input.

Parallelism
Sets the angular tolerance for determining if the faces are parallel.

Coplanarity
Sets the allowed distance between the faces for being coplanar.

General options
Contains the option of erasing the input entities.

Erase input solids
When this option is checked, the input solid entities will be erased on the completion of the command.

Outer wall options
This section contains the options for creating the outer walls.

Create individual walls per floor
Creates separate outer walls for each floor.

L-connect nearby outer walls
Creates mitered corner connections between nearby walls.

Create single wall shell
Creates the outer wall of the whole structure as one single 3D solid entity.

Slice outer wall shell per floor
Creates the outer wall of the structure as many 3D solid entities. The outer wall will be sliced by a horizontal plane at each floor level.

Split outer walls in separate pieces
Creates separate 3D solid entities as outer walls. The outer wall will be split into many 3D solid entities, one for each side of the structure.

7.30 BIMLINEARSOLID command

Creates a chain of linear solids.

Shape Lite Pro Mechanical BIM

Icon: 

7.30.1 Options within the command

Last point
Press Enter to use the most recently specified point.

Follow
Draws the next linear solid at the same angle as the previous one.

Rectangular
Switches to a default rectangular profile.

Circular
Switches to a default circular profile.

Library
Opens the Profiles dialog box.

select in Model
Allows to select a profile in the model.

adJust profile
Allows to adjust the dimensions of the default rectangular and circular profile.

Note: For rectangular profile you may change the width and the height and for circular profile the radius.

Angle
Draws at a user-specified angle and distance.

Length
Draws at a user-specified distance and angle.

Undo
Erases the last linear solid.

Quarter turn
Rotates the profile 90° counter clockwise.

Rotate
Allows to rotate the profile by a user defined angle.

Positive values rotate the profile counter clockwise.

choose other Profile
Allows to use a different profile for the next linear solid.

7.31 BIMLIST command

Lists the selected entities.

Shape Lite Pro Mechanical BIM

7.31.1 Description

Lists all the selected entities in the Command line, displaying their handle, material and GUID.

7.31.2 Method

Run the command and select the BIM entities to list.

7.32 BIMMULTISELECT command

Selects linear solids.

Shape Lite Pro Mechanical BIM

Icon: 

7.32.1 Description

Selects linear solids with coplanar and/or parallel axes.

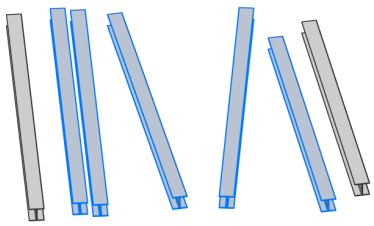
7.32.2 Method

Selects linear solids with coplanar and/or parallel axes. If a face of a linear solid is selected, the corresponding faces of the linear solids with coplanar/parallel axes are selected.

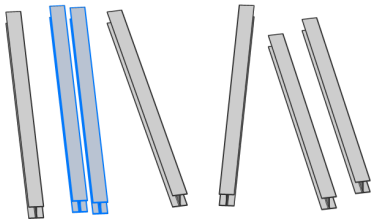
Note: Select a different face by repeatedly pressing the TAB key to cycle through all faces of the selected solid.

7.32.3 Options

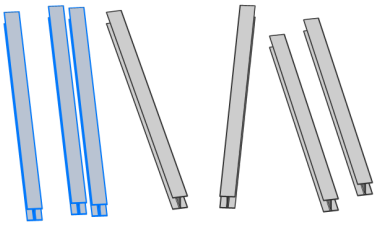
all Coplanar axes
Selects all solids of which the axis is coplanar with the axis of the selected solid.



coplAnar and parallel axes
Selects all solids of which the axis is coplanar and parallel with the axis of the selected solid.



all Parallel axes
Selects all solids of which the axis is parallel with the axis of the selected solid.



Note: Press the Ctrl key to cycle through the command's options. Hot Key Assistant must be set ON.

7.33 BIMPARAMETRIZEDETAIL command

Generates parameters for a detail.

Shape Lite Pro Mechanical BIM

Icon:

7.33.1 Description

Generates parameters for a detail in order to easily change dimensions, angles, etc.

Note: Use this command in the detail file, not in a project. Detail files are saved in the folder, and its subfolders, which is defined by the DETAILSPATH system variable. By default, this is C:\ProgramData \Bricsys\Details\.

Note: The parameters and constraints can be edited in the Mechanical Browser.

7.34 BIMPATCH command

Allows to replace part of a generated section.

Shape Lite Pro Mechanical BIM

Note: BIMPATCH cuts out part of geometry and replaces it by a block reference. In this cutting process, entity data linked to the 3D model are lost. Therefore, starting from V21.2, this command is no longer available.

7.34.1 Description

Allows you to specify a rectangular area on a block that was previously generated and modify it.

7.34.2 Method

Patches can be edited same as blocks, using the REFEDIT command.

When the corresponding BIM section is updated, BricsCAD will check if the generated section result within that rectangle is identical to the result which was generated at the time the BIM patch was created:

- If not, then the user edited BIM patch is preserved but the boundary is colored red.
- If it is, the user edited BIM patch is preserved.

Intersection points and endpoints of curves originating from 3D objects are used to find the position of a BIM patch rectangle. When objects are modified or moved in the 3d model in such way that the position of these intersection points change, but the geometry inside the BIM patch rectangle is not affected, then BricsCAD will relocate the user edited BIM patch.

Note: The command is valid in model space only.

Note: In order to use the command, the GENERATEASSOCVIEWS system variable must be set ON.

This command cuts out part of geometry and replaces it by a block reference. The cut-out is generated as a nested block in the BIM section. BIM patch blocks are named Patch1, Patch2, etc. The rectangular boundary of a BIM patch is placed on layer BIMPATCH_BOUNDARY.

7.35 BIMPROFILES command

Opens the Profiles dialog box.

Shape Lite Pro Mechanical BIM

Icon:

7.35.1 Description

Opens the Profiles dialog box to create and modify BIM profiles.

7.36 BIMPROJECTINFO command

Opens the BIM project info dialog box.

Shape Lite Pro Mechanical BIM

7.36.1 Description

Open the BIM project info dialog box to view and modify BIM project information.

7.37 BIMPROPAGATE command

Maps details related to selected base solids to similar base solids in the model.

Shape Lite Pro Mechanical BIM

Icon:

7.37.1 Description

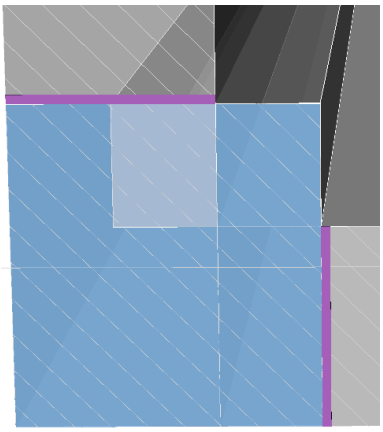
Maps details (solids, holes, finishing geometry, etc.) related to selected base solids to similar base solids in the model and optionally outlines the detail in a grid.

7.37.2 Method

Select reference solids and, optionally, detail entities (solids, block references, faces, edges, etc.).

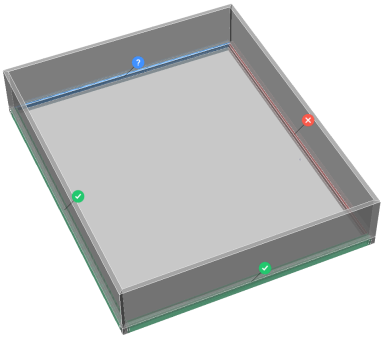
It can be used to copy details throughout your model.

Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.



Note: Hovering over the widget will reveal more options.

Note: If the PROPAGATESEARCHSPACE system variable is ON, you can select a search space which allows you to limit the places that will be propagated.

7.37.3 Options within the command

Block

Propagates the detail as a block.

Copy
Propagates the detail as a copy.

Select search space
Select the space(s) to limit the places that will be propagated.

Entire drawing
Selects the entire drawing as a search space.

No
Does not accept the detail.

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
Displays the **Save Drawing As** dialog box, which allows you to save the detail.

save detail to Library
Displays the **Save Detail** dialog box, which allows you to save the detail to library.

7.38 BIMPROPAGATECORNER command

Propagates details.

Shape Lite Pro Mechanical BIM

Icon:

7.38.1 Description

Propagates details connected to three planar base solids (e.g. corners).

7.38.2 Method

Select at least three planar reference solids that form a 3D corner. The detail will be copied to similar corners.

7.38.3 Options

Select detail objects
Select additional solids, blocks or faces to be included in the corner detail.

yes, copy as Block
This option is available only when detail objects are selected. A block is created from the selected detail objects, which is then copied.

Note: The default block names are Block, Block 1, Block 2, ... You can rename blocks in the Blocks category in the Drawing explorer. See the EXPBLOCKS command.

yes, plain Copy
This option is available only when detail objects are selected. The detail objects are copied as is.

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
A SAVE STANDARD FILE dialog box displays.
You are prompted to specify a file name to save the detail.

save detail to Library
The SAVE DETAIL dialog box displays.
See the BIMCREATEDETAIL command.

Apply
Accept or reject suggestions by clicking on the tick.

Note: See the Apply option in the BIMPROPAGATE command.

7.39 BIMPROPAGATEEDGES command

Propagates a detail along the edge of a planar solid.

Shape Lite Pro Mechanical BIM

Icon:

7.39.1 Description

It propagates railings, gutters, borders, wall caps, etc.

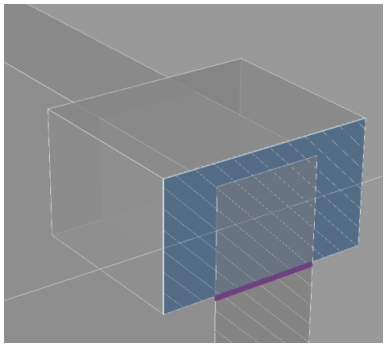
Note: The detail of BIMPROPAGATEEDGES will always be a 2D slice through a linear detail.

7.39.2 Method

Select planar reference solid to which the edge detail is related and the detail entities to be copied.

It can be used to copy details along edges of planar solids.

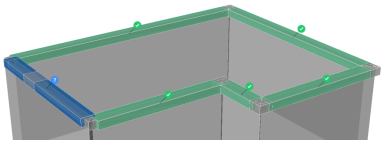
Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.

- A red X mark means that the suggestion will not be applied.



7.39.3 Options within the command

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
Displays the Save Drawing As dialog box, which allows you to save the detail.

save detail to Library
Displays the Save Detail dialog box, which allows you to save the detail to library.

Yes
Only edges with similar orientation are selected.

No
All minor faces of similar planar solids are selected.

Outside reference solid
The propagated detail is placed outside the base solid. The volume of the base solids is not modified.

Inside reference solid
The propagated detail is placed inside the base solids. The volume of the detail is subtracted from the base solids.

7.40 BIMPROPAGATEFROMFILE command

Opens the Select source file dialog box.

Shape Lite Pro Mechanical BIM

7.40.1 Description

Opens the Select source file dialog box to select a dwg file to propagate in the current drawing.

After selecting the file and choosing **Open**, you are prompted at the Command line. The prompt sequence depends on the type of detail you selected: **Planar**, **Linear**, **Pattern**, **Edge** or **Corner**. For more information see the relevant BIMPROPAGATE commands.

7.41 -BIMPROPAGATEFROMFILE command

Propagates a saved detail.

Shape Lite Pro Mechanical BIM

7.41.1 Description

Propagates a saved detail on all suitable connections in your project.

Note: The prompt sequence depends on the selected detail type: planar, linear, patter, edge or corner.

7.41.2 Method

Specify the full path and file name of the saved detail.

You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.

7.42 BIMPROPAGATELINEAR command

Propagates connections between linear elements.

Shape Lite Pro Mechanical BIM

Icon:

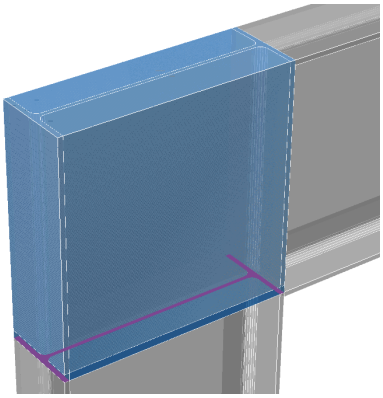
Linear elements accepted: beams, columns, pipes, ducts and their connections to walls and slabs.

7.42.1 Method

Select the linear or planar reference solids that form the connection and the detail entities to be copied.

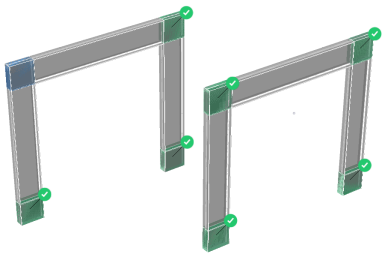
It can be used to copy detailed connections between two or more linear solids.

Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.



7.42.2 Options within the command

Block
Propagates the detail as a block.

Copy
Propagates the detail as a copy.

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
Displays the Save Drawing As dialog box, which allows you to save the detail.

save detail to Library
Displays the Save Detail dialog box, which allows you to save the detail to library.

7.43 BIMPROPAGATEPATTERN command

Propagates a single element on a flat surface to multiple locations and grids.

Shape Lite Pro Mechanical BIM

7.43.1 Description

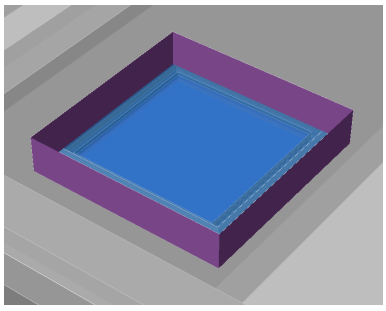
It propagates: lights, light switches, windows, air diffusers, etc.

7.43.2 Method

Select planar solid to which detail is related and the detail entities to be propagated.

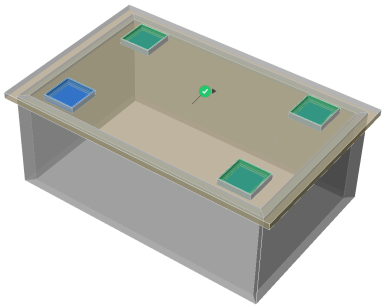
It can be used to copy objects to similar locations or over a certain pattern or grid on top of a planar solid.

Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.



Note: Hover over the tick to toggle between Similar location and Grid options.

7.43.3 Options within the command

Inflate first
Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail
Displays the Save Drawing As dialog box, which allows you to save the detail.

save detail to Library
Displays the Save Detail dialog box, which allows you to save the detail to library.

7.44 BIMPROPAGATEPLANAR command

Propagates connections between planar elements.

Shape Lite Pro Mechanical BIM

Icon:

Planar elements accepted: walls, slabs, roofs.

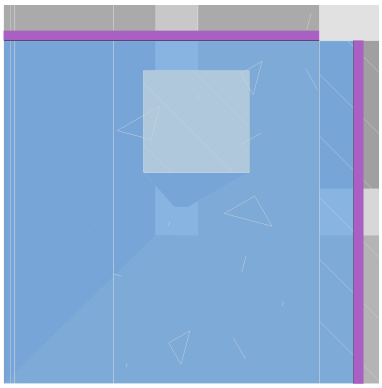
Note: The detail of BIMPROPAGATEPLANAR will always be a 2D slice through a linear detail.

7.44.1 Method

Select planar reference solids that form the connection and, optionally, detail entities to be copied as part of the connection.

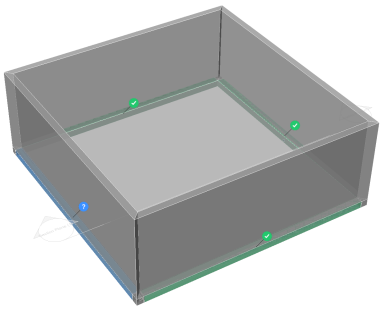
It can be used to copy detailed connections between two or more planar entities.

Note: The detail area to be copied is displayed in blue. The reference contact area is displayed in purple.



You can accept or reject suggestions by clicking on the tick.

- A green check mark means that the suggestion will be applied.
- A question mark means that the suggestion will not be applied due to one of several possible reasons.
- A red X mark means that the suggestion will not be applied.



7.44.2 Options within the command

Inflate first

Expands the sensitive zone to include similar solids which are not connected yet in the model. The edges of the initially detected detail are moved outward over a specified distance.

Save detail

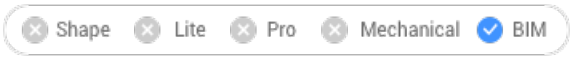
Displays the Save Drawing As dialog box, which allows you to save the detail.

save detail to Library

Displays the Save Detail dialog box, which allows you to save the detail to library.

7.45 BIMPROPERTIES command

Opens the **BIM Properties** dialog box.



Icon: 

7.45.1 Description

Opens the **BIM Properties** dialog box to create and manage BIM property definitions.

7.46 BIMPYTHON command

Allows Python Scripting and Querying of a BIM model.

Shape Lite Pro Mechanical BIM

7.46.1 Method

Opens the Standard Open File - Python script dialog box and runs the chosen script.

Note: Python version shipped with BricsCAD is increased to 3.9.6.

7.47 BIMQUICKBUILDING command

Transforms solids into buildings.

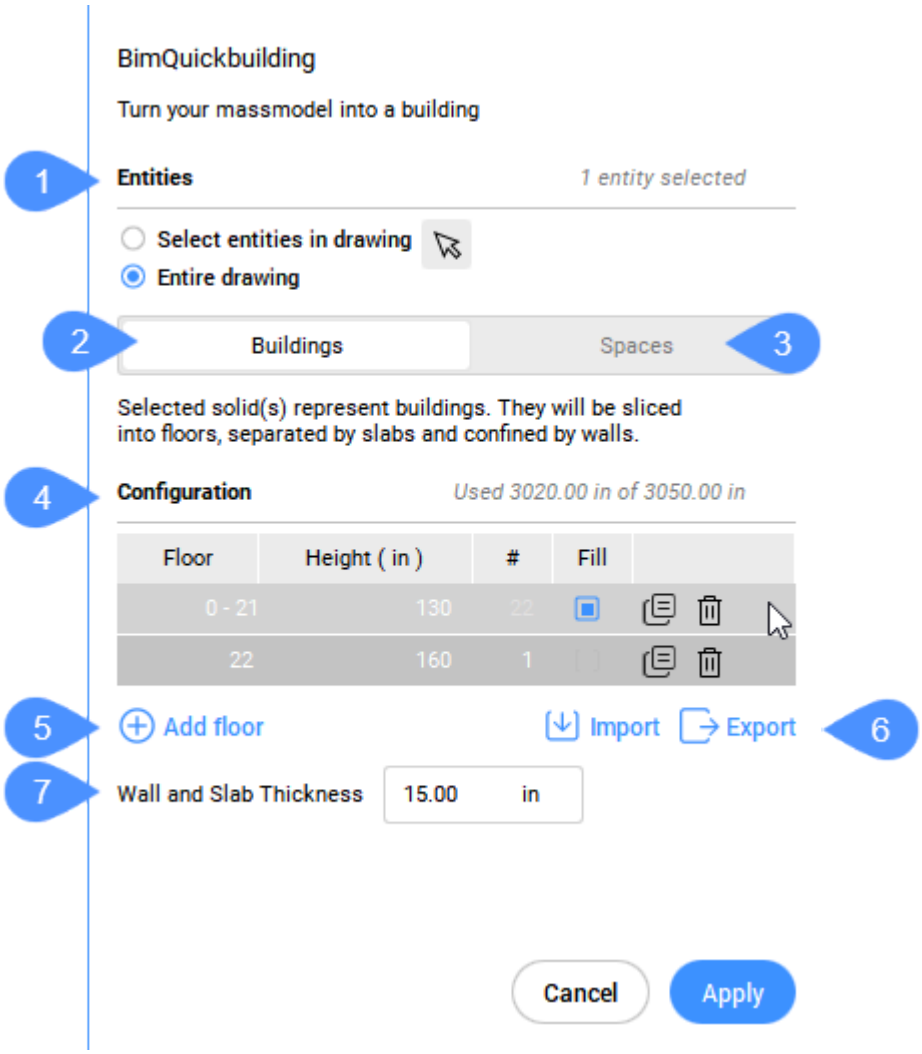
Shape Lite Pro Mechanical BIM

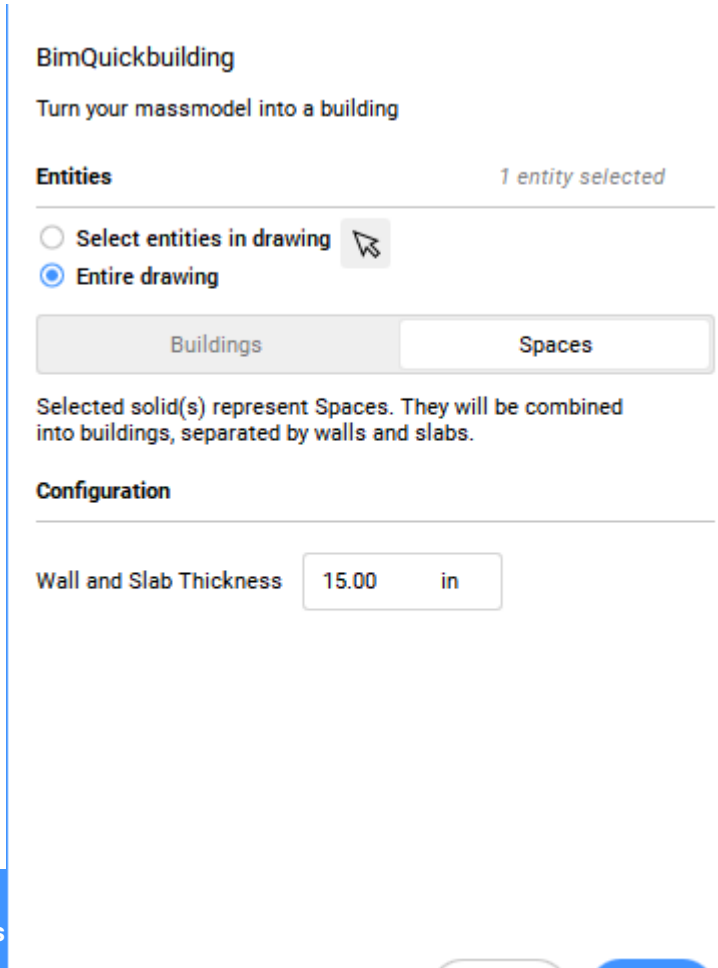
7.47.1 Description

Creates a separate drawing for the generated BIM building and in this drawing walls, slabs and roofs, as well as spatial elements such as floors and spaces will be automatically created.

This command also opens the **BimQuickBuilding** command panel.

7.47.2 BimQuickBuilding command panel


BimQuickbuilding
Turn your massmodel into a building
1
Entities
1 entity selected
Select entities in drawing
Entire drawing
2
Buildings
Spaces
3
Selected solid(s) represent buildings. They will be sliced into floors, separated by slabs and confined by walls.
4
Configuration
Used 3020.00 in of 3050.00 in
Floor
Height (in)
#
Fill
0 - 21
130
22
22
160
1
5
Add floor
Import
Export
6
7
Wall and Slab Thickness
15.00
in
Cancel
Apply


BimQuickbuilding
Turn your massmodel into a building
Entities
1 entity selected
Select entities in drawing
Entire drawing
Buildings
Spaces
Selected solid(s) represent Spaces. They will be combined into buildings, separated by walls and slabs.
Configuration
Wall and Slab Thickness
15.00
in

2 Building mode

3 Spaces mode

4 Configuration

5 Add floor

6 Import / Export

7 Wall and Slab Thickness

Select entities:

- **Select entities in the drawing**: choose which entities to select.
- **Entire drawing**: by default, the panel selects all 3D solids in the model space.

Buildings mode

Overlapping input solids will be unionized and for each resulting solid a building will be created. Additional input is needed to divide the building in multiple storeys. They will be sliced into floors, separated by slabs and confined by walls.

Configuration

Displays the elevation table.

- **Floor**: displays the storey number.
- **Height**: sets and displays the storey height (elevation value).
- **#**: sets and displays the number of equal storeys height (number of floors).
- **Fill**: if checked, an automatic number of storeys are generated according to the elevation input (**Height**) value and the remaining available height (elevation value).
- : duplicates a storey.
- : deletes a storey.

Add floor

Adds a storey.

Import / Export

Imports from or exports to an elevation table as .csv format file.

Wall and Slab Thickness

Sets and displays the value of the wall and slab thickness.

Note: The elevation input (**Height**) and **Wall and Slab Thickness** respects the insertion units of the drawing.

Spaces mode

The individual input solids will be considered as spaces in a building. For each group of touching solids, a building will be created. Each resulting building will consist of floor slabs and inner walls, corresponding to the touching face pairs of the input solids.

7.47.3 Options within the command line

change Selection

Select entities in the drawing, as, by default, the entire drawing is selected.

Import
Uses a .csv or .txt file to configure the storey heights of the building.

Note: For instance, a text line for 2 storeys configuration can be:

```
Floor;Height- mm;Amount;Fill
 0 - 10;3250.000000;11;Yes
 12 - 16;5000.000000;5;No
```

sPaces
Generates spaces for buildings.

7.48 BIMQUICKDRAW command

Creates and edits spaces and buildings.

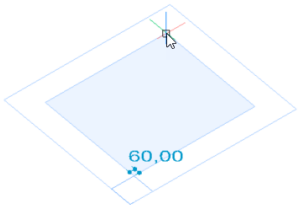
Shape Lite Pro Mechanical BIM

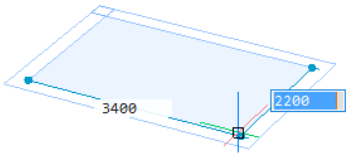
Icon:

7.48.1 Description

Quickly creates and edits rectangular and L-shaped spaces and buildings. The layout of the QuickDraw cursor represents the space being created. It also indicates the alignment with existing walls and distances from existing solids.

Note: If dynamic input (DYN) is set ON, the dimensions are displayed, as the spaces are created. The dimensions can also be entered manually.





Note: The default settings can be changed in the Settings dialog box.

7.48.2 Options

Floor to Floor Distance (Floor Height)
Sets the floor to floor distance. The value is saved to the FLOORTOFLOORDISTANCE system variable and will be used until set differently.

Note: The default value is 3250mm.

Wall Width
Sets the width of the walls being created. The width is saved to the WALLWIDTH system variable and will be used until set differently.

Note: The default value is 250mm.

Slab thickness
Sets the slab thickness of the room. The thickness is saved to the SLABTHICKNESS system variable and will be used until set differently.

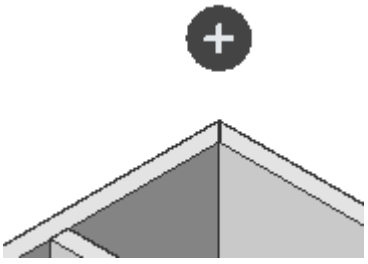
Note: The default value is 250mm.

Auto adapt sizes
Allows to copy the wall width and height from an existing wall, thus overriding the current value of the WALLWIDTH system variable.

- ON: The width and height are copied from the highlighted wall when you place the QuickDraw cursor against a wall to specify the first corner of a new room. When two walls of different height or width highlight (corner), the dimensions of the wall that highlights first are copied.
- OFF: The values of the FLOORTOFLOORDISTANCE and WALLWIDTH system variables are used.

Add story
Adds a story to the existing building:

- 
  Full copy of the top floor.
- 
  Copies the slab and the outer walls of the top floor.
- 
  Finish the building with flat roof.



7.49 BIMREASSOCIATE command

Automatically re-associates invalid Tags and Dimensions to the underlying geometry.

7.49.1 Description

Select a BIM section viewport in order to automatically re-assign invalid Tags and Dimensions to the underlying geometry.

Note: The BIM "Associativity" property indicates whether a tag is associated.

Note: The BIM "Associative coloring" property visualizes an unassociated tag in red, if true. It doesn't change the color property.

7.50 BIMRECALCULATEAXIS command

Recalculates and repositions the axis of linear building elements.

Shape Lite Pro Mechanical BIM

Icon: 

7.50.1 Description

Recalculates the axis of linear building elements and repositions it back on the center line of a linear element. When the command is run, select all linear solids for which to recalculate the axis.

7.51 BIMREGENERATETAGS command

7.52 BIMROOM command

The BIMROOM command is obsolete and has been replaced by the BIMSPACE command.

Shape Lite Pro Mechanical BIM

Icon: 

Note: You can still open BIM rooms, but it is not possible to create new BIM rooms or update their geometry.

Note: Use the Space option of the BIMCLASSIFY command to convert a BIM room to a BIM space.

7.53 BIMSECTION command

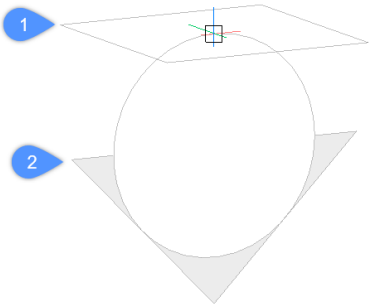
Creates a BIM section entity.

Shape Lite Pro Mechanical BIM

Icon: 

7.53.1 Description

Creates a BIM section plane entity (1) in the XY-plane of the current coordinate system (WCS or UCS),with the view direction (2) in the negative Z-direction of the current coordinate system.



Note:

If dynamic UCS (UCSDETECT) is ON, the section plane aligns to the face of a 3D solid under the cursor.

BIM section entities are created on a dedicated BIM_SECTIONS layer.

7.53.2 Methods

There are three types of BIM section entities that can be created using the BIMSECTION command:

- Plan section type: displays a horizontal section plane.
- Section section type: displays a vertical section that cuts though the model.
- Elevation section type: displays the exterior elevation view.

Note: You can change the BIM section type in the Properties panel.

Note: The GENERATEASSOCVIEWS system variable controls if VIEWSECTION produces 2D drawings which are persistently associated with the source 3D model. If GENERATEASSOCVIEWS is ON, associative dimensions are updated automatically when the 3D model is modified and BIMSECTIONUPDATE is executed.

7.53.3 Options

Select a point to place section
Specify a point.

Note: The section plane displays dynamically parallel to UCS XY-plane, through the cursor position.

Specify distance
Type a distance or specify a point.

Note: It is recommended to have Dynamic Dimensions (DYN) active. It allows to enter the distance in the dynamic entry field.

Note: The section plane is defined through the point or at the specified offset from the first point. The section line is parallel to the X-axis of the UCS or dynamic UCS and through the point at the specified offset from the first point.

Note: Only the section line and the callouts of a BIM Section entity display. When highlighted or selected the section plane, section boundary and/or section volume display.

set clipping oN
Sets the clip display property to ON.

Note: This option is set by default. If the Hot Key Assistant (HKA) system variable is ON, press the Ctrl key to toggle the clip display property off.

Note: Multiple sections can have the clip display ON simultaneously. This property can be saved in a model view (see the VIEW command).

set clipping Off
Sets the clip display property to OFF.

Note: You can change the clip display property in the Properties panel.

Detail
Creates a Detail section type.

Based on an existing section
Creates a Detail section type based on an existing section. The base rectangle of the detail section will be parallel to the section plane of the selected section.

Interior
Creates interior elevations and a floor plan of a space.

Note: The BIM/Interior elevation property of a space is set to ON. Use the BIMSECTIONUPDATE command to update the interior elevation.

Scale
Sets the Scale property of the viewport in the drawing file created by the BIMSECTIONUPDATE command.

Note: The default Scale is saved through the SECTIONSCALE user preference (default value is 0.02) in the Settings dialog box.

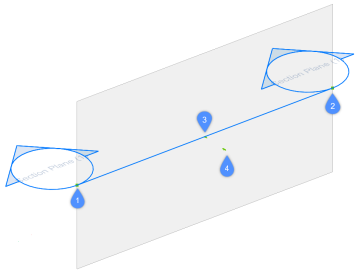
Reflected ceiling
Creates the plan of a ceiling projected onto the section plane.

Note: A reflected ceiling plan shows the size and location of light fixtures and other structures on the ceiling. The clip display property of a reflected ceiling section is OFF by default.

7.53.4 Grips editing

BIM section entities can be edited through grips, depending on their state.

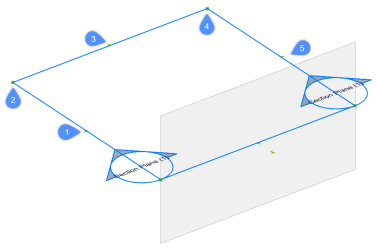
- Plane state:
-



  - Start grip (1) allows to move the section and the position of the Start callout.
  - End grip (2) allows to modify the orientation of the section and the position of the End callout.
  - Center grip (3) allows to move the section and the position of the Mid callout.

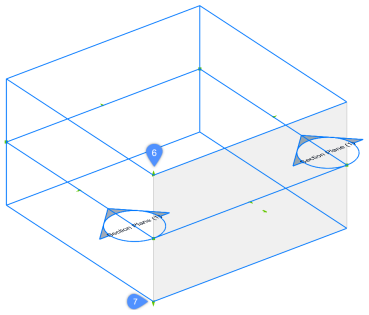
- The arrow (4) flips the view direction.

- Boundary State:



- Midpoint grip (1) allows to move the section entity.
- Corner grip (2 & 4) allows to deform the section boundary/volume.
- Midpoint grip (3 & 5) allows to stretch the section boundary/volume.

- Volume State:



- (6) allows to move the top plane of the section entity.
- (7) allows to move the bottom plane of the section entity.

7.54 BIMSECTIONOPEN command

Opens the drawing file related to a BIM section entity.

Shape Lite Pro Mechanical BIM

Icon:

7.54.1 Description

Opens the drawing file related to a BIM section entity, or the 3D BIM model related to a BIM section drawing.

7.54.2 Method

There are two methods to use BIMSECTIONOPEN command:

- Select a BIM section entity in Model Space.
- Select a BIM section result (a viewport) in Paper Space.

7.55 BIMSECTIONUPDATE command

Updates the result of a BIM section entity.

Shape Lite Pro Mechanical BIM

Icon: 

7.55.1 Description

Generates or updates the result of a BIM section in a drawing.

Note: The GENERATEASSOCATTRS system variable controls the generation of associative data on 3D solids during modeling. When set ON, 3D solids contain associative data from the moment of creation. This allows BIMSECTIONUPDATE to produce drawings for which dimensions and tags can be automatically updated when the 3D model has been modified.

Note: When GENERATEASSOCVIEWS system variable is ON, the BIMSECTIONUPDATE command will generate/update the section indicators, grid callouts, and story callouts.

Tags are no longer automatically generated with BIMSectionUpdate command.

Story indicators, section indicators and grid callouts are generated in paper space now and essentially are BIM tags.

7.55.2 Method

There are two methods to use the BIMSECTIONUPDATE command:

- Generate the result of a BIM section entity.

  Note: If the **Interior** option of the BIMSECTION command was used during the creation of the BIM section, BIM spaces can also be selected.

- Update the result of a BIM section.

  Note: Select a BIM section entity in the Model Space or a viewport in the Paper Space layout.

7.56 BIMSETLOADBEARINGDIRECTION command

Sets the loadbearing direction for BIM slab solids.

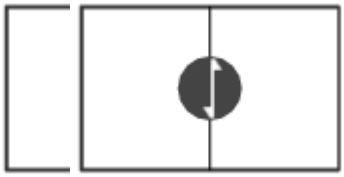
Shape Lite Pro Mechanical BIM

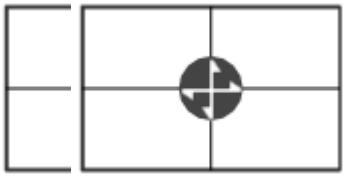
Icon: 

7.56.1 Method

An icon is displayed in the center of the slab after selecting a slab solid. Set the loadbearing direction for BIM slab solids by clicking on the icon.

None	Short	Long	Both





7.57 BIMSETREFERENCEFACE command

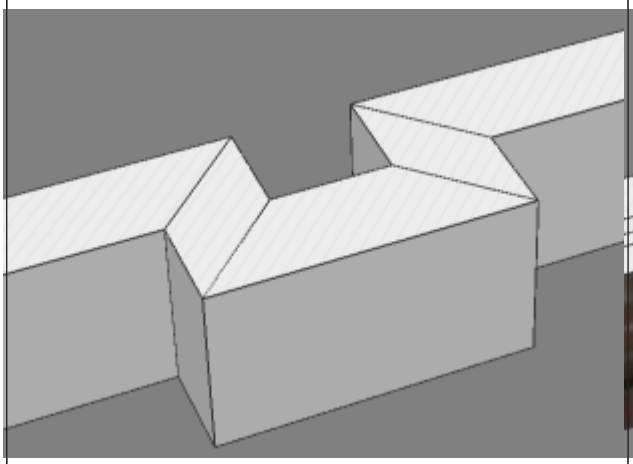
Defines a reference and an opposing face to control the ply layout of a composition.

Shape Lite Pro Mechanical BIM

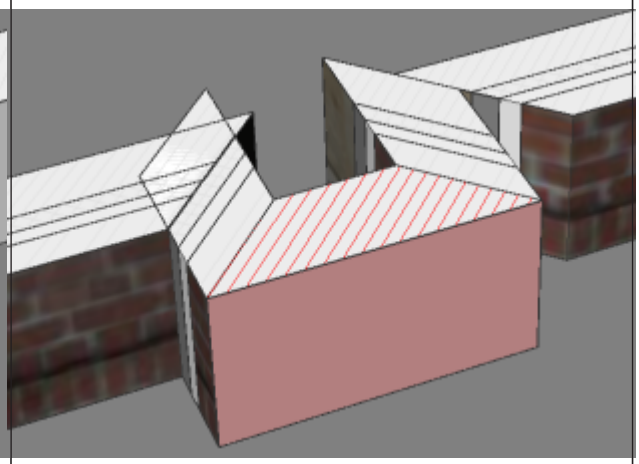
7.57.1 Methods

The reference face can be set manually by selecting a face. If the opposing face is parallel to the reference face, a suggestion for the opposing face is given and can either be accepted or be changed.

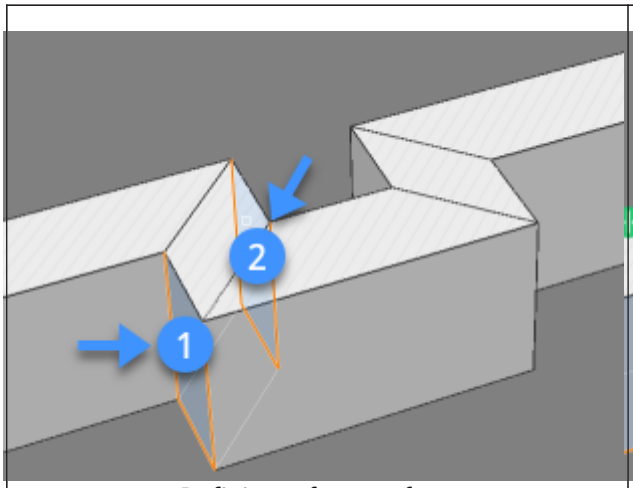
Note: If one of the plies of the composition has a variable thickness, the reference and opposing face can be non-parallel. In this case the fixed thickness plies are set out starting from the reference face, and the remainder of the solid is filled by the variable ply.



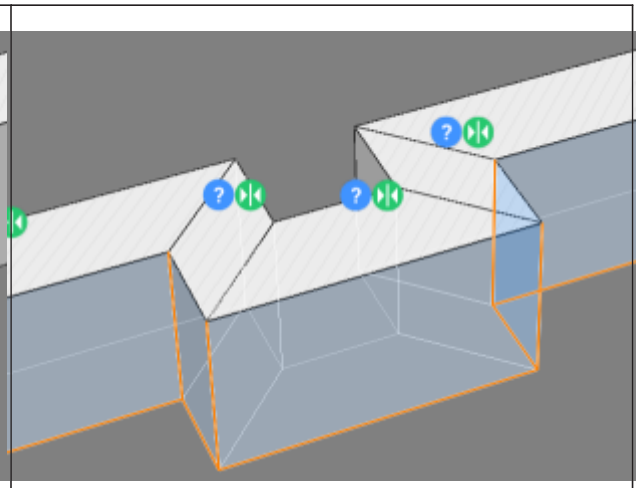
Solids without composition



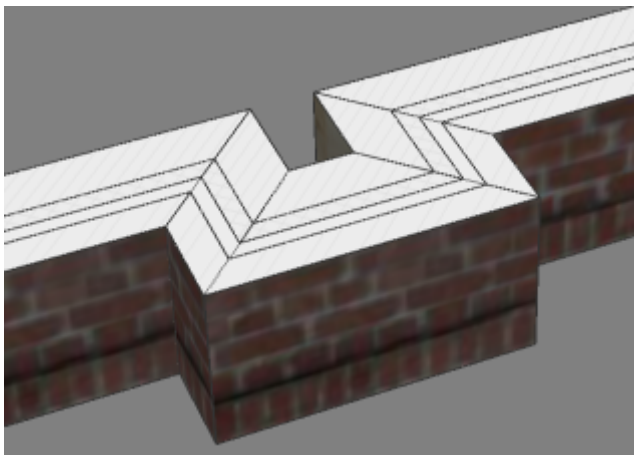
Multiply composition assigned
Reference faces are either wrong or not found



Defining reference face
(1) and opposing face (2)



Attaching a composition
Reference faces highlight



Result

7.57.2 Options

Select manually
Selects the opposite face manually.

7.58 BIMSPACE command

Creates BIM Space entities from enclosed boundaries.

Shape Lite Pro Mechanical BIM

Icon:

7.58.1 Description

Creates BIM Space entities by picking a point inside an enclosed area defined by space bounding entities.

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 Yes. If the Centerline property of a space bounding solid is set to Yes, the center of the solid is used as the space boundary.

A label is created at the geometric center of space.

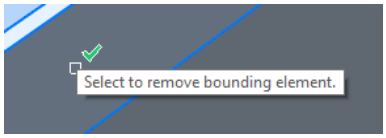
Note: The Space option of the BIMCLASSIFY command converts any 2D or 3D entity into a BIM Space entity.

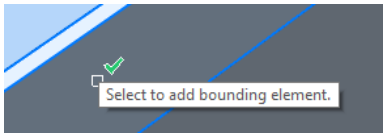
The height of the space is defined by:

- The top face of the highest bounding entity.
- The Default Space Height user preference , when all bounding entities are 2D linear entities.

7.58.2 Options

Edit
Launches the Edit mode, which allows to add or remove space bounding entities in a Space by clicking them.





Note: After editing, launch the BIMUPDATESPACE command to update the space.

Note: If the update Method property of the space is Manually, the space cannot be updated by the BIMUPDATESPACE command. This can be solved by setting this property to Automatic.

Note: The BIM Spaces properties can be edited in the Properties panel.

BIM Space Properties

Representation
Sets the visual representation of the BIM Space:

- Footprint: Space stamp only.
- Solid: Transparent 3D solid. The height of the solid is copied from the surrounding wall solids.

Status
Specifies the status of the Space.

Note: If the space is out-of-date, an exclamation mark icon is displayed next to the space stamp. This can be solved with the BIMUPDATESPACE command.

Name
Sets the name of the space, which is also visible on the space stamp.

Description
Describes the space.

Building
Specifies the building to which the space is assigned to.

Story
Specifies the story to which the space is assigned to.

Interior elevations
Controls the display of Interior Elevations. If interior elevations are not created yet, the property is disabled.

Note: Additional properties can be created using the BIMPROPERTIES command.

7.59 BIMSPATIALLOCATIONS command

Allows to create and edit site, buildings and stories.

Shape Lite Pro Mechanical BIM

Icon: 

7.59.1 Description

Displays the SPATIAL LOCATIONS MANAGER dialog box .

7.59.2 Options

Dialog
Opens the SPATIAL LOCATIONS MANAGER dialog box.

add Building
Adds a new building to the model.

Rename Building
Renames a building.

add Story
Adds a new story in the selected building.

Rename Story
Renames a story.

sEt story elevation
Sets an elevation for an existing story by entering the elevation value or select a point.

Point
Sets the point for story elevation.

7.60 BIMSPLIT command

Splits segmented solids into separated solids.

Shape Lite Pro Mechanical BIM

Icon: 

7.60.1 Description

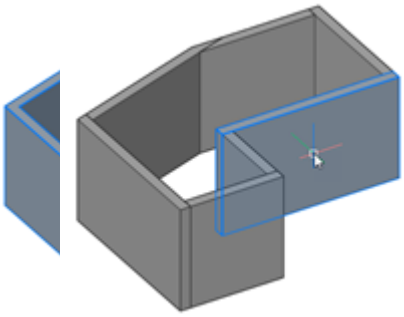
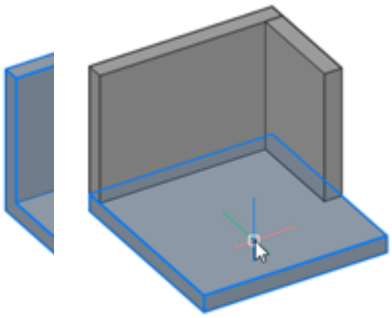
Automatically splits segmented solids into separate solids and allows to split a solid using a selection of cutting faces. Specifically designed to recognize planar & linear shapes such as walls, slabs & columns it can help you split your mass model into separate entities, which is necessary for a typical BIM workflow.

7.60.2 Method

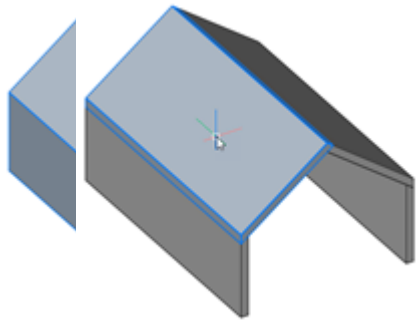
Select a solid for auto splitting

One method is to select an entire solid and let BIMSPLIT decide automatically where the solid should be cut. This will be done along logical cutting faces, yielding results as shown in the table below.

Note: For complex geometry BIMSPLIT may not give the expected result. In that case, try simplifying by first splitting the geometry into smaller parts manually.

Solid selected…	**…split into 7 separate solids**
	
Solid selected…	**…split into 3 separate solids**
	

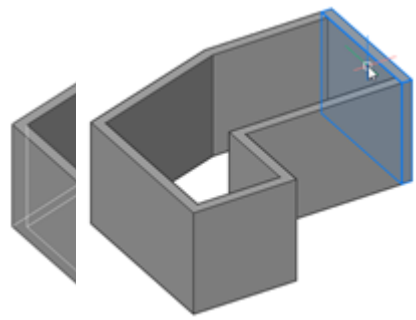
Solid selected ...	**...split into 4 separate solids**

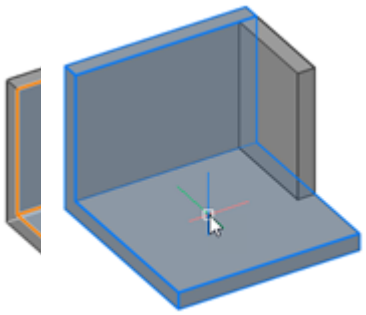


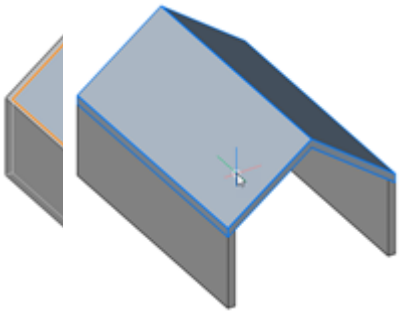
Select cutting faces for manual splitting

Another method is to select cutting faces manually. This behavior is similar to the SLICE command, the main difference being that it is possible to select multiple cutting faces. Adjacent cutting faces will keep their respective parts of the solids intact.

One cutting face selected ...	**... split into 2 separate solids**



Two cutting faces selected...	...split into 2 separate solids
	

Two cutting faces selected...	...split into 3 separate solids
	

Note: Inserts, such as windows and doors, which existed in the split solid are removed.

7.61 BIMSTAIR command

Creates stairs.

Shape Lite Pro Mechanical BIM

Icon:

7.61.1 Description

Creates stairs between 2 floor slabs or any empty space and it will create a stair. This command is using the default floor-to-floor-distance as height.

The stairs display automatically when the cursor is moved. The type of stair being created depends on the position of the cursor with respect to the position of the selected start point.

Note: The staircase is created as a parametric block. The Stair Type is assigned automatically and can be edited in the Properties panel. Editing this property does not affect the geometry of the staircase.

7.61.2 Methods

There are two methods to create stairs using the BIMSTAIR command:

- Select a horizontal face of a solid.
- ***Note:*** If both the start plane and the end plane of a stair are horizontal faces of a solid, the stair height is set to the difference between Z-values of start plane and the end plane.
- Select a void area.
- ***Note:*** If the start plane of a stair is a horizontal face of a solid and the end plane is a void area, the stair height is set to the difference between Z-values of start plane and the end plane. If the start plane is also a void area, then the stair height is set to the current value of the FLOORTOFLOORDISTANCE system variable.

7.61.3 Options

Quarter-turn
Turns the direction of the stair cursor with 90 degrees.

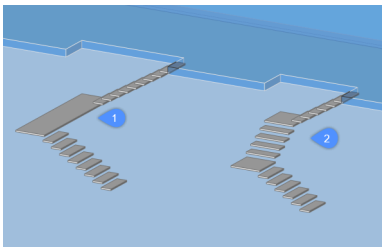
Note: Press Ctrl key to turn the stair cursor with 90 degrees.

Flip
Turns the direction of the stair cursor with 180 degrees.

Note: Press Ctrl key to turn the stair cursor with 180 degrees.

U-shape type
Switches between two different U-shape type stairs:

- • Double-flight U-shaped stairs (1): with a single rectangular landing solid bridging the perpendicular distance between both opposing stair flights.
- • Triple-flight U-shaped stairs (2): with two square landing solids between each pair of subsequent stair flights.

- 

Note: Press Ctrl key to cycle the possible stair types with respect to the current position of the stair cursor.

Note: The HOTKEYASSISTENT must be set ON to use Ctrl key functions.

Stair settings

Opens the Settings dialog box, where the parameters of the stair can be set.

- **Stair width:**Default width of the stairs
- **Preferred tread length**: Preferred tread length of the stairs
- **Ideal riser height**: Ideal riser height of the stairs
- **Head room**: Sets the minimum clearance for the opening to be created
- **Step thickness**: Thickness of the individual steps
- **Nosing**: Horizontal distance of overlap between subsequent treads
- **Final tread length**: Length of the final tread, this always in regard to the top tread of the stair
- **Automatically flip/quarter-turn**: Turn on if you want to automatically flip or turn when you hover over the slab with your cursor and snap the blue StairCursor to the walls (the end of the arrow can then never be facing a wall)
- **Final tread**: Controls creation of a tread at the elevation of the top bounding slab.

Note: For all properties, parameters will be created when creating the stair, driving the constraints of the parametric stair block. These parameters can be then changed individually afterward in the Parameters section of the Properties panel.

7.62 BIMSTRETCH command

Stretches the ends of linear solids.

Shape Lite Pro Mechanical BIM

7.62.1 Description

Stretches the ends of linear solids while maintaining previously made connections between other linear solids in the overall structure.

7.62.2 Method

There are two methods to use BIMSTRETCH command:

- Stretch structural connections.

Note: Stretching elements that are classified as Beam, Column or Member.

- Stretch MEP connections.

Note: Stretching elements that are classified as Flow segments.

Before using the BIMSTRETCH command make sure that:

- 'Display sides and ends' is enabled.
- 'Display Axes' is enabled.
- Structural profiles are connected with StructuralConnect.
- MEP connections are connected with FlowConnect.

Note: Before running the command, select the ends of the entities that will be modified.

7.62.3 Options within the command

Base Point
Specifies new basepoint to start the displacement from.

Copy
Makes a copy of the connected entities and places them in the new position.

Undo
Undoes the last action.

Exit
Exits the command.

7.63 BIMSTRUCTURALCONNECT command

Connects structural profiles.

Shape Lite Pro Mechanical BIM

Icon: 

7.63.1 Description

Connects structural profiles such as beams, columns and members.

7.63.2 Method

There are two methods to use BIMSTRUCTURALCONNECT command:

- Select two solids.
- Select multiple solids.

Note: The axes of the solids must be coplanar.

7.63.3 Options

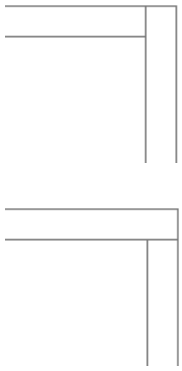
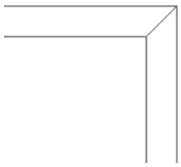
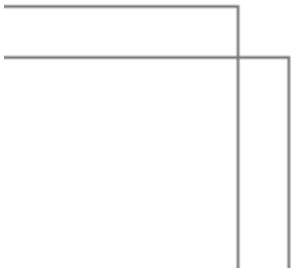
Select base solid
Specifies the base solid.

Note: : This option is only available for L-connection.

Switch

Allows to choose between different types of L-connections:

- Butted connection
- Mitered L- connection
- Touching/disconnected connection

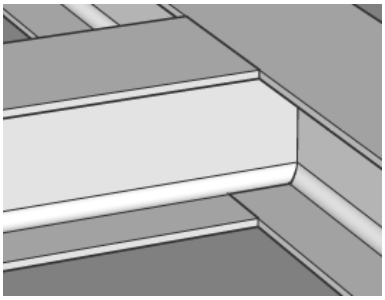
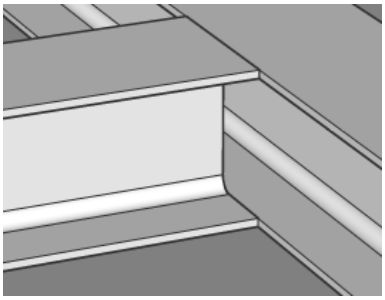
Butted Connection	Mitered L-connection	Touching connection
		

Note: : Press the Crtl key to cycle between the connection types. Hot Key Assistant must be set ON.

Note: If a T-connection is created, there are no options available.

Connection cut type

Depending on the DMCONNECTIONCUTTYPE system variable value, the connection type can be smooth or planar.

Smooth connection	Planar connection
	

7.64 BIMTAG command

Creates associative tags.

Shape Lite Pro Mechanical BIM

Icon: 

7.64.1 Description

Creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model.

Note: This command can only be used on section viewports.

Note: The GENERATEASSOCVIEWS system variable must be ON when generating or updating BIM tags.

7.64.2 Method

The tags are created automatically for all building elements in the selected viewport.

Note: Tags are created as multileader entities using multileader styles in _SectionTag.dwg. The content of such a multileader style can be a field referring to a property of BIM entity type or a block with attributes. Each attribute refers to a property of a BIM entity type. BIM entity types are linked to a multileader style in _TagTypeToStyle.xml. All source files can be found in the Support folder, for example: *C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V22x64\en_US\Support\Bim\Sections*.

Note: Section indicators, story indicators and grid callouts are generated automatically within section update.

7.64.3 Options within the command

Auto
Creates tags automatically for all sectioned building entities in the selected viewport, including section indicators, story indicators and grid callouts.

Manual
Creates tags for selected entities in sections and elevations.

Box selection
Creates tags for all entities selected in sections and elevations.

Note: Both Manual and Box Selection allow you to tag background entities.

Current mleader style
Allows to use the current multileader style instead of the default style for selected sectioned BIM elements.

change Tag type
Changes the type of the selected tags.

cOmposition
Tag with all materials of composition of the sectioned BIM element.

Load-bearing directions
Inserts a loadbearing direction symbol when a generated line of a slab solid is selected. Make sure the load bearing direction is set by the BIMSETLOADBEARINGDIRECTION command.

Spot-elevation
Inserts a tag that indicates the elevation of the identified point in elevations and vertical sections.

Note: The following options are available only if tags have already been assigned to the section.

Update existing
Updates the existing tags in the selected viewport. The position of relocated tag blocks is preserved.

update existing and add New
Updates the existing tags in the selected viewport and adds new tags.

Regenerate all
Updates all tags in the selected viewport. Repositioned tag blocks are moved to their default location.

7.65 BIMUPDATESPACE command

Recalculates a BIM space which is out-of-date.

Shape Lite Pro Mechanical BIM

Icon: 

7.65.1 Description

Recalculates a BIM space which is out-of-date, for example, after adding or removing bounding entities.

After the command is run, select all spaces to update.

Note: Only spaces that have the Update Method property set to Automatic can be processed.

7.66 BIMUPDATETHICKNESS command

Re-applies the overall thickness of a composition to the solid.

Shape Lite Pro Mechanical BIM

Icon: 

7.66.1 Description

Re-applies the overall thickness of a composition to the solid by selecting the solid.

Note: Only solids that need to be updated are processed.

7.66.2 Options within the command

Symmetrical Update
Allows to update the thickness symmetrically or with respect to the reference face (See BIMATTACHCOM-POSITION command).

Note: If the thickness is updated symmetrically, the position of the axis of the solid is preserved.

7.67 BIMWINDOWCREATE command

Creates a fully parametric window or opening based on a closed contour.

Shape Lite Pro Mechanical BIM

Icon: 

7.67.1 Description

Creates parametric windows using as profile any entity such as a boundary, a closed polyline or grid.

Note: A series of parameters is created, which can be edited in the Parameters section of thePropertiespanel.

7.67.2 Options within the command

Window

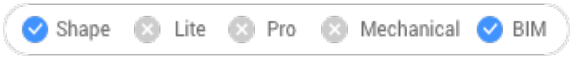
Creates a BIM Window entity. An opening in the solid on which the profile entity lies is created automatically. The Choose Window Style dialog box opens and the windows style can be chosen.

Opening

Creates an opening in the solid on which the profile entity lies. The opening is created by a solid on the BIM_SUBTRACT layer. This layer is frozen by default. Thaw the layer to select the opening.

7.68 -BIMWINDOWCREATE command

Creates a fully parametric window or opening based on a closed contour.



7.68.1 Description

Creates parametric windows using as profile any entity such as a boundary, a closed polyline or grid.

Note: This command operates at the Command line.

7.68.2 Options

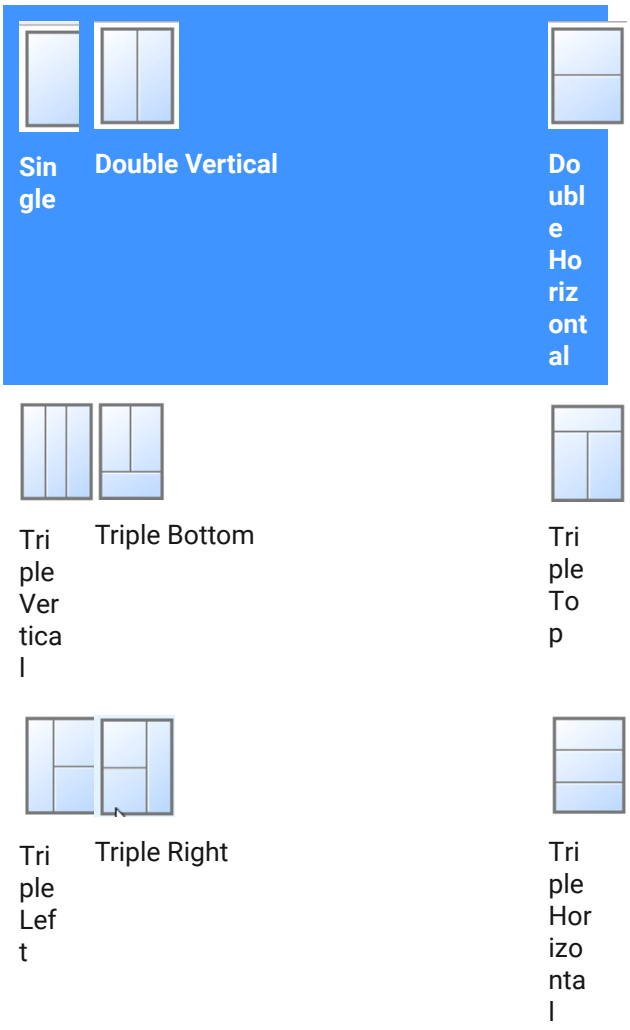
wIndow

Creates a BIM Window entity. An opening in the solid on which the profile entity lies is created automatically.

Opening

Creates an opening in the solid on which the profile entity lies. The opening is created by a solid on the BIM_SUBTRACT layer. This layer is frozen by default. Thaw the layer to select the opening.

Panel type



7.69 BIMWINDOWPRINT command

7.70 BIMWINDOWUPDATE command

Updates the opening made by a window or door in a solid.

Shape Lite Pro Mechanical BIM

Icon:

7.70.1 Description

Updates the opening created by a window or door in case the definition changed in such way that the opening was not correctly updated automatically.

7.70.2 Options within the command

MOde

Allows to select the placement face, whether Automatic or Manual.

7.71 BLADE command

Opens the BricsCAD LISP Advanced Development Environment (BLADE).

Shape Lite Pro Mechanical BIM

7.71.1 Description

Opens the BricsCAD LISP Advanced Development Environment (BLADE) to interactively edit and debug LISP applications. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

7.72 BLCOMPOSITIONS command

Opens the Compositions dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

7.72.1 Description

Open the Compositions dialog box to create and manage BIM compositions.

7.73 BLIPMODE command

Toggles the BLIPMODE system variable.

Shape Lite Pro Mechanical BIM

Alias: BM

7.73.1 Description

Toggles the BLIPMODE system variable to show or hide marker blips that indicate where you picked in the drawing. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'BLIPMODE.

- On: turns on the BLIPMODE system variable
- Off: turns off the BLIPMODE system variable
- Toggle: changes the BLIPMODE system variable to the opposite of the current setting

7.74 BLMATERIALS command

Opens the Physical Materials dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

7.74.1 Description

Opens the Physical Materials dialog box to create and edit material definitions in the library or project.

7.75 BLOCK command

Opens the Block definition dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: B

7.75.1 Description

Opens the Block definition dialog box to create a block definition in the current drawing.

7.76 -BLOCK command

Groups entities into a block.

Shape Lite Pro Mechanical BIM

Alias: -B

7.76.1 Method

Specify whether the block will follow annotative scaling and match the orientation of the viewport.

Note: Optionally, include REFERENCECURVES during the selection of entities which allows to automatically align a block during insertion.

7.76.2 Options within the command

Name for new block
Specifies the name of a new block.

?
Lists the names of blocks already in the drawing, so that an existing name won't be used.

Note: If an existing name is entered, BricsCAD asks you if you want to redefine the block. If it is redefined, all the instances of that block name will be changed.

Insertion point for new block
Pick a point in the drawing or enter X, Y and Z coordinates.

Note: The Z coordinate is optional, default is 0.

Annotative
Specifies if the block will follow annotative scaling.

Orient relative to sheet in paper space viewports
Determines if the annotative blocks match the orientation of the layout.

7.77 BLOCKICON command

Creates and updates preview bitmaps of blocks and stores them in the drawing.

Shape Lite Pro Mechanical BIM

7.77.1 Method

Type the name(s) of the block(s) to be processed. Separate block names by commas. You can use wild cards (? Or *).

7.77.2 Options within the command

*

Select all blocks.

7.78 BLOCKIFY command

Converts identical set of entities (2D or 3D) to block references.

Shape Lite Pro Mechanical BIM

Icon: 

7.78.1 Description

Searches the drawing for an identical set of entities (2D or 3D) and replaces them with block references.

7.78.2 Method

There are three methods to use BLOCKIFY command:

- Convert 2D and 3D entities to blocks.
- Convert equally shaped solids to blocks.
- Optionally replace block patterns by arrays.

7.78.3 Options within the command

match Equal solids

Searches the drawing (or the selection) for equally shaped solids and replaces them with newly created block definitions. In case a solid matches an existing block definition, the replacing block reference points to that block definition.

Note: This option is not available in the Lite license level.

Specify block name or <ENTER to use default name>

Specify a block name or use the default name. If you choose to use the default name, block definitions with names like **Block1**, **Block2** etc. are created.

Note: If BLOCKIFY is used on BIM entities, it checks the BIM Name property of all equal-shaped BIM classified solids and tries to find the longest common substring to use it as the name of the block definitions.

find Collections

Identifies collections and eventually nested blocks are created. This option opens the Find Collections panel.



Note:

- This option only works on block references. If the model also contains other entities, these will not be taken into consideration.
- With a large number of input entities, calculating all possible collections can take a long time. Press the Esc key to cancel the calculation at any time, and choose from the collections found so far.

Maximum gap

Increases the allowed gap between entities to find a more complex collection.

For example: If a drawing contains multiple block references of desks and chairs, you can find a collection of both elements by setting the maximum gap between a desk and a chair.

Recalculate
Recalculates the collections based on the new maximum gap.

Collections
Displays the found collections.

Note: A message is displayed if the model contains overlapping collections. Overlapping collections are collections that have at least one solid in common. These collections are highlighted if the **Highlight overlapping collections** option is ticked.

Details
Tick the **Show** option to highlight one collection at a time and cycle through them by using the arrows. Use the **zoom in** button to extend the view to the shown collection.

Remove
Removes the current collections.

Reset
Resets to the initially found collections, by re-adding any collections that were removed.

Note: After creating a block reference, the collections are recalculated, taking into account the newly created block reference. The solids that are part of the newly created block reference could previously appear in a different collection, but now they are part of a single block reference and always have to appear together. This can decrease the choices for creating more collections.

match existing Blocks
Searches the drawing (or the selection) for 2D geometry matching existing block definitions.

Settings
Opens the Settings dialog box at the blockify settings.

Note: Selecting search space prompt is only available if **Use entire drawing as search space** is not ticked.

Note: Selecting insertion point prompt is only available if **Use default block insertion point** is not ticked.

Note: Specifying block name prompt is only available if **Use default block name** is not ticked.

7.79 BLOCKREPLACE command

Replaces block definitions.

Shape Lite Pro Mechanical BIM

Icon:

7.79.1 Method

This command displays the **Block Replace** dialog box.

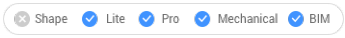
Note:

- Modified (anonymous) instances of dynamic blocks cannot be selected by name.
- Externally dependent blocks cannot be replaced.

- When a block with attributes is replaced by a block without attributes, the replaced block reference keeps the old attributes.
- When a block without attributes is replaced by a block with attributes, the replaced block references doesn't insert new attributes.
- When an annotative block is replaced with a non-annotative block, the block reference keeps the old context data manager and annotation scales in the extension dictionary.
- When a non-annotative block is replaced with an annotative block, BricsCAD® adds context data immediately according to CANNOSCALE.
- When a dynamic block is replaced by a non-dynamic block or another dynamic block, the old extension dictionary **AcDbBlockRepresentation** is removed from the block reference.

7.80 -BLOCKREPLACE command

Replaces block definitions.



7.80.1 Method

There are two methods to replace block definitions:

- By replacing all block instances.
- By replacing particular block instances.

Note:

- Modified (anonymous) instances of dynamic blocks cannot be selected by name.
- Externally dependent blocks cannot be replaced.
- When a block with attributes is replaced by a block without attributes, the replaced block reference retains the old attributes.
- When a block without attributes is replaced by a block with attributes, the replaced block references doesn't insert new attributes.
- When an annotative block is replaced with a non-annotative block, the block reference keeps the old context data manager and annotation scales in the extension dictionary.
- When a non-annotative block is replaced with an annotative block, BricsCAD® adds context data immediately according to CANNOSCALE.
- When a dynamic block is replaced by a non-dynamic block or another dynamic block, the old extension dictionary **AcDbBlockRepresentation** is removed from the block reference.

7.80.2 Options within the command

Enter name of block to be replaced

Enter the name of the block to be replaced. Note that modified (anonymous) instances of dynamic blocks cannot be selected by name.

?

Lists the block names.

select object (=)
Select a block reference with the desired name. Note that modified (anonymous) dynamic blocks cannot be selected.

options (*)
Lists the replace options.

by block Name
Enter a block name to replace. Note that modified (anonymous) dynamic blocks cannot be selected by name.

Selected block instances
Selects any block instances, including modified (anonymous) dynamic block instances to be replaced.

SIMilar inserts
Replaces all modified (anonymous) and unmodified instances of dynamic blocks and all instances of non-dynamic blocks selected.

Enter block name or file to replace selected entities
Enter a block name or file to replace selected entities as follow:

- blockname: name of a block already defined in the current drawing;
- fileName.dwg: name of an external .dwg or .dxf file (file name will be used as a new block name);
- blockName=fileName.dwg: when new block name should be different than fileName.

Note: fileName.dwg can have blocks with the same names as in the current drawing. In this case blocks from file do not re-define blocks with the same names already present in the current drawing.

Note: The file name should include the entire path if the file is not located in the support file search path.

Enter a block name or a file name to replace selected entities. The file name should include the entire path if the file is not located in the support file search path.

Purge unreferenced when finished?
Removes unreferenced block instances.

7.81 BLOCKTOXREF command

Replaces all instances of a block with an Xref.

Shape Lite Pro Mechanical BIM

Icon: 

7.81.1 Method

Select one or more block references to replace with Xref/s.

This command displays the Block to Xref dialog box.

Note: The block name for the Xref is generated from the file name. If an Xref (or overlay) with the same name already exists in the drawing, the block is replaced using existing Xref.

7.82 -BLOCKTOXREF command

Replaces all instances of a block with an Xref at the command line.

Shape Lite Pro Mechanical BIM

7.82.1 Method

Select one or more block references to replace with Xref/s.

Note: The block name for the Xref is generated from the file name. If an Xref (or overlay) with the same name already exists in the drawing, the block is replaced using existing Xref.

7.82.2 Options within the command

Enter block name of the block to be replaced with an xref
Enter a block name.

?
Lists the block names.

select objects (=)
Select a block reference with the desired block name.

options (*)
Lists the replacement options.

by block Name
Enter a block name to replace.

Selected block instances
Select the block instance(s) to replace.

SIMilar inserts
Replaces all instances of a block and all its modified (anonymous) dynamic blocks.

Note: If non-dynamic block is selected, then all instances with the same block name is replaced with Xref.

Note: This option acts like the BLOCKREPLACE command.

with Conversion to xref

Yes
First saves the block and then replaces the block with the newly created Xref.

No
The block is replaced with an Xref.

Ask for filename for newly created Xref

Yes
Saves the block before replacing it with an Xref.

No
A new drawing is created next to the original drawing with the same name as the block name.

Enter an xref file (~ to open file dialog)
Enter the name of an xref file or enter ~ to open the Select an Xref file dialog box.

7.83 BMBALLOON command

Creates associative balloon entities for assembly components.

Shape Lite Pro Mechanical BIM

Icon:

7.83.1 Description

Creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. Each balloon contains an index of the component in the associated BOM-table created by BMBOM command.

Note: Balloons are created as Multileader entities using the current Multileader style. Create a Multileader style with the **Content** set to **Block** and adjust the Scale property to control the size of the balloon frame.

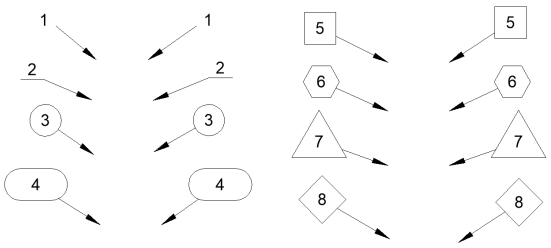
7.83.2 Options within the command

select other Table

Allows to select a different BOM-table in case multiple tables exist in the layout. The most recently inserted BOM-table is selected by default.

select balloon Style

Allows to define the shape of the balloon.



Note: Predefined balloon styles from _BalloonStyles.dwg can be used, as well as any custom user-defined styles.

Note: The PROMPTMENU variable allows the selection of the balloon styles.

Auto mode

Allows automatic placement of balloons on all components in a particular drawing view.

placement Mode

Specifies whether multiple balloons with the same values are generated or not.

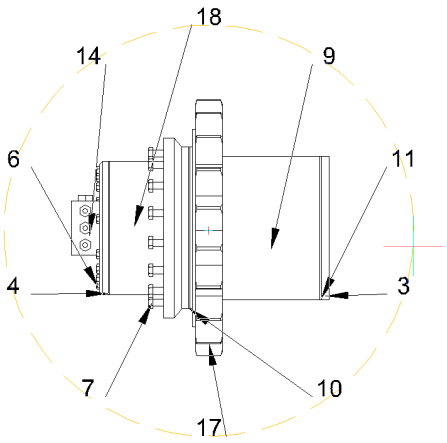
Unique

Only unique values are generated on one view.

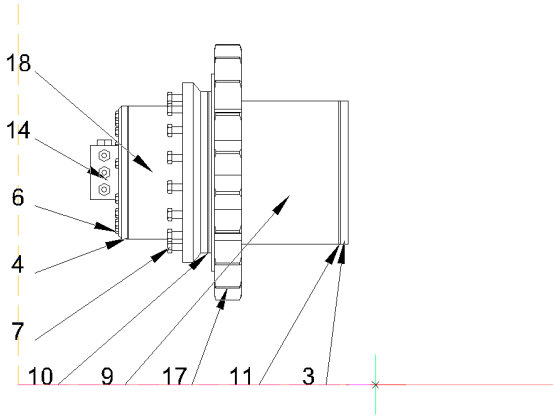
All

All values are generated on one view.

Circle
Aligns leaders along a circle.



Polyline
Aligns leaders along a polyline.



Back
Returns to previous prompt.

7.84 BMBOM command

Inserts a Bill of Materials (BOM) table in the current drawing.

Shape | Lite | Pro | Mechanical | BIM

Icon:

7.84.1 Options within the command

Name
Specifies the table name.

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 BOM of the top-level components only.

Bottom level
Creates a BOM of the subcomponents.

Hierarchical
Creates a hierarchical BOM listing all (sub)components.

Load from template
Opens the **Select file with the BOM table template** which allows you to select a .bom file to create a BOM from a template.

Save as template
Opens the **Select file to save BOM table template** which allows you to save the current BOM configuration as a template.

LAyout
Changes layout where the table will be placed.

Configure
Allows to add additional columns to your BOM table.

Description
Adds a description column.

dEnsity
Adds a density column.

Volume
Adds a volume column.

Mass
Adds a mass column.

parameterS
If multiple inserts of the same parametric component exist in the assembly, they will be grouped in different BOM rows depending on the values of the parameters.

mAterials
Adds a material column.

Thickness
Adds a thickness column.

Note: If a component is a sheet metal part, this column shows the thickness of the part. Otherwise, it is empty.

More
Displays more configuration options.

Toggle column
Adds or removes columns from the BOM.

add Formula field
Adds a formula field.

add Thumbnails
Adds a thumbnail column

Name
Sets the name of the column.

Width
Sets the thumbnails width in pixels

Height
Sets the thumbnails height in pixels

View type
Sets the orientation of the camera.

Orthographic
Allows to choose an orthographic projection view.

Isometric
Allows to choose an isometric projection view

viSual style
Sets the visual style for thumbnails.

BacKGround color
Sets the thumbnails background color.

Transparent
Sets transparent background for thumbnails.

seleCt color
Opens the **Select color** dialog box to select a color.

Folder
Opens the **Select image folder** which allows you to choose a folder for thumbnail images.

Add column
Adds a thumbnail column with specified parameters to the BOM table.

Set sorting
Sets the sorting mode for the BOM.

Automatic
Sets the automatic sorting for the table.

No sorting
Disables the sorting for the BOM.

Custom sorting
Specifies the column for sorting.

Toggle sorting column
Sets the sorting mode for columns.

column Properties
Configures columns.

Visibility
Sets the visibility of the column.

ON
Makes the column visible.

OFF
Makes the column invisible.

Width
Sets the column width.

Auto
Sets the automatic width of the column.

Prefix
Sets the prefix for numbers in the Number column.

Suffix
Sets the suffix for numbers in the Number column.

Delimiter
Sets the delimiter between levels in numbers for hierarchical BOM tables.

Numeration type
Sets the numeration type for hierarchical BOM tables.

Full numbers
Numbers for each level are combined using the provided delimiter and shown in the Number column.

Short numbers
Only numbers for the current level of each part are used. This mode is supposed to be used with Level column.

Continuous numbers
All parts are numbered continuously, regardless of their level.

Footer type
Sets the footer type.

None
Removes the column from the footer.

Sum
Adds the total sum to the footer.

Average
Adds the average value to the footer.

MINimum
Adds the minimal value to the footer.

MAXimum
Adds the maximal value to the footer.

Note: It is possible to add footers to formula or template columns as well.

Note: A different unit and format for footer values can be set.

Aggregate function
Sets the aggregate function.

None
Removes an aggregate function from the column.

Sum
Shows the total sum of values in the group.

Average
Shows the average value of values in the group.

MINimum
Shows the lowest value in the group.

MAXimum
Shows the highest value in the group.

CONcatenate
Combines all values in the group using the provided delimiter.

CONCatenate with count
Combines all values in the group with their number using the provided delimiters, prefix and suffix.

sEttings
Configures the aggregated column.

value Separator
Sets the delimiter between concatenated values.

count Position
Sets the position of the number of encounters for concatenated values.

After value
The number of instances is placed after the corresponding value.

bEfore value
The number of instances is placed before the corresponding value.

Count separator
Sets the delimiter between a value and its number of encounters.

count PREfix
Sets the prefix that will be added before the number of encounters.

count SUFfix
Sets the suffix that will be added after the number of encounters.

Units
Configures units and how they display.

unit Mode
Sets the units for the column values.

Best for All
The best unit for all values.

BEst for each value
The best unit for each value.

set Fixed unit
Choose the unit manually.

unit Format
Sets how the units display.

Title
Places the unit symbol in the column title.

Same cell
Places the unit symbol in the same cell as the value.

separate Column
Places the unit symbol in a separate column.

do Not show units
Hides the units symbol.

title Format
Sets the format for the column title for the Title option.

fOrmat string
Sets the format string for column values.

Role
Sets the column role for the column.

Regular
Sets the regular role for the column.

Number
Sets the number role for the column. (So it will be used as the number source for the BMBALLON command).

nAme
Sets the name role for the column. (So it will be used as the name source for the BMBALLON command).

Quantity
Sets the quantity role for the column. (So it will be used as the quantity source for the BMBALLON command).

table sEttings
Configures the properties of a BOM table.

footer Title
Sets the title of the footer row.

Filter
Sets the table filter.

Property set
Specify the properties to add from.

Mechanical only
Only mechanical properties of components and instances will be available.

all Except coordinates
All properties of mechanical components and instances will be available, including properties of associated database entities, except coordinates.

All
All properties of mechanical components and instances will be available, including properties of associated database entities.

Grouping mode
Specifies the grouping mode for the current table.

Auto
Groups the parts with respect to their definitions and parameters.

by COmponents and columns
Groups the parts with respect to their definitions and properties used in the table.

by Columns only
Groups the parts with respect to their properties used in the table only. (So one row may correspond to completely different parts if their properties displayed in BOM are the same).

Counting mode
Specifies the counting mode for hierarchical BOM.

by Document
Sets the counting mode to count all instances.

by parent Component
Sets the counting mode to count instances in parent component.

Max level
Allows to specify the maximal level of parts in hierarchical BOM tables.

Selection set
Allows to specify a selection set linked to this BOM table.

Entire model
Sets entire model as the selection set.

Subassembly
Sets a selected subassembly as the selection set.

Custom selection
Sets a custom selection of top-level components as the selection set.

drawing View
Associates this BOM table with selected drawing views.

layout viewPOrt
Associates this BOM table with selected layout viewports.

fixed coRner
Changes the fixed corner of the table. The fixed corner will remain in position during table changes.

Note: Press the **Tab** key to cycle the corners of the table. The table position will change to place the fixed corner under the cursor, thus helping to align the table with other objects in the drawing.

Top Left
Sets top left corner of the table as fixed.

Top Right
Sets top right corner of the table as fixed.

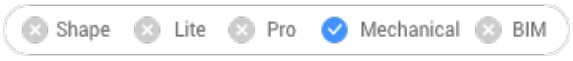
Bottom Left
Sets bottom left corner of the table as fixed.

Bottom Right
Sets bottom right corner of the table as fixed.

Back
Returns to the previous prompt.

7.85 BMBOMEDIT command

Edits an existing Bill of Materials (BOM) table in the current drawing.



7.85.1 Options within the command

APPLY
Applies the current changes.

DISCARD
Discards the current changes.

Note: See the BMBOM command for the full command options.

7.86 BMBOMPANELCLOSE command

Closes the BOM manager panel.

Shape Lite Pro Mechanical BIM

7.86.1 Description

Closes the BOM manager panel to hide it from the current workspace. If the BOM manager panel is stacked when you close it, the BOM manager tab or icon is removed from the stack.

7.87 BMBOMPANELOPEN command

Opens the **BOM manager** panel.

Shape Lite Pro Mechanical BIM

7.87.1 Description

Opens the **BOM manager** panel to display it in the current workspace. The **BOM manager** panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the **BOM manager** panel can be either floating, docked or stacked.

7.88 BMCONNECT command

Connects two components by creation of 3D constraints between their connection entities. Also, under the **complete flange Assembly** option, a gasket and a bolting assembly are inserted, resized, and connected to a pair of flanges.

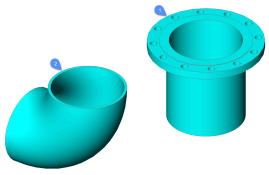
Shape Lite Pro Mechanical BIM

Icon: 

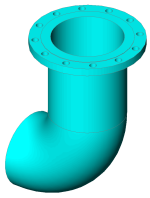
7.88.1 Method

Select components to connect
Select the component to connect. (1)

Select target component
Select the component to connect to. (2)



The command automatically creates a set of 3D constraints to correctly connect the parts and maintain their relative position for any further modification.



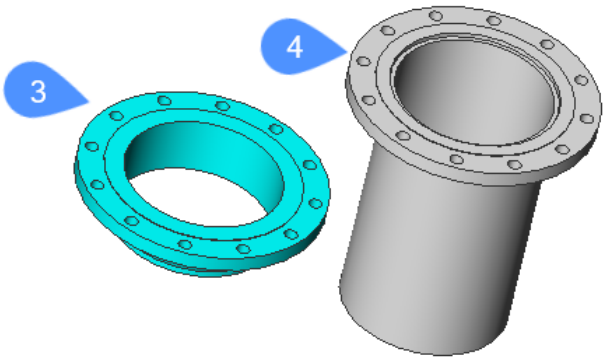
7.88.2 Option within the command

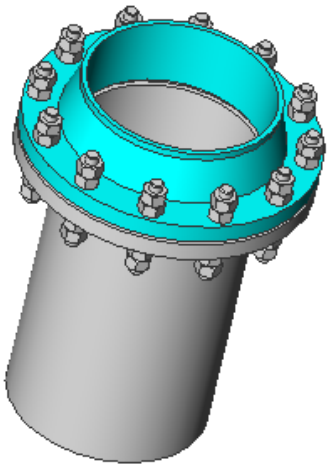
Flip

The next pair of connection point and line is selected for the component (1)

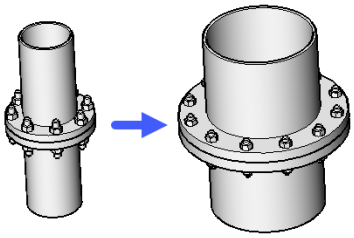
complete flange Assembly

A gasket and a bolting assembly can be selected to automatically connect the flanges (3) and (4).





If the flanges', the gaskets' the bolting assembly's Size and Rating parameters are linked to the global Size and Rating parameters, the flange assembly will be updated properly when these global parameters are changed.



Gasket
Allows to choose a gasket for the flange assembly.

list available gaskets (?)
Lists all available gaskets in the command line.

Path
Allows to choose a custom gasket from the Select a gasket file (Standard Open File) dialog box.

Clear
Removes the chosen gasket.

BOlting assembly
Selects a bolting assembly.

Note: Bolting is a parametric assembly of fasteners for one pair of flanges' bolting holes, e.g., a stud and 4 nuts. All nuts are of the same type. Two of these nuts can be suppressed, according to NutsNumber parameter.

Note: Bolting Assembly is a parametric assembly that contains a polar array of boltings (and connection 2d entities). Its Size and Rating parameters can be linked to the corresponding global parameters.

Note: Flange Assembly is an assembly of the connected pair of flanges, gasket, and bolting assembly.

Path
Allows to choose an existing bolting assembly from the Select a bolting assembly file (Standard Open File) dialog box.

Generate
Generates a boltings assembly from a bolting.

Path
Allows to choose an existing bolting from the Select a bolting file (Standard Open File) dialog box.

Generate
Generates a bolting.

Select a stud

list available studs (?)
Lists all available studs in the command line.

Path
Allows to choose a custom stud from the Select a stud file (Standard Open File) dialog box.

Select a nut

list available nuts (?)
Lists all available nuts in the command line

Path
Allows to choose a custom nut from the Select a nut file (Standard Open File) dialog box.

Specify the bolting name
Gives a name to the bolting generated above and save it in the Select the bolting file dialog box.

Specify the bolting assembly name
Gives a name to the bolting assembly generated above and save it in the Select the bolting file dialog box.

Keep the bolting file
Specifies if keep the bolting file

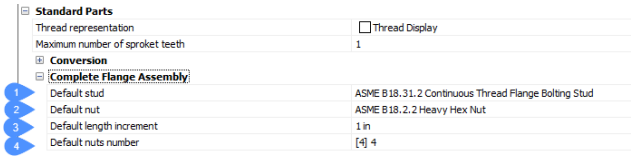
Select nuts number
Specifies the nuts number.

Input length increment
Specifies the length increment of the stud.

Auto
Applies the default stud, nut, length increment, nuts number for the boltings assembly.
A temporary bolting assembly is generated and is chosen. Its file will be removed at the end of the command execution.

Clear
Removes the chosen bolting assembly.

SETtings
Opens the Settings dialog box:



1 BoltingAsmDefaultStud - default stud to generate bolting assembly.

2 BoltingAsmDefaultNut - default nut to generate bolting assembly.

3 BoltingAsmDefaultLengthIncrement - default length increment for default stud.

4 BoltingAsmDefaultNutsNumber - default nuts number for bolting assembly.

Back
Returns to the previous command line options.

Finish
Creates the assembly of flanges, gasket and bolting assembly.

7.89 BMCONVERT command

Converts BIM Linear solids with circular profiles to Mechanical assemblies.

Shape Lite Pro Mechanical BIM

7.89.1 Method

Select entities to convert and choose the proper styles for pipes.

7.89.2 Options within the command

Pipe
Allows you to select a style for the pipe segment.

Elbow
Allows you to change the styles for the pipe elbows (45 deg and 90 deg).

Reducer
Allows you to change the styles for the pipe reducers (concentric and eccentric).

Splitter
Allows you to change the styles for the pipe tee and pipe cross.

?
Lists available styles

Note: Use **;** to select multiple styles.

Use current
Uses the current style.

SETtings
Opens the Settings dialog box with the **Conversion** section expanded.

7.90 BMCREATECOMPONENT command

Creates library blocks.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The BMCREATECOMPONENT command has been replaced by the CREATELIBRARYBLOCK command.

7.91 -BMCREATECOMPONENT command

7.92 BMDEPENDENCIES command

Lists all files, containing component definitions inserted in the assembly, in the Command line.

Shape Lite Pro Mechanical BIM

Icon: 

7.92.1 Method

The files are automatically listed in the Command line once the command is run.

7.93 BMDISSOLVE command

Dissolves a mechanical component inserted in the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

7.93.1 Description

Dissolves a mechanical component inserted in the current drawing by selecting all entities to dissolve manually.

Note: Only components at the first level, directly branching from the root component, in the mechanical hierarchy can be dissolved. To dissolve a component at a lower level, its parent component(s) must be dissolved first.

All subcomponents of a dissolved component are moved up one level in the mechanical hierarchy.

Dissolving a component insert does not affect the source drawing of the component.

The selection set must contain component inserts only. If not, a message displays in the Command line: "No components selected."

7.94 BMEXPLODE command

Creates a block with an exploded representation of the current assembly.

Shape Lite Pro Mechanical BIM

Icon: 

7.94.1 Description

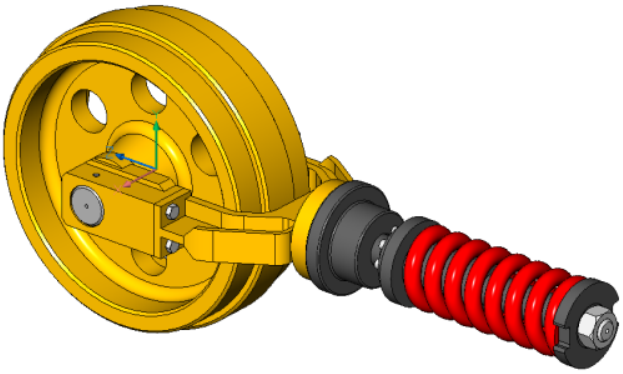
Creates a block with an exploded representation of the current assembly. This block can be inserted at any location.

Note: Local mechanical components based on blocks and solids are now supported in exploded views, but not yet the mechanical external references.

7.94.2 Method

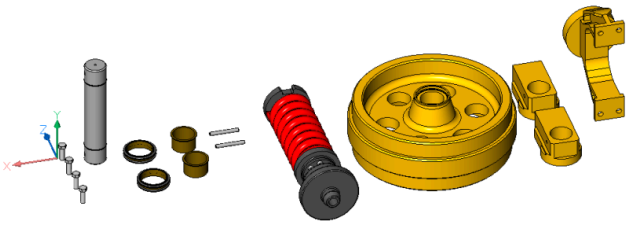
It supports two levels of the exploded representation: top and bottom. Top-level means that the assembly will be exploded down to top level components. Bottom-level means that the assembly will be exploded down to bottom level components.

The steps to create an exploded representation are automatically completed and are displayed in the Mechanical Browser. You can add, delete and reorder the steps from the Mechanical Browser. Animate a single step of an explosion or the entire sequence.

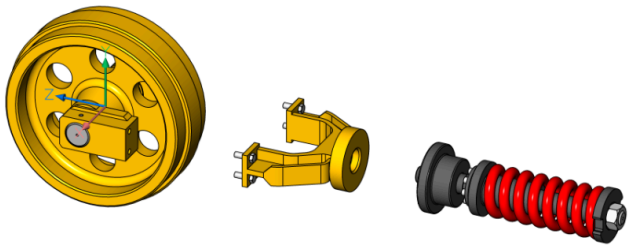


7.94.3 Options

Table by types
Groups identical parts on the same row and different parts on different rows.



Linear
Automatically creates exploded representations of assemblies in a given direction, taking into account possible physical collisions between components.

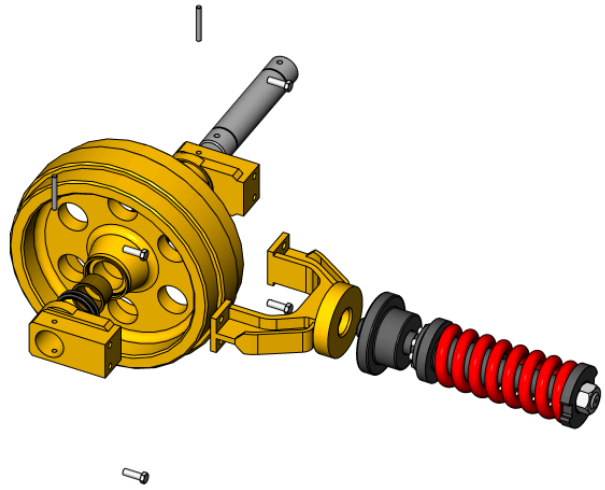


Automatic
Automatically determines the directions for each part, taking into account possible physical collisions between components.

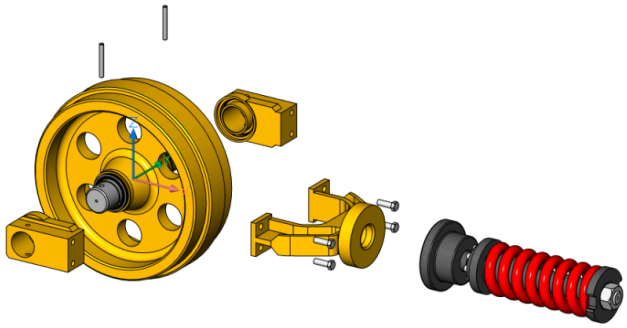
enable Leader follower
Enables the detection of parts that should move in a coherent manner. If enabled, parts that are inserted into other parts will follow the movement of the base parts they are inserted into.

use Current projection for gap
Enables the gap calculation with respect to the current camera orientation. It minimizes the probability that some parts will be hidden by other parts on a 2D view with the current camera orientation.

Auto
Enables automatic gap calculation.



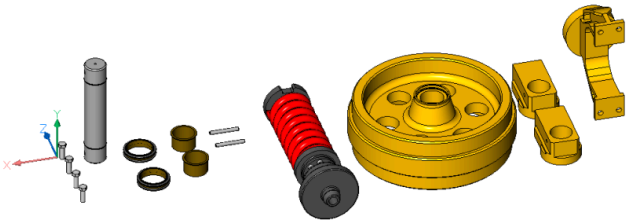
Manual
Selects the manual mode. This creates an exact copy of the assembly, ready to create a customized exploded view.



Update
Updates solids and block references in exploded view block with model space objects.

Settings
Gives access to the command settings.

Top
If selected, the assembly will be exploded using only top-level components.



Bottom
If selected, the assembly will be exploded down to bottom-level components.



set Name
Set the name for the exploded representation.

BAck
Returns to the previous prompt.

Edit
Opens the exploded representation for editing.

Generate drawing views
Generates drafting views from an exploded representation.

Finish
Concludes the command.

7.95 BMEXPLODEMOVE command

Moves selected parts to form an exploded representation in a given direction.

Shape Lite Pro Mechanical BIM

Icon:

7.95.1 Description

Moves 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).

This command also has an automatic mode. This determines the direction for each selected part automatically (similar to the Automatic option of the BMEXPLODE command).

7.95.2 Options within the command

Automatic
Enables automatic mode. An automatically exploded representation of the selected parts is created.

Entire model
Selects the entire model.

Linear
Enables the linear explode mode.

7.96 BMEXPLODESTEPEDIT command

Manages the exploded view steps.

Shape Lite Pro Mechanical BIM

7.96.1 Description

Allows to create, delete, merge, copy, split and reorder exploded view steps as well as to save the current step and to change the current step.

7.96.2 Method

Select an exploded view created with BMEXPLODE command and open it for edit (from MECHANICALBROWSER or with -BEDIT command).

Perform the necessary operations to get the desired exploded representation following the below options. Do not forget to save the current step.

Then, save changes using BCLOSE command with the option Save.

7.96.3 Options

Add step
Adds a new step after the last one. The new step will be automatically set as current. All unsaved modifications of the exploded representation will be lost.

cOpy step
Copy an existing step. Step names are case sensitive.

merGe step with previous
Merges a step with the previous one and removes the previous step.

Move step
Moves a step up (-) or down (+) in the list.

Remove step
Removes a step as well as associated movements of parts. The positions of parts in the exploded representation will be updated accordingly.

reName step
Renames a step.

Save current state
Saves the current positions of parts in the current step.

set Current step
Sets the step as the current and updates the parts in the exploded representation according to this step.

sPlit step
Splits the step to a sequence of steps, each of them corresponding to exactly one part.

list steps (?)
Lists all steps.

7.97 BMEXTERNALIZE command

Converts local components to external components.

Shape Lite Pro Mechanical BIM

Icon: 

7.97.1 Description

Local Mechanical components are converted to external components with the **Save Mechanical Component** dialog box.

Local Mechanical blocks are converted to external references with the **Save Block** dialog box.

This procedure is repeated for each selected local component.

7.97.2 Options

Entire model
A file dialog displays for each local component in the assembly.

ASk for file names
Toggles the option to ask for file names of externalized components or to generate them automatically.

Yes
File names need to be entered.

No
File names are generated automatically.

> ***Note:*** some symbols used in component names cannot be used in file names and will be automatically replaced with an underscore.

7.98 BMFORM command

Creates a new mechanical component or a mechanical block, and inserts it into the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

7.98.1 Description

Creates a new mechanical component as an internal reference in the current drawing. The selected solid(s) are removed from the current drawing.

> ***Note:*** When the MECHANICALBLOCKS system variable is enabled (set to 1), BMFORM creates mechanical blocks by default.

> ***Note:*** If necessary, run the BMMECH command first to initialize the mechanical structure in the current drawing.

7.98.2 Options

Local
Creates a new component and inserts it as a block reference into the current drawing.

> ***Note:*** When MECHANICALBLOCKS system variable is enabled (set to 1), creates an internal mechanical block. Use BEDIT command to edit block definition.

External
Creates a new drawing which contains the selected solid(s) and can be saved with the **Save Drawing As** dialog box. The selected solid(s) are removed from the current drawing. The new drawing is inserted as an external reference in the current drawing.

mechanical Component
Creates a new mechanical component.

mechanical Block
Creates a new mechanical block.

Template settings
Opens the **Settings** dialog box at the BMFORMTEMPLATEPATH system variable to choose a template file.

7.99 BMHARDWARE command

Opens the Library panel.

Shape Lite Pro Mechanical BIM

Icon: 

7.99.1 Description

Opens the Library panel to display it in the current workspace. The Library panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Library panel can be either floating, docked or stacked.

7.100 -BMHARDWARE command

Inserts a standard hardware part as a mechanical component in the current drawing.

Shape Lite Pro Mechanical BIM

7.100.1 Method

This command can only be executed via the options in the Command line.

Note: To initialize the mechanical structure in the current drawing, run the BMMECH command.

Note: This command is obsolete and creates non-parametric standard hardware parts. To insert parametric standard hardware, use the Components panel. This contains parametric Standard Mechanical Components.

7.100.2 Options within the command

Parametric
Allows to switch on/off the creation of parametric standard parts.

create Custom
Allows to create custom standard parts from prepared tables. Opens the **Select configuration table file** dialog box that allows to select your created .TXT file (or files).

Edit inserted entity
Edits the parameters of the selected insert.

SMART insert
Allows to automatically connect a Piping standard part to an existing Piping standard part by creating a proper set of 3D constraints between them and by copying expressions for the parameters of existing inserted part to the new part.

Rotate component
Specifies the rotation angle by entering a value.

set Base point
Specifies a new base point for the insert.

Name
Renames the insert by entering a name.

insertion Type
Specifies the insertion type.

Local
Inserts the part locally.

External
Inserts the part externally.

Flip
Inserts the selected insert in the opposite face of the solid.

mUltiple
Allows you to insert multiple copies by entering multiple insertion points.

Array
Creates an associative array of components.

Change target 3d solids
Allows you to choose the new set of target solids. Any current component-based features will be removed and new ones will be created for the selected solids.

cleaR
Removes the existing component-based features with their geometry. See the BMUNLINK command.

Select all affected 3d solids
Automatically chooses solids. A solid is selected if any solids from the BC_SUBTRACT layer or if any layer from the BC_UNITE touches or intersects it.

7.101 BMHIDE command

Hides an inserted mechanical component.

Shape Lite Pro Mechanical BIM

Icon:

7.101.1 Description

Hides a mechanical component inserted in the current drawing.

The command affects the visibility of the selected inserts only. Hidden inserts will still be considered by commands such as BMBOM, BMMASSPROP etc.

7.101.2 Options

Component name
Type the component name of the component insert(s) you want to hide. All inserts of the specified component will be hidden.

Insert name
Type the insert name of the component insert you want to hide.
If you want to hide a nested component insert (subcomponent insert) first type the insert name of the parent component insert, then the component insert name separated by a slash (/).

7.102 BMINSERT command

Opens the Select file to insert dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

7.102.1 Description

Opens the Select file to insert dialog box to select a dwg or rfa file to insert into the current drawing.

After selecting the file and choosing **Open**, the Properties panel is displayed, and you are prompted at the Command line.

7.102.2 Select insertion point

Specify a point in the current drawing where the entity will be inserted. You can change the insertion before it is inserted using the Properties panel and Command line options.

Additional options: [Edit inserted entity/SMART insert/Rotate component/set Base point/Name/insertion Type/Flip/mUltiple/Change target 3d solids].

7.102.3 Options within the BMINSERT command

Edit inserted entity
Enables you to change the parameter expressions for the inserted entity. Continue editing individual parameters until you press Enter to end the option. This option is also available in the Hot Key Assistant.

Enter the parameter name or press ENTER to finish
Specify the name of the parameter.

Enter expression
Specify the expression for the parameter.

Smart insert
Enables you to connect a Piping standard part to an existing Piping standard part. It automatically creates appropriate 3D constraints between the two parts and copies expressions for the parameters of the existing part to the new part. This option is also available in the Hot Key Assistant.

Select entity to insert to or
Hover the cursor over the Piping standard part you want to connect to.
Additional options: [switch Back/Rotate component/Flip/mUltiple/selection options (?)]

Rotate component
Enables you to change the rotation angle for the inserted entity.

Specify rotation angle of the insert
Specify the rotation angle.

Set base point
Enables you to change the basepoint for the inserted entity.

New base point <0,0,0>
Specify a new basepoint for the entity.

Name
Enables you to change the instance name for the inserted entity.

Component insert name <DefaultName>
Enter a name for the inserted entity.

Insertion type
Enables you to change the insertion type for the inserted entity.

Specify component type [Local/External] <External>
Choose Local or External

- Local: references an entity definition within the current drawing. See Block Reference.
- External: references an entity definition outside the current drawing. See External Reference.

Flip
Enables you to flip the direction for the inserted entity.

Multiple
Enables you to insert multiple copies of the same entity by specifying an insertion point for each instance or creating an Array.

Select insertion point
Specify a point in the current drawing where the entity will be inserted. Continue inserting entities until you press Enter to end the command.
Additional options: [Edit inserted entity/Rotate/Base point/Name/Insertion Type/Flip/Array/Change target 3d solids]

Array
Enables you to create an associative array of the inserted entity.

Specify base point for the array
Specify a point.

Enter the distance between columns
Specify the distance between columns
Additional options: [Direction]

Enter the distance between rows
Additional options: [Single row/Rectangular/Direction]

Select an end point for creating array
Specify a point.

Accept resulting array
Press Enter to accept the array.
Additional options: [Base point/Columns/Rows/Accept] <Accept>:

Change target 3d solids
Enables you to apply the inserted entity to existing 3D solids in the current drawing.

Select target 3D solids
Select the 3D solids to which you want
Additional options: [cleaR/Select all affected 3d solids]

Select all affected 3d solids
All solids intersecting or touching solids in the BC_SUBTRACT and BC_UNITE layers of the inserted entity are affected.

Clear
Clears the selection set to ensure no solids are affected by the inserted entity.

7.103 -BMINSERT command

Inserts sheet metal form features and components in the current drawing.

Shape Lite Pro Mechanical BIM

7.103.1 Description

Inserts sheet metal form features, mechanical and BIM components in the current drawing.

Note: This command inserts a *.RFA file as a BIM component.

This command operates at the Command line. Enter the path of the file to be inserted and choose one of the options. For more details about the options see BMINSERT command.

7.104 BMLINK command

Changes target 3D solids of component-based features.

Shape Lite Pro Mechanical BIM

Icon:

7.104.1 Description

Changes the target 3D solids of component-based features by adding or removing target 3D solids.

7.104.2 Options within the command

Change target 3d solids
Specifies a set of target solids. Component-based features will be removed and new ones will be created for selected solids.

Append
Adds new solids to set of target solids. Component-based features will be updated and new ones will be created for the selected solids.

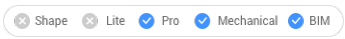
cleaR
Removes the existing component-based features and their geometry (see the BmUnlink command).

Select all affected 3d solids
This mode is similar to **Change target 3d solids** but solids are chosen automatically.

Note: A solid will be chosen if any solids from BC_SUBTRACT layer intersects it or if any other solid from the BC_UNITE layer touches or intersects it.

7.105 BMLOCALIZE command

Converts external components to local components.



Icon: 

7.105.1 Description

Converts external components to local components by selecting the components manually.

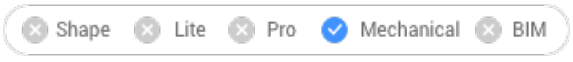
If multiple inserts of the same external component exist in the model, all inserts are converted to local components.

7.105.2 Options within the command

Entire model
Converts all external components in the model to local components.

7.106 BMMASSPROP command

Computes mass properties for the current model using densities of materials assigned to its components.



Icon: 

Note: BmMassProp will format the values of mass, volume and linear nature with their corresponding units only if the PROPUNITS system variable has the corresponding bits set to 1 or the corresponding flag in the Settings dialog for the Property units field is checked.

7.106.1 Method

The density values used for the mass property calculations are taken from materials defined by the Material property of the components and subcomponents, which can be either inherited from the main component of the assembly or set explicitly. The density of the main component is defined by the materials set in the Material field in the Mechanical Browser dialog. If a material does not have an

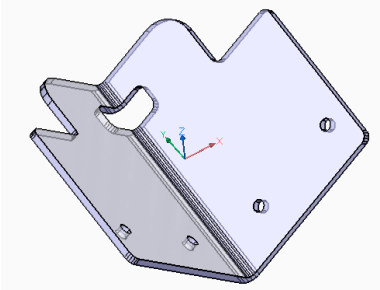
assigned density, or its density is less or equal to zero, an error message displays, and the mass properties will not be computed. All materials involved in the assembly must have non-zero non-negative densities.

Selects the 3D solids and subcomponents or the entire model and will get a Report in the Prompt History window:

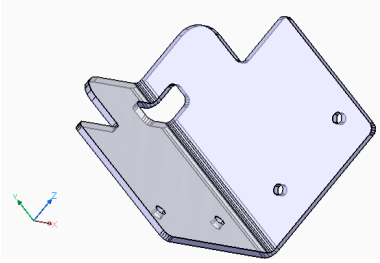
```
---------------- Component ----------------
Mass: 6.7757 kg
Volume: 846.9586 cm³
Centroid: X= -267.9141 mm Y= 46.2431 mm Z= -435.3534 mm
Moments of inertia: X= 1.3564 kg·m² Y= 1.8427 kg·m² Z= 0.5730 kg·m²
Products of inertia:
XY: -0.0808 kg·m²
YZ: -0.1535 kg·m²
ZX: 0.7934 kg·m²
Radii of gyration: X= 447.4159 mm Y= 521.4979 mm Z= 290.802 mm
Principal moments and X-Y-Z directions about centroid:
I: 0.0551 kg·m² along X= 0.0000 Y= -0.7071 Z= 0.7071
J: 0.0570 kg·m² along X= 0.9906 Y= 0.0970 Z= 0.0969
K: 0.0899 kg·m² along X= -0.1371 Y= 0.7004 Z= 0.7004
```

7.106.2 Options

Yes
Aligns the UCS with the principal moment axes.



No
Keeps the current UCS.



7.107 BMMECH command

Converts the current drawing into a mechanical component.

Shape Lite Pro Mechanical BIM

Icon:

7.107.1 Description

If the drawing contains block references or external references, they can also be converted into inserts of local and external components. If mechanical blocks are enabled, solids and regular named blocks can be converted to mechanical entities and blocks correspondingly as well.

If the drawing is a mechanical component already, BMMECH does nothing.

Note: Mechanical blocks are enabled if the MECHANICALBLOCKS system variable is set to 1.

7.107.2 Options within the command

create Legacy components
(Only if Mechanical blocks are enabled) Changes the command mode to create mechanical (legacy) components.

create mechanical Blocks
(Only if Mechanical blocks are enabled) Changes the command mode to create mechanical blocks and mechanical entities. If there is no root component in the document, the document will become a mechanical block itself.

Yes - to convert all
If the drawing contains block references or external references, blocks are converted to internal components and external references are converted to external components. If the command is in Mechanical blocks mode, block references, external references and solids in the model space will be converted to mechanical blocks and entities.

Note: Blocks converted to mechanical (legacy) components are not available in the INSERT command. Use BMINSERT to insert blocks as internal components. However, mechanical blocks are available in the INSERT command, as they are regular blocks with mechanical data attached.

Blocks - to convert blocks
(Only if Mechanical blocks are enabled). If the drawing contains block references or external references, they will be converted to mechanical blocks. Solids from the model space will not be converted in that mode.

No - to skip conversion
The mechanical structure is initialized but blocks and external references are not converted.

7.108 BMNEW command

Creates a mechanical component as a new drawing file.

Shape Lite Pro Mechanical BIM

Icon: 

7.108.1 Description

Creates a mechanical component by automatically opening a new drawing file.

Note: When the MECHANICALBLOCKS system variable is enabled (ON), a mechanical root block is created instead.

7.109 BMOPEN command

Opens the source drawing of an external mechanical component.

Shape Lite Pro Mechanical BIM

Icon: 

7.109.1 Description

Opens the source drawing of an external mechanical component by selecting it in the model, in order to change or visualize it.

7.110 BMOPENCOPY command

Opens a copy of a component insert as a new drawing.

Shape Lite Pro Mechanical BIM

Icon: 

7.110.1 Description

Opens a copy of a selected external mechanical component in a new drawing.

If you open a copy of a parametric mechanical component, the current values of the various parameters are applied to the copy.

After saving it you may replace the original component with the newly created drawing using BMREPLACE command.

7.111 -BMPARAMETERS command

Allows to list and edit the parameters of inserted components and associative arrays.

Shape Lite Pro Mechanical BIM

7.111.1 Description

Allows to list and edit the parameters of inserted components and associative arrays. By pressing Enter, the parameter expression can be changed.

Note: Parameters of components nested inside an array can be changed or assigned to an expression.

7.111.2 Options within the command

Edit
Edits the value of a parameter.

Delete
Allows to reset a previously edited parameter to its default value.

Link to parameter
Assigns parameters as an expression for a parameter of a dimensional constraint (both 2D and 3D) associated with an entity which has been put into an associative array.

?

Lists the parameters of the selected components in the Prompt History window.

Note: Press F2 to display the Prompt History window.

7.112 BMPROPERTIES command

Creates, edits and deletes property definitions and organizes properties in property sets.

Shape Lite Pro Mechanical BIM

Icon: 

7.112.1 Description

Opens the **Mechanical Properties** dialog box to create and manage property definitions.

7.113 BMPOUT command

Saves the current view as a BMP file.

Shape Lite Pro Mechanical BIM

7.113.1 Description

Opens the Save Bitmap dialog box to save the current view (model or paper space) as a bitmap BMP file.

7.114 BMRECOVER command

Recovers a broken mechanical structure.

Shape Lite Pro Mechanical BIM

7.114.1 Description

Recovers a broken mechanical structure selected through the Select file dialog box. The command overwrites the original drawing.

Note: The command differs from the basic Audit and Recover commands. It is recommended to use BMRECOVER to mend issues with a MCAD drawing.

7.115 BMREPLACE command

Replaces a component insert.

Shape Lite Pro Mechanical BIM

7.115.1 Description

Replaces a mechanical component inserted in the current drawing with another one selected from Select file component dialog box.

7.115.2 Options within the command

SIMilar inserts
Allows to replace similar inserts automatically, depending on the chosen option.

- **Yes**: replaces all similar inserts of the selected mechanical component(s) in assembly.
- **No**: replaces the selected component(s) only.

component Type
Allows to choose the insertion type for a replacement component. The following options are available:

- **Local**: the new component will be local.
- **External**: the new component will be external.
- **Keep as is**: the new component inherits the insertion type from the replaced component.
- **component Default**: the new component will be linked as specified in its source file.

See External and local components in the User Guide.

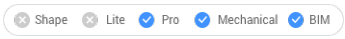
Parameter changes
Allows to control which parameter values will be used after the replacement is complete.

- **Yes - reapply**: parameter values of the replaced component are applied.
- **No - use replacement as is**: parameter values of the replacement component are applied.

from FIle
Allows to select a .dwg file from the Select file for component dialog box for the replacement component.

7.116 BMSHOW command

Shows hidden mechanical components from the drawing.



7.116.1 Description

Shows previously hidden mechanical components inserted in the current drawing.

7.116.2 Options within the command

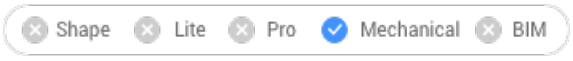
show ALl
Shows all component inserts.

COmponent name
Type the component name of the component insert(s) you want to show. This affects all inserts of the specified component.

Insert name
Type the insert name of the component insert you want to show.
If you want to show a nested component insert (subcomponent insert), first type the insert name of the parent component insert, then the component insert name separated by a forward slash (/).

7.117 BMTRAILINGLINES command

Creates all necessary trailing lines for selected parts.



7.117.1 Description

Creates all necessary trailing lines by selecting entities.

Note: An exploded view needs to be selected before the command can be run.

7.117.2 Options within the command

use Origin points
Switches the mode used to choose the reference point for parts to the origin (insertion point) of the component.

use Center points
Switches the mode used to choose the reference point for parts to the center point of the bounding box of the component.

Entire model
Selects the entire model and starts the processing.

7.118 BMUNLINK command

Breaks the connection between a component insert and a 3D solid retaining the opening.

Shape Lite Pro Mechanical BIM

7.118.1 Description

Breaks the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening by selecting the components.

7.119 BMUNMECH command

Converts mechanical components.

Shape Lite Pro Mechanical BIM

7.119.1 Description

The command converts the current mechanical component into a plain drawing.

Note: The command applies to drawings that are a mechanical component only.

7.119.2 Options within the command

Yes
Removes the mechanical structure from the drawing, by turning it into a plain drawing.
Inserts of local components are converted to block references (the component name becomes the block name). Inserts of external components are converted to external references.

No
Aborts the command.

7.120 BMUPDATE command

Reloads all referenced components from external files and updates BOM tables.

Shape Lite Pro Mechanical BIM

7.120.1 Description

The command:

- Reloads referenced components from external files (BmInsert).
- Rebuilds component based features, sheet metal form features and BIM windows/doors components (see BC_SUBTRACT layer, Form Features, BmInsert).
- Updates the Bill Of Material (BmBom).
- Regenerates drawing views produced out of 3D model (ViewBase).

Note: If the BMAUTOUPDATE system variable is ON, external assembly components are reloaded on opening the file.

7.120.2 Options within the command

Mode
Defines the new placement face.

Note: Each inserted component-based feature or BIM Window component has an associated placement face of the correspondent 3D solid the feature is attached to. This is the face which will be used for feature placement in the BMINSERT command. The associated placement face of the feature can be lost in modeling or copying operations or it can be absent if the feature has been inserted in an empty area. The BMUPDATE command allows to re-associate the feature with its placement face.

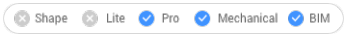
Automatic
The placement face for the feature is detected automatically.

Manual
Select a new placement face manually.

Entire model
Updates the entire model.

7.121 BMVSTYLE command

Applies a visual style to a mechanical component inserted.



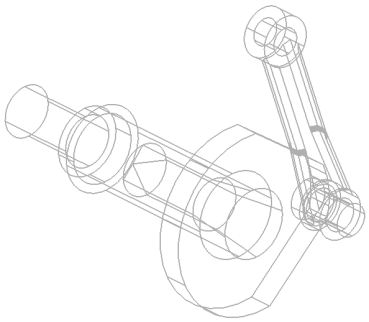
7.121.1 Description

Applies a chosen visual style to all or only some inserted components by typing ALL or the component's names inserted, separated by slashes (/). Component insert names are case sensitive.

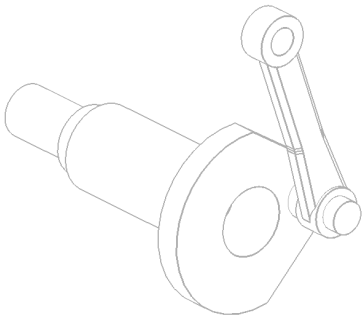
7.121.2 Options

by Viewport
Applies the current visual style of the viewport.

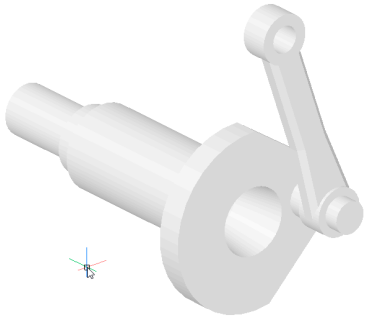
Wireframe
Applies the 3D Wireframe visual style.



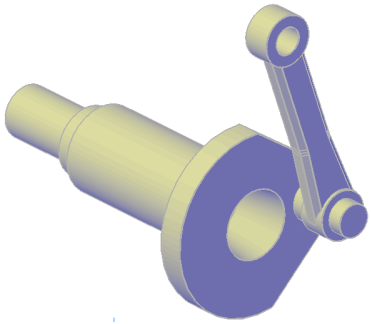
Hidden
Applies the Hidden visual style.



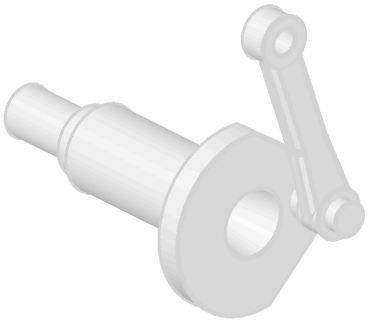
Realistic
Applies the Realistic visual style.



Conceptual
Applies the Conceptual visual style.



Modeling
Applies the Modeling visual style.



by Name
Applies a named visual style.

7.122 BMXCONVERT command

Converts X-Hardware solids to mechanical components.

Shape Lite Pro Mechanical BIM

7.122.1 Description

Converts X-Hardware solids in the current drawing to mechanical components.

7.123 BOUNDARY command

Creates closed polylines from bounding entities.

Shape Lite Pro Mechanical BIM

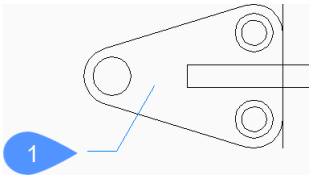
Icon:

Alias: BO, BPOLY

7.123.1 Description

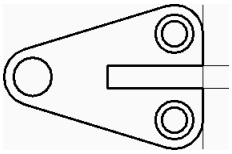
Creates closed polylines, defined by surrounding entities. Options allow you to pick an internal point, specify the boundary set, and detect islands.

Original entities:



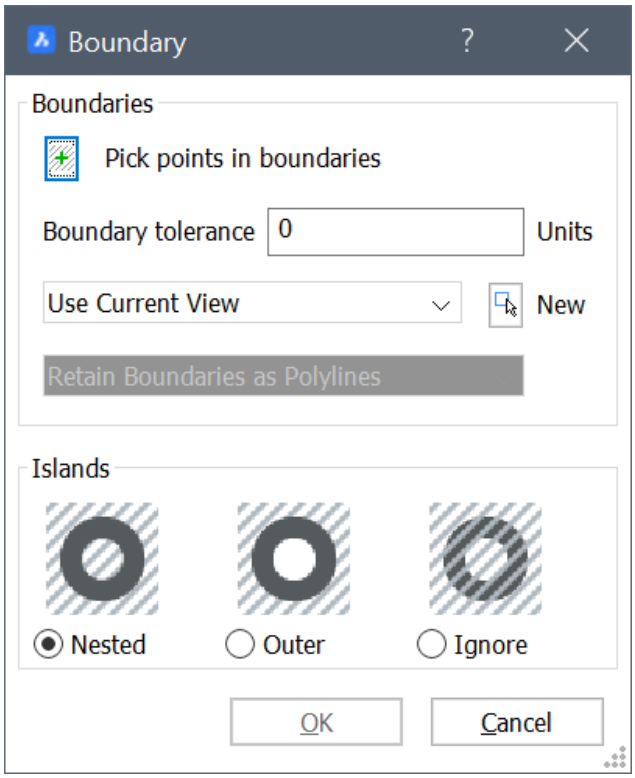
1 Internal point

Resulting polylines:



7.123.2 Boundary dialog box

Begin creating boundaries by selecting options in the Boundary dialog box.



Boundaries

Specify boundary options.

Pick points in boundary

Specify a point inside an enclosed area for which you want to create a boundary. You can continue picking additional points until you press Enter to return to the Boundary dialog box.

Additional options: [Select entities/Undo]

Boundary tolerance

Specify the largest gap that can exist in the boundary for BricsCAD to consider the area closed. When boundary tolerance is 0, no gaps can exist.

Boundary set

Specify where BricsCAD should search for entities that make up the boundary.

- **Use Current View**: search all entities in the current viewport.
- **Use Boundary Set**: search only the current selection set.

- **New**: create a new selection set. Press Enter to finish selecting entities and return to the Boundary dialog box.

Retain Boundaries as Polylines [Read only]
Indicates how boundaries are retained.

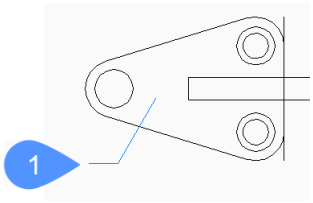
Islands
Specify island detection. An island is an enclosed area inside a boundary.

Nested
Treat every island as a boundary.

Outer
Create a boundary of only the outermost entities.

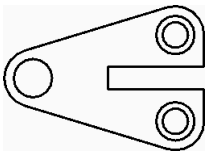
Ignore
Ignore the innermost areas. A boundary is created between the outermost area and islands.

Original entities:

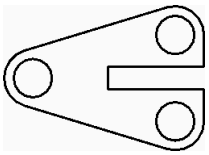


1 Internal point

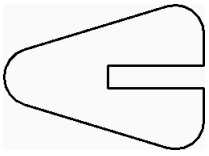
Nested:



Outer:



Ignore:

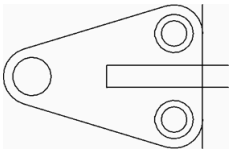


7.123.3 Options within the BOUNDARY command

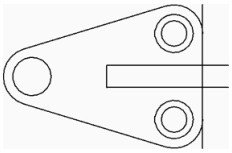
After you begin creating boundaries, the following options may be available:

Select entities
Select entities to use as boundaries.

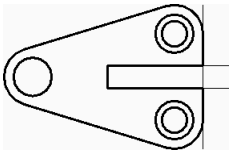
Original entities:



Selected entities:



Resulting polylines:



Undo
Undo the last internal pick point and continue picking points to specify additional enclosed areas.

7.124 -BOUNDARY command

Creates closed polylines from bounding entities.

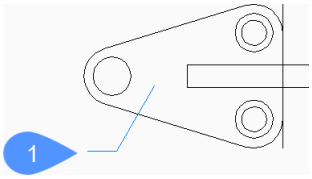
Shape Lite Pro Mechanical BIM

Alias: -BO

7.124.1 Description

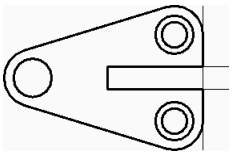
Create closed polylines, defined by surrounding entities using the Command line. Options allow you to pick an internal point, specify the boundary set, and detect islands. See the BOUNDARY command to access similar functionality using a dialog box.

Original entities:



1 Internal point

Resulting polylines:



7.124.2 Methods to create a boundary

There is one method to begin creating a boundary:

- Internal point

Internal point
Begin creating boundaries by specifying a point inside an enclosed area for which you want to create a boundary. You can continue picking additional points until you press Enter to end the command.
Alternate options: [Advanced/Undo]

7.124.3 Options within the -BOUNDARY command

After you begin creating boundaries, the following options may be available:

Advanced
Choose to modify boundary options or island detection.

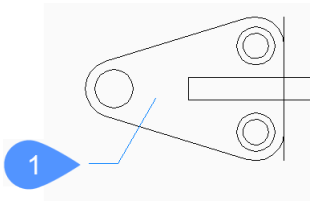
Boundary
Specify where BricsCAD should search for entities that make up the boundary.

- **New**: create a new selection set of entities that make up the boundary.
- **Everything**: select all entities in the current viewport.

Island
Specify the island detection. An island is an enclosed area inside a boundary.

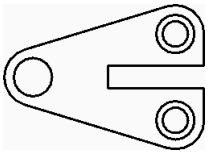
- **Yes**: treat every island as a boundary.
- **No**: create a boundary of only the outermost entities.
- **Outer only**: ignore the innermost areas. A boundary is created between the outermost area and islands.

Original entities:

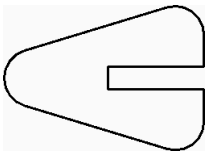


1 Internal point

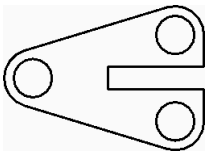
Yes:



No:



Outer only:



Exit
Return to the previous prompt.

Undo
Undo the last internal pick point and continue picking points to specify additional enclosed areas.

7.125 BOX command

Creates a 3D solid in the shape of a box.

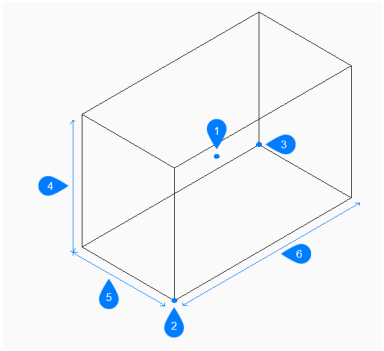
Note: In BricsCAD Lite, which doesn't support 3D Solids, the BOX command launches the AI_BOX command.

Shape Lite Pro Mechanical BIM

Icon:

7.125.1 Description

Create a 3D solid in the shape of a rectangular or square box. Choose from a combination of options including corner, center, length, width, height, and cube.



1 Center of box

2 Corner of box

3 Opposite corner

4 Height

5 Width

6 Length

7.125.2 Methods to create a box

This command has 2 methods to begin creating a box:

- Set corner of box

- Center

Set corner of box
Begin creating a box by specifying a corner for the base of the box then:

Set opposite corner
Specify the opposite corner for the base of the box to apply both the length and width. The box is created parallel to the x and y axes.
Additional options: [Cube/Length of side]

Height of box
Specify the height of the box.
Additional option: [2Point]

Center
Begin creating a box by specifying the center of the box then:

Set corner of box
Specify a corner on the midplane of the box to apply both the length and width. The box is created parallel to the x and y axes.
Additional options: [Cube/Length of side]

Height of box
Specify the height of the box.
Additional option: [2Point]

7.125.3 Options within the BOX command

After you begin creating a box, the following options may be available:

Cube
Specify a single distance to use for the length, width, and height of the box.

Length of side
Specify the length of the side of the box then:

Width of box
Specify the width of the box.

Height of box
Specify the height of the box.
Additional option: [2Point]

2Point
Specify the height of the box by picking any two points.

7.126 BREAK command

Removes a portion of an entity.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: BR

You can break arcs, circles, ellipses, lines, polylines, rays and infinite lines.

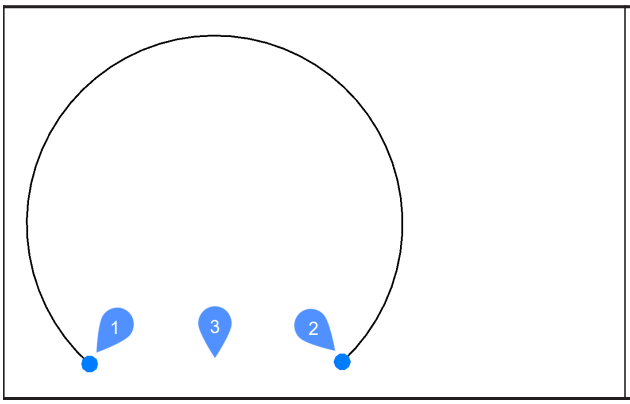
Note: Breaking a circle converts it to an arc. A ray is broken into a ray and a line. An infinite line is broken into two rays.

7.126.1 Method

When breaking entities, you must specify two points for the break. The portion between the two break points is removed.

On arcs and circles, the break occurs in the counter-clockwise direction from first to second break points.

Note: By default, the point you use to select the entity becomes the first break point.



1 First break point
2 Second break point
3 Removed portion

7.126.2 Options within the command

First break point
Specifies the start of the portion of the entity to remove.

@
Specifies that the first and second break points are at the same location on the entity. The selected entity is broken into two connected parts.

7.127 BROWSER command

Opens the default Web browser.

Shape Lite Pro Mechanical BIM

7.127.1 Description

Opens the default Web browser to browse the internet starting from a specified URL. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

7.128 BSAVEAS command

Opens the Save block definition dialog box.

Shape Lite Pro Mechanical BIM

7.128.1 Description

Open the Save block definition dialog box to copy the current block definition to a new name. This command is only available from the Block Editor.

8. C

8.1 CAL command

Opens the BricsCAD Calculator.

Shape Lite Pro Mechanical BIM

8.1.1 Description

Opens the BricsCAD Calculator to perform common calculation and conversion operations. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

8.2 CALLOUT command

Shape Lite Pro Mechanical BIM

8.2.1 Description

This is a service command which is not supposed to be entered directly by the user. It is used by the program when a context menu item is selected.

8.3 CAMERA command

Places camera glyphs in drawings that point at target points, and creates named views.

Shape Lite Pro Mechanical BIM

Icon: 

8.3.1 Method

Specify the location of the camera and the location of the target (the point at which the camera looks).

A camera glyph displays in the drawing to indicate the position of the camera.

Note: A camera glyph displays in the drawing to indicate the position of the camera.

8.3.2 Options

?

Displays a list of existing cameras. Press Enter to list all existing cameras.
Use wildcards (* or ?) to list a selection of cameras. For example, Cam* lists all cameras names that start with 'cam' and ?a* lists all camera names of which the second letter is 'a'

Name

Names the new camera.

LOcation

Positions the camera by picking a point in the drawing or typing the x, y, z coordinates in the Command line.

Height
Sets the height (z coordinate) of the camera.

Target
Positions the target, the point at which the camera looks by picking a point in the drawing or typing the x, y, z coordinates in the command line.

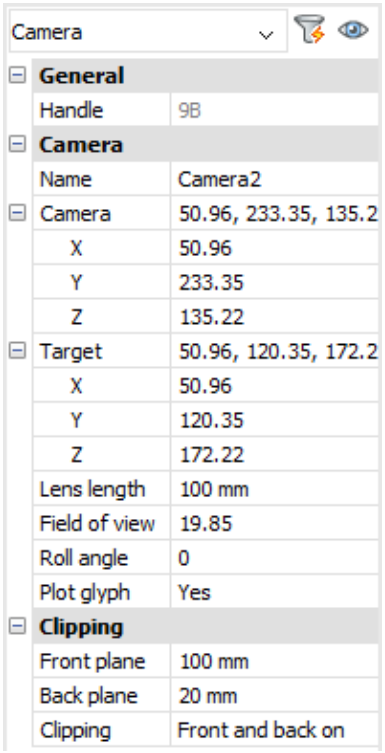
LEns
Defines the lens length. A smaller number, such as 20, provides a wider field of view, while a higher number, such as 200, provides a closer look, like a zoom lens on a camera.

CLipping
Defines the front and back clipping planes, which cut off the view.

View
Sets the camera as current view.

Note: Properties of a camera view can be edited:

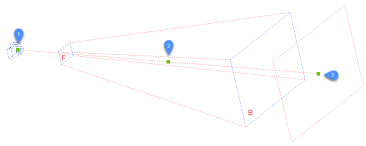
- in the dialog box displayed by the VIEW command.
- in the Properties panel after selecting the camera glyph in the drawing.
-



Grips Editing
Cameras can be edited directly through grips:
Select the camera glyph in the drawing. Notice that it has three grips. Front (F) and back (B) clipping planes display in a blue dotted line.
Drag the grips to perform the following actions:

- Position the camera (1).
- Move the camera definition as a whole (2).

- Position the target (3).



8.4 Команда CENTER

Включает/выключает объектную привязку Центр.

Shape Lite Pro Mechanical BIM

Значок:

8.4.1 Description (Описание)

Включает/выключает объектную привязку Центр, которая выполняет привязку к центру объекта. Вы можете запустить эту команду с помощью командной строки. Команда CENTER изменяет значение системной переменной OSMODE. Вы также можете запустить эту команду при выполнении другой команды, чтобы включить привязку Центр и отключить остальные объектные привязки только для текущей операции. В этом случае значение системной переменной OSMODE не изменяется.

8.5 CENTERDISASSOCIATE command

Breaks the association of a centerline entity with two selected lines, or else a center mark with a circle or an arc.

Shape Lite Pro Mechanical BIM

Icon:

8.5.1 Description

Breaks the association of centerline and center mark entities with the associated entities, such as lines, arcs, and circles.

8.6 CENTERLINE command

Creates a centerline.

Shape Lite Pro Mechanical BIM

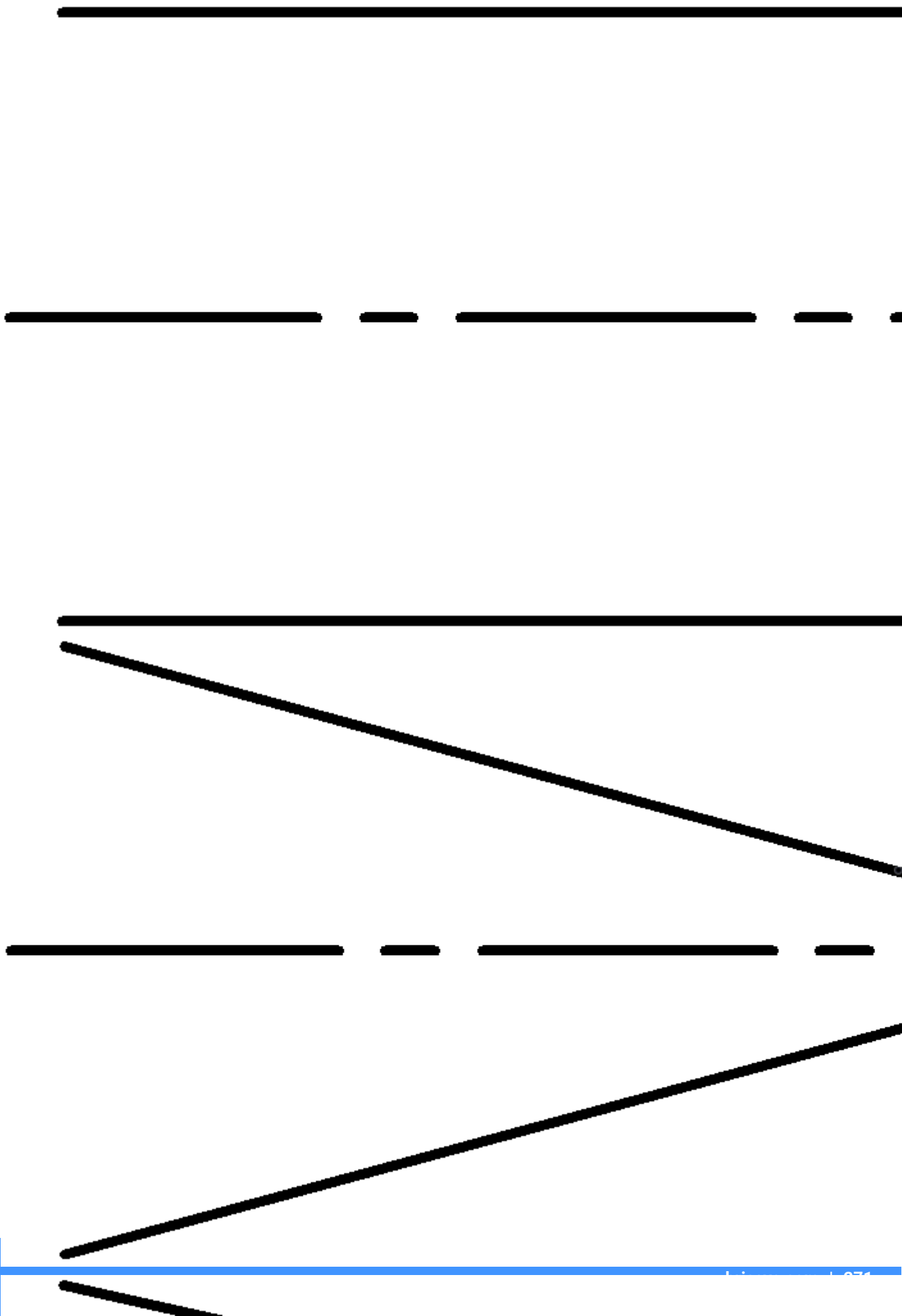
Icon:

Alias: CL

8.6.1 Description

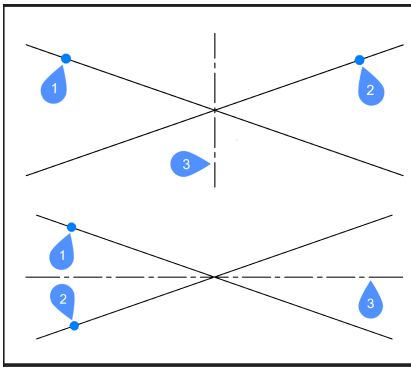
Creates a centerline geometry associated with two selected lines or polyline segments.

Note: Centerlines are drawing reference entities to indicate axes of symmetry.



8.6.2 Method

When applying a centerline to the intersection of two lines, the location of the picked points determines the direction of the centerline.

	1 First segment selected. 2 Second segment selected. 3 Resulting centerline.

The lines and polyline segments can be selected inside blocks and in drawing view viewports. You can pick two segments of the same polyline.

Note: The centerline is associative, so when one or both lines are moved, the centerline repositions itself.

Note: Centerlines can be edited through grips. It can be moved and lengthened by dragging the grips. The CENTERRESET command can be used to reset the centerline.

8.7 CENTERMARK command

Creates a center mark.

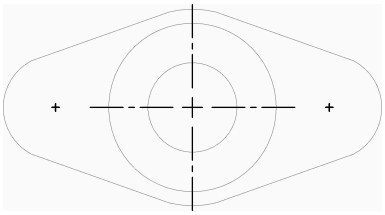
Shape Lite Pro Mechanical BIM

Icon:

Alias: CM

8.7.1 Description

Creates a center mark associated with a selected circle, arc, or polyarc.



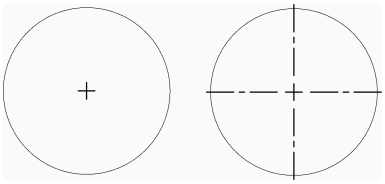
8.7.2 Methods to create a center mark

There is one method to begin creating a center mark:

- Select circle or arc

Select circle or arc

Begin creating a center mark by selecting a circle, arc or polyarc. Depending on the current value of the CENTERMARKEXE system variable, the center mark is drawn with or without extension lines. Additional center settings further control the appearance of centermarks.



The center mark is associative, so when the arc or circle is moved or resized, the centermark follows.
Additional options: [selection options (?)]

8.7.3 Options within the CENTERMARK command

After you begin creating a center mark, the following option may be available:

8.8 CENTERREASSOCIATE command

Associates a centerline entity with two selected lines, or else a center mark with a circle or an arc.

Shape Lite Pro Mechanical BIM

Icon:

8.8.1 Description

Selects the center mark or a centerline to associate with their specific entities.

If you select a center mark you have to specify a circle or an arc (circular entity) to associate with.

If you select a centerline you have to specify two lines to associate with.

8.9 CENTERRESET command

Resets centerline and center mark entities.

Shape Lite Pro Mechanical BIM

Icon:

8.9.1 Description

Centerlines are reset by the program when one of the associated lines is moved. Center marks are reset automatically when the associated circle or arc is moved or when their radius or diameter is modified.

8.10 CHAMFER command

Creates chamfers at intersections, defined by two lengths, or a length and an angle.

Shape Lite Pro Mechanical BIM

Icon:

Alias: CHA

8.10.1 Methods

There are four methods to create chamfers:

- Create chamfer by specifying two distances.
- Create chamfer by length and angle.
- Create chamfers along a polyline.
- Create chamfers on a 3D solid edge (obsolete, replaced by DMCHAMFER command).

Note: : The command cannot place a chamfer between two polylines, but it can place a chamfer between a line and a polyline, even a closed polyline.

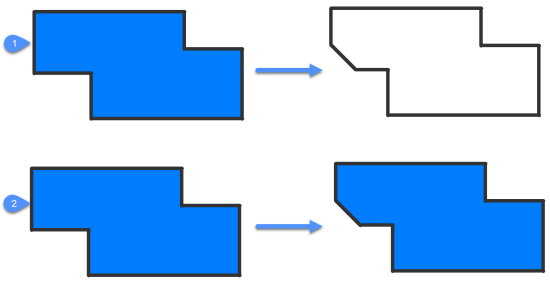
Note: To connect two entities without creating a chamfer, hold the Shift key when selecting the second entity. This acts like a combined Trim-Extend command.

8.10.2 Options

chamfer Settings
Opens the Settings dialog box at the chamfer parameters.

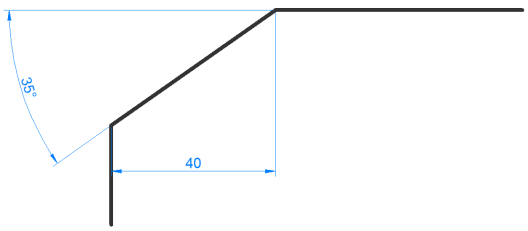
Polyline
Chamfers the vertices where two segments meet of the selected polyline (if applicable).

Note: Creating chamfers on a hatch boundary created with individual lines (1) results in removal of hatch associativity. Associativity is maintained if the boundary is defined from a polyline (2).

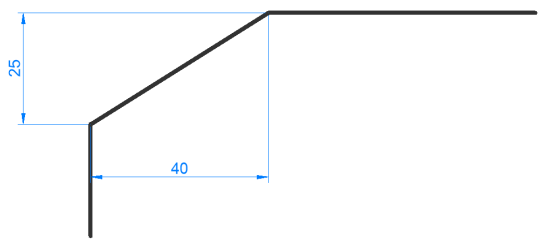


Angle
Changes the method to length-angle.

Note: BricsCAD measures the angle from 0 degrees, the X axis.



Distance
Changes the method to distance-distance.



mEthod
Specify between angle and distance methods.

Note: The program will keep using the same method for determining the chamfer until the method is changed again.

Trim
Determines if the entities are trimmed or extended to meet the endpoints of the chamfer line.

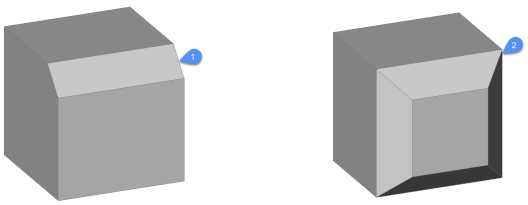
Undo
Undoes the last chamfer when in multiple mode.

Multiple
Create multiple chamfers with the same settings. Press the Esc key to exit the command.

Create 3D chamfer
Create 3D chamfer by selecting the edge of a 3D solid or surface.

Enter surface selection option
Select between surfaces adjacent to the selected edge.

Select edge or Loop
Select edge (1) to be chamfered or loop around selecting all tangential edges belonging to the base surface (2).



8.11 CHANGE command

Changes the properties of entities, through the Command line. This command has largely been superseded by the PROPERTIES panel.

Shape Lite Pro Mechanical BIM

Icon:

Alias: -CH

8.11.1 Method

Choose one or more entities in the drawing and choose a change point which changes the size of the entity, depending on what the entity is:

- Lines - moves the endpoint closest to the pick point.
- Arcs - changes the length closest to the pick point.
- Circles - changes the diameter.
- Text - moves the text.
- Blocks - moves the block.

Note: This option has no effect on polylines.

8.11.2 Options

special Entities
Specifies options for changing the properties of rays and infinite lines (xlines):
Point change for Rays and Infinite Lines affects:

change Angles
Changes the angle.

definition Points
Changes the location of the points defining the rays and infinite lines.

No changes
Leaves entities unchanged.

Properties…
Color/Elevation/LAyer/LineType/linetype Scale/Line Weight/ Thickness/TRansparency/Material/Annotative

For more explanation, see the Properties panel.

8.12 CHECKSTANDARDS command

Checks the current drawing for inconsistencies that violate the standards.

Shape Lite Pro Mechanical BIM

8.12.1 Methods

The CHECKSTANDARDS command is used to view and manage the violations of standards in the current drawing.

This command allows the user to fix or ignore each reported violation. The user can hide or show the ignored violations. If the ignored violations are hidden, they will no longer be reported as violations. See the STANDARDSOPTIONS system variable.

The user can choose to be notified about the standards violations. When the notification option is on, the user will be notified with an alert when a violation occurs. See the STANDARDSVIOLATION system variable.

The purpose of this command is to make some changes in the current drawing based on the standards that are defined for the drawing.

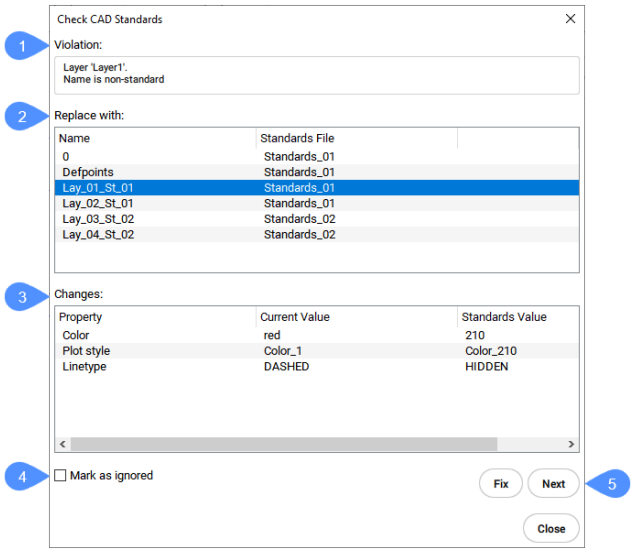
A standard is a set of properties of a named object in the standards file.

The properties of named objects in the current drawing are compared with the definitions in the standards file. When the named object does not comply with any standard, a violation occurs. In this case, the user can choose to apply a replacement. A replacement assigns a standard to a named object in the current drawing. All the properties of this named object are replaced by the values in the standard.

This command has an associated dialog box that manages all necessary operations.

8.12.2 CHECKSTANDARDS dialog box

The CHECKSTANDARDS dialog box allows to select the options for the CHECKSTANDARDS command.



1 Violation

2 Replace with

3 Changes

4 Mark as ignored

5 Fix / Next / Close

Violation

This section contains a description of the current violation. The current violation includes a target named object in the current drawing that does not comply with the standards. The description includes the target named object type and name and the first property of the target named object.

Replace with
This section contains a list of standards that can be applied to the target named object. An entry of the list includes the name of the source named object in the standards, and the standards file name where this source named object is located. Clicking on an entry will highlight it and make it active.

Changes
This section displays the changes that can be made to the target named object. It contains a table that displays the properties of the target named object, in the current drawing. For each property, the table displays the current value and the standards value. The current value is the property value that is currently assigned to the target named object. The standards value is the property value defined in the standard. When the user chooses to make the change, the properties of target named object will take the values in the chosen standard.

Mark as ignored
This option allows to ignore a certain violation. The user can choose to hide the ignored violations. See the STANDARDSOPTIONS system variable.

Fix / Next / Close
This section contains the buttons assigned to the actions that can be taken.

Fix
Applies the selected standard to the target named object.

Next
Proceeds with the next violation in the current drawing.

Close
Closes the dialog box.

8.13 CHPROP command

Changes the properties of the selected entity.

Shape Lite Pro Mechanical BIM

8.13.1 Description

Changes the properties of the selected entities, with fewer options than the CHANGE command. You can change the properties for one or multiple entities.

Note: This command is largely superseded by the properties panel.

8.13.2 Options within the command

Color
Changes the color of the entity.

COlorbook
Select colorbook to append.

LAyer
Changes the layer to which the entities are assigned.

LineType
Changes the linetype. Linetype must be loaded into the drawing.

linetype Scale
Changes the linetype scale.

Line Weight
Changes the lineweight.

Thickness
Changes the thickness.

TRansparency
Changes the transparency.

Material
Applies a material.

Annotative
Sets the annotative property.

8.14 CHSPACE command

Moves entities from paper space to model space and vice versa.

Shape Lite Pro Mechanical BIM

Note: This command can only be used in a Paper Space.

8.14.1 Method

There are two methods to use CHSPACE command:

- Moves entities from paper space to model space.
- Moves entities from model space to paper space.

8.14.2 Options within the command

Note: The following options are available only with multiple viewports active

Select TARGET viewport
Select target viewport as reference.

Note: The scale factor is the inverse of custom scale of the selected target viewport. For example, if the custom scale = 1/2 (0.5), then the scale factor = 2.

Select SOURCE viewport
Select source viewport as reference.

Note: The scale factor equals the custom scale of the selected source viewport.

8.15 CIRCLE command

Creates circle entities.

Shape Lite Pro Mechanical BIM

Icon:

Alias: C

8.15.1 Description

Creates a circle from a combination of options including the center, radius, two points, three points or tangent to entities. You can also convert an arc to a circle.

8.15.2 Methods

There are 7 different methods to create circle entities:

Center-Radius

Center-Diameter

2 Point

3 Point

Tangent-Tangent-Radius

Turn Arc into circle

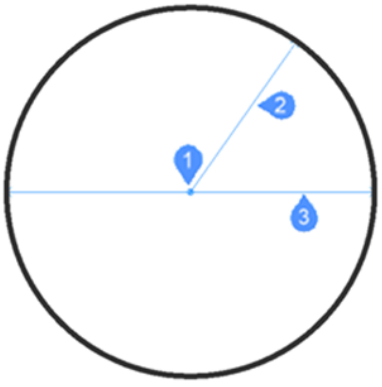
Multiple circles

8.15.3 Options within the CIRCLE command

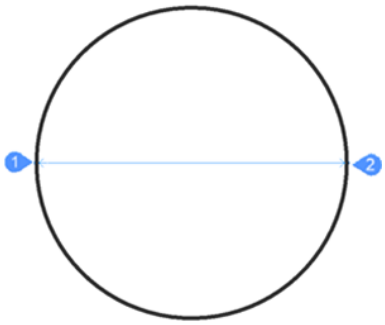
Select center of circle
Specify a point (1)

Set Radius
Specify a point (2)

Set Diameter
Specify a point (3)

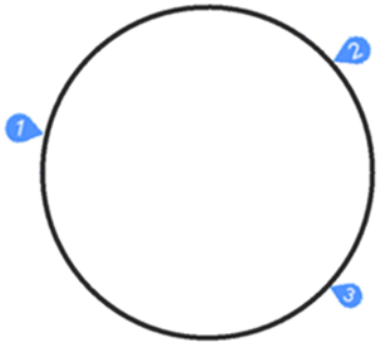


2P
Specify 2 diametrically opposite points (1 and 2) to define the circle.



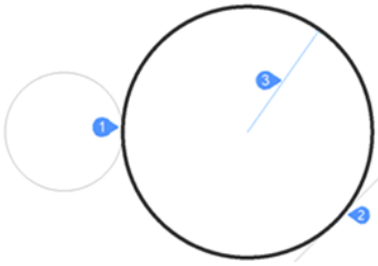
3P

Specify 3 points to define the circle. Tangent entity snap allows defining a circle by picking 3 tangent points to other entities. The TanTanTan method is also available by AI_CIRCTAN command.



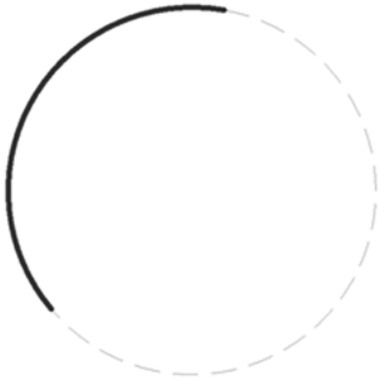
Tangent-Tangent-Radius

Define the circle by two tangent points (to other entities – 1 and 2) and its radius. When you specify a radius that is not possible with the selected tangents, you are prompted to specify the second tangent and the radius again.

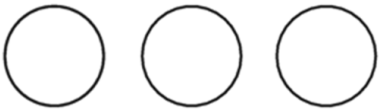


Turn Arc into circle

Converts an arc entity into a circle by extending it to 360 degrees.



Multiple circles
Create multiple circles with the same size using one of the listed methods above. Press Enter to end the command.



8.16 CIVIL3DIMPORT command

Creates BricsCAD civil entities from an Autodesk® Civil 3D drawing.

Shape Lite Pro Mechanical BIM

8.16.1 Description

Opens the **Import Civil 3D** dialog box which allows you to import AutoCAD® Civil 3D entities from a selected .dwg file, or to convert them directly from the current drawing.

The following objects from AutoCAD® Civil 3D can be imported, which are in BricsCAD® also created as Civil entities:

- Cogo points and point groups
- TIN Surfaces
- TIN Volume Surfaces
- Grid Surfaces
- Alignments (both Alignments by PI and Alignments by Elements are supported)
- Profile Views
- Profiles (Vertical Alignments)

8.17 CIVILDWGEXPORT command

Exports BricsCAD Civil entities as native CAD entities.

Shape Lite Pro Mechanical BIM

Icon: DWG

8.17.1 Description

Opens the **Save Drawing As** dialog box which allows you to export Civil entities as native CAD entities in a new drawing.

8.18 CIVILPOINT command

Creates **Civil Points**.

Shape Lite Pro Mechanical BIM

Icon:

8.18.1 Description

Civil Points are defined in BricsCAD® as a Civil entities, which are represented by symbols and labels. In addition to XYZ coordinates, Civil Points can have a variety of attributes assigned, including point number, point name, raw (field) description and full (expanded) description.

Use the Properties panel to edit the properties of an individual point or a group of selected points.

For improved organization, Civil Points can be collected into point groups based on certain criteria determined by filters.

You can use the basic BricsCAD® commands to edit Civil Points in a drawing. For example, COPY, COPYCLIP, PASTE, MOVE, ROTATE...

You can use Civil Points as input objects for creating a TIN Surface, which is automatically updated when editing points.

8.18.2 Method

There are two methods to insert Civil Points:

- Insert a single point by specifying its location in a drawing.
- Import multiple points from an ASCII point file.

  Note: In addition to the basic ASCII file, points can also be imported from a LandXML file using the LANDXMLIMPORT command.

8.18.3 Options within the command

Specify point location
Allows you to specify the location of a new individual point in a drawing.

Import points from file
Opens the Open LandXML file dialog box which allows you to import Civil Points from an ASCII point file. Currently, only XYZ coordinates are imported from an ASCII point file, without additional attributes that may be included in the input file. The input point file can have any extension, for example .xyz, .txt, .asc...

Enter point description
Allows you to specify a description for the Civil Point.

Enter point elevation
Allows you to specify an elevation for the Civil Point.

Enter point file delimiter

Allows you to specify a character that separates the point data (attributes) in the specified ASCII point file.

Columns in the input file can be delimited by any character (delimiter), which is specified by the user when importing the file.

set Style

Allows you to specify a point symbol style when inserting a new point.

Note: More information on point styles and labels can be found in the **Working with Civil points** article.

?

Lists the names of the available point symbol styles in the Command line.

8.19 CIVILPOINTATTRIBUTES command

Adds or removes user-defined attributes to Civil Points.

Shape Lite Pro Mechanical BIM

Icon:

8.19.1 Options within the command

Select civil points

Allows you to select Civil Points to add user-defined attributes.

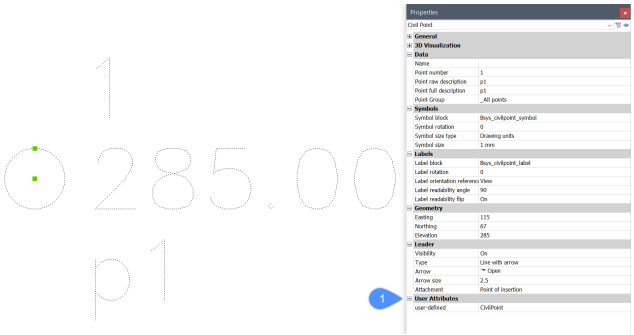
? to list user attributes

Lists the existing user-defined attributes on Civil Points in the Command line.

Add attribute

Adds attributes for the selected Civil Point by specifying a key (name) and a value.

User defined attributes are added to the **User Attributes** (1) section in the Properties panel.

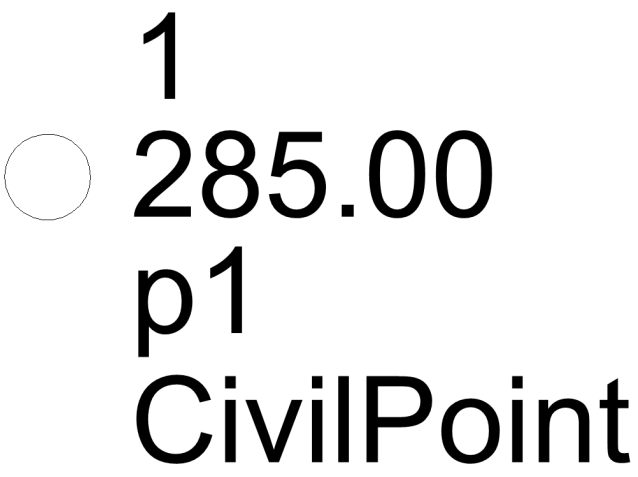


You can use the block editor to add a new custom attribute to the Civil point **Symbol block**. The new attribute is added by writing the attribute key in angle brackets: **<user-defined>**. The attribute key can be written as Text or as an Attribute definition.

User defined attribute added in the block editor:

<Number>
<Elevation>
<Full Description>
<user-defined>

The Civil point Symbol block displayed in the Layout view:



Remove

Removes user-defined attributes on Civil Points based on the specified attribute keys.

8.20 CIVILPOINTGROUP command

Creates **Civil Point** groups.

Shape Lite Pro Mechanical BIM

Icon: 

8.20.1 Description

Depending on the specified filters, points can be included or excluded from the point group.

8.20.2 Method

There are two ways to use the command:

- Create a new point group and add filters to include or exclude points from the group.
- Edit filters of an existing point group.
- With this method, you can specify the name of the existing point group in the first step and then you can edit the filters.

8.20.3 Options within the command

Enter point group name

Allows you to enter the name of a new point group for which you will create a new filter. It also allows you to enter the name of an existing point group for which you want to edit the filters.

filter to Include

Allows you to create a new or to edit an existing filter in which you specify the criteria for including points in the selected point group.

First, select the name of existing Civil Point attribute in the Command line. Next, specify either an individual attribute value, multiple values, a range of values, or any combination of the listed options, based on which points are included in the desired point group. When specifying multiple attribute values, a comma (,) without a space is used for separation.

In addition to entering full attribute values/keys, filters can be defined in other ways:

- Only specify the first character of the name and the "*" sign, such as "a*", where the filter will take into consideration all names starting with "a".
- Specify the first and last character with an intermediate sign "*", such as "a*z", where the filter will take into consideration all names starting with "a" and ending with "z".
- Specify sign "*" and a last character, such as "*z", where the filter will take into consideration all names ending with "z".
- Specify individual characters, missing characters are replaced with the "*" sign, such as "a*b*" where the filter will take into consideration all names starting with "a", and with the third character "b".

Note:

- Multiple filters can be added to the existing point group.
- Once an individual filter is created in a particular drawing, it also affects all the points that are inserted later in that drawing.

filter to Exclude

Allows you to create a new or to edit an existing filter in which you specify the criteria for excluding points from the selected point group.

The procedure of adding a new filter is exactly the same as described in the **filter to Include** section.

? to list point groups

Lists existing point groups in the drawing on the Command line.

Specify point numbers to include

Allows you to enter the point numbers to be included in the specified filter.

Specify point numbers to exclude

Allows you to enter the point numbers to be excluded from the specified filter.

Name

Allows you to enter the point names to be included in the specified filter, if the option **filter to Include** is selected in the previous step.

Allows you to enter the point names to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

Elevation

Allows you to enter the point elevations to be included in the specified filter, if the option **filter to Include** is selected in the previous step.

Allows you to enter the point names to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

An example of an Elevation filter with multiple attribute values, including a range of values, looks like this: "100-200,>400". This filter includes all points with an elevation range between 100 and 200 meters and at the same time all points with an elevation higher than 400 m.

Let's add a new filter to our sample point group to include points based on points raw description: "Manhole".

Based on the additional filter, only points that meet criteria of both filters: Elevation="100-200,>400" and Raw Description="Manhole" are included in the example point group.

Full description

Allows you to enter the point full descriptions to be included in the specified filter, if the option **filter to Include** is selected in the previous step.

Allows you to enter the point full descriptions to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

Raw description

Allows you to enter the point raw descriptions to be included in the specified filter, if the option **filter to Include** is selected in the previous step.

Allows you to enter the point raw descriptions to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

User attribute Key

Allows you to enter the user-defined attribute keys (attribute names) to be included in the specified filter, if the option **filter to Include** is selected in the previous step.

Allows you to enter the user-defined attribute keys (attribute names) to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

User attribute Value

Allows you to enter the user-defined attribute keys (attribute names) to be included in the specified filter, if the option **filter to Include** is selected in the previous step.

Allows you to enter the user-defined attribute keys (attribute names) to be excluded from the specified filter, if the option **filter to Exclude** is selected in the previous step.

8.21 Команда CIVILPOINTGROUPEDIT

Редактирует группы **точек Civil**.

Shape Lite Pro Mechanical BIM

Значок: 

8.21.1 Description (Описание)

Эта команда позволяет изменить название выбранной группы точек или удалить ее.

8.21.2 Опции команды

Введите группу точек для редактирования

Позволяет ввести имя группы точек для редактирования.

? для списка групп точек
Отображает в командной строке перечень групп точек, существующих в чертеже.

? для списка точек
Отображает в командной строке все точки, входящие в выбранную группу точек.

Удалить
Позволяет удалить выбранную группу точек.

Переименовать
Позволяет переименовать выбранную группу точек.

8.22 CLEANSCREENOFF command

Displays user interface elements that were hidden by the CLEANSCREENON command.

Shape Lite Pro Mechanical BIM

Note: Do not use the numeric keypad. On AZERTY keyboards: do not press the Shift key when hitting the number key.

8.23 CLEANSCREENON command

Makes the drawing area larger by hiding elements of the user interface.

Shape Lite Pro Mechanical BIM

Note: Do not use the numeric keypad. On AZERTY keyboards: do not press the Shift-key when hitting the number key.

8.23.1 Method

Elements in the user interface are hidden according to the values of the CLEANSCREENOPTIONS system variable.

8.24 CLEANUNUSEDVARIABLES command

Purges parametric variables not used by constraint expressions and not linked to dimensions.

Shape Lite Pro Mechanical BIM

8.24.1 Description

Automatically deletes parametric variables not used by constraint expressions and not linked to dimensions when the command is run.

8.25 CLIPDISPLAY command

Toggles the Clip Display property of section planes and BIM section entities.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The display of the section entity and be toggled also through the Properties panel.

8.26 CLOSE command

Closes the current drawing.

Shape Lite Pro Mechanical BIM

8.26.1 Description

Closes the current drawing after it has been saved. If changes have been made since the last save, a BricsCAD dialog box offers you the opportunity to save the drawing before closing it.

8.27 COLOR command

Opens the Select color dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: COL, COLOUR, DDCOLOR, DDCOLOUR, SETCOLOR

8.27.1 Description

Opens the Select color dialog box to specify the current color.

8.28 -COLOR command

Sets the current working color, through the Command line.

Shape Lite Pro Mechanical BIM

Alias: -COL, -COLOUR

8.28.1 Methods

Enter the name, number, or RGB value of a color:

- Specify a color name: Red, Yellow, Green, Cyan, Blue, Magenta, White, ByLayer, or ByBlock.
- Specify a number from 0 to 256, which represents the Index Color.
- Specify a value for each of red, green, and blue, which represents the True Color. The range is from 0 to 255. For example, white is 255,255,255 and gray is 128,128,128.
- Specify a bookcolor name.

8.29 COMMANDLINE command

Opens the Command line panel.

Shape Lite Pro Mechanical BIM

8.29.1 Description

Opens the Command line panel to display it in the current workspace. The Command line panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Command line panel can be either floating, docked or stacked.

8.30 COMMANDLINEHIDE command

Closes the Command line panel.

Shape Lite Pro Mechanical BIM

8.30.1 Description

Closes the Command line panel to hide it from the current workspace. If the Command line panel is stacked when you close it, the Command line tab or icon is removed from the stack.

8.31 COMMANDS command

Lists the names of the commands.

Shape Lite Pro Mechanical BIM

8.31.1 Description

Lists the names of commands available in the program according to your license level, first the name in English followed by the localized command name.

8.31.2 Options

Internal
Lists command names native to BricsCAD.

External
Lists command names from add-on applications.

Both
List both internal and external command names.

8.32 COMMUNICATORINFO command

Checks whether the Communicator add-on is correctly installed and provides a diagnostic report.

Shape Lite Pro Mechanical BIM

The command is available on Windows only.
Until V21, the major versions of BricsCAD and Communicator should match (e.g. Communicator V20.2.x works with BricsCAD V20.2.x).
Starting with V21 the compatibility between minor releases is supported (e.g. Communicator V21.1.x will work with BricsCAD V21.2.x).

8.32.1 Description

Switches to the Prompt History window and then provides a report.

When Communicator is not installed, then the command reports:

Brief check result: Communicator installation: FAILED

8.33 COMPONENTSPANELCLOSE command

This command is deprecated.

Use LIBRARYPANELCLOSE instead.

8.34 COMPONENTSPANELOPEN command

This command is deprecated.

Use LIBRARYPANELOPEN instead.

8.35 Команда CONE

Создает трехмерное твердое тело в форме конуса.

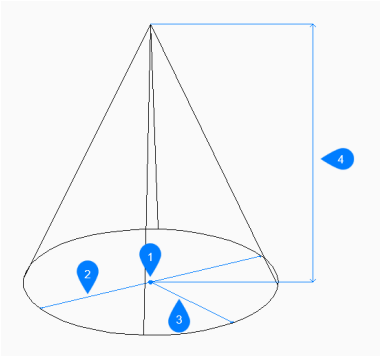
Note: В BricsCAD Lite, который не поддерживает трехмерные твердые тела, команда CONE запускает команду AI_CONE.

Shape Lite Pro Mechanical BIM

Значок:

8.35.1 Description (Описание)

Создает трехмерное твердое тело в форме круглого или эллиптического конуса. Выберите один из вариантов, включая центр, радиус, диаметр, 3 точки, 2 точки, касательные, конечные точки оси и высоту.



1 Центр грани

2 Диаметр

3 Радиус

4 Высота

8.35.2 Методы создания конуса

Эта команда имеет 5 методов для начала создания конуса:

- Средняя точка

- 3 точки
- 2 точки
- Касательные и радиус
- Эллиптический

Средняя точка
Начните создание круглого конуса с указания центра основания:

Радиус основания конуса
Укажите радиус основания конуса.
Дополнительная опция: [Диаметр]

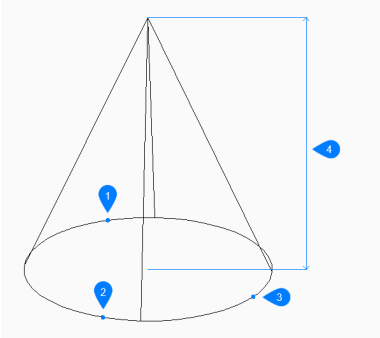
Укажите высоту
Укажите высоту конуса.
Дополнительные параметры: [2Точка/Конечные точки осей /Радиус верхнего основания].

3 точки
Начните создание круглого конуса с указания первой из трех точек на окружности его основания:

Вторая точка
Укажите вторую точку на окружности.

Третья точка
Укажите третью точку на окружности.

Укажите высоту
Укажите высоту конуса.
Дополнительные параметры: [2Точка/Конечные точки осей /Радиус верхнего основания].



1 Точка 1

2 Точка 2

3 Точка 3

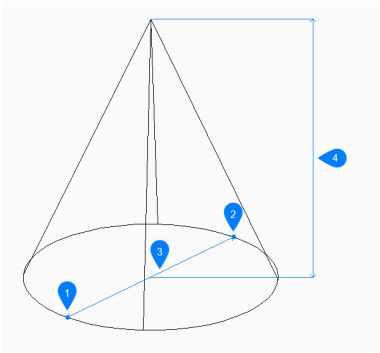
4 Высота

2 точки
Начните создание круглого конуса с указания первой из двух точек на окружности его основания:

Вторая точка диаметра
Укажите вторую точку на диаметре. Эти две точки определяют диаметр основания.

Укажите высоту
Укажите высоту конуса.

Дополнительные параметры: [2Точка/Конечные точки осей /Радиус верхнего основания].



1 Точка 1

2 Точка 2

3 Диаметр

4 Высота

Касательные и радиус

Начните создание круглого конуса с выбора точки касания на первом объекте:

Укажите на объекте вторую точку касания

Выберите точку касания на втором объекте.

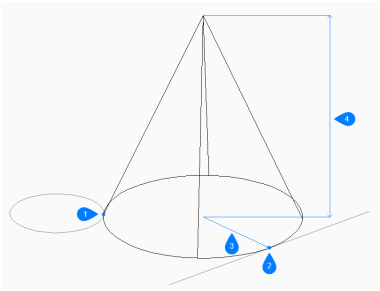
Радиус окружности

Укажите допустимый радиус для основания. Если вы укажете радиус, который невозможен при выбранных касательных, вам будет предложено снова указать касательные и радиус.

Укажите высоту

Укажите высоту конуса.

Дополнительные параметры: [2Точка/Конечные точки осей /Радиус верхнего основания].



1 Касательная точка 1

2 Касательная точка 2

3 Радиус

4 Высота

Эллиптический

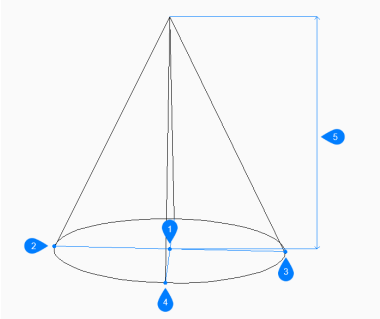
Начните создание эллиптического конуса, указав первый конец оси эллипса, затем:

Альтернативный вариант: [Центр]

Конечная точка оси эллипса
Укажите второй конец оси эллипса.

Вторая точка оси эллипса
Укажите радиус второй оси.

Укажите высоту
Укажите высоту конуса.
Дополнительные параметры: [2Точка/Конечные точки осей /Радиус верхнего основания].



1 Центр грани

2 Первая точка оси эллипса

3 Конечная точка оси эллипса

4 Конец второй оси

5 Высота

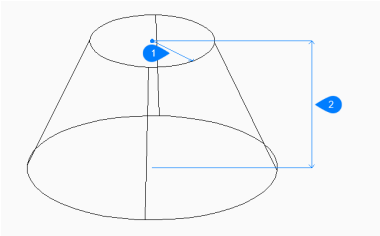
8.35.3 Опции команды

После начала создания конуса могут быть доступны следующие опции:

2 точки
Укажите высоту конуса как расстояние между любыми двумя точками.

Конечные точки осей
Укажите конечную точку оси для определения высоты и ориентации конуса в трехмерном пространстве. Центр основания используется в качестве конечной точки второй оси.

Радиус верхнего основания
Укажите радиус для вершины конуса. Любой радиус больше нуля (0) создает конус с плоской вершиной.



1 Радиус верхнего основания

2 Высота

Диаметр
Укажите диаметр основания конуса.

8.36 CONSTRAINTBAR command

Shows and hides constraint bars.

Shape Lite Pro Mechanical BIM

Icon: 

8.36.1 Description

Shows and hides constraint bars next to entities constrained with geometric constraints.

Note: Constraint Bars are initially hidden when a drawing is opened.

8.36.2 Options

Show
Displays the constraint bar next to the entities you selected.

Hide
Hides the constraint bar next to the entities you selected.

Reset
Relocates constraint bars to their default positions, which is near the midpoint of the entity.

8.37 CONTENTBROWSERCLOSE command

Closes the Content browser panel.

Shape Lite Pro Mechanical BIM

8.37.1 Description

Closes the Content browser panel to hide it from the current workspace. If the Content browser panel is stacked when you close it, the Content browser tab or icon is removed from the stack.

8.38 CONTENTBROWSEROPEN command

Opens the Content browser panel.

Shape Lite Pro Mechanical BIM

8.38.1 Description

Opens the Content browser panel to display it in the current workspace. The Content browser panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Content browser panel can be either floating, docked or stacked.

8.39 CONVERTCTB command

Opens the Select color-dependent plot style table file dialog box.

Shape Lite Pro Mechanical BIM

8.39.1 Description

Opens the Select color-dependent plot style table file dialog box to select a ctb file to convert to an stb file.

8.40 CONVERTPOLY command

Converts 2D and 3D polylines between modern lightweight and classic polylines (heavyweight) definitions.

Shape Lite Pro Mechanical BIM

8.40.1 Method

The main benefit of this command is to reduce the drawing file size by using lightweight polyline definitions.

This command does not convert the following types of polylines:

- Curve-fit.
- Splined polylines.
- Polylines with extended object data stored on their vertices.

Note: BricsCAD® ignores ineligible entities.

You may not need to use this command in the following situations:

- When editing commands apply to both classic and lightweight polylines.
- You can also use the PLINETYPE system variable to specify if classic polylines are automatically converted to lightweight polylines when an older drawing is opened. This system variable determines which style of polyline is created in new drawings.

Note: You can enter this command transparently during commands ('convertpoly).

8.40.2 Options

Heavy
Converts lightweight and 3D polylines to heavyweight ones.
This might be necessary to make drawings compatible with certain software.

Light
Converts heavyweight and 3D polylines to lightweight ones.
After selecting, you are prompted:
Split 3d polyline into planar polylines if it is not planar?[Yes/No] <Yes>:

- **Yes**: 3D polylines are converted.
- **No**: 3D polylines are not converted.

3dPoly
Converts lightweight and heavyweight polylines to 3D polylines.

8.41 CONVERTPSTYLES command

Converts the current drawing from Color Dependent (CTB) to Named (STB) plot style mode and vice versa.

Shape Lite Pro Mechanical BIM

Note: A drawing can use either CTB or STB plot styles, but not both.

8.41.1 Description

Converts the drawing's plot style tables before converting the drawing, using the CONVERTCTB command.

8.42 CONVTOMESH command

Converts entities to mesh entities.

Shape Lite Pro Mechanical BIM

8.42.1 Description

Converts valid 2D and 3D entities to Mesh entities. Valid entities include:

- 3D Solid
- Surface
- Polygon Mesh
- Region
- Closed Polyline

8.42.2 Method to convert to mesh

There is one method to convert entities to Mesh entities:

- Select entities

Select Entities
Begin converting an entity to a Mesh entity by selecting a valid entity then:

Select entities
Select another valid entity or press Enter to complete the command.

8.43 CONVTOSOLID command

Converts entities to solid entities.

Shape Lite Pro Mechanical BIM

8.43.1 Description

Converts valid 2D and 3D entities to 3D Solid entities. Valid entities include:

- Watertight 3D Mesh
- Watertight 3D Surface
- 3D Polygon Mesh
- 3D Polyface Mesh
- Circle with thickness

- Closed Polyline of non-zero thickness and uniform width

8.43.2 Method to convert to solid

There is one method to convert entities to 3D Solid entities:

- Select entities

Select Entities
Begin converting an entity to a 3D Solid entity by selecting a valid entity then:

Select entities
Select another valid entity or press Enter to complete the command.

8.44 CONVTOSURFACE command

Converts entities to Surface entities.

Shape Lite Pro Mechanical BIM

8.44.1 Description

Converts valid 2D and 3D entities to Surface entities. Valid entities include:

- 2D Solid
- 3D Solid
- Region
- Open Polyline with non-zero thickness and zero width
- Line with non-zero thickness
- Arc with non-zero thickness
- Circle with non-zero thickness
- Mesh
- Planar 2D Face

8.44.2 Method to convert to surface

There is one method to convert entities to Surface entities:

- Select entities

Select Entities
Begin converting an entity to a Surface entity by selecting a valid entity then:

Select entities
Select another valid entity or press Enter to complete the command.

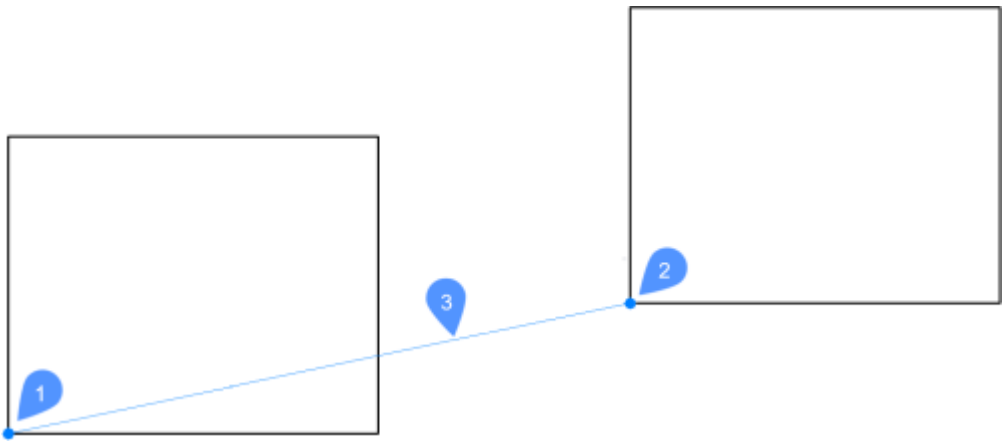
8.45 COPY command

Makes copies of entities.

Shape Lite Pro Mechanical BIM

Icon:

Alias: CO, CP



1 Base point

2 Second point

3 Displacement distance

8.45.1 Method

This command allows you to make a single or multiple copies of entities by entering a base point and a displacement vector.

8.45.2 Options within the command

Displacement
Specify the displacement vector (the distance at which to position the copy). 'Vector' means that you specify the distance and angle at the same time.

Note: When Dynamic Input Mode is active, you can type a distance and angle in the dynamic input fields.

mOde
Toggle between single and multiple copying modes.

Array
Specifies the number of copies to create and the distance between each copy or the distance between the first and last copies.

Multiple
In single-copy mode only: Switches to multiple-copy mode.

Undo
In multiple-copy mode only: Undoes the last copy operation.

Repeat
Repeats the copy using the same displacement.

Exit
In multiple-copy mode only: Exits this command.

8.46 COPYBASE command

Copies entities to the Clipboard.

Shape Lite Pro Mechanical BIM

Icon: 

8.46.1 Description

Copies entities to the Clipboard with a base point for pasting into the same drawing or another drawing.

Note: When you paste entities copied with the COPYBASE command, it uses a user-defined base point.

8.47 COPYCLIP command

Copies entities to the Clipboard.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The PictureExportScale system variable sets the resolution of the image when the exported geometry is pasted in bitmap format, such as in a Word document.

8.47.1 Description

Copies selected entities to the Clipboard for pasting into drawings and other documents.

8.48 COPYEDATA command

Copies extended entity data from one entity to others.

Shape Lite Pro Mechanical BIM

Icon: 

8.48.1 Method

Enter the name of the application to which the entity data belongs, select the entity from which to copy Entity Data and one or more entities to which to copy Entity Data (that will receive the edata).

Extended Entity Data can be created with EDITEDATA command.

8.48.2 Options

? to list application names

Lists the names of applications loaded in the current drawing.

8.49 COPYGUIDED command

Copies entities usingguide curves.

Shape Lite Pro Mechanical BIM

Icon:

8.49.1 Description

Automatically aligns copied entities with relevant geometry, using temporary guide curves.

8.49.2 Method

The command can be executed in two ways:

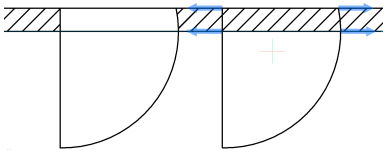
Pre-pick mode
First select entities, then launch the command.

Note: The entities to be copied are displayed in green.

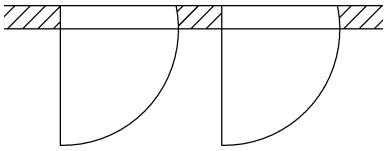
Post-pick mode
Launch the command, then select the entities using a selection window.

Note: All entities that fall completely within the selection window are included in the copy selection set and are displayed in yellow. Entities that cross the selection window are used as guide curves and are displayed in blue.

Blue arrows indicate the anchor points and direction of the guide curves. The copied entities will only align with geometry that matches the number of guide curves and distances between them.



Click to place the copy or enter a distance in the dynamic entry fields.



8.49.3 Options within the command

Polygonal
Creates a polygonal selection window.

Rectangular
Creates a rectangular selection window.

Region
Pastes the clipped area inside the selection window.

Entities
Pastes the entities inside the selection window.

Save detail
Allows to save a detail as a block/Parametric block.

Note: Displays the Write block to file dialog box.

switch to 3D mode
This option links the COPYGUIDED command to the COPYGUIDED3D functionality.

Note: See the COPYGUIDED3D command for more information.

Manually specify source faces
Sets the source faces manually.

Reset
Resets the selection entirely or alters the automatic selection.

Automatic detection of source faces
Sets the recognition of source faces to automatic.

Disable display of source faces
Toggles the display of source faces.

switch back to 2D mode
Switches back to 2D mode, if 3D mode was previously selected.

8.50 COPYGUIDED3D command

Copies 3D Solids or Block references from one location to another, using reference faces of the user's choice.

Shape Lite Pro Mechanical BIM

8.50.1 Method

Selecting a solid or a block activates the automatic detection of connections and source faces. These source faces should not intersect with the detail extents.

Only faces with analytical geometry (planar, cylindrical, spherical, conical and toroidal) are taken into account.

To find insertion points, the algorithm uses faces in a similar way as how COPYGUIDED uses curves.

The detail can transform to fit a new position. The transformation of the detail can consist of translations, rotations and mirrorings.

The detail can be visualized in two colors:

- Blue means the detail extent is in **Free mode** without any possibilities to insert.
- Green means the algorithm has recognized a placement for the detail extents, the transformation has been found and the detailed has already been transformed in the preview. Therefore, the next mouse-click will insert the detail using the found transformation.

When a possible insertion has been detected, press the Ctrl key to cycle through all detected alternatives for the current cursor placement. Next, hold down the Shift key to fix a set of faces, to prevent the command from searching for these faces.

8.50.2 Option within the command

Manually specify source faces
Sets the source faces manually.

Reset
Resets the selection entirely or alters the automatic selection.

Automatic detection of source faces
Sets the recognition of source faces to automatic.

Disable display of source faces
Toggles the display of source faces.

8.51 COPYHIST command

Copies all text from the Command line history to the Clipboard.

Shape Lite Pro Mechanical BIM

Note: The ScrlHist system variable determines how many lines of command history the Prompt History window retains.

8.52 COPYTOLAYER command

Copies entities to another layer.

Shape Lite Pro Mechanical BIM

8.52.1 Description

Creates duplicates of selected entities on a layer specified by the user. You can specify a different location for the duplicated entities.

8.52.2 Method

There are two methods to copy to layer:

- Select entity on target layer.
- Select target layer in the Copy to Layer dialog box.

8.52.3 Options within the command

Name
Displays the Copy To Layer dialog box that allows selecting a target layer.

Displacement
Specify the displacement vector (the distance at which to position the copy). 'Vector' means that you specify the distance and angle at the same time.

Note: When Dynamic Input Mode is active, you can type a distance and angle in the dynamic input fields.

8.53 CORRIDOR command

Creates a corridor from a corridor template.

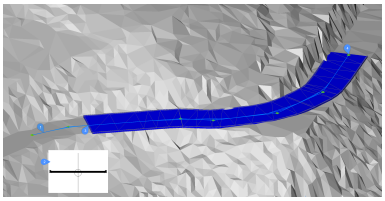
Shape Lite Pro Mechanical BIM

Icon: 

8.53.1 Methods

There are two methods for creating a corridor:

1 **Creates a corridor using a corridor template along an 3D alignment line.**
   - Select 3d alignment used as a corridor baseline (1).
   - Select corridor template (2).
   - Enter region start station (the starting point along the 3D alignment line) (3) .
   - Enter region end station (the ending point along the 3D alignment line) (4).



2 **Creates a simple corridor using a corridor template along a path defined by picking horizontal alignment PI points.**
   - Select TIN Surface
   - Select corridor template
   - Pick horizontal alignment PI point: pick as many horizontal PI points needed and press Enter to finish.

8.53.2 Options

create Simple corridor
Creates a simple corridor using a corridor template along a path defined by picking horizontal alignment PI points.

Predefined
Sets the predefined corridor template and specifies the template position.

Change template
Allows to change the corridor template.

8.54 CORRIDOREDIT command

Adds or removes regions for a selected corridor.

Shape Lite Pro Mechanical BIM

8.54.1 Method

There are two methods to use CORRIDOREDIT command:

- Add regions for a selected corridor.
- Remove regions for a selected corridor.

8.54.2 Options within the command

Add Region
Allows you select a template for a selected corridor.

Enter region start station
The starting point along the 3D alignment line.

Enter region end station
The ending point along the 3D alignment line.

Note: The available intervals are displayed in the command window.

Remove Region
Select region to remove.

enter Index
Select index to be removed.

Note: Index is calculated from region start station to region end station.

8.55 CORRIDOREXTRACT command

Extracts a 3D solid, 3D mesh, TIN surface, 3D polyline or outer boundaries from a corridor.

Shape Lite Pro Mechanical BIM

8.55.1 Description

Extracts a 3D solid, 3D mesh, TIN surface, 3D polyline or outer boundaries from a corridor. Corridor template element (CTE) is defined by points, links and shapes and each of these can contain multiple codes. Corridor region applies template (a collection of CTEs) at incremental stations along it's baseline. Subsequent points, links or shapes are stitched together to form a newly extracted entity. Points are used to extract 3D polylines and outer boundaries for each region separately. Similarly shapes are used to extract 3D solids or 3D meshes in each region. Whereas links are used to create TIN surfaces, but they are always combined from all regions into a single TIN surface, and can only be specified using codes.

8.55.2 Method

There are three methods to use CORRIDOREXTRACT command:

- Extract all 3D solids, 3D meshes, 3D polylines, TIN surface or outer boundaries.
- Extract individual 3D solids, 3D meshes, 3D polylines TIN surface or outer boundaries.
- Extract 3D solids, 3D meshes, 3D polylines, TIN surface or outer boundaries by code.

8.55.3 Options within the command

Mesh
Extract a 3D mesh.

All Shapes
Extract 3D mesh or 3D solid from all shapes.

by Codes
Extract 3D mesh or 3D solid by codes.

Solid
Extract a 3D solid.

Tin
Specify the links to extract

? to list all link codes
Lists all link codes in command line.

liSt selected link codes
Lists the selected link codes.

Polylines
Extract a 3D polyline.

All Stringlines
Extract a 3D polyline from all stringlines.
Stringlines are lines that connect points with the same code value along the corridor baseline.

by Codes
Extract a 3D polyline from all stringlines by code.

outer Boundaries
Extract the outer boundary as a 3D polyline.

Create dynamic link
Specify if the created boundary or stringline should be linked to the original corridor.

8.56 CORRIDORTEMPLATE command

Creates a corridor template.

Shape Lite Pro Mechanical BIM

Icon: 

8.56.1 Method

Pick a point to create a corridor template.

Add corridor template elements to existing corridor template.

8.56.2 Options within the command

Add template element
Select a corridor template element that you wish to append.

8.57 CORRIDORTEMPLATEELEMENT command

Creates a corridor template element from polylines with a specified base point and orientation (left, right, none).

Shape Lite Pro Mechanical BIM

Icon: 

8.57.1 Description

The geometry of the corridor template element is created on the basis of the specified polylines, which can later be edited with the command CORRIDORTEMPLATEELEMENTEDIT.

Input polylines can contain a large number of segments. An individual segment can be drawn as a line or arc. The ARCTESSELLATIONTEMPLATEELEMENT system variable controls the Mid-Ordinate Distance parameter which approximates arc segments.

8.57.2 Method

Select polylines to create a corridor template element and specify the base point.

8.57.3 Options

None
No orientation.

Right
Right orientation.

Left
Left orientation.

8.58 CORRIDORTEMPLATEELEMENTEDIT command

Edits corridor template elements by adding or removing components and targets.

Shape Lite Pro Mechanical BIM

Icons:

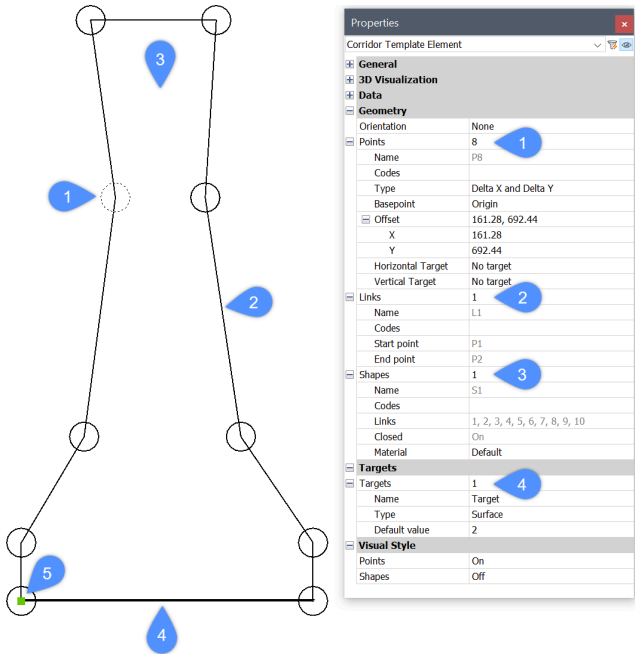


8.58.1 Description

Edits corridor template elements by adding or removing components (points, links, shapes) and targets to an existing template element.

Note:

- After insertion, points (1), links (2), shapes (3) and targets (4) can be also edited in the Properties panel.
- The currently selected point and link are highlighted.
- The green square (5) represents the origin of the corridor template element.



8.58.2 Options within the command

Add Point

Adds a new point to an existing template element by specifying its location.

Basepoint

Allows to specify a basepoint, relative to which the position of the new point is determined. A new point can be inserted directly after the specified basepoint or as the last point of the selected template element.

by Index

Allows to enter an index which corresponds to the point template element component (point, link, shape) by the index. The index represents the sequence number of the template element component.

Point P1 has an index value of 1, point P2 value has an index value of 2, and so on. Link L1 has an index value of 1, link L2 has an index value of 2, and so on. The same applies to shapes.

Add new points directly after the basepoint?

Toggles between Yes or No.

Unset basepoint

Unsets basepoint, which is set to the origin for the new point.

unset insertion Index

Unsets insertion index, which is set to the last value for the new point.

Let's take an example from the figure above, where the template element consists of 10 points, named P1 to P10. Assume that we want to insert a new point directly after the specified basepoint P5. If we select the **unset insertion Index** option in the command line, the new point will get an index value equal to 11, so the name of new point will be P11. If this option is not selected, the new point will get an index value of 6 and the point name will be P6. Existing points from P6 onwards are re-indexed and renamed.

insert bEfore

Adds new point before a specified point.

If a new point is added to the template element before the existing point P5, the new point will get an index value equal to 5, and the name P5. The existing point will be re-indexed and renamed to P6.

Links between existing points and a newly added point are not created automatically, so these points need to be re-linked.

insert After

Adds new point after a specified point. The index of a newly added point is one unit higher than the index of an existing selected point.

If a new point is added to the template element after the existing point P5, the new point will get an index value equal to 6, and the name P6. Existing points from P6 onwards are re-indexed and renamed.

Links between existing points and a newly added point are not created automatically, so these points need to be re-linked.

Remove Point

Removes the template element point by clicking on it or by typing an index. The remaining points are re-indexed accordingly after editing.

Add Link

Creates a link between selected points.

Remove Link

Removes the template element link by clicking on it or by typing an index. The remaining links are re-indexed accordingly after editing.

Add Shape

Creates shapes from template element links.

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 in the Properties panel for a realistic representation of the Corridor model.

Remove Shape

Remove the template element shape by clicking on it or by typing an index. The remaining shapes are re-indexed accordingly after editing.

Add Target

Allows you to create **Surface**, **Horizontal** and **Vertical** targets.

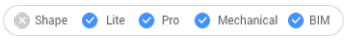
When the geometry of an individual corridor template element must follow a specific surface, horizontal or vertical object, a target is created and added to the corresponding point on the corridor template element.

Remove Target

Removes template element target by clicking on it or by typing an index.

8.59 CPAGESETUP command

Opens the Page setup dialog box.

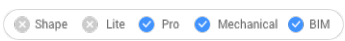


8.59.1 Description

Opens the Page setup dialog box to view and modify the page setup for the current layout or model space.

8.60 CREATELIBRARYBLOCK command

Opens the Add block to library dialog box.



Icon:

8.60.1 Description

Opens the Add block to library dialog box to create a block definition and add it to the library.

8.61 -CREATELIBRARYBLOCK command

Creates and categorizes components.

Shape Lite Pro Mechanical BIM

8.61.1 Description

Adds entities to the Library panel in the form of components that are categorized and displayed as thumbnails. This command is meant for macros.

The component will be added to an existing category or to a new category. If a specific category is open in the Library panel, then the command will display the option to save in that category or a new category.

Categories are listed in alphabetical order:

- 1 - Building
- 2 - Doors
- 3 - Furnishing elements
- 4 - Holes
- 5 - Landscape
- 6 - Mechanical electrical plumbing flow connection points
- 7 - People
- 8 - Sheet metal
- 9 - Standard parts
- 10 - Transportation
- 11 - Windows
- 0 - Add a new category
- <rootCat (current category)> - add component to the current category

A thumbnail is created from the desired view:

- TFL - Top front left isometric view
- TFR - Top front right isometric view
- TBL - Top back left isometric view
- TBR - Top back isometric right view
- T - Top view

8.62 CREATETHUMBNAIL command

Creates a thumbnail image of the current drawing.

Shape Lite Pro Mechanical BIM

8.62.1 Description

Creates custom thumbnail views to show the content of drawing files by file managers and other non-CAD programs. The thumbnails are small raster preview images, and usually are created automatically of the current view as the drawing is saved.

8.62.2 Method

Accept the thumbnail frame by zooming and panning the drawing to the position in which you prefer to save the thumbnail image. Opens the Save Drawing As dialog box to save the thumbnail image.

8.63 Команда CUILOAD

Открывает диалоговое окно Дополнительные файлы меню

Shape Lite Pro Mechanical BIM

8.63.1 Description (Описание)

Открывает диалоговое окно Дополнительные файлы меню для загрузки и выгрузки дополнительных файлов меню.

8.64 CUIUNLOAD command

Opens the Customization groups dialog box.

Shape Lite Pro Mechanical BIM

8.64.1 Description

Opens the Customization groups dialog box to load and unload customization groups.

8.65 CUSTOMIZE command

Opens the Customize dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: CUI

8.65.1 Description

Opens the Customize dialog box to customize the user interface.

8.66 CUTCLIP command

Copies entities to the Clipboard, and then erases them from the drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The PICTUREEXPORTSCALE system variable sets the resolution of the image when the exported geometry is pasted in bitmap format, such as in a Word document.

8.66.1 Description

Copies selected entities to the Clipboard for pasting into drawings and other documents. It automatically erases the selected entities from the drawing.

8.67 CYLINDER command

Creates a 3D solid in the shape of a cylinder.

Note: In BricsCAD Lite, which doesn't support 3D solids, the CYLINDER command launches the AI_CYLINDER command.

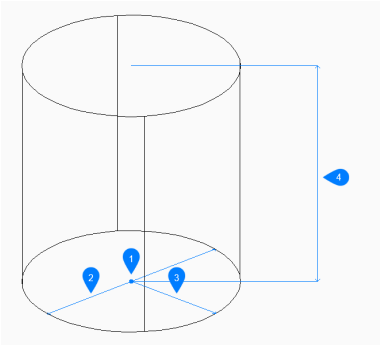
Shape Lite Pro Mechanical BIM



Alias: CYL

8.67.1 Description

Creates a 3D solid in the shape of a circular or elliptical cylinder. Choose from a combination of options including center, radius, diameter, 3 point, 2 point, tangents, axis endpoints, and height.



1 Center

2 Diameter

3 Radius

4 Height

8.67.2 Methods to create a cylinder

This command has 5 methods to begin creating a cylinder:

- Center point
- 3 Point
- 2 Point
- Tangent tangent radius
- Elliptical

Select center point

Begin creating a circular cylinder by specifying the center of the base then:

Radius of base of cylinder
Specify the radius of the cylinder.
Additional option: [Diameter]

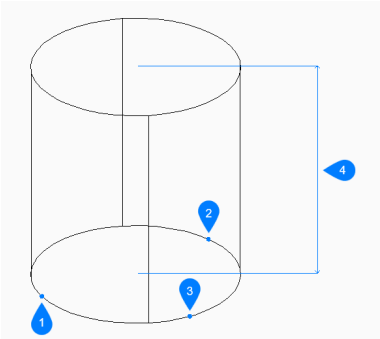
Specify height
Specify the height of the cylinder.
Additional options: [2Point/Axis endpoint]

3 Point
Begin creating a circular cylinder by specifying the first of three points on the circumference of its base then:

Second point
Specify the second point on the circumference.

Third point
Specify the third point on the circumference.

Specify height
Specify the height of the cylinder.
Additional options: [2Point/Axis endpoint]



1 Point 1

2 Point 2

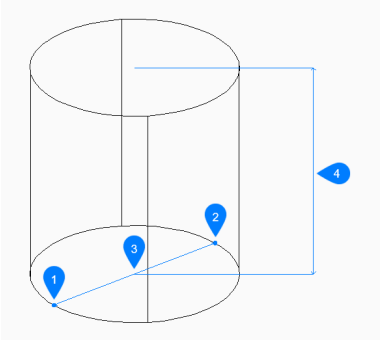
3 Point 3

4 Height

2 Point
Begin creating a circular cylinder by specifying the first of two points on the circumference of its base then:

Second end of diameter
Specify the second point on the diameter. The two points define the diameter of the base.

Specify height
Specify the height of the cylinder.
Additional options: [2Point/Axis endpoint]



1 Point 1

2 Point 2

3 Diameter

4 Height

Tangent Tangent Radius

Begin creating a cylinder by selecting a tangent point on the first entity then:

Specify point on object for second tangent

Select a tangent point on the second entity.

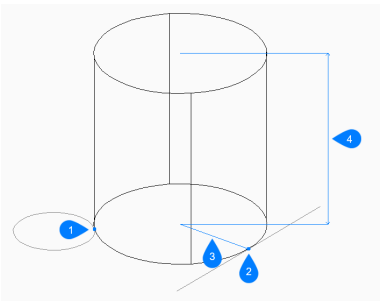
Radius of circle

Specify a valid radius for the base. If you specify a radius that is not possible with the selected tangents, you are prompted to specify the tangents and radius again.

Specify height

Specify the height of the cylinder.

Additional options: [2Point/Axis endpoint]



1 Tangent point 1

2 Tangent point 2

3 Radius

4 Height

Elliptical

Begin creating an elliptical cylinder by specifying the first end of the ellipse axis then:

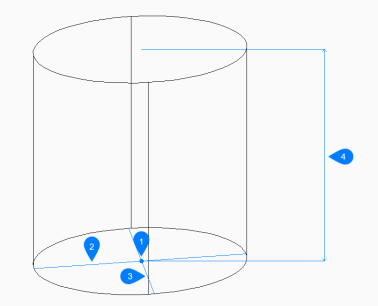
Alternate option: [Center]

Second end of ellipse

Specify the second end of the ellipse axis.

Other end of ellipse
Specify the radius of the other ellipse axis.

Specify height
Specify the height of the cylinder.
Additional options: [2Point/Axis endpoint]



1 Center

2 First axis

3 Second axis

4 Height

8.67.3 Optionswithin the CYLINDER command

After you begin creating a cylinder, the following options may be available:

2Point
Specify the height of the cylinder as the distance between any two points.

Axis endpoint
Specify axis endpoint to define the height and orientation of the cylinder in 3D space. The center of the base is used as the other axis endpoint.

Diameter
Specify the diameter of the cylinder’s base.

9. D

9.1 DATAEXTRACTION command

Opens the Wizard page dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

9.1.1 Description

Opens the Wizard page dialog box to export entity properties, block attributes, and drawing information to a table or external data file.

9.2 -DATAEXTRACTION command

Inserts a data extraction table.

Shape Lite Pro Mechanical BIM

9.2.1 Method

Select the data extraction (.dxd) file from the Standard Open File dialog box and the insertion point for the table.

9.3 DATALINK command

Opens the Datalink manager dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

9.3.1 Description

Opens the Datalink dialog box to create and manage datalinks in the current drawing.

9.4 DATALINKUPDATE command

Synchronizes the linked data in tables in the current drawing with data in the linked source file.

Shape Lite Pro Mechanical BIM

Icon: 

9.4.1 Method

There are two methods to use DATALINKUPDATE command:

- Update the data link existing in a table in the drawing.
- Update the linked data in an external file.

9.4.2 Options within the command

Update data link
Updates the linked data in a table in the drawing with data that has been changed in the external source file.

Write data link
Updates the linked data in an external file with data that has been changed in a table in the drawing.

Select objects
Prompts you to select table entities.

Update all
Synchronizes all linked data in all tables in the drawing.

9.5 DATASMITHCONNECT command

Creates a connection to which Twinmotion and/or Unreal Engine can connect.

Shape Lite Pro Mechanical BIM

Icon: 

9.5.1 Description

Creates a datasmith connection, which can be picked up by Twinmotion or Unreal Engine to establish a Direct Link.

If a connection was already established for another drawing, it will be removed and replaced by the new connection.

Note:

- After establishing a connection to Twinmotion with Direct Link, the model is automatically synchronized the first time.
- Use the DATASMITHSYNC command to synchronize the latest changes of the current drawing to the Direct Link.

9.6 DATASMITHEXPORT command

Exports the current drawing to a udatasmith file.

Shape Lite Pro Mechanical BIM

Icon: 

9.6.1 Description

Opens the **Export drawing to** dialog box to save data from the current drawing to a udatasmith file, which can be used in Twinmotion and/or Unreal Engine.

9.7 DATASMITHSYNC command

Sends changes to Twinmotion and/or Unreal Engine since the last call to the connection.

Shape Lite Pro Mechanical BIM

Icon:

9.7.1 Description

If a datasmith connection for a Direct Link with Twinmotion and/or Unreal Engine exists, it synchronizes it to reflect the latest changes in the current drawing.

Note: When there is no connection yet, use the DATASMITHCONNECT command first.

9.8 DBLIST command

Lists information about all entities in the drawing (short for "database listing").

Shape Lite Pro Mechanical BIM

9.8.1 Description

Displays information about every entity from the drawing.

The information can refer to handle, current space, layer, color, etc.

Note: PressF2to see the data listed in the Prompt History window. For drawings with many entities, the list can take a long time to finish. PressEscto stop the command.

9.9 DCALIGNED command

Constrains the distance between two entities.

Shape Lite Pro Mechanical BIM

Icon:

9.9.1 Description

Constrains the shortest distance between a point and an entity. Constrains the length of a line, a polyline segment, or an arc. Constrains the distance between two lines, making them parallel. Only the distance is constrained (locked); entities can still be moved (in tandem), rotated, and so on. Aligned constraints look like aligned dimensions.

9.9.2 Method

There are three methods to begin creating an aligned constraint:

- Entity
- Point & line
- 2Lines

9.9.3 Options within the command

Entity

Constraints the length of a line, a polyline segment or the chord of an arc.

Note: The entity can still be modified (moved, rotated), even if the length is constrained.

Point & line
Constraints the perpendicular distance between a point and a line or polyline segment.

Note: A valid constraint point is usually the same geometric location as entity snaps, such as the ends and middles of lines, centers of circles and arcs, and so on.

Note: The constrained distance can be edited directly or with the Properties panel.

2Lines
Constrains the distance between two straight linear segments. If the entities are not parallel, then the entity picked second is rotated around its midpoint to be parallel to the first entity.

Note: The constrained distance can be edited directly or with the Properties panel.

9.10 DCANGULAR command

Constrains angles.

Shape Lite Pro Mechanical BIM

Icon:

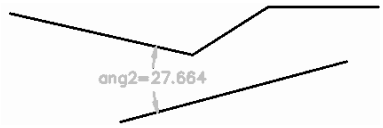
9.10.1 Description

Constrains the angle between two lines or straight polyline segments; also constrains the angles of arcs, polyline arcs, and any three constraint points.

9.10.2 Methods

There are three methods to begin constraining angles:

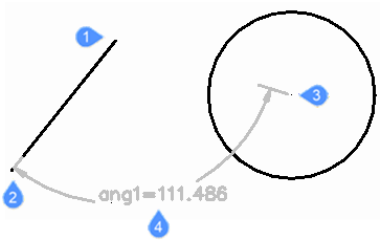
Between two straight segments
Constrains the angle between two segments.



Of arcs or polyline arcs
Constrains the angle of an arc between its endpoints.



Between three valid constraint points
Valid constraint points are at the same geometric locations as entity snaps, such as the ends and middles of lines, centers and quad points of circles and arcs, and so on.



1 Angle vertex

2 First angle constraint point

3 Second angle constraint point

4 Dimension location

9.11 DCCONVERT command

Turns associative dimensions into constraints.

Shape Lite Pro Mechanical BIM

Icon:

9.11.1 Description

Converts dimensions to the appropriate dimensional constraints, such as linear dimensions into linear constraints, or diameter dimensions into diameter constraints. Dimensional constraints are colored gray.



9.12 DCDIAMETER command

Constrains diameters.

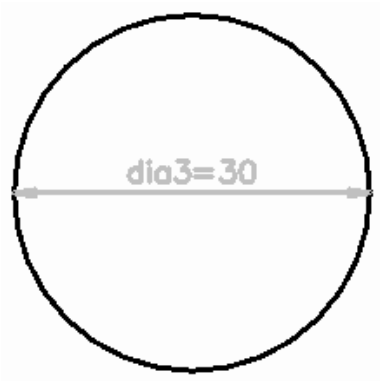
Shape Lite Pro Mechanical BIM

Icon:

9.12.1 Description

Constrains the diameters of circles, arcs, and polyline arc segments.

Although the diameter of the entity is fixed (constrained), the entity can be moved, rotated, and so on.



9.13 DCDISPLAY command

Toggles dimensional constraint visibility.

Shape Lite Pro Mechanical BIM

Icon:

9.13.1 Description

Toggles the display of dimensional constraints (attached to selected entities) between visible and hidden. All dimensional constraints are initially hidden when a drawing containing them is opened. Many constraints in a drawing can clutter it, and so it is useful to hide them.

9.14 DCHORIZONTAL command

Constrains entities horizontally.

Shape Lite Pro Mechanical BIM

Icon:

9.14.1 Description

Horizontally constrains the distance between two points, or the length of a single entity.

Note: "Horizontal" means the constraint lies in the X direction of the current coordinate system.

9.14.2 Method

There are two methods to begin constraining horizontally:

- By distance: constrain the distance between two entities horizontally.
- ***Note:*** Valid constraint points are located at same geometric locations as entity snaps, such as the ends and middles of lines, centers and quad points of circles and arcs, and so on.
- By entity: constrain an entity horizontally.

  Note: Constrains the following entities horizontally: Line, Arc, Polyline segment, Polyline arc segment.

9.14.3 Options within the command

Entity
Constrains the horizontal distance between the endpoints of an entity.

9.15 DCLINEAR command

Constrains linearly.

Shape Lite Pro Mechanical BIM

Icon: 

9.15.1 Description

Constrains the distance between two points, or the length of a single entity, to the horizontal or vertical, depending how the cursor is moved, as in the DIMLINEAR command.

Note: "Horizontal" means the constraint lies in the X direction of the current coordinate system, and "vertical" in the Y direction.

9.15.2 Method

There are two methods to begin constraining linearly:

- By distance: constrain the distance between two entities vertically or horizontally.

  Note: Valid constraint points are found at the same geometric locations as entity snaps, such as the ends and middles of lines, centers and quad points of circles and arcs, and so on.

- By entity: constrain the length of an entity between its endpoints, vertically or horizontally.

  Note: Constrains one of the following entity types horizontally or vertically: Line, Arc, Polyline segment, Polyline arc segment.

The program applies a vertical or horizontal constraint, depending on how the cursor is moved during the command.

9.15.3 Options within the command

Entity
Constrains the horizontal or vertical distance between the endpoints of an entity.

9.16 DCRADIUS command

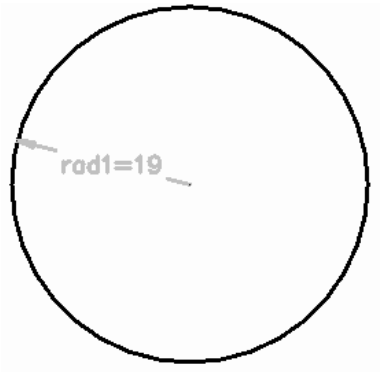
Constrains radii.

Shape Lite Pro Mechanical BIM

Icon: 

9.16.1 Description

Constrains the radius of circles, arcs, or polyline arc segments. Although the radius of the entity is fixed (constrained), the entity can be moved, rotated, and so on.



9.17 DCVERTICAL command

Constrains entities vertically.

Shape Lite Pro Mechanical BIM

Icon:

9.17.1 Description

Vertically constrains the distance between two points, or the length of an entity.

Note: "Vertical" means the constraint lies in the Y direction of the current coordinate system.

9.17.2 Method

There are two methods to begin constraining vertically:

- By distance: constrain the distance between two entities vertically.

  Note: Valid constraint points are found at the same geometric locations as entity snaps, such as the ends and middles of lines, centers and quad points of circles and arcs, and so on.

- By entity: constrain an entity vertically.

  Note: Constrains any of the following entities vertically: Line, Arc, Polyline segment, Polyline arc segment.

To constrain the distance between two entities vertically, select a valid constraint point on each entity and enter the constraint distance. The value controls the distance between the two entities.

9.17.3 Options within the command

Entity

Constrains the vertical distance between the endpoints of an entity.

9.18 DDATTE command

Edits the values of attributes through a dialog box. This command is superseded by the BATTMAN command.

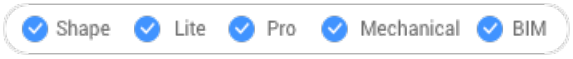
Shape Lite Pro Mechanical BIM

9.18.1 Description

Opens the Edit Attributes dialog box where may change the value of the attribute text.

9.19 DDEDIT command

Edits single-line text, multi-line text (mtext), dimension text, attribute definition and leader text.



Alias: ED

9.19.1 Description

Allows to edit single-line texts, multi-line text(mtext), dimension text, attribute definition and leader text.

If the selected entity is represented by multi-line text, dimension text or leader text, the Text formatting window opens.

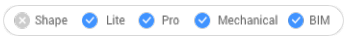
If the selected entity is represented by an attribute definition, the Edit Attribute Definition dialog box opens.

If the selected entity is represented by single-line text, the in-place editor is displayed (this editor has no User Interface).

Note: Change the value of TEXTED system variable to change the editing method of a single-line text.

9.20 DDEMODES command

Sets default values for creating entities.



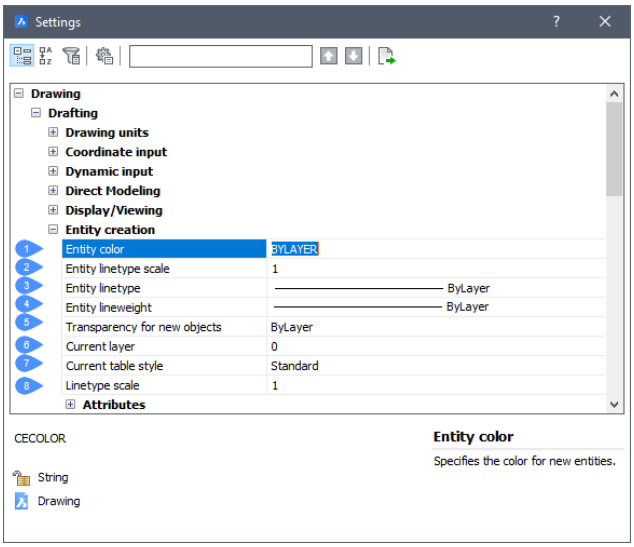
9.20.1 Description

Displays the Entity Creation section of the Settings dialog box.

Note: This command is superseded by the Settings command.

9.20.2 Method

Displays Settings dialog box at the Entity Creation section:



1 Entity color

2 Entity linetype scale

3 Entity linetype

4 Entity lineweight

5 Transparency for new objects

6 Current layer

7 Current table style

8 Linetype scale

9.20.3 Options

Entity Color

Specifies the default color of new entities; initial value is ByLayer, which means that entity colors are governed by layer properties.

Entity Linetype Scale

Specifies the default scale factor for linetypes; initial value is 1.0000.

Entity Linetype

Specifies the default linetype for new entities; initial value is ByLayer, which means that entity linetypes are governed by layer properties.

Entity Lineweight

Specifies the default linetype for new entities; initial value is ByLayer, which means that entity lineweights are governed by layer properties.

Current Layer

Specifies the default layer for new entities; initial value is layer 0.

Current Table Style

Specifies the initial value of the table style name for new tables.

Linetype Scale
Specifies the global linetype scale factor; this factor affects entity linetype scale factors.

9.21 DDFILTER command

Creates a selection set of the entities.

Shape Lite Pro Mechanical BIM

Note: This command is superseded by the SELECT command.

9.21.1 Description

Creates a selection set that include the selected entities, so you may access it by entering "P" (previous) the next time a command prompts you "Select entities."

9.22 DDGRIPS command

Opens the Settings dialog box with the **Grips** category expanded.

Shape Lite Pro Mechanical BIM

Alias: GR

9.22.1 Description

Opens the Settings dialog box with the **Grips** category expanded to view and modify relevant system variables.

9.23 DDPTYPE command

Opens the Settings dialog box with the **Points** category expanded.

Shape Lite Pro Mechanical BIM

9.23.1 Description

Opens the Settings dialog with the **Points** category expanded to view and modify relevant system variables.

9.24 DDSELECT command

Opens the Settings dialog box with the **Entity selection** category expanded.

Shape Lite Pro Mechanical BIM

Alias: SE

9.24.1 Description

Opens the Settings dialog box with the **Entity selection** category expanded to view and modify relevant system variables.

9.25 DDSETVAR command

Opens the Settings dialog box.

Shape Lite Pro Mechanical BIM

9.25.1 Description

Opens the Settings dialog box to view and modify system variables. Most, but not all, system variables are available in the Settings dialog box. You can modify all system variables using the SETVAR command.

9.26 DDSTRACK command

Opens the Settings dialog box with the **Snap tracking** category expanded.

Shape Lite Pro Mechanical BIM

9.26.1 Description

Opens the Settings dialog box with the **Snap tracking** category expanded to view and modify relevant system variables.

9.27 DDVPOINT command

Opens the Set viewpoint dialog box.

Shape Lite Pro Mechanical BIM

Alias: SETVPOINT, VIEWCTL, VP

9.27.1 Description

Opens the Set viewpoint dialog box to specify a 3D viewpoint for the current viewport.

9.28 DEFAULTSCALELIST command

Opens the Default scale list dialog box.

Shape Lite Pro Mechanical BIM

9.28.1 Description

Opens the Default scale list dialog box to view and manage the default scale list. The default scale list is stored in the registry. When you reset the current drawing's scale list, it updates to match the default scale list.

9.29 DELAY command

Delays execution of commands.

Shape Lite Pro Mechanical BIM

9.29.1 Description

This command is used to delay the execution of the next command.

Note: Meant to use with scripts.

9.29.2 Options within the command

Milliseconds to delay

Specifies the amount of time BricsCAD waits before continuing to the next command in the script.

Note: Enter a number between 0 and 2,147,483,627(approximately 24 days).

9.30 DELCONSTRAINT command

Removes dimensional and geometrical constraints from selected entities.

Shape Lite Pro Mechanical BIM

9.30.1 Method

Select one or more entities or type ALL to select all entities in the drawing to delete all their dimensional and geometric constraints.

Note: To remove individual constraints one at a time:

- Dimensional constraints: select the dimensional constraint, and then press ERASE command.
- Geometric constraints: click the tiny x on the constraint bar: 

9.31 DELEDATA command

Deletes extended entity data for specific applications from the selected entities (short for "delete entity data").

Shape Lite Pro Mechanical BIM

9.31.1 Description

Specifies the name of the application to which the entity data belongs and selects one or more entities from which Entity Data will be deleted.

9.31.2 Options

? to list application names

Lists names of applications loaded in the current drawing.

9.32 DESIGNTABLE command

Imports .csv files that contain sets of parameter values or else creates them from scratch.

Shape Lite Pro Mechanical BIM

Icons: 

9.32.1 Description

Imports .csv files that contain sets of parameter values or creates them from scratch. Parameters are then added to the Mechanical Browser panel and to parametric component properties in Properties panel.

Note: This command can be entered transparently during the commands (‘designtable).

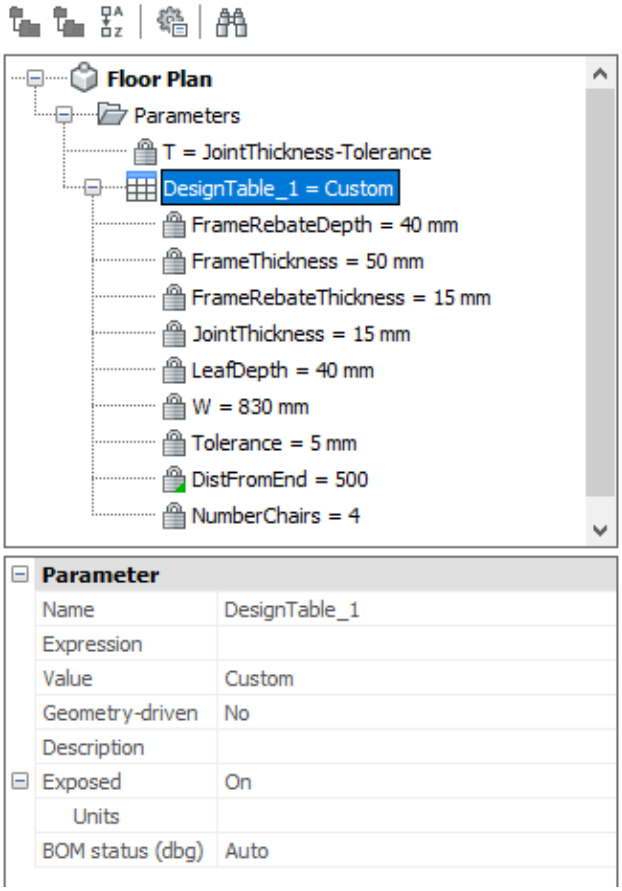
9.32.2 Options

From file

Creates a design table by importing a .csv file (comma-separated values file). Make sure the list separator character on your system is the same as the one used in the CSV file.

Displays the Select Design Table File dialog box from where you will select the .csv file and open it.

Note: Enter the MECHANICALBROWSEROPEN command to view the constraints:



Empty
Creates an empty design table from scratch.

Note: Enter the names of the parameters to include in the design table or choose to add all parameters that have a constant value. The constant parameters do not depend on another parameter.

9.33 -DESIGNTABLEEDIT command

Edits design tables.

Shape Lite Pro Mechanical BIM

Note: This command can be entered transparently during the commands (‘designtable).

9.33.1 Options

eXport
Exports the content of a design table entity in .csv format file.

Replace
Replaces the content of a design table with data from a .csv format file.

Delete
Deletes a design table.

Configurations
Creates, removes, or views configurations.

Save current
Creates a new configuration as a copy of an existing one.

Remove
Removes a configuration.

? to list
List the names of design tables attached to the current drawing.

Parameters
Manages parameters:

Unlink
Unlinks parameters from the design table.

Link
Links parameters to the design table.

Clear expressions
Clears expressions (values and formulae) from parameters.

? to list columns
Lists parameters.

Apply configuration
Creates a new row using the current values of variables.

? to list
Lists design tables attached to the current drawing.

9.34 DETAILSPANELCLOSE command

Closes the Details panel.

Shape Lite Pro Mechanical BIM

9.34.1 Description

Closes the Details panel to hide it from the current workspace. If the Details panel is stacked when you close it, the Details tab or icon is removed from the stack.

9.35 DETAILSPANELOPEN command

Opens the Details panel.

Shape Lite Pro Mechanical BIM

9.35.1 Description

Opens the Details panel to display it in the current workspace. The Details panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Details panel can be either floating, docked or stacked.

9.36 DGNEXPORT command

Exports the current drawing to a DGN file format.

Shape Lite Pro Mechanical BIM

9.36.1 Description

Opens the Export drawing as dialog box to export the current drawing to a MicroStation DGN File (*.dgn) format.

9.37 -DGNEXPORT command

Exports the current drawing to MicroStation DGN File (*.dgn)format in the command line.

Shape Lite Pro Mechanical BIM

9.37.1 Method

Specify the full path and file name for the exported DGN file.

Note: Type ~ (tilde) to display the Export Drawing As dialog box, which allows you to specify the folder and the name for the DGN file.

9.37.2 Options within the command

DGN
Converts reference to DGN.

DWG
Keeps reference to DGN.

Bind
Binds reference file to the parent DGN.

Detach
Detaches external reference.

9.38 DGNIMPORT command

Opens the Import file dialog box.

Shape Lite Pro Mechanical BIM

9.38.1 Description

Opens the Import file dialog box to select a dgn file to import into the current drawing.

9.39 DGNIMPORTOPTIONS command

Opens the Settings dialog box with the **DGNIMPORT** category expanded.

Shape Lite Pro Mechanical BIM

9.39.1 Description

Opens the Settings dialog box with the **DGNIMPORT** category expanded to view and modify relevant system variables.

9.40 DIGITALSIGN command

Applies a digital signature (block of encrypted information) to the drawing.

Shape Lite Pro Mechanical BIM

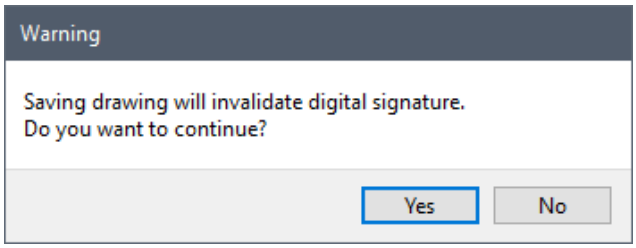
9.40.1 Method

Opens the Digital Signatures dialog box where you can attach a digital signature to the drawing in order to validate the origin, authenticity and unaltered state of the file since the digital signature was applied.

Note: The digital signature remains valid after renaming the drawing.

Note: The SIGWARN system variable controls the display of a dialog with the signature content when a drawing with a digital signature is opened.

Note: After the DIGITALSIGN command is launched, a digital signature is applied every time when saving the drawing until it is closed. The next time the signed drawing is opened and modified, a warning message appears when saving.



9.41 DIM command

Creates multiple dimension types in a single workflow.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMENSION

9.41.1 Description

Allows to create multiple dimensions or types of dimensions at once.

Note: Isometric view can be dimensioned and reflect the actual true geometry size.

9.41.2 Options within the command

HORizontal
Places horizontal linear dimensions, same as the DimLinear command.

VErtical
Places vertical dimensions, same as the DimLinear command.

ALigned
Places linear dimensions that are aligned to entities, same as the DimAligned command.

ANgular
Places angular dimensions that measure angles, same as the DimAngular command.

Leader
Places leaders, same as the DimLeader command.

OBlique
Changes the angle of extension lines, same as the DimEdit command.

ROtated
Places linear dimensions at an angle, same as the DimLinear command.

CEnter
Places marks at the centers of circles and arcs, same as the DimCenter command.

Diameter
Places diameter dimensions on circles and arcs, same as the DimDiameter command.

RAdius
Places radial dimensions on circles and arcs, same as the DimRadius command.

Baseline
Places multiple linear and angular dimensions from the same base point, same as the DimBaseline command.

COntinue
Continues linear and angular dimensions from the last endpoint, same as the DimContinue command.

ORdinate
Places x and y ordinate measurements from an origin point, same as the DimOrdinate command.

Position
Repositions the dimension text, same as the DimTEdit command.

DIStribute
Spaces selected dimensions equally. There are two options between which to chose how the dimensions will be distributed.

Equal
Distributes equally all the dimensions selected.

Offset
All selected dimensions are distributed at a specified offset distance.

UPdate dimensions
Applies the current dimension style to a selection of dimension entities; see Apply option of the -DimStyle command.

variable STatus
Lists the status of all dimension variables in the Prompt History window.

OVerride
Overrides the values of the current dimension style, same as DimOverride command.

SEttings...
Opens the Drawing explorer | Dimension Styles dialog box which allows you and modifying dimension styles, same as the DIMSTYLE command.

LAyer
Sets a different layer for the default layer on which dimensions are draw.

Note: Dimensions are created on the layer which is specified by the DIMLAYER system variable.

9.42 DIM1 Command

Executes a single dimension command at the 'Dimensioning command:' prompt.

Shape Lite Pro Mechanical BIM

9.42.1 Description

Allows to create a single type of dimension, as specified in the Command line.

9.42.2 Options within the command

HORizontal
Places horizontal linear dimensions, same as the DimLinear command.

VErtical
Places vertical dimensions, same as the DimLinear command.

ALigned
Places linear dimensions that are aligned to entities, same as the DimAligned command.

ANgular
Places angular dimensions that measure angles, same as the DimAngular command.

Leader
Places leaders, same as the DimLeader command.

OBlique
Changes the angle of extension lines, same as the DimEdit command.

ROtated
Places linear dimensions at an angle, same as the DimLinear command.

CEnter
Places marks at the centers of circles and arcs, same as the DimCenter command.

Diameter
Places diameter dimensions on circles and arcs, same as the DimDiameter command.

RAdius
Places radial dimensions on circles and arcs, same as the DimRadius command.

Baseline
Places multiple linear and angular dimensions from the same base point, same as the DimBaseline command.

COntinue
Continues linear and angular dimensions from the last endpoint, same as the DimContinue command.

ORdinate
Places x and y ordinate measurements from an origin point, same as the DimOrdinate command.

Position
Repositions the dimension text, same as the DimTEdit command.

DIStribute
Spaces selected dimensions equally. There are two options between which to chose how the dimensions will be distributed.

Equal
Distributes equally all the dimensions selected.

Offset
All selected dimensions are distributed at a specified offset distance.

UPdate dimensions
Applies the current dimension style to a selection of dimension entities; see Apply option of the -DimStyle command.

variable STatus
Lists the status of all dimension variables in the Prompt History window.

OVerride
Overrides the values of the current dimension style, same as DimOverride command.

SEttings...
Opens the Drawing explorer | Dimension styles dialog box which allows you and modifying dimension styles, same as the DIMSTYLE command.

LAyer
Sets a different layer for the default layer on which dimensions are draw.

Note: Dimensions are created on the layer which is specified by the DIMLAYER system variable.

9.43 DIMALIGNED command

Creates an aligned dimension.

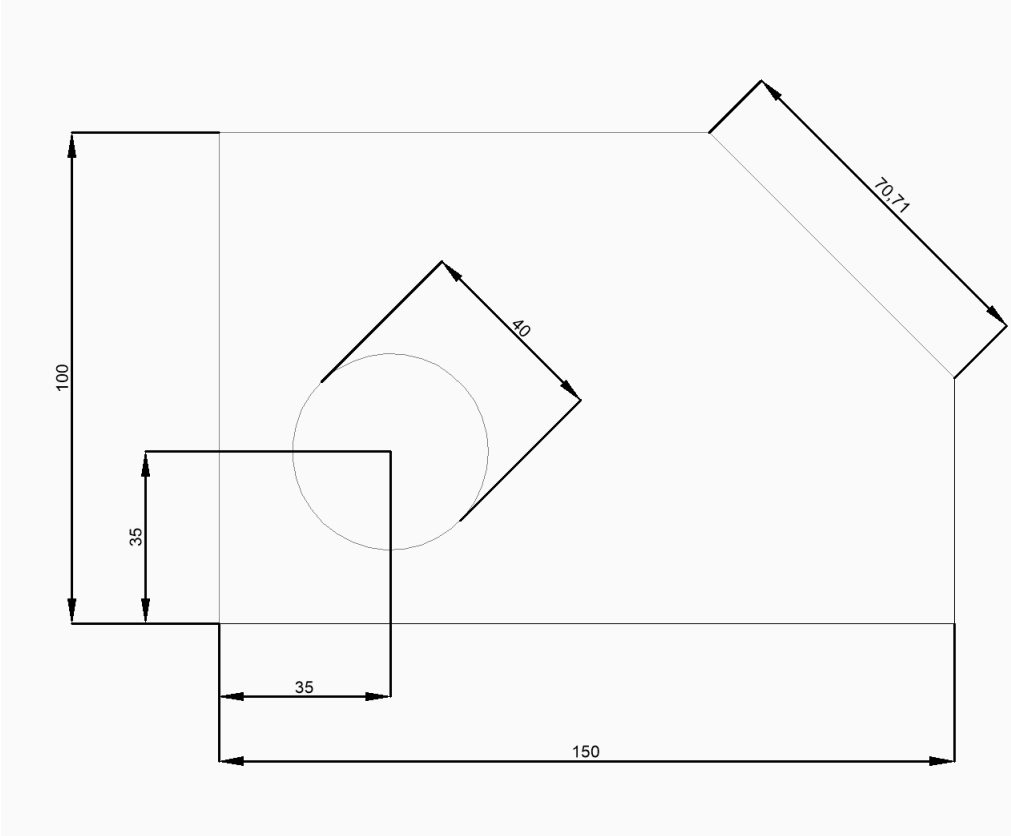
Shape Lite Pro Mechanical BIM

Icon: 

Alias: DAL, DIMALI

9.43.1 Description

Createsa dimension that aligns with the origin points of the extension lines. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



Note: Isometric view can be dimensioned and reflect the actual true geometry size.

9.43.2 Methods to createan aligned dimension

This command has 2 methods to begin creating an aligned dimension:

- Origin of first extension line
- Select entity

Origin of first extension line

Begin creating an aligned dimension by specifying a point for the first extension line then:

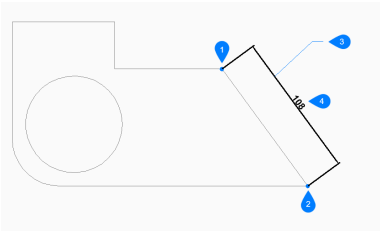
Origin of second extension line

Specify a point for the second extension line.

Location of dimension line

Specify the location of the dimension line. The dimension is placed at the same distance from each of the extension line origins.

Additional options: [Angle/Text]



1 Origin of first extension line

2 Origin of second extension line

3 Location of dimension line

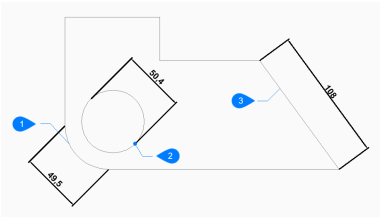
4 Aligned dimension

Select entity

Begin creating an aligned dimension by pressing Enter then:

Select entity to dimension

Select a line, polyline segment, arc, or circle to dimension. When selecting a line, the endpoints of the line are used for the extension origins. When selecting an arc, the endpoints of the arc are used for the extension origins. When selecting a circle, the selection point on the circle is used for the first extension origin and the opposite point on the diameter of the circle is used for the second extension origin.



1 Arc entity

2 Selection point of circle entity

3 Polyline entity

Additional option: [selection options (?)]

Location of dimension line

Specify the location of the dimension line. The dimension is placed at the same distance from each of the extension line origins.

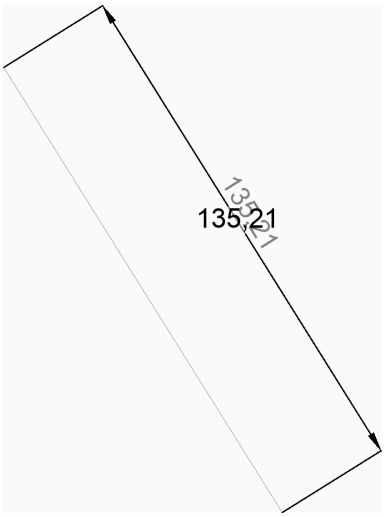
Additional options: [Angle/Text]

9.43.3 Options within theDIMALIGNEDcommand

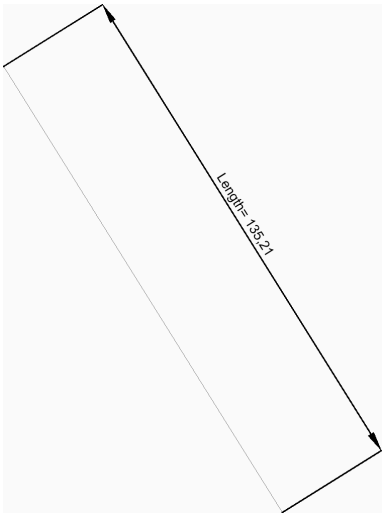
After you begin creating analigned dimension, the following options may be available:

Angle

Specify the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text
Enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



9.44 DIMANGULAR command

Creates an angular dimension.

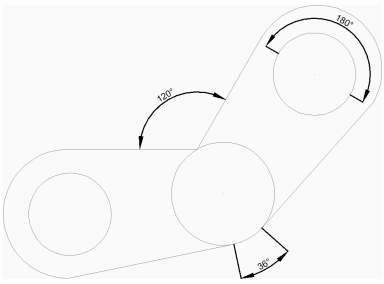
Shape Lite Pro Mechanical BIM

Icon:

Alias: DAN, DIMANG

9.44.1 Description

Creates an angular dimension by selecting an entity or specifying the vertex and both sides of the angle. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



9.44.2 Methods to create an angular dimension

This command has 2 methods to begin creating an angular dimension:

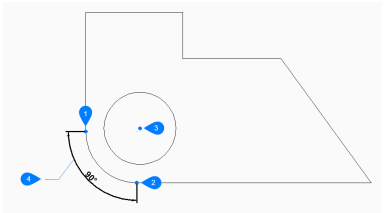
- Select line, arc, or circle
- Press ENTER to specify angle

Select line, arc or circle
Begin creating an angular dimension by selecting a line, arc, or circle to dimension then:
Additional options: [selection options (?)]
If you selected a line or line segment

Other line for angular dimension
Specify another line segment.

Location of dimension arc
Specify the location of the dimension arc.
Additional options: [Angle/Text]
If you selected an arc or arc segment

Location of dimension arc
Specify the location of the dimension arc.



1 First side of angle

2 Other side of angle

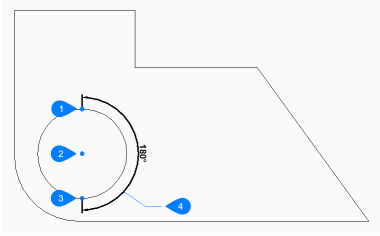
3 Vertex of angle

4 Location of dimension arc

Additional options: [Angle/Text]
If you selected a circle

Other side of angle
Specify a point to define one side of the angle. The point where you first selected the circle defines the other side of the angle.

Location of dimension arc
Specify the location of the dimension arc.



1 Select circle

2 Other side of circle

3 Vertex of angle

4 Location of dimension arc

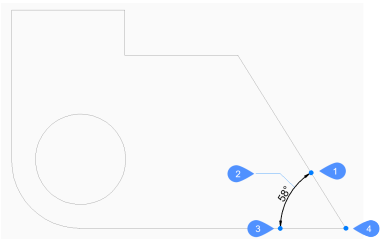
Additional options: [Angle/Text]

Press ENTER to specify angle
Begin creating an angular dimension by specifying the vertex of the angle then:

First side of angle
Specify a point to define one side of the angle.

Other side of angle
Specify a point to define the other side of the angle.

Location of dimension arc
Specify the location of the dimension arc.



1 First side of angle

2 Location of dimension arc

3 Other side of angle

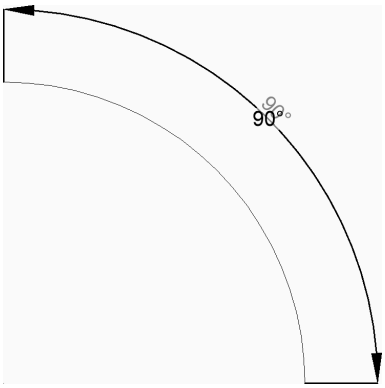
4 Vertex of angle

Additional options: [Angle/Text]

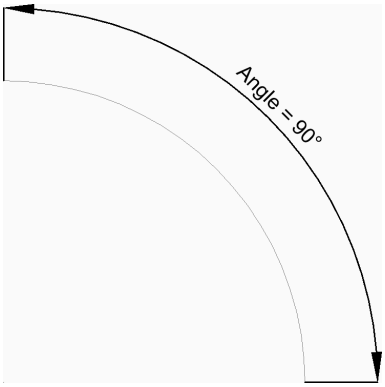
9.44.3 Options within the DIMANGULAR command

After you begin creating an angular dimension, the following options may be available:

Angle
Specify the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text
Enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



9.45 DIMARC command

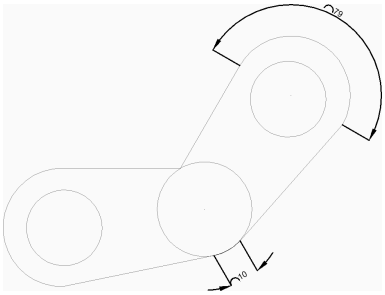
Creates an arc length dimension.

Shape Lite Pro Mechanical BIM

Icon:

9.45.1 Description

Createsa dimension that measure the length of an arc or polyarc. The dimension is based on the current dimension style. Options allow you to include a leader and specify the angle and contents of the dimension text.



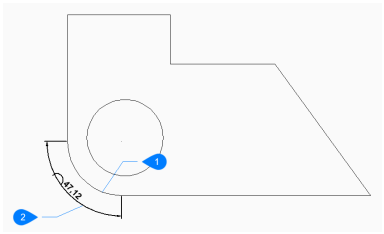
9.45.2 Methods to createan arc length dimension

There is one method to begin creating an arc length dimension:

- Select arc or polyline arc segment

Select arc or polyline arc segment
Begin creating an arc length dimension by selecting an arc or polyline arc segment then:

Location of dimension arc
Specify a point to locate the position of the dimension arc.
Additional options: [Angle/Text/Partial/Leader]



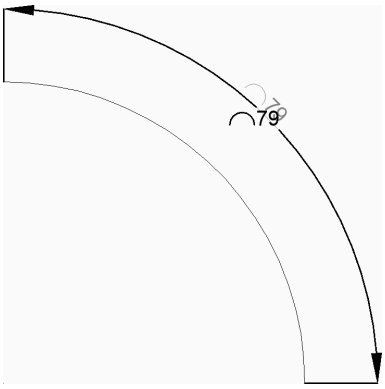
1 Arc segment

2 Location of dimension arc

9.45.3 Options within theDIMARCcommand

After you begin creating anarc length dimension, the following options may be available:

Angle
Specify the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



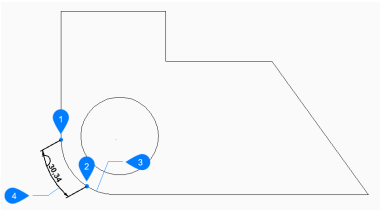
Text
Enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



Partial
Specify a point for the first extension line then:

Specify second point for arc length dimension
Specify a point for the second extension line.

Location of dimension arc
Specify the location of the dimension line.
Additional options: [Angle/Text/Partial]



1 First point of arc length dimension

2 Second point of arc length dimension

3 Arc segment

4 Location of dimension arc

Leader
Toggle on a leader from the dimension text to the arc.

No leader
Toggle off the leader from the dimension text to the arc.

9.46 DIMBASELINE command

Creates stacked dimensions from the same baseline.

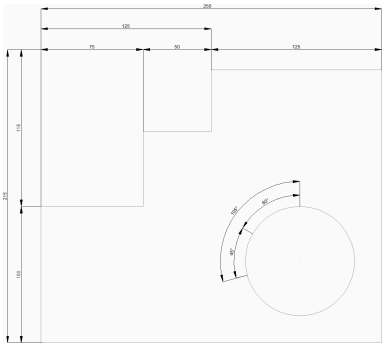
Shape Lite Pro Mechanical BIM

Icon:

Alias: DBA, DIMBASE

9.46.1 Description

Creates stacked linear, angular or ordinate dimensions from the same baseline as an existing dimension. The dimensions are based on the current dimension style and the dimension spacing is specified by the DIMDLI variable.



9.46.2 Methods to create baseline dimensions

This command has 2 methods to begin creating baseline dimensions:

- Origin of next extension line
- Select starting dimension

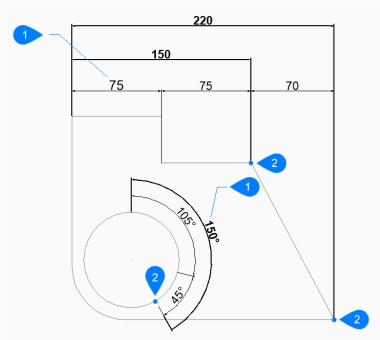
Origin of next extension line

Begin creating baseline dimensions from the last linear, angular, or ordinate dimension by specifying a point for the next extension line.

For linear and angular dimensions, the first extension line of each baseline dimension coincides with the first extension line of the previous dimension.

Continue placing dimensions until you press Enter to end the command.

Additional options: [Select starting dimension/Undo]



1 Existing dimension

2 Origin of next extension line

Select starting dimension

Begin creating a baseline dimension by selecting an existing linear, angular, or ordinate dimension then:

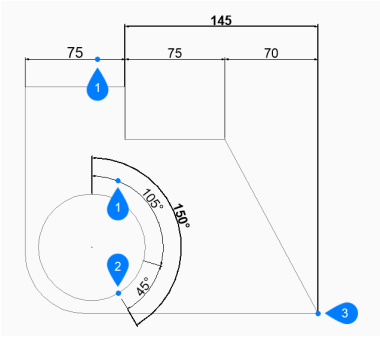
Origin of next extension line

Specify a point for the next extension line.

For linear and angular dimensions, the first extension line is coincides with the extension line nearest to where you selected the existing dimension.

Continue placing dimensions until you press Enter to end the command.

Additional options: [Select starting dimension/Undo]



1 Starting dimension

2 Origin of next extension line

9.46.3 Options within the DIMBASELINE command

After you begin creating a baseline dimension, the following option may be available:

Undo
Undo the last dimension and continue drawing from the previous dimension.

9.47 DIMBREAK command

Breaks dimension lines, extension lines, and leaders at the locations where they cross other entities.

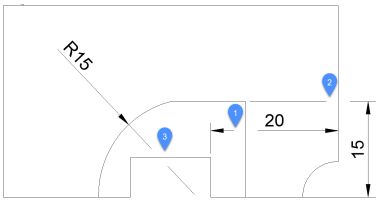
Shape Lite Pro Mechanical BIM

Icon:

9.47.1 Description

Allows breaking entities such as dimension lines, extension lines or leaders by a certain object and it can also remove breaks.

The dimension entity can be broken on different entities: the dimension line itself (1), on the extension line (2) or on the leader (3).



9.47.2 Options

Multiple
Allows multiple dimension entities to be broken in the desired spots.

Auto
Breaks all the selected dimension entities at all intersections found with other entities.

Remove
Removes all dimension breaks from the selected dimension entities.

Manual
The width of the dimension break can be manually defined. This option is not available in Auto mode.

9.48 DIMCENTER command

Creates a center mark.

Shape Lite Pro Mechanical BIM

Icon:

9.48.1 Description

Create a center mark at the center of a circle, arc or polyarc. See the CENTERMARK command to create associative center marks.

9.48.2 Methods to create a center dimension

There is one method to begin creating a center mark.

- Select arc or circle to dimension

Select arc or circle to dimension
Begin creating a center dimension by selecting an arc, polyline arc segment, or circle. The DIMCEN system variable controls the length and appearance of the center mark lines.
Additional options: [selection options (?)]

9.48.3 Options within the DIMCENTER command

After you begin creating a center dimension, the following options may be available:

9.49 DIMCONSTRAINT command

Applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

Shape Lite Pro Mechanical BIM

9.49.1 Method

Select an associative dimension or else choose an option to place a dimensional constraint.

The associative dimension is converted to the dimensional constraint of the same type. This option is equivalent to the DCCONVERT command.

9.49.2 Options

LInear
Constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. This option is equivalent to the DCLINEAR command.

Horizontal
Constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. This option is equivalent to the DCHORIZONTAL command.

Vertical
Constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. This option is equivalent to the DCVERTICAL command.

Aligned
Constrains the distance between two points. This option is equivalent to the DCALIGNED command.

ANgular
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. This option is equivalent to the DCANGULAR command.

Radial
Constrains the radius of a circle or an arc. This option is equivalent to the DCRADIUS command.

Diameter
Constrains the diameter of a circle or an arc. This option is equivalent to the DCDIAMETER command.

9.50 DIMCONTINUE command

Creates dimensions in a continuous line or arc.

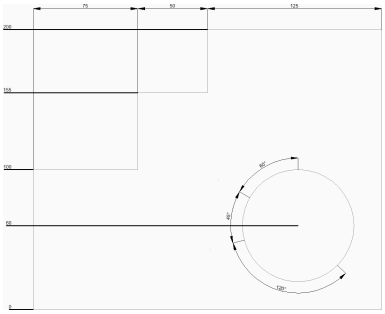
Shape Lite Pro Mechanical BIM

Icon:

Alias: DCO, DIMCONT

9.50.1 Description

Creates a continuous line or arc from an existing linear, angular, or ordinate dimension. The dimensions are based on the current dimension style.

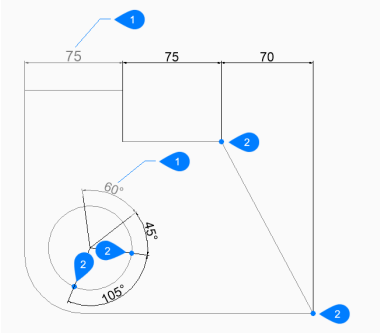


9.50.2 Methods to createcontinuous dimensions

This command has 2 methods to begin creating continuous dimensions:

- Origin of next extension line
- Select starting dimension

Origin of next extension line
Begin creating continuous dimensions from the last linear, angular, or ordinate dimension by specifying a point for the next extension line.
For linear and angular dimensions, the first extension line of each continuous dimension coincides with the second extension line of the previous dimension.
Continue placing dimensions until you press Enter to end the command.
Additional options: [Select starting dimension/Undo]

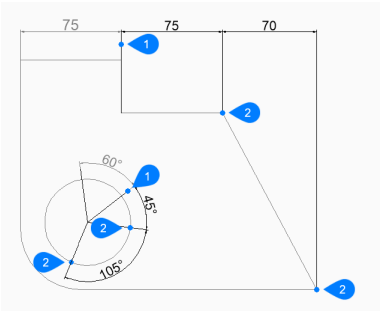


1 Last dimension

2 Origin of next extension line

Select starting dimension
Begin creating a continuous dimension by selecting an existing linear, angular, or ordinate dimension then:

Origin of next extension line
Specify a point for the next extension line.
For linear and angular dimensions, the first extension line coincides with the extension line nearest to where you selected the existing dimension.
Continue placing dimensions until you press Enter to end the command.
Additional options: [Select starting dimension/Undo]



1 Starting dimension

2 Origin of next extension line

9.50.3 Options within theDIMCONTINUEcommand

After you begin creating a continuous dimension, the following options may be available:

Undo
Undo the last dimension and continue drawing from the previous dimension.

9.51 DIMDIAMETER command

Creates a diametric dimension.

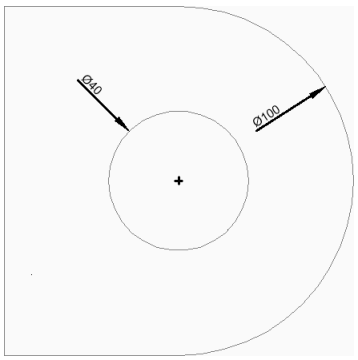
Shape Lite Pro Mechanical BIM

Icon: ⌀

Alias: DDI, DIMDIA

9.51.1 Description

Creates a diametric dimension for an arc, polyarc, or circle. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



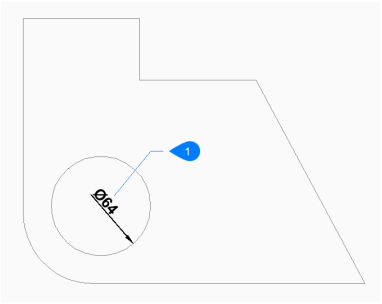
9.51.2 Methods to create a diametric dimension

There is one method to begin creating a diametric dimension:

- Select arc or circle to dimension

Select arc or circle to dimension
Begin creating a diametric dimension by selecting an arc, polyline arc, or circle then:

Location of dimension line
Specify the location of the dimension line.
Additional options: [Angle/Text]

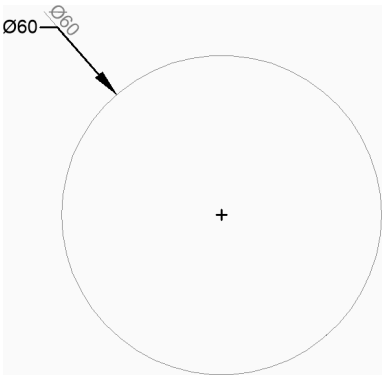


1 Location of dimension line

9.51.3 Options within the DIMDIAMETER command

After you begin creating a diametric dimension, the following options may be available:

Angle
Specify the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text

Enter text to override the default dimension text which includes the measured length of the diameter. You can use two angle brackets <> to display the measured length of the diameter in addition to other text.



9.52 DIMDISASSOCIATE command

Removes the associativity of selected dimension entities.

Shape Lite Pro Mechanical BIM

Icon:

9.52.1 Description

Disassociates the selected dimension entities and prompts you in the Command line the number of disassociated dimensions.

9.53 DIMEDIT command

Edits particular dimension elements.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DED, DIMED

9.53.1 Description

Edits the position, angle, and wording of dimension text, and changes the angle of extension lines.

9.53.2 Methods

There are four methods to edit dimensions:

- Edit text
- Oblique lines

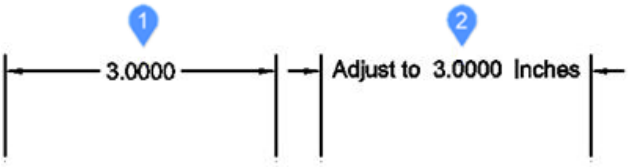
- Rotate text
- Restore text

9.53.3 Options within the DIMEDIT command

Edit text

This option will allow you to edit the existing dimension text.

Use any selection mode to choose the dimension entities to be edited. Use the <> symbols to indicate the default text (1), so that you can insert text in front or behind the default, such as “Adjust to <> inches” (2).



Oblique lines

This option will rotate (obliques or slants) the extension lines of selected linear dimensions.

The obliquing angle is measured counterclockwise from the positive x-axis.

Select linear dimensions (1) and enter the obliquing angle (2). The extension lines (3) will become oblique.

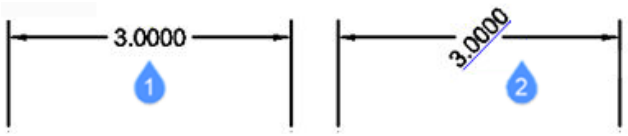


Rotate text

This option will rotate the dimension text.

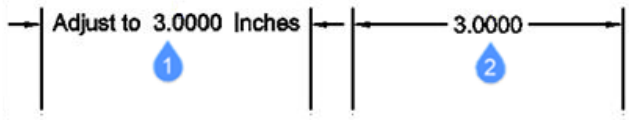
The dimension text rotation angle is measured counterclockwise from the positive x-axis.

Select the dimension text (1) and enter the angle of dimension text (2).



Restore text

This option will restore the dimension text (1) back to its original position (2). This option does not restore edited text or obliqued extension lines.



9.54 DIMLEADER command

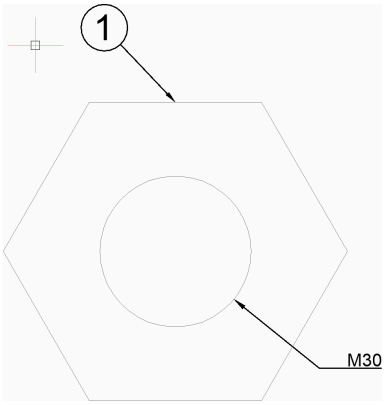
Creates a leader.

Shape Lite Pro Mechanical BIM

Icon: ↙A

9.54.1 Description

Creates a leader by specifying a sequence of points. The leader is based on the current dimension style. Options allow you to specify the format and annotation of the leader.



9.54.2 Methods to createa leader

There is one method to begin creating a leader:

- Start of leader

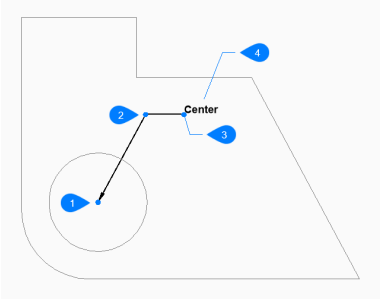
Start of leader
Begin creating a leader by specifying a start point then:

Next point
Specify the next vertex of the leader.

To point
Specify the next vertex. You can continue adding unlimited vertices until you press Enter to access the annotation option.
Additional options: [Format/Undo/Annotation]

Annotation
Enter the first line of annotation text and press Enter or press Enter without entering text to access dimension text options.

Next line of annotation text
Enter the next line of annotation text. You can continue adding lines of text until you press Enter to end the command.



1 Start of leader

2 Next point (vertex)

3 To point

4 Annotation

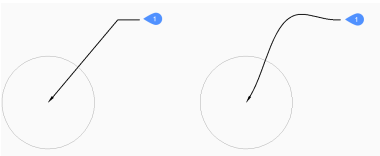
9.54.3 Options within theDIMLEADERcommand

After you begin creating a leader, the following options may be available:

Format

Specify if the leader line includes an arrow and whether it has spline or straight segments:

- Arrow: draw the arrowhead.
- None: does not draw the arrowhead.
- Spline: draw the leader line as a spline.
- Straight: draw the leader line as straight segments.
- Exit: exit the Format options.



1 Center

Undo

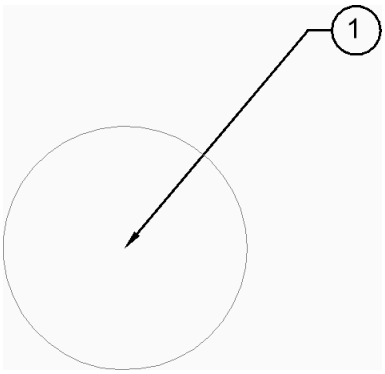
Undo the last leader line segment and continue drawing from the previous one.

Dimension text options [Block/Copy/None/Tolerance/MText]

Specify the type of annotation to use for the leader:

Block

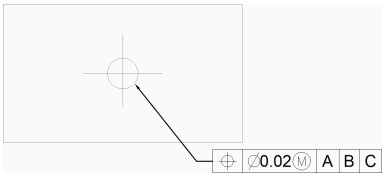
Specify the name of a block in the drawing.
Alternate option: [? to list blocks in drawing/open file Dialog]



Copy
Select an mtext, text, block reference or tolerance entity in the drawing.

None
The leader line is created without annotations.

Tolerance
Specify tolerance using the Geometric Tolerance dialog box.



MText
Enter annotation text using the MText editor.

? to list blocks in drawing
Enter * to list the names of all block definitions in the current drawing. You can also use * as a wildcard with other characters.

Open file Dialog
Select a .dwg file from the file dialog box to use as an annotation block.

9.55 DIMLINEAR command

Creates a linear dimension.

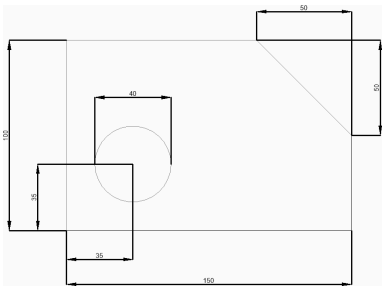
Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMHORIZONTAL, DIMLIN, DIMROTATED, DIMVERTICAL, DLI

9.55.1 Description

Creates a linear dimension that is horizontal, vertical, or rotated. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



9.55.2 Methods to createa linear dimension

This command has 2 methods to begin creating a linear dimension:

- Origin of first extension line
- Select entity

Origin of first extension line

Begin creating a linear dimension by specifying a point for the first extension line then:

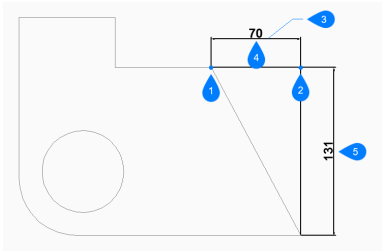
Origin of second extension line

Specify a point for the second extension line.

Location of dimension line

Specify the location of the dimension line. If the two extension points you specified are aligned vertically or horizontally, you are restricted to placing a vertical or horizontal dimension, respectively. If the two extension points you specified are not vertically or horizontally aligned, you can drag the cursor to place either a vertical or a horizontal dimension.

Additional options: [Angle/Text/Horizontal/Vertical/Rotated]



1 Origin of first extension line

2 Origin of second extension line

3 Location of dimension line

4 Horizontal dimension

5 Vertical dimension

Select entity

Begin creating a linear dimension by pressing Enter then:

Select entity to dimension

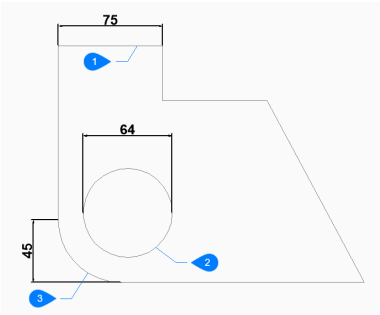
Select a line, polyline segment, arc, or circle to dimension. When selecting a line, the endpoints of the line are used for the extension origins. When selecting an arc, the endpoints of the arc are used for the extension origins. When selecting a circle, the quadrants of the circle are used for the extension origins.

Additional option: [selection options (?)]

Location of dimension line

Specify the location of the dimension line. If the endpoints of the entity you selected are aligned vertically or horizontally, you are restricted to placing a vertical or horizontal dimension, respectively. If the endpoints of the entity you selected are not aligned vertically or horizontally, you can drag the cursor to place either a vertical or a horizontal dimension.

Additional options: [Angle/Text/Horizontal/Vertical/Rotated]



1 Line entity

2 Circle entity

3 Arc entity

9.55.3 Options within the DIMLINEAR command

After you begin creating a linear dimension, the following options may be available:

Horizontal

Specify the location of the dimension to place it in the drawing. This creates a horizontal dimension regardless of how the endpoints are aligned.

Vertical

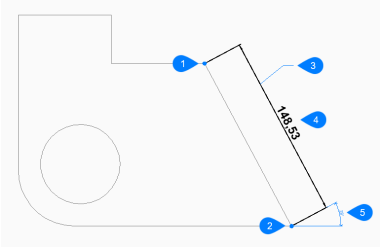
Specify the location of the dimension to place it in the drawing. This creates a vertical dimension regardless of how the endpoints are aligned.

Rotated

Specify an angle for the dimension then:

Location of dimension line

Specify the location of the dimension line. The dimension is rotated from its default (vertical or horizontal) location by the specified angle.



1 Origin of first extension line

2 Origin of second extension line

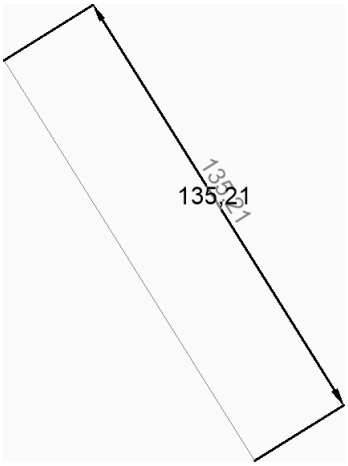
3 Location of dimension line

4 Rotated dimension

5 Angle of dimension line

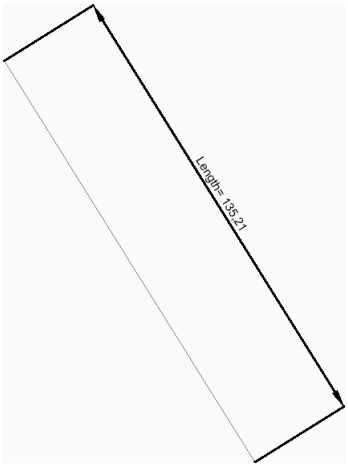
Angle

Specify the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text

Enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



9.56 DIMORDINATE command

Creates an ordinate dimension.

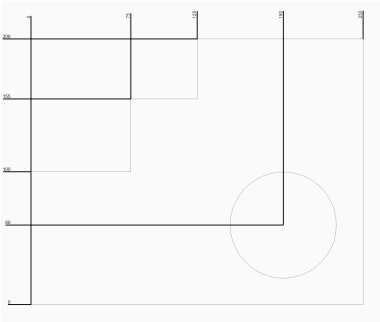
Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMORD, DOR

9.56.1 Description

Create an ordinate dimension that measures the x or y distance from the current UCS origin to a specified point in the drawing. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



9.56.2 Methods to create an ordinate dimension

This command has 1 method to begin creating an ordinate dimension:

- Select point for ordinate dimension

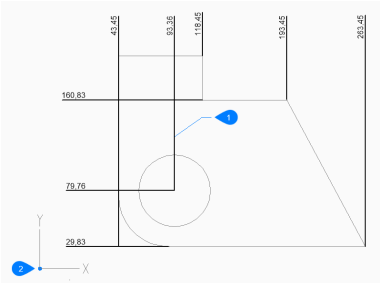
Select point for ordinate dimension

Begin creating an ordinate dimension by specifying a start point for the leader then:

Leader endpoint

Specify an endpoint for the leader. The direction you drag from the start point determines whether you create an X or Y ordinate.

Additional options: [XDatum/YDatum/Text/Angle]



1 Location of ordinate line

2 UCS Basepoint

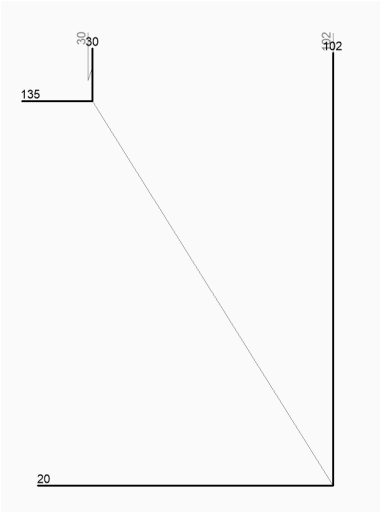
9.56.3 Options within the DIMORDINATE command

After you begin creating an ordinate dimension, the following options may be available:

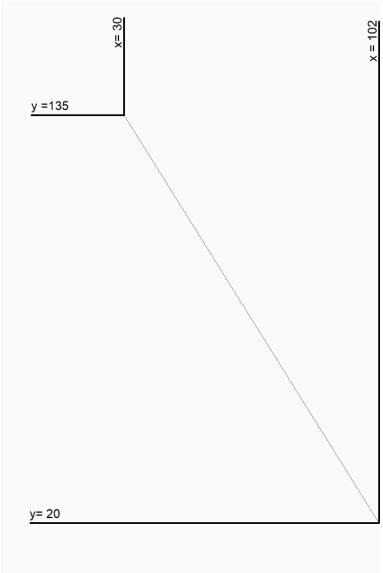
XDatum
Specify the endpoint for an X ordinate regardless of which direction you drag from the start point.

YDatum
Specify the endpoint for a Y ordinate regardless of which direction you drag from the start point.

Angle
Specify the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text
Enter text to override the default dimension text which includes the measured length of the dimension. You can use two angle brackets <> to display the measured length of the dimension in addition to other text.



9.57 DIMOVERRIDE command

Overrides the values of the current dimension style.

Shape Lite Pro Mechanical BIM

Alias: DIMOVER, DOV

9.57.1 Description

Allows the override of dimension style values in a selected dimension.

Note: This command is superseded by the Override function of the DIMSTYLE command.

9.57.2 Options within the command

Clear
Resets the values of dimension variables, clearing the overrides.

9.58 DIMRADIUS command

Creates a radial dimension.

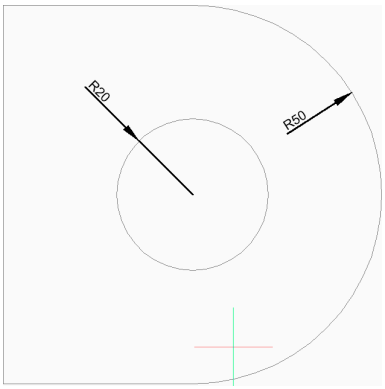
Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMRAD, DRA

9.58.1 Description

Creates a radial dimension for an arc, polyarc, or circle. The dimension is based on the current dimension style. Options allow you to specify the angle and contents of the dimension text.



9.58.2 Methods to create a radial dimension

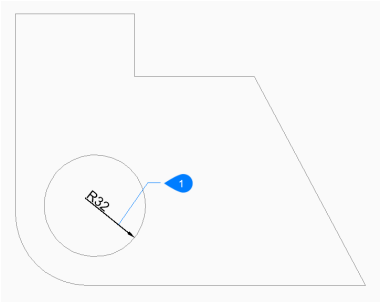
This command has 1 method to begin creating a radial dimension:

- Select arc or circle to dimension

Select arc or circle to dimension
Begin creating a radial dimension by selecting an arc, polyline arc, or circle then:

Location of dimension line
Specify the location of the dimension line.

Additional options: [Angle/Text]

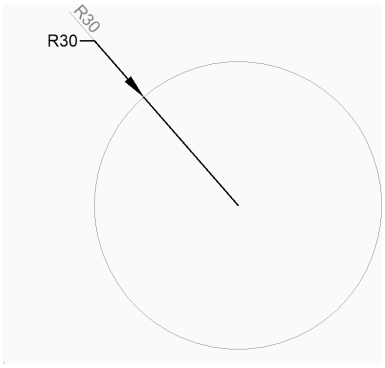


1 Location of dimension line

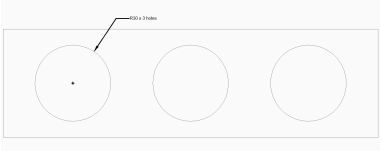
9.58.3 Options within the DIMRADIUS command

After you begin creating a radial dimension, the following options may be available:

Angle
Specify the angle of the dimension text. A value of 0 aligns the dimension text with the dimension line. Any other value rotates the dimension text relative to the x-axis of the current UCS.



Text
Enter text to override the default dimension text which includes the measured length of the radius. You can use two angle brackets <> to display the measured length of the radius in addition to other text.



9.59 DIMREASSOCIATE command

Reassociates or associates dimensions to entities or points on entities.

Shape Lite Pro Mechanical BIM

Icon:

9.59.1 Description

Selected dimension entities highlight one by one with prompts to specify association points or entities for the currently highlighted dimension entity.

Note: A marker displays at the current position of the edited dimension point:

- 
  indicates a non-associated dimension point.
- 
  indicates an associated dimension point.

9.59.2 Methods

The DIMREASSOCIATE command can be used in six different ways:

- Linear (rotated and aligned) - Specify an entity or two lines.
- Diameter - Select arc, circle or polyline arc.
- Radius - Select arc, circle or polyline arc.
- Angular - Select 2 lines or 3 points, an Arc or a circle.
- Ordinate - Specify feature location.
- Leader - Specify leader association point.

9.59.3 Options

Disassociated

Prompts to reassociate ALL disassociated dimension entities in the drawing. All disassociated dimensions highlight one by one.

9.60 DIMREGEN command

Updates associative dimensions.

Shape Lite Pro Mechanical BIM

9.60.1 Description

Updates ALL associative dimensions in the drawing file.

9.61 DIMSTYLE command

Opens the **Drawing Explorer** dialog box with **Dimension Styles** selected.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: D, DDIM, DIMSTY, DS, DST, EXPDIMSTYLES, SETDIM

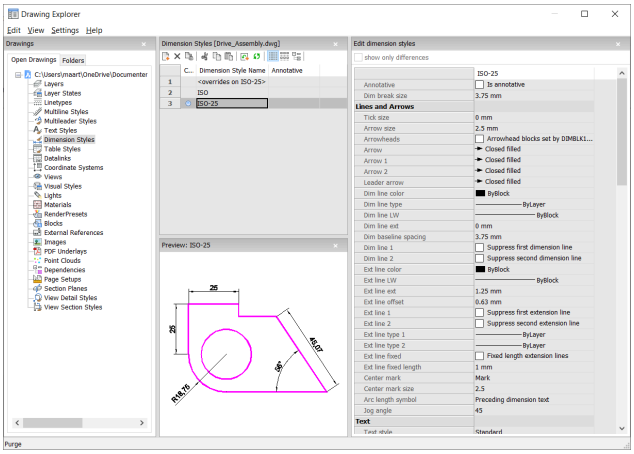
9.61.1 Description

Opens the **Drawing Explorer** dialog box with **Dimension Styles** category selected to view and modifydimension stylesinthe current drawing.

9.61.2 Method

There are two methods when using DIMSTYLE command:

- <Overrides>: overrides the values of individual dimension variables without creating a new style.
- Standard: lists the dimension settings for the dimension style named "Standard".



9.61.3 Context Menu Options

The following options appear when you right-mouse click on a dimension style:

New
Loads additional Dimension Styles definitions into the drawing.

Delete
Deletes Dimension Styles definitions from the drawing. The following dimension styles cannot be deleted:

- <overrides on ISO-25>/<overrides on Standard>
- ISO-25/Standard

Rename
Renames the Dimension Styles. The following dimension styles cannot be renamed:

- <overrides on ISO-25>
- ISO-25

Select All
Selects all Dimension Styles definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected Dimension Style as current.

Save overrides to current style
Saves the override variables of the selected dimension style to the current dimension style.

Save to new style
Creates a new dimension style based on the selected dimension style.

New child style
Creates a new dimension child style. A dimension style can have up to 6 child styles: Linear, Angular, Radius, Diameter, Ordinate, and Leader. If defined, then the child style is used for the corresponding dimension type.
Dimension child styles share all settings with the parent style, except for the properties that are defined explicitly differently.

9.61.4 Options within the command

Edit dimension styles

Show only differences
If multiple styles are selected, compares the selected styles, and shows the differences only.

Name
Names the dimension style. The following dimension styles cannot be renamed:

- <overrides on ISO-25>
- ISO-25

Annotative
Sets the annotative property of the dimension style.

Dim break size
Sets the value of the gap created by the DIMBREAK command.

Lines and Arrows

Lines and Arrows	
Tick size	0 mm
Arrow size	2.5 mm
Arrowheads	Arrowhead blocks set by DIMBLK1...
Arrow	Closed filled
Arrow 1	Closed filled
Arrow 2	Closed filled
Leader arrow	Closed filled
Dim line color	ByBlock
Dim line type	ByLayer
Dim line LW	ByBlock
Dim line ext	0 mm
Dim baseline spacing	3.75 mm
Dim line 1	Suppress first dimension line
Dim line 2	Suppress second dimension line
Ext line color	ByBlock
Ext line LW	ByBlock
Ext line ext	1.25 mm
Ext line offset	0.63 mm
Ext line 1	Suppress first extension line
Ext line 2	Suppress second extension line
Ext line type 1	ByLayer
Ext line type 2	ByLayer
Ext line fixed	Fixed length extension lines
Ext line fixed length	1 mm
Center mark	Mark
Center mark size	2.5
Arc length symbol	Preceding dimension text
Jog angle	45

Tick size

Determines the size of tick marks drawn instead of arrowheads for linear, radius and diameter dimensioning. If the value is zero, arrowheads are drawn.

Arrow size

Determines the size of arrowheads of dimension lines and leader lines.

Arrowheads

Controls whether dimension line arrowhead blocks are set by DIMBLK or DIMBLK1 or DIMBLK2.

Arrow

Determines the name of the block displayed at the ends of dimension lines and leader lines. The block name can be either a standard name or refer to a user-defined arrowhead block.

Arrow 1

Specifies the style of arrowhead for the starting end of the dimension line.

Arrow 2

Specifies the style of arrowhead for the other end of the dimension line.

Leader arrow

Specifies the style of arrowhead for the starting end of the leader line.

Dim line color

Specifies the color of the dimension line; choose from:

- Any Index Color
- Any True Color

- ByBlock
- ByLayer

Dim line type
Specifies the line type of the dimension line. Choose from any line type loaded into the current drawing. To access additional linetypes, click Load..., and then choose one from the **Load Linetypes** dialog box. See the LINETYPE command.

Dim line LW
Specifies the lineweight of the dimension line. Choose from any lineweight supported by the program.

Dim line ext
Specifies the distance that the dimension line extends beyond the extension lines.

Dim baseline spacing
Specifies the default distance between dimensions added with the DIMBASELINE command.

Dim line 1
Toggles the display of the first half of the dimension line (between the starting extension line and the text).

Dim line 2
Toggles the display of the second half of the dimension line (between the other extension line and the text).

Ext line color
Specifies the color of the dimension line. To select additional colors, click Select color... and choose one from the **Select Color** dialog box.
See the COLOR command.

Ext line LW
Specifies the lineweight of the extension line. Choose from any lineweight supported by the program.

Ext line ext
Specifies the distance that the extension lines extend beyond the dimension line.

Ext line offset
Specifies the offset distance between the object and the start of the extension lines.

Ext line 1
Toggles the display of the first extension line.

Ext line type 1
Specifies the line type of the first extension line. You can choose from any line type loaded into the current drawing.
To access additional linetypes, click Load... . The **Load Linetypes** dialog box is displayed where you can choose a linetype.

Ext line type 2
Specifies the line type of the second extension line. You can choose from any line type loaded into the current drawing.
To access additional linetypes, click Load... . The **Load Linetypes** dialog box is displayed where you can choose a linetype.

Ext line 2
Toggles the display of the second extension line.

Ext line fixed
Determines whether fixed length extension lines are used.

Ext line fixed length
Specifies the total length of the extension lines.

Center mark
Specifies the type of center mark:

- Mark
- Line
- None

Center mark size
Specifies the size of the center mark.

Arc length symbol
Specifies the location of the arc length symbol:

- Preceding dimension text.
- Above dimension text.
- Not displayed.

Jog angle
Specifies the angle of the jog (default is 45 degrees). Enter another angle.

Text

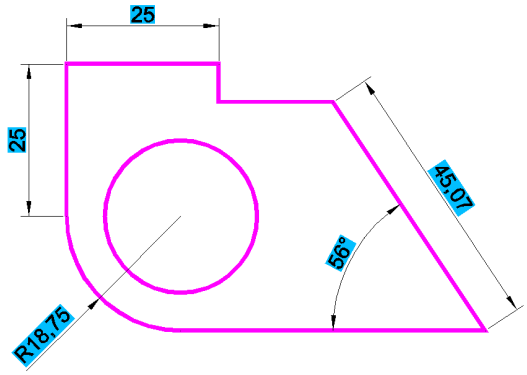
Text	
Text style	Standard
Text color	ByBlock
Text fill	No fill
Text fill color	ByBlock
Text height	2.5 mm
Draw frame around text	Draw frame around text
Text position vertical	Above
Text position horizontal	Centered
Text offset	0.625
Text vertical offset	0
Text inside align	Horizontal
Text outside align	Horizontal
Text view direction	Left to right

Text style
Specifies the style for dimension text; can use only styles created with the STYLE command.

Text color
Specifies the color of the text. Click Select color... to select additional color from the **Select Color** dialog box.

Text fill
Determines whether the dimension text has a rectangular background filled with color.

- No fill: does not apply a background fill.
- Background: uses the background color of the drawing, usually white or black.
- Color: uses the color specified by the Text Fill Color option.



Text fill color
Specifies the background fill color when Text Fill is set to Color. You can choose a color from the drop-down list or choose Select Color..., and choose a color from the **Select Color** dialog box.

Text height
Specifies the height of the text.

Draw frame around text
Draws a rectangle around the text.

Text position vertical
Justifies the text vertically relative to the dimension line:

- Centered: centers the text on the dimension line.
- Above: places the text above the dimension line.
- Outside: places text outside of the extension lines.
- JIS: places text above the dimension line, according to the Japanese Industry Standard.
- Below: places the text below the dimension line.

Text position horizontal
Justifies the text horizontally relative to the extension lines:

- Centered: centers the text between the extension lines.
- First Extension Line: places the text near the first extension line.
- Second Extension Line: places the text near the second extension line.
- Over First Extension: places the text over the first extension line.
- Over Second Extension: places the text over the second extension line.

Text offset
Specifies the size of gap between the dimension line and the text.

Text vertical offset
Sets the vertical position of dimension text above or below the dimension line.

Text inside align
Justifies text when located between extension lines:

- Aligned with Dimension Line: aligns text with dimension line.
- Horizontal: forces text to be always horizontal.

Text outside align

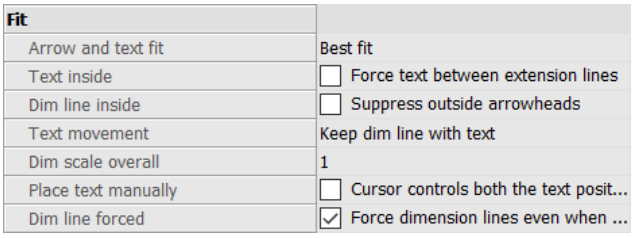
Justifies text when located outside of the extension lines:

- Aligned with Dimension Line: aligns text with dimension line.
- Horizontal: forces text to be always horizontal.

Text view direction

Sets the dimension text reading direction.

Fit



Arrow and text fit

Specifies where text and arrows should be located when there is insufficient space for both between the extension lines:

- Text and Arrows: forces both text and arrows between the extension lines.
- Arrows Only: forces arrows to stay within extension lines; moves text outside when there is insufficient space.
- Text Only: forces text to stay within extension lines; moves arrows outside when there is insufficient space.
- Best Fit: requires the program to figure out where to place text and arrows, depending on the space between extension lines.

Text inside

Specifies whether the text is forced between extension lines:

- On: forces text between extension lines.
- Off: draws text outside if there is insufficient space between extension lines.

Dim line inside

Specifies whether the dimension line is forced between extension lines:

- On: forces dimension line always between extension lines.
- Off: draws dimension line outside if there is insufficient space between extension.

Text movement

Specifies what happens when text is moved away from its default location:

- Keep dim line with text: moves the dimension line with the text, stretching the extension lines.
- Move text, add leader: draws a leader between the text and the dimension line.
- Move text, no leader: does not draw a leader.

Dim scale overall

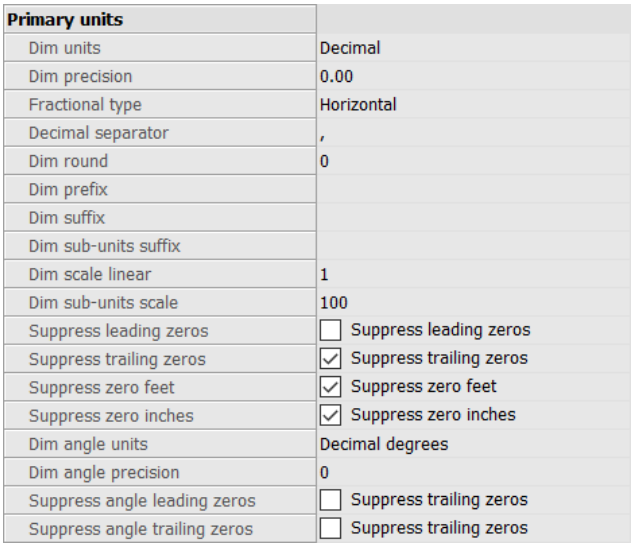
Specifies the overall scale factor for dimensions. This affects the size of arrows and text only.

This setting is not editable for annotative dimension styles. It is recommended to set Dim Scale Overall = 1 for annotative dimension styles.

Place text manually
Toggles whether the user must always specify the text location when creating dimensions.

Dim line forced
Forces the dimension line to be always drawn; forces leaders to be drawn with the DIMDIAMETER and DIMRADIUS commands.

Primary units



Primary units	
Dim units	Decimal
Dim precision	0.00
Fractional type	Horizontal
Decimal separator	,
Dim round	0
Dim prefix	
Dim suffix	
Dim sub-units suffix	
Dim scale linear	1
Dim sub-units scale	100
Suppress leading zeros	☐ Suppress leading zeros
Suppress trailing zeros	☑ Suppress trailing zeros
Suppress zero feet	☑ Suppress zero feet
Suppress zero inches	☑ Suppress zero inches
Dim angle units	Decimal degrees
Dim angle precision	0
Suppress angle leading zeros	☐ Suppress trailing zeros
Suppress angle trailing zeros	☐ Suppress trailing zeros

Dim units
Specifies the display units for dimensions:

- Scientific: scientific notation, such as 1.2345E+01
- Decimal: metric, such as 1.2345
- Engineering: feet and decimal inches, such as 1'-2.3456"
- Architectural: feet and fractional inches, such as 1'-2 1/16"
- Fractional: fractional inches only; no feet, such as 14 1/16"
- Windows Desktop: uses the units set by Windows

Dim precision
Specifies the precision of units, either decimal places or fractional accuracy.

Fractional type
Specifies how fractions are stacked:

- Horizontal: stacks fractions vertically, with a horizontal separator line.
- Diagonal: stacks diagonally, with a diagonal separator line.
- None: does not stack fractions, placed horizontally with a slash separator.

Decimal separator
Specifies the character used to indicate the decimal point; can be any character.
North American countries use the period; European countries use the comma.

Dim round
Specifies the rounding of decimal numbers. The range goes from none to 8 decimal places.

Dim prefix
Specifies a prefix text that appears in front of the dimension text, if any.

Dim suffix
Specifies a suffix text that appears after the dimension text, if any.

Dim sub-units suffix
Specifies suffix text in case the subunit is used. For example, enter 'cm' to display 0.96 as 96cm.

Note: suppress leading zeros must be set to Yes to display dimension distances less than one unit in sub units.

Dim scale linear
Specifies the scale factor for linear dimension values, such as 25.4 for changing inches to millimeters. Positive values apply to dimensions in both model space and paper space. Negative values apply to paper space dimensions only.

Dim sub-units scale
Sets the scale factor for the sub-units. For example, enter 10 if the drawing unit is cm, and the subunit suffix is mm.

Note: suppress leading zeros must be set to Yes to display dimension distances less than one unit in subunits.

Suppress leading zeros
Toggles the display of zeros in front of the decimal point. For example, 0.23 is displayed as:

- On: suppresses the leading zero, such as .23
- Off: permits the leading zeros, such as 0.23

Suppress trailing zeros
Toggles the display of zeros after the decimal point. For example, 1.2300 is displayed as:

- On: suppresses trailing zeros, such as 1.23
- Off: allows trailing zeros, such as 1.2300

Suppress zero feet
Toggles the display of zero feet. For example, 0'-3" is displayed as:

- On: suppresses zero feet, such as 3"
- Off: allows zero feet, such as 0'-3"

Suppress zero inches
Toggles the display of zero inches; for example, 1'-0" is displayed as:

- On: suppresses the display of zero inches, such as 1'
- Off: allows zero inches, such as 1'-0"

Dim angle units
Specifies the format of units in angular dimensions:

- Decimal Degrees: 360 degrees in a circle. For example, 123.45 degrees
- Deg/Min/Sec: degrees, minutes, seconds. For example, 123d 12' 45.67"
- Grads: 400 grads in a circle. For example, 230g
- Radians: 2pi radians in a circle. For example, 1.5r

Dim angle precision
Specifies the number of decimal places. The range goes from 0 to 8.

Suppress angle leading zeros
Toggles the display of zero degrees. For example, 0.1234 degrees is displayed as:

- On: suppresses zero degrees, such as .1234
- Off: allows zero degrees, such as 0.1234

Suppress trailing leading zeros
Toggles the display of zeros after degrees. For example, 0.1200 degrees is displayed as:

- On: suppresses trailing zeros, such as 0.12
- Off: allows trailing zeros, such as 0.1200

Alternate units

Alternate units	
Alt enabled	☐ Enable alternate units
Alt units	Decimal
Alt precision	0.000
Alt scale factor	0.03937007874
Alt sub-units scale	100
Alt round	0
Alt prefix	
Alt suffix	
Alt sub-units suffix	
Alt suppress leading zeros	☐ Alt suppress leading zeros
Alt suppress trailing zeros	☐ Alt suppress trailing zeros
Alt suppress zero feet	☑ Alt suppress zero feet
Alt suppress zero inches	☑ Alt suppress zero inches

Alt enabled
Toggles the display of alternate units:

- On: second, alternative units displayed to the right of primary units.
- Off: only primary units are displayed.

Alt units
Specifies the alternate unit type for linear dimensions:

- Scientific: scientific notation, such as 1.2345E+01.
- Decimal: metric, such as 1.2345
- Engineering: feet and decimal inches, such as 1'-2.3456"
- Architectural Stacked: such as 4'-6.61"
- Fractional Stacked: such as 54 1/2
- Architectural: feet and fractional inches, such as 1'-2 1/16"
- Fractional: fractional inches only; no feet, such as 14 1/16"
- Windows Desktop: uses the units set by Windows

Alt precision
Specifies the precision of alternate units, either decimal places or fractional accuracy.

Alt scale factor
Specifies the multiplier for alternate values, such as 25.4 for showing millimeters (alternate units) next to inches (primary units).

Alt sub-units scale
Specifies the multiplier for alternate values, such as 25.4 for showing millimeters (alternate units) next to inches (primary units).

Alt round
Specifies the rounding of alternate numbers. The range goes from zero to 8 decimal places.

Alt prefix
Specifies a prefix text that appears in front of the alternate text, if any.

Alt suffix
Specifies a suffix text that appears after the alternate text, if any.

Alt sub-units suffix
Specifies suffix text in case the subunit is used.

Note: suppress leading zeros must be set to Yes to display alternate distances less than one unit in subunits.

Alt suppress leading zeros
Toggles the display of zeros in front of the decimal point. For example, 0.23 is displayed as:

- On: suppresses the leading zero, such as .23
- Off: permits the leading zero, such as 0.23

Alt suppress trailing zeros
Toggles the display of zeros in front of the decimal point. For example, 1.2300 is displayed as:

- On: suppresses trailing zeros, such as 1.23
- Off: allows trailing zeros, such as 1.2300

Alt suppress zero feet
Toggles the display of zero feet; for example, 0'-3" is displayed as:

- On: suppresses zero feet, such as 3" Off: allows zero feet, such as 0'-3"

Alt suppress zero inches
Toggles the display of zero inches. For example, 1'-0" is displayed as:

- On: suppresses the display of zero inches, such as 1'
- Off: allows zero inches, such as 1'-0"

Tolerances

Tolerances	
Tolerance display	☐ Display tolerance
Limits display	☐ Generate dimension limits as defa...
Tolerance precision	0.00
Tolerance limit lower	0
Tolerance limit upper	0
Tolerance text height	1 mm
Tolerance position vertical	Bottom
Tolerance suppress leading zeros	☐ Tolerance suppress leading zeros
Tolerance suppress trailing zeros	☑ Tolerance suppress trailing zeros
Tolerance suppress zero feet	☑ Tolerance suppress zero feet
Tolerance suppress zero inches	☑ Tolerance suppress zero inches
Alt tolerance precision	0.000
Alt tolerance suppress leading zeros	☐ Alt tolerance suppress leading zeros
Alt tolerance suppress trailing zeros	0
Alt tolerance suppress zero feet	☑ Alt tolerance suppress zero feet
Alt tolerance suppress zero inches	☑ Alt tolerance suppress zero inches

Tolerance display
Toggles the display of tolerance text.

Limits display
Sets the default text to dimension limits.

Tolerance precision
Specifies the display precision of tolerance text. The range goes from 0 to 8 decimal places or 1/1 to 1/256 inch.

Tolerance limit lower
Specifies the value of the upper tolerance.

Tolerance limit upper
Specifies the value of the lower tolerance.

Tolerance text height
Specifies the height of the tolerance text.

Tolerance position vertical
Locates the tolerance text relative to the dimension text:

- Bottom: aligns tolerance text with the bottom of the dimension text.
- Middle: aligns tolerance text with the middle of the dimension text.
- Top: aligns tolerance text with the top of the dimension text.

Tolerance suppress leading zeros
Toggles the display of zeros in front of the decimal point. For example, 0.23 is displayed as:

- On: suppresses the leading zero, such as .23
- Off: permits the leading zero, such as 0.23

Tolerance suppress trailing zeros
Toggles the display of zeros behind the decimal point. For example, 1.2300 is displayed as:

- On: suppresses trailing zeros, such as 1.23
- Off: allows trailing zeros, such as 1.2300

Tolerance suppress zero feet
Toggles the display of zero feet; for example, 0'-3" is displayed as:

- On: suppresses zero feet, such as 3"
- Off: allows zero feet, such as 0'-3"

Tolerance suppress zero inches
Toggles the display of zero inches. For example, 1'-0" is displayed as:

- On: suppresses the display of zero inches, such as 1'
- Off: allows zero inches, such as 1'-0"

Alt tolerance precision
Specifies the number of decimal places for tolerances in alternate units.

Alt tolerance suppress leading zeros
Toggles the display of zeros in front of the decimal point. For example, 0.23 is displayed as:

- On: suppresses the leading zero, such as .23
- Off: permits the leading zero, such as 0.23

Alt tolerance suppress trailing zeros
Toggles the display of zeros behind the decimal point. For example, 1.2300 is displayed as:

- On: suppresses trailing zeros, such as 1.23
- Off: allows trailing zeros, such as 1.2300

Alt tolerance suppress zero feet
Toggles the display of zero feet; for example, 0'-3" is displayed as:

- On: suppresses zero feet, such as 3"
- Off: allows zero feet, such as 0'-3"

Alt tolerance suppress zero inches
Toggles the display of zero inches. For example, 1'-0" is displayed as:

- On: suppresses the display of zero inches, such as 1'
- Off: allows zero inches, such as 1'-0"

9.62 -DIMSTYLE command

Creates and modifies dimensions styles at the Command line.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: -DST

9.62.1 Method

Run the command to create a new dimension style by choosing one of the options. The command also prompts the name of the current dimension style.

9.62.2 Options within the command

?
Lists the names of all dimension styles loaded into the current drawing, other than Standard.

ANnotative
Sets the annotative property of the dimension style.

Apply
Changes or restores the style of existing dimensions by applying the current style.

Save
Saves the current dimension settings as a style.

Note: This option is useful for saving overrides as a style.

STatus
Displays the current value of every dimension variable.

Variables
Lists the values of dimension variables belonging to the selected dimension.

Restore
Resets the values of dimension variables to those of the selected dimension.

9.63 DIMSTYLESET command

Reports the current dimension style in the Command line.

Shape Lite Pro Mechanical BIM

9.63.1 Method

The name of the current dimension style is also shown in the BricsCAD application window as Current Dimension Style in the Status bar.

9.64 DIMTEDIT command

Changes the position of dimension text.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DIMTED

9.64.1 Description

Allows you to change the position and angle of the text within a dimension entity.





9.64.2 Methods

This command offers four methods to change the position of dimension text.

- Rotate the dimension text.
- Move the dimension text to the left.
- Move the dimension text to the right.
- Center the dimension text.

9.64.3 Options

Angle
Rotates the dimension text from the original dimension text position (1) to the rotated dimension text position (2).

Left
Moves the dimension text next to the left extension line of the dimension, depending on the original positioning of the dimension.

Center
Centers the dimension text between the two extension lines of the dimension.

Right
Moves the dimension text next to the right extension line of the dimension, depending on the original positioning of the dimension.

REstore
Resets the rotation angle of the dimension text to 0 degrees and does not change repositioned dimension text.

9.65 DISH command

This command is obsolete and exists for backward compatibility only. Use the AI_DISH command instead.

Shape Lite Pro Mechanical BIM

Icon:

9.66 DIST command

Reports the distance and angle between two points.

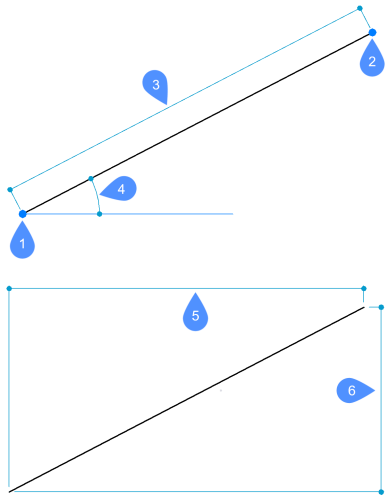
Shape Lite Pro Mechanical BIM

Icon:

Alias: DI

9.66.1 Method

The distance can be measured between two points or multiple points.



1 Starting point for distance
2 End point
3 Distance
4 Angle in XY plane
5 X Distance
6 Y Distance

Note: When DRAGSNAP = ON and the Dynamic Dimensions are enabled, the distance is displayed in the dynamic entry field when hovering over the second point using an appropriate entity snap.

9.66.2 Options within the command

Multiple points
Allows to measure the total distance between multiple points.

Arc
Allows to specify an arc distance.

Angle
Specify the angle for the arc.

Radius
Specify the radius of the arc.

Center
Specify the center of the arc. Hold CTRL key to switch direction.

Direction
Specify the direction of the arc.

Line
Allows to specify a line distance.

Length
Allows to add a distance.

Undo
Removes the last point.

Total
Prints the total length in the Command line and concludes the command.

9.67 DISTANTLIGHT command

Places distant lights for renderings.

Shape Lite Pro Mechanical BIM

Icon:

9.67.1 Description

Places distant lights for use with renderings. Distant lights represent sources that are far off, such as the sun, and so does not display a glyph in drawings.

Note: having DEFAULTLIGHTING=1 will open a dialog box where you are asked.

9.67.2 Options

Name
Specifies a name for the light.

Intensity factor
Indicates the relative brightness of the light.

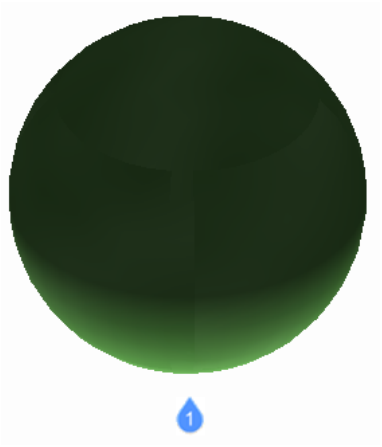
Status
Toggles the light on and off

Photometry
Specifies the parameters for color and intensity.

shadoW
Specifies the look of shadows, if any.

filterColor
Sets the color of the light.

Vector
The direction of the light beams can be defined by a vector, whose XYZ tuple determines the direction of the light, starting at infinity. The default direction points up, in the Z direction.
Unlike regular vectors, this lighting vector specifies only direction, not magnitude.



1: Distant light illuminating from default vector direction.

9.68 DIVIDE command

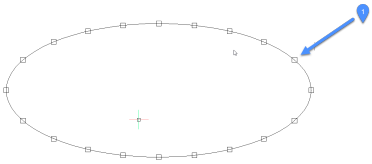
Places a specific number of points or blocks spaced equally along an entity.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DIV

9.68.1 Method



1. Points placed evenly divided along entity

The command allows creating and placing points or blocks along the perimeter of an entity, at equal distances.

The following entities can be divided:

- Line
- Polyline
- Spline
- Arc
- Circle
- Ellipse

Note: It might be necessary to adjust the Point display mode settings (DDPTYPE) to display the points correctly.

9.68.2 Options

Number of segments
An integer number between 2 and 32767 can be specified. This will represent the number of segments, the equally-divided distances between points.

Note: Entering 2 for the number of segments places a single point or block at the midpoint of the entity, as the entity is divided into two segments. There is always one more segment than points placed, as no point or block is placed at the start or endpoints of the entity.

insert Block
Place equally-spaced blocks along the entity, instead of points. You can specify the name of the block or use ? to list available blocks.

Yes – align blocks
Rotates blocks for them to match the local orientation of the entity, such as along an arc.

No – do not align
The blocks maintain their orientation (they do not rotate).

9.69 DMANGLE3D command

Creates an angle constraint between faces and/or edges of 3D solids, WCS subentities and subentities of a coordinate system of a block.

Shape Lite Pro Mechanical BIM

Icons: 

9.69.1 Description

Creates an angle constraint between faces and/or edges of 3D solids, WCS subentities, and subentities of a coordinate system of a block. Creates a Planar Angle constraint by default: coordinate planes of the WCS are used as third reference objects whenever possible. It also allows to control the top angle of a cone. Press the TAB key to select obscured geometry.

9.69.2 Options

cone Angle constraint
Allows you to control the top angle of a cone by specifying the angle between the axis and the cone face (= cone half angle). Values must be smaller than 90°.

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block. You can choose from: X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane.

Set reference entity
Allows you to manually specify the reference entity:

Keep as is
allows you to keep the proposed reference object;

WOrld
allows you to select the WCS subentity as an argument;

BLock
allows you to choose as an argument the subentity of the coordinate system of the block;

None
creates Angle constraint instead of Planar Angle constraint, without third argument.

Geometry-driven
Accepts the current value and creates a geometry-driven constraint.

9.70 DMAUDIT command

Analyzes and repairs errors.

Shape Lite Pro Mechanical BIM

Icon: 

9.70.1 Description

It is a powerful tool to analyze and automatically fix problems in 3D geometry supported by ACIS kernel (3D solids, surfaces).

Note: It is recommended to run DMAUDIT whenever 3D geometry is imported.

9.70.2 Methods

There are two methods to audit the drawing by selecting one or more entities or the entire model:

- Check problems
- Fix problems

9.70.3 Options within the command

Check
Checks the 3D geometry in the drawing for errors without fixing them.

Fix
Repairs errors.

Settings
Audits the drawing in function of one of the options you select.

Back
Returns to the main prompt of the command.

Report panel
Redirects the output of the command to the Report panel.

Note: Depending on the value of REPORTPANELMODE variable, it might be necessary to click on the red exclamation mark in the lower right corner to see more details in the Report panel.

Copy to file
Reports the output of the command in a file.

Audit level
Sets the strictness of the check.

Low
Runs basic analysis (for example, catching common fatal topology errors).

High
Goes deep (for example, catching self-intersections in solid bodies).

Medium
This is the default level.

Coincident faces
Checks and fixes coincident faces. Requires audit level set to High.

Nested blocks
Checks and fixes entities in nested blocks.

9.71 DMAUDITALL command

Analyzes and repairs errors in drawings inserted as external references

Shape Lite Pro Mechanical BIM

9.71.1 Description

It is a powerful tool to analyze and automatically fix problems in 3D geometry supported by ACIS kernel (3D solids, surfaces).

Note: It is recommended to run DMAUDITALL whenever 3D geometry is imported.

9.71.2 Method

There are two methods to audit the drawing:

- Check problems.
- Fix problems.

9.71.3 Options within the command

Check
Checks the 3D geometry in the drawing for errors without fixing them.

Fix
Repairs errors.

Back
Returns to the main prompt of the command.

report Panel
Redirects the output of the command to the Report panel.

Note: Depending on the value of REPORTPANELMODE variable, it might be necessary to click on the red exclamation mark in the lower right corner to see more details in the Report panel.

copy to File
Reports the output of the command in a file.

audit Level
Sets the strictness of the check.

Low
Runs basic analysis (for example, catching common fatal topology errors).

High
Goes deep (for example, catching self-intersections in solid bodies).

Medium
This is the default level.

Coincident faces
Checks and fixes coincident faces. Requires audit level set to High.

9.72 DMCHAMFER command

Creates equal and variable distance chamfers between adjacent faces sharing a sharp edge.

Shape Lite Pro Mechanical BIM

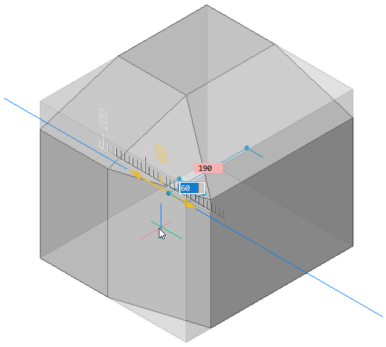
Icons: 

9.72.1 Description

Creates equal or variable distance chamfers between adjacent faces that share a sharp edge.

After selecting the edges to create chamfer you may choose to specify a constant symmetrical offset or one of the available advanced options.

The chamfer is applied dynamically, and the manipulator is displayed.



9.72.2 Options

Allows to create variable chamfers. All advanced mode chamfer procedures start by creating a symmetrical offset, which is then modified by specifying offsets and/or angles. Before specifying the final value, you can press the TAB key to modify previously specified values.

ASymmetrical
Allows you to specify two offsets.

ANgular
Allows you to specify an offset and an angle.

Variable Symmetrical
Allows you to specify two offsets.

VAriable aSymmetrical *
Allows you to specify four offsets.

VAriable aNgular *
Allows you to specify two pairs of offsets and an angle.

* Creates a curved chamfer face.

9.73 DMCOINCIDENT3D command

Applies a coincident constraint between two subentities of 3D entities.

Shape Lite Pro Mechanical BIM

Icons: 

9.73.1 Method

Applies a coincident constraint between two edges, two faces, an edge and a face or a vertex and a face or edge of two different solids or surfaces.

Select the two cylindrical (circular or elliptical), spherical or conical surfaces between which a concentric constraint will be applied. Press the TAB key to select obscured geometry.

9.73.2 Options

Autoconstrain
Automatically creates a coincident constraint between each pair of connected faces in selection of solids. You can check which constraints are created in the Mechanical Browser panel or in Parameter and Constraints panel.

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.74 DMCONCENTRIC3D command

Applies a concentric constraint between two circular subentities of 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.74.1 Method

The concentric constraint could be applied between any two circular surfaces or circular edges of a 3D entities (cylindrical (circular or elliptical), spherical or conical surfaces and their edges) or between a subentity of a 3D entity and a 2D circular entity.

Select the two circular subentities between which the concentric constraint will be applied. Press the TAB key to select obscured geometry.

9.74.2 Options within the command

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.75 DMCONSTRAINT3D command

Applies geometric and dimensional constraints for 3D entities.

Shape Lite Pro Mechanical BIM

9.75.1 Description

Applies geometric and dimensional constraints between faces, surfaces or edges of 3D entities.

The geometric constraints create a specific relationship between two subentities/entities, like parallelism, perpendicularity, tangency, coincidence, concentricity, fixed or rigid.

The dimensional constraints restrain the value of a dimension of an entity or between entities, like radius, angle or distance.

Select the subentities between which you will apply a dimensional or geometric constraint.

Press the TAB key to select obscured geometry.

9.75.2 Options

New
Allows to create a new parameter.

Fix
Applies a fixed constraint to a solid or to an edge or a face of a solid. The option is equivalent to the DMFIX3D command.

Coincident
Applies a coincident constraint between two edges, two faces, or an edge and a face of two different solids. The option is equivalent to the DMCOINCIDENT3D command.

PAth
Allows a point to move along a curve by specifying the curve's parameters with an expression. The option is equivalent to the DMPATH3D command.

CONcentric
Applies a concentric constraint between two cylindrical, spherical or conical surfaces. The option is equivalent to the DMCONCENTRIC3D command.

Parallel
Applies a parallel constraint between two faces of a solid or of different solids. The option is equivalent to the DMPARALLEL3D command.

PErpendicular
Applies a perpendicular constraint between two faces of a solid or of different solids. The option is equivalent to the DMPERPENDICULAR3D command.

Tangent
Applies a tangent constraint between a face and a curved surface of different solids. The option is equivalent to the DMTANGENT3D command.

RIgidset
Allows to define a set of entities or subentities as a rigid body. The option is equivalent to the DMRIGIDSET3D command.

Distance
Applies a distance constraint between two subentities of a solid or of different solids. The option is equivalent to the DMDISTANCE3D command.

Radius
Applies a radius constraint to cylindrical surfaces or circular edges. The option is equivalent to the DMRADIUS3D command.

Angle
Applies an angle constraint between two faces of a solid or of different solids. The option is equivalent to the DMANGLE3D command.

Edit
Allows to edit the named constraint

NodeId
Specify the constraint by its ordered number.

Enable
Toggles the selected constraint Enable/Disable.

change Arguments
Allows to change the 3D constraint arguments.

Replace argument
Specify the index of the argument to be replaced and its replacement.

WOrld
Select a reference coordinate system subentity.

BLock
Select a block reference.

Exclude argument
Specify the index of the argument to be excluded.

Note: This option is available only for RigidSet constraints.

Add arguments
Specify the position where the new arguments to be added.

Note: This option is available only for RigidSet constraints.

Append arguments
The new argument is added at the end of the list.

Edit Arguments
Allows to edit the list of 3D constraint arguments by selecting entities or by choosing one of the options.

Note: This option is available only for RigidSet constraints.

Rename
Allows to rename a constraint.

Delete
Allows to delete a constraint.

?
Lists the constraints and their status.

9.76 DMCOPYFACES command

Copies a feature from a 3D solid.

Shape Lite Pro Mechanical BIM

Icon: 

9.76.1 Description

Copies a set of 3D solid faces forming a protrusion or depression feature, such as holes, pockets, extrusions, ribs, and their combinations within the same 3D solid or from one 3D solid to another.

9.76.2 Options within the command

Select entities
Select all faces of the entire feature.

Base point
Specify the base point.

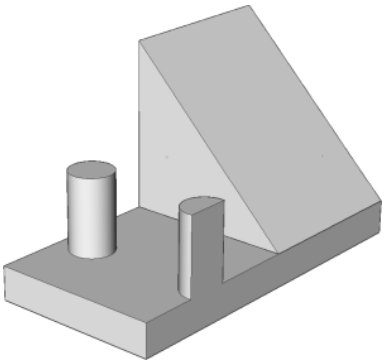
Insertion point
Allows you to select an insertion point.

Rotate
Rotates the faces around the Z axis of the UCS.

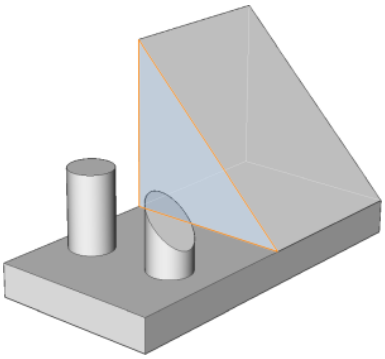
mUltiple
Insert multiple copies of a feature.

MOde
Toggles between various cutting options.

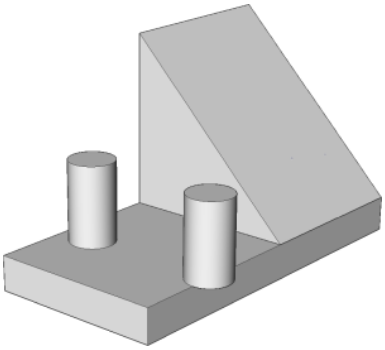
Placement Face
Cuts the copied feature so that it suits the placement face.



Select Faces
Cuts the copied feature so that it suits the selected face.



No Cut
Does not cut the copied feature.



9.77 DMDEFORMCURVE command

Deforms 3D solids or surfaces by replacing their edges with given curves.

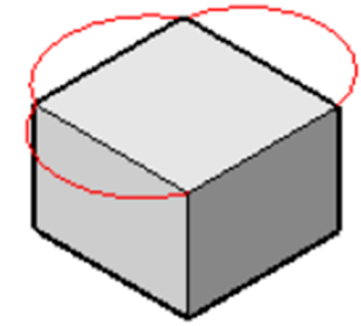
Shape Lite Pro Mechanical BIM

Icon:

9.77.1 Description

Deforms one or more connected faces of a 3D solid or a surface by replacing their edges with given curves.

Select the edges that will be replaced with the target curves to get the new 3d solid or surface.





9.77.2 Options

add Faces
Allows to select additional faces, other than the faces adjacent to selected edges, to deform.

Multiple edges
Allows the deformation of a chain of edges to a single target curve.

9.78 DMDEFORMMOVE command

Deforms 3D solids, surfaces or regions by moving and/or rotating their edges.

Shape Lite Pro Mechanical BIM

Icon:

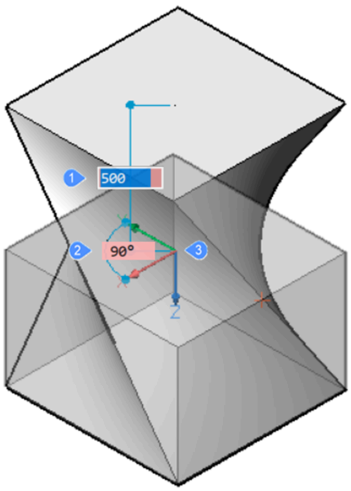
9.78.1 Description

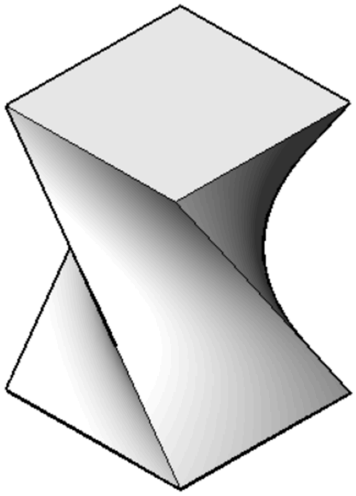
Deforms connected faces of a 3D solid or a surface by moving and/or rotating their edges.

To specify the deformation do one of the following:

- Specify a point in the drawing.
- Type a value in the dynamic entry fields.
- Hit the TAB key to switch between the distance (1) and angle (2) fields.

The edges are rotated around an axis passing through the base point (3) in the deformation direction.





Dynamics (left) and final result (right)

9.78.2 Options

Base point
Allows to specify the base point. The deformation axis passes through the base point.

Direction
Allows to specify the direction of the deformation axis. You may choose to define the direction by specifying two points in the drawing or choose one of the UCS axis.

additional Faces
Allows to select additional faces to be deformed, in addition to ones adjacent to the selected edge(s).

9.79 DMDEFORMPOINT command

Deforms a region, face(s) of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction.

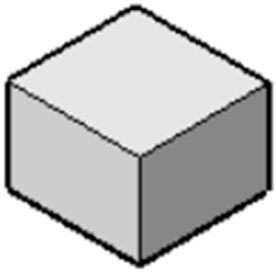
Shape Lite Pro Mechanical BIM

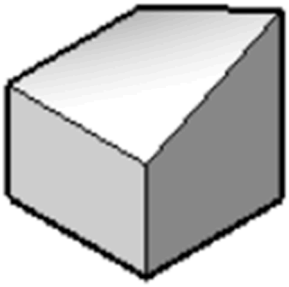
Icon:

9.79.1 Description

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. The initial continuity between deformed faces (G1 – tangent faces, or G2 – curvature continuity) is kept when deformed.

Select a region or a face or some connected faces of a 3D solid or surface, then specify the deformation point and the deformation value to deform the entity.





9.79.2 Options

change Parameters
Choose a method to change parameters.

Alpha
Specifies the resistance to stretching. It is a second order tensor which can be described with three numbers. Each value must be either 0 or a positive number:

- alpha U: U-direction resistance,
- alpha V: V-direction resistance,
- alpha theta: the angle between U and V principal directions of the surface and the material property directions.

Beta
Specifies the resistance to bending. Similarly to Alpha, Beta is defined as a triple: beta U, beta V, beta theta. Each value must be either 0 or a positive number.

Gamma
Specifies the resistance to the rate of change in bending. The value must be either 0 or a positive number.

Delta
Specifies the resistance to deviations from the default shape. The value must be either 0 or a positive number.

set Target Point
The entities are deformed towards the specified point.

set Direction
The selected entities are deformed dynamically in the specified direction.

change Base point
Select a new deformation point.

9.80 DMDELETE command

Erases entities from the drawing.

Shape Lite Pro Mechanical BIM

Icon: 

The DMDELETE command was integrated into the ERASE command.

9.81 DMDISTANCE3D command

Applies a distance constraint between two entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.81.1 Description

Applies a distance constraint between two subentities of the same solid or of different solids, as well as WCS entities, points, cylinders, and spheres.

Select the two entities/subentities for which the distance constraint will be applied by specifying the distance value. Press the TAB key to select obscured geometry.

9.81.2 Options

Use as
Gives the option to specify how the distance must be measured between cylindrical and spherical faces. The default measurement mode is Axis.

Boundary
The boundary of the argument.

Axis
An argument that has an axis; circle, cylinder, cone, or torus.

Center
An argument that has a central point; circle, sphere or torus.

Vertex
A cone vertex only.

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

Geometry-driven
Accepts the current value and creates a geometry-driven constraint.

9.82 DMEXTRUDE command

Creates 3D solids or surfaces by extruding entities.

The DMEXTRUDE command has been unified with the EXTRUDE command. Use the EXTRUDE command instead.

9.83 DMFILLET command

Creates a smooth fillet between adjacent faces sharing a sharp edge.

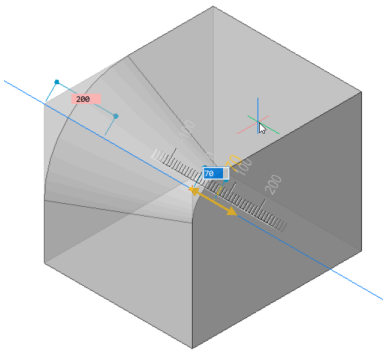
Shape Lite Pro Mechanical BIM

Icon:

9.83.1 Description

Creates constant or variable radius fillets between adjacent faces that are sharing a sharp edge. After selecting the edges, you may choose to specify a constant radius to fillet edges or one of the available advanced options.

The fillet is applied dynamically and the manipulator is displayed.



9.83.2 Options

Variable radius mode
Allows to create a variable fillet between adjacent faces that share a sharp edge(s).

Note: All advanced mode fillet procedures start by creating a constant radius fillet, which is then modified by specifying the first, second radius or intermediate points. Before specifying the final value of the radius you can press the TAB key to modify previously specified values.

Intermediate radius mode
Allows you to introduce different values for radius for intermediate points.

9.84 DMFIX3D command

Applies a fixed constraint to 3D entity.

Shape Lite Pro Mechanical BIM

Icon: 

9.84.1 Description

Applies a fixed constraint to a solid or to an edge or a face of a solid.

9.84.2 Options

Multiple
Allows to select multiple entities or subentities like:

- 3D entities: all faces and edges of the 3D solid or surface will be fixed.
- Faces: the selected faces will be fixed and cannot be moved or rotated.
- Edges: the selected edges will be fixed. The adjacent faces can be rotated, but not moved.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.85 DMGROUP command

Creates group features, which are named persistent collections of faces and edges of 3D solids and surfaces.

Shape Lite Pro Mechanical BIM

9.85.1 Description

Creates group features, which are named persistent collections of faces and edges of 3D solids and surfaces.

A group feature can be coupled with its description, which can be used to store manufacturing instructions or any other information with your model. Group features usually survive after modification of 3D solids/surfaces and can be accessed or created in Lisp scripts and BRX applications.

9.85.2 Options

New
Creates a new group.

Edit
Allows to edit an existing group feature by calling its name.
Use Shift to select entities from this selection set.

Dissolve
Allows to delete an existing group feature by calling its name.

Note: Group feature names are case sensitive.

9.86 DMMOVE command

Moves entities or sub-entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.86.1 Description

Moves solids, faces, edges or vertices of a solid or inserts using a vector.

You can select one or more entities. Press Enter to stop selecting. To specify the base and the endpoint you can choose one of the following methods:

- Specify a point by clicking the left mouse button.
- Type a value in the dynamic input field.

The selected geometry moves dynamically.

9.87 DMPARALLEL3D command

Applies a parallel constraint between subentities of 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.87.1 Method

Select two faces of a solid or two faces of different solids or surfaces or a face and an edge of two different solids between which the parallel constraint will be applied. Press the TAB key to select obscured geometry.

9.87.2 Options

WOrld

Allows you to select the WCS subentity as an argument.

BLock

Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.88 DMPATH3D command

Allows a point to move along a curve by specifying the curve's parameters.

Shape Lite Pro Mechanical BIM

Icon: 

9.88.1 Method

Select any curve entity (polyline, 3D polyline, spline or helix) as the path and a point/vertex from a 3D solid and specify the value of the curve parameter. Press the TAB key to select obscured geometry.

9.88.2 Options

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

Geometry-driven
Accepts the current value and creates a geometry-driven constraint.

9.89 DMPERPENDICULAR3D command

Applies a perpendicular constraint between two subentities of two 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.89.1 Description

Applies a perpendicular constraint between faces or edges of a solid, or of different solids or surfaces.

Select the two subentities of the 3D entities between which the perpendicular constraint will be applied. Press the TAB key to select obscured geometry.

9.89.2 Options

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.90 DMPUSHPULL command

Adds or removes volume from a solid by moving highlighted faces.

Shape Lite Pro Mechanical BIM

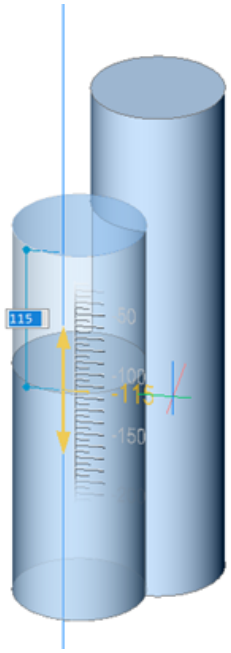
Icon: 

9.90.1 Description

Adds volumes to or removes volume from a solid using the cursor movement or direct distance input. You can use this command to edit any face of a 3D solid or a surface. You can select more faces or detected boundaries. Press Enter to complete the selection set.

Note: Plies of multiply compositions are selectable if the Select Faces of the SELECTIONMODES system variable is set.

The selected geometry moves dynamically. Using the Manipulator you can do one of the following:



- You can press the TAB key to select a different reference face. All faces which are parallel to the face being push/pulled are recognized, beginning with the first ones. Press and hold the SHIFT key when pressing the TAB key to cycle reference faces in reverse order. To select a reference face on a different solid, move the cursor over that face, then hit the TAB key. The selected face is highlighted.
- Click a point in the model.
- Type a value in the dynamic distance field.
- Move the cursor to adjust the value in the dynamic distance field. You can optionally zoom in/out to set the snap increment size of the ruler.

9.90.2 Options within the command

Enable subtract
Solids being push/pulled are subtracted from other 3D solids they intersect.

Disable subtract
Solids being push/pulled are not subtracted from other 3D solids they intersect.

Note: When the system variable DMPUSHPULLSUBTRACT = 1, the solid being pushpulled is subtracted from interfering solids. Pressing the Ctrl key allows you to override the DMPUSHPULLSUBTRACT system variable.

Note: If the Hotkey Assistant (HKA) is ON the Hotkey Assistant widget displays, indicating whether the command is in subtract mode or not.

9.91 DMRADIUS3D command

Applies a radius constraint to circular 3d entities or subentities.

Shape Lite Pro Mechanical BIM

Icon: 

9.91.1 Description

Applies a radius constraint to circular edges, cylindrical surfaces, spheres and tori by indicating the radius value or using geometry-driven.

9.91.2 Method

Select a cylindrical or sphere surface or a circular edge and specify the radius value or keep the geometry-driven. In case of selecting a torus surface you may constrain the rotation radius or/and the tube radius.

Press the TAB key to select obscured geometry.

9.91.3 Options

Rotation radius
Specify the rotation radius to be constrained.

Tube radius
Specify the tube radius to be constrained.

Geometry-driven
Accept the current value and create a geometry-driven constraint.

9.92 DMREPAIR command

Analyzes and repairs errors.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The DMREPAIR command has been replaced by the DMAUDIT command.

9.93 DMREVOLVE command

Creates 3D solids or surfaces by revolving 2D entities about an axis.

Shape Lite Pro Mechanical BIM

Icons: 

9.93.1 Description

Note: The *DMREVOLVE* command has been unified with the *REVOLVE* command. Use the *REVOLVE* command instead.

9.94 DMRIGIDSET3D command

Allows to define a set of entities or subentities as a rigid body.

Shape Lite Pro Mechanical BIM

Icon: 

9.94.1 Method

Select a set of edges, faces or 3D entities that will be linked as a rigid body.

All members of a rigid set are moved (DmMove command) and rotated (DmRotate command) together, while their relative positions within the rigid set are preserved.

Press the TAB key to select obscured geometry.

9.94.2 Options

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose, as an argument, the subentity of the coordinate system of the block.

9.95 DMROTATE command

Rotates the selected geometry around an axis.

This command is deprecated. Use the ROTATE3D command instead.

9.96 DMSELECT command

Selects edges and faces of 3D solids or surfaces based on their geometric properties.

Shape Lite Pro Mechanical BIM

9.96.1 Description

Allows to select a group of faces, which form a geometric feature, such as a protrusion, a depression, a blend, or a blend network. You can combine different options of this command, e.g. select all blends, which radius is less than given value. Select the faces or edges of which characteristics will be considered as pattern for the selection set.

9.96.2 Options

Selection
Allows to select a set of subentities:

sUbset
Only faces/edges in the subset will be selected.

Sample
The preselected set of faces/edges should be considered as samples, of which the geometric characteristics (length, area, radius) will be used as parameters for the selection.

sEed
The preselected set of faces /edges should be considered as seed elements for the selection of features such as blends, blend networks, protrusion or depressions.

Attribute

Radius
Selects faces/edges by radius.

Area
Selects faces by area.

Length
Selects edges by length.

Type
Selects faces/edges by their geometric type.

Exclude short fillets
Excludes short fillets from the selection, which might be useful when working with imported sheet metal parts.

Then choose between:

Current sample
To use the value of the sample face/edge.

Value
Type a value in the Command line.

> ***Note:*** : In case of a non-zero value, you may filter entities as Less/Less or eQual/Equal/Greater or eQual/Greater.

Relation

Coincident
Selects coincident faces/edges.

Parallel
Selects parallel faces/edges.

coAxial
Selects coaxial faces/edges.

Smooth
Selects the smooth side of a part.

Primitive

Face
Selects faces.

Edge
Selects edges.

Loop
selects edge loops.

> ***Note:*** The preselection will determine the result. It is necessary to select a set of connected faces. The command will then select all edges, which describe the border of the selection set; internal edges will be skipped. If the result contains multiple loops and you want only one, the preselection should contain a set

of connected faces one edge of the desired output loop. If none of the loops of the selection set contains the selected edge, the output will be empty.

edge Network
Finds an adjacent edge which has similar convexity for a given input edge.

feaTure

Protrusion
Selects protrusions.

Depression
Selects depressions.

Blend
Selects blends.

blend Network
Selects blend networks.

9.97 DMSELECTEDGES command

Selects the edges of faces and solids.

Shape Lite Pro Mechanical BIM

Icon: 

9.97.1 Method

Select the faces or solids from which all edges will be in the selection set.

Press and hold the CTRL key, then click the edges to be removed from the selection set.

9.98 DMSIMPLIFY command

Simplifies the geometry and topology of 3d solid entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.98.1 Description

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.

Note: It is recommended to always run this command on imported 3d solid geometry.

9.98.2 Options

Entire model
Simplifies the entire model.

SEttings
Allows to access the command settings.

Geometry
Define the geometrical settings.

Topology
Define the Topology settings.

ON
Switches topological/ geometrical simplification on.

OFF
Switches topological/ geometrical simplification off.

9.99 DMSIMPLIFYALL command

Simplifies the geometry in solids, within drawings inserted as external references.

Shape Lite Pro Mechanical BIM

9.99.1 Description

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.

Note: It is recommended to always run this command on imported drawings with inserted external references.

9.99.2 Options

Entire model
Simplifies the entire model.

SEttings
Allows to access the command settings.

Geometry
Define the geometrical settings.

ON
Switches topological/geometrical simplification on.

OFF
Switches topological/geometrical simplification off.

Topology
define the Topology settings.

Auto
Simplifies the detected geometry or topology.

9.100 DMSTITCH command

Converts a set of region and surface entities that bound a watertight area to a 3d solid.

Shape Lite Pro Mechanical BIM

Icon: 

9.100.1 Method

Select the surfaces to stitch and convert them in a 3D solids or surfaces.

Note: If DMREPORTPANEL system variable is set ON, the result is reported in the Report panel instead of the Command line.

9.100.2 Options

MOde
Allows to set the validation mode either manual or automatic.

Auto
Creates only valid 3d solid or surface entities without free or non-manifold edges.

SOlid
Creates invalid 3d solid entities.

SUrface
Creates invalid surfaces.

Note: Creating an invalid 3d solid entities could be useful for understanding the reason of a failure. The command reports problems (free or non-manifold edges) and highlights them. You can either accept the invalid 3d solid by pressing Enter or refuse it by pressing Cancel.

Tolerances
Type a value or accept Auto.

Note: The selected validation mode is stored and re-used for future calls of the command.

9.101 DMTANGENT3D command

Applies a tangent constraint between circular subentities of 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

9.101.1 Description

Applies a tangent constraint between a face and a curved surface of different 3D entities or a surface and WCS reference coordinate system or a surface and a reference coordinate system of a block reference.
Press the TAB key to select obscured geometry.

9.101.2 Options

WOrld
Allows you to select the WCS subentity as an argument.

BLock
Allows you to choose as an argument the subentity of the coordinate system of the block.

9.102 DMTHICKEN command

Creates 3D solids by thickening surfaces, their faces, faces of 3D solids, regions and wire entities.

Shape Lite Pro Mechanical BIM

Icon:

9.102.1 Description

Creates 3D solids by thickening surfaces, their faces, faces of 3D solids, regions and wire entities by specifying the thickness value.

The following wire entities are accepted: lines, polylines, circles, ellipses, arcs, helices, splines.

Note: If you thicken a multi-face surface of several adjacent faces of one 3D solid/surface, then the corresponding faces remain adjacent on the opposite side of the new 3D solid(s). This differs the DMTHICKEN command from DMEXTRUDE.

Note: When a wire entity is selected, a circle with the radius of the specified thickness value is swept along the wire.

9.102.2 Options

SIngle side
Adds the thickening in the specified direction.

Both sides
Adds thickening in both directions.

9.103 DMTHREAD command

Creates threads on cylindrical faces, which are represented according to drafting standards in drawing views.

Shape Lite Pro Mechanical BIM

Icon:

9.103.1 Description

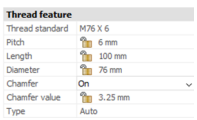
Creates a thread feature in the model that is represented according to drafting standards in drawing views or section view created with VIEWBASE and VIEWSECTION commands.

Note: This command can be entered transparently during commands with ‘dmthread.

9.103.2 Method

Selecting a cylindrical face and specifying the pitch and length values a thread feature will be created, that will be represented according to drawing standards in drawing views. With VIEWUPDATE command the changes will be applied in the drawing views.

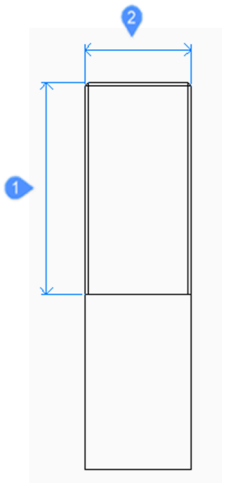
Thread features display in the Mechanical browser , where you can edit the thread parameters.

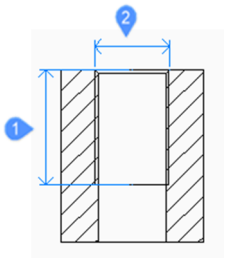


Note: Male and female thread types are supported.

Note: This command operates only in Model Space.

Below you can see the result of VIEWBASE and VIEWSECTION commands.





1 Length

2 Diameter

9.104 DMTWIST command

Twists a portion of an entity around an axis.

Shape Lite Pro Mechanical BIM

Icon:

9.104.1 Description

Modifies 3D solids, surfaces or regions by twisting a portion defined by two points around an axis.

Note: You can twist for an angle smaller than 360 degrees.

9.104.2 Method

It is possible to specify the continuity between the deformed and the fixed portion of the entities. You can choose between sharp, smooth and in-between.

9.104.3 Options within the command

Start point of twisting axis
Specify a point where the twisting deformation will start.

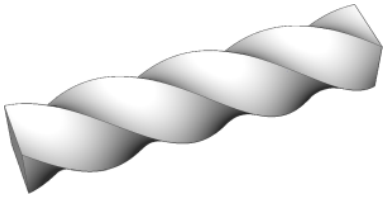
End point of twisting axis
Specify a point where the twisting deformation will end.

Start point of twisting
Specify a point where the twisting will begin.

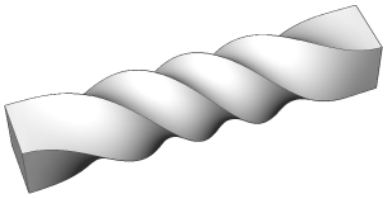
Twist angle
Specify an angle dynamically or type a value.

Continuity
Allows to control whether the transition between the deformed and non-deformed part is sharp or smooth.

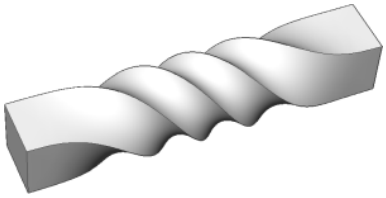
G0
No continuity.



G1
Tangent faces.



G2
Curvature continuity.



9.105 DMUPDATE command

Forces 3d constraints to update.

Shape Lite Pro Mechanical BIM

Icon: 

9.105.1 Description

Updates all 3d constraints applied to all 3D entities existing in the drawing.

9.106 DOME command

This command is obsolete and exists for backward compatibility only. Use instead the AI_DOME command.

Shape Lite Pro Mechanical BIM

Icon:

9.107 DONUT command

Creates a closed polyline in the shape of a donut.

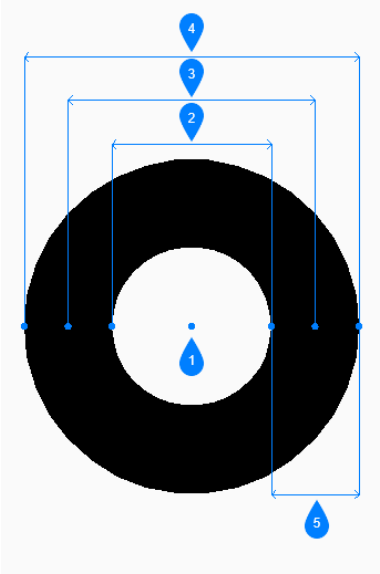
Shape Lite Pro Mechanical BIM

Icon:

Alias: DO, DOUGHNUT

9.107.1 Description

Creates a closed polyline in the shape of a donut from a combination of options including the inside diameter, outside diameter, center and width.



1 Center

2 Inside diameter

3 Diameter

4 Outside diameter

5 Width

9.107.2 Methods to create a donut

This command has 4 methods to begin creating a donut:

- Inside diameter of a donut
- 2 Point
- 3 Point
- Tangent Tangent Radius

Inside diameter of donut
Begin creating a donut by specifying its inside diameter then:

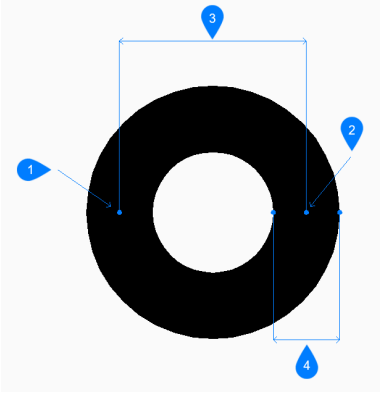
Outside diameter of donut
Specify the outside diameter of the donut.

Center of donut
Specify the center of the donut to place it in the drawing. You can continue placing donuts of the same size until you press Enter to end the command.

2 Point
Begin creating a donut by specifying its width then:

First point on diameter
Specify a point on the diameter.

Second point on diameter
Specify the second point on the diameter.



1 Point 1

2 Point 2

3 Diameter

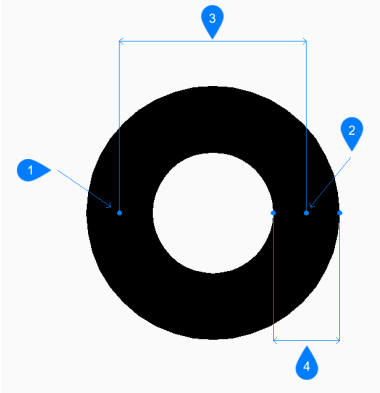
4 Width

3 Point
Begin creating a donut by specifying its width then:

First point on donut
Specify a point on the circumference of an imaginary circle that runs through the center of its width.

Second point
Specify the second point on the circumference of the imaginary circle.

Third point
Specify the third point on the circumference of the imaginary circle.



1 Point 1

2 Point 2

3 Point 3

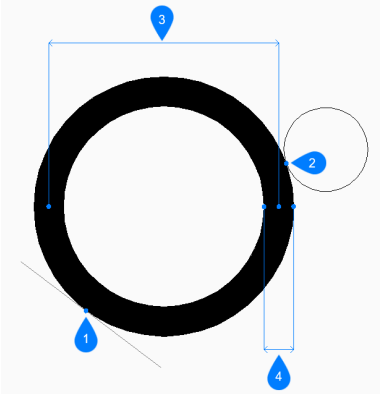
4 Width

Tangent Tangent Radius
Begin creating a donut by selecting a tangent point on the first entity then:

Specify point on object for second tangent on donut
Select a tangent point on the second entity.

Width of donut
Specify the width of the donut.

Diameter of donut
Specify a valid diameter for the donut. If you specify a diameter that is not possible with the selected tangents, you are prompted to specify the tangents and diameter again.



1 Tangent point 1

2 Tangent point 2

3 Diameter

4 Width

9.108 DRAG command

Moves 3D solids.

Shape Lite Pro Mechanical BIM

Icon:

9.108.1 Description

Moves 3D solids perpendicular to a selected face by dragging that face. Optionally preserves connections to other solids.

9.108.2 Method

After selecting one or more planar face(s):

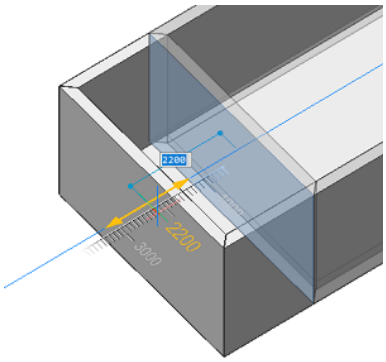
- Type a value in the dynamic distance field.

Note: Toggle Dynamic Input ON from the Status bar.

- Type a value in the Command line.
- Move the cursor to adjust the value in the dynamic distance field.

Note: The selected face(s) move(s) dynamically. The manipulator displays the distance from the current position of the selected face in the dynamic entry field.

Note: Select a different reference face by repeatedly pressing TAB key to cycle all parallel faces of the solid. All parallel faces under the cursor are recognized.



9.108.3 Options within the command

Disable connectivity mode
The connectivity is not kept.

Note: This mode is set as default.

Note: Press Ctrl key to toggle the enable connectivity mode.

Enable connectivity mode
The connectivity is kept.

Note: Press Ctrl key to toggle the disable connectivity mode.

Optionally the Hot Key Assistant can be toggled On, from the status bar, to display the current used connectivity mode.

Note: If the Hot Key Assistant is not displayed, right click on the toggle button and check its configuration.

9.109 DRAGMODE command

Controls the appearance of objects.

Shape Lite Pro Mechanical BIM

9.109.1 Description

Controls the appearance of the objects while being dragged. The command is no longer necessary and is kept only for compatibility.

9.109.2 Options within the command

ON
Turns on DRAGMODE.

Note: Dragged objects are always shown.

OFf
Turns off DRAGMODE.

Note: Dragged objects and the dragline are never shown on the screen.

Auto
Turns on DRAGMODE.

Note: Dragged objects are always shown.

9.110 DRAWORDER command

Changes the display order of overlapping entities.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: DR

9.110.1 Description

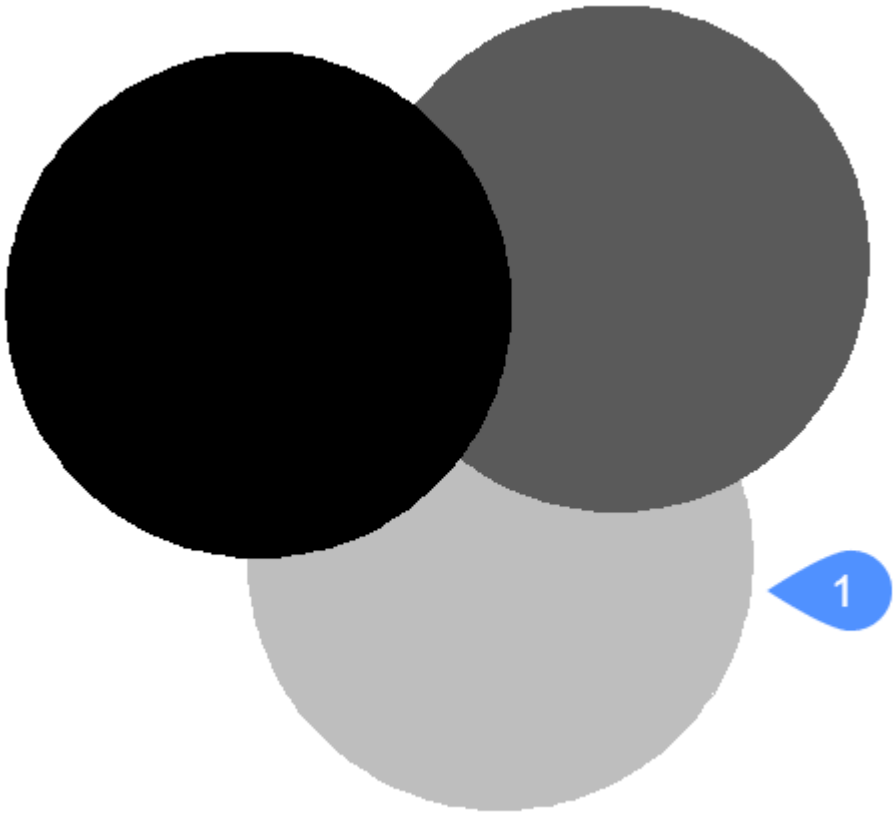
This command is used to display entities over the top of other entities that would otherwise overlap or hide them. This command is meant to work with overlapping objects. While this command works with non-overlapping entities, it has no effect on them.

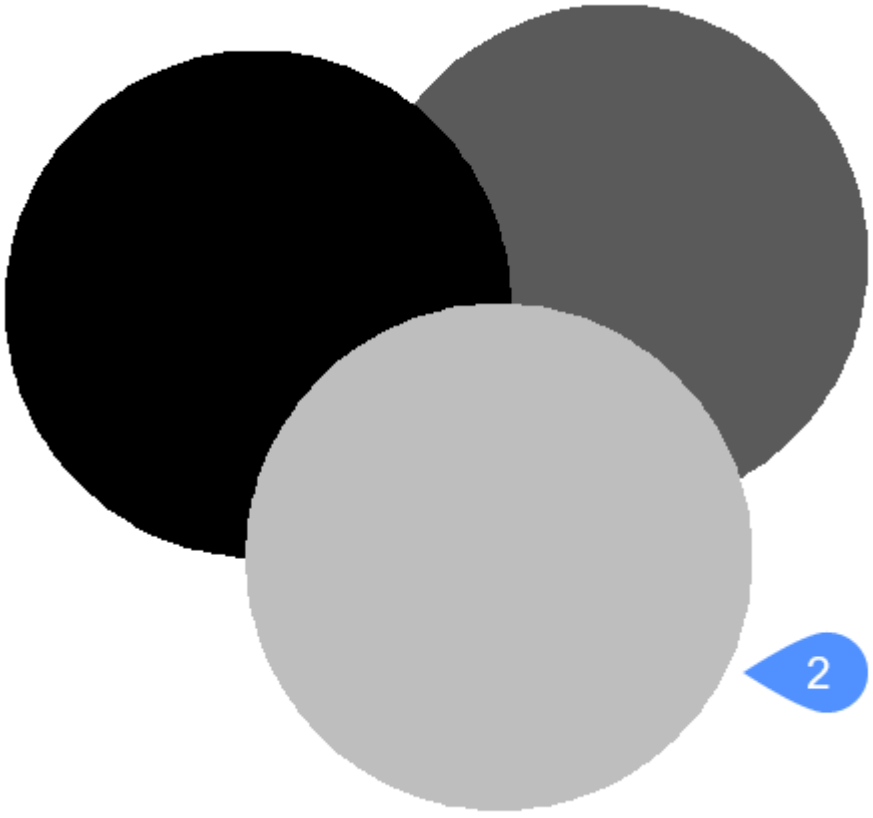
Note: Hatches and text entities in BricsCAD already have handy commands controlling their display order:HatchToBack places all hatching below all other entities, andTextToFrontdisplays all text above all entities.

9.110.2 Method

This command offers 5 methods to change the draw order of overlapping entities:

- Above
- Under
- Bring to front
- Bring to back
- Clear all orders


1


2

9.110.3 Options within the command

Above
Moves the visual display of selected entities above the other entities; this option does not necessarily move them to the top (front) of display order.

Under
Moves the visual display of selected entities under the other entities; this option does not necessarily move them to the bottom (back) of the display order.

Clear all orders
Clears the assigned display orders, with entities displayed as they were originally created.

Send to Back
Moves the selected entities' display order below all other overlapping entities (1).

Bring to Front
Moves the selected entities' display order above all other overlapping entities (2).

9.111 DRAWORDERBYLAYER command

Opens the Open layer list file dialog box.

Shape Lite Pro Mechanical BIM

9.111.1 Description

Opens the Open layer list file dialog box to select an lst file that controls the display order of entities based on their layers.

9.112 DSETTINGS command

Opens the Settings dialog box.

Shape Lite Pro Mechanical BIM

Alias: DDRMODES, RM

9.112.1 Description

Open the Settings dialog box to view and modify system variables. Most, but not all, system variables are available in the Settings dialog box. You can modify all system variables using the SETVAR command.

9.113 DTEXT command

Creates a single-line text entity.

Shape Lite Pro Mechanical BIM

Alias: DT

See the TEXT command.

9.114 DUMPSTATE command

9.115 DVIEW command

Changes the 3D viewpoint interactively and turns on perspective mode (short for "dynamic view").

Shape Lite Pro Mechanical BIM

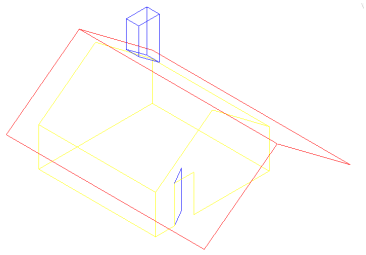
Icon:

Alias: DV

Note: The PERSPECTIVE system variable toggles the perspective property of a view.

9.115.1 Method

Select entities or use DVIEWBLOCK that displays the below built-in 3d block .



9.115.2 Options

rotate CAmera
Specifies the camera (eye) location.

rotate TArget
Specifies the target (look at) location.

Twist view
Sets the twist (view rotation about Z axis) angle.

Distance to target
Specifies the camera to target distance.

Note: This option turns on perspective mode.

x y z POints
Specifies the camera and target location.

CLipping
Sets the clipping distances, which cut off the front and back parts of the model. The front and back clipping planes are positioning perpendicular to the imaginary line between the camera and target.

Note: The clipping planes can be set using the slider bar.

Front clipping
Sets the front clipping plane which obscures objects located between it and the camera.

frONt clipping on
Switch on front clipping at the current clipping distance.

front clipping OFf
Switch off front clipping.

front clipping at Eye
Sets the front clipping plane at the camera location.

Back clipping
Sets the back clipping plane which obscures objects behind it:

back clipping ON
Switch on back clipping at the current clipping distance

back clipping OFf
Switch off back clipping

clipping Off
Removes the clipping planes.

perspective Off
Turns off perspective mode. Use the Distance option to turn perspective mode on.

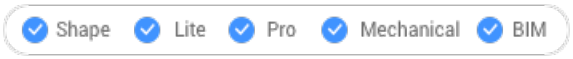
Hide
Removes hidden lines from the model.

PAn
Interactively pans around the drawing.

Zoom
Zooms in and out interactively (enter a number to change the zoom level).

9.116 DWFOUT command

Saves the drawing in DWF and other formats.



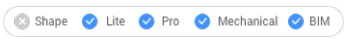
9.116.1 Description

Saves the drawing in DWF and other formats (short for "drawing Web format output"). DWFOUT is an alias for theEXPORTcommand.

Note: DWF files can be viewed with the DesignReview software, available free from www.autodesk.com/designreview.

9.117 DWGCODEPAGE command

Changes the language code for text in drawings.



9.117.1 Description

Text on computers is defined by the Unicode system, which determines the alphabet used to display text, such as English or Turkish. DWGCODEPAGE changes the code number that specifies the alphabet. This command does not affect the language displayed by the user interface.

9.118 DWGCOMPARE command

Compares a selected drawing with the current drawing.

Shape Lite Pro Mechanical BIM

Icon:

9.118.1 Description

Compares a selected drawing with the current drawing, showing the differences between the two.

9.118.2 Method

The Open drawing file dialog box is displayed. Select a drawing with which to compare the current drawing.

The command looks for differences in the same entities between the current drawing and the drawing opened for comparison.

BricsCAD reports in the Command line if any differences have been found.

When the program finds differences, entities are shown in these colors:

- Entities added - green (color is set by variable CMPCLRNEW).
- Entities removed - red (set by variable CMPCLRMISS).
- Entities changed in the original drawing - gray (by variable CMPCLRMOD1).
- Entities changed in the comparison drawing - yellow (variable CMPCLRMOD2).

Note: The colors are set by the Settings dialog box. Search for variables that begin with CMPCLR.

As well, you can see a list of differences in the Structurepanel. Open it with theSTRUCTUREPANELcommand and ensure the Default CST configuration file is loaded. Expand theComparisonnode in the Structure panel.



Note: The ENDCOMPARE command unloads the comparison drawing.

9.118.3 Options within the command

Limit

Sets the limit on how many entities to compare in the drawing.

9.119 DWGPROPS command

Opens the Drawing properties dialog box.

Shape Lite Pro Mechanical BIM

9.119.1 Description

Opens the Drawing properties dialog box to view and modify drawing properties.

9.120 DXFIN command

Opens the Load DXF file dialog box.

Shape Lite Pro Mechanical BIM

9.120.1 Description

Opens the Load DXF file dialog box to select a dxf file to import into the current drawing.

9.121 DXFOUT command

Opens the Select DXF file dialog box.

Shape Lite Pro Mechanical BIM

Alias: DX

9.121.1 Description

Opens the Select DXF file dialog box to save data from the current drawing to a dxf file.

9.121.2 Method

After choosing **Save** in the dialog box, you are prompted at the Command line:

Enter decimal places of accuracy (0-16)
Specify the accuracy of real numbers in the dxf file. Some CNC (computerized numerical control) machines require 4 decimal places in dxf files.
Alternate options: [Entities/Binary/Version]

Entities
Specifies the entities to export. If unspecified, all entities are exported.

Binary
Specifies binary format for the dxf file. If unspecified, the dxf file is exported to ASCII (text) format.

Version
Specifies the DXF version number. If unspecified, the dxf file is exported to the most current version. Exporting to an older version can cause some entities or properties to be lost.

10. E

10.1 EATTEDIT command

Edits attributes contained in a single block.

Shape Lite Pro Mechanical BIM

Icon:

Alias: ATE

10.1.1 Description

Edits the values and most properties of all attributes contained in a single block, through the Attribute Editor dialog box.

10.2 EDGESURF command

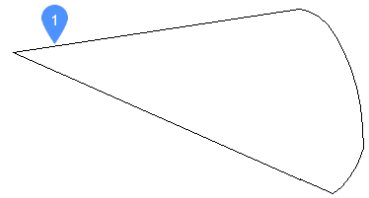
Creates 3D polygon mesh patches between four linear entities (short for “edge-defined surface”).

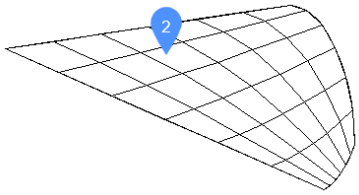
Shape Lite Pro Mechanical BIM

Icon:

10.2.1 Method

Create an edge surface (2) by selecting each four connected linear entities (1) for it.





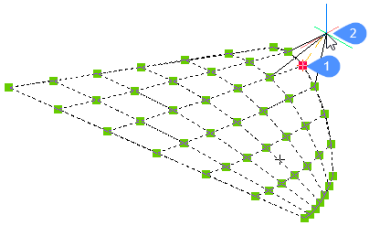
The four entities must be:

- Linear: open entities, such as lines, arcs, open splines, and open polylines.
- Connected: touching or overlapping, so that they form a closed area.

10.2.2 Grips Editing

Edge meshes can be edited directly through grips:

Drag a grip (1) to stretch (2) the adjacent faces.



10.3 EDITEDATA command

Edits extended entity data.

Shape Lite Pro Mechanical BIM

Icon:

10.3.1 Method

Select an entity to display the Create or Edit Entity Data dialog box.

10.4 ELEV command

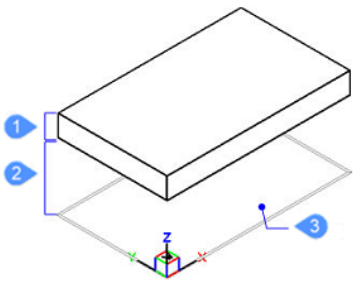
Changes the elevation and thickness of selected objects.

Shape Lite Pro Mechanical BIM

Icon:

10.4.1 Description

Changes the elevation (as measured from the x,y-plane in the z direction) and thickness (as measured from the elevation) of selected entities.



1 Thickness
2 Elevation
3 x,y-plane

10.4.2 Options within the command

New current elevation

Specifies the elevation above or below the x,y-plane. For a negative elevation, a negative value can be entered.

Note: The elevation is reported on the status bar as the z coordinate.

New current thickness
Specifies the thickness. Enter a negative value for thickness that are drawn downwards.

Note: The thickness begins at the elevation.

Note: The thickness is the extrusion distance, in the z direction, of non-3D entities, such as: points, lines, polylines, arcs and circles. When thickness is not 0:

- Points become vertical lines.
- Lines and open polylines become vertical planes.
- Arcs become curved surfaces.
- Circles become open-ended cylinders.
- Closed polylines become open-ended tubes.

10.5 ELLIPSE command

Creates an ellipse or elliptical arc.

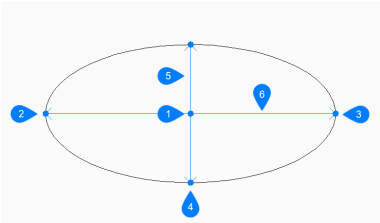
Shape Lite Pro Mechanical BIM

Icon:

Alias: EL

10.5.1 Description

Create an ellipse or elliptical arc from a combination of options including the center, axis endpoints, axis radius, rotation angle, and included angle.



1 Center
2 First end of ellipse axis
3 Second end of ellipse axis
4 End of other axis
5 Major axis
6 Minor axis

10.5.2 Methods to create an ellipse

This command has 3 methods to begin creating an ellipse:

- First end of ellipse axis
- Arc

- Center

If the PELLIPSE variable is set to 1, the Arc option is not available.

First end of ellipse axis
Begin creating an ellipse by specifying a point on the ellipse's first axis then:

Second end of axis
Specify the end point of the ellipse's first axis.

Set other axis
Specify the radius of the other axis.
Additional option: [Rotation]

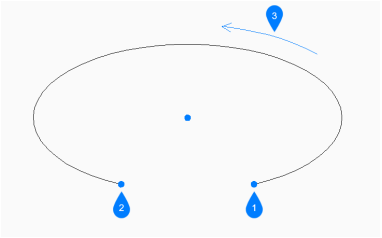
Arc
Begin creating an elliptical arc by specifying a point on the ellipse's first axis then:
Additional options: [Center]

Second end of axis
Specify the end point of the ellipse's first axis.

Set other axis
Specify the radius of the other axis.
Additional options: [Rotation]

Set start angle of arc
Specify the start angle to define the arc's starting point. The angle is measured counter-clockwise from the positive x-axis.
Additional option: [Parameter]

End angle
Specify the end angle to define the arc's endpoint.
Additional options: [Parameter/Included angle]



1 Start angle

2 End angle

3 Included angle

Center
Begin creating an ellipse by specifying its center point then:

Endpoint of axis
Specify an endpoint of the ellipse's first axis, measured from the center.

Set other axis
Specify an endpoint of the other axis, measured from the center.
Additional option: [Rotation]

10.5.3 Additional options within the ELLIPSE command

After you begin creating an ellipse, the following options may be available:

Included angle
Specify the included angle, measured from the starting point.

Parameter
Specify values for the parametric formula that defines the elliptical arc:
p(b) = c + a * cos(u) = b * sin(u)
where:

- a = major axis.
- b = minor axis.
- c = center of the ellipse.
- u = point along the arc.

Rotation
Specify the angle of the ellipse around the major axis. The angle can vary between 0 and 89.9. An angle of 0 degrees draws a circle. An angle of 89.9 degrees draws a very thin ellipse.

10.6 ENABLEASSOCVIEWS command

Adds the service data to the drawing and its Xrefs.

Shape Lite Pro Mechanical BIM

10.6.1 Description

It affects only associativity of the section result.

Note: If this command is not called and both GENERATEASSOCVIEWS and GENERATEASSOCATTRS are turned off, the result will be visually the same, but not associative.

10.6.2 Method

After the data is added, BIMSECTIONUPDATE and VIEWBASE commands cause no extra modifications in the referenced drawings.

Note: If the drawing contains a Xref file which requires modification, a message will appear to confirm modification of the Xref file.

10.6.3 Options within the command

Proceed
Toggles between Yes or No.

10.7 ENDCOMPARE command

Exits a drawing that was in comparison mode.

Shape Lite Pro Mechanical BIM

Icon: 

10.7.1 Description

Exits a drawing that was in comparison mode begun by the DWGCOMPARE command.

10.8 ENDPOINT command

Toggles the Endpoint entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

10.8.1 Description

Toggles the Endpoint entity snap to enable or disable snapping to endpoint. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

10.9 ERASE command

Erases entities from the drawing; erases the faces and edges of 3D solids.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: DELETE, E

10.9.1 Method

Select entities, edges or faces to be removed.

Select the edges of an opening in a surface to delete the opening.

Select the edge(s) between coplanar faces of a 3D solid to remove the edges. Alternatively use DMSIMPLIFY command to remove redundant edges.

Select the interior faces of an opening in a 3D solid to remove the opening.

10.10 ETRANSMIT command

Opens the Drawing explorer dialog box with **Dependencies** selected.

Shape Lite Pro Mechanical BIM

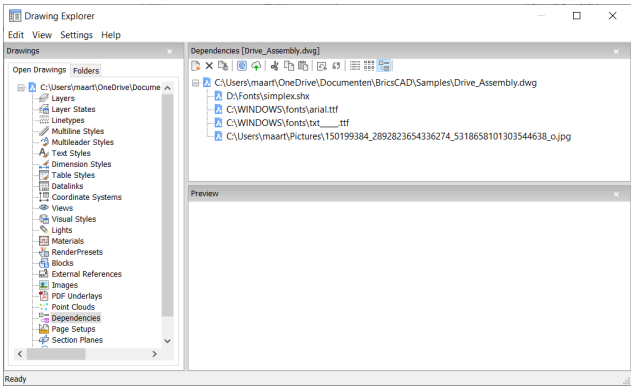
Icon: 

10.10.1 Description

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.

10.10.2 Methods

Opens theDrawing explorer dialog boxwithDependenciescategory selected to view and modify all dependenciesinthe current drawing.



10.10.3 Context Menu Options

eTransmit

Starts the eTransmit procedure. The eTransmit dialog box opens, which lists the files to include.

Upload to Bricsys 24/7

Opens the Upload to Bricsys 24/7 dialog box to allow you upload the drawing and its dependencies onto Bricsys 24/7.

10.10.4 Options within the command

New

Add another drawing to the eTransmit procedure. The File dialog box opens to let you choose another drawing.

Delete

Deletes Dependency definitions from the drawing.

10.11 -ETRANSMIT command

Creates a package ZIP file at the Command line.

Shape Lite Pro Mechanical BIM

10.11.1 Description

Creates a package ZIP file at the Command line, consisting the current drawing file and all its dependent files.

10.11.2 Options within the command

Create

Creates a package ZIP file that contains the drawing and its support files.

Yes

Automatically creates a package ZIP file that has the same name as the drawing. On subsequent usage of the command, the package will be saved in the previously used folder path.

No
Opens a File dialog box to let you select a folder and specify a file name.

Settings
Prompts you for changes to the settings in the Command line.

Save format
Specifies the format of the drawing files in the eTransmit package.

Output format
Specifies whether to create a ZIP file or copy the files to the output folder.

FOlder structure
Specifies how files are organized into folders.

FIle list
Determines which files are included in the transmittal package.

10.12 EXECUTETOOL command

Repeats the last tool used from the Tool palettes panel.

Shape Lite Pro Mechanical BIM

10.13 EXPBLOCKS command

Opens the **Blocks** category of the Drawing explorer dialog box for inserting, deleting, and creating blocks.

Shape Lite Pro Mechanical BIM

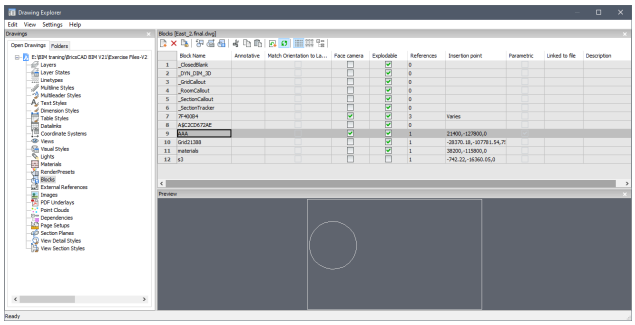
Icon:

Alias: XB

Note: Non-Uniformly scaled blocks can be exploded only if EXPLMODE system variable is ON.

10.13.1 Description

Displays the Blocks category of the Drawing explorer dialog box:



10.13.2 Context Menu Options

New
Creates new blocks; dismisses the Drawing explorer dialog box, and the starts the -BLOCK command. See BLOCK command.

Delete
Deletes the block definition; available only for those block definitions not inserted in the drawing.

Insert
Inserts the selected block; dismisses the Drawing explorer dialog box, and the starts the -INSERT command. See INSERT command.

Insert External
Inserts DWG files as blocks; starts the INSERT command, and then displays the INSERT BLOCK dialog box. See INSERT command.

Save Block
Exports the selected block as a DWG file; displays the Save Block dialog box.
Right-click a block to display the shortcut menu, and then choose Options.

Add to current Toolpalette
Adds the block to the current Tool Palettes.

Options
Displays the Drawing explorer options dialog box to allow you to align, set the scale or the rotation of the inserted blocks.

Align with view
When checked, inserts the block to face the camera (blocks are rotated about their local Z-axis to face the camera).

Note: Parametric blocks cannot be cut, copied ore renamed.

Rename
Renames the selected block.

Select All
Selects all blocks definitions.

Invert Selection
Inverts the selected blocks. For instance, if you have one block selected, then this option deselects it, and then selects all other blocks.

10.13.3 Options within the command

Block Name
Specifies the name of the block.

Annotative
When checked, sets the Annotative property.

Match Orientation to Layout
When checked, sets the Match Orientation property of annotative blocks.

Face Camera
When checked, inserts the block to face the camera (blocks are rotated about their local Z-axis to face the camera). See the CAMERA command.

Explodable
When checked, blocks of this definition can be exploded into its component parts by the EXPLODE command. Non-Uniformly scaled blocks can be exploded only if EXPLMODE system variable is ON.

References
Reports the number of insertions of the block in the current drawing; number is set by the program, and so you cannot change it.

Insertion Point
Reports the insertion point of the block; reports "Varies" when the block has been inserted in the drawing more than once.

Parametric
Indicates if the block is parametric or not.

Linked to file
Shows file path for External Parametric blocks

Description
Optional description of the block.

10.14 EXPFOLDERS command

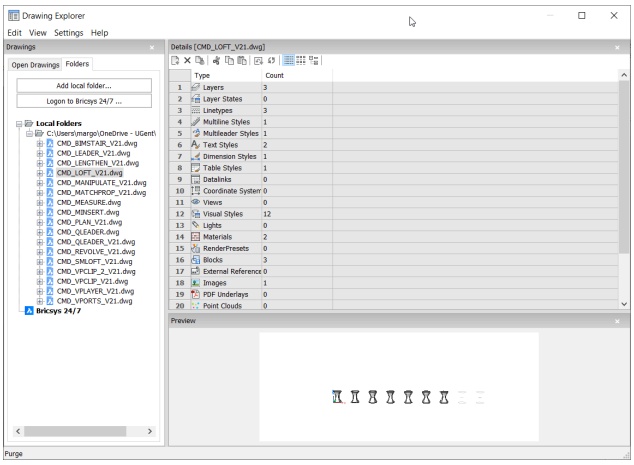
Opens the Drawing explorer dialog box with the **Folders** tab selected.

Shape Lite Pro Mechanical BIM

Icon:

10.14.1 Description

Opens the Drawing explorer dialog box on the Folders tab.



Note: The categories (such as Layers, Blocks, etc.) can be copy-pasted from Folders tab to Open Drawings tab.

10.14.2 Context Menu Options

Select All
Selects all items.

Invert selection
Deselects the current selection and vice versa.

10.14.3 Options within the command

Add local folder
Opens a Choose a Folder dialog box.

Logon to Bricsys 24/7
Displays the Bricsys 24/7 dialog box to log on.

10.15 EXPIMAGES command

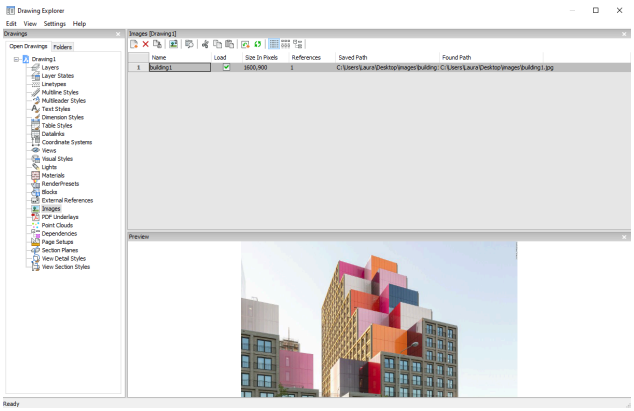
Opens the Drawing explorer dialog box with **Images** selected.

Shape Lite Pro Mechanical BIM

Icon:

10.15.1 Description

Displays the Images category of the Drawing explorer dialog box:



Supports the following formats:

BMP: Windows bitmap

ECW: Enhanced Compressed Wavelet format

GIF: CompuServe graphic image format

JPG, JPEG, and JPEG2000 (JP2, j2k) : Joint Photographic Experts Group (common among digital cameras)

MTI: Multi-resolution Tiled Image (Bricsys raster format for large images)

PCX: PC Paintbrush

PNG: Portable network graphics

SID: MrSID is an acronym for Multi-resolution Seamless Image Database

TGA: Targa

TIF, TIFF: Tagged Image File Format

10.15.2 Context menu options

New
Attaches raster image files to the drawing. dismisses the Drawing explorer dialog box, and the starts the IMAGEATTACH command. See IMAGEATTACH command.

Delete
Removes the image from the drawing. You may need to use the REGEN command to update the screen.

Insert
Inserts more images in the drawing, like the New button. Displays the Attach Raster Image dialog box. See IMAGEATTACH command.

Run extended search for missing attachments
Runs an extended search for missing attachments.

Clear cash
Clears the folder where temporary image cache files are stored.

Relative path
Shows the Saved Path as a relative path.

Absolute path
Shows the Saved Path as an absolute path of the image located in the drawing folder.

Filename as a path
Replaces the saved path with the file name.

10.15.3 Options within the command

Name
Specifies the name of the image. This usually is the image's file name.

Load
Toggles whether the image is displayed in the drawing:
On: displays the image in the drawing
Off: displays only the image's frame, which is turned off by the IMAGEFRAME command

Size in Pixels
Reports the size of the image in pixels, horizontally and vertically.

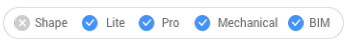
References
Reports the number of times the image occurs in the drawing.

Saved Path
Reports the path to the image file when first loaded. When the path cannot be found, click the Browse button which will display the Choose a File dialog box and locate the missing image file.

Found Path
Reports the current path to the image file. This path should match the Saved Path in most cases.

10.16 EXPLAYERS command

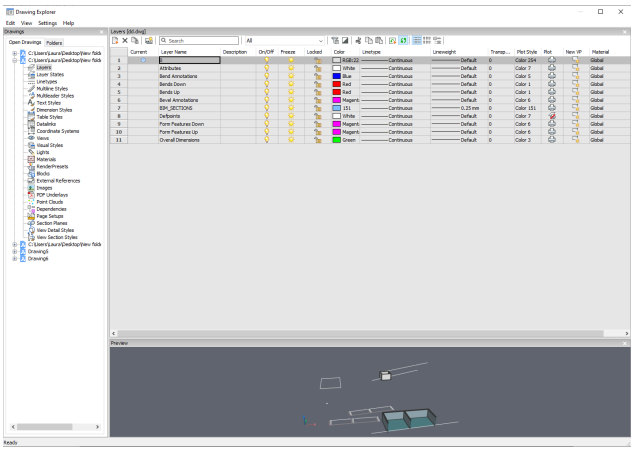
Opens the Drawing explorer dialog box with **Layers** selected.



Icon:

10.16.1 Description

Opens theDrawing explorer dialog boxwithLayerscategory selected to view and modifylayers in the current drawing.



Note: When you open the Drawing explorer - Layers dialog when in paper space, additional columns are displayed: VP Freeze, VP Color, VP Linetype, VP Lineweight, and VP Plot Style, where VP is short for viewport.

Note: By default, these columns match the corresponding model space settings, but viewport layer properties can be set differently for each layout and for each paper space viewport. If the SHOWLAYERUSAGE variable is on, then the**Layer Usage**icons indicates whether viewport settings for the current layout and paper space viewport are different from model space settings:

- : current layer with viewport overrides.
- : layer with viewport overrides.
- : empty layer with viewport overrides.

Note: If no paper space viewport is active, then the settings apply to entities in the current layout. If a viewport is active, then viewport settings override the BYLAYER properties in the current viewport.

Note: The VP Freeze setting allows you to control the visibility of model space entities in the current viewport. The New setting controls whether a layer is either Frozen or Thawed in the new paper space viewports.

10.16.2 Context Menu Options

New

Create a new layer definition into the drawing. The new layer that is created copies the properties of the currently selected layer.

Delete

Deletes layer definitions from the drawing. The following layer definitions cannot be deleted:

- Layer 0

- Defpoints
- Layers in use
- The current layer

Merge to…
Opens the Merge Layers dialog box. Here you can choose a target layer where the selected layers will be merged to.
The following layer definitions cannot be merged:

- Layer 0
- Defpoints
- The current layer

Note: you can only merge layers which are in use. If there aren't any entities assigned to a layer, the merge option will delete the selected layer.

Remove from group
Removes layers from the group they are assigned to.

Create Group Filter From Selection
Creates a group with the selected layers assigned to it.

Rename
Renames the selected layer.
Following layers can not be renamed:

- Layer 0
- Defpoints

Select All
Selects all layers.

Invert selection
Deselects the current selection and vice versa.

Remove Viewport Overrides
Removes overrides from properties that were set differently for viewports.

From Selected Layers

- For current viewport: removes overrides from the current viewport
- For all viewports: removes overrides from all viewports in the drawing

From All layers
Removes the overrides form all layers in the drawing for either the current viewport or all viewports.

Isolate Selected Layers
Dims the entities on all which are not selected. Use the LAYUNISO command to unisolate layers.

10.16.3 Options within the command

Filters
You can think of layer filters as "layer groups," groups of layers that you need to show or turn on-or-off all at once.

New Properties Filter
Creates a new properties filter. Displays the Layer Property Filter dialog box.

New Group Filter
Creates a new group filter.

Layers state
Toggles the layers state of all the layers in the selected group.

- On: toggles the layers on.
- Off: toggles the layers off.
- Thaw: thaws the layers.
- Freeze: freezes the layers.
- Lock: locks the layers.
- Unlock: unlocks the layers.

Isolate layers
Isolates the layers in the selected group.

- Current viewport: removes overrides from the current viewport.
- All viewports: removes overrides from all viewports in the drawing.

Add layers from selection
In the selected layer group filter, the layers from the selected objects in the drawing are added.

Replace with layers from selection
The layers from the selected group filter are replaced with the layers of the selected objects in the drawing.

Add layers from panel selection
Adds the layers which are selected in the layers panel.

Paste with Layers
Pastes the copied or cut group, including its layers.

Convert to Group
Converts a property filter to a group filter.

10.17 EXPLODE command

Reduces complex entities to simpler ones.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: X

10.17.1 Description

Reduces complex entities to simpler ones, such as blocks into constituent parts, and polylines into lines and arcs.

Note: Entities on frozen and locked layers are not exploded.

10.17.2 Method

The result depends on the entities you selected:

- Polylines become lines and arcs. Width is lost.
- Regions become lines, arcs and splines.
- Dimensions becomes lines, text, and solids (arrowheads).
- Blocks become individual entities and nested blocks. You may need to use Explode a second time.
- ***Note:*** Only blocks that have Explodable property set can be exploded. Non-Uniformly scaled blocks can be exploded only if EXPLMODE=ON.
- ByBlock colors and linetypes may change.
- Attributes revert to attribute definitions.
- 3D surface entities become 3D faces.
- Meshes become 3D faces.

Note: Simple entities, such as line and circles, cannot be exploded.

10.18 EXPLORER command

Opens the Drawing explorer dialog box.

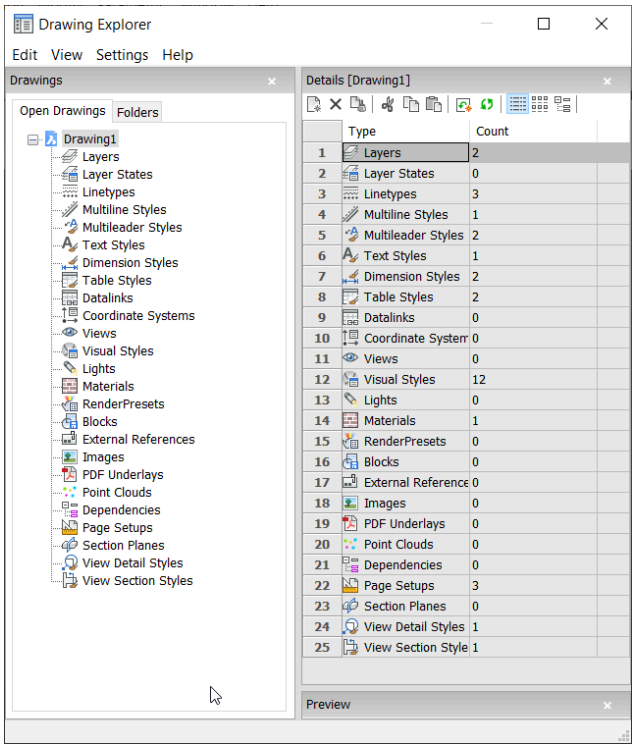
Shape Lite Pro Mechanical BIM

Icon: 

10.18.1 Description

Opensthe Drawing explorer dialog box on the last used tab or categoryto manage definitions and reference content that is used in the drawing.

Note: Press F1 to access help. To close, Press X.



10.18.2 Options within the menu bar

View

Check boxes

Toggles the visibility of specific items in the Drawing Explorer dialog box on and off.

Tree/Icons Details

Selects a visual style to list all definitions of the selected drawing.

Regen

Regenerates all definitions.

Regen at each change

Regenerates definitions each time a change has been made.

Hide xref symbols

Toggles the visibility of xref symbols on and off.

Settings

Restore default layout

Restores the modified layout of the Drawing explorer to the default layout.

Options

Displays the Drawing explorer options dialog box. Options regarding inserting a block can be defined.

Columns

Columns can be switched on and off and repositioned by drag-and-drop:

- Move: drag it by its header to a new location.
- Change width: drag the separator line back and forth.

- Restore width: double-click the separator line.

Show all columns
Displays all columns including the ones that were hidden before.

Restore Column positions
Returns columns to their original position.

10.18.3 Context Menu Options

New
Creates a new item with a generic name, such as NewLayer1 or NewStyle1.

Delete
Deletes the selected item from the drawing. When the selected item is in use, for example a layer with entities, several options may be available.

Delete
Deletes the selected layer and all entities on it.

Change
Changes the characteristic of all entities on the item to delete to another characteristic item. Afterward, the item will be deleted.

Cancel
Cancels the deleting operation so that no changes are made.

Note: Some automatically generated definitions cannot be deleted.

Cut
Cuts the selected item to the clipboard.

Copy
Copies the selected item to the clipboard.

Note: The options Cut and Copy can only be used when additionalitems were already made and cannot be usedon automatically generated items.

Paste
Pastes the copied item from the clipboard. When the same name item already exits, several options may be available:

Purge
Removes unused named entities from drawings, such as unused layers and linetypes.

Copy and replace
Replaces the original item in the current drawing with the item that is being copied.

Don’t copy
The item will not be pasted in the current drawing.

Copy, but keep both
The item will be pasted into the drawing with a new name ‘NAME’1.

Cancel
Cancels the Paste operation.

Note: This option cannot be used before cutting or copying an item.

Rename
Renames the item.

Note: Some automatically generated definitions cannot be renamed.

Select All
Selects all items.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected item as current.

10.19 EXPORT command

Opens the **Export drawing as** dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: DWFOUT, EXP

10.19.1 Description

Opens the Export Drawing As dialog box to save the data from the current drawing to a variety of file formats.

Supported file formats for export, depends on the license level, are:

BricsCAD Lite

- **FBX binary file** (.fbx)
- **FBX ASCII file** (.fbx)
- **AutoCAD Drawing** (dwg)
- **AutoCAD ASCII DXF** (dxf)
- **Drawing Template** (.dwt)
- **Bitmap file**(2) (.bmp)
- **Windows Meta File** (.emf)(1) (2)
- **Enhanced Windows Meta File** (.wmf)(1) (2)
- **3D DWF v6.01** (.dwf)
- **Binary DWF v6.0** (.dwf)
- **Zipped ASCII Encoded 2D Stream DWF v6.0** (.dwf)
- **Compressed DWF v5.5** (.dwf)
- **Binary DWF v5.5** (.dwf)

- **ASCII DWF v5.5** (.dwf)
- **Compressed DWF v4.2** (.dwf)
- **Binary DWF v4.2** (.dwf)
- **ASCII DWF v4.2** (.dwf)
- **XPS DWFx** (.dwfx)
- **Scalable Vector Graphics** (.svg)
- **Lithography** (.stl)
- **Collada file**(.dae)
- **Micro Station DGN File** (.dgn)

BricsCAD Pro

- **Unreal Datasmith** (.udatasmith)(1) (2)
- **Rhino file** (.3dm)(1) (2)
- **3D PDF**: .pdf
- **Assembly ACIS file** (.asat)
- **Assembly ACIS file** (.asab)

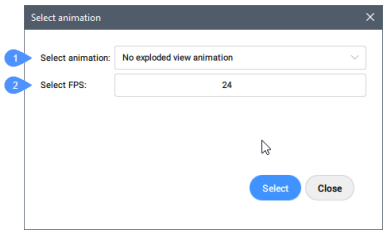
BricsCAD BIM / Mechanical / Ultimate

- **IFC2x3 file** (.ifc)
- **IFC4 file** (.ifc)
- **IFC4x1** (.ifc)

(1) Not available for BricsCAD® for Linux.

(2) Not available for BricsCAD® for macOS.

Note: If the EXPORT3DPDFWRITER system variable is set to 1 (Internal 3D PDF writer), and you save the drawing as a 3D PDF file, the **Select animation** dialog box is displayed:



Choose an animation to export (1) and set the required number of frames per second (fps) (2), or select **No exploded view animation** to export the document without animation.

Note: Export to **Binary DWF v6.0** (.dwf) takes into account only the **Plot area** of the layout or modelspace defined in the Page Setup. So everything outside the plot area will be removed and or clipped away.

Note: The EXPORTMODELSPACE, EXPORTPAPERSPACE and EXPORTPAGESETUP system variables currently are dummies for .dwf exports.

Note: More 3D formats are available through a separate module, **BricsCAD Communicator**. You can download it from the Bricsys website.

The import and export procedures using Communicator are controlled through a series of user preferences. See the Communicator section on the Settings dialog box.

Note: To export to a PDF format, use the EXPORTPDF command.

10.20 EXPORTLAYOUT command

Opens the Export layout to drawing dialog box.

Shape Lite Pro Mechanical BIM

10.20.1 Description

Opens the Export layout to drawing dialog box to save data from the current drawing to a dwg file. The current layout specifies what data to include in the drawing. After the drawing is created, a BricsCAD dialog box allows you to choose if you want to open the new drawing.

10.21 EXPORTPDF command

Opens the Export drawing as dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

10.21.1 Description

Open the Export drawing as dialog box to save data from the current drawing to a PDF file.

10.22 EXPPDFS command

Attaches PDF files as underlays to the current drawing through the Drawing Explorer.

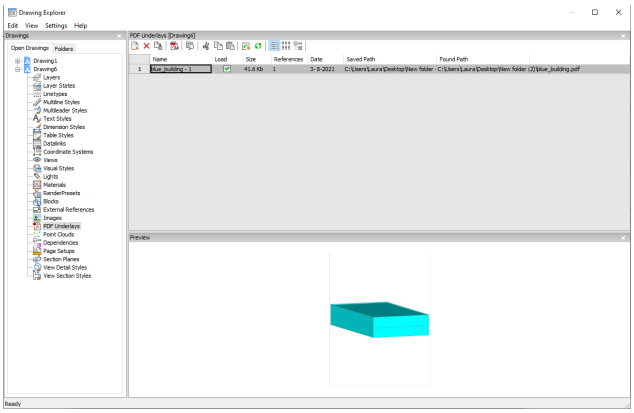
Shape Lite Pro Mechanical BIM

Icon: 

Note: To import PDF files as drawing entities, use the PDFIMPORT command.

10.22.1 Description

Displays the PDF Underlays category of the Drawing explorer dialog box:



10.22.2 Option within the command

Name

Reports the name of the PDF insert. This name is generated by the program, but can be edited by pressing double click on the name or right-click and choose Rename from the shortcut menu.

Load

Toggles loading of the PDF:

On: loads the PDF, and makes it visible.

Off: unloads the PDF, making it invisible.

Size

Reports the size of the PDF file.

References

Reports the number of times the PDF is attached to the drawing.

Date

Reports the date of the file. This can be useful in determining whether you ware working with the most recent revision.

Saved Path

Reports the path to the PDF file when first loaded. When the path cannot be found, click the Browse button which will display the Choose a File dialog box and locate the missing PDF file.

Found Path

Reports the current path to the PDF file. This path should match the Saved Path in most cases.

10.22.3 Context menu options

New

Attaches a PDF file to the current drawing. From the Select PDF Underlay File dialog box open a PDF file, specifies the parameters in the Attach PDF Underlay dialog box and an insertion point to locate the lower-left corner of the PDF.

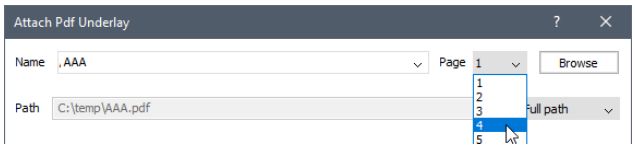
Delete

Detaches the selected PDF attachments without warning.

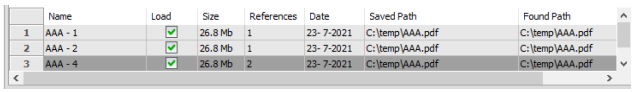
Purge

Purges selected PDF files from the drawing; available only for those PDF file definitions that are not inserted in the drawing.

Insert
Attaches additional pages from a multi-page PDF file already attached to the current drawing. From the Attach Pdf Underlay dialog box choose a page number from the Page drop-list.



The added page appears in Drawing explorer with the page number as a suffix to its name.



Run extended search for missing attachments
Runs an extended search for missing attachments.

Clear cache
Clear Cache for PDFs.

Relative path
Shows the Saved Path as a relative path.

Absolute path
Shows the Saved Path as an absolute path of the PDF file located in the drawing folder.

Filename as a path
Replaces the saved path with the file name.

10.23 EXPUCS command

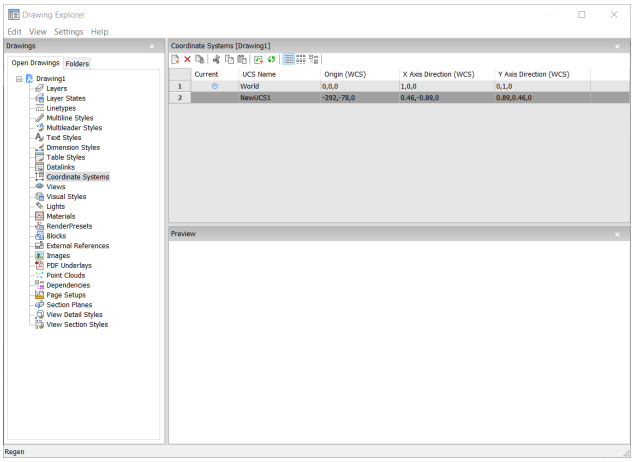
Creates, renames and deletes named UCS's through the Drawing explorer dialog box.

Shape Lite Pro Mechanical BIM

Alias: DDUCS, UC

10.23.1 Description

Opensthe Drawing explorer dialog box on the Coordinate systems category:



10.23.2 Options within the command

Current
Indicates the current UCS.

UCS Name
Lists the names of UCS's in the drawing; click to rename.

Origin (WCS)
Reports the origin (0,0,0) of the UCS in x,y,z coordinates of the WCS; cannot be edited.

X Axis Direction (WCS)
Reports the direction of the x axis in WCS coordinates.

Note: Cannot be edited.

Y Axis Direction (WCS)
Reports the direction of the y axis in WCS coordinates.

Note: Cannot be edited.

10.23.3 Context Menu Options

New
Creates additional UCS into the drawing through the model space.

Delete
Deletes Coordinate system from the drawing. The World coordinate system definitions cannot be deleted.

Rename
Renames the Coordinate System.

Select All
Selects all Coordinate System definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected Coordinate System as current.

10.24 EXPXREFS command

Combines several xref-related commands through the Drawing explorer.

Shape Lite Pro Mechanical BIM

Icon: 

10.24.1 Description

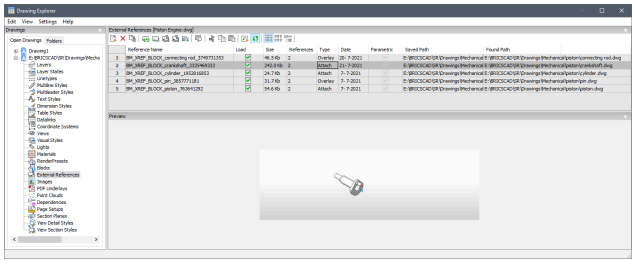
This command attaches DWG files to the current drawing and controls the state of the attachments (short for "external reference").

Note: The XDWGFADECTL system variable sets the fading of external references. Values between 0 (no fading) and 90 are accepted.

Note: The BINDTYPE system variable influences the behavior of the Bind option of the -XREF command.

10.24.2 Method

Displays the External Reference section of the Drawing explorer:



10.24.3 Option within the command

Reference Name

Reports the name of the xref insert. This name is assigned by the program, but could be edited by pressing double click on the name or right-click and choose the Rename from the shortcut menu.

Load

Toggles loading of the xref:

On: loads the xref, and makes it visible

Off: unloads the xref, making it invisible

Size

Reports the size of the xref DWG file. Very large files can slow down the system.

References

Reports the number of times the xref is attached to the drawing.

Type

Reports whether the xref is attached or overlaid, how they treat xrefs that have xrefs themselves:

Attach: all xrefs are displayed.

Overlay: only the first xref of nested xrefs is displayed.

Date

Reports the date of the file. This can be useful in determining whether you ware working with the most recent revision.

Parametric

Indicates if the Xref drawings is parametric or not.

Saved Path

Reports the original path to the xref DWG file when first loaded. When the path cannot be found, click the Browse button which will display the Choose a File dialog box and locate the missing DWG file.

Path for parametric blocks cannot be changed.

Found Path

Reports the current path to the xrefs DWG file. This path should match the Saved Path in most cases.

10.24.4 Context menu options

Attach Xref
Attaches DWG files as xrefs to the current drawing. From the Attach External Reference dialog box browse and choose a dwg file and specify the parameters and an insertion point to locate the xref file. See -XREF command.

Detach Xref
Erases the selected xref from the drawing without warning. This is equivalent to using the Detach option.

Reload Xref
Reloads the selected xref. This is useful when the original DWG file has changed and you want to see the updated version in your drawing.

Unload Xref
Unloads the selected xref. This hides the xref from view. You can also click the checkboxes in the Load column.

Bind Xref
Binds the xref to the current drawing to make the xref part of the drawing. References to the xref disappear from the Drawing explorer. (This option is not available for xrefs that are unloaded.)
This option sets the BINDTYPE system variable OFF (Traditional binding behavior), thus influencing the behavior of the Bind option of the -XREF command.

Insert Xref
Converts xrefs into blocks. This is like using the Insert command to insert external DWG files in drawings. References to the xref disappear from the Drawing explorer. (This option is not available for xrefs that are unloaded.)

Open Xref
Opens the xref DWG file for editing. See the XOPEN command.

Run extended search for missing attachments
Runs an extended search for missing attachments.

Relative path
Shows the Saved Path as a relative path.

Absolute path
Shows the Saved Path as an absolute path of the Xref file located in the drawing folder.

Filename as a path
Replaces the saved path with the file name.

10.25 EXTEND command

Extends or trims one or more open entities to a boundary entity.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: EX

10.25.1 Method

Choose one or more entities to use as boundaries entity (2). These are the entities to which subsequently picked entities are extended or press ENTER to choose all entities in the drawing as boundaries. There is no need to pick individual entities as boundaries, except in certain cases.

Chooses the entity to be extended (1). When another end of an entity could be extended to a boundary, BricsCAD extends the closest end to your pick point.



1 Entity to be extended

2 Boundary entity

3 Extended entity

Note: Switches to trim mode: press and hold the Shift key to select the part of an entity to be trimmed at the intersection with the nearest boundary entities. See the TRIM command.

10.25.2 Options

Fence
Allows fence selection, an irregular selection window.

Crossing
Allows crossing rectangle selection. The entities that cross the specified rectangle are extended.

Edge mode
Toggles edge mode.

Projection
Specifies how the entity is projected to the boundary.

No projection
Extends only those entities that intersect real boundaries in 3D space.

xy plane of Ucs
Projects entities and boundaries to the x,y - plane of the current UCS, and then extends the projected entities that would intersect projected boundaries.

current View
Projects entities into the current view, and then extends them accordingly.

eRase
Deletes selected entities.

10.26 EXTENSION command

Toggles the Extension entity snap.

Shape Lite Pro Mechanical BIM

Icon:

10.26.1 Description

Toggles the Extension entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

10.27 EXTRACTBLOCKS command

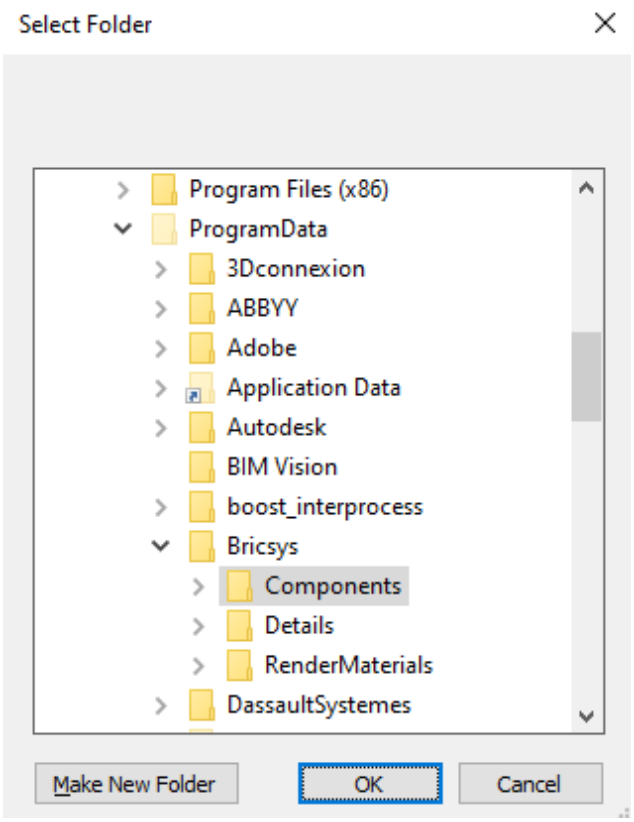
Extracts blocks to individual files.

Shape Lite Pro Mechanical BIM

10.27.1 Method

This command extracts a block instance from a drawing, creating a separate .dwg file in a user-specified location.

After selecting a block to extract, the Select Folder dialog box displays to choose the location for the new created .dwg file.



10.28 EXTRUDE command

Creates 3D solids or surfaces by extruding entities.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: EXT

10.28.1 Description

Creates 3D solids or surfaces by extruding open or closed 2D entities, faces of 3D solids, regions or closed boundaries.

Note:

- The SELECTIONPREVIEW system variable must be set to 2 or 3 to highlight faces.
- Depending on the value of the DELOBJ system variable, the source entities are either retained or deleted. If not, you are prompted whether you want the entities deleted or not.

10.28.2 Method

There are two methods to extrude entities:

- Create 3D solids.
- Create surfaces.

10.28.3 Options within the command

MOde
Allows you to create either solids or surfaces.

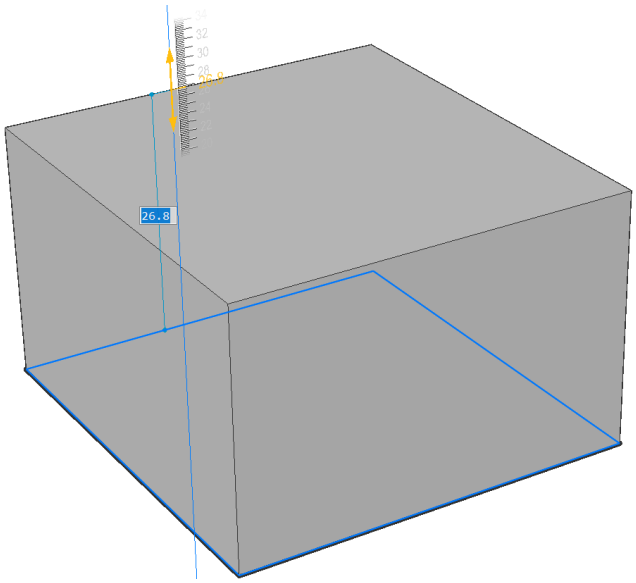
SOlid
Creates 3D solids.

SUrface
Creates surfaces.

Note: EXTRUDE does not extrude open 2D entities as 3D solids because they cannot be watertight.

Specify height of extrusion
Allows you to set a height of extrusion. The extrusion height is measured perpendicularly to the source entity.

Note: You can specify the height dynamically, using the **Manipulator**, or typing a distance value.



Direction

Allows you to specify the extrusion direction.

Path

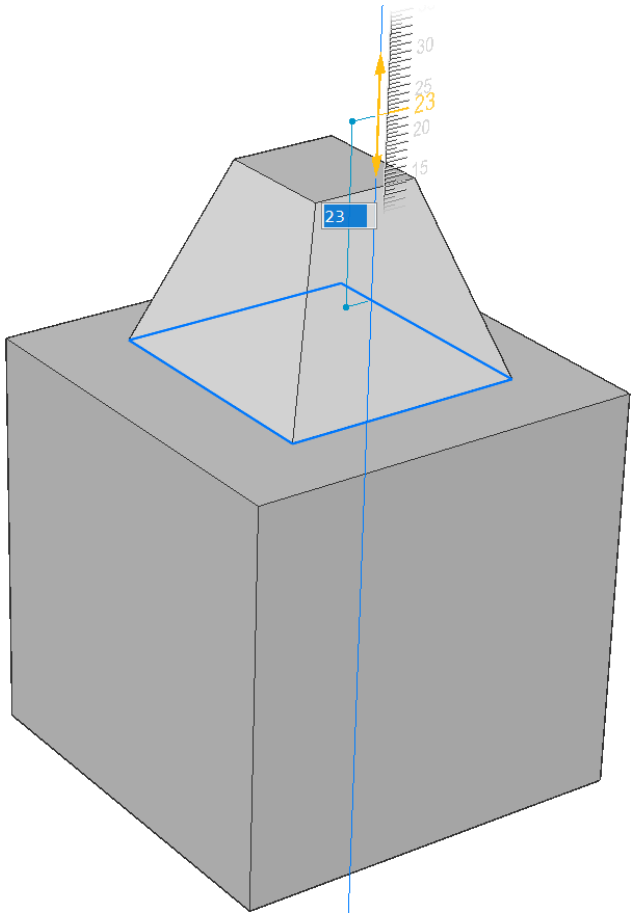
Specify the extrusion height and taper by another entity. The program uses the path to determine how to extrude the source entity.

Note: The path entity cannot lie in the same plane as the extrusion entity.

Taper angle

Specifies the taper angle for the extrusion. The angle is measured from the extrusion direction. A negative value tapers outwards.

Note:The angle must be shallow enough so that the sloping sides do not intersect after the top of the extrusion.



Auto

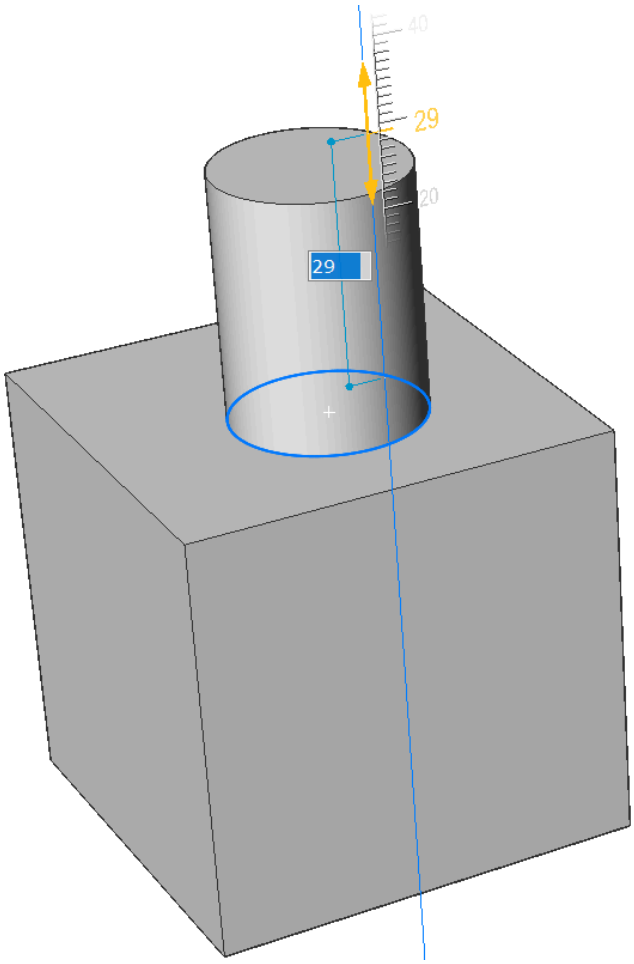
The result depends on the extrusion direction and the value of the EXTRUDEOUTSIDE, EXTRUDEINSIDE, INTERSECTEDENTITIES and UNITESURFACES system variables.

Note: The default value of all above Extrude mode system variables depends on the Workspace:

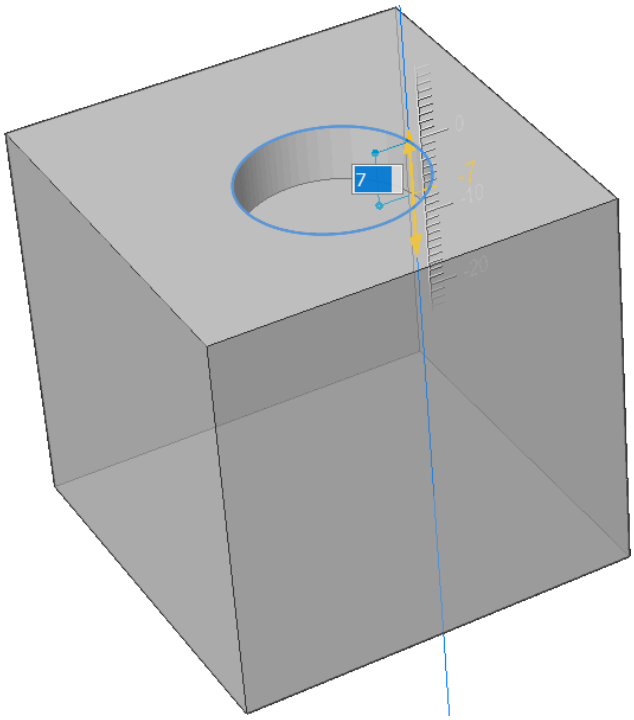
- **0** for Draft and Modeling.
- **1** for Mechanical and BIM.

For example, in the Mechanical an BIM workspaces if all Extrude Mode system variables have the default value, the extrude result is like below:

- When you highlight an entity and extrude it to the outward, a new volume is added.

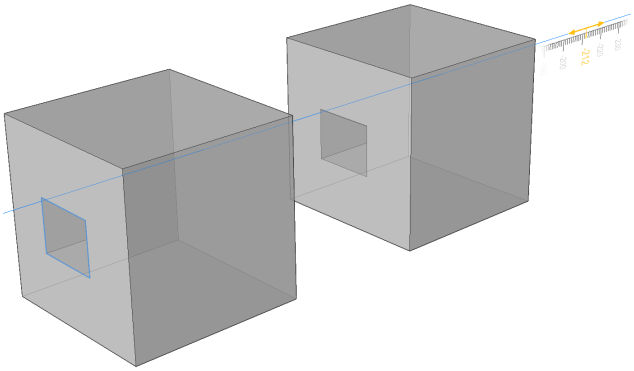


- When you highlight an entity and extrude it to the inward, a volume is subtracted from the main solid.



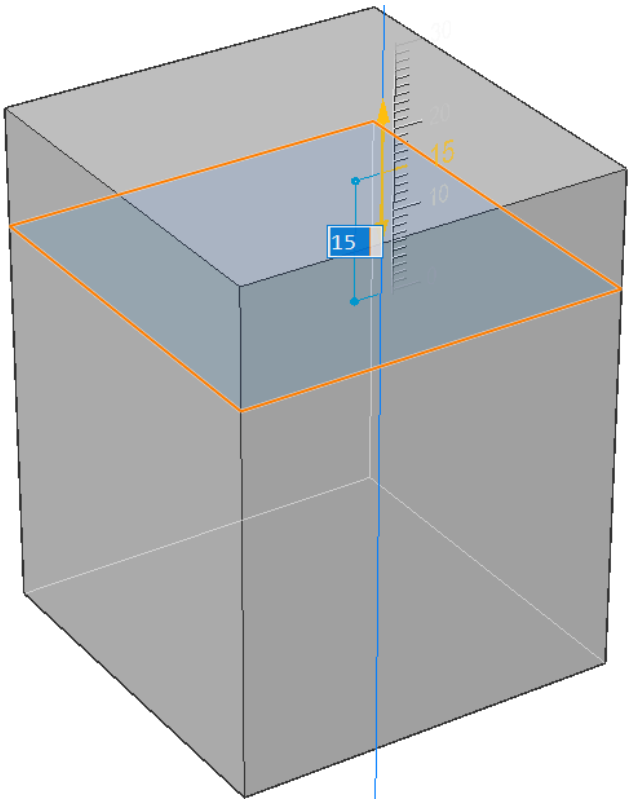
Subtract

The 3D solid is subtracted from each interfering existing solid.



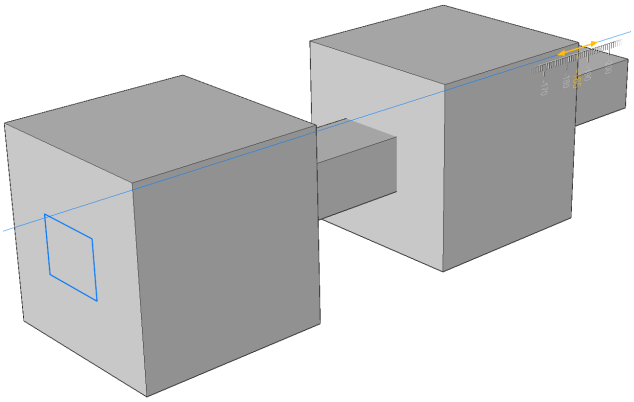
Create

A new volume or surface is created, regardless of the extrusion direction.



Unite

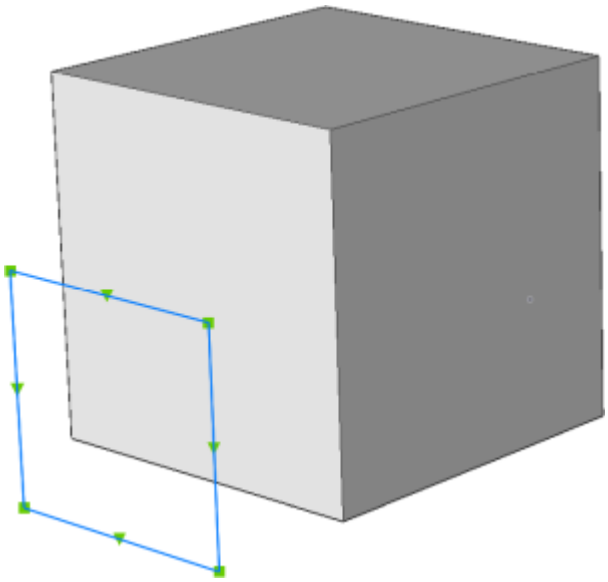
The new 3D solid is unified with each interfering existing solid.



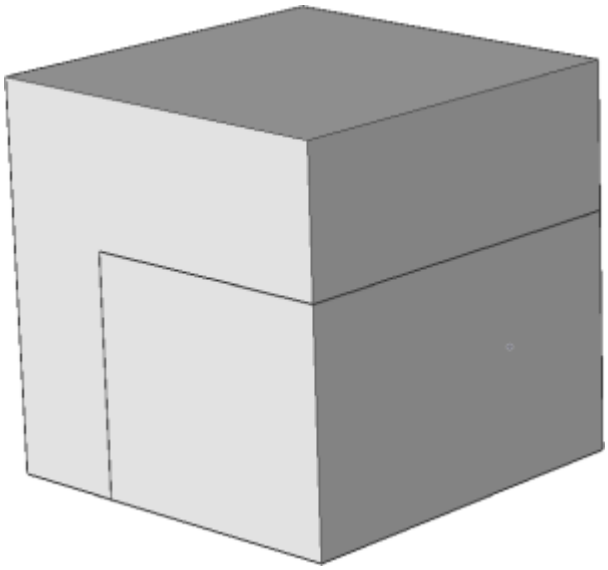
Slice

Cuts through solids with the extruded surface.

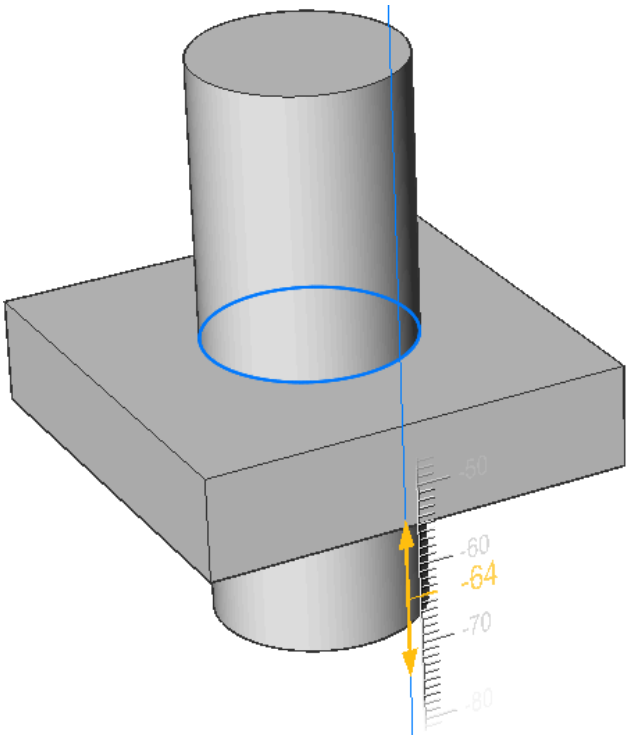
- Select an entity to be extruded.



- The extruded surface cuts through solids.



Both sides
Extrudes symmetrically in both sides.



ALign
Allows you to select or to define an axis.

2Points
Define the extrusion direction by specifying two points.

Entity
Allows you to select an axial entity.

Last
Uses the previous axis.

View
Allows you to specify a point on view direction.

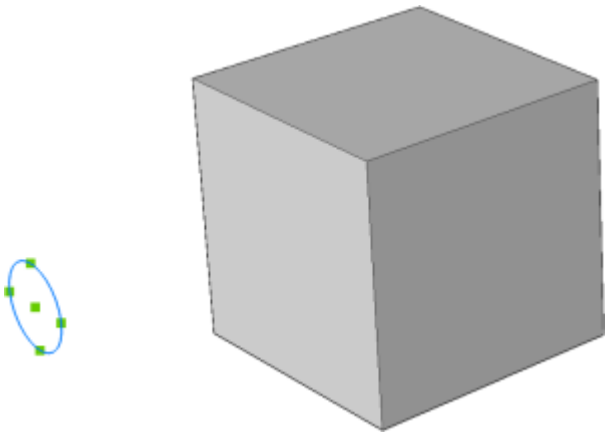
Xaxis
Sets the extrusion direction parallel to the X axis of the current UCS.

Yaxis
Sets the extrusion direction parallel to the Y axis of the current UCS.

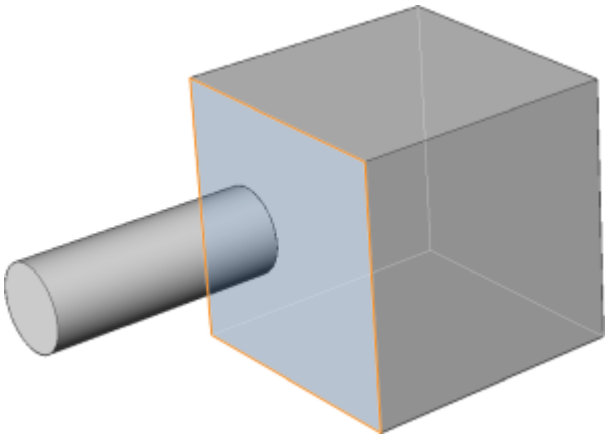
Zaxis
Sets the extrusion direction parallel to the Z axis of the current UCS.

set Limit
Allows to limit the extrusion by the face of a solid.

- Select an entity to be extruded.



- Select a face as limit of the extrusion.



Note: If the HOTKEYASSISTANT system variable is ON, the **Hotkey Assistant** widget displays. Repeatedly press the Ctrl key during the dynamic display of the extrusion to cycle the various options

11. F

11.1 FBXEXPORT command

Exports 3D entities in the current drawing to FBX format.

Shape Lite Pro Mechanical BIM

Note: If you wish to export 2D entities, you must first give them a thickness.

11.1.1 Method

There are two methods to export to FBX format:

- Export all visible entities.
- Export only the selected entities.

11.1.2 Options within the command

Selected
Select the entities to be exported.

Visible
Exports all visible entities.

Note: This option excludes the entities on frozen or off layers, and those that are not visible in the current viewport.

Select
Prompts to select among entities, lights, cameras and materials.

All
Exports all entity types.

Embedded
Includes the texture files into the FBX file.

Reference to file
Adds a reference link in the FBX file to the location of the texture files.

Copy to file
Copies the texture files to the destination folder, separately from the FBX file.

Note: Texture files are not included in the FBX files when they are very large or when a common set of texture files are used by rendering and animation projects.

Enter path to export fbx file:
Specify the path to the folder in which to place the FBX file, or press Enter key to accept the path provided.

Note: Typing ~ displays the FBX export dialog box, which allows you to select a folder.

11.2 -FBXEXPORT command

Exports 3D entities in the current drawing to FBX format.

Shape Lite Pro Mechanical BIM

See the FBXEXPORT command.

11.3 FIELD command

Opens the Field dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

11.3.1 Method

Open the Field dialog box to create a field in the current drawing. After you select the field and choose **OK**, you are prompted at the Command line:

Specify start point
Specify a point in the current drawing where the field will be inserted.
Additional options: [Height/Justify]

Height
Specify the height of the field text.

Justify
Specify a justification for the field text.

11.4 FILEOPEN command

Opens files from the Command line.

Shape Lite Pro Mechanical BIM

11.4.1 Description

Opens drawing (DWG), template (DWT), and interchange (DXF) files from the command line, replacing the current drawing.

11.4.2 Options within the command

Save the changes to drawing file?
Specifies whether the current drawing should be saved or not.

Open Drawing
Type the name of the drawing, including path.

Note: Enter ~ to display the Open Drawing dialog box.

11.5 FILES command

Opens the operating system's file manager to access files.

Shape Lite Pro Mechanical BIM

11.6 FILL command

Toggles the FILLMODE system variable.

Shape Lite Pro Mechanical BIM

11.6.1 Description

Toggles the FILLMODE system variable to specify the display of filled 2D entities including polyline, hatch, solid, and trace entities. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'FILLMODE. You must use REGEN or REGENALL to see the change.

- On: turns on the FILLMODE system variable
- Off: turns off the FILLMODE system variable

11.7 FILLET command

Fillets intersections; joins intersecting lines with an arc of any suitable radius.

Shape Lite Pro Mechanical BIM

Icon:

Alias: F

11.7.1 Description

Sets a radius value for fillet between two entities.

The entities could be:

- Lines, including parallel lines
- All vertices of a single 2D polyline; two polylines cannot be filleted
- Rays, including parallel rays
- Infinite lines, including parallel infinite lines
- Arcs

Note: Press and hold the Shift key when select the second entity to create a corner (radius = 0). The selected entities are extended or trimmed at the intersection point. The selected part of the entities is kept.

Note: If the second entity is parallel to the first entity the entities are connected by a semicircle. If the parallel entities are not of equal length, then the second entity is extended or trimmed when Trim Mode = Trim.

11.7.2 Options

fillet Settings

Displays the Chamfer/Fillet section of the Settings dialog box:



1 Fillet radius - sets the radius of the fillet arc.

2 Trim mode - toggles whether the entities are trimmed back to meet the fillet radius.

Polyline
Fillets the all vertices of a polyline.

Radius
Toggles edge mode.

Trim
Sets the trim mode for chamfers and fillets:

Trim
Trims or extends the selected entities.

No trim
Creates the chamfer or fillet, but leaves the selected entities unchanged.

Undo
Undoes the last fillet when in Multiple mode.

Multiple
Lets to create additional fillets using the same settings without having to restart the command.

11.8 FIND command

Opens the Find and replace dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

11.8.1 Description

Opens the Find and replace dialog box to find and optionally replace strings of text in the current drawing. It searches for text in blocks, attributes, dimensions, and hyperlinks.

11.9 FITARC command

Fits an arc or a circle to entities.

Shape Lite Pro Mechanical BIM

11.9.1 Description

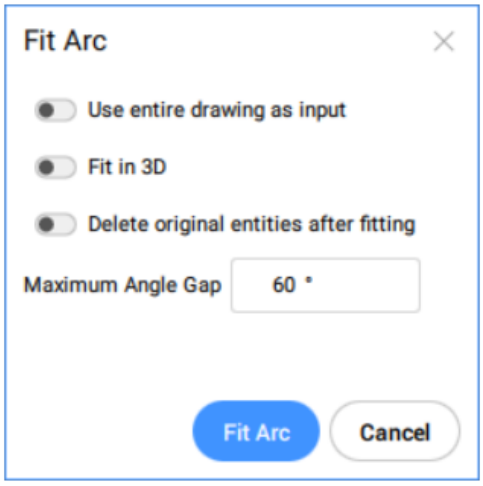
Draws an arc or circle fitted to one or more entities. The command minimizes accumulated perpendicular distances to the points of the selected entity(ies).

A realistic use case is a production company of advertisement cardboard. Typically, the 2D CAD files to pass to the cutting machine are imported from other design software, where polylines and splines are the typical 2D curve entity types. Take the example where the machine has to cut a half-circle in the cardboard, and the half-circle is represented by a polyline with a lot of small subsequent linear segments. When passing this drawing to the cutting machine, the machine will make a sequence of many of small cuts. After each cut, the machine has to stop, change orientation and restart. For each stop-change-restart action, there is a chance for errors. This would not only lead to a much longer cutting time, also the

resulting cut will be suboptimal and look rough. In this case, FITARC could be used to turn this polyline into an ARC, leading to one fast and smooth cut operation.

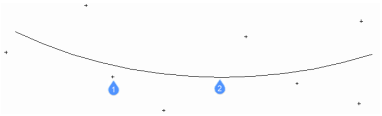
11.9.2 Methods

This command opens the **Fit Arc** command panel.



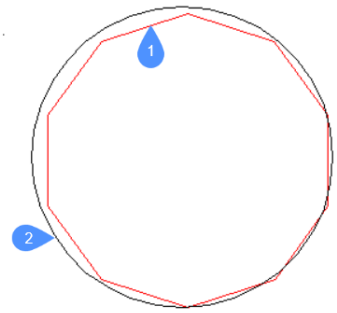
The options within the FITARC command are the same as the options in the **Fit Arc** command panel.

The entities and the results of this command are shown in the examples below:



1 Points used by the command to fit the arc.

2 Arc fitted to the points.



1 Polyline used by the command to fit the arc.

2 Arc fitted to the polyline.

11.9.3 Options within the command

use Entire drawing
Use all entities in the drawing as input.

Fit in 3d
This option applies when one or more input entities do not lie in the XY-plane.
If checked, the resulting arc or circle is created in 3D space, otherwise the input entities are projected onto the XY-plane first.

> ***Note:*** The XY plane used for projection is the one of the current UCS (User Coordinate System). This allows the user to make the fit in any plane.

Delete original entities after fitting
When this option is set to **Yes**, the initial entities are deleted.

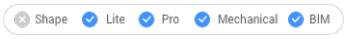
maximum Gap angle
Defines the threshold value for the maximum angle gap between the characteristic points of the input entities. If the maximum gap angle is larger than the threshold value, an arc is created. Otherwise, a circle is created.

selection options (?)
Allows to choose a selection method. See the SELECT command.

> ***Note:*** The options **Use entire drawing**, **Fit in 3d**, and **Delete original entities after fitting** can be set using the system variable FITLINEFITARCMODE. The option **Maximum gap angle** can be set using the FITARCMAXGAP system variable.

11.10 FITLINE command

Fits a line to entities.



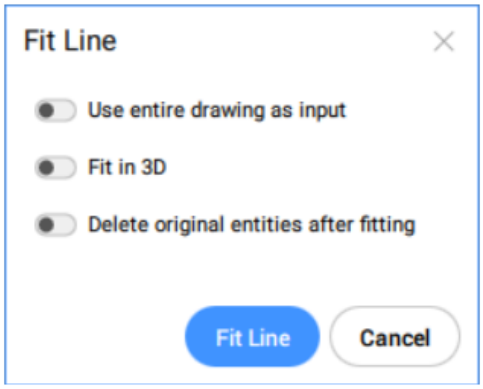
11.10.1 Description

Draws a line fitted to one or more entities. This command minimizes accumulated perpendicular distances to the points of the selected entities.

The FITLINE command works on any input entity type, so can be applied in various real life use cases. You can use it to draw a line which optimally matches a set of scanned points in the XY-plane. You can use FITLINE to find the optimal line that fits a multi-segment polyline or spline curve that goes up and down.

11.10.2 Methods

The command opens the **Fit Line** command panel.



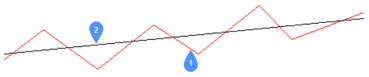
The options in the FITLINE command are the same as the options in the **Fit Line** command panel.

The entities and the results of this command are shown in the examples below:



1 Points used by the command to fit the line.

2 Line fitted to the points.



1 Polyline used by the command to fit the line.

2 Line fitted to the polyline.



1 Spline used by the command to fit the line.

2 Line fitted to the spline.

11.10.3 Options within the command

use Entire drawing

Use all entities in the drawing as input.

Fit in 3d

This option applies when one or more input entities do not lie in the XY-plane.

If checked, the resulting line is created in 3D space, otherwise the input entities are projected onto the XY-plane first.

Note: The XY plane used for projection is the one of the current UCS (User Coordinate System). This allows the user to make the fit in any plane.

Delete original entities after fitting
When this option is set to Yes, the initial entities are deleted.

selection options (?)
Allows to choose a selection method. See the SELECT command.

Note: The options **Use entire drawing**, **Fit in 3d**, and **Delete original entities after fitting** can be set using the FITLINEFITARCMODE system variable.

11.11 FITPOLYLINE command

Fits polylines to entities.

Shape Lite Pro Mechanical BIM

Icon: 

11.11.1 Description

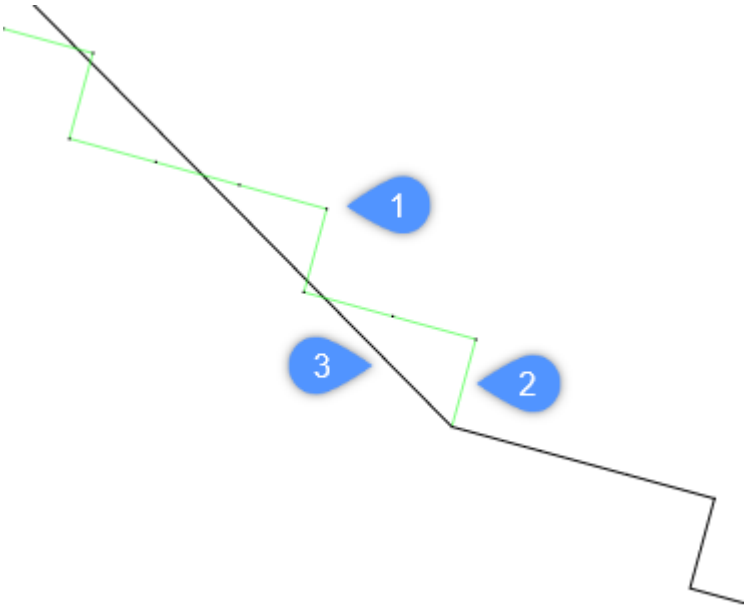
Draws one or multiple polyline entities, fitted to the characteristic points of the input entities. The command minimizes accumulated perpendicular distances to the points of the selected entities. The command works with every type of entity, as long as it has characteristic points.

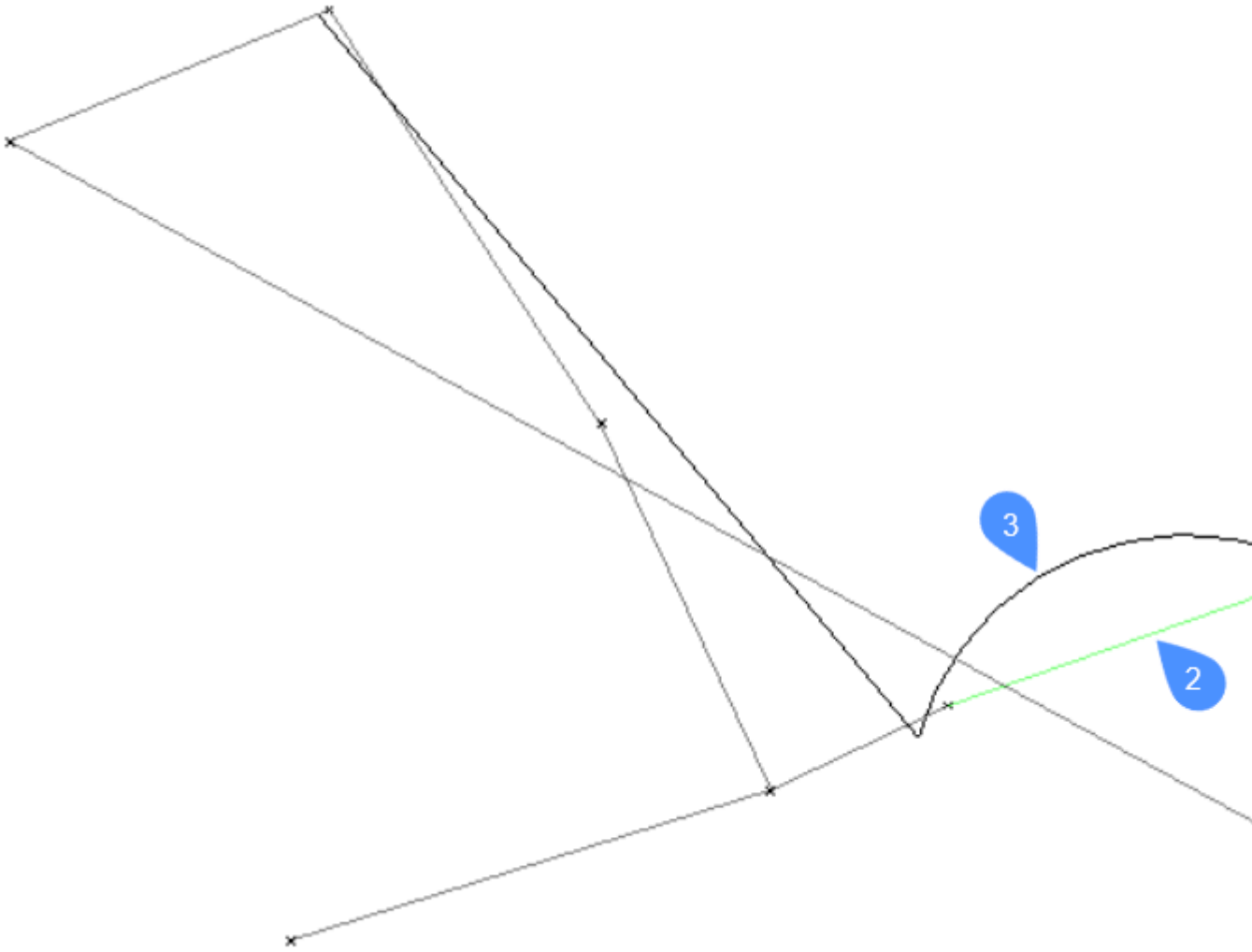
11.11.2 Method

There are two ways to define the input to the FITPOLYLINE command:

- Select fitting entities: choose the entities to which the polyline should be fitted.
- Use entire drawing: select all non-frozen entities in the current viewport.

Pick a point for Start of polyline. The guideline is used as a visual aid for the user, helping to pick the envisioned polyline vertices.





1 The points used by the command to fit the polyline.

2 The suggested guideline.

3 The polyline.

11.11.3 Options within the command

Start the polyline
Choose a point and start the polyline.

fit Lines
Fits a linear polyline segment to the points between the last fixed polyline vertex and the mouse position.

Note: See the FITLINE command.

fit Arcs
Fits an arc polyline segment between the points between the last fixed polyline vertex and the mouse position.

Note: See the FITARC command.

Best fit
The algorithm automatically decides whether a linear or an arc polyline segment is best suited to fit the points between the last fixed polyline vertex and the mouse position.

Note: If the Hot Key Assistant system variable is set ON, press the Ctrl key to cycle between the fit options.

Undo
Undoes the last action.

Close polyline
Closes the polyline.

start New polyline
Starts a new polyline.

Note: Press the Esc key once to end the current polyline and start a new polyline.

Note: Press the Esc key twice to finish the command.

Finish
Finishes the command.

11.12 FLATSHOT command

Creates a flattened, hidden line representation of a 3D model.

Shape Lite Pro Mechanical BIM

Icon: 

11.12.1 Description

Opens the **Flatshot** dialog box to create a flattened, hidden line representation of a 3D model as a block or a new drawing.

11.13 FLATTEN command

Flattens 2D and 3D entities.

Shape Lite Pro Mechanical BIM

Icon:

11.13.1 Description

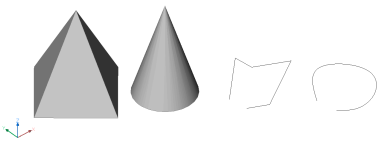
Flattens 2D and 3D entities by projecting them onto the XY-plane of the current view.

Note: The PERSPECTIVE system variable must be set to 0.

11.13.2 Method

The entities will be projected onto the XY plane of the current view. In top view, the entities will be projected onto the XY-plane of the current coordinate system (WCS or UCS).

Entities before FLATTEN command:



Entities after FLATTEN command:



- Pyramid converted to a square with its diagonals.
- Cone becomes a circle with a point in the center.
- 3D polyline and spline are converted to 2D polylines.

11.13.3 Options within the command

new ucs ELevation

Specifies the new elevation for the resulting 2D entities.

Note: The elevation and thickness of the 2D entities can be changed with the ELEV command.

allow EXplode

Explodes compounded entities, such as 3D solids.

11.14 FLIPLINE command

Shape Lite Pro Mechanical BIM

11.14.1 Methods

This command is to be used with 2D parametric blocks.

The FLIPLINE command allows to draw a line that will be used as a reference axis for flipping the entities of the block. After it has been created, the flip line will have a green color and will be moved to a special layer named FLIPLINES.

The FLIPLINE command will automatically create a special type of parameter and will associate it to the flip line entity. This parameter has two possible values, **Not Flipped** and **Flipped**.

The flip of the block will occur when the value of the flip parameter is changed. This parameter can be accessed in **Parameters and Constraints** panel. Another way to change the flip parameter is to select the block and go to the **Properties** panel, in **Parameters** section.

When the flip parameter takes the value **Not Flipped**, the block is in its default configuration, as it was drawn. When the flip parameter value is changed to **Flipped**, the block is flipped around the flip line.

Note: The construction of the flip line is equivalent with drawing a line using the **Line** command.

11.14.2 Options within the command

Angle
Specify the angle of the line segment.

Length
Specify the length of the line segment.

Undo
Undo the last line segment and continue drawing from its previous start point.

Close
Automatically draw a line from the end point of the last segment to the start point of the first segment.

11.15 FROM command

Allows you to enter relative coordinates with respect to a reference point whenever a command prompts you to enter a point.

Shape Lite Pro Mechanical BIM

Icon: 

Note: From is a command modifier, not a command, and so it is entered only during a drawing or editing command.

Note: This command is useful for starting entities offset from another entity, such as inserting a door relative to the end of a wall.

11.15.1 Options within the command

Base point
Pick a point or enter coordinates to specify the base point.

Offset or regular point
Enter the offset by specifying relative coordinates, using the @x,y,z format

12. G

12.1 GCCOINCIDENT command

Creates a coincident geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.1.1 Description

Creates a coincident constraint to ensure a point on an entity remains coincident with a specified point or entity.

12.1.2 Methods to create a coincident constraint

This command has 3 methods to begin creating a coincident constraint:

- Select first point
- Entity
- Autoconstrain

Select first point

Begin creating a coincident constraint by specifying a point on an entity then:

Select second point

Specify a point on an entity to make coincident with the first point. The point on the first entity maintains its position while the second entity adjusts, as necessary, to become coincident.

Additional Options: [Entity]

Entity

Begin creating a coincident constraint by selecting an entity then:

Select a point

Specify a point on an entity to make coincident with the first entity. The first entity maintains its position while the point on the second entity adjusts, as necessary, to become coincident.

Additional Options: [Multiple]

Autoconstrain

Create coincident constraints by selecting all entities to which you want to apply relevant coincident constraints.

12.1.3 Optionswithin the GCCOINCIDENT command

After you begin creating a coincident constraint, the following option may be available:

Multiple

Create multiple coincident constraints until you press Enter to end the command.

12.2 GCCOLINEAR command

Creates a colinear geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.2.1 Description

Creates a colinear constraint to ensure two or more linear entities remain colinear.

12.2.2 Methods to create a colinear constraint

This command has 2 methods to begin creating a colinear constraint:

- Select first entity
- Multiple

Select first entity
Begin creating a colinear constraint by selecting a linear entity then:

Select second entity
Select a linear entity to make colinear to the first one. The first entity maintains its position while the second entity adjusts, as necessary, to become colinear.

Multiple
Begin creating more than one colinear constraint by specifying the Multiple option then:

Select first entity
Select a linear entity then:

Select entity to make collinear to the first
Select a linear entity to make colinear to the first one. You can continue adding colinear constraints to other entities until you press Enter to end the command. The first entity maintains its position while subsequent entities adjust, as necessary, to become colinear.

12.3 GCCONCENTRIC command

Creates a concentric geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.3.1 Description

Creates a concentric constraint to ensure two circular or elliptical entities remain concentric.

12.3.2 Method to create a concentric constraint

This command has 1 method to begin creating a concentric constraint:

- Select first entity

Select first entity
Begin creating a concentric constraint by selecting a circular or elliptical entity then:

Select second entity
Select a circular or elliptical entity to make concentric with the first entity. The first entity maintains its position while the second entity moves, as necessary, to become concentric.

12.4 GCENTER command

Toggles the Geometric center entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

12.4.1 Description

Toggles the Geometric entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

12.5 GCEQUAL command

Creates an equal geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

12.5.1 Description

Creates an equal constraint to ensure circular entities maintain equal radii or linear entities maintain equal lengths.

12.5.2 Methods to create an equal constraint

This command has 2 methods to begin creating an equal constraint:

- Select first entity
- Multiple

Select first entity
Begin creating an equal constraint by selecting an arc, circle, line, or polyline segment then:

Select second entity
Select a similar entity to make equal to the first. The first entity maintains its radius or length while the radius or length of the second entity updates to match the first entity.

Multiple
Begin creating more than one equal constraint by specifying the Multiple option then:

Select first entity
Select an arc, circle, line, or polyline segment.

Select entity to make equal to the first
Select a similar entity to make equal to the first. Continue selecting similar entities until you press Enter to end the command. The first entity maintains its radius or length while the radius or length of all subsequent entities update to match the first entity.

12.6 GCFIX command

Creates a fixed geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.6.1 Description

Creates a fixed constraint on a point or entity to maintain its position in the drawing.

12.6.2 Methods to create a fixed constraint

This command has 2 methods to create a fixed constraint:

- Select point
- Entity

Select point
Create a fixed constraint by specifying a point on an entity. The point maintains its position while other geometry moves, as necessary, to maintain other constraints.

Entity
Create a fixed constraint by selecting an entity. The entity maintains its position while other geometry moves, as necessary, to maintain other constraints.

12.7 GCHORIZONTAL command

Creates a horizontal geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

12.7.1 Description

Creates a horizontal constraint to ensure a linear entity or pair of points remains parallel to the x-axis.

12.7.2 Methods to create a horizontal constraint

This command has 2 methods to begin creating a horizontal constraint:

- Select an entity
- 2 Points

Select an entity
Create a horizontal constraint by selecting a linear entity to make parallel to the x-axis.

2 Points
Begin creating a horizontal constraint by specifying the first point to make parallel to the x-axis then:

Select Second Point:
Specify the second point to make horizontal to the x-axis. The first point maintains its position while the second point moves to become horizontal with the first point.

12.8 GCPARALLEL command

Creates a parallel geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.8.1 Description

Creates a parallel constraint to ensure linear entities remain parallel to each other.

12.8.2 Methods to create a parallel constraint

This command has 1 method to create a parallel constraint:

- Select first entity

Select First Entity
Begin creating a parallel constraint by selecting a linear entity then:

Select Second Entity
Select a second linear entity to make parallel to the first. The first entity maintains its position while the second entity moves, as necessary, to become parallel to the first entity.

12.9 GCPERPENDICULAR command

Creates a perpendicular geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.9.1 Description

Creates a perpendicular constraint to ensure linear entities remain perpendicular to each other.

12.9.2 Methods to create a perpendicular constraint

This command has 1 method to begin creating a perpendicular constraint:

- Select first entity

Select First Entity
Begin creating a perpendicular constraint by selecting a linear entity then:

Select Second Entity
Select a second linear entity to make perpendicular to the first. The first entity maintains its position while the second entity moves, as necessary, to become perpendicular to the first entity.

12.10 GCSMOOTH command

Creates a smooth geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.10.1 Description

Creates a smooth constraint to ensure two splines maintain fluid geometric continuity between them.

12.10.2 Methods to create a smooth constraint

This command has 1 method to begin creating a smooth constraint:

- Select first spline entity

Select first spline curve
Begin creating a smooth constraint by selecting a spline curve then:

Select second curve
Select a second spline entity. The first spline maintains its position while the second spline stretches, as necessary, to smoothly connect to the first spline.

12.11 GCSYMMETRIC command

Constrains a symmetric geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.11.1 Description

Creates a symmetric constraint to ensure two entities remain symmetric with respect to a selected line.

12.11.2 Methodsto create a symmetric constraint

Therearetwomethodsto begin creating asymmetricconstraint:

- Selectfirstentity
- 2Points

Select First Entity
Begin creating a symmetric constraint by selecting a 2D entity then:

Select Second Entity
Select an entity to make symmetric to the first entity then:

Select Line of symmetry:
Select a line to act as a mirror line between the two entities. The first entity maintains its position while the second entity adjusts, as necessary, to become symmetric about that line.

2Points
Begin creating a symmetric constraint by selecting a valid point on a 2D entity then:

Select second point
Select a second valid point on a 2D entity to make symmetric with the first point.

Select line of symmetry
Select a line to act as a mirror line between the two entities. The first point maintains its position while the second point adjusts, as necessary, to become symmetric about that line.

12.12 GCTANGENT command

Creates a tangent geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

12.12.1 Description

Creates a tangent constraint to ensure curved entities remain tangent to another curved or linear entity.

12.12.2 Methods to create a tangent constraint

This command has 1 method to begin creating a tangent constraint:

- Select first entity

Select First Entity
Begin creating a tangent constraint by selecting a linear or curved entity then:

Select Second Entity
Select a second entity to make tangent to the first. The first entity maintains its position while the second entity moves, as necessary, to become tangent to the first entity. At least one curved entity must be selected per tangent constraint.

12.13 GCVERTICAL command

Creates a vertical geometric constraint on 2D entities.

Shape Lite Pro Mechanical BIM

Icon:

12.13.1 Description

Creates averticalconstraint toensure alinear entity or pair of pointsremains parallel to the y-axis.

12.13.2 Methods to create a vertical constraint

This command has 2 methods to begin creating averticalconstraint.

- Select an entity
- 2 Points

Select an entity
Create a vertical constraint by selecting a linear entity to make parallel to the y-axis.

2 Points
Begin creating a vertical constraint by specifying the first point to make parallel to the y-axis then:

Select Second Point:
Specify the second point to make horizontal to the y-axis. The first point maintains its position while the second point moves to become vertical with the first point.

12.14 GENERATEBOUNDARY command

Generates a closed polyline or spline around the boundary of a flat, closed area.

Shape Lite Pro Mechanical BIM

Icon:

12.14.1 Description

Generates a closed polyline or spline around the boundary of a planar closed area, a planar face of a 3D solid or around the area of a hatch pattern.

12.14.2 Method

Select a planar enclosed point to generate a boundary by picking a point inside one of the following entities:

- Closed planar area
- Planar face on a 3D solid; to select the face of a 3D solid hold down the Ctrl key.
- Hatch pattern

The resulting boundary are:

- Polyline - when the boundary is polygonal (has corners, like a square)
- Spline - when the boundary is curved, like a spline.

12.15 GEOGRAPHICLOCATION command

Opens the Geographic location dialog box.

Shape Lite Pro Mechanical BIM

Icon:

Alias: GEO

12.15.1 Description

Opens the Geographic location dialog box to apply a longitude and latitude to a point in the current drawing.

12.16 GEOIMPORT command

Imports a drawing with respect to the geographic location.

Shape Lite Pro Mechanical BIM

12.16.1 Description

Imports a drawing with respect to the geographic location in the source and the target drawing.

Note: The geographic location must be defined in both the source and the target drawing.

12.16.2 Method

This command displays the Open drawing file dialog box, which allows you to select either a Standard Drawing File (*.dwg) or a Drawing Exchange Format (*.dxf).

12.17 GEOMCONSTRAINT command

Applies geometric relationships between entities, on entities, and on valid constraint points.

Shape Lite Pro Mechanical BIM

12.17.1 Description

Constraints keep entities in a fixed position, such as perpendicularly or vertically.

Note: Geometric constraints can be applied to the following entities and their constraint points:

Entity Type	Valid Constraint Points
Lines	Endpoints, midpoint
Arcs, elliptical arcs	Endpoints, center point, midpoint
Circles, ellipses	Center point
Polyline segments	Endpoints, vertices, midpoints
Polyline arcs	Endpoints, vertices, midpoints, center points
Splines	Endpoints
Inserted entities: blocks, xrefs, text, mtext, attributes, tables	Insertion points

12.17.2 Options

Horizontal

Constrains entities or pairs of points to lie parallel to the X-axis of the current coordinate system. See the GCHORIZONTAL command.

Vertical

Constrains entities or pairs of points to lie parallel to the Y-axis of the current coordinate system. See the GCVERTICAL command.

Perpendicular

Constrains two entities to lie perpendicularly to each other. See the GCPERPENDICULAR command.

PArallel

Forces two entities to be parallel to one other. See the GCPARALLEL command.

Tangent
Constrains two entities to maintain a point of tangency to each other or their extensions. See the GCTANGENT command.

SMooth
Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. See the GCSMOOTH command.

Coincident
Applies a coincident geometrical constraint to two points or constrains a point to an entity. See the GCCOINCIDENT command.

CONcentric
Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. See the GCCONCENTRIC command.

COLlinear
Forces entities to be collinear. See the GCCOLLINEAR command.

Symmetric
Constrains two entities or points to lie symmetrically with respect to a selected line. See the GCSYMMETRIC command.

Equal
Constrains circular entities to the same radius, or linear entities to the same length. See the GCEQUAL command.

Fix
Constrains points and entities at a fixed position. See the GCFIX command.

12.18 GOTOSTART command

Displays the Start Tab.

Shape Lite Pro Mechanical BIM

12.18.1 Description

Opens or activates the Start Tab.

12.18.2 Method

In the Start tab you can:

- Open a recent drawing.

Note: The number of available recent files is controlled by the value of the RECENTFILES system variable.

- Create a new drawing.

Note: Click on Template to choose what template is used when creating the new drawing.

- Open a drawing using the Open file dialog box.
- Access BricsCAD tutorials and sample drawings.

Note: Internet access is required.

- Access the Third-Party Application catalog on the Bricsys website.

Note: Internet access is required.

12.19 GRADIENT command

Opens the Hatch and gradient dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

12.19.1 Description

Opens the Hatch and gradient dialog box to create a hatch entity in the current drawing.

12.20 -GRADIENT command

Fills closed areas with a gradient fill.

Shape Lite Pro Mechanical BIM

12.20.1 Description

Fills closed areas with solid fills in gradients of one or two colors in a variety of patterns.

Note: 3D entities cannot be filled in gradients.

12.20.2 Method

There are two methods to fill in gradients:

- Single color
- Two colors

12.20.3 Options within the command

hatch Properties
Allows to specify a hatch pattern.

?
Lists the names of all hatch patterns. Press F2 to see the list in the prompt history window.

Solid fill
Specify a solid-color filled hatch.

User defined
Allows you to customize the hatch pattern.

Angle for lines
Specify the angle for the pattern lines.

Space between standard pattern lines
Specify the space between the hatch lines.

Cross-hatch
The pattern is repeated at 90 degrees to the original.

Gradient
Allows to specify a gradient pattern.

angle for the gradient
Allows to specify an angle for the gradient fill.

Center the gradient
Centers the gradient.

?
Lists the names of all gradient patterns. Press F2 to see the list in the prompt history window.

Single color
Specify a new gradient color and tint.

shade or tint value
Specify a number between 0 and 1 (0 = dark, 1 = light).

COlorbook
Enter Color Book name.

Two color
Specify two new gradient colors.

Remove boundaries
Removes boundary entities.

Advanced options
Allows you to set advanced options for the hatch.

Boundary set
Specify the entities to consider while creating boundaries.

Retain boundary
Determine whether the temporary boundary is retained after the command ends.

Island detection
Toggle whether islands are hatched over or not. Islands represent interior boundaries.

Specify style
Specifies how islands are treated.

Nested
Alternating islands are hatched, beginning with the outermost one.

Outer
Only the outermost region is hatched. Interior islands are not hatched.

Ignore
Islands are ignored and hatched over.

set Associativity
Toggles whether hatches are associative, so they do update/not update their geometry alongside the boundaries.

boundary Tolerance
Specify the largest gap the program ignores when hatching a boundary that is not fully closed.

Note: A value of 0 means that no gaps in the boundary are tolerated by the program.

separate Hatches
Specifies to create separate hatches for each closed area or a single hatch entity for all.

Draw order
Specifies whether the hatch pattern appears visually above/below overlapping entities or its boundary.

Origin
Specify a new value for the hatch origin.

Annotative
Sets the annotative property for the hatch. It applies the current annotative scale, defined by CANNOS-CALE system variable.

Layer
Specify the layer on which the hatch should be placed.

Transparency
Specify a value between 0 and 90 for the transparency.

Note: A value of 0 means fully opaque. The transparency level is limited to 90% to avoid confusion with layers that are frozen or turned off.

ByLayer
Applies the value of the transparency property of the layer on which the hatch resides.

ByBlock
The transparency value is controlled by block.

Use current
Applies the current transparency value as defined by the CETRANSPARENCY system variable.

Note: The transparency value for new hatches is saved by the HPTRANSPARENCYsystem variable.

Undo
Removes selected boundaries from the selection.

12.21 GRADIENTBKGOFF command

Turns off the background gradient.

Shape Lite Pro Mechanical BIM

12.21.1 Description

Turns off the background gradient colors (used by all visual styles, except for 2D Wireframe) to display the default background color.

12.22 GRADIENTBKGON command

Turns on the background gradient.

Shape Lite Pro Mechanical BIM

12.22.1 Description

Turns on the background gradient colors, which are used by all visual styles (except for 2D Wireframe).

12.23 GRADING command

Creates a graded surface on existing topo surfaces.

Shape Lite Pro Mechanical BIM

Icon:

12.23.1 Description

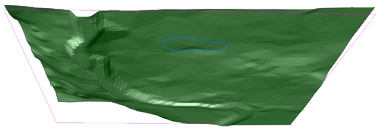
Creates a graded surface between a selected entity and a TIN surface or an offset or slope graded surface from a selected entity.Select the entity to define a graded surface, select the target TIN surface to create a graded surface and move the mouse to adjust the slope of the grading.

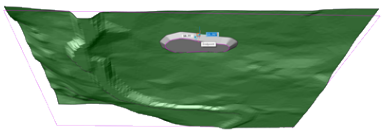
12.23.2 Options

sIOpe/offset

Sets either the slope or offset option of the grading.

Note: The offset-slope projection method requires input entity, offset, and slope.





12.24 GRADINGBALANCE command

Balances grading cut and fill volumes.

Shape Lite Pro Mechanical BIM

Icon:

12.24.1 Description

Balances grading 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.

12.25 GRAPHICOVERRIDE command

Allows you to temporarily modify the appearance of entities in 3D model space.

Shape Lite Pro Mechanical BIM

Icon: 

12.25.1 Description

GRAPHICOVERRIDE is a 3D drawing customization feature that allows you to temporarily override the visual appearance of a specific selection of objects based on BIM properties.

The command opens the **3D Graphic Override** dialog box.

12.26 GRAPHSCR command

Switches to the graphic screen.

Shape Lite Pro Mechanical BIM

12.26.1 Description

Switches from the programs Prompt History window to the drawing window.

Prompt History window is displayed by the TEXTSCR command or by the F2 key.

12.27 GRID command

Toggles the display of the grid and sets some of its properties.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: G

12.27.1 Method

Specify the grid spacing value. It makes the x and y grid spacing the same.

12.27.2 Options within the command

ON
Turns on the display of the grid.

OFf
Turns off the display of the grid.

Snap
Synchronizes the grid spacing with the current snap spacing, so that the two are the same.

Aspect
Sets the aspect ratio of the grid by specifying the horizontal and vertical spacing.

12.28 GROUP command

Opens the Entity grouping dialog box.

Shape Lite Pro Mechanical BIM

Icon:

12.28.1 Description

Opens the Entity grouping dialog box to view and manage named groups of entities in the current drawing.

12.29 -GROUP command

Creates and modifies groups of entities.

Shape Lite Pro Mechanical BIM

12.29.1 Description

Creates and modifies named groups of entities at the Command line.

12.29.2 Options within the command

?

Lists named and unnamed groups in the drawing.

Note: Unnamed groups are prefixed with *A, followed by an incremented number, such as *A3.

Order

Reverses the order of entities in a group.

Add

Adds entities to a group.

Remove

Removes entities from a group.

Explode

Ungroups the selected group.

reName

Renames groups.

Selectable

Toggles the selectivity of groups.

Create

Creates new groups.

13. H

13.1 HATCH command

Opens the Hatch and gradient dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: BH, H

13.1.1 Description

Opens the Hatch and gradient dialog box to create a hatch entity in the current drawing.

13.2 -HATCH command

Fills closed 2D areas with repeating patterns or solid colors.

Shape Lite Pro Mechanical BIM

Alias: -BH, -B

Note: 3D entities cannot be hatched.

13.2.1 Method

There are two methods to apply a hatch pattern:

- Specify internal point.
- Select entities.

Note: Entities which are not visible in the current viewport when the command is launched cannot be selected. Partly visible entities can, however, be selected.

13.2.2 Options within the command

hatch Properties
Allows to specify a hatch pattern.

?
Lists the names of all hatch patterns. Press F2 to see the list in the prompt history window.

Solid fill
Specify a solid-color filled hatch.

User defined
Allows you to customize the hatch pattern.

Angle for lines
Specify the angle for the pattern lines.

Space between standard pattern lines
Specify the space between the hatch lines.

Cross-hatch
The pattern is repeated at 90 degrees to the original.

Gradient
Allows to specify a gradient pattern.

angle for the gradient
Allows to specify an angle for the gradient fill.

Center the gradient
Centers the gradient.

?
Lists the names of all gradient patterns. Press F2 to see the list in the prompt history window.

Single color
Specify a new gradient color and tint.

shade or tint value
Specify a number between 0 and 1 (0 = dark, 1 = light).

COlorbook
Enter Color Book name.

Two color
Specify two new gradient colors.

Remove boundaries
Removes boundary entities.

Advanced options
Allows you to set advanced options for the hatch.

Boundary set
Specify the entities to consider while creating boundaries.

Retain boundary
Determine whether the temporary boundary is retained after the command ends.

Island detection
Toggle whether islands are hatched over or not. Islands represent interior boundaries.

Specify style
Specifies how islands are treated.

Nested
Alternating islands are hatched, beginning with the outermost one.

Outer
Only the outermost region is hatched. Interior islands are not hatched.

Ignore
Islands are ignored and hatched over.

set Associativity
Toggles whether hatches are associative, so they do update/not update their geometry alongside the boundaries.

boundary Tolerance
Specify the largest gap the program ignores when hatching a boundary that is not fully closed.

Note: A value of 0 means that no gaps in the boundary are tolerated by the program.

separate Hatches
Specifies to create separate hatches for each closed area or a single hatch entity for all.

Draw order
Specifies whether the hatch pattern appears visually above/below overlapping entities or its boundary.

Origin
Specify a new value for the hatch origin.

Annotative
Sets the annotative property for the hatch. It applies the current annotative scale, defined by CANNOS-CALE system variable.

Layer
Specify the layer on which the hatch should be placed.

Transparency
Specify a value between 0 and 90 for the transparency.

Note: A value of 0 means fully opaque. The transparency level is limited to 90% to avoid confusion with layers that are frozen or turned off.

ByLayer
Applies the value of the transparency property of the layer on which the hatch resides.

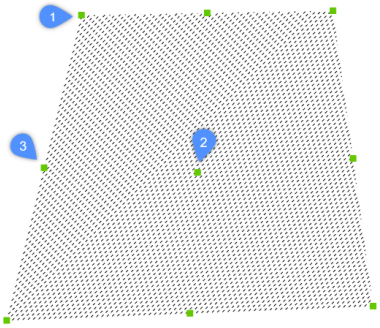
ByBlock
The transparency value is controlled by block.

Use current
Applies the current transparency value as defined by the CETRANSPARENCY system variable.

Note: The transparency value for new hatches is saved by the HPTRANSPARENCYsystem variable.

Undo
Removes selected boundaries from the selection.

Note: Hatches can be edited directly through grips.



- Drag the central grip (2) to move the hatch.
- Drag the vertex (1) or a midpoint grip (3) to change the hatch boundary.

13.3 HATCHEDIT Command

Edits hatches through a dialog box.

Shape Lite Pro Mechanical BIM

Icon:

Alias: HE

13.3.1 Description

Selects a hatch pattern or a gradient fill and edit them through Hatch Edit dialog box.

Note: The Properties panel can also be used to edit hatches and gradient fills.

13.4 -HATCHEDIT command

Edits the boundaries of hatch entities at the Command line.

Shape Lite Pro Mechanical BIM

13.4.1 Description

Edits the boundaries of hatch and gradient entities at the Command line.

13.4.2 Method

There are two methods to use -HATCHEDIT command:

- Select a hatch entity.
- Select a gradient entity.

13.4.3 Options within the command

Disassociate
Removes the associative property of the selected hatch or gradient entity.

ADd boundaries
The area of a hatch is modified by adding boundaries.

Specify internal point
The boundary is determined from an existing enclosed area around the specified point. The selected hatch is then associated with that boundary.

Select entities
The boundary is determined from the selected entities that form an enclosed area. The selected hatch is then associated with that boundary.

Remove boundary
The area of a hatch is modified by removing boundaries.

ASsociate
Associate the selected hatch or gradient entity to a different boundary set.

13.5 HATCHEDITEXT command

Edits the vertices of a hatch or gradient entity.

Shape Lite Pro Mechanical BIM

13.5.1 Description

Edits the vertices of a hatch or gradient entity interactively through the command line.

13.5.2 Options within the command

Add vertex
Adds vertices by stretching selected grip points.

Remove vertex
Removes vertices.

convert to Line
Converts arc segments to line segments.

convert to ARc
Converts line segments to arc segments.

13.6 HATCHGENERATEBOUNDARY command

Generates a boundary around a hatch or gradient fill.

Shape Lite Pro Mechanical BIM

Icon: 

13.6.1 Description

Creates a boundary made of a polyline around the selected hatches or gradient fill.

13.7 HATCHTOBACK command

Moves all hatch entities in the drawing behind all other overlapping entities.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command selects all hatch patterns in the current drawing automatically.

13.8 HELIX command

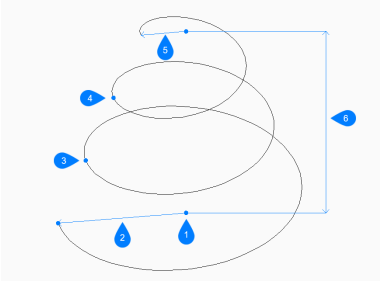
Creates a 2D spiral or a 3D helix.

Shape Lite Pro Mechanical BIM

Icon: 

13.8.1 Description

Creates a 2D spiral or a 3D helix from a combination of options including center, radius, diameter, height, axis endpoints, turns, distance between turns, and twist.



1 Center of base

2 Base radius

3 Start turn 2

4 Start turn 3

5 Top radius

6 Height

13.8.2 Methods to create a helix

This command has 1 method to begin creating a helix:

- Specify center point of base

Specify center point of base
Begin creating a helix by specifying the center of the base then:

Specify base radius
Specify the radius of the helix's base.
Additional option: [Diameter]

Specify top radius
Specify the radius of the helix's top.
Additional option: [Diameter]

Specify helix height
Specify the height of the helix.
Additional options: [Axis endpoint/Turns/Turn height/Twist]

13.8.3 Options within the HELIX command

After you begin creating a helix, the following options may be available:

Diameter
Specify the diameter of the base or top of the helix.

Axis endpoint
Specify the axis endpoint to define height and orientation of the helix in 3D space. The center of the base is used as the other axis endpoint.

Turns
Specify the number of revolutions for the helix, between 1 and 500. The default is 3.

Turn height
Specify the distance between each revolution of the helix. The turn height and the number of turns define the height of the helix.

Twist
Specify which direction the helix rotates.

- CW: clockwise
- CCW: counterclockwise

13.9 HELP command

Opens the BricsCAD Help Center.

Shape Lite Pro Mechanical BIM

Icon: 

13.9.1 Description

Opens the BricsCAD Help Center Description to learn more about BricsCAD commands, system variables, and workflows. It opens in an external application window, using the default Web browser, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

13.10 HELPSEARCH command

Searches online help from the Command line.

Shape Lite Pro Mechanical BIM

13.10.1 Description

Searches the online BricsCAD help pages and then displays the result in your computer's default Web browser; works at the Command line.

13.11 HIDE command

Removes hidden lines from 3D entities.

Shape Lite Pro Mechanical BIM

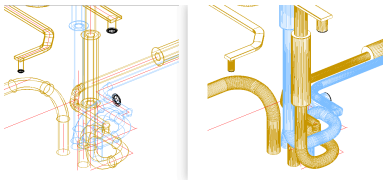
Icon: 

Alias: HI

13.11.1 Method

This command offers only one method to remove hidden lines. The command is run automatically after typing “HIDE” in the Command line and hitting enter.

Displays no prompt in the Command line; the program immediately removes hidden lines.



Note: To return to wireframe display, change the visual style to 2D Wireframe.

13.12 HIDEOBJECTS command

Hides selected entities.

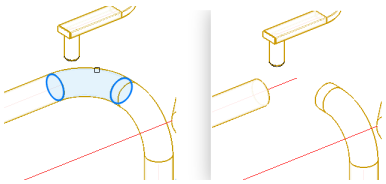
Shape Lite Pro Mechanical BIM

Icon:

13.12.1 Description

Hides selected entities, like using the Freeze option of the Layer command but for individual entities. Hidden entities optionally stay hidden between drawing sessions.

Note: To return hidden objects to view, use the UNISOLATEOBJECTS command.



Note: The OBJECTISOLATIONMODE system variable controls whether the hidden state is saved or not.

Note: The HIDEOBJECTS command is enabled under BEDIT and REFEDIT sessions.

13.13 HYPERLINK command

Links entities and web pages.

Shape Lite Pro Mechanical BIM

13.13.1 Description

Links selected entities with fields and web pages through Edit Hyperlink dialog box.

13.14 -HYPERLINK command

Links entities and web pages.

Shape Lite Pro Mechanical BIM

13.14.1 Description

Links entities with fields and web pages at the Command line.

Note: This command is meant for use by macros and LISP routines.

13.14.2 Options within the command

Remove
Removes hyperlinks from entities.

Insert
Adds a hyperlink to one or more entities.

Enter URL
Specify the path and name of the file, networked file, or location on the Internet.

Enter named location
Specifies a bookmark name.

Note: The # character specifies locations (bookmarks) within files or Web pages.

Enter description
Describes the hyperlink.

Overwrite
Replaces the existing hyperlink with the new one or leaves the existing hyperlink as it is.

13.15 HYPERLINKOPTIONS command

Toggles the display of the hyperlink.

Shape Lite Pro Mechanical BIM

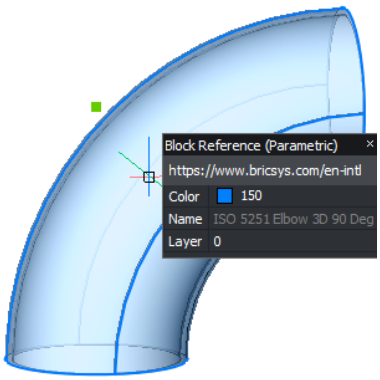
13.15.1 Description

Toggles the display of the hyperlink cursor and URL tooltip and adds Hyperlink to the shortcut menu.

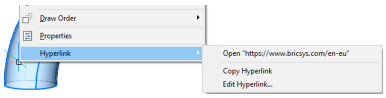
13.15.2 Options within the command

Display hyperlink cursor, tooltip and shortcut menu
Toggles the display of the "hyperlink" icon, tooltip displaying the URL, and the Hyperlink sub-menu on shortcut menus.

Note: The text of the tooltip is specified by the HYPERLINK command's Hyperlink Description option.



Hyperlink menu
Right- click on the selected entity that contains hyperlinks, the Hyperlink item is added to the menu.



Open URL
Opens the location specified by the URL: a location on the Internet or a file with the associated application.

Note: This option executes the URL command.

Copy Hyperlink
Copies the URL to the Clipboard.

Note: The URL 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.

14. I

14.1 ID command

Reports the x,y,z coordinate.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: IDPOINT

14.1.1 Description

Reports the x,y,z coordinates of a specified point using the current coordinate system.

14.2 IEMBED command

Embeds an attached bitonal .tiff image file to the current drawing.

Shape Lite Pro Mechanical BIM

14.2.1 Method

Usually, the images attached to a drawing are saved in a separate file. This command saves a render image within the current drawing.

Note: To reverse this action use the IUNEMBED command.

14.3 IFCEXPORT command

Exports a BIM model to IFC.

Shape Lite Pro Mechanical BIM

14.3.1 Description

Exports a BIM model to IFC, partially or entirely.

Note: The import and export of Alignments and TIN Surfaces with IFC 4X1 is supported.

14.3.2 Method

Select the entities to be exported or press **Enter** to export the entire model.

14.4 IMAGE command

Opens the Attachments panel.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: IM

14.4.1 Description

Opens the Attachments panel to display it in the current workspace. The Attachments panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Attachments panel can be either floating, docked or stacked.

14.5 IMAGEADJUST command

Adjusts the properties of images through the Properties panel.

Shape Lite Pro Mechanical BIM

Alias: IAD

Keyboard Shortcuts: **click the image frame**

14.5.1 Method

After selecting one or more image entities by their frames the Properties panel will display the Raster Image properties.

The Image Adjust properties are:

Brightness
Lightens and darkens the image:

- 0 - very dark or black
- 50 - normal
- 100 - very light or white

Contrast
Increases and decreases the contrast::

- 0 - very low contrast
- 50 - normal
- 100 - very high contrast

Fade
Fades the image:

- 0 - no fade
- 100 - fully faded

14.6 -IMAGEATTACH command

Attaches raster images to the drawing.

Shape Lite Pro Mechanical BIM

14.6.1 Method

Introduce the path and file name of the image file, the insertion point, the scale factor and rotation factor to insert the raster image in the drawing.

14.6.2 Options

Specifies how the program should remember the path to the image, which is saved in the drawing:

Full path
The full path of the image file is saved such as D:\BricsCAD Training\EN\Exercises\Gearbox.png

Relative path
The path relative to the drawing folder is saved such as ..\Exercises\Gearbox.png

No path
The path is not saved, and so the program searches for the image in the drawing folder or in support search file path that could be added with SRCHPATH command.

use Geocoding info
Uses geocoding data to determine the image's insertion point, scale, and rotation angle.

Geocoding file path
Specifies the name of a PGW file that contains the geocoding data.

Embedded geocoding info
Use the geocoding information embedded in the image file.

Xy scales factors
Specifies the scale factors for the X and Y directions independently.

Rotation angle
Specifies the rotation angle.

14.7 IMAGEATTACH command

Opens the Select image file dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: IAT

14.7.1 Description

Opens the Select image file dialog box to select an image file to reference into the current drawing. After selecting the file and choosing **Open**, the Attach Raster Image file dialog box displays. It enables you to specify where and how to attach the image.

14.8 IMAGECLIP command

Clips images with rectangular or polygonal shapes.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: ICL

14.8.1 Method

Select the image by clicking on its frame and create a new clipping boundary.

Clipped images can be edited directly through grips.

Note: An image can have just one clipping boundary; creating a new one erases the old one.

Note: This command can be entered transparently during commands ('imageclip).

14.8.2 Options

ON
Turns on the clipping and displays the clipping boundary.

OFf
Turns off the clipping and hides the clipping boundary.

Invert
Inverts the clipping mode so that the image is clipped on the outside or the inside the clipping boundary.

Delete boundary
Removes the clipping boundary.

New boundary
Creates a new clipping boundary, and turns on clipping.

Select polyline
Creates a clipping boundary by closing a selected polyline.

Polygonal
Creates a polygonal boundary.

Undo
Undraws the last polygonal segment.

Rectangular
Creates a rectangular clipping boundary.

14.9 IMAGEFRAME command

Toggles the IMAGEFRAME system variable.

Shape Lite Pro Mechanical BIM

Icon:

14.9.1 Description

Toggle the IMAGEFRAME system variable to change the visibility and plotting for image frames. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'IMAGEFRAME.

- 0: sets the IMAGEFRAME system variable to 0.
- 1: sets the IMAGEFRAME system variable to 1.

- 2: sets the IMAGEFRAME system variable to 2.

14.10 IMAGEQUALITY command

Specifies the display quality of attached images.

Shape Lite Pro Mechanical BIM

Icon: 

14.10.1 Description

Specifies the display quality of attached images to increase performance or image resolution.

- Draft: increases performance by reducing color resolution, image size, and memory usage. This does not affect the quality of plotted images.
- High: increases image quality resulting in decreased performance for large images.

14.11 -IMPORT command

Imports geometry from external files at the command prompts.

Shape Lite Pro Mechanical BIM

Note: This command opens all file formats supported by the IMPORT command.

14.11.1 Method

Specify the full path and file name of an importable file.

Note: Type ~ (tilde) to display the Import file dialog box, which allows you to select a file to import.

Note: File types available for importing are the same as for the **IMPORT** command.

14.12 IMPORT command

Imports the geometry from external files into the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: IMP

14.12.1 Description

Opens the **Import file** dialog box to select a supported file type to import into the current drawing.

The supported file types to import are:

- **Drawing Exchange Format** (.dxf; .dwg)
- **Windows Metafile Formats*** (.wmf; .emf; .wmz; .emz)[(1) (2)]
- **Collada** (.dae)

- **Micro Station DGN File** (.dgn)

  Formats available with the BIM add-on:

- **Wavefront Object file** (.obj)
- **Rhino file**(.3dm)**(1) (2)**
- **SketchUp file** (.skp)**(1) (2)**

  Note: The Trimble SketchUp files are imported as subdivision meshes (entities are imported as named blocks, render material definitions and the material mappings are imported on the meshes). The ConvToSolid and ConvToMesh commands provide easy conversion between meshes and solids.

- **IFC file** (.ifc; .ifczip)
- **Revit family** (.rfa)
- **Revit Project** (.rvt)

  (1) Not available in BricsCAD® for Linux.

  (2) Not available in BricsCAD® for macOS.

Note: BricsCAD® currently supports the import of **Revit 2015-2021** files only. 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.

Note:

More 3D format files are available through a separate module, BricsCAD Communicator. You can download it from the Bricsys website.

The import and export procedures using Communicator are controlled through a series of user preferences. See the Communicator section in the Settings dialog box.

Note: If the imported assembly has physical materials assigned to its parts, the materials can be imported together with the assembly structure. For each material in the imported file, a corresponding material will be created in the target document material library, and its name, density, specific heat and thermal conductivity will be copied from the source material. If the imported file contains several materials with the same name, the first one will be used. If the document already has a material with the same name in its material library, this material will be used instead.

Materials copied to the document material library will not be erased with Undo operations.

14.13 IMPRINT command

Imprints 2D entities onto planar faces of 3D solids and surfaces to create additional edges.

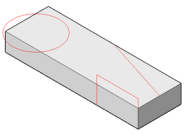
Shape Lite Pro Mechanical BIM

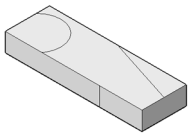
Icon: 

Note: Use dynamic UCS (DUCS) to draw on the faces of 3D objects (UCSDETECT system variable =1).

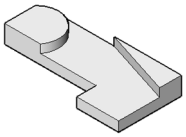
14.13.1 Method

Select a 3D solid that has at least one flat face, a surface or a region entity. Select the source objects, 2D entities lying on or intersecting a planar face of the selected entity and after each choose if delete or keep the source object.





With the entities imprinted on the solid, you can use commands like EXTRUDE or DMPUSHPULL to manipulate the newly created faces, as shown below.



14.14 INSERT command

Opens the Insert block dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: DDINSERT, I

14.14.1 Description

Opens the Insert block dialog box to insert a block instance from a block definition. The block definition may exist in the current drawing or as an external dwg file.

14.15 -INSERT command

Inserts blocks through prompts in the Command line. Also prompts for attribute values, if any are part of the block definition.

Shape Lite Pro Mechanical BIM

Alias: -I

14.15.1 Description

Inserts blocks through prompts in the Command line. And prompts for attribute values, if any are part of the block definition.

14.15.2 Method

Specify the name of the block to insert or press Enter to accept the name of the previously inserted block and specify the insertion point, the scale factor, the rotation angle for the inserted bock. BricsCAD prompts in the Command line the block inserted units, e.g. Units: Millimeters.

Note: The current drawing can be inserted as a bock by typing the drawing name as block name.

Note: If the block contains attributes, the value of the ATTDIA (Attribute Dialog) system variable determines whether you are prompted in the Command line to set the attributes (ATTDIA=0) or through the Edit Attributes dialog box (ATTDIA=1).

14.15.3 Options

? to list blocks in drawing

Lists the names of all blocks in the current drawing.
Enter part of a name to view the names of certain blocks. You can use wildcards like ? for an individual character and * for all characters.

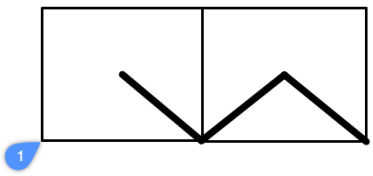
Note: If necessary, press F2 key to view the full list; see TEXTSCR command. You must restart the -INSERT command to insert the block.

~ to open the file dialog

Displays the Insert Block dialog box . Choose a DWG or DXF file and open it.

insertion point (1)

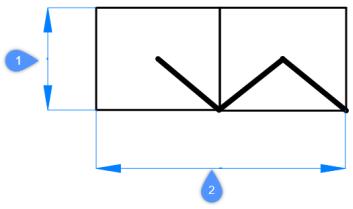
Specify the X-, Y- and Z- coordinates or pick a point for the block's insertion point (lower left corner). The Z coordinate is typically left at 0 for 2D blocks.



Scale

Scales the block:

- 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.



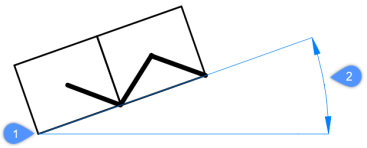
X scale (2)
Specifies the scale factor along the x axis; enter a negative value to mirror the block about the y axis.

Y scale (1)
Specifies the scale factor along the y axis; enter a negative value to mirror the block about the x axis.

Z scale
Specifies the scale factor along the z axis.

Rotate
Specifies the rotation angle (2) of the block about its insertion point (1), starting with the x-axis as 0 degrees:

- **Positive angles** - rotate the block counter-clockwise.
- **Negative angles** - rotate the block clockwise.



Multiple blocks
Inserts multiple session of the block in the specified insertion points.

Corner
Indicates the size of the block by specifying a second corner of a rectangle. The insertion point is the first corner.

Note: Blocks can be edited directly through grips.

14.16 INSERTALIGNED command

Inserts blocks aligned to entities.

Shape Lite Pro Mechanical BIM

Alias: INSAL

14.16.1 Description

Inserts blocks aligned with entities and optionally mirrors them. Hover the cursor over an existing entity in order to align the block to its geometry. Entity snap does not need to be on, but placement is more accurate when it is.

14.16.2 Options

Block to insert

- Enter block name
- ~ - opens a dialog box to choose an external drawing to insert
- ? - lists the existing block definitions.

Block mirroring control point
Moves the cursor in order to flip the block about its insertion point.

X/Y/Z Scale
Sets the corresponding scale of the inserted block.

Multiple
Inserts multiple instances of the block.



1 Block being inserted

2 Block aligned with entity

3 Block mirrored about the entity

14.17 INSERTION command

Toggles the Insertion entity snap.

Shape Lite Pro Mechanical BIM

Icon:

14.17.1 Description

Toggles the Insertion entity snap to enable or disable snapping to the insertion point of objects. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

14.18 INSERTOBJ command

Opens the Insert object dialog box.

Shape Lite Pro Mechanical BIM

Alias: IO

Note: This is a Windows only command.

14.18.1 Description

Opens the Insert object dialog box to insert an OLE object into the current drawing.

14.19 INTERFERE command

Shows volumes and areas of interference between two sets of ACIS entities.

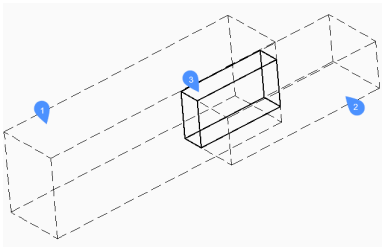
Shape Lite Pro Mechanical BIM

Icon:

Alias: INF

Note: This command can be entered transparently ('interfere) during the commands.

14.19.1 Method



Specifies the first (1) and the second (2) set of ACIS entities: choose one or more 3D solids and/or 2D regions.

You can respond with ALL to place all ACIS entities into a set and later use the Check first Set option to check them against each other. This obviates the need to create two sets of entities.

The entities in the first set are compared with the entities in the second set and results an interference volume (3) or area which is created in layer defined by INTERFERELAYER system variable that is stored in the registry and has initial value for "Interferences".

Note: The Pro edition additionally shows volumes of interference between two sets of ACIS solids and then, optionally, creates new ACIS solids from the common portions of pairs of intersecting solids, placing them on layer "Interferences".

The interference solid(s) remain(s) in the drawing when the command ends.

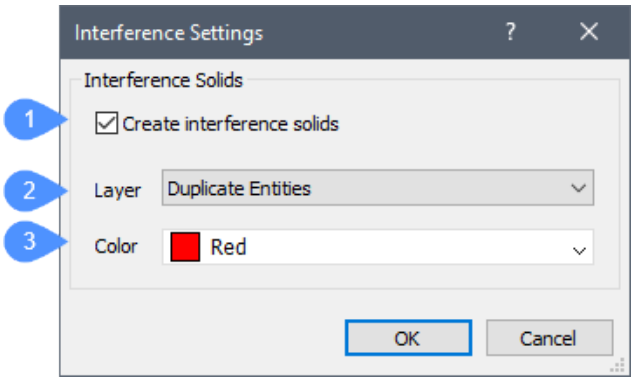
Note: Entities which are on the layer that is set by the INTERFERELAYER system variable are not accepted when selecting entities, as it is considered the output layer only.

Note: When you select an interference entity in the model or in the STRUCTURE panel , the originating entities highlight along with the selected interference entity.

14.19.2 Options

Nested selection
Selects ACIS entities inside blocks or xrefs.

Settings
Displays a dialog box:



1 **Create interference solids**: when ticked, new solids are created from the interference area or volume.

2 **Layer**: specifies the layer on which interference entities are created. Default = layer stored in INTERFERELAYER system variable. Do not select a layer of selected entities as it is considered the output layer only. It is recommended to select an empty layer or accept the default layer.

3 **Color**: specifies the color of the interference entities; the color of the Interference layer is changed to match this color.

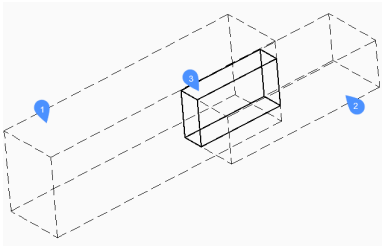
checK first set
Checks the interference of entities in the first set, when it contains two or more entities, and then creates interference entity(ies) of them.

14.20 -INTERFERE command

Shows volumes and areas of interference between two sets of ACIS entities.

Shape Lite Pro Mechanical BIM

14.20.1 Method



Specifies the first (1) and the second (2) set of ACIS entities: choose one or more 3D solids and/or 2D regions.

You can respond with ALL to place all ACIS entities into a set and later use the Check first Set option to check them against each other. This obviates the need to create two sets of entities.

The entities in the first set are compared with the entities in the second set and results an interference volume (3) or area which is created in layer defined by INTERFERELAYER system variable that is stored in the registry and has initial value for "Interferences".

Note: The Platinum edition additionally shows volumes of interference between two sets of ACIS solids and then, optionally, creates new ACIS solids from the common portions of pairs of intersecting solids, placing them on layer "Interferences".

The interference solid(s) remain(s) in the drawing when the command ends.

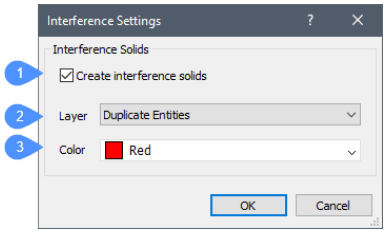
Note: Entities which are on the layer that is set by the INTERFERELAYER system variable are not accepted when selecting entities, as it is considered the output layer only.

Note: When you select an interference entity in the model or in the STRUCTURE panel , the originating entities highlight along with the selected interference entity.

14.20.2 Options

Nested selection
Selects ACIS entities inside blocks or xrefs.

Settings
Displays a dialog box:



1 Create interference solids - when ticked on, makes new solids from the interference area or volume.

2 Layer - specifies the layer on which interference entities are created. Default = layer stored in INTERFERELAYER system variable. Do not select a layer of selected entities as it is considered the output layer only. It is recommended to select an empty layer or accept the default layer.

3 Color - specifies the color of the interference entities; the color of the Interference layer is changed to match this color.

checK first set
Checks the interference of entities in the first set, when it contains two or more entities, and then creates interference entity(ies) of them.

Create interface solid?
Choose whether to create interface solid or not.

14.21 INTERSECT command

Performs Boolean intersection operations on 3D solids and 2D regions.

Shape Lite Pro Mechanical BIM

Icon:

Alias: IN

Note: In the BricsCAD® Lite license level, the command applies to region entities only.

14.21.1 Description

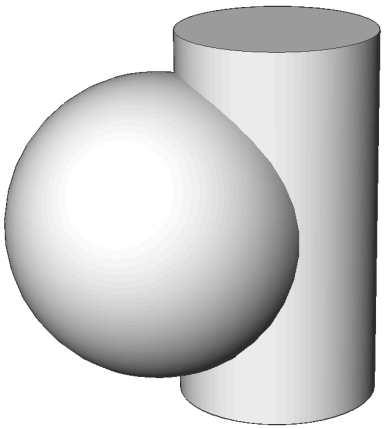
Performs Boolean intersection operations on 3D solids and 2D regions by removing all but portions in common.

Note: If solids and regions do not intersect, BricsCAD erases them.

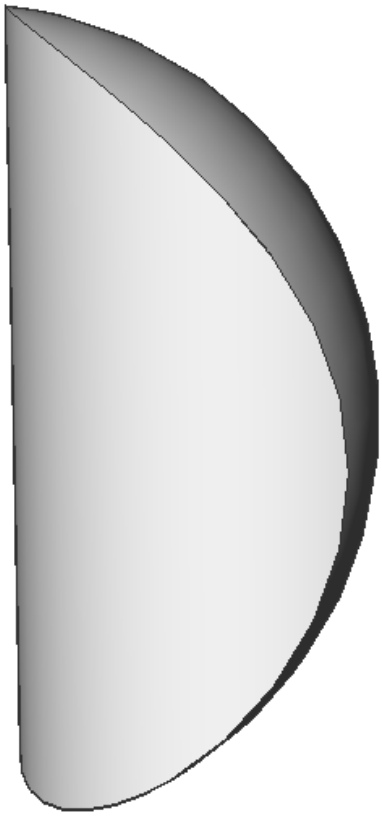
14.21.2 Method

Specify the 3D solids and/or 2D regions to be intersected. The program removes all but the volumes and areas in common from the selected entities.

The entities selected:



Result:



14.22 INTERSECTION command

Toggles the Intersection entity snap.

Shape Lite Pro Mechanical BIM

14.22.1 Description

Toggles the Intersection entity snap to enable or disable snapping to intersection. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

14.23 ISAVEAS command

Saves images.

Shape Lite Pro Mechanical BIM

14.23.1 Description

Saves images to your computer through the Save Image File dialog box.

14.24 ISOLATEOBJECTS command

Hides all but the entity(ies) selected.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: ISOLATE

14.24.1 Description

Only selected entities are displayed. Other entities are hidden.

Note: Use the UNISOLATEOBJECTS command to make the hidden entities visible again.

Note: The OBJECTISOLATIONMODE variable controls what happens with the hidden state of the entities that weren't selected to be hidden.

Note: The ISOLATEOBJECTS command is enabled under BEDIT and REFEDIT sessions.

14.25 ISOPLANE command

Toggles the SNAPISOPAIR system variable.

Shape Lite Pro Mechanical BIM

Alias: IS

14.25.1 Description

Toggles the SNAPISOPAIR system variable to specify the drafting plane for isometric drawings. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'ISOPLANE.

- Left: sets the SNAPISOPAIR system variable to Left.
- Right: sets the SNAPISOPAIR system variable to Right.
- Top: sets the SNAPISOPAIR system variable to Top.
- Toggle: changes the SNAPISOPAIR system variable to the next setting. It cycles in order from the current setting (left-top-right-left).

14.26 IUNEMBED command

Converts an embedded image to an attached one.

Shape Lite Pro Mechanical BIM

14.26.1 Description

Converts an embedded image into an attached image by exporting the embedded image to a file (which the image entity is then attached to).

Note: This command reverts the action of IEMBED command.

15. J

15.1 JOIN command

Joins 2D entities at their common endpoints.

Shape Lite Pro Mechanical BIM

Icon:

15.1.1 Description

Joins lines, lightweight and old-style 2D polylines, 3D polylines, arcs, elliptical arcs, polyarcs, splines, and helices at their common endpoints.

Note: The type of the resulting entity depends on the types of the input entities and on their coplanarity.

15.1.2 Method

Two 2D polylines are joined into a single:

- 2D polyline- when they are coplanar.
- 3D polyline- when they are not coplanar and if they are composed only of straight segments.
- Spline- when they are not coplanar and if at least one of the polylines has a bulged (arc) segment.

A 2D polyline and a 3D polyline are joined into a single:

- 2D polyline: when they are coplanar.
- 3D polyline: when they are not coplanar and if the 2D polyline only has straight segments.
- Spline: when they are not coplanar and if the 2D polyline has at least one arc segment.

A line and a circular arc (or a polyline with arcs) are joined into a single:

- 2D polyline: when they are coplanar.
- Spline: when they are not coplanar.

The following pairs of entities are joined into a spline:

- Line and an elliptical arc.
- Spline and another open entity, such as an elliptical arc or polyline.
- Helix and another open entity, such as a line or arc.

Even when there are gaps between these entities, the command joins them into a single entity:

- Collinear lines: joined into a single line.
- Coplanar arcs (with same radii and center points): joined into an arc or a circle.
- Coplanar elliptical arcs (with same major and minor axes): joined into an elliptical arc or ellipse.

Circular and elliptical arcs are joined in a counterclockwise direction starting from the source arc.

Note: Collinear means the entities lie in the same imaginary line. Coplanar means the entities lie in the same plane.

16. K

16.1 KEEPME command

Adds changed entities to the source drawing, when two somewhat different drawings are being compared visually.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The command works only during a session begun by the DWGCOMPARE command.

16.1.1 Method

Select one or more entities or else type ALL to select all entities in the drawing to add them to the source drawing.

17. L

17.1 LANDXMLEXPORT command

Exports Civil entities to a LandXML file.

Shape Lite Pro Mechanical BIM

17.1.1 Description

Allows you to select the BricsCAD Civil entities you want to export to a LandXML file. Once the entities are selected, the **Save LandXML file** dialog opens, where you can specify the location and name of the output LandXML file.

17.1.2 Method

Select Civil points, TIN surfaces, gradings or Horizontal Alignments and 3D Alignments in the drawing and specify the output file.

17.2 LANDXMLIMPORT command

Creates TIN surfaces or alignments from a LandXML file.

Shape Lite Pro Mechanical BIM

Icon: 

17.2.1 Description

Opens the **Open LandXML file** dialog box to select an .xml file to import.

The following BricsCAD Civil entities can be imported from the specified LandXML file: Civil Points, Surfaces, Horizontal Alignments (both Alignments by PI and Alignments by Elements are supported), and 3D Alignments.

17.2.2 Options within the command

Draw Breaklines as Polylines
Specifies if breaklines are imported as polylines.

Note: This option is available when importing TIN surfaces with breaklines.

The units in the drawing are different than those in the XML file. What would you like to do:
This option is available if the units in the current drawing do not match the units in the input XML file.

Scale
Scales the XML file units.

Import without scaling
Imports the XML file without scaling.

Cancel
Cancels the command.

17.3 LAYCUR command

Moves selected entities to the current layer.

Shape Lite Pro Mechanical BIM

Icon: 

17.3.1 Description

Moves selected entities to the current layer without having to specify the name of the current layer.

You are prompted:

Select entities to be moved to the current layer

Select one or more entities you want to move to the current layer. You can continue selecting entities until you press Enter to end the command. The Command line indicates how many entities were moved and to what layer:

 # entities were moved to the current layer ("LayerName1").

17.4 -LAYER command

Manages layers through the Command line.

Shape Lite Pro Mechanical BIM

Alias: -LA

17.4.1 Description

Creates layers and layer states and changes their properties.

Note: The current layer cannot be turned off and frozen.

17.4.2 Options within the command

?
Lists the names of layers in the current drawing.

New layer
Creates new layers.

Note: You can create several new layers by separating each name with a comma (,).

Make new current layer
Creates a new layer, and then makes it current.

Set layer as current
Sets a layer as the current one.

Rename
Renames the layer.

Color
Changes the color of layers.

Linetype
Changes the linetype of layers.

LineWeight
Changes the lineweight of layers.

TRansparency
Sets the entity transparency factor between 0 (fully opaque) and 90 (fully transparent).

MATerial
Sets the material property for all entities on the layer.

Plot
Changes the plot status of layers.

stAte
Manages layer states.

Save
Saves the current layer state.

Restore
Restores a saved layer state.

Edit
Edits a saved layer state.

reName
Renames a saved layer state.

Delete
Deletes a saved layer state.

Import
Displays the Import Layers States dialog box, which allows you to import a layer state from a file into the current drawing.

eXport
Exports a saved layer state in the current drawing to a layer state file.

turn layer ON
Turns layers on that were previously turned off.

turn layer OFf
Turns layers off to hide their entities from view.

Freeze
Freezes layers so that they are hidden from view.

Thaw
Thaws layers that were previously hidden with the Freeze option.

LOck
Locks layers so that their entities remain visible but cannot be edited.

Unlock
Unlocks layers that were previously locked.

17.5 LAYER command

Opens the Layers panel.

Shape Lite Pro Mechanical BIM

Alias: DDLMODES, LA

17.5.1 Description

Opens the Layers panel to display it in the current workspace. The Layers panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Layers panel can be either floating, docked or stacked.

17.6 LAYERP command

Restores layer properties to their previous state.

Shape Lite Pro Mechanical BIM

Icon: 

17.6.1 Description

Restores layer properties to their previous state one change at a time. This command only works when the LAYERPMODE system variable is enabled.

There are no prompts. The Command line indicates: Previous layer settings were restored.

17.7 LAYERSPANELCLOSE command

Closes the Layers panel.

Shape Lite Pro Mechanical BIM

17.7.1 Description

Closes the Layers panel to hide it from the current workspace. If the Layers panel is stacked when you close it, the Layers tab or icon is removed from the stack.

17.8 LAYERSPANELOPEN command

Opens the Layers panel.

Shape Lite Pro Mechanical BIM

17.8.1 Description

Opens the Layers panel link to display it in the current workspace. The Layers panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Layers panel can be either floating, docked or stacked.

17.9 LAYERSTATE command

Opens the Drawing explorer dialog box with **Layer States** selected.

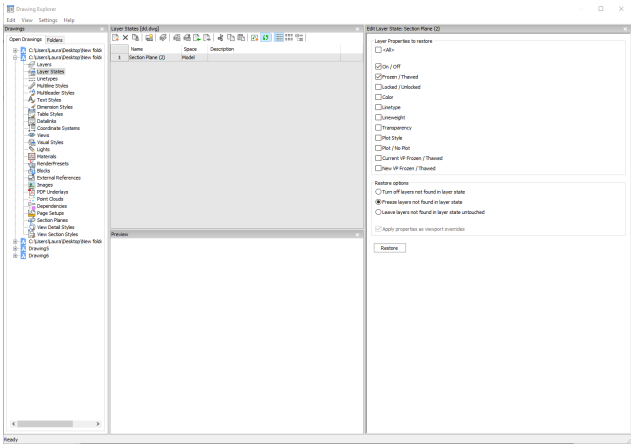
Shape Lite Pro Mechanical BIM

Icon:

Alias: LAS

17.9.1 Description

Opensthe Drawing explorer dialog box on the Layer States category to manage layer states of layers that are used in the selected drawing.



17.9.2 Options within the command

<All>

Selects or deselects all properties. See the LAYER command for the meaning of the properties

Restore options

Determines what happens with layers not found in the layer state such as those added after the layer state was created.

Apply properties as viewport overrides

Applies the layer state as viewport overrides (see VP Layer properties) to the current layout viewport.

Note: This option is only available on a layout within a layout viewport.

17.9.3 Context Menu Options

New

Creates a new layers state with a generic name.

Delete

Deletes the selected layer state from the drawing.

Note: Layer states that are used in the drawing will be deleted without warning.

Hide xref symbols

Toggles the visibility of xref symbols on and off.

Edit Layer State
Displays the Edit Layer State dialog box to change the status of the layer state.

Add
Adds layers to layer state

Remove
Remove layers from layer state.

Restore
Restores the selected layer state to the default state.

Note: The properties of the layer will change after closing the Drawing Explorer dialog box.

Overwrite
Overrides the settings of the selected layer state with new settings changed by the Edit Layer State panel.

Import
Imports layer states from a LAS file through the Layer States dialog box.

Export
Exports layer states to a LAS file through the Layer States dialog box.

Note: The layer state file can be imported into other drawings or send to clients.

Rename
Renames the item.

Select All
Selects all items.

Invert selection
Deselects the current selection and vice versa.

17.10 LAYFRZ command

Freezes layers of selected entities.

Shape Lite Pro Mechanical BIM

Icon:

17.10.1 Description

Freezes layers of selected entities to hide all entities that are on the same layers as the selected entities.

17.10.2 Method

You are prompted:

Select an entity on the layer to be frozen

Select one or more entities on layers you want to freeze. You can continue selecting entities until you press Enter to end the command. The Command line indicates which layers have been frozen:

- **This layer has been frozen: LayerName1**
- **This layer has been frozen: LayerName2**

If any of the selected entities are on the current layer, the Command line indicates that layer can't be frozen:

Cannot freeze layer: LayerName3. It is the current layer.

Additional options: [Settings/Undo/selection options (?)]

17.10.3 Optionswithin theLAYFRZcommand

After you begin the LAYFRZ command, the following options may be available:

Settings
Choose to modify viewport or block selection settings.

Viewports
Specify the behavior for layout viewports.

- **Off**: freezes layers in all viewports. This is the default option.
- **VPFreeze**: freezes layers in the current viewport.

Block selection
Specify the behavior for blocks and external references.

- **Selection**: freezes the layer of selected entities including blocks and external references. It ignores layers of entities within the block or external reference regardless of where you pick the block. This is the default option.
- **Block**: freezes the layer of the selected block/xref or nested block/xref. This option requires you to select entities by picking each one individually. Where you pick, on a parent or nested block/xref, determines which layer is frozen.
- **Entity**: freezes the layer of the selected entity within a block/xref or nested block/xref. This option requires you to select entities by picking each one individually. Where you pick, on a parent or nested block/xref, determines which layer is frozen.
- **None**: this option behaves the same as the Selection option.

Undo
Undo the previous LAYFRZ operation.

Selection options

17.11 LAYISO command

Isolates the layers of selected entities.

Shape Lite Pro Mechanical BIM

Icon: 

17.11.1 Description

Isolates the layers of selected entities lock or hide all entities except those that are on the same layers as the selected entities.

17.11.2 Method

You are prompted:

Select entities on the layer(s) to be isolated

Select one or more entities on layers you want to isolate. You can continue selecting entities until you press Enter to end the command. If all selected entities are on the same layer, the Command line indicates which layer has been isolated and makes it the current layer:

This layer has been isolated: LayerName1. This layer is current: LayerName1.

If the selected entities are on more than one layer, the Command line indicates how many layers have been isolated.

Number of isolated layers: #.

If the selected entities don't include one from the current layer, the current layer is changed to one of the isolated layers.

Additional options: [Settings/selection options (?)]

17.11.3 Optionswithin theLAYISOcommand

Settings

Specify the behavior of layers that are not isolated.

Lock

Lock layers that are not isolated. This is the default option.

Off

Hide entities on layers that are not isolated. You can specify how entities are hidden for layout viewports.

- **Off**: freezes layers in all viewports. This is the default option.
- **VPFreeze**: freezes layers in the current viewport.

Selection options

17.12 LAYLCK command

Locks the layer of a selected entity.

Shape Lite Pro Mechanical BIM

Icon: 

17.12.1 Description

Opens the Layers panel to display it in the current workspace. The Layers panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Layers panel can be either floating, docked or stacked.

17.12.2 Method

Lock the layer of a selected entity to prevent entities on that layer from being edited.

You are prompted:

Select an entity on the layer to be locked

Select an entity on the layer you want to lock.

The Command line indicates which layers have been locked:

- **This layer has been locked: LayerName1**
- **This layer has been locked: LayerName2**

Entities on locked layers are faded by default. You can change layer fading with the LAYLOCKFADECTL system variable.

17.13 LAYMCUR command

Changes the working layer to that of the entity selected (short for "layer make current").

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SETLAYER

17.13.1 Method

Choose one entity whose layer will be made the current layer.

17.14 LAYOFF command

Turns off layers of selected entities.

Shape Lite Pro Mechanical BIM

Icon: 

17.14.1 Description

Turns off layers of selected entities to hide all entities that are on the same layers as the selected entities.

You are prompted:

17.14.2 Select an entity on the layer to be turned off

Select one or more entities on layers you want to turn off. You can continue selecting entities until you press Enter to end the command. The Command line indicates which layers have been turned off:

- **This layer has been turned off: LayerName1**
- **This layer has been turned off: LayerName2**

If any of the selected entities are on the current layer, you are prompted:

This layer is current: LayerName3. Do you want to turn off current layer?

Specify if you want to turn the current layer off.

- **Yes**: turn off the current layer.
- **No**: do not turn off the current layer.

Additional options: [Settings/Undo/selection options (?)]

17.14.3 Optionswithin theLAYOFFcommand

After you begin the LAYOFF command, the following options may be available:

Settings
Choose to modify viewport or block selection settings.

Viewports
Specify the behavior for layout viewports.

- **Off**: turns off layers in all viewports. This is the default option.
- **VPFreeze**: freezes layers in the current viewport.

Block selection
Specify the behavior for blocks and external references.

- **Selection**: turns off the layer of selected entities including blocks and external references. It ignores layers of entities within the block or external reference regardless of where you pick the block. This is the default option.
- **Block**: turns off the layer of the selected block/xref or nested block/xref. This option requires you to select entities by picking each one individually. Where you pick, on a parent or nested block/xref, determines which layer is turned off.
- **Entity**: turns off the layer of the selected entity within a block/xref or nested block/xref. This option requires you to select entities by picking each one individually. Where you pick, on a parent or nested block/xref, determines which layer is turned off.
- **None**: this option behaves the same as the Selection option.

Undo
Undo the previous LAYOFF operation.

Selection options

17.15 LAYON command

Turns on all layers in the drawing.

Shape Lite Pro Mechanical BIM

Icon: 

17.15.1 Description

Turns on all layers in the drawing to view and edit entities on those layers.

There are no prompts. The Command line indicates: all layers have been turned on.

Note: Entities on frozen layers are only visible if you also thaw the layer. Entities on locked layers are only editable if you also unlock the layer.

17.16 LAYOUT command

Creates, copies, renames, and deletes layouts.

Shape Lite Pro Mechanical BIM

Icon: 

17.16.1 Method

You can create up to 255 layouts in a single drawing. Each layout represents a sheet of paper.

17.16.2 Options within the command

?
Lists the names of layouts already defined in the drawing.

Set
Switches to the specified layout.

New
Creates new layouts.

Copy
Creates a new layout by making a copy of an existing layout.

Delete
Erases layouts from the drawing.

Rename
Renames layouts.

SAve
Saves the mentioned layout as DWG or DXF files format.

Template
Imports layouts from DWG, DWF, or DXF drawing files. The Insert Layout(s) dialog box displays, which allows you to select a layout name.

neXt
Displays the next layout.

Previous
Displays the previous layout.

17.17 LAYOUTMANAGER command

Opens the Layout manager dialog box.

Shape Lite Pro Mechanical BIM

17.17.1 Description

Opens the Layout manager dialog box to view and manage layouts in the current drawing.

17.18 LAYTHW command

Thaws all layers in the drawing.

Shape Lite Pro Mechanical BIM

Icon: 

17.18.1 Description

Thaws all layers in the drawing to view and edit entities on those layers.

There are no prompts. The Command line indicates: all layers have been thawed.

Note: Entities on layers are turned off are only visible if you also turn on the layer. Entities on locked layers are only editable if you also unlock the layer.

17.19 LAYTRANS command

Opens the Layer translator dialog box.

Shape Lite Pro Mechanical BIM

17.19.1 Description

Opens the Layer translator dialog box to translate layer properties by mapping to other layers.

17.20 LAYULK command

Unlocks the layer of a selected entity.

Shape Lite Pro Mechanical BIM

Icon: 

17.20.1 Description

Unlocks the layer of a selected entity to enable entities on that layer to be edited.

17.20.2 Method

You are prompted:

Select an entity on the layer to be unlocked

Select an entity on the layer you want to unlock. The Command line indicates which layers have been locked:

- **This layer has been locked: LayerName1**
- **This layer has been locked: LayerName2**

17.21 LAYUNISO command

Restores isolated layers to their previous state.

Shape Lite Pro Mechanical BIM

Icon: 

17.21.1 Description

Restores the Lock, On/Off, and VP Freeze properties of isolated layers as they were before you used the LAYISO command.

There are no prompts. The Command line indicates: Layers isolated by LAYISO command have been restored.

17.22 LCONNECT command

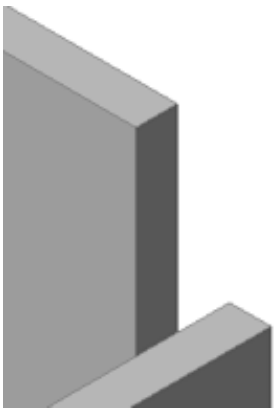
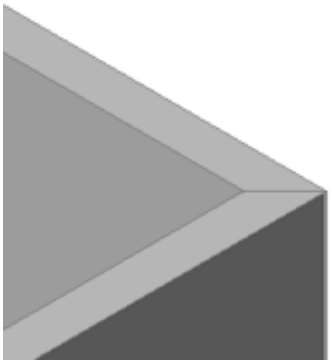
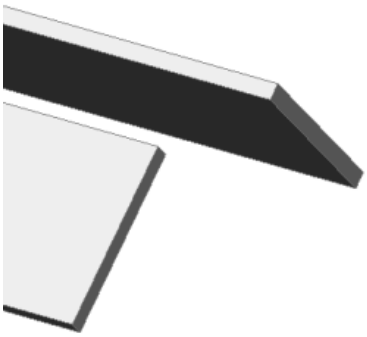
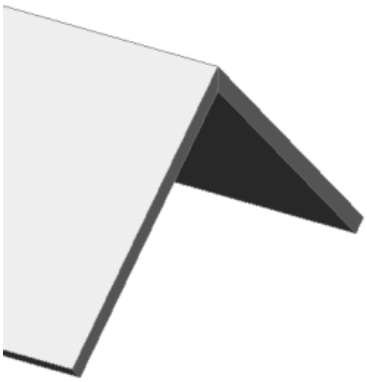
Creates or modifies an L-connection between solids.

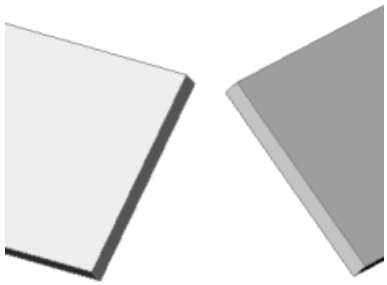
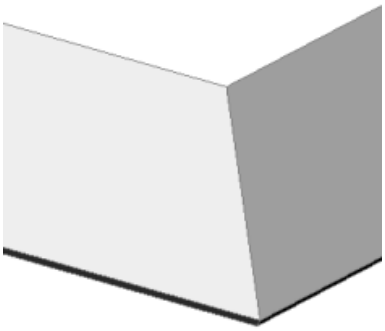
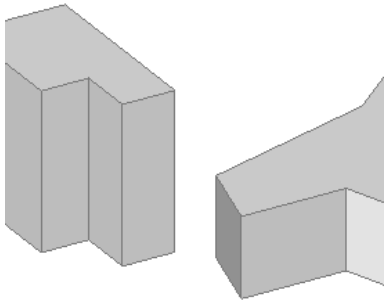
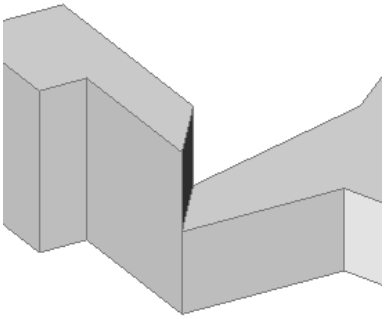
Shape Lite Pro Mechanical BIM

Icon:

17.22.1 Description

Creates or modifies an L-connection between solids. Optionally disconnects L-connected solids.

Before	After
	
	

Before	After
	
	

17.22.2 Options

Select entities to connect
Manually select the two entities you want to connect.

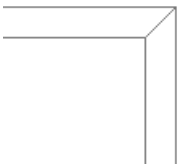
Switch
Allows to choose between two butted L-connections:



or



or a mitered L-connection:



or a touching (disconnected) connection:



Note: If the HOTKEYASSISTANT is set ON, press Ctrl key to cycle between the connection types.

17.23 LEADER command

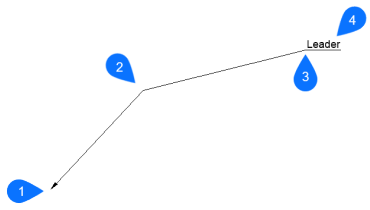
Draws leaders.

Shape Lite Pro Mechanical BIM

Alias: LE, LEAD

17.23.1 Description

Draws leaders by specifying several points.



1 Start point

2 Next point

3 To point

4 Annotation

Note: After you start creating a leader, options regarding the annotation may be available.

17.23.2 Options

Format
Specifies the leader style.

Arrow
Draws an arrowhead at the leader's start point (default).

None
Does not draw an arrowhead.

SPline
Draws the leader as a spline.

STraight
Draws the leader as a straight-line segment (default).

Undo
Undoes the last leader line segment.

Annotation
Starts adding text to the end of the leader line.

Note: The annotation is created as an MTEXT entity.

Note: The annotation is independent of the leader line. When moving a leader, make sure to include the annotation in the selection set.

Block
Select a block in the drawing or upload a block file from the computer.

Copy
Select a Mtext, text, block reference or tolerance object from the drawing to use as annotation of the leader.

None
The leader command closes without annotation.

Tolerance
Enter the annotation as a tolerance notation through the geometric tolerance dialog box.

Mtext
Enter annotation as Mtext through a text formatting toolbar.

17.24 LENGTHEN command

Changes the length of open objects such as lines, polyline segments and arcs.

Shape Lite Pro Mechanical BIM

Icon:

Alias: EDITLEN, LEN

17.24.1 Method

There are four methods to change an object's length.

- Dynamic
- Increment
- Percent
- Total length

Note: By selecting an object, the current length is listed in the Command line.

17.24.2 Options

DYnamic
Specify a point to define the starting point of the object.

Note: The direction of the object won't change.

edit Mode
Returns to original prompt to switch lengthening modes.

Increment
Changes the length by a specified amount.

Angle
Changes the angle by a specified amount.

Percent
Changes the length of entities by a percentage.

Note: : For example, enter 25 (for 25%), and a 1"-long line is shorted to 0.25". Enter 150%, and the 1" line becomes 1.5".

Total
Enter the new total length of the object.

17.25 LIBRARYPANELCLOSE command

Closes the Library panel.

Shape Lite Pro Mechanical BIM

17.25.1 Description

Closes the Library panel to hide it from the current workspace. If the Library panel is stacked when you close it, the Library tab or icon is removed from the stack.

17.26 LIBRARYPANELOPEN command

Opens the Library panel.

Shape Lite Pro Mechanical BIM

17.26.1 Description

Opens the Library panel to display it in the current workspace. The Library panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Library panel can be either floating, docked or stacked.

17.27 LICENSEMANAGER command

Opens the BricsCAD license manager dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

17.27.1 Description

Opens the BricsCAD License manager dialog box to view and manage your BricsCAD license.

17.28 LIGHT command

Places light glyphs in drawings for generating more realistic renderings.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: LIGHTING

Note: Set DEFAULTLIGHTING system variable OFF to take into consideration the active lights sources defined in the drawing. Otherwise, only the default lighting will be used.

17.28.1 Options

Point
Creates a point light.

Note: See the POINTLIGHT command for the meaning of its options.

Spot
Creates a spot light.

Note: See the SPOTLIGHT command for the meaning of its options.

Web
Creates a web light.

Note: See the WEBLIGHT command for the meaning of its options.

Distant
Creates a distant light.

Note: See DISTANTLIGHT command for the meaning of its options.

17.29 LIGHTLIST command

Opens the Drawing explorer dialog box with **Lights** selected.

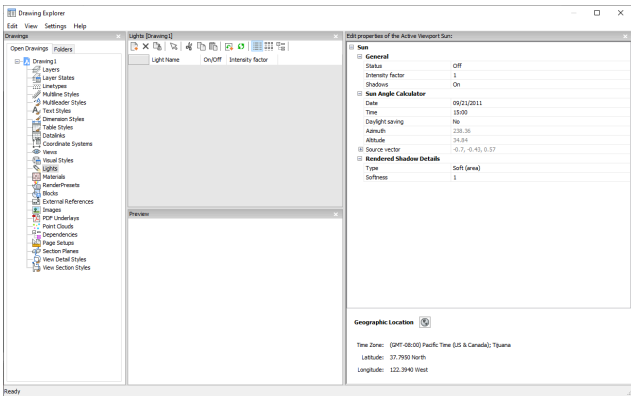
Shape Lite Pro Mechanical BIM

Icon: 

Alias: LL

17.29.1 Description

Opensthe Drawing explorer dialog on the Lights category to manage the lights in the selected drawing.



17.29.2 Options within command

General
Defines general settings of the sun.

Intensity factor
Specifies an intensity factor that makes the light brighter or dimmer, at its source.

Sun Angle Calculator
Defines the sun angle by entering date, time and location information.

Rendered Shadow Details
Defines shadow rendering settings.

Graphic Location
Defines the graphic location.

Note: For more information about the properties of a distance light named Sun, which simulates sunlight, see SUNPROPERTIES command.

17.29.3 Context Menu Options

New
Creates a new layers state with a generic name.

Delete
Deletes the selected layer state from the drawing.

Note: Layer states that are used in the drawing will be deleted without warning.

Select lights in the drawing
Selects a light through the model space of the drawing.

Rename
Renames the item.

Select All
Selects all items.

Invert selection
Deselects the current selection and vice versa.

17.30 LIMITS command

Sets an imaginary boundary for the extents of the drawing and optionally, of the grid.

Shape Lite Pro Mechanical BIM

Icon: 

17.30.1 Description

The command creates an imaginary rectangle boundary inside the drawing area.

Drawing outside of the area defined by this command is restricted when the LIMCHECK system variable is On. "Selected point is outside limits. Please select another point." will be reported in the Command line.

The imaginary boundary also limits the grid display when the first flag of the GRIDDISPLAY variable is not set.

17.30.2 Options

Lower left corner
Specifies the lower left corner of the limits.

Upper right corner
Specifies the upper right corner of the limits.

ON
Turn on the limits for your drawing area.

OFf
Turn off the limits for your drawing area.

17.31 LINE command

Creates line segments.

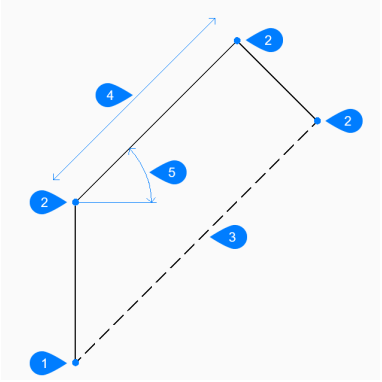
Shape Lite Pro Mechanical BIM

Icon: 

Alias: 3DLINE, L

17.31.1 Description

Creates a series of individual line entities by specifying the start and endpoint of each segment. Options allow you to specify the angle, undo, and close the geometry.



1 Start

2 End

3 Close

4 Length

5 Arc

17.31.2 Methods to begin a line

This command has 3 methods to begin creating a line segment:

- Start of line
- Last point
- Follow

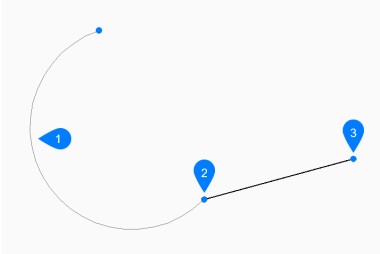
You can continue adding unlimited segments until you press Enter to end the command.

Start of line
Begin creating a line by specifying the start point then:

Set end point
Specify the end point of the line segment.
Additional options: [Angle/Length/Follow/Close/Undo]

Last point
Begin creating a line from the last point picked then:

Set end point
Specify the end point of the line segment.



1 Last drawn arc

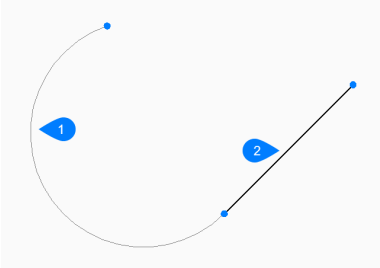
2 Last point

3 End point

Additional options: [Angle/Length/Follow/Close/Undo]

Follow
Begin creating a line from the last drawn arc or line segment, following its angle, then:

Length of line
Specify the length of the line. Picking the endpoint only determines the length since the angle follows the previous segment.



1 Last drawn arc

2 Follow

Additional options: [Angle/Length/Follow/Close/Undo]

17.31.3 Options within the LINE command

After you begin creating a line, the following options may be available:

Angle
Specify the angle of the line segment.

Length
Specify the length of the line segment.

Undo
Undo the last line segment and continue drawing from its previous start point.

Close
Automatically draw a line from the end point of the last segment to the start point of the first segment.

17.32 -LINETYPE command

Loads, sets, and creates linetypes in the Command line.

Shape Lite Pro Mechanical BIM

Alias: -LT

17.32.1 Method

You can create a new linetype by specifying a description (up to 47 characters long) and a linetype definition.

The linetype definition consists on a series of numbers separated by commas:

- Dashes: are shown by positive numbers.
- Spaces: are shown by negative numbers.
- Dots: are shown by number zero.

Note: After creating a new linetype, you need to load it to be accessible.

17.32.2 Options within the command

?
Displays the Select Linetype File dialog box, which allows you to select a LIN file.

Create
Displays the Create or Append Linetype File dialog box, which allows you to select a LIN file to add a new linetype.

Load
Displays the Select Linetype File dialog box, which allows you to load a linetype definition.

Set
Sets current a loaded linetype.

17.33 LINETYPE command

Opens the Drawing explorer dialog box with **Linetypes** selected.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: DDLTYPE, EXPLTYPES, LT

17.33.1 Description

Opens the**Drawing explorer** dialog boxwith the **Linetypes**category selected to view and modifylinetypes in the current drawing.

New drawings contain at least these linetypes: Continuous, ByLayer, and ByBlock.

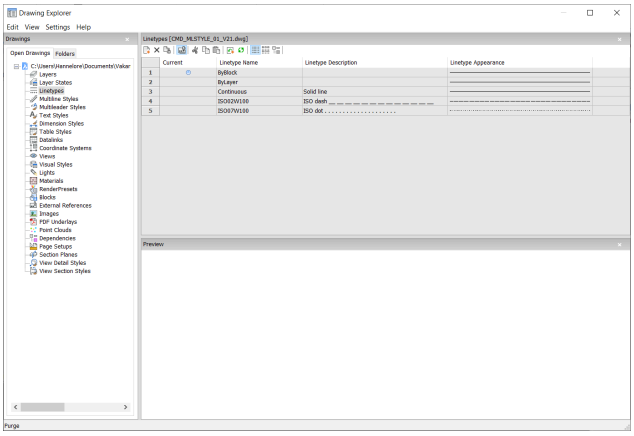
- **Continuous** linetype: displays entities with a solid, unbroken line.
- **ByLayer** linetype: displays entities with the linetype assigned to the current layer.
- **ByBlock** linetype: displays entities with the **Continuous** linetype until the entities are combined into a block definition.

Note:

- If you need to control the linetype of a certain part of a block, you can assign the **ByBlock** value to that part of the block. This means that this part of the block does not have any value assigned until it is inserted into a drawing. You should assign the **ByBlock** value to entities before creating the block or change it in the block editor after.
- When the block is inserted into the drawing, it displays the current linetype of the drawing for those entities.

Any other linetype must be loaded into the drawing before it can be used: click the **New** button to load the linetype.

To start new drawings with all linetypes loaded, create and save the drawing as a DWT template file.



17.33.2 Context Menu Options

New
Loads additional linetypes into the drawing. Displays the **Load Linetypes** dialog box.

Delete
Deletes linetypes from the drawing.

Note: The following linetypes cannot be deleted:

- Continuous
- ByLayer
- ByBlock
- Any linetype in use

Rename
Renames the selected linetype.

Note: The following linetypes cannot be renamed:

- ByBlock
- ByLayer
- Continuous

Select all
Selects all linetypes.

Invert selection
Deselects the current selection and vice versa.

Set Current
Sets the selected linetype as the current one.

Hide xref symbols
Toggles the visibility of xref symbols on and off.

17.34 LIST command

Lists the properties of selected entities.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: LI, LS

17.34.1 Description

Returns a list of properties of the selected entities. A long lists of data could be generated, so press F2 to switch to the Prompt History window. When listing gets too long, press Esc to end it.

Following properties are listed for each entity:

- Entity type
- Layer, Color, Linetype, lineweight: if these properties are not specifically defined, the information will be set to 'ByLayer'.
- Bounding box coordinates

17.34.2 Method

This command offers two methods to display the properties of a selected entity.

- Selecting entities before running the LIST command: Returns the properties of the selected entity.
- Running the LIST command before selecting entities: Allows to choose between different options before selecting the entities.

17.34.3 Options

SOrt
Entities will be listed in sort of varying properties.

SEquential
Entities will be listed in the order by which they were selected.

Tracking
Specifies the number of lines of text to be displayed.

add to set (+)
Extra entities can be added, to list their properties.

subtract from set (-)
Allows to select the entity you want to remove from the selection.

Previous selection
Select the option in the Command line to go back to the previous selection.

select by PROperties
Lists the properties of entities with the same specified properties. Entities with the same property value for properties such as color, layer, linetype, name, thickness, type, value, width, handle, location, can be selected.

selection methoDs
Changes the selection method, same as the PICKAUTO system variable works.

17.35 LIVESECTION command

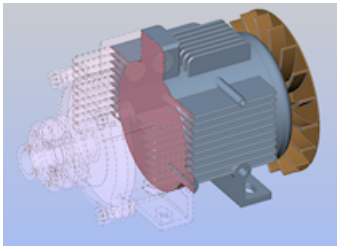
Toggles section views on/off.

Shape Lite Pro Mechanical BIM

Icon:

17.35.1 Description

Turns the display of the Live Section property of section planes on and off. When on, the insides of 3D models can be viewed.



Live sections differ from regular section planes by being interactive. This command requires at least one section plane in a drawing that was made with the SECTIONPLANE command.

Note: If the section was on, then it is turned off -- and vice versa.

Note: It is recommended to use the Clip Display property instead of Live Section. The Clip Display status can be set for multiple section entities simultaneously.

17.36 LOAD command

Opens the Load shape file dialog box.

Shape Lite Pro Mechanical BIM

17.36.1 Description

Opens the Load shape file dialog box to select an shx file to load into the current drawing.

17.37 LOFT command

Creates 3D solids or 3D surfaces.

Shape | Lite | Pro | Mechanical | BIM

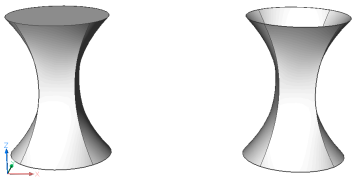
Icon:

17.37.1 Description

Creates 3D solids or 3D surfaces passing through 2 selected cross sections.

17.37.2 Options

Mode
Determines if the resulting loft is a solid or a surface.



Solid
Creates lofts as solid.

Surface
Creates lofts as surfaces.

Guides
Uses both cross-sections and guide lines between the selected cross sections to create the loft.

Note: When DELOBJ system variable is set to 2, the selected guide entities are deleted.

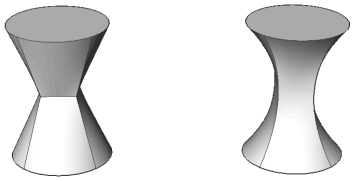
Path
Specify the path curve.

Cross sections only
Uses only cross section entities to create the loft, and no guides.

Settings
Sets variables that affect how the loft is constructed.

Ruled
Draws straight surfaces between the cross sections, features sharp edges at each cross-section.

Smooth fit
Draws smooth surfaces between the cross sections.



Normal to

Draws surfaces normal to the chosen cross sections. Select one of the options to become one of the following results:

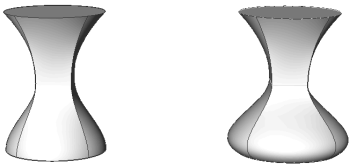
- Start
- Both
- End
- All



Draft angles

Specifies the angles at the starting and ending cross-sections to modify the loft shape.

Note: The angle option specifies the angle at which the loft starts from a cross section. The magnitude option sets the relative distance of the surface from the cross section in the direction of the draft angle before surface starts to bend towards next section.



17.38 LOGFILEOFF command

Turns off log file recording.

Shape Lite Pro Mechanical BIM

17.38.1 Description

Turns off log file recording.

Note: Log files record all prompts from the program and all of your keyboard input. It does not record mouse actions or other non-keyboard activities.

17.39 LOGFILEON command

Turns on log file recording.

Shape Lite Pro Mechanical BIM

17.39.1 Description

Turns on log file recording.

Note: Log files record all prompts from the program and all your keyboard input. It does not record mouse actions or other non-keyboard activities.

BricsCAD records all command text to the LOG file located in the folder specified by the LogFilePath system variable. The default name of the file is "drawingname_year-month-date_hour-minutes-seconds.log", such as drawing1_2029-08-31_08-32-46.log. Setting the LOGFILEMODE system variable to 1 has the same effect as the LOGFILEON command. You can open the resulting log file in Notepad or another text editor.

17.40 -LOGIN command

Authenticates to Bricsys server.

Shape Lite Pro Mechanical BIM

17.40.1 Description

Opens the Login dialog box to authenticate to Bricsys server.

17.41 -LOGOUT command

Logs out of the Bricsys account.

Shape Lite Pro Mechanical BIM

17.41.1 Description

Logs the user out of the Bricsys account, which was used to access online services.

17.42 LOOKFROM command

Toggles LookFrom widget.

Shape Lite Pro Mechanical BIM

Alias: NAVVCUBE

17.42.1 Description

Changes the display of the LookFrom widget between on and off, and accesses the settings for displaying the widget.

17.42.2 Methods

There are two methods to control the widget:

- ON/OFf - toggle the widget's display.
- Settings - adjust the widget's look, by accessing the LookFrom Control section of the Settings dialog box.

17.43 LWEIGHT command

Opens the Settings dialog box with the **Lineweights** category expanded.

Shape Lite Pro Mechanical BIM

17.43.1 Description

Opens the Settings dialog box with the **Lineweights** category expanded to view and modify relevant system variables.

18. M

18.1 MAIL command

Opens the default email client.

Shape Lite Pro Mechanical BIM

18.1.1 Description

Opens the default email client to automatically start a new email with the current drawing attached. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

18.2 MANIPULATE command

Launches the manipulator widget to rotate, move, copy, mirror, and/or scale 2D entities and 3D models.

Shape Lite Pro Mechanical BIM

Icon:

18.2.1 Methods

There are multiple methods to access the manipulator.

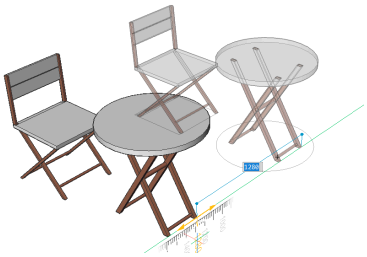
- Manipulate command
- Select entities and press enter.
- Quad
- Long press an object
- Make sure that the MANIPULATOR system variable is set correctly.

18.2.2 Options

Select an axis

Moves the selection of entities along the selected axis.

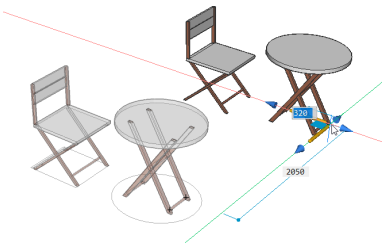
Select an axis and enter the value of the displacement or define the new position by specifying a point.



Select a plane

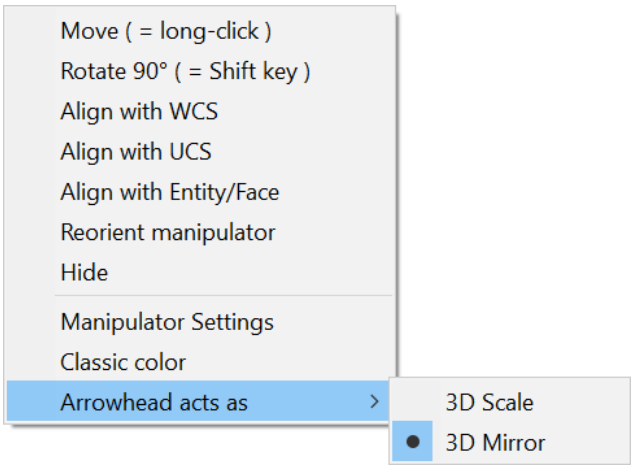
Moves the selection of entities in the selected plane.

Select a plane and enter the value of the displacement or define the new position by specifying a point.



Select an arrowhead

Mirrors or scales the selection. By right-clicking when hovering over the manipulator widget, a contexts menu with other options displays. Here, 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.

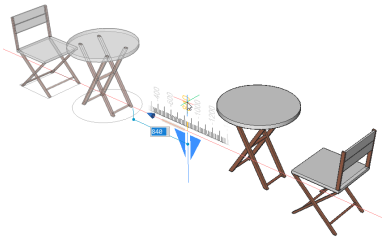


Figure 1: 3D Mirror

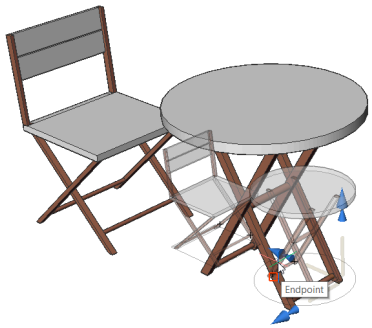
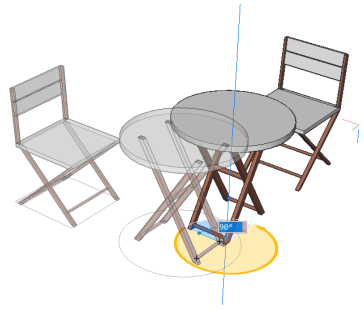


Figure 2: 3D Scale

Select a rotation arc

Rotates the selection around one of the manipulator axes. Enter the rotation angle value or specify by clicking on a point in the drawing.



Select the anchor handle

- If MANIPULATORHANDLE = 0: Moves the manipulator. Specify a point to relocate.
- If MANIPULATORHANDLE = 1: Moves the selected entities unrestrictedly.
- Long-click the manipulator handle and move it to a new location.

Note: To copy the original entity so that a new entity is created, hold down the Crtl key before starting the axis or make sure the copy option is enabled.

18.3 MAPCONNECT command

Allows to set up a connection with a Web Map Service.

Shape Lite Pro Mechanical BIM

18.3.1 Description

Displays the Map connect dialog box.

Note: Before adding map data to a drawing, you must run the GEOGRAPHICLOCATION command to define the geographic location of the drawing.

18.4 MAPTRIM command

Trims entities by a selected boundary.

Shape Lite Pro Mechanical BIM

Icon:

18.4.1 Description

Offers an easy and fast way to trim drawing entities, either inside or outside a specified boundary.

Note: The command enables to erase or ignore entities that cannot be trimmed, such as blocks and texts.

18.4.2 Method

There are two methods to define a boundary:

- Select an entity as a trim boundary.

- Define a polygon.

Entities accepted as trim boundary:

- Closed polylines
- Circles
- Closed splines
- Ellipses

Note: Entities that cannot be trimmed:

- Leaders
- Blocks
- Texts
- Mtexts
- 3d faces
- Solids
- Wipeouts
- Raster images
- Surfaces
- Regions
- Dimensions
- Hatches
- 3d solids
- Polyface meshes
- Polygon meshes

18.4.3 Options within the command

Select a trim boundary
Allows you to select an existing entity in the drawing as the boundary polygon.

Define
Allows you to define points as the boundary polygon.

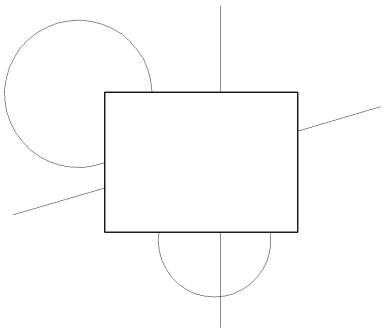
Undo
Undoes the last endpoint of line.

Select entities to trim
Allows you to selects entities to trim in the drawing.

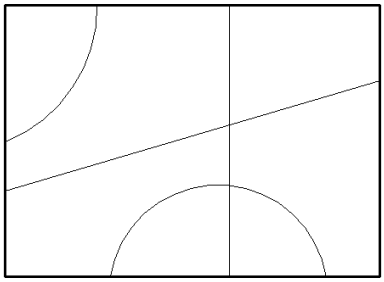
Automatic
Automatically selects all entities in the drawing to be trimmed.

Note: **Automatic** is the default option for entity selection.

Inside
Trims the entities inside the defined boundary.



Outside
Trims the entities outside the defined boundary.



Note: **Outside** is the default option to trim entities.

Ignore
Ignores the entities that cannot be trimmed.

Delete
Deletes the entities that cannot be trimmed.

Reference
If the **Inside** option was chosen, the entity that cannot be trimmed is deleted only if its insertion point lies inside the specified boundary.
If the **Outside** option was chosen, the entity that cannot be trimmed is deleted only if its insertion point lies outside the specified boundary.

18.5 MASSPROP command

Reports mathematical properties of 3D and 2D regions.

Shape Lite Pro Mechanical BIM

18.5.1 Description

Reports the area, perimeter and other mathematical properties of 3D solid models and 2D region entities.

Note: All other entities are ignored by the program.

18.5.2 Options

Write analysis to a file?
Decides whether to save the report into a *.mpr file or not.

Note: If Yes, the Create Mass and Area Properties File dialog box opens. The *.mpr file can be opened in any text editor. MPR is short for Mass Properties Report.

18.6 MATBROWSERCLOSE command

Closes the Render materials panel.

Shape Lite Pro Mechanical BIM

18.6.1 Description

Closes the Render materials panel to hide it from the current workspace. If the Render materials panel is stacked when you close it, the Render materials tab or icon is removed from the stack.

18.7 MATBROWSEROPEN command

Opens the Render materials panel.

Shape Lite Pro Mechanical BIM

Alias: MATB

18.7.1 Description

Opens the Render materials panel to display it in the current workspace. The Render materials panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Render materials panel can be either floating, docked or stacked.

18.8 MATCHPERSPECTIVE command

Matches the current model space view with a background image.

Shape Lite Pro Mechanical BIM

18.8.1 Description

Matches the viewpoint of model space with the apparent perspective view provided by a background image, after three or more pairs of matching points are selected.

Before starting this command, place an image in the drawing with the BACKGROUND command, and then switch the viewpoint to perspective mode with the PERSPECTIVE variable.



18.9 MATCHPROP command

Copies selected properties and styles from one entity and applies it to one or more entities.

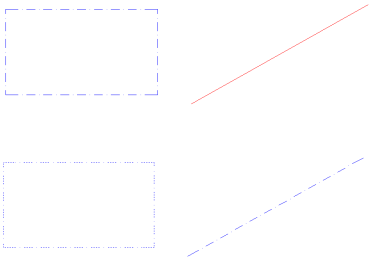
Shape Lite Pro Mechanical BIM

Icon:

Alias: MA

18.9.1 Method

Select the entity whose properties will be copied and the entities to apply the properties to.



18.9.2 Options

Settings

Specifies which settings to be copied. The Match Properties dialog box lists applicable properties along with their current value.

Check all properties that may be applied on other entities so not all properties are automatically matched.

18.10 MATERIALASSIGN command

Assigns current material to entities.

Shape Lite Pro Mechanical BIM

18.10.1 Description

Applies a material definition to filled entities. These materials make the results of visual styles and renderings look more realistic. They do not define physical properties, such as mass. Among visual styles, materials are displayed by the Modeling, Realistic, and Xray styles.

18.10.2 Method

Before a material can be assigned, one must be chosen. There are three methods of picking, assigning, and removing materials to and from entities.

Picking Materials

Follow one of these methods to specify the material to be applied:

- Specify a material with the CMATERIAL command:
- (Hidden option.) After starting the MATERIALASSIGN command, hold down the ALT key:

Use the dropper widget to select an entity that already has a material assigned to it.



Assigning Materials

Use the paintbrush glyph to apply the material to one or more valid entities.



*Valid entities*are 3D solids and 3D surfaces, as well as “solid filled” 2D entities, such as regions and traces. Materials cannot be assigned to unfilled entities, such as circles, nor to open entities, such as arcs, nor to areas that appear to be filled, such as those filled with hatches or gradients.

To apply the material to just one face instead of the entire entity, hold down the CTRL key.

Choose more entities, or undo the last material assignment or end the command.

Removing Materials

To remove materials, apply the **BYLAYER** material to entities or to layers.

18.11 MATERIALMAP command

Adjusts the placement of texture images.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SETUV

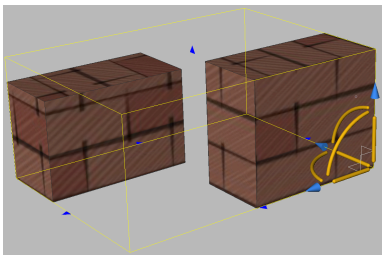
18.11.1 Description

Adjusts the placement of texture images onto 3D solids, 3D faces, wide polylines, and polygon meshes to create a more realistic look, such as matching a brick pattern to the wall of a house.

18.11.2 Method

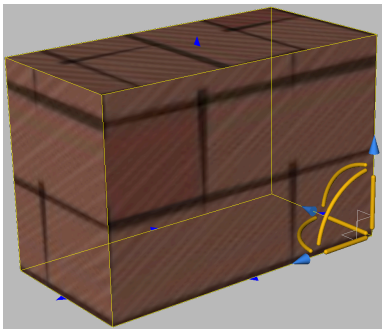
The selected entities are included in a yellow bounding box. Blue grips and the manipulator widget appear:

- Drag the blue grips to change the scale of the material horizontally and vertically.
- Click different parts of the manipulator to move, scale, and rotate the material.



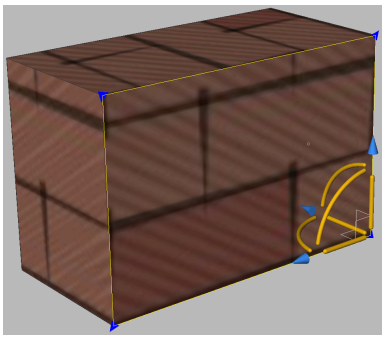
18.11.3 Options within the command

Box
Wraps the texture around the six faces of the bounding box.

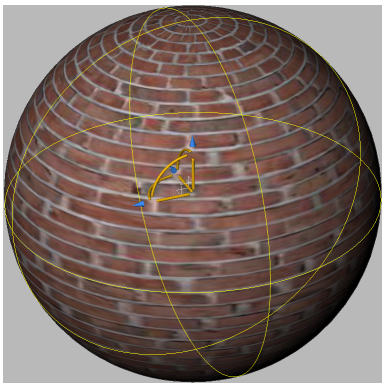


Planar
Aligns the texture to a single face.

Note: To select a face of a 3D solid, tick the Select Faces option of the SELECTIONMODES variable.

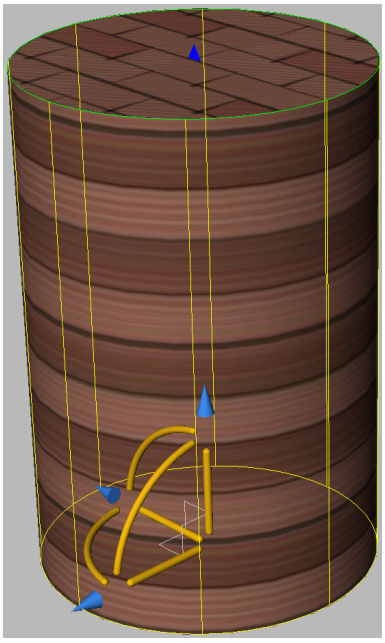


Spherical
Aligns the texture to a spherical shape. The top and bottom edges of the image are compressed to a point at the north and south poles of the sphere.



Cylindrical

Aligns the texture to a cylinder shape. The vertical edges of the image are wrapped together. The height of the image is scaled along the axis of the cylindrical shape.



sWitch mapping mode

Switches to a different mapping mode.

copY mapping to

Applies mapping from the source entity or face to the selected entities, thus duplicating the mapping, including any adjustments, to other entities. All entities share the same mapping origin, axes and scale. As a result, the texture image is extended seamlessly from one entity to another.

Reset mapping

Restores the default mapping.

18.12 MATERIALS command

Opens the Drawing explorer dialog box with **Materials** selected.

Shape Lite Pro Mechanical BIM

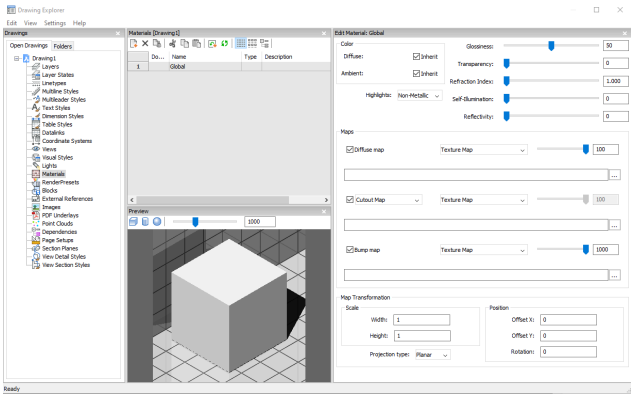
Icon:

Alias: FINISH, MAT, RMAT

18.12.1 Method

Open the Drawing explorer dialog box with **Materials** category selected to view and modify Materials in the current drawing.

Note: Materials provided by RedWay ( ) cannot be edited initially and so their settings are grayed-out, except for Map Transformation. To edit these materials, right-click a Redway material, and then choose Convert to regular material from the context menu.



18.12.2 Options within the command

Preview options

Cube
Displays a preview of the selected material applied to a cube.

Cylinder
Displays a preview of the selected material applied to a cylinder.

Sphere
Displays a preview of the selected material applied to a sphere.

Light Intensity
Sets the light intensity in the Preview window.

Color

Diffuse
Sets the diffuse color. You can click the colored tile to choose a color in the Select Color dialog box. When Inherit is checked on, the entity color is applied.

Ambient
Sets the ambient color. You can click the colored tile to choose a color in the Select Color dialog box. When Inherit is checked on, the entity color is applied.

Highlights
Sets the highlight property of the material. You can choose between Non-metallic or Metallic.

Glossiness
Defines the glossiness of the material surface. You can choose a number in the range of 0 – 100.

Transparency
Defines the transparency of the material surface. You can choose a number in the range of 0 – 100.

Refraction Index
Defines the refraction index of the material surface. You can choose a number in the range of 1.00 – 3.00.

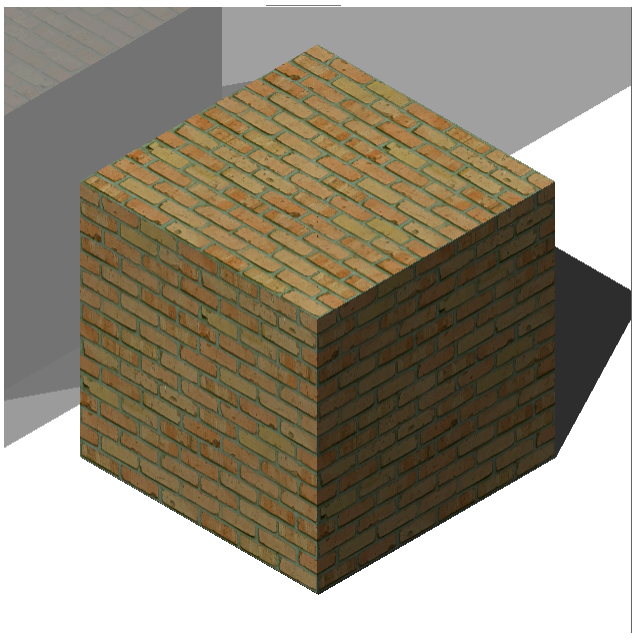
Self-Illumination
Defines the self-illumination of the material surface. You can choose a number in the range of 0 – 100.

Reflectivity
Defines the reflectivity of the material surface. You can choose a number in the range of 0 – 100.

Maps
Texture maps add detail to a surface, which is not included in the 3D model itself.

Note: TheTextureMapPath user preference defines the search path for texture map images. In the BricsCAD program folder exist three subfolders underTextures, each containing a number of texture files of the same name. Images in folder1have a size of 256 x 256 pixels, folder2contains images of 512 x 512 pixels, images in folder3have a size of 1024 x 1024 pixels. If theDiffuse mapsetting of a material uses the image name only (not path), you can control the quality of a rendered image by setting theTextureMapPathuser preference to folder1,2or3.

Diffuse map
Diffuse maps apply a texture to the surface of a material such as wood grains, bricks, or tiles. You can select a texture bitmap file with the Browse button. The selected Texture Map can be applied as Transparency Map or as Cutout Map.

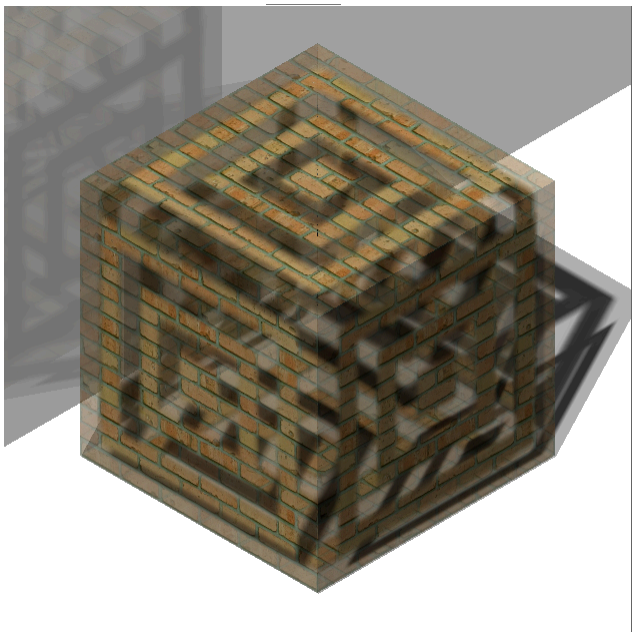


Blend factor
Sets the intensity of the texture map. In this way you can blend the texture map with the Color settings. You can set the blend factor in a range of 0 to 100.

Transparency Map

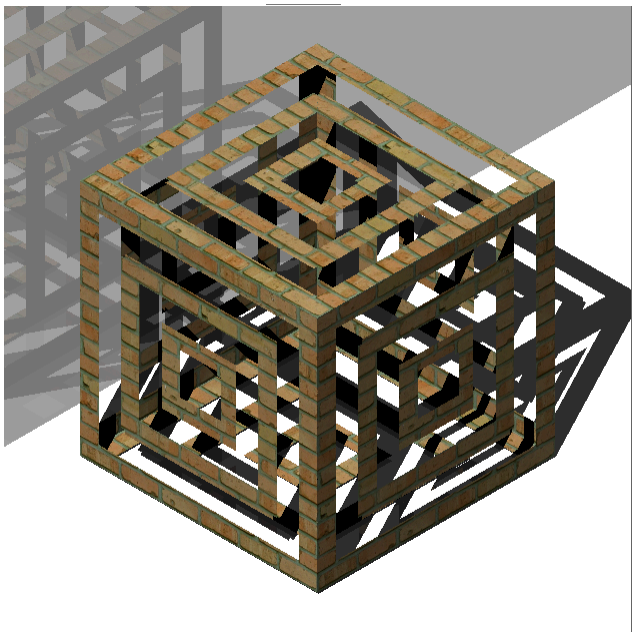
If Transparency Map is selected, the selected Texture Map image defines a transparency map. The image is recommended to be a grayscale image. White pixels will be invisible, black pixels will be opaque and gray pixels will be transparent.

Note: If you select a colored image, it will be converted to a grayscale image in the background. The alpha channel of the image is ignored.



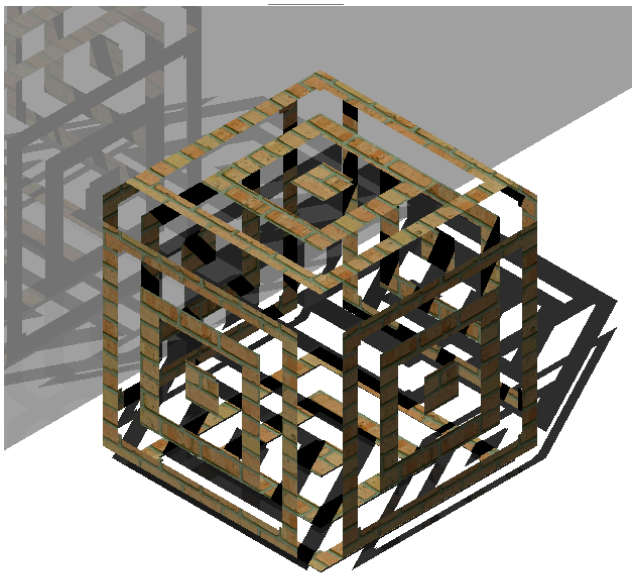
Cutout Map

If Cutout Map is selected, then the selected transparency map defines cutouts. White pixels are visible, and black pixels are invisible. The image is recommended to be a bitonal black-and-white image, without grays.



Cutout Map Inverted

Inverts the Cutout Map effect. Black pixels are visible and white pixels are invisible.



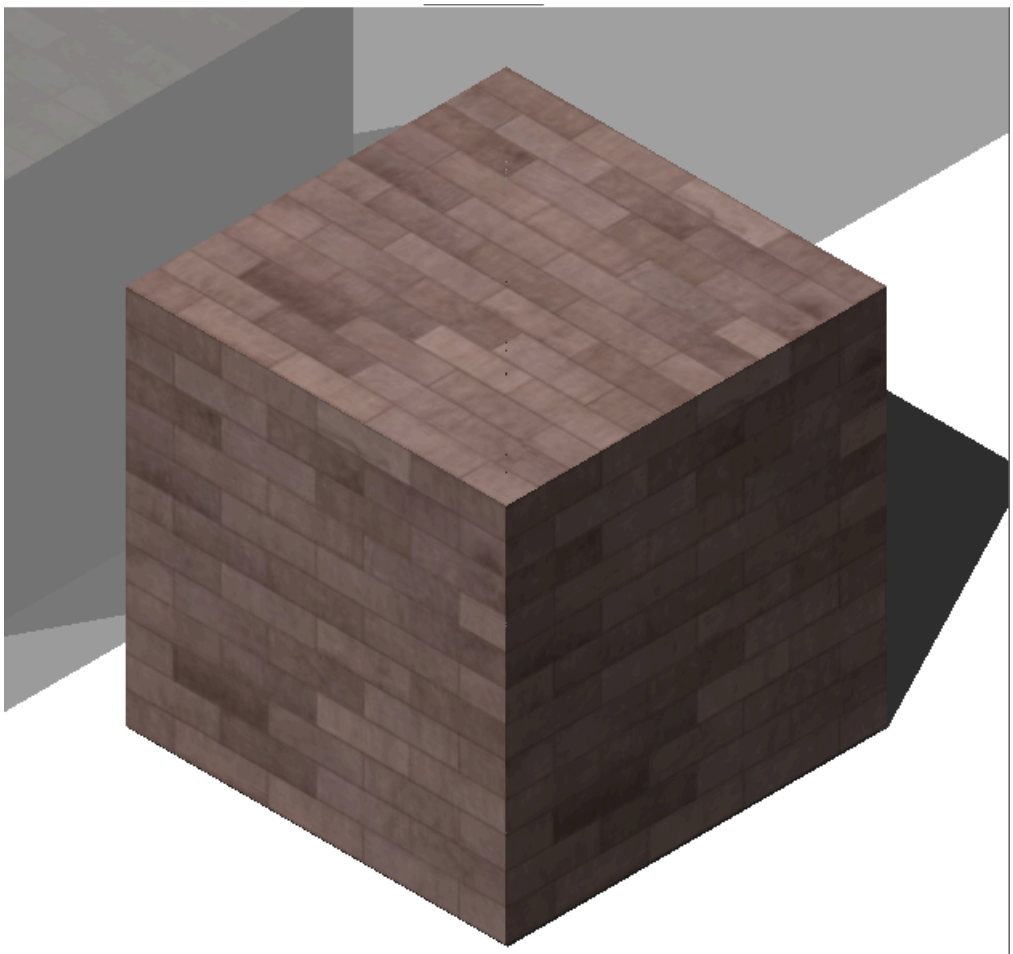
Note: If you select a colored or grayscale image, it will be converted to a black-and-white image in the background. The alpha channel of the image is ignored.

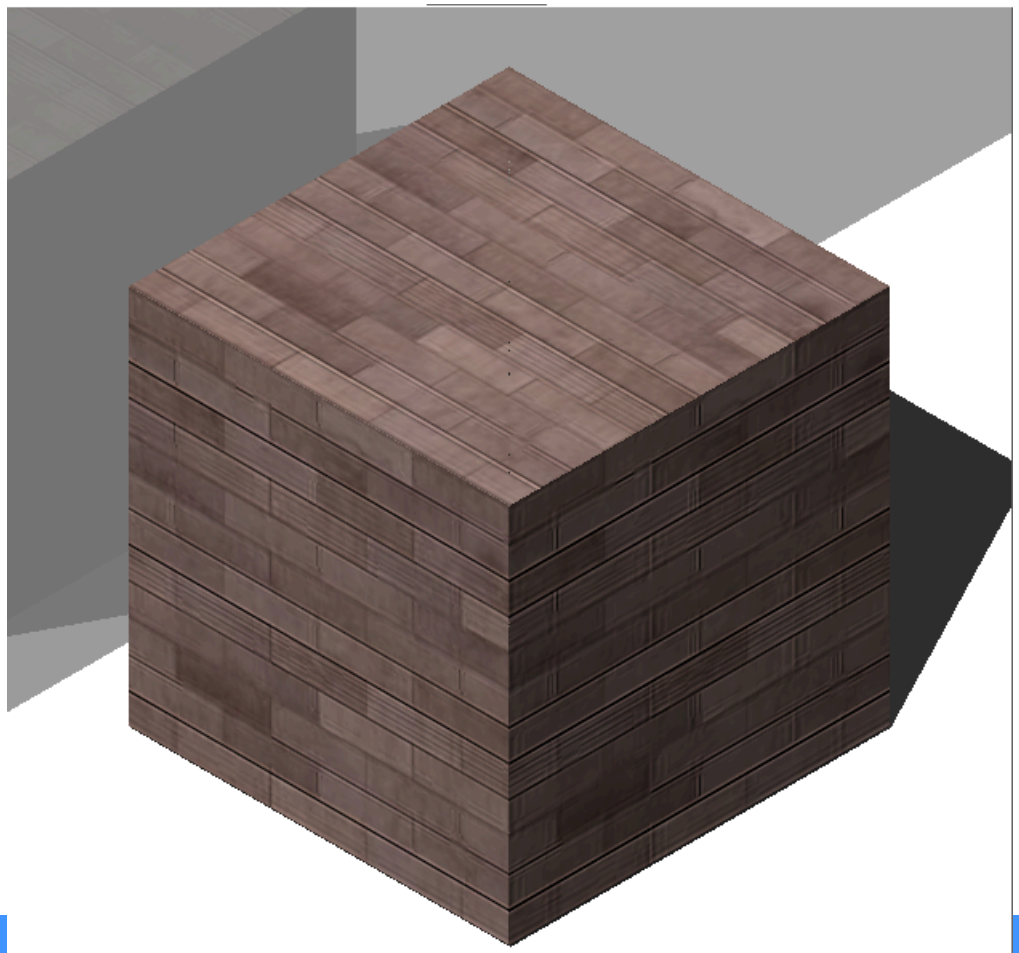
Blend factor

The transparency values in the texture will approach the scalar Transparency setting as the blend factor approaches 0. This means that if the Blend factor is 100, then the transparency value will be controlled completely by the Texture Map image. If the Blend factor is 0, the transparency texture will be completely ignored and the scalar Transparency value will be used.

Bump map

Simulates bumps and wrinkles on the surface of an object. The result is a bumpy surface although the surface of the underlying object is not modified.





Left: Diffuse map only. Right: Bump Map applied

Blend factor
Defines the amount of bump which is applied to the material.

Map Transformation

Scale
The texture map image is applied at a size of 1 drawing unit multiplied by the Width and Height factors.
E.g. if the Width and Height factors or both set to 10, the size of the texture image is 10 x 10 drawing units.

Position

Offset
Texture maps are tiled starting from the origin of the WCS.
To adjust the tiling, you can define an Offset X and Offset Y. The offsets are expressed in drawing units.

Rotation
Sets the rotation of the texture maps.

Projection type
Defines the way the material is projected onto the object.

18.12.3 Context menu options

New
Creates a new material.

Delete
Deletes material definitions from the drawing. The following material definition cannot be deleted:

- Global
- Materials in use

Rename
Renames the material.

Note: Following material cannot be renamed: Global.

Select All
Selects all material definitions.

Invert selection
Deselects the current selection and vice versa.

Convert to regular material
Converts materials to regular material and makes it possible to edit them.

Add material(s) to library
Adds materials to the material library so you can use them in other files as well.

18.13 MATLIB command

Opens the Render materials panel.

Shape Lite Pro Mechanical BIM

18.13.1 Description

Opens the Render materials panel to display it in the current workspace. The Render materials panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Render materials panel can be either floating, docked or stacked.

18.14 MEASURE command

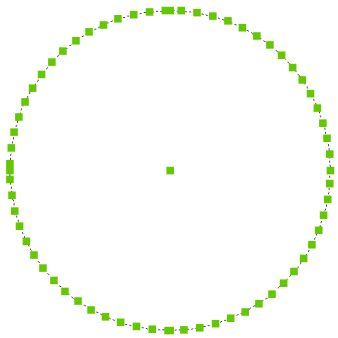
Places points or blocks along entities at equally-spaced specified distance.

Shape Lite Pro Mechanical BIM

Icon:

18.14.1 Method

Specify the entity you want to measure and enter a block distance value or insert a block. On the entity the points are places at equally-spaced specified distance.



Note: Points are usually invisible, so use the PdMode variable to make them larger.

18.14.2 Options within the command

Insert block
Inserts a block instead of a point.

Align block
Rotates the blocks to match the entity alignment.

18.15 MECHANICALBROWSERCLOSE command

Closes the Mechanical browser panel.

Shape Lite Pro Mechanical BIM

18.15.1 Description

Closes the Mechanical browser panel to hide it from the current workspace. If the Mechanical browser panel is stacked when you close it, the Mechanical Browser tab or icon is removed from the stack.

18.16 MECHANICALBROWSEROPEN command

Opens the Mechanical browser panel.

Shape Lite Pro Mechanical BIM

Icon: 

18.16.1 Description

Opens the Mechanical browser panel to display it in the current workspace. The Mechanical browser panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Mechanical browser panel can be either floating, docked or stacked.

18.17 MENU command

Opens the Choose a customization file dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

18.17.1 Description

Opens the Choose a customization file dialog box to select a cui, cuix, mnu, mns, or icm file to load. These customization files modify the BricsCAD user interface.

18.18 MENULOAD command

Opens the Customization groups dialog box.

Shape Lite Pro Mechanical BIM

18.18.1 Description

Opens the Customization groups dialog box to load and unload customization groups.

18.19 MENUUNLOAD command

Opens the Customization Groups dialog box.

Shape Lite Pro Mechanical BIM

18.19.1 Description

Opens the Customization Groups dialog box to load and unload customization groups.

18.20 MIDPOINT command

Toggles the Midpoint entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

18.20.1 Description

Toggles the Midpoint entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

18.21 MINSERT command

Inserts a block as a rectangular array.

Shape Lite Pro Mechanical BIM

18.21.1 Description

Inserts a block as a rectangular array. It is a combination of the –INSERT and -ARRAY commands and creates a MInsert Block entity.

18.21.2 Options within the command

Block to insert
Specify the name of the block to be inserted.

~
Displays the Insert Block dialog box.

Insertion point for block
Specify the insertion point of the selected block.

Scale
Allows to set the X and Y scale factor equally.

Corner
Specifies the size of the block by picking a second point.

X scale
Specifies the scale factor of the block in the x direction.

Y scale
Specifies the scale factor of the block in the y direction.

Z scale
Specifies the scale factor of the block in the z direction.

Rotate
Specifies the rotation angle of the block around its insertion point.

Multiple blocks
Allows to insert a rectangular array of multiple blocks.

Number of rows in the array
Specifies the number of rows.

Note: To insert a linear array, enter 1. To draw the array downwards, enter a negative number.

Number of columns in the array
Specifies the number of columns.

18.22 MIRROR command

Mirrors entities about a mirror line in a 2D plane.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: MI

Note: MIRRTEXT system variable determines whether text is mirrored by the MIRROR command or not.

18.22.1 Description

The entities are mirrored about a mirror line, which you define by specifying two points.

Note: To mirror vertically or horizontally, press the Shift-key when specifying the second point or use Polar Tracking.

18.22.2 Options within the command

Delete the original entities?
Determines whether the original entities are erased or not.

18.23 MIRROR3D command

Mirrors entities about a mirror plane in 3D space.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: 3DMIRROR, 3DM

18.23.1 Description

Creates a mirror copy of selected entities in three-dimensional space.

18.23.2 Options within the command

Entity
Defines the mirror plane by extrusion direction.

Note: This option does not work with flat entities or 3D solids. To give an entity an extrusion, use the Properties panel's Thickness option.

Last
Reuses the last mirror plane.

View
Defines the mirror plane as the current view plane.

Z axis
Defines the mirror plane by the z axis and a point on the view plane.

X-Y plane
Specifies the mirror plane as the x,y plane.

Y-Z plane
Specifies the mirror plane as the y,z plane.

Z-X plane
Specifies the mirror plane as the z,x plane.

3 points
Defines the mirror plane by three points.

Delete the original entities?
Determines whether the original entities are erased or not.

18.24 MLEADER command

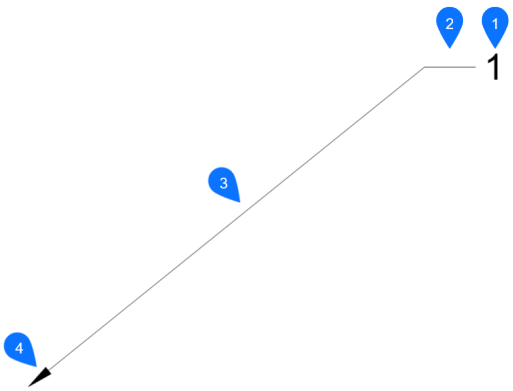
Creates multileader entities using the current multileader style.

Shape Lite Pro Mechanical BIM

Icon:

18.24.1 Description

Creates a multileader with three components: a leader line with an optional arrowhead, a landing line and the content, such as text or a block. Multileaders support more than one leader line per content.



1 Mtext/block

2 Landing

3 Segment

4 Arrowhead

18.24.2 Methods

There are three methods to begin creating a multileader:

- leader arrowHead first

- leader Landing first
- Content first

18.24.3 Options with the command

leader arrowHead first
Create a multileader by first specifying the location of the arrowhead.

leader Landing first
Create a multileader by first specifying the location of the leader landing.

Content first
Create a multileader by first specifying the location of the content.

Leader type
Sets the leader type: straight, spline or none.

Straight
The leader is drawn from straight segments.

SPline
A spline is drawn using the pick points as the control points.

None
No leader is drawn.

leader lAnding
Specify whether to draw a landing or not.

Content type
Specifies the content type of the multileader.

Block
Uses the block name you insert.

Mtext
Uses mtext; displays the MText editor.

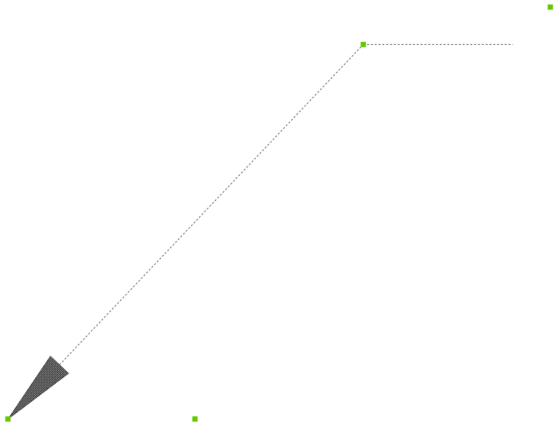
None
Draws no content

Maxpoints
Enter the maximum points.

First angle
Enter the first angle constraint.

Second angle
Enter the second angle constraint.

Grip editing
By clicking the green squares, you can edit the grip points of the multileader.



18.25 MLEADERALIGN command

Aligns the balloons of two or more multileaders.

Shape Lite Pro Mechanical BIM

Icon:

18.25.1 Description

Aligns the balloons of two or more multileaders to each other, to a polyline or array around a circle. You can also specify the spacing between multileaders.

18.25.2 Method

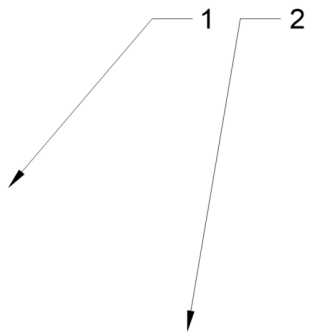
This command offers five methods to align multileaders:

- byMleader
- onpolyLine
- Parallel
- Spacing
- Circle

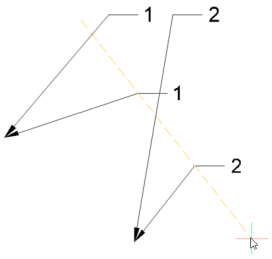
18.25.3 Options within the command

byMleader

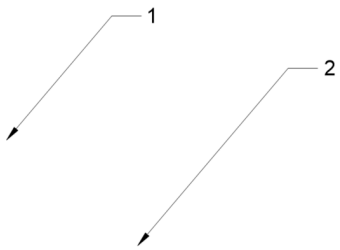
Align the landings of the multileaders to the landing of a selected multileader.



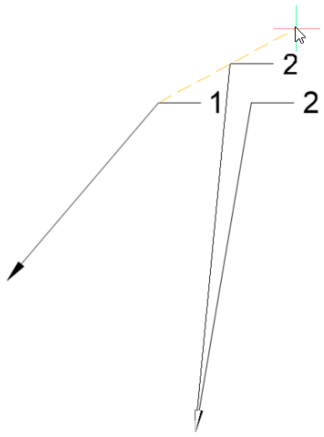
onpolyLine
Align multileaders along an imaginary polyline.



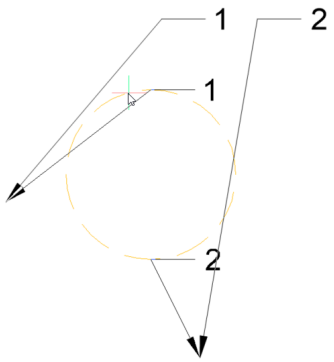
Parallel
Align the segment of the multileaders parallel to the segment of a selected multileader.



Spacing
Specify the spacing between multileaders.



Circle
Arrays the multileaders around a circle.



18.26 MLEADERCOLLECT command

Collects two or more multiline leaders into a single leader.

Shape Lite Pro Mechanical BIM

Icon:

18.26.1 Description

Collects two or more multiline leaders into a single leader in a horizontal or vertical arrangement. The leaders can be sorted alphabetically.

Note: The command only works with mleaders that have blocks as content.

18.26.2 Method

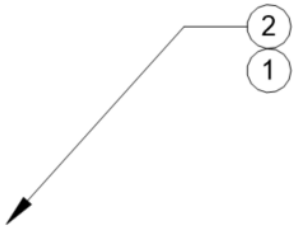
There are five methods to collect multileaders:

- Vertical
- Horizontal

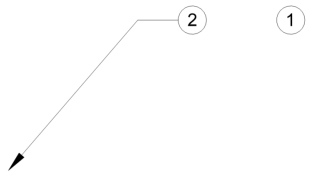
- Wrap
- Sorting
- Collapsing

18.26.3 Options within the command

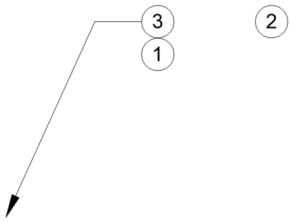
Vertical
Arranges the content blocks vertically to a single leader.



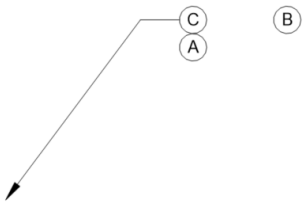
Horizontal
Arranges the content blocks horizontally to a single leader.



Wrap
Arranges the content blocks horizontally to a maximum width, and then places the remaining blocks in the row below.



Sorting
Sorts the content by name.



None
Does not sort the blocks.

Ascending
Sorts from A to Z.

Descending
Sorts from Z to A.

Collapsing
Arranges the content blocks horizontally to a single leader.



18.27 Команда MLEADEREDIT

Редактирование мультивыносок.

Shape Lite Pro Mechanical BIM

Значок:

18.27.1 Description (Описание)

Добавляет и удаляет выноски в/из мультивыноски.

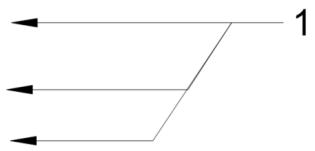
18.27.2 Способы редактирования

Доступны два способа редактирования мультивыносок:

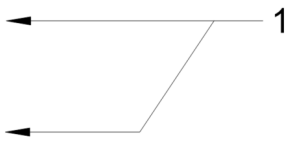
- Добавить выноски
- Удалить выноски

18.27.3 Опции команды

Добавить выноски
Добавляет выноски к выбранной мультивыноске.



Удалить выноски
Удаляет выноски из выбранной мультивыноски.



18.28 MLEADEREDITEXT command

Edit leaders from multileader entities.

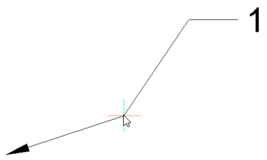
Shape Lite Pro Mechanical BIM

18.28.1 Description

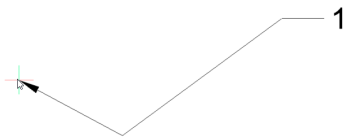
Adds and removes leader lines, adds and removes vertices, and changes the landing length of a multileader entity (short for "multiline leader edit extended").

18.28.2 Options within the command

Add vertex
Adds a vertex to the leader line.

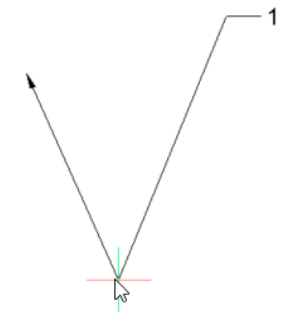


add vertex at ENd
Adds a vertex and a new leader line at the arrowhead.

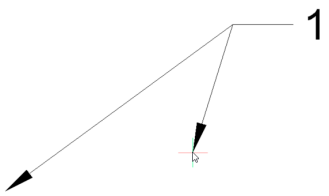


Remove vertex
Removes a vertex from the leader line.

Stretch vertex
Stretch a leader line by moving a vertex.



add Leader
Adds one or more leader lines to an existing mleader.



rEmove Leader
Removes a leader and associated vertex from the leader line.

lengthen LAnding
Changes the length of the landing.



18.29 MLEADERSTYLE command

Opens the Drawing explorer dialog box with **Multileader Styles** selected.

Shape Lite Pro Mechanical BIM

18.29.1 Description

Opens the Drawing explorer dialog box with **Multileader Styles** selected to view and modify multileader styles in the current drawing.

18.30 MLINE command

Creates an mline.

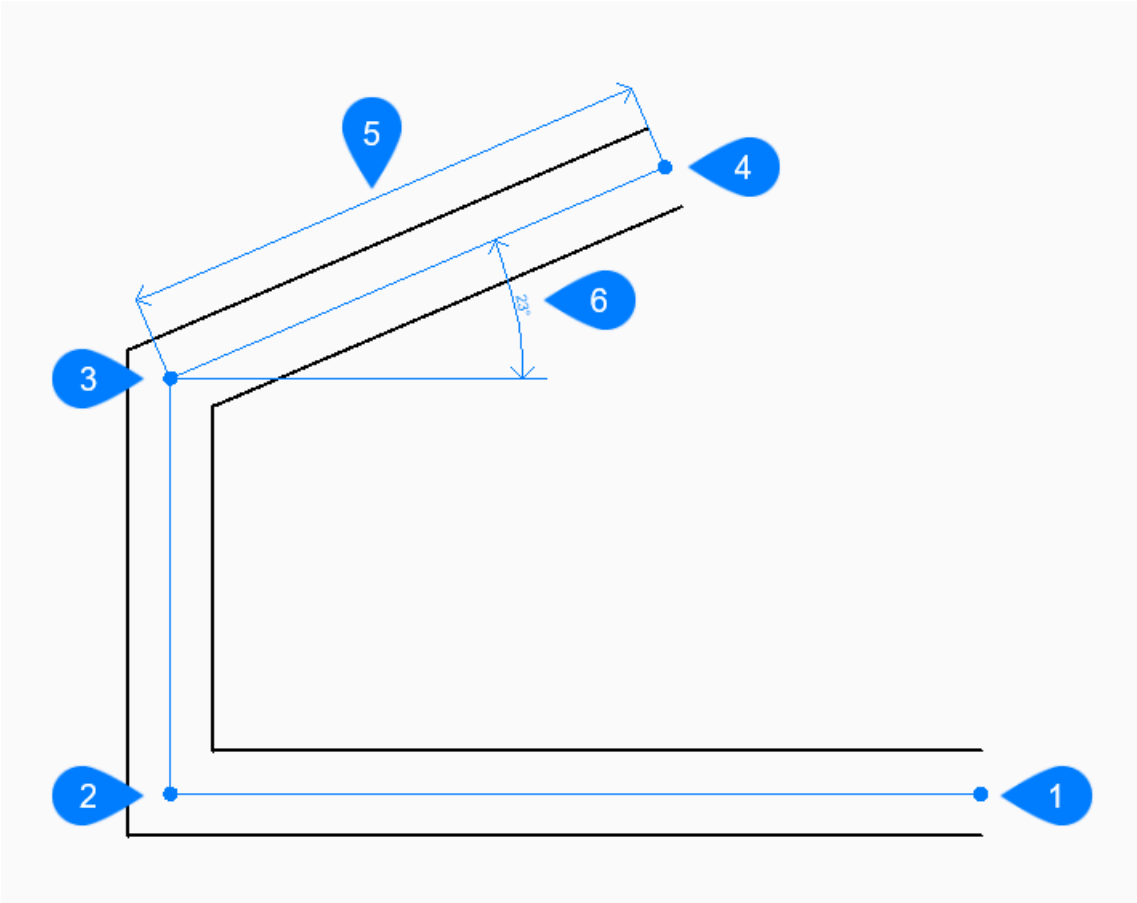
Shape Lite Pro Mechanical BIM

Icon:

Alias: ML

18.30.1 Description

Creates a single mline entity with multiple parallel lines and segments by specifying the start and endpoint of each segment. The mline is based on the current multiline style. Options allow you to specify the justification and scale.



1 Start

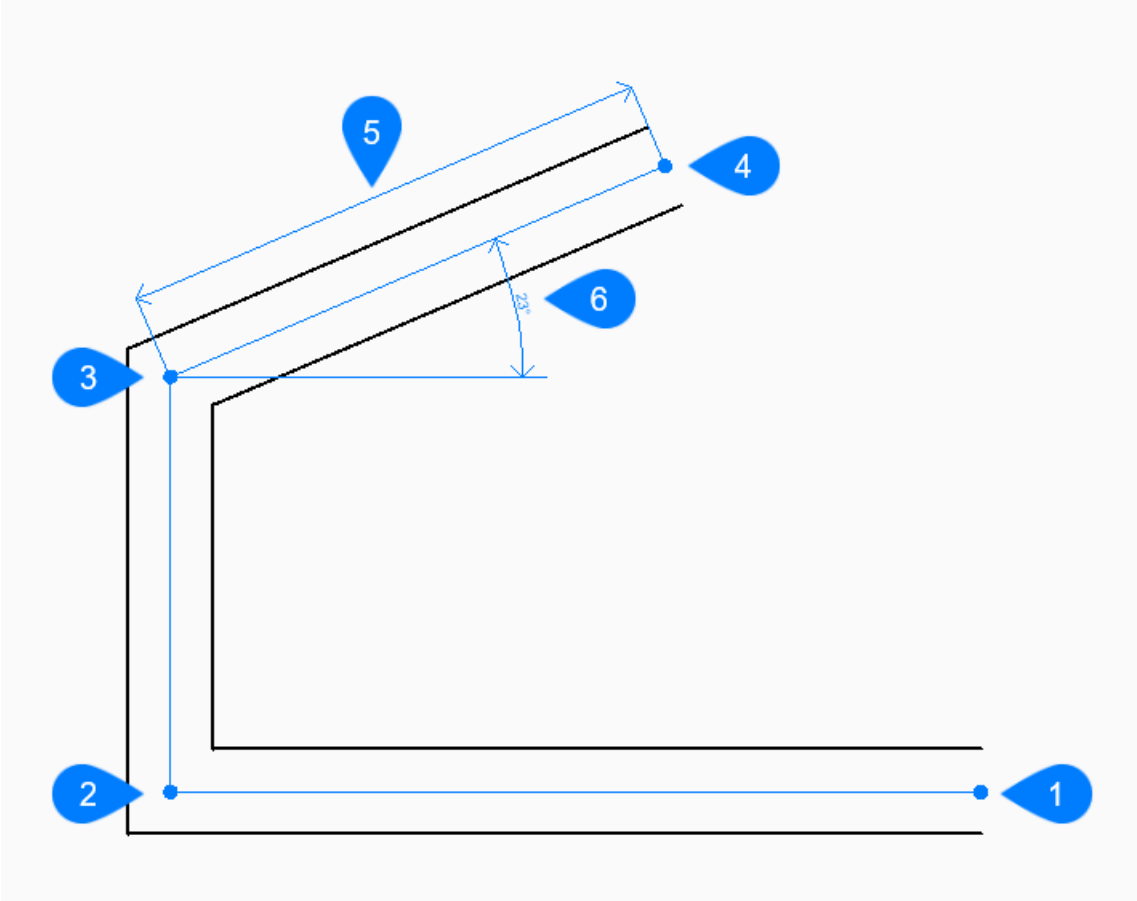
2 Next

3 Next

4 End

5 Length

6 Angle



1 Start

2 Next

3 Next

4 End

5 Close

18.30.2 Methods to create an mline

This command has 3 methods to begin creating an mline:

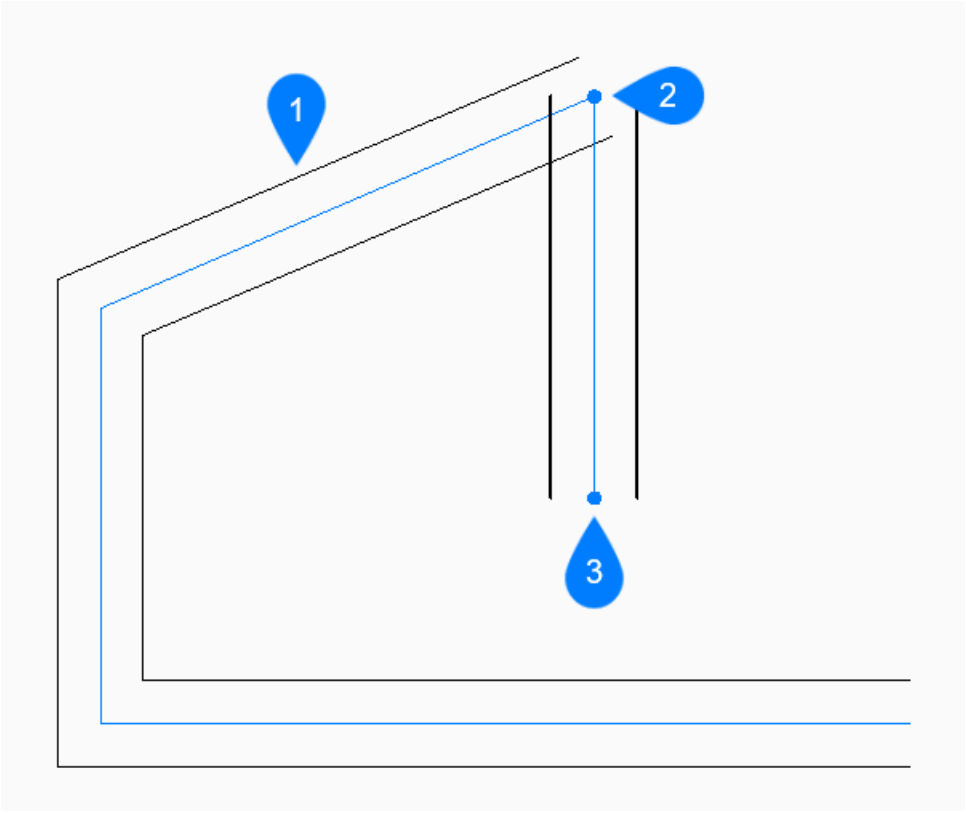
- Start of line
- Last point
- Follow

You can continue adding unlimited segments until you press Enter to end the command.

Start of line
Begin creating an mline by specifying the start point then:
Additional options: [Justification/Scale/STyle/Follow]

Set end point
Specify the endpoint of the mline segment.
Additional options: [Angle/Length/Follow/Close/Undo]

Last point



1 Last drawn segment

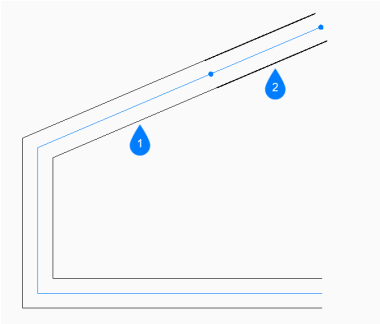
2 Last point

3 End point

Begin creating an mline from the last point picked then:

Set end point
Specify the endpoint of the mline segment.
Additional options: [Angle/Length/Follow/Close/Undo]

Follow



1 Last drawn segment

2 Follow

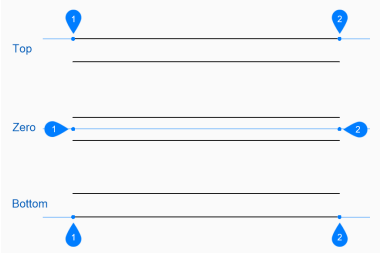
Begin creating an mline from the last drawn line segment, following its angle, then:

Length of Line
Specify the length of the mline segment. Picking a point only determines the length since the angle follows the previous segment.

18.30.3 Options within the MLINE command

After you begin creating an mline, the following options may be available:

Justification
Specify the justification of the mline relative to the start point.



1 Start

2 End

The justification is saved in the CMLJUST variable.

Scale
Specify the overall scale of the mline.
The scale is saved in the CMLSCALE variable.

Style
Specify the style to use for the mline as defined by the MLSTYLE command. The style is saved in the CMLSTYLE variable.
Additional option: [? to list available styles]

[? to list available styles]
Type ? to view a list of all multiline styles that are defined in the drawing.

Angle
Specify the angle of the mline segment measured counterclockwise from the positive x-axis.

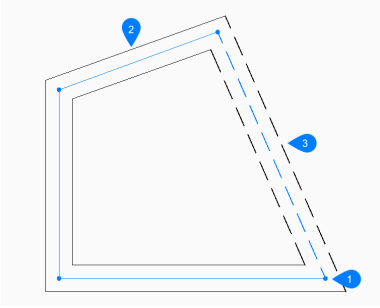
Length of line
Specify the length of the mline segment.

Length
Specify the length of the mline segment.

Angle of line
Specify the angle of the mline segment measured counterclockwise from the positive x-axis.

Undo
Undo the last mline segment and continue drawing from its previous start point.

Close
Automatically draw an mline segment from the end point of the last segment to the start point of the first segment.



1 Start/End

2 Last drawn segment

3 Close

18.31 MLSTYLE command

Opens the Drawing explorer dialog box with **Multiline Styles** selected.

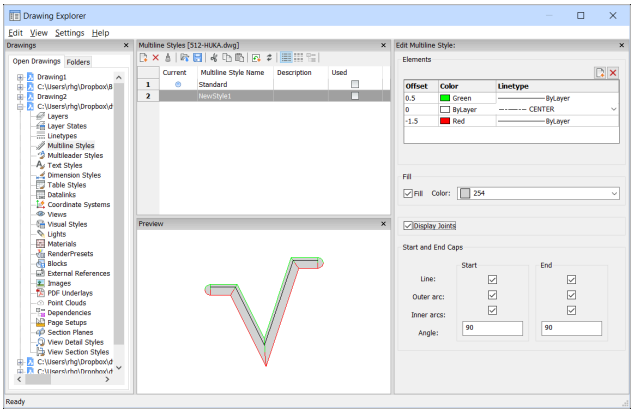
Shape Lite Pro Mechanical BIM

Icon:

18.31.1 Description

Opens theDrawing explorer dialog boxwithMultiline Styles categoryselected to view and modify multiline styles in the current drawing.

Note: Some properties defined by this command can be overridden by options in the MLine command.



18.31.2 Options within the command

Elements

Edits the elements of the multiline.

Add

Adds a line to the multiline definition using the default properties.

- Offset = 0 drawing units
- Color = Bylayer
- Linetype = Bylayer

Delete
Erases a line from the multiline definition without warning.

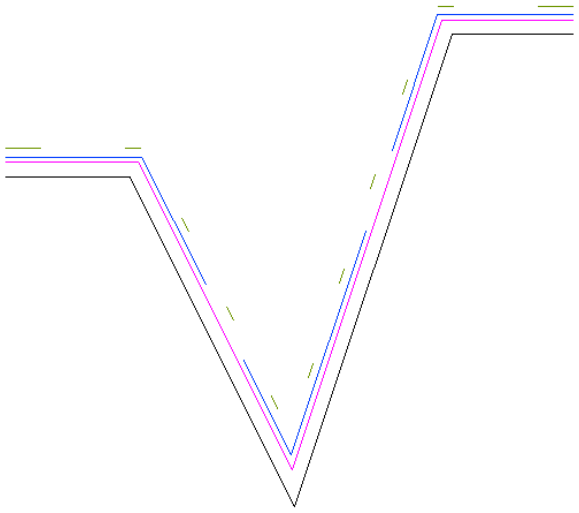
Note: Following multiline style cannot be deleted: The Standard Style.

Offset
Specifies how far each line is from the center line of the mline.

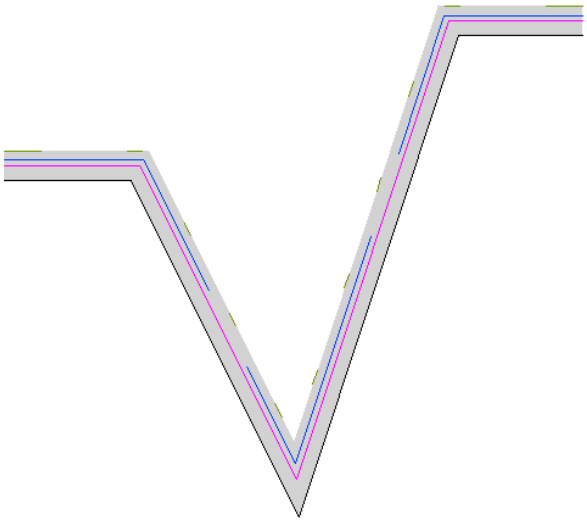
Color
Specifies the color of each line.

Linetype
Specifies the linetype for each line. You can choose a linetype from the list or choose load to access additional linetypes.

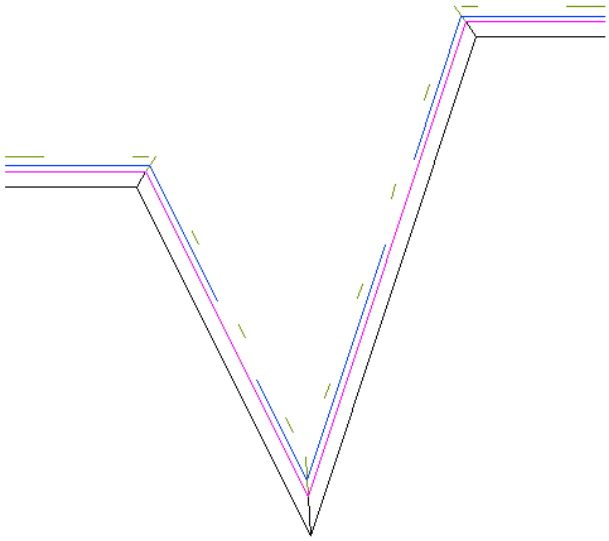
Note: As you add elements and change properties, the preview pane shows what the mline now looks like. If the changes do not load, you should click regen or close and reopen the dialog box.



Fill
When turned on, a solid fill floods the mline between the outer lines.

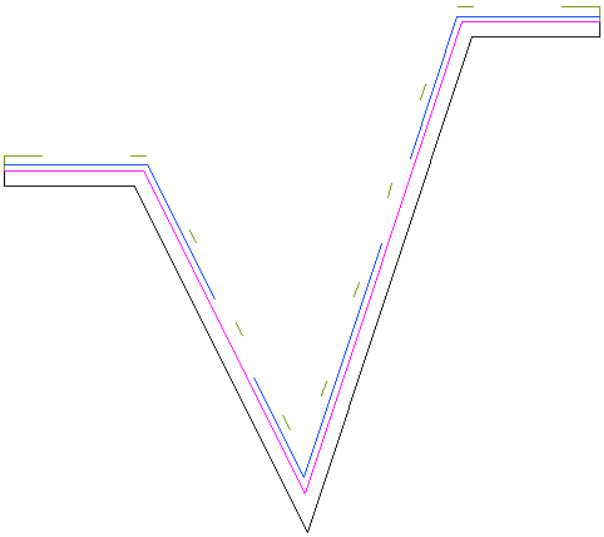


Display Joints
When selected, the mline draws joints (diagonal lines) between segments.

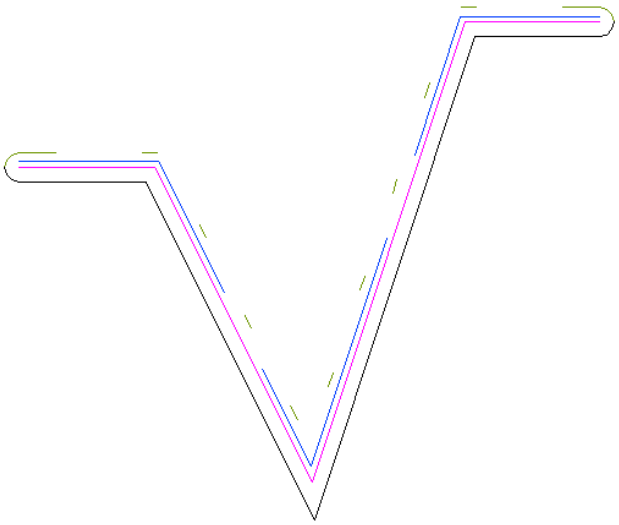


Start and End Caps
Closes the end segments of open multilines by a variety of methods.

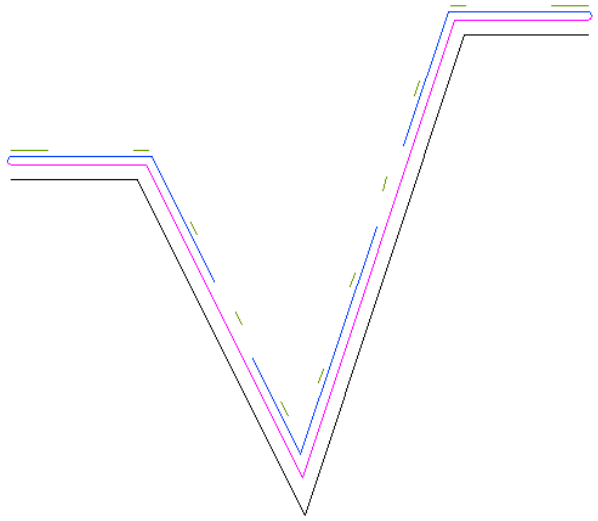
Line
Draws a straight line to close the end segments.



Outer arc
Draws an arc which connect the outer lines of the multiline to close the end segments.



Inner arcs
Draws arcs which connect the inner lines of the multiline to close the end segments.



Angle
Sets the angle of the outer and inner arcs.

18.31.3 Context Menu Options

New
Creates a new multiline style definition. Displays the New Multiline Style dialog box.

Delete
Deletes multiline style definitions from the drawing. The following multiline style definitions cannot be deleted:

- Standard style
- The styles in use

Load from mln file
Loads multiline styles from a multiline (*.mln) file. Displays the Load Multiline Styles dialog box.

Save to mln file
Saves all multiline styles in the drawing to a multiline (*.mln) file for use by other CAD systems. Displays the Save Multiline Styles file dialog box.

Rename
Renames the multiline style.

Note: Following multiline style cannot be renamed: Standard style.

Select All
Selects all multiline style definitions.

Invert selection
Deselects the current selection and vice versa.

18.32 MODELERPROPERTIES command

Opens the Settings dialog box with the **Modeler properties for ACIS** category expanded.

Shape Lite Pro Mechanical BIM

18.32.1 Description

Opens the Settings dialog box with the **Modeler properties for ACIS** category expanded to view and modify relevant system variables.

18.33 -MODELERPROPERTIES command

ACIS modeler settings.

Shape Lite Pro Mechanical BIM

18.33.1 Description

Specifies settings for the ACIS modeler.

18.33.2 Method

There are 14 methods to set ACIS modeling properties:

- view Properties
- USe facetres
- Surface tolerance
- Normal tolerance
- Max grid lines
- Edge length
- grid aspect Ratio
- Adjust mode
- Grid mode
- Triangulation mode
- min U grid lines
- min V grid lines
- mass properties aCcuracy
- check Level

Note: Turning on FACETRES turns off all other settings.

To adjust the Hidden Line Accuracy setting, use the MODELPROPERTIES command.

18.33.3 Options within the command

view Properties
Reports the value of each setting.

USe facetres
Reads the value of the FACETRES variable to set the smoothness of shaded and rendered scenes.

Surface tolerance
Sets the value for the SPANORMALTOL variable.

Normal tolerance
Sets the value for the SPASURFACETOL variable.

Max grid lines
Sets the value for the SPAMAXNUMGRIDLINES variable.

Edge length
Sets the value for the SPAMAXFACETEDGELENGTH variable.

grid aspect Ratio
Sets the value for the SPAGRIDASPECTRATIO variable.

Adjust mode
Sets the value for the SPAADJUSTMODE variable.

Grid mode
Sets the value for the SPAGRIDMODE variable.

Triangulation mode
Sets the value for the SPATRIANGMODE variable.

min U grid lines
Sets the value for the SPAMINUGRIDLINES variable.

min V grid lines
Sets the value for the SPAMINVGRIDLINES variable.

mass properties aCcuracy
Sets the value for the MASSPROPACCURACY variable.

check Level
Sets the value for the SPACHECKLEVEL variable.

18.34 MOVE command

Moves entities.

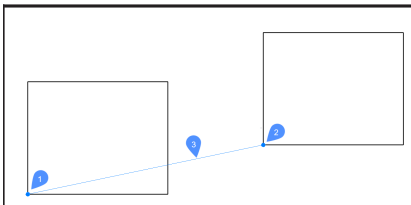
Shape Lite Pro Mechanical BIM

Icon:

Alias: M

18.34.1 Description

Moves entities by specifying the starting point and the destination.



1 Base point
2 Second point
3 Displacement distance

18.34.2 Options within the command

Displacement
Specify the displacement vector (the distance at which to position the copy). 'Vector' means that you specify the distance and angle at the same time.

Note: When Dynamic Input Mode is active, you can type a distance and angle in the dynamic input fields.

18.35 MOVEEDATA command

Moves extended entity data from one entity to one or more others.

Shape Lite Pro Mechanical BIM

Icon: 

18.35.1 Method

Enter the name of the application to which the entity data belongs, select the entity from which to move Entity Data and one or more entities to which Entity Data will be moved (that will receive the edata).
Extended Entity Data can be created with EditEData command.

18.35.2 Options

? to list application names
Lists names of applications loaded in the current drawing.

18.36 MOVEGUIDED command

Moves entities usingguide curves.

Shape Lite Pro Mechanical BIM

Icon: 

18.36.1 Description

Automatically aligns moved entities with relevant geometry, using temporary guide curves.

18.36.2 Method

The command can be executed in two ways:

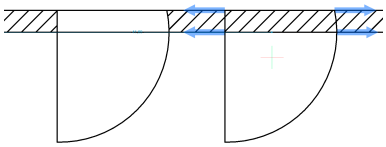
Pre-pick mode
First select entities, then launch the command.

Note: The entities to be copied are displayed in green.

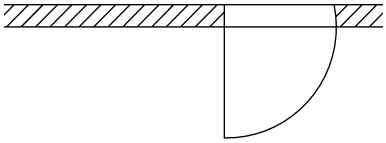
Post-pick mode
Launch the command, then select the entities using a selection window.

Note: All entities that fall completely within the selection window are included in the copy selection set and are displayed in yellow. Entities that cross the selection window are used as guide curves and are displayed in blue.

Blue arrows indicate the anchor points and direction of the guide curves. The moved entities will only align with geometry that matches the number of guide curves and distances between them.



Click to place the moved entity or type a distance in the dynamic entry fields.



18.36.3 Options within the command

Polygonal
Creates a polygonal selection window.

Rectangular
Creates a rectangular selection window.

Region
Pastes the clipped area inside the selection window.

Entities
Pastes the entities inside the selection window.

18.37 MSLIDE command

Opens the Create snapshot dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: MSNAPSHOT

18.37.1 Description

Opens the Create snapshot dialog box to save data from the current drawing to an sld file.

18.38 MSPACE command

Switches from paper space to model space viewports.

Shape Lite Pro Mechanical BIM

Alias: MS

18.38.1 Description

Switches from paper space to model space viewport in a layout tab. This command has a shortcut menu that opens by right-clicking on M:Layout on the Status Bar.

Note: This command operates only when a layout tab is in paper space mode. To reverse the action of this command double-click outside the viewport border, or use PSPACE command.

18.39 MTEXT command

Places text in a bounding box.

Shape Lite Pro Mechanical BIM

Icon:

Alias: MT, T

18.39.1 Description

Places formatted, paragraph text in a bounding box that limits the extent of the text.

After specifying the limits of the text bounding box, the Text Formatting toolbar opens.

You can edit Mtext bounding boxes directly using grips:



1 Drag to change the width of the bounding box.

2 Drag to change the height of the bounding box.

18.39.2 Options within the command

Justification
Specifies the justification (horizontal alignment) of the text inside the bounding box.

Rotation angle
Specifies the rotation angle of the text block. Positive angles rotate the text block counter-clockwise.

text Style
Specifies the text style to use. See Style command.

text Height
Specifies the height of the text.

Note: This option is unavailable when the style specifies the text height.

Direction
Specifies the direction in which the bounding box is expanded.

Left-to-right
The text is placed at the left and expands the bounding box to right.

Top-to-bottom
The text is placed at the top and expands the bottom of the bounding box.

By-Style
Uses the direction defined by the text style.

Width
Specifies the width of the bounding box.

Note: When set to 0, the box has "infinite" width.

Line spacing
Defines the line spacing of the text.

At least
Specifies the minimum line spacing factor.

Exactly
Specifies the exact line spacing factor.

Columns
Defines the amount of columns.

No columns
No columns are created.

Static
A fixed amount of columns with fixed dimensions is created.

Dynamic
A dynamic amount of columns is created. The amount of columns depends on the length of the text.

18.40 -MTEXT command

Creates multiline text with the Command line.

Shape Lite Pro Mechanical BIM

Places formatted, paragraph text in a bounding box that limits the extent of the text.

Note: The text does not appear in the drawing until you end the command.

18.40.1 Description

Creates a multiline text by specifying the first corner and the opposite corner for block of text.

18.40.2 Options within the command

Justification
Specifies the horizontal alignment of the text inside the bounding box. Choose between: Top-Left, Top-Center, Top-Right, Middle-Left, Middle-Center, Middle-Right, Bottom-Left, Bottom-Center, Bottom-Right.

Rotation angle
Specifies the rotation angle of the text block.

Note: Positive angles rotate the text block counter-clockwise.

text Style
Specifies the text style to use for the mtext.

?
Lists all styles defined in the current drawing.

text Height
Specifies the height of the text.

Note: This option is unavailable when the style specifies the text height.

Direction
Specifies the direction in which the bounding box is expanded.

Left-to-right
The text is placed at the left and expands the bounding box to right.

Top-to-bottom
The text is placed at the top and expands the bottom of the bounding box.

Width
Specifies the width of the bounding box.

Note: When set to 0, the box has "infinite" width.

Line spacing
Specifies the line spacing of the mtext.

At least
The lines of text will be automatically adjusted, based on the highest character in the line.

Exactly
The line spacing will be the same for all lines of the mtext.

Columns
Specify the properties of the columns.

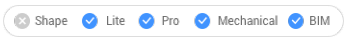
No columns
Sets no column to the mtext.

Static
Specify total width, number of columns, gutter (space between columns) and height of the columns.

Dynamic
Specify width, gutter (space between columns) and height of the columns.

18.41 MTP command

Allows to specify a point as the midpoint.



18.41.1 Description

Allows to specify a point as the midpoint between two points whenever a command prompts to enter a point.

Note: This is a transparent command.

18.41.2 Method

This command works only with commands that prompts to enter a point.

Note: This command sets entity snaps, similar to OSNAP command.

18.41.3 Options within the command

First point of mid
Identify the first reference point.

Second point of mid
Identify the second reference point.

18.42 MULTIPLE command

Repeats commands.

Shape Lite Pro Mechanical BIM

18.42.1 Description

Repeats commands until you press the Esc key.

Note: This command is useful for commands that do not repeat automatically.

18.43 MVIEW command

Creates one or more viewports in Paper space (short for "make viewports") to display entities drawn in Model space. Each viewport can have its own settings, shown below.

Shape Lite Pro Mechanical BIM

Icon:

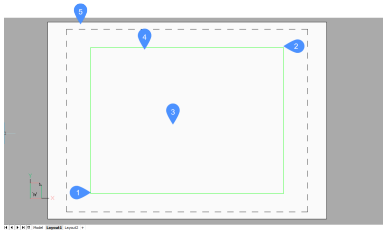
Alias: MV

Note: This command operates only in Paper Space.

Note: To create viewports in Model space, use the VPORTS command.

18.43.1 Description

Specifies the first and the opposite corners to create a rectangular viewport.



1 First corner

2 Opposite corner

3 Viewport

4 Viewport border

5 Paper space (Layout)

18.43.2 Options

ON
Turns on viewports that are turned off.

OFf
Turns off viewports.
This option hides the content of the viewport; the viewport border remains visible. To hide the viewport border, place it on a separate layer, and then freeze the layer.

Lock
Locks the viewport's scale factor.
Set the scale factor with the ZOOM command's nXP option.

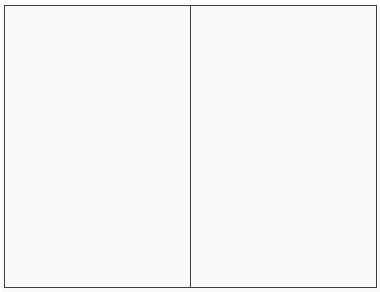
Fit
Draws a rectangular viewport that fits the current layout.

create 2 viewports
Draws two rectangular viewports.

Horizontal
Draws two horizontal viewports of equal size.



Vertical
Draws two vertical viewports of equal size.



create 3 viewports
Draws three rectangular viewports.

Above
Draws one viewport above two side-by-side viewports.



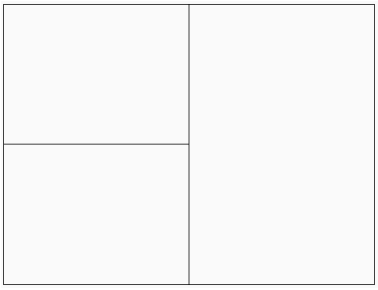
Below
Draws one viewport below two side-by-side viewports.



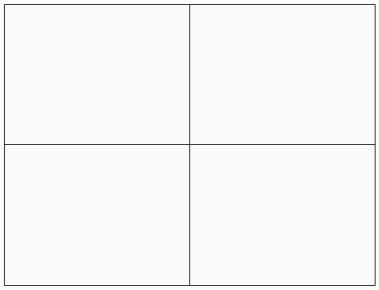
Left
Draws one viewport to the left of two stacked viewports.



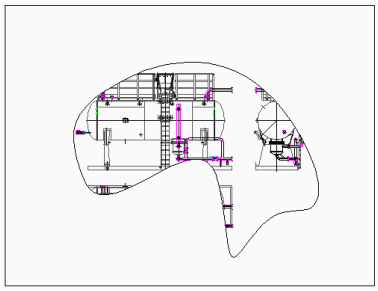
Right
Draws one viewport to the right of two stacked viewports.



create 4 viewports
Draws four rectangular viewports of equal size.



Object
Converts an object into a viewport border.
The viewport is traced over the object; the original object remains in the drawing. This option is useful for ensuring a viewport border precisely matches an outline.



Polygonal
Draws non-rectangular viewports made of lines and arcs.

draw Arcs
Enters arc drawing mode; options from ARC command are activated.

draw Lines
Enter line drawing mode; options from LINE command are activated.

Distance
Draws the next line segment at a specified distance and angle.

Follow
Draws the next line segment at the same angle.

18.44 MVSETUP command

Creates multiple paper space viewports and edits them.

Shape Lite Pro Mechanical BIM

18.44.1 Description

Creates multiple paper space viewports and then aligns, rotates and scales them.

Note: This command operates differently, depending on whether it starts in Model tab or Layout tab.

Note: This command can be entered transparently.

18.44.2 Method

There are two methods to use MVSETUP command:

- In Model tab.
- In Layout tab.

18.44.3 Options within the command

Yes
Switches to Layout tab.

No
Stay in Model tab.

Align
Pans the display in a viewport to align it with a base point in a reference viewport.

Note: Depending on the relative position of the two viewports, the display of the panned viewport might be shifted outside of the viewport boundary.

Angled
Pans the view in the second viewport is panned over the specified distance and angle.

Horizontal alignment
Pans the view in the second viewport vertically to align the base point and the anchor point horizontally.

Vertical alignment
Pans the view in the second viewport horizontally to align the base point and the anchor point vertically.

Rotate view
Rotates the view in the viewport by the specified angle.

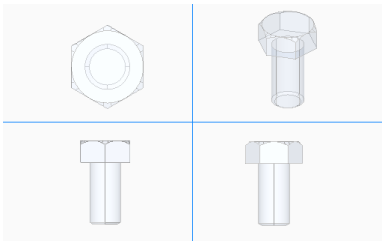
Note: When the VPROTATEASSOC variable is ON, the display in a viewport is rotated if the viewport is rotated.

Undo
Undoes the previous action and returns to the options prompt.

Create
Creates viewports in the current layout.

Delete objects
Deletes viewport entities.

2 - std. engineering
Creates 4 viewports preset to top, front, right and south-east isometric viewpoints.



3 - array of viewports
Creates a desired number of viewports.

Scale viewports
Specifies the scaling inside viewports.

Interactively
Scale each viewport separately.

Uniform
Scale all viewports equally.

Undo
Undoes all operations executed by the current MVSETUP command session.

Scientific
Scales the viewport on a scientific scale.

Decimal
Scales the viewport on a decimal scale.

Engineering
Scales the viewport on an engineering scale.

Architectural
Scales the viewport on an architectural scale.

Metric
Scales the viewport on a metric scale.

19. N

19.1 NAVIGATE command

Changes the way you navigate in BricsCAD.

Shape Lite Pro Mechanical BIM

Icon:

19.1.1 Description

Walks and flies through 3D models with the keyboard, as used in computer games.

19.1.2 Method

There are two methods to navigate through the model:

- Keyboard option: Hover the mouse cursor over the target location and then hold down a key to navigate.
- Mouse option: Hold down left button and drag the mouse.

Pressing right click brings up the Navigation Settings dialog box.

19.1.3 Keyboard options

A/Left arrow
Moves to the left.

W/Up arrow
Moves forward.

S/Down arrow
Moves backwards.

D/Right arrow
Moves to the right.

F
Toggles flight mode on and off.

ESC
Exits the command.

19.1.4 Mouse options

Hold down left button and drag the mouse
Look around mode.

Right click
Brings up the Navigation Settings dialog box.

19.2 NEAREST command

Toggles the Nearest entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

19.2.1 Description

Toggles the Nearest entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

19.3 NETLOAD command

Opens the Netload dialog box.

Shape Lite Pro Mechanical BIM

19.3.1 Description

Opens the Netload dialog box to select a dll file and load the application.

19.4 NEW command

Opens the Select template dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

19.4.1 Description

Opens the Select template dialog box to select a dwt or dwg file to use as a template for creating a new drawing.

19.5 NEWSHEETSET command

Opens the Create new sheet set dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

19.5.1 Description

Opens the Create new sheet set dialog box to create a new sheet set.

19.6 NEWWIZ command

Opens the Create new drawing dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

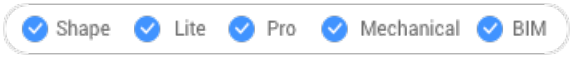
Alias: DDNEW

19.6.1 Description

Open the Create new drawing dialog box to create a new drawing using a wizard.

19.7 NODE command

Toggles the Node entity snap.



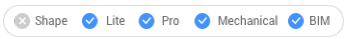
Icon:

19.7.1 Description

Toggles the Node entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

19.8 NONE command

Disables all 2D entity snaps.



Icon: 

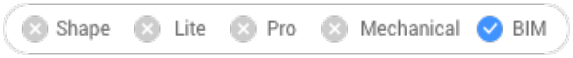
19.8.1 Description

Disables all 2D entity snaps to prevent the cursor from snapping to 2D entities. You can launch this command at the Command prompt to turn off running entity snaps. Doing so changes the value of the OSMODE system variable to zero. You can also launch this command within another command to turn off entity snaps only for the current operation. This does not change the value of the OSMODE system variable.

This command displays no prompts and has no options.

19.9 NUMBER command

Creates incremental number tags for BIM entities.



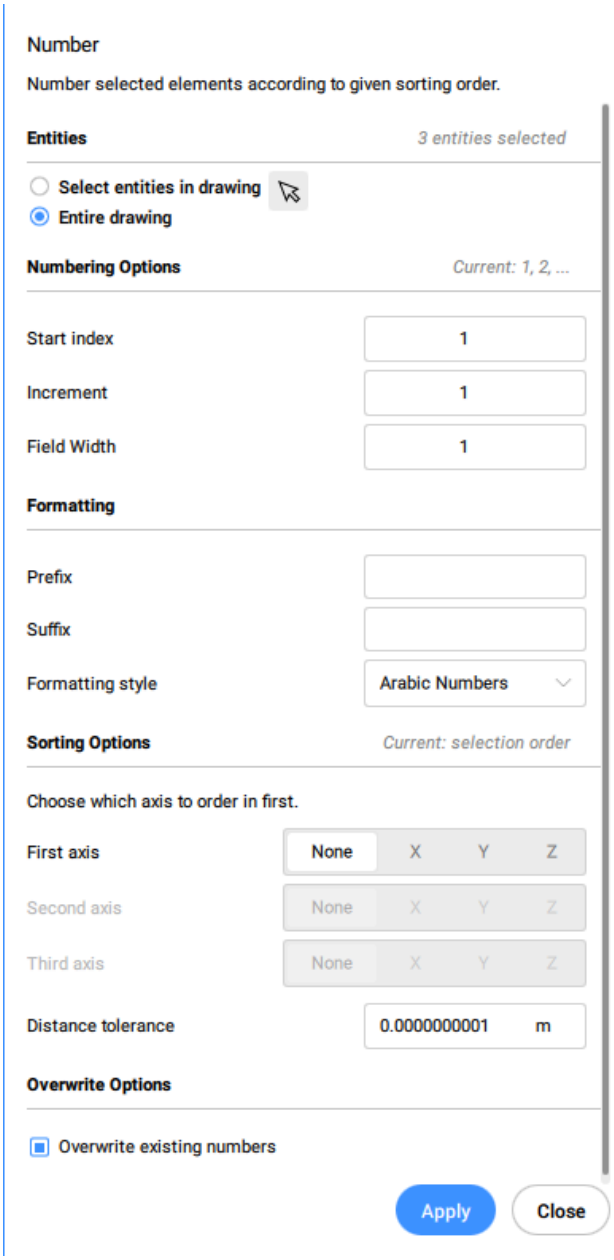
Icon: 

19.9.1 Description

Provides the first index for the selected entities.

Note: Integer numbers are accepted.

The command opens the Number command panel.



Select entities:

- **Select entities in the drawing**: choose which entities to select.
- **Entire drawing**: by default, the panel selects all BIM entities in the model space.

Numbering Options

- **Start index:** sets the index from which to start the numbering.
- **Increment**: sets the increment used for numbering
- **Field Width**: sets the total length of the number field. This will add the appropriate amount of leading zeros in front of Arabic numbers, so that the length of the resulting string is equal to the field width.

Formatting

- **Prefix**: sets a prefix for the numbering.
- **Suffix**: sets a suffix for the numbering.
- **Formatting style**: choose a number style from the drop-down list.

Sorting Options
Allows to specify which axis to order first.

X
Based on the value of the x-coordinate, from low to high.

Y
Based on the value of the y-coordinate, from low to high.

Z
Based on the value of the z-coordinate, from low to high.

None
Uses selection order, unless the option 'Entire Drawing' was chosen, in which case it will order from oldest to most recent.

Distance Tolerance
Sets the distance tolerance for sorting in comparison. Two solid's centroids within a given tolerance are considered equal, and they will be numbered in selection order.

Overwrite Options
Allows to overwrite or to keep existing numbers.

19.9.2 Options within the command line

Increment
Sets the increment for the numbering.

Prefix
Specifies a prefix for the numbering.

Suffix
Specifies a suffix for the numbering.

Number style
Defines the number style.

0
Define Arabic numbers (1, 2, 3, ...).

1
Define roman uppercase numbers (I, II, III, ...).

2
Define roman lowercase numbers (i, ii, iii, ...).

3
Define uppercase letters (A, B, C, ...).

4
Define lowercase letters (a, b, c, ...).

Field Width
Sets the total length of the number field. This will add the appropriate amount of leading zeros in front of Arabic numbers.

Entities sorting
Allows to specify the sorting order.

X
Based on the value of the x-coordinate, from low to high.

Y
Based on the value of the y-coordinate, from low to high.

Z
Based on the value of the z-coordinate, from low to high.

None
Uses selection order, unless the option **Entire Drawing** was selected, in which case it will order from oldest to most recent.

Tolerance
Sets the distance tolerance for sorting in comparison. Two solid's centroids within a given tolerance are considered equal, and they will be numbered in selection order.

Overwrite Numbers
Allows to overwrite or to keep existing numbers.

Kept
Keeps existing numbers.

Overwritten
Overwrites existing numbers.

Change Selection
Allows to change the selection set.

20. O

20.1 OBJECTSCALE command

Adds or removes scale factors.

Shape Lite Pro Mechanical BIM

Icon: 

20.1.1 Description

Adds or removes scale factors used with annotative entities through Annotation Object Scale dialog box.

Note: Annotative scaling adjust scale factors of annotative items, such as text and hatch patterns, to match the plot scale.

20.2 -OBJECTSCALE command

Adds or removes scale factors.

Shape Lite Pro Mechanical BIM

20.2.1 Description

Adds or removes scale factors used with annotative entities through Annotation Object Scale dialog box.

Note: Annotative scaling adjust scale factors of annotative items, such as text and hatch patterns, to match the plot scale.

20.2.2 Options within the command

Select annotative entities

Select one or more items using any selection method.

Note: Write all in the Command line to select all entities in the drawing. The program filters out ineligible entities automatically.

Add

Adds annotation scales to the selected annotative entities.

Note: Enter ? to list the annotation scale factors available.

Delete

Deletes annotation scales from the selected annotative entities.

?

Displays a list of available annotation scales as defines in the scale list.

Note: See SCALELISTEDIT command.

	Scale Name	Paper Units	Drawing Units	Effective Scale
1:	100:1	100.0000	1.0000	100.0000
2:	10:1	10.0000	1.0000	10.0000
3:	1:1	1.0000	1.0000	1.0000
4:	1:10	1.0000	10.0000	0.1000
5:	1:100	1.0000	100.0000	0.0100
6:	1:125	1.0000	125.0000	0.0080
7:	1:16	1.0000	16.0000	0.0625
8:	1:2	1.0000	2.0000	0.5000
9:	1:20	1.0000	20.0000	0.0500
10:	1:25	1.0000	25.0000	0.0400
11:	1:250	1.0000	250.0000	0.0040
12:	1:30	1.0000	30.0000	0.0333
13:	1:4	1.0000	4.0000	0.2500
14:	1:40	1.0000	40.0000	0.0250
15:	1:5	1.0000	5.0000	0.2000
16:	1:50	1.0000	50.0000	0.0200
17:	1:8	1.0000	8.0000	0.1250
18:	2:1	2.0000	1.0000	2.0000
19:	4:1	4.0000	1.0000	4.0000
20:	8:1	8.0000	1.0000	8.0000

20.3 OFFSET command

Creates parallel copies.

Shape Lite Pro Mechanical BIM

Icon:

Alias: O

20.3.1 Description

Creates parallel copies of 2D entities and 3D solid faces.

Note: The OFFSETGAPTYPE system variable determines how potential gaps between segments are treated when polylines are being offset.

Note: The SELECTIONMODES system variable determines how potential conflicts are handled.

Note: When curved entities are offset, the command changes the radii of the copies.

20.3.2 Method

There are four methods to create an offset:

- Enter offset distance
- Through point
- Erase
- Layer

20.3.3 Options within the command

Enter offset distance
Specifies the distance between the original entity and its parallel copy.

Note: The offset distance is saved to the OFFSETDIST variable.

Both Sides
Places offset copies on both sides of the entity.

Multiple
Repeats the command until you press ESC.

Through point
Specifies the offset distance by picking two points.

Multiple
Repeats the command until you press ESC.

Erase
Deletes the source entity after the offset operation.

Note: The status is saved to the OFFSETERASE variable with the drawing, but is then reset (turned off) with the next drawing.

Layer
Specifies the layer on which to place the copied entity.

Current
Places the offset entity on the current layer.

Source
Places the offset on the same layer as that of the source entity.

20.4 OLELINKS command

Opens the Links dialog box.

Shape Lite Pro Mechanical BIM

20.4.1 Description

Opens the Links dialog box to view and manage OLE links.

20.5 OLEOPEN command

Opens the selected OLE object.

Shape Lite Pro Mechanical BIM

20.5.1 Description

Opens the selected OLE object in the source application for editing. (short for "object linking and embedding linking"). If the link between the drawing and the source application is broken, the OLE object cannot open.

Note: Use the InsertObj command to insert OLE objects in drawings.

Note: The command is only available on the windows platform.

20.6 ONWEB command

Opens the default Web browser.

Shape Lite Pro Mechanical BIM

20.6.1 Description

Opens the default Web browser to browse the internet starting from the BricsCAD home page. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

20.7 OOPS command

Restores the last erased entity.

Shape Lite Pro Mechanical BIM

Alias: UNDELETE, UNERASE

20.7.1 Description

Restores the last erased entity, including those erased by theBlockcommand. If there are no erased entities in the drawing, BricsCAD indicates: There is nothing to un-erase.

This command displays no prompts and has no options.

20.8 OPEN command

Opens the Open file dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

20.8.1 Description

Opens the Open file dialog box to select a file to open in the drawing editor.

Supported open file formats are:

- **Standard Drawing File** (.dwg)
- **Drawing Exchange Format** (.dxf)
- **Template Format** (.dwt)
- **Standards File** (.dws)
- **Windows Metafile Formats** (.wmf; .emf; .wmz; .emz)(1) (2)
- **Collada** (.dae)
- **Micro Station DGN File** (.dgn)

Formats available with the BIM add-on:

- **Rhino file** (.3dm)(1) (2)
- **SketchUp file**(.skp)(1) (2)
- **IFC file** (.ifc; .ifczip)
- **Revit Family** (.rfa)(1) (2)
- **Revit Project** (.rvt)(1) (2)

(1) Not available in BricsCAD for Linux.

(2) Not available in BricsCAD for macOS.

20.9 OPENSHEETSET command

Opens the Select a sheet set file dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

20.9.1 Description

Opens the Select a sheet set file dialog box to select a dst file to open in the Sheet Sets panel.

20.10 -OPENSHEETSET command

Opens a sheet set file.

Shape Lite Pro Mechanical BIM

20.10.1 Description

Opens a .dst sheet set file, and then displays the Sheet Sets panel which lists all drawings and sheets associated with the set; meant for use by macros. An example of path and file name of a .dst sheetset file is *F:\work\Documentation\Documentation\project.dst*.

20.11 OPTIMIZE command

Corrects inaccuracies in the drawing, for 2D entities or 3D entities.

Shape Lite Pro Mechanical BIM

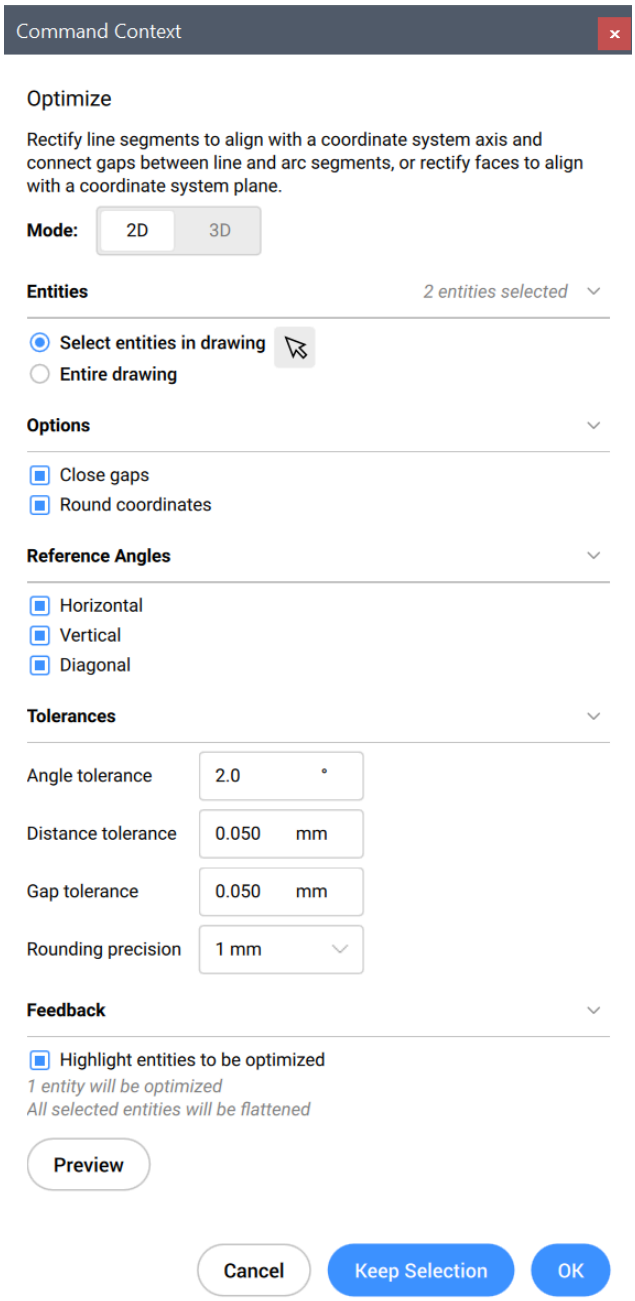
Icon: 

20.11.1 Description

This command works with either 2D entities or 3D entities. The supported 2D entities are line, arc, and polyline. The command makes corrections such as small gaps between lines or near vertical, horizontal and diagonal lines.

The only 3D entities supported are solids. The command makes corrections such as making faces that are near parallel to the orthogonal planes being coplanar to these planes; and making faces that belong to different solids to be coplanar to one another.

This command opens the **Optimize** command panel.



Note:

- The options within the OPTIMIZE command are the same as in the **Optimize** command panel.
- The **Optimize** command panel allows you to toggle On/Off the live Feedback. When turned on, it highlights the entities that match the selected options and displays the number of entities that will be optimized.

20.11.2 Methods

There are two methods to begin optimizing entities:

Entire drawing
When this option is selected, all non-frozen entities in the current viewport are used as input.

selection options (?)
Allows to choose a selection method. See the SELECT command.

20.11.3 Options within the command, in 2D mode

change reference Angles
This option determines which entities will be optimized based on their orientation.

Horizontal lines
Corrects near horizontal lines according to the angle tolerance.

Vertical lines
Corrects near vertical lines according to the angle tolerance.

45-Degree lines
Corrects near 45° lines according to the angle tolerance.

All
Corrects near horizontal, near vertical and near 45° lines according to the angle tolerance.

None
Does not correct entities based on their orientation.

change Tolerances
Allows you to set the tolerances for the parameters used during the optimization.

Angle
Sets the angle tolerance, in degrees, with respect to the World Coordinate System (WCS). Near horizontal, vertical or diagonal lines within this angle tolerance will be optimized.

Distance
Sets the distance tolerance (see **Note**). Parallel lines within this distance will be merged.

Rounding precision
Sets the rounding precision (see **Note**). This option rounds the coordinates to the specified number of digits following the decimal.

Gap tolerance
Sets the gap tolerance (see **Note**). Gaps between collinear lines within the gap tolerance will be filled.

Note: The values are expressed in insertion units (see the INSUNITS system variable).

change Options
Determines if gaps will be closed and if the drawing will be rounded.

close Gaps
If this option is enabled, the gaps between collinear lines that are smaller than the gap tolerance will be closed.

Round coordinates
If this option is enabled, it rounds the coordinates to the rounding precision.
If this option is disabled, the drawing will not be rounded.

Note: This action is controlled by the **Rounding precision** setting

enable All
Enables both **close Gaps** and **Round coordinates** options.

change Input selection
Allows you to make a new selection of entities to be used by the command.

apply settings and PReview
Makes a preview of the rectified segments. You can accept or reset the result.

Keep selection
Clicking this button will cancel the command without optimizing, but with keeping the highlighted lines selected. This option allows you to select suboptimal lines in a drawing.

switch to 3d Mode
Switches the mode to 3D.

20.11.4 Options within the command, in 3D mode

change reference Plane
Determines which plane will be the referenced for correcting faces position. You can choose XY plane, YZ plane, XZ plane or all of them.

change Tolerances
This option allows setting the tolerances for the parameters used during the optimization.

Angle
Sets the angle tolerance, in degrees, for relative position of faces and planes.

Distance
Sets the distance tolerance (see **Note**).
The faces within the tolerance will be made coplanar, parallel or perpendicular, according to the settings.

Rounding precision
Sets the rounding precision (see **Note**). This option rounds the coordinates to the specified number of digits following the decimal.

Note: The values are expressed in insertion units (see the INSUNITS system variable).

change Options
Changes the options for correcting faces position.

make faces Coplanar
Makes the faces in the selection coplanar, with respect to the values in the **Tolerances** section.

rectify faces PArallel to reference planes
Makes the faces in the selection parallel to the selected reference planes, with respect to the values in the Tolerances section.

rectify faces PErpendicular to reference planes
Makes the faces in the selection perpendicular to the selected reference planes, with respect to the values in the Tolerances section.

Round coordinates
If this option is enabled, it rounds the coordinates to the rounding precision.

If this option is disabled, the drawing will not be rounded.

Note: This action is controlled by the **Rounding precision** setting

enable All
All the options for 3D faces (coplanar, parallel, perpendicular and round) will be used by the command.

change Input selection
Allows making a new selection of entities to be used by the command.

apply settings and PReview
Makes a preview of the rectified faces. You can accept or reset the result.

Keep selection
Clicking this button will cancel the command without optimizing, but with keeping the highlighted entities selected. This option allows you to select suboptimal entities in a drawing.

switch to 2d Mode
Switches the mode to 2D.

20.12 OPTIONS command

Opens the Settings dialog box with the **Program options** category expanded.

Shape Lite Pro Mechanical BIM

Icon:

Alias: CFG, CONFIG, OP, PREFERENCES, PREFS

20.12.1 Description

Opens the Settings dialog box with the **Program options** category expanded to view and modify relevant system variables.

20.13 ORTHOGONAL command

Toggles the ORTHOMODE system variable.

Shape Lite Pro Mechanical BIM

Alias: OR, ORTHO

20.13.1 Description

Toggles the ORTHOMODE system variable to specify if the cursor is constrained to dragging at 90 degree angles. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'ORTHOGONAL.

- On: turns on the ORTHOMODE system variable.
- Off: turns off the ORTHOMODE system variable.
- Toggle: changes the ORTHOMODE system variable to the opposite of the current setting.

20.14 OSNAP command

Opens the Settings dialog box with the **Entity snap mode** category expanded.

Shape Lite Pro Mechanical BIM

Alias: DDESNAP, DDOSNAP, OS, SETESNAP

20.14.1 Description

Opens the Settings dialog box with the **Entity snap mode** category expanded to view and modify relevant system variables.

20.15 -OSNAP command

Toggles entity snap modes.

Shape Lite Pro Mechanical BIM

Alias: -OS, ESNAP

20.15.1 Description

Toggles entity snap modes at the Command line (short for "object snap").

Note: Entity snaps help you draw and edit precisely by snapping the cursor to the nearest geometric feature.

20.15.2 Options within the command

NEArest
Toggles nearest entity snap mode. Snaps to the nearest point on any entity.

Note: Can also toggle with the NEAREST command.

ENDpoint
Toggles endpoint entity snap mode. Snaps to the endpoints of open entities, such as lines, arcs, open polylines, and open splines.

Note: Can also be toggled with the ENDPOINT command.

MIDpoint
Toggles midpoint entity snap mode. Snaps to the midpoint of open entities.

Note: Can also be toggled with the MIDPOINT command.

CENter
Toggles center entity snap mode. Snaps to the center of circular entities, such as circles, arcs, and polyarcs.

Note: Can also be toggled with the CENTER command.

GeometriC cEnter
Toggles geometric center entity (GCenter) snap mode. Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids.

Note: Can also be toggled with the GCENTER command.

PERpendicular
Toggles perpendicular entity snap mode. Snaps to the perpendicular of one entity to another.

Note: Can also be toggled with the PERPENDICULAR command.

TANgent
Toggles Tangent entity snap mode. Snaps to the tangencies of circular entities.

Note: Can also be toggled with the TANGENT command.

QUAdrant
Toggles quadrant entity snap mode. Snaps to the quadrant points of circular entities, such as circles, arcs, and polyarcs.

Note: Can also be toggled with the QUADRANT command.

INSertion point
Toggles insertion entity snap mode. Snaps to the insertion point of blocks and text.

Note: Can also be toggled with the INSERTION command.

POInt
Toggles node entity snap mode. Snaps to point entities.

Note: Can also be toggled with the NODE command.

EXTension
Toggles extension entity snap mode. Snaps to the intersection of open entities, as if they were extended to the point of intersection.

Note: Can also be toggled with the EXTENSION command.

PARallel
Toggles parallel entity snap mode. Snaps to the parallel point of entities. Allows to draw lines, polyline segments, xlines and rays parallel to another entity.

Note: Can also be toggled with the PARALLEL command.

INTersection
Toggles intersection entity snap mode. Snaps to the intersection of any pair of entities.

Note: Can also be toggled with the INTERSECTION and 3DINTERSECTION commands.

APParent intersection
Toggles Apparent entity snap mode. Snaps to the apparent intersection of entities in 3D space.

Note: Can also be toggled with the APPARENT and 2DINTERSECTION commands.

NON
Clears entity snaps.

ON
Turns all entity snap modes on.

OFF
Turns all entity snap modes off.

Note: Modes can also be turned off with the NONE command.

20.16 OVERKILL command

Deletes duplicate and overlapping entities and combines duplicate blocks.

Shape Lite Pro Mechanical BIM

20.16.1 Description

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 and optionally purges duplicate block definitions.

After selecting the entities, Delete Duplicate Entities dialog box is displayed.

20.17 -OVERKILL command

Deletes duplicate and overlapping entities and combines duplicate blocks.

Shape Lite Pro Mechanical BIM

20.17.1 Description

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 and optionally purges duplicate block definitions.

20.17.2 Options within the command

Done
Executes the overkill procedure.

Ignore
Sets the properties to ignore.

None
Properties are taken into account to determine if overlapping entities should be combined.

All
All properties are taken into account.

Color, etc.
The colors of overlapping entities are considered. If the colors match, then they are combined or deleted.

TOlerance
Sets the tolerance for the comparison process.

Note: When is set to 0, entities must match completely before being evaluated in the overkill process.

optimize Plines
Segments of selected polylines are evaluated individually, duplicate vertices and segments are removed. If lines or arcs duplicate a polyline segment, one of them is deleted, which might break a polyline.

segment witDth
Specifies if the width property of polyline segments is ignored.

Break polyline
Polyline segments are kept intact even if the duplicates are deleted.

Yes
Optimizes polyline entities.

No
Polyline entities are not optimized.

combine parTial overlap
Partially overlapping entities are unified into a single entity.

combine Endtoend
Collinear lines and polyline segments having coinciding endpoints are unified into a single entity or segment.

Associativity
Associative entities are not processed.

delete or Move duplicates
Determines what to do with duplicate entities.

Move
Entities are moved to Overkill Layer.

Note: This layer is set by the OVERKILLLAYER system variable. By default, it is set to Duplicate Entities.

Combine duplicate block definitions
Searches the drawing for duplicate block definitions and replaces all such block instances by the most recent of these duplicates.

Purge duplicate block definitions
Deletes duplicate block definitions.

21. P

21.1 PACKAGE command

Creates a package ZIP file of the drawing and its dependencies.

Shape Lite Pro Mechanical BIM

See the -ETRANSMIT command.

21.2 PAGESETUP command

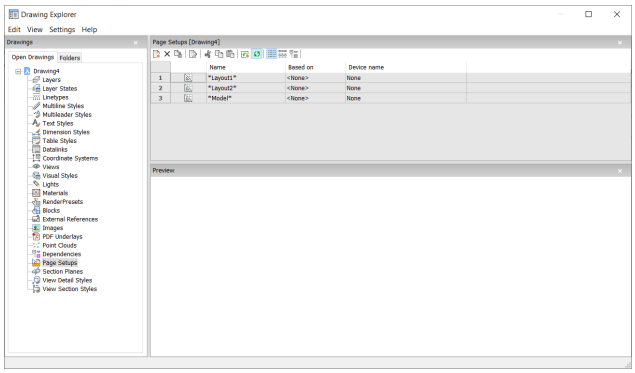
Opens the Drawing explorer dialog box with **Page Setups** selected.

Shape Lite Pro Mechanical BIM

Icon: 

21.2.1 Description

Opens the Drawing explorer dialog box with **Page Setups** category selected to view and modify page setups in the current drawing.



21.2.2 Context Menu Options

New

Loads additional page setup definitions into the drawing. Displays the New Page Setup dialog box.

Delete

Deletes page setup definitions from the drawing. The following page setup definitions cannot be deleted:

- The Model tab
- The last layout

Edit Page Setup

Edit the selected page setup. Displays the Page Setup dialog box. Here you can edit the properties of the selected page setup.

Rename

Renames the layout tabs.

Note: Following linetypes cannot be renamed:

- The Model tab
- The Layout tabs

Select All
Selects all page setup definitions.

Invert selection
Deselects the current selection and vice versa.

21.3 PAN command

Moves the entire drawing.

Shape Lite Pro Mechanical BIM

Icon:

Alias: P

21.3.1 Description

Moves the entire drawing in real- time within the current viewport.

Note: Real-time means that the drawing moves as the mouse moves.

Note: The PERSPECTIVE system variable must be set to 0.

After running the command, the cursor changes into a hand icon. Hold down the left mouse key and then move the mouse to pan the drawing or right-click to open the Pan context menu.

Note: Pan the drawing also with the scroll bars. Turn them on with the SCROLLBAR command.

21.4 -PAN command

Moves the entire drawing.

Shape Lite Pro Mechanical BIM

Alias: -P

21.4.1 Description

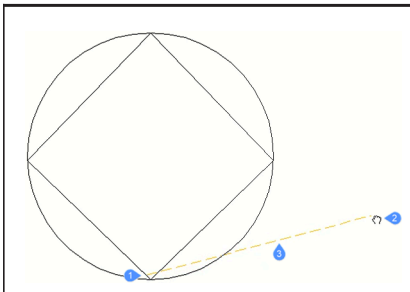
Moves the entire drawing within the current viewport.

Note: The PERSPECTIVE system variable must be set to 0.

21.4.2 Method

There are two methods to use -PAN command:

- Pick a base point and a displacement point.
- Choose one of the predefined options.

	1 Pan base point. 2 Pan cursor which specifies the pan displacement point. 3 Pan displacement.

21.4.3 Options within the command

Pan base point
Specifies the pan distance's starting point.

Pan displacement point
Specifies the pan distance's ending point.

Note: The drawing is panned by 5% or 100% of the width of the current viewport.

Left
Pans the drawing 5% to the right.

Right
Pans the drawing 5% to the left.

Up
Pans the drawing 5% downwards.

Down
Pans the drawing 5% upwards.

PaGe Left
Pans the drawing 100% to the right.

Note: You can also press Shift + Left cursor key.

PaGe Right
Pans the drawing 100% to the left.

Note: You can also press Shift + Right cursor key.

PaGe Up
Pans the drawing 100% upwards.

Note: You can also press Shift + Up cursor key.

PaGe Down
Pans the drawing 100% downwards.

Note: You can also press Shift + Down cursor key.

21.5 PANELIZE command

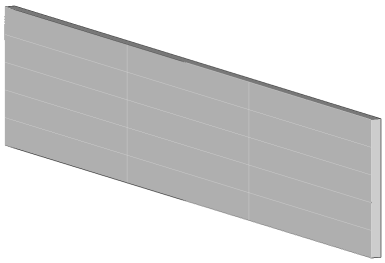
Creates a grid on the face of a 3D Solid, as a block reference.

Shape Lite Pro Mechanical BIM

Icon:

21.5.1 Description

Creates a custom grid on the face of a 3D Solid, by mentioning the length and the number of panels.



21.5.2 Options

Length U panels
Sets the length of the panels in U direction (height).

Length V panels
Sets the length of the panels in V direction (width).

Number U panels
Number of panels in U direction (rows).

Number V panels
Number of panels in V direction (columns).

21.6 PARALLEL command

Toggles the Parallel entity snap.

Shape Lite Pro Mechanical BIM

Icon:

21.6.1 Description

Toggles the Parallel entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

21.7 -PARAMETERS command

Manages dimensional-constraint parameters.

Shape Lite Pro Mechanical BIM

21.7.1 Description

Creates, edits, renames, and deletes parametric equations made of dimensional constraints and user-defined variables; works at the Command line.

21.7.2 Options

?
Lists all dimensional constraints and user variables in the current drawing, showing the name, expression and current value of each dimensional constraint or user variable.

New
Creates a user variable.

Edit
Edits the expression for the dimensional constraint or user variable.

Rename
Renames a dimensional constraint or user variable.

Delete
Deletes a dimensional constraint or user variable.

21.8 PARAMETERSPANELCLOSE command

Closes the Parameters and constraint panel.

Shape Lite Pro Mechanical BIM

21.8.1 Description

Closes the Parameters and constraint panel to hide it from the current workspace. If the Parameters and constraint panel is stacked when you close it, the Parameters and constraint tab or icon is removed from the stack.

21.9 PARAMETERSPANELOPEN command

Opens the Parameters and constraints panel.

Shape Lite Pro Mechanical BIM

Icon: 

21.9.1 Description

Opens the Parameters and constraints panel to display it in the current workspace. The Parameters and constraints panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Parameters and constraints panel can be either floating, docked or stacked.

21.10 PARAMETRICBLOCKIFY command

Automatically replaces sets of entities, matching a selected parametric block or a set of parametrized entities in a model, by parametric block references.

Shape Lite Pro Mechanical BIM

Icon: 

21.10.1 Description

The PARAMETRICBLOCKIFY command replaces sets of entities in the current drawing by parametric blocks. To make the replacements, this command uses some input entities. There are two kinds of input entities:

- A reference to a block definition containing parametrized entities.
- A set of parametrized entities.

The overall workflow of this command include the following steps:

- First, the command defines some input entities to be used for comparison. These input entities can be parametric block or sets of parametrized entities.
- The geometry of each input entity is driven by some parameters. A configuration is defined by the values of the parameters.
- After the input entities have been defined, the command searches the current drawing for sets of entities with a geometry that match the geometry of one of the configurations.
- A match occurs when the input entities can be configured exactly as the set of entities which that input is compared to.
- After a match has been found, the set of entities will be replaced by a parametric block. The parameters will have the values specific to the configuration for which the match has been found.

Note:

- The BLOCKIFYMODE system variable defines the behavior of the PARAMETRICBLOCKIFY command. Depending on the value of the system variable, some options are not available.
- The PARAMETRICBLOCKIFY command does not support 2D parameter and constraints. It will not make any replacement when used with 2D parametric blocks or sets of parametrized 2D entities. An exception is the case when the geometry of a set of entities is exactly equal to the geometry of the entities inside the 2D parametric block definition. For such an exact match, the command will make the replacement.

21.10.2 Methods

There are three methods of using PARAMETRICBLOCKIFY command:

- Using a single parametric block selected in the drawing.
- Using a set of parametrized entities selected in the drawing.
- Using the block definitions from the library

Using a single parametric block

- Select the parametric block to create the input entity.
- Search through the current drawing for sets of entities that match the input entity.
- A match is when the input entity can be configured exactly as the set of entities being searched.
- When a match is found, the set of entities will be replaced by an instance of the parametric block.

- If there is no match, the command makes no changes to the drawing.

Using a set of parametrized entities

- Select a set of parametrized entities to create the input.
- Search through the current drawing database for sets of entities that match the input.
- A match is when the input entities can be configured exactly as the set of entities being searched.
- When at least one match is found, the input entities will be converted into a parametric block definition. The input and the matching sets of entities will then be replaced by instances of the newly created parametric block.
- If there is no match, the command makes no changes to the drawing.

Using block definitions from the library

- The inputs used in this case are the parametric blocks in the library.
- In this case, there are many inputs. Each parametric block in the library is an input.
- Search through the current drawing for sets of entities that match one of the inputs.
- A match is when the one of the inputs can be configured exactly as the set of entities being searched.
- When a match is found, the set of entities will be replaced by an instance of the parametric block that matched.
- If there is no match, the command makes no changes to the drawing.

21.10.3 Options within the command

Select input parametric block or parametrized entities
This option allows to make a selection. The type of selection will determine which method will be used by the command.

Settings
Displays the BLOCKIFYMODE system variable in the Settings dialog box. For more information see the BLOCKIFYMODE system variable.

selection options (?)
Allows to choose a selection method.

check Library
This option allows to select blocks outside the drawing to be used in the search for matching.

Use Library components path or [select Folder/Library] <Library>
This prompt allows to select either the library or a specified folder as the source of parametric blocks to be used by the command.
The default option is **Library**. This option will use the blocks in the library as inputs to match.

Note:

- The library search path is *<Install_Folder>/UserDataCache/Support/<Language>/BIM/Components*. This path does not include the standard parts.
- The **Library** option also supports library components that are not parametrized.
- If the **Select Folder** option is selected, the user can select a folder containing .dwg files. In this case, the geometry in each of the files will be used as input.

Select search space or [Entire drawing/selection options (?)] <Entire drawing>:
This option allows to select where the search will be performed. The user can select an area of the drawing or the entire drawing.

Select parametric block insertion point or [use Default point] <use Default point>:
This option is available only for the parametrized entities method.
It allows to select an insertion point for the parametric block that is to be created after a match has been found. The default point is calculated by the command depending on the geometry of the selected parametrized entities.

Specify parametric block name or <ENTER to use default name>:
This option is available only for the parametrized entities method.
It allows the user to enter the name of the parametric block that is to be created. The default block name is **Param_Block** to which an incrementing number is added.

21.11 PARAMETRICSTRETCH command

This command defines a parameter to be used by a stretch operation on some entities of a parametric block.

Shape Lite Pro Mechanical BIM

Icon: 

21.11.1 Methods

This command allows to stretch some entities in a block. The appearance of the block can be controlled by changing the value of the parameter associated with the stretch operation.

The goal of this parametric stretch action is to simplify the stretching of entities in the block. Usually, the same behavior could be obtained with parameters and constraints, but it would take more time and effort to obtain the same behavior.

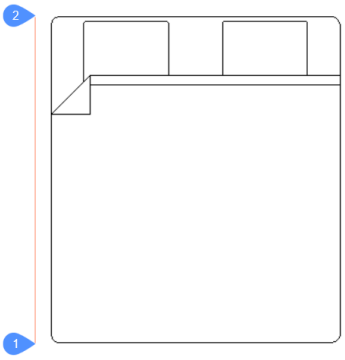
The PARAMETRICSTRETCH command can be launched from the command line by typing _PARAMETRICSTRETCH. Another way to launch this command is from the ribbon in a **Drafting** workspace. Go to tab **Parametric** > **Parametric Blocks panel**.

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.

21.11.2 Options within the command

Base point of displacement
A location point for the origin of the stretch vector.

Second point of displacement
A location point for the head of the stretch vector.
For example, when defining a parametric block for a bed symbol, the stretch vector could look like this:



The point marked with 1 is the base point of displacement (the origin of the vector) and the point marked with 2 is the second point of displacement (the head of the vector). The two points are located on a vertical line at a distance equal to the length of the bed. The points can be placed on the actual geometry if this is more convenient for the user. Here, for visualization purposes, the points have been placed on the side of the block.

The parametric stretch vector will have a soft red color and will be placed on a special layer named PARAMETRIC_STRETCH_VECTORS.

Construct stretch frame: pick first point of rectangular frame [Polygonal]

The first corner of the rectangular stretch frame.

Opposite corner

The second corner of the rectangular stretch frame .

Polygonal

Allows to enter a series of points that will set a polygonal stretch frame.

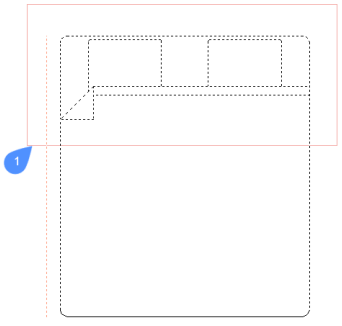
Note: The vertices inside the selection contour (rectangle or polygon) will be moved according to the stretch vector.

Select/deselect entities which can be affected by the stretch [selection options (?)]:

Allows to edit the selection of entities to be affected by the stretch action.

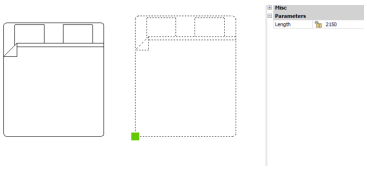
By default, all the entities which have stretch points inside the stretch frame are selected. Using this option, the user can change the selection. All the selected entities will be highlighted.

At this step, the screen will look like this:



The contour marked with 1 is the stretch frame. The vertices that are inside this contour and belong to a selected entity, will be affected by the parametric stretch.

Enter name for the stretch parameter <Stretch>:
Allows to give a name to the stretch parameter. The default name is **Stretch**.
For this example, the stretch parameter is named **Length**.
To see the block at work, open a new drawing and insert the block. After insertion, with the block selected, go to the **Properties** panel in the **Parameters** section, and change the value of the stretch parameter, **Length** in this case.
The block entities that were selected for stretch will adjust according to the new value of the stretch parameter.



21.12 PARAMETRIZE command

Applies constraints automatically.

Shape Lite Pro Mechanical BIM

21.12.1 Description

Automatically adds geometric constraints and parametric equations to 3D solid geometry. The resulting parameters and geometric constraints can be viewed and edited in the Mechanical Browser (MECHANICALBROWSEROPEN command).

21.13 PARAMETRIZE2D command

Automatically applies 2D geometric and dimensional constraints to selected 2D geometry.

Shape Lite Pro Mechanical BIM

21.13.1 Description

Select the 2D entities to automatically add dimensional and geometrical constraints.

The parameters and constraints can be edited in the Parameters and constraints panel or in Mechanical browser.

21.14 PASTEBLOCK command

Pastes CAD entities from the Clipboard as a block.

Shape Lite Pro Mechanical BIM

Icon:

Note: Before you can use this command, first use the COPYCLIP or COPYBASE commands to copy entities from the current drawing, or even from another CAD program.

21.14.1 Description

This command can only paste CAD entities into the drawing as block entities. Other Clipboard content may be pasted as OLE entities.

21.15 PASTECLIP command

Pastes entities from the Clipboard into the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Note: Before you can use this command, first use the COPYCLIP or COPYBASE commands to copy entities from the current drawing, or even from another CAD program.

21.15.1 Method

This command can only paste CAD entities into the drawing as entities.

When the Clipboard contains non-BricsCAD entities:

- Non-BricsCAD entities, such as raster images, are pasted as OLE entities.
- If non-BricsCAD text (including LISP and Diesel code) is pasted in the Command line, the program executes the text as commands.

21.15.2 Options within the command

Rotate
Specifies the angle by which to rotate the entities.

Scale
Specifies the scale factor.

Mirror
Mirrors the entities about a mirror line in a 2D plane.

21.16 PASTEORIG command

Pastes entities from the clipboard into a different drawing.

Shape Lite Pro Mechanical BIM

Icon: 

Note: Before you can use this command, first use the COPYCLIP or COPYBASE commands to copy entities from the current drawing.

21.16.1 Description

Pastes CAD entities from the Clipboard into a different drawing using the coordinates of the entities in the source drawing. PASTEORIG comes from "paste at original coordinates".

Note: This command does not paste entities back into the source drawing.

21.17 PASTESPEC command

Opens the Paste special dialog box.

Shape Lite Pro Mechanical BIM

Icon:

Alias: PA

Note: This is a Windows only command.

21.17.1 Description

Opens the Paste special dialog box to specify how entities from the clipboard are pasted into the current drawing. This command is only available on the Windows platform.

21.18 PDF command

Opens the Attachments panel.

Shape Lite Pro Mechanical BIM

Icon:

21.18.1 Description

Opens the Attachments panel to display it in the current workspace. The Attachments panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Attachments panel can be either floating, docked or stacked.

21.19 PDFADJUST command

Adjusts graphical properties of a PDF underlay.

Shape Lite Pro Mechanical BIM

Icon:

21.19.1 Description

Adjusts the face, contrast and monochrome level of one or more PDF underlays attached to the current drawing.

Note: It can be easier to use the Properties panel's Underlay Section to adjust the parameters.

21.19.2 Options

Fade

Sets the fade setting of the PDF underlay(s).

- 0: minimum fade: Underlay PDF is fully displayed
- 100: maximum fade: Underlay PDF is barely visible

Contrast
Sets the contrast setting of the PDF underlay(s).

- 0: minimum contrast: both dark and light elements are displayed medium gray
- 100: maximum contrast: dark elements are displayed dark, light elements are displayed light

Monochrome
Toggles the monochrome setting of the PDF underlay(s).

Yes
Turns colors into shades of gray.

No
Keeps the PDF as-is.

21.20 PDFATTACH command

Opens the Select PDF underlay file dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

21.20.1 Description

Opens the PDF underlay file dialog box to select a PDF file to reference into the current drawing. After selecting the file and choosing **Open**, the Attach PDF Underlay dialog box displays. It enables you to specify where and how to attach the PDF file.

21.21 -PDFATTACH command

Attaches a PDF file as an underlay.

Shape Lite Pro Mechanical BIM

21.21.1 Description

Attaches a PDF file as an underlay into the current drawing through the command bar.

21.21.2 Options

PDF Underlay file
Specifies the file name of the PDF file to attach by entering the path of PDF name.

Note: To open the File dialog box, enter ~ .

PDF underlay page number
Specifies the page number of a multi-page PDF file to insert.

Note: This option does not appear when the PDF has only one page.

Insertion point
Specifies the insertion point in the drawing. This can be done in 2 ways.

- Picking a point
- Entering x, y coordinates

Scale
Specifies the size of the PDF underlay by entering a scale factor or by moving the cursor.

Size
Sets the size of the insert dynamically. The first point is the insertion point. By moving the cursor, the program ghosts the size and position of the underlay relative to the insertion point.

XY Scale factors
Scales the underlay separately in X and Y direction differently.

Rotation Angle
Specifies the angle of the underlay.

21.22 PDFCLIP command

Clips a PDF underlay with a boundary.

Shape Lite Pro Mechanical BIM

Icon: 

21.22.1 Description

Clips a PDF underlay with a rectangular or polygonal boundary and inverts the boundary.

21.22.2 Options

PDF Underlay file
Selects the PDF underlay to clip by clicking the PDF border.

ON
Turns the clipping boundary on. The area outside the clipping Boundary is hidden from the view.

OFF
Turns the clipping boundary off. All of the PDF underlay is visible. The clipping boundary is saved.

Delete
Deletes an existing clipping boundary.

Invert
Inverts the clipping boundary. If the PDF underlay outside the clipping boundary is hidden before it will be visible afterward and the PDF underlay inside the clipping boundary is hidden. Use this option to invert it, so that the portion of the PDF inside the boundary is hidden, and that outside is visible.

Note: : You may find it more convenient to use the Misc section of the Properties panel to turn the clipping on and off, and to invert it.

New
Draws a new clipping boundary. The type needs to be specified.

Polygonal
Draws multi-sided clipping boundaries. The start and next point are specified by picking points.

Note: To undo the last point, press U.

Rectangular
Draws rectangular clipping boundary by picking two opposite corners.

21.23 PDFIMPORT command

Imports a PDF and converts its content to simple CAD entities.

Shape Lite Pro Mechanical BIM

Icon: 

21.23.1 Method

This command also converts a PDF underlay already attached to the drawing to CAD entities.

Different conversions are made depending on the type of the PDF entity.

PDF entity	CAD Entity
Straight and curved lines	Polylines and splines
SHX text	Polylines
TrueType text	MText
Solid-filled areas and wide lines	Hatches with 50% transparency
Raster images	Raster images
Layers	Layers
Lines with linetypes	Polyline segments

There are 2 methods to convert a PDF to CAD entities

- PDF Underlay
- PDF file

21.23.2 Options

21.23.3 Underlay options

Specify area
Picks the first two points to convert a rectangular area of an underlay to CAD entities.

Polygonal
Picks three or more points to convert a polygonal area of an underlay to CAD entities.

Note: When picking points that are not on the PDF, BricsCAD ignores those points so more points need to be picked.

All
Selects the entire PDF underlay for conversion to CAD entities.

Settings
Displays the PDF Import Settings section of the Setting dialog box.

Keep
Keeps the PDF underlay in place, in addition to converting its vector portions to CAD entities.

Detach
Detaches the PDF underlay so that it is no longer seen in the drawing but is still attached to the drawing.

Unload
Unloads the PDF underlay so that it is no longer seen the drawing but is still attached to the drawing.

21.23.4 Import file Options

Page number
Imports a specific page from the PDF file. When pressing enter, the first page will be imported.

?
Lists the page number in the PDF document.

Note: To view the content of the pages before importing them, use the File Managers preview window.

Settings
Displays the PDF Import Settings section of the Settings dialog box.

Insertion point
Specifies the insertion point in the drawing.

Scale
Specifies the scaling factor or the inserted geometry.

Rotation
Specifies the rotation angle.

Note: The angle needs to be entered and cannot be specified by picking points in the drawing.

21.24 -PDFIMPORT command

Imports a PDF and converts its content to simple CAD entities.

Shape Lite Pro Mechanical BIM

For more information see PDFIMPORT command.

21.25 PDFLAYERS command

Toggles the display of layers in PDF underlays.

Shape Lite Pro Mechanical BIM

21.25.1 Description

Toggles the display of layers in PDF underlays. After entering the command and selecting a PDF underlay, the dialog box Underlay Layers displays. Here you can edit the display of the layers of the PDF.

21.26 PDFOPTIONS command

Opens the Settings dialog box with the **PDF export** category expanded.

Shape Lite Pro Mechanical BIM

21.26.1 Description

Opens the Settings dialog box with the **PDF export** category expanded to view and modify relevant system variables.

21.27 PEDIT command

Edits polylines.

Shape Lite Pro Mechanical BIM

Icon:

Alias: EDITPLINE, PE

21.27.1 Description

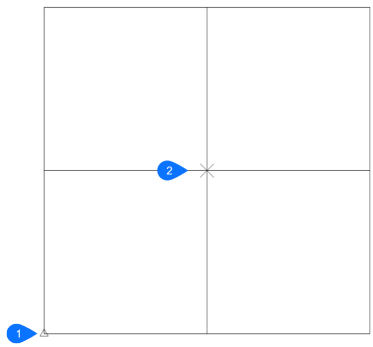
Edits polylines, 3D polylines, and 3D meshes (short for "polyline edit") and converts 2D entities into polylines.

Note: The options of this command change, depending on the entity you are editing. You can edit single or multiple 2D polylines, single 3D polylines, 3D meshes and single or multiple lines, arcs, circles, splines or helixes.

21.27.2 Options

Edit vertices

Edits the vertices.

	1 Triangle marker at the start of the polyline. 2 X marker at the current vertex.
	1 Triangle marker at the home vertex. 2 X marker at the current vertex.

Note: It can be easier to edit the vertices with grips editing.

Next vertex
Moves the X marker to the next vertex. When the marker reaches the last vertex of a closed polyline, it does not move to the "next" (first) vertex.

Previous vertex
Moves the X marker to the previous vertex.

Angle
Changes the angle of the current segment. If it is an arc segment, the angle (curvature) of the arc is changed. If it is a line segment, it is turned into an arc. To convert an arc into a line segment, enter an angle of zero degrees.

Note: angle > 0 is counterclockwise, angle = 0 is straight, angle < 0 is clockwise.

Break
Removes segments from the polyline between the two vertices marked with X markers. This action turns closed polylines into open polylines and open polylines into two polylines, with a gap between them.

Insert vertex
Inserts a vertex. A drag line previews the location of the new vertex.

Left
Moves the X marker to the vertex on the left.

Right
Moves the X marker to the vertex on the right.

Up
Moves the X marker to the higher-numbered vertex.

Down
Moves the X marker to the lower-numbered vertex.

Next
Choose the next vertex.

Previous
Choose the previous vertex.

Select
Moves second X marker directly to the selected vertex.

Go
Break the polyline between the first and second marked vertices.

Note: It is much easier to use the BREAK or TRIM commands to remove part of a polyline.

Move
Moves the current vertex. The drag line previews the new location of the vertex.

Note: It is much easier to move the vertices of a polyline with grips editing.

Regen
Regenerates the polyline to show changes made by the editing actions of this command.

SElect
Moves the X marker directly to another vertex that you select. This works faster than using the Next and Previous options.

Straighten
Removes all segments between two vertices. When you select the start and end points of a multi-segment polyline, the entire polyline is straightened, becoming a single segment; this also applies to closed polylines.

Next
Moves the second X marker to the next vertex.

Previous
Moves the second X marker to the previous vertex.

Select
Moves second X marker directly to another selected vertex.

Go
Replaces the segments between the two marked vertices with a single segment.

Tangent
Edits the tangent direction of the vertex.

Width
Changes the width of the current segment. The current segment is the segment that lays between the X-marked vertex and the next vertex.

Close
Closes the polyline by adding a segment between the start and endpoints. When the polyline is closed, this option reads Open.

Open
Opens closed polylines by removing the last segment drawn to close the polyline. When the polyline is open, this option reads Close.

Decurve
Reverses the effects of the Fit and Spline options.

Desmooth
Removes the Bezier-spline smoothing of 3D meshes.

Fit
Fits a curve to the polyline.

Note: To unfit the polyline, use the Decurve option.

Join
Adds open entities to the selected polyline.

Note: This option works only with open and with cojoined entities. Cojoined means that the endpoints of entities meet, such as when drawn with ENDpoint object snap or with the Last point option. The joined entities take on the source polyline's properties, such as color, width, and layer.

Fuzz distance
When endpoints are within the fuzz distance, the entities are extended or trimmed.

Jointype
Specify the type of joining.

Extend
Extends or trims segments to the nearest endpoints.

Add
Adds linear segments between the nearest endpoints.

Both
Extends or trims if possible, otherwise adds linear segments.

M close
Closes the 3D mesh in the m direction. When the mesh is closed, this option reads M open.

M open
Opens the 3D mesh in the m direction. When the mesh is open, this option reads M close.

N close
Closes the 3D mesh in the n direction. When the mesh is closed, this option reads N open.

N open
Open the 3D mesh in the n direction. When the mesh is open, this option reads NClose.

Linetype mode
Determines how linetypes appear on polylines.

On
Linetypes start and stop at the polyline's start and endpoints.

Off
Linetypes start and stop at each vertex.

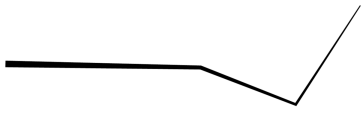
Note: This option is stored in the PLINEGEN variable.

Reverse direction
Reverses the direction of the polyline: the starting point is now its end point, and vice versa. The polyline does not change its look with this operation, except that the triangle marker moves to the other end of open polylines. This option affects operations that depend on the direction of a polyline, such as vertex editing.

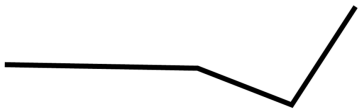
Spline
Converts the polyline into a spline.

Note: all width information is lost. Use the Width option to reapply width. The spline is a Bezier-spline whose smoothing is defined by the SPLINETYPE system variable.

Taper
Tapers the width of the entire polyline from one end to the other.



Width
Changes the width of all segments.



Note: This option overrides tapered widths.

Undo
Undoes the last action.

21.28 PEDITEXT command

Edits vertices and segments of a polyline.

Shape Lite Pro Mechanical BIM

Icons: 

21.28.1 Description

Edits vertices and segments of a polyline interactively, without first selecting the polyline.

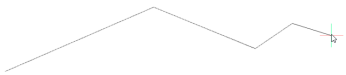
Note: This command was meant for use with the Quad cursor. See the related procedure Editing a polyline using the Quad on how to use this command with the Quad.

21.28.2 Options

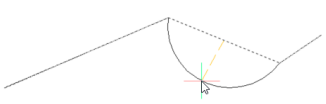
Add vertex
Adds a vertex to the selected segment.



add vertex at the End
Adds a new vertex at the end of the polyline.

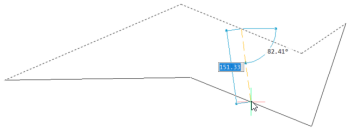


adjust Bulge
Modifies the bulge factor of the selected segment.

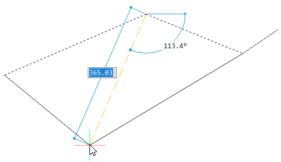


Remove segment
Deletes the selected segment.

Stretch segment
Moves the selected segment.



Stretch Vertex
Moves the selected vertex.



21.29 PERPENDICULAR command

Toggles the Perpendicular entity snap.

Shape Lite Pro Mechanical BIM

Icon:

21.29.1 Description

Toggles the Perpendicular entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

21.30 PFACE command

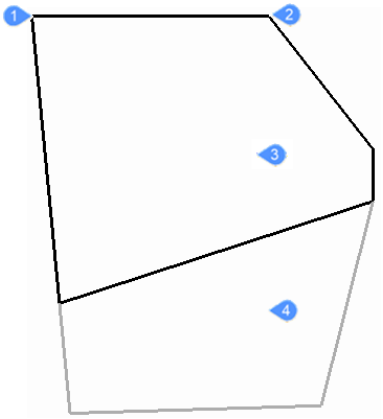
Draws multi-sided meshes.

Shape Lite Pro Mechanical BIM

Icon:

21.30.1 Description

Draws multi-sided polyface meshes in 3D; meant for use by macros.



1 Vertex 1

2 Vertex 2

3 Face 1

4 Face 2

21.30.2 Options

Invisible
(Hidden option.) Specify an invisible edge by entering a negative number.

Color
Specify the color of the edge by entering a color name, number, or color book name.

Layer
Specify the layer name for the edge by entering the name of a layer.

21.31 PLACEVIEW command

Import named views from a source drawing.

Shape Lite Pro Mechanical BIM

21.31.1 Description

Places named views from a source drawing into a paper space layout of the current drawing.

Note: This command works only in paper space, and is meant for use with sheet sets.

Note: It is not allowed to place a named view in a layout of the source drawing.

21.31.2 Methods

There are two methods to place named views:

- Command line: start the command by typing placeview in the Command line.
- Mouse: drag a model view from the Content Browser panel onto a paper space layout.

21.31.3 Options

Enter file name
Enter the file name from the source file from which the views will be imported.

Enter view name to place
Enter the name of the view you want to import.

?
Prints a list of the named views in the Prompt History window.

Enter view origin point
Specify the location of the lower left corner of the named view.

21.32 PLAN command

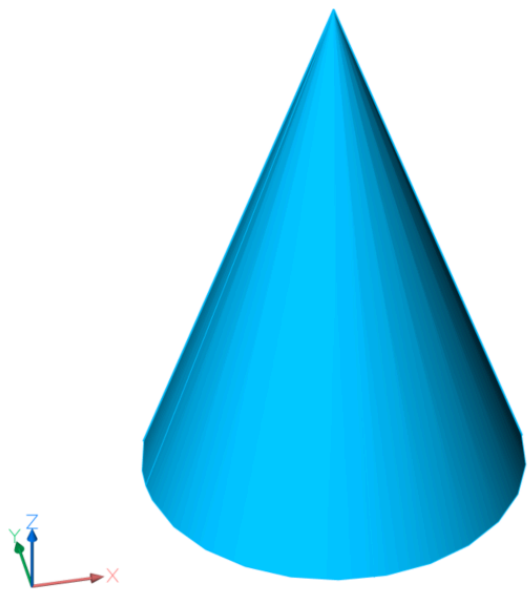
Displays the plan viewpoint of drawings.

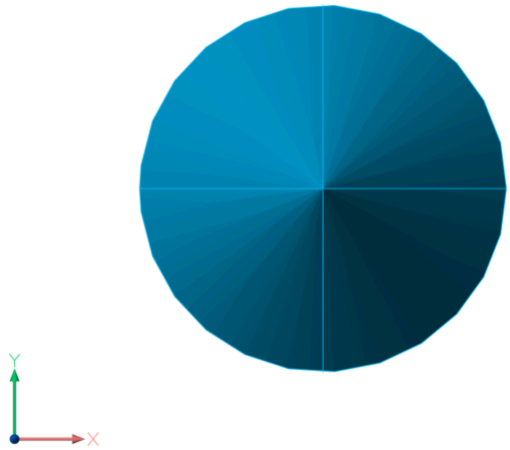
Shape Lite Pro Mechanical BIM

Icon:

21.32.1 Description

Shows the 3D model in plan view which looks straight down on the x,y-plane.





21.32.2 Options

UCS
Displays the plan view of a named UCS or Dynamic UCS after entering a name or placing the cursor over a 3D solid.

Note: To be able to run this command the second way, make sure the system variable DUCS (insert link) is turned on.

?
Lists the names of UCSs in the current drawing.

World
Displays the plan view of the World Coordinate System.

Current
Displays the plan view of the current UCS.

21.33 PLINE command

Creates a polyline.

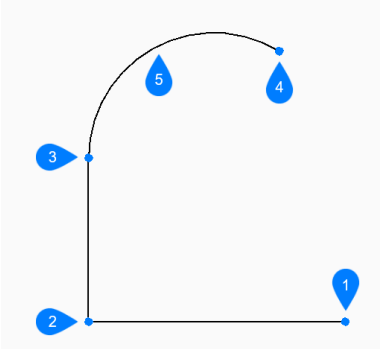
Shape Lite Pro Mechanical BIM

Icon:

Alias: PL, POLYLINE

21.33.1 Description

Creates a single polyline entity with multiple line and arc segments by specifying the start and end point of each segment. Options allow you to switch between line and arc segments, apply widths, undo, and close the geometry.



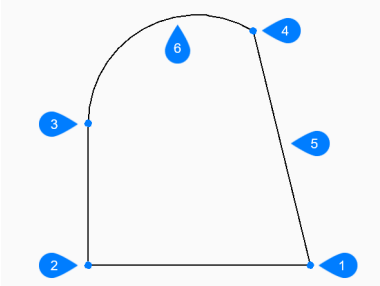
1 Start

2 Next

3 Next

4 End

5 Arc segment



1 Start/End

2 Next

3 Next

4 Next

5 Close

6 Arc segment

21.33.2 Methods to create a polyline

This command has 3 methods to begin creating a polyline:

- Start of polyline
- Last point
- Follow

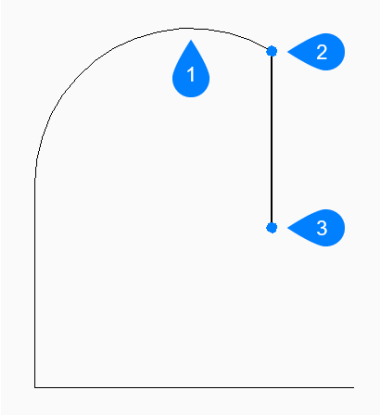
You can continue adding unlimited segments until you press Enter to end the command.

Start of polyline
Begin creating a polyline by specifying a start point then:

Set next point
Specify the next vertex of the polyline.
Additional options: [draw Arcs/Close/Distance/Follow/Halfwidth/Width/Undo]

Last point
Begin creating a polyline from the last point picked then:

Set next point
Specify the next vertex of the polyline.



1 Last drawn segment

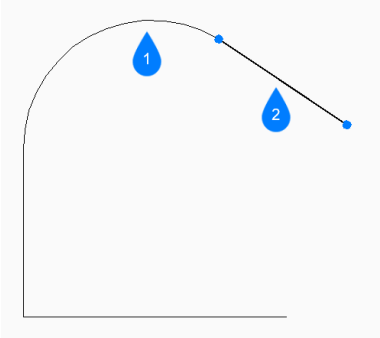
2 Last point

3 End point

Additional options: [draw Arcs/Close/Distance/Follow/Halfwidth/Width/Undo]

Follow
Begin creating a polyline from the last drawn arc or line segment, following its angle, then:

Set next point
Specify the length of the segment. Picking a point only determines the length since the angle follows the previous segment.



1 Last drawn segment

2 Follow

21.33.3 Options within the PLINE command

After you begin creating a polyline, the following options may be available.

Draw arcs
Specify the end point of the polyline arc segment. The arc is drawn tangent to the previous segment.
Additional options: [draw Lines/Angle/CEnter/Direction/Halfwidth/Radius/Second point/Width]

Angle
Specify the included angle of the arc segment then:

Set end of arc
Specify the next vertex of the polyline.
Additional options: [Center/Radius]

Center
Specify the center point of the arc segment then:

Set end of arc
Specify the next vertex of the polyline.
Additional options: [Angle/Length of chord]

Direction
Specify the direction of the arc segment then:

End of arc
Specify the next vertex of the polyline.

Radius
Specify the radius of the arc segment then:

Set end of arc
Specify the next vertex of the polyline.
Additional options: [included Angle]

Second point
Specify a point along the arc's circumference.

Draw Lines
Specify the end point of the line segment.
Additional options: [draw Arcs/Close/Distance/Follow/Halfwidth/Width/Undo]:

Distance
Specify the distance for the line segment then:

Angle of segment
Specify the angle of the line segment.

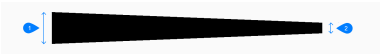
Halfwidth
Specify the starting and ending half-width of the segment, as well as all segments that follow -- until you change the width or halfwidth.



1 Halfwidth

The width is saved in the PLINEWID variable.

Width
Specify the starting and ending width of the segment, as well as all segments that follow -- until you change the width or halfwidth.



1 Start width

2 End width

The width is saved in the PLINEWID variable.

Undo
Undo the last polyline segment and continue drawing from its previous start point.

Close
Automatically draw a polyline segment from the end point of the last segment to the start point of the first segment. This ends the command.

21.34 PLOT command

Opens the Print dialog box.

Shape Lite Pro Mechanical BIM

21.34.1 Description

Opens the Print dialog box to specify print options and preview or print the current drawing.

21.35 -PLOT command

Plots drawings to printers and to files.

Shape Lite Pro Mechanical BIM

21.35.1 Description

Plots drawings to printers and to files, through the Command line.

Note: This command is meant for scripts and routines.

21.35.2 Method

Choose whether to proceed a detailed plot configuration.

21.35.3 Options

Detailed plot configuration?
Choose whether to change the plot configuration.

Yes
Prompts 16 options to configure the plot.

No
Prompts 6 options.

Enter a layout name or ?
Specifies the name of the layout to plot. Enter ? to list the layouts in the current drawing.

Enter an output device name or ?
Specifies the name of the output device (plotter or file). Enter ? to list the available plot devices.

Enter paper size or ?
Specifies the name of the paper size. Enter ? to list the available media sizes.

Enter paper units
Specifies the paper units: inches or millimeters.

Enter drawing orientation
Specifies the drawing orientation: portrait or landscape.

Plot upside down?
Plots the drawing upside down.

Note: This option is useful when paper with a drawing border is loaded backwards in the printer.

Enter plot area
Specifies which area of the drawing to plot.

Display
Plots the area shown in the current viewport.

Extents
Plots the extents of the drawing, ensuring all non-frozen entities are plotted.

Limits
Plots the limits of the drawing, as defined by the Limits command.

View
Plots the current view, or a named view. See View command.

Window
Plots a rectangular area defined by the user.

Enter plot scale
Specifies the plot scale factor.

Note: The Fit option calculates the scale factor automatically.

Enter plot offset
Specifies the offset distance for the print.

Note: 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 and negative values move the drawing down and to the left.

Note: 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.

Plot with plot styles?
Toggles the use of plot styles.

Note: Plot styles override lineweight settings.

Enter plot style table name
Specifies the plot style table to use, which assigns properties to "pens," colors, and entities.

Note: Enter ? to list the plot styles available in the drawing.

Plot with lineweights?
Toggles the use of lineweights.

Note: This option is unavailable when Plot with Plot Styles is turned on.

Yes
Mimics lineweights assigned to the drawing in the print.

Enter shade plot
Specifies the shade plot. Overrides the visual style of the current view when printing model space.

Note: 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.

Write plot to a file?
Determines whether the plot is sent to a file. If Yes, displays the Create plot file dialog box.

Note: The plot is saved in a .plt file.

Save changes to layout?
Toggles saving the plot parameters to the model or layout tab.

Proceed with plot?
Toggles plotting of the drawing.

Enter a page setup name
Specifies the name of a page setup.

21.36 PLOTSTAMP command

Opens the Plot stamp dialog box.

Shape Lite Pro Mechanical BIM

21.36.1 Description

Opens the Plot stamp dialog box to add information about drawings to the edge of the plotted drawing.

21.37 PLOTSTYLE command

Sets the current plot style.

Shape Lite Pro Mechanical BIM

21.37.1 Description

The command opens the Current Plot Style dialog box where you can set the current plot style.

Note: This command operates only when plot styles are enabled in drawings.

21.38 PLOTTERMANAGER command

Create and edit plotter settings.

Shape Lite Pro Mechanical BIM

21.38.1 Description

Opens a generic system dialog box PlotConfig where you can choose a plotter configuration file or where you can create and edit customized parameter PC3 files for printers and other output devices.

21.39 PNGOUT command

Opens the Save PNG dialog box.

Shape Lite Pro Mechanical BIM

21.39.1 Description

Opens the Save PNG dialog box to save data from the current drawing to a png file. The current view specifies what data to include in the image.

21.40 POINT command

Creates a point.

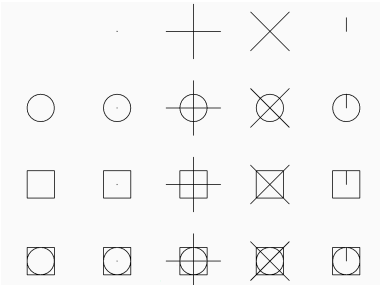
Shape Lite Pro Mechanical BIM

Icon:

Alias: PO

21.40.1 Description

Creates one or multiple points, controlling their size and style.



21.40.2 Option to create a point

Create a single point, represented by a dot, by specifying the location of point.

Location of point
Specify the location of the point.
Alternate options: [point Settings.../Multiple points]

21.40.3 Additional options within the POINT command

Before you begin creating a point, the following options may be available.

Point settings
Access the Settings dialog box to specify the display mode and size for points.

Multiple points
Draw multiple points until you press enter to end the command.

21.41 POINTCLOUD command

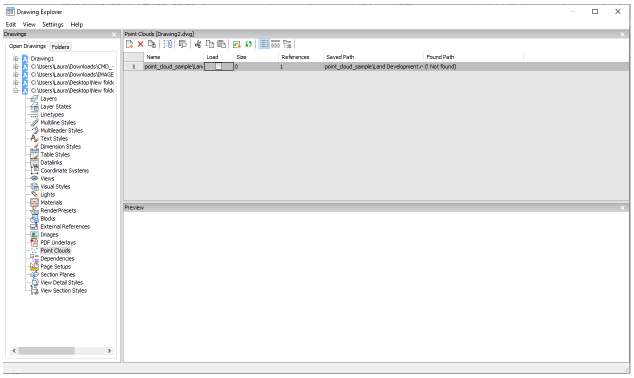
Manages Point Cloud files.

Shape Lite Pro Mechanical BIM

Icon:

21.41.1 Description

Displays the Point Clouds category of the Drawing explorer:



21.41.2 Option within the command

Name
Reports the name of the Point Cloud file insert.

Load
Toggles loading of the Point Cloud file.

- On: loads the Point Cloud, and makes it visible
- Off: unloads the Point Cloud, making it invisible

Size
Reports the size of the Point Cloud file. Very large files can slow down the system.

References
Reports the number of times the Point Cloud is attached to the drawing.

Saved Path
Reports the original path to the point cloud file when first loaded. When the path cannot be found, click the Browse button which will display the Choose a File dialog box and locate the missing point cloud file.

Found Path
Reports the current path to the point cloud file. This path should match the Saved Path in most cases.

21.41.3 Context menu options

New
Attaches a point cloud file to the current drawing, like the POINTCLOUDATTACH command.

Delete
Removes the selected point cloud from the current drawing.

Insert
Inserts additional copies of the existing point cloud files into the drawing; displays the Attach Point Cloud dialog box, like the POINTCLOUDATTACH command.

Select all
Selects all linetype definitions.

Invert selection
Deselects the current selection and vice versa.

Run extended search for missing attachments
Runs an extended search for missing attachments.

21.42 POINTCLOUDALIGN command

Rotates the point cloud to optimally align it with the X and Y axes.

Shape Lite Pro Mechanical BIM

Icon: 

21.42.1 Description

Rotates the point cloud to optimally align it with the X and Y axes in the world coordinate system. This enables you to efficiently create and edit entities relative to the point cloud using common tools such as orthogonal and polar tracking.

21.42.2 Options within the command

Select point cloud entity
Allows to select a point cloud entity if the current drawing has more than one point cloud attached.

> ***Note:*** If the current drawing has only one point cloud attached, it is automatically selected for the align operation.

First corner point
Specify the first corner of a rectangular selection window.

Specify opposite corner
Specify the opposite corner of a rectangular selection window.

21.43 POINTCLOUDATTACH command

Attaches point cloud files to the current drawing.

Shape Lite Pro Mechanical BIM

Icon:

21.43.1 Method

Select a file or a folder, specify Point cloud data name, give unit for point coordinate and then attach it from the Attach Point Cloud dialog box.

The point cloud file or folder is preprocessed in the background and a balloon message arising when is ready.

Note: The POINTCLOUDCACHEFOLDER system variable sets the folder where the point cloud data are stored.

21.43.2 Options within the command

File
Opens the Pre-process point cloud data file(s) dialog box from where you have to select a point cloud file to attach.

Folder
Opens the Choose input folder dialog box from where you have to select a folder to import.

21.44 -POINTCLOUDATTACH command

Attaches point cloud files to the current drawing.

Shape Lite Pro Mechanical BIM

21.44.1 Description

Attaches point cloud files (that were previously converted to BricsCAD's BPT format) to the current drawing through the Command line.

21.44.2 Options within the command

Specify point cloud data to attach
Specify point cloud link to attach to the current drawing.

Enter displacement vector
Enter displacement vector coordinates or specify on screen.

Rotation angle
Specify rotation angle.

Scale factor
Specify scale factor.

21.45 POINTCLOUDCOLORMAP command

Colorizes the point cloud.

Shape Lite Pro Mechanical BIM

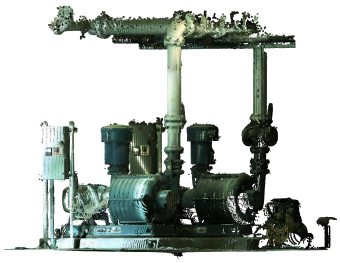
Icon:

21.45.1 Description

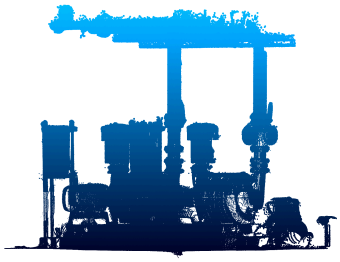
Colorizes the point cloud based on a range of colors.

21.45.2 Options within the command

Scan
Assigns color based on the color data collected by the scan.



Elevation
Assigns color based on the elevation of the points using a specified color map.



Spectrum
Assigns a range of colors based across the color spectrum.

Earth
Assigns a range of colors from light brown to dark brown.

Hydro
Assigns a range of colors from light blue to dark blue.

grAy
Assigns a range of colors from light gray to dark gray.

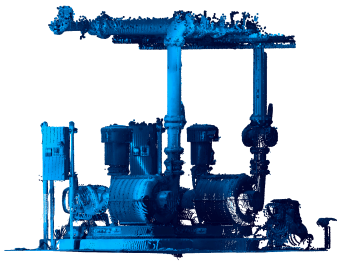
Reds
Assigns a range of colors from light red to dark red.

Greens
Assigns a range of colors from light green to dark green.

Blues
Assigns a range of colors from light blue to dark blue.

Intensity
Assigns colors according to their intensity value based on the specified color map.

Note: When the point cloud file contains no intensity data, then the colors are assigned by height, regardless of which option you select.

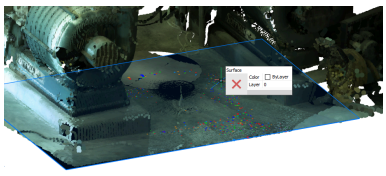


PlaneDeviation

The deviation colors points in the point cloud according to their distances to the selected planes.

Note: This option is used to verify how well points fit with given or fitted entities.

Note: The colors are influenced by the two tolerance levels: Plane Tolerance OK and Plane Tolerance Warning. They can be found in the Properties Panel . All points that are on a distance less than Plane Tolerance OK from the plane, are colored green. Points that are on a further distance than Plane Tolerance OK, but smaller than Plane Tolerance Warning, are colored orange (on one side of the plane) or light blue (on the other side of the plane). Points that are even further than Plane Tolerance Warning are colored red (on one side) or dark blue (on the other side).



Note: This option works with regions, surfaces and solids.

21.46 POINTCLOUDCOMPRESS command

Compresses a point cloud cache entry.

Shape Lite Pro Mechanical BIM

21.46.1 Method

Opens the **Please provide a point cloud cache** dialog box where you can select a point cloud cache folder. Inside the selected folder, data files at the relative path Data/*.pnt will be compressed. The extension of the data files will change from .pnt to .pnz.

The compression ratio is displayed on the Command line.

Note: If FILEDIA =0, you have to specify the location of the point cloud cache folder in the Command line.

Note: Manual compressing and decompressing a point cloud cache file may be needed in case of data exchange between BricsCAD® V21 and V22 versions. BricsCAD® V22 can work seamlessly with compressed or uncompressed data, but BricsCAD® V21 only works with uncompressed data.

Note: Set the POINTCLOUDCACHEFOLDER system variable as you prefer.

21.47 POINTCLOUDCROP command

Creates a crop boundary on a point cloud.

Shape Lite Pro Mechanical BIM

Icons: 

Creates a crop boundary on an attached point cloud to limit the displayed points to a specific area. If the current drawing has only one point cloud attached, it is automatically selected for the crop operation. If the current drawing has more than one point cloud attached, you are prompted so select which point cloud to crop.

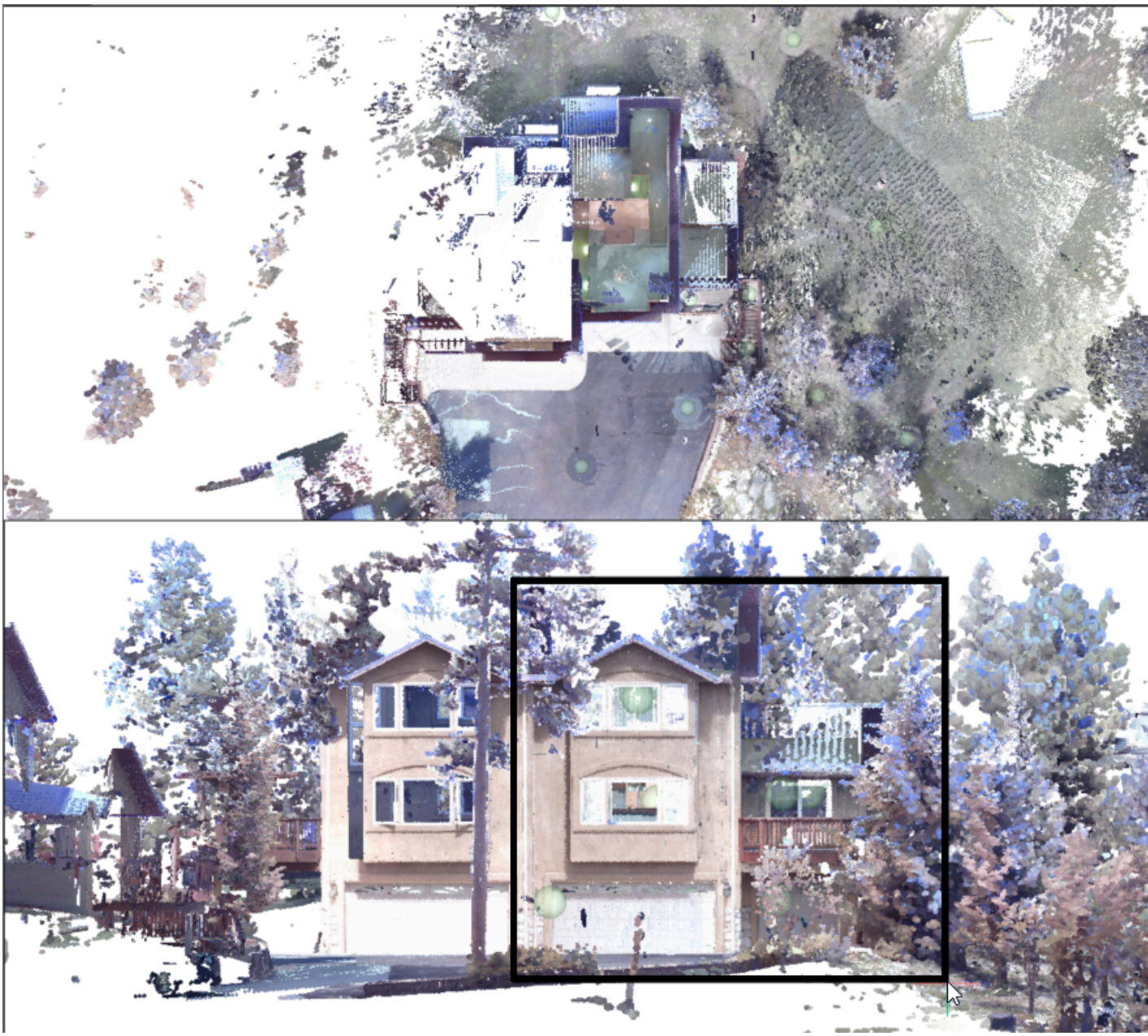
21.47.1 Method

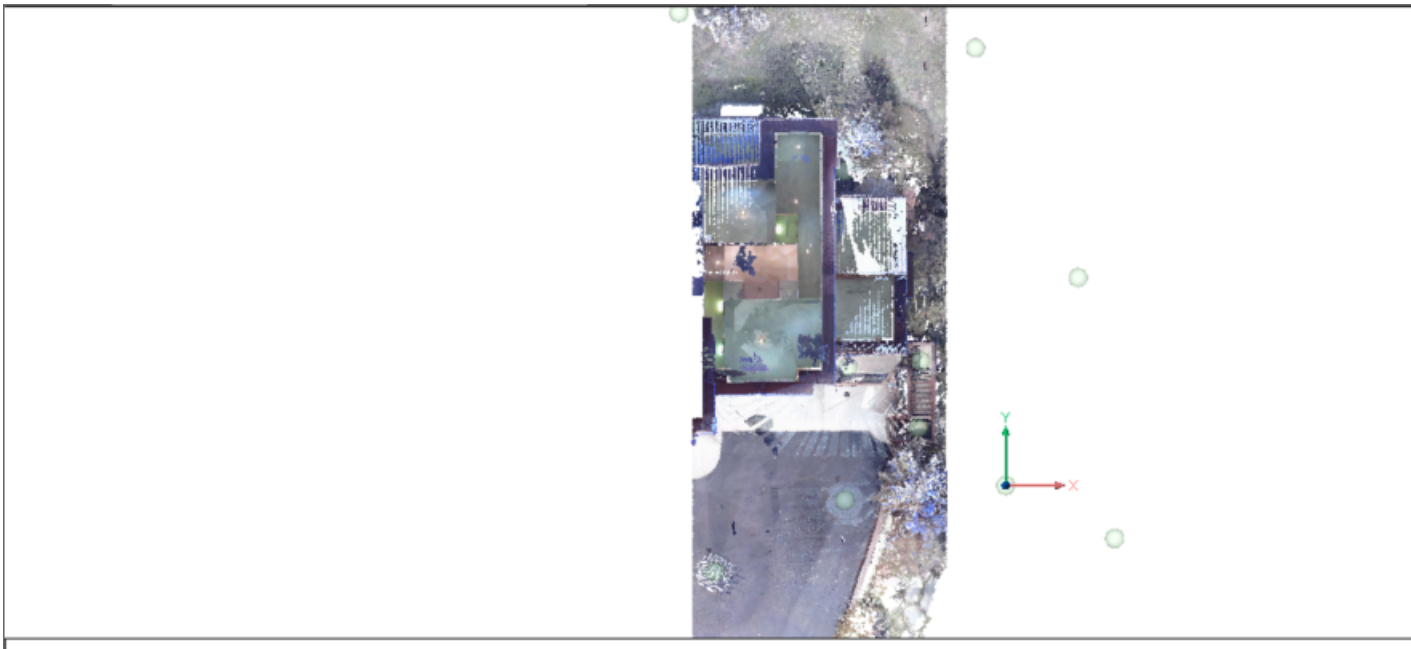
Crops a point cloud entity, displaying only the part of the point cloud inside or outside a prismatic or cylindrical volume.

21.47.2 Options within the command

Rectangular
Creates a rectangular prismatic cropping.

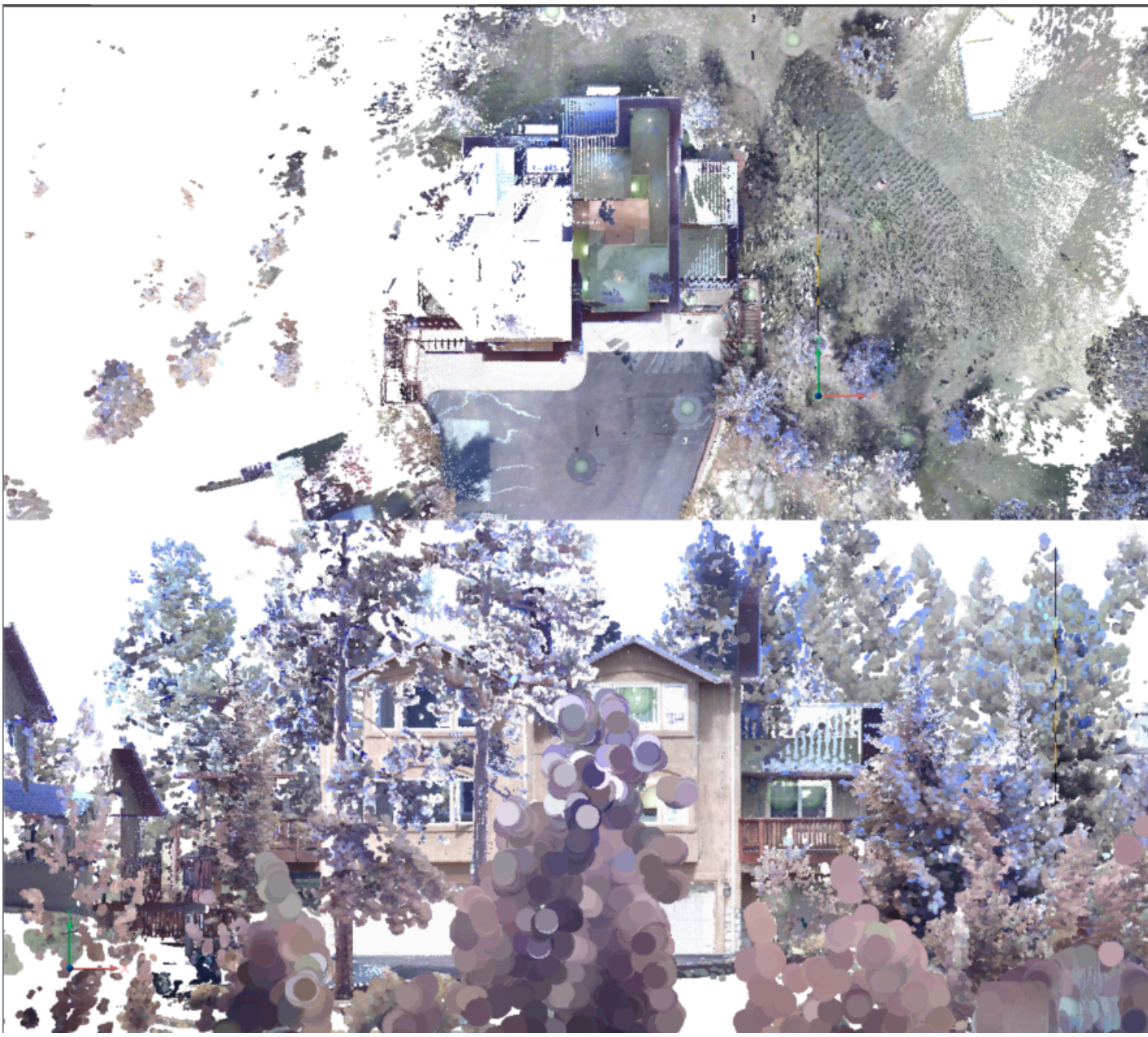
Note: The crop boundary creates a rectangular volume of points perpendicular to the view in which you defined it.

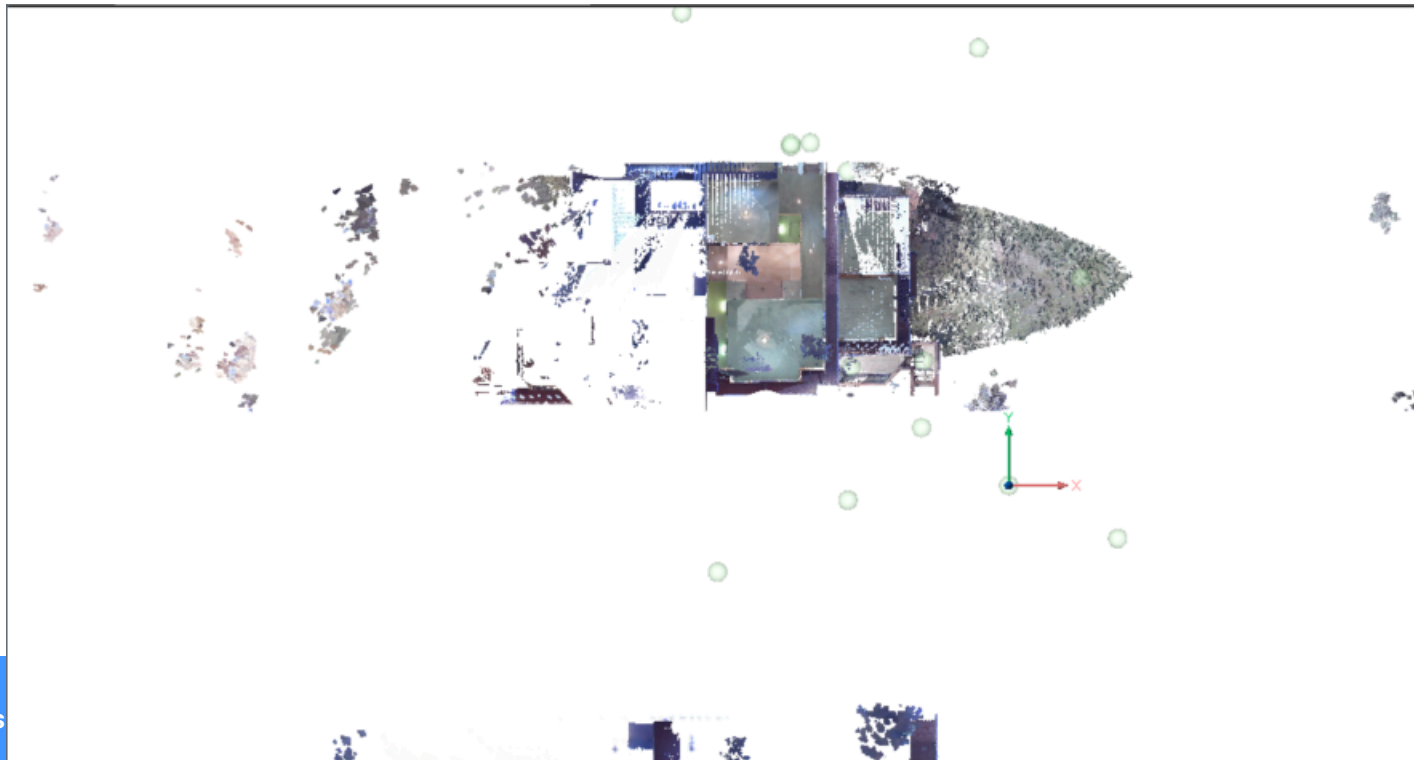




Circular
Creates a cylindrical cropping

Note: The crop boundary creates a circular volume of points perpendicular to the view in which you defined it.



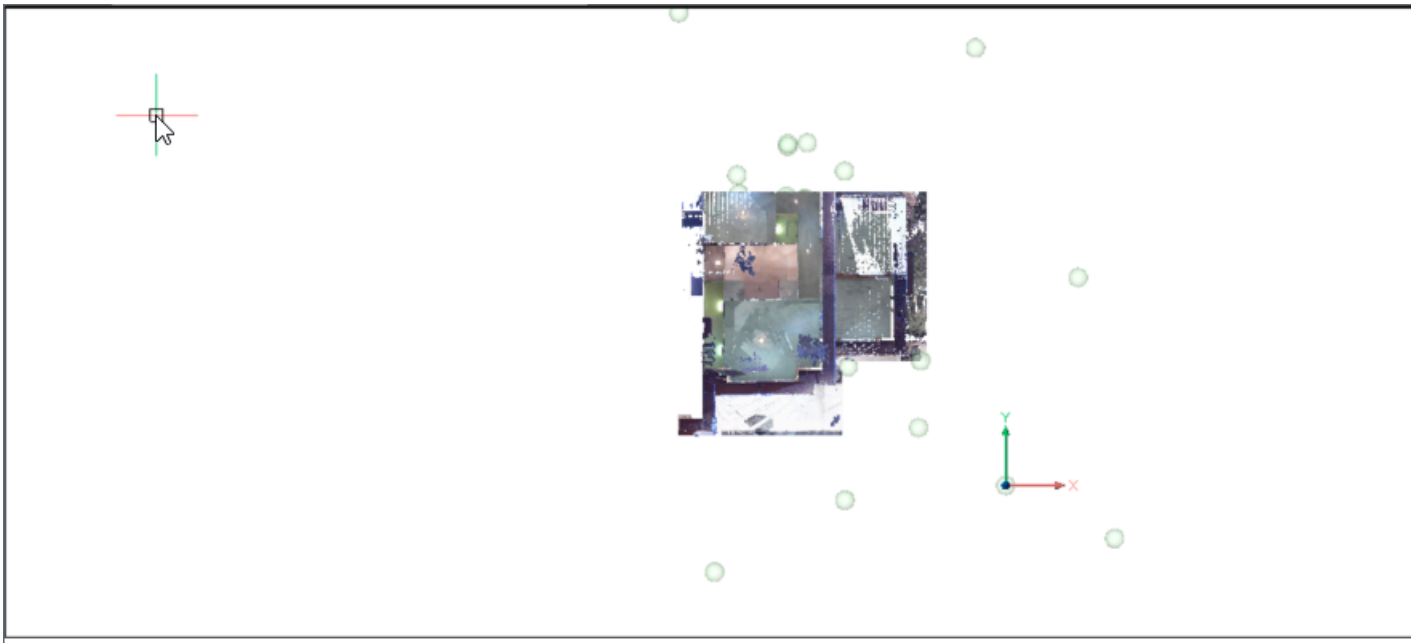


Polygonal
Creates a polygonal prismatic cropping.

Note: The crop boundary creates a polygonal volume of points perpendicular to the view in which you defined it.







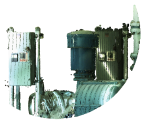


Invert
Inverts the current cropping. Toggles between Inside/Outside.

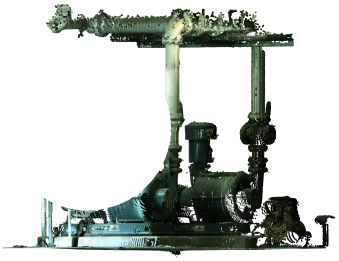




Inside
Displays the part of the point cloud inside the volume only.



Outside
Displays the part of the point cloud outside the volume only.



showToggle
Toggles the current cropping on/off.

removeLast
Removes the most recently added cropping volume.

removeAll
Removes all cropping.

21.48 POINTCLOUDCROPSOLID command

Converts an extruded solid/polysolid to a crop solid.

Shape Lite Pro Mechanical BIM

Icon:

Note: Crop solids can be used to crop point clouds in 3D and can be modified like any other BricsCAD solid.

21.48.1 Method

There are two methods to convert solids or polysolids to a crop solid:

- Inside
- Outside

21.48.2 Options within the command

Inside
Converts the selected solids into inside crop solids.

Outside
Converts the selected solids into outside crop solids.

Note: You can switch the Inside/Outside cropping property of a crop solid from the Properties panel.

21.49 POINTCLOUDDECOMPRESS command

Decompress a point cloud cache entry.

21.49.1 Method

Opens the **Please provide a point cloud cache** dialog box where you can select a point cloud cache folder. Inside the selected folder, data files at the relative path Data/*.pnz will be decompressed. The extension of the data files will change from .pnz to .pnt.

The compression ratio is displayed on the Command line.

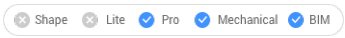
Note: If FILEDIA =0, you have to specify the location of the point cloud cache folder in the Command line.

Note: Manual compressing and decompressing a point cloud cache file may be needed in case of data exchange between BricsCAD® V21 and V22 versions. BricsCAD® V22 can work seamlessly with compressed or uncompressed data, but BricsCAD® V21 only works with uncompressed data.

Note: Set the POINTCLOUDCACHEFOLDER system variable as you prefer.

21.50 POINTCLOUDDELETEITEM command

Opens the Select Items To Delete From Cache dialog box.



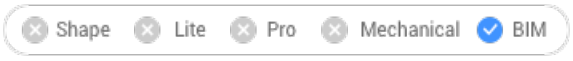
Icon: 

21.50.1 Description

Opens the Select Items To Delete From Cache dialog box to delete point cloud files from the cache.

21.51 POINTCLOUDDETECTFLOORS command

Generates volume sections.



Icon: 

21.51.1 Description

Generates volume sections for each floor found in a point cloud representing a building.

21.51.2 Options within the command

Full
Automatically detects floors for the entire Point Cloud.

Selection
Specifies the Point Cloud area where to detect floors.

21.52 POINTCLOUDDEVIATION command

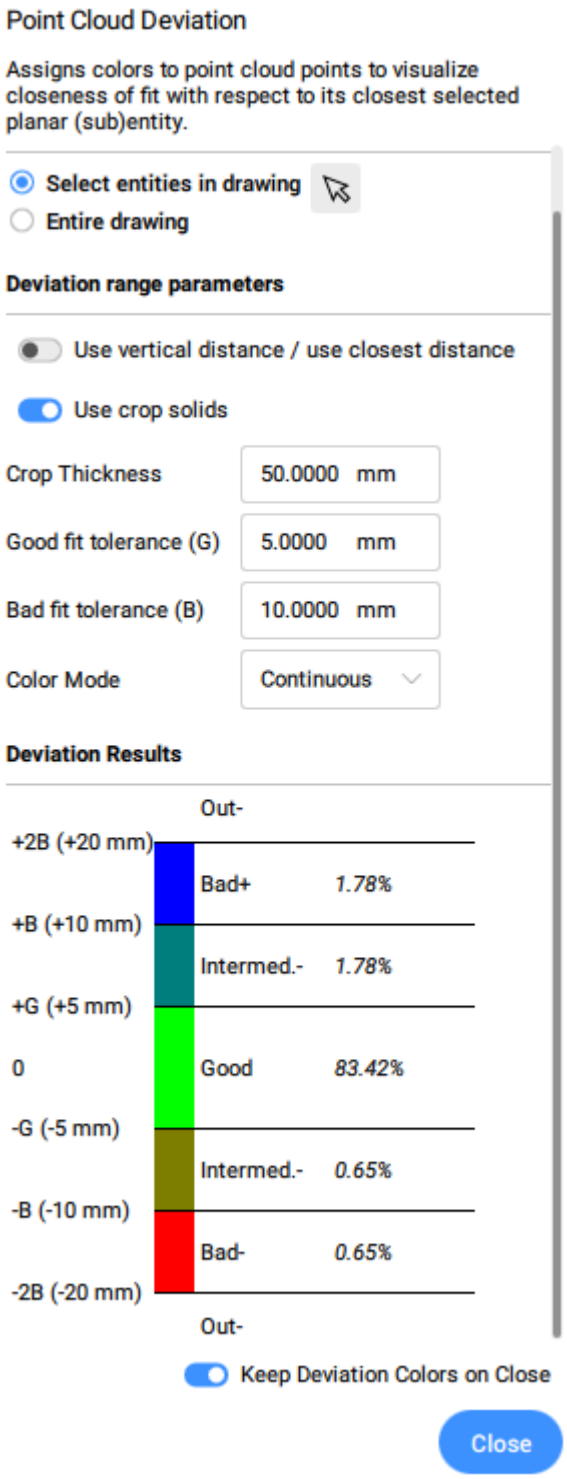
Assigns colors to point cloud points to visualize closeness of fit with respect to its closest selected planar (sub)entity. The selected entities are isolated using crops while the command panel is open.

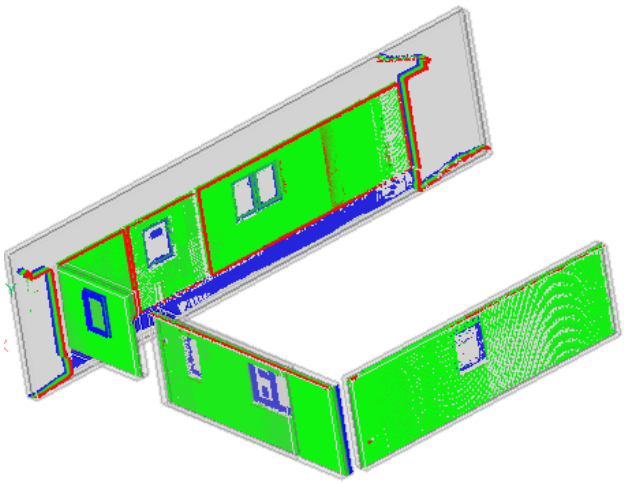
Shape Lite Pro Mechanical BIM

Icon:

21.52.1 Method

After selecting the planar surface or the solid from point cloud entities, the Point Cloud Deviation command panel is opened.


Point Cloud Deviation
Assigns colors to point cloud points to visualize closeness of fit with respect to its closest selected planar (sub)entity.
Select entities in drawing
Entire drawing
Deviation range parameters
Use vertical distance / use closest distance
Use crop solids
Crop Thickness
50.0000 mm
Good fit tolerance (G)
5.0000 mm
Bad fit tolerance (B)
10.0000 mm
Color Mode
Continuous
Deviation Results
Out-
+2B (+20 mm)
Bad+ 1.78%
+B (+10 mm)
Intermed.- 1.78%
+G (+5 mm)
0
Good 83.42%
-G (-5 mm)
Intermed.- 0.65%
-B (-10 mm)
Bad- 0.65%
-2B (-20 mm)
Out-
Keep Deviation Colors on Close
Close



21.52.2 Options within the command and panel

Entities

Select entities in the drawing
Selects specific geometric entities in the drawing.

Entire drawing
Selects all geometric entities in the drawing.

Deviation range parameters

Note: Parameters previously set by the user are remembered.

Use vertical distance / use closest distance
Vertical distance only takes into account the difference in z-coordinate (this is relevant in civil).
Closest distance just takes into account the closest distance to the nearest planar entity, perpendicular to that entity.

Use crop solids
Toggles the crop of the solids.

Crop Thickness
Defines a thickness for a solid that is used around the geometry to compare the position of the point cloud points.

change Tolerances

Good fit tolerance (G)
Sets the good fit (OK) tolerance value.

Bad fit tolerance (B):
Sets the bad fit (warning) tolerance value.

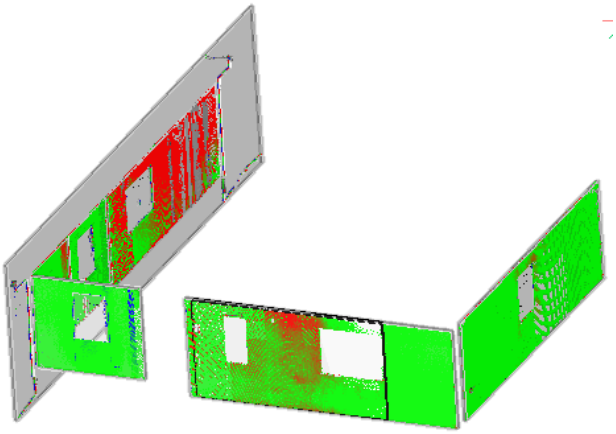
Colormode

Continuous
Spreads colors continuously

Discrete
Spreads colors discontinuously.

change Input selection
Defines a new geometry entities selection.

Deviation Results
Displays the point cloud distribution in percentage and colors.



Keep Deviation Colors on Close
Toggles the visualisation of the point cloud distribution in colors after closing the panel.

21.53 POINTCLOUDEXPORT command

Exports a cropped selection of a point cloud to a .pts, .Hspc or .laz file.

Shape Lite Pro Mechanical BIM

Icon:

21.53.1 Description

Saves data from the current drawing to the specified file format [Pts/Hspc/Laz].

Note: The HSPC file format is a proprietary format developed by Hexagon VCH (Visual Computing Hub).

21.54 POINTCLOUDFITCYLINDER command

Creates cylinders in Point Clouds.

Shape Lite Pro Mechanical BIM

Icon:

21.54.1 Description

This command can only operate in **Bubble Viewer**.

21.54.2 Method

Open **Bubble Viewer** and choose two points along the axis to create a cylinder.

After selecting the second point, the cylinder is inserted in model space. Press **Ctrl+A** keys to sync the drawing view to match the **Bubble Viewer** and see the new inserted cylinder.

21.54.3 Options within the command

Accept
Accepts the new inserted cylinder.

Decline
Does not accept the new inserted cylinder.

21.55 POINTCLOUDFITPLANAR command

Creates a planar surface or a solid from a point cloud point.

Shape Lite Pro Mechanical BIM

Icon: 

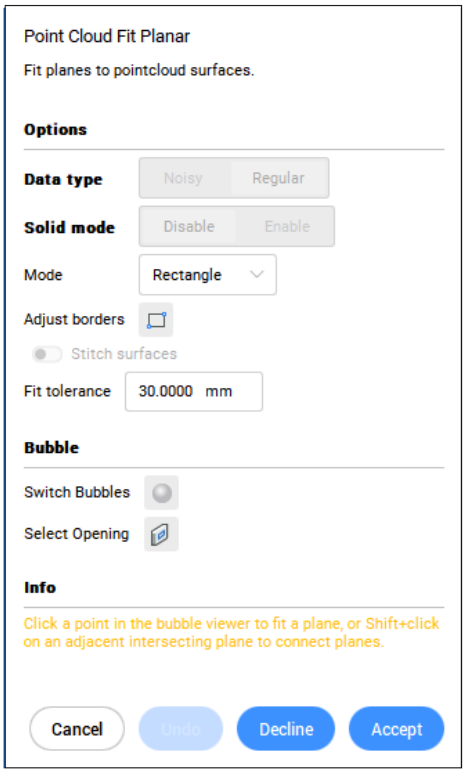
21.55.1 Method

Searches a point cloud for a dense set of planar points around selected seed point(s) and creates a planar surface or a solid if possible.

Note: Enables to create multiple surfaces or solids until you cancel.

Note: You can execute the command in model view or in a bubble view. Double click the bubble in which you want to start the command before you launch the command.

This command opens the **Point Cloud Fit Planar** command panel.



21.55.2 Options within the command

Select point cloud entity
Specifies point cloud entity.

Data type

- **Noisy**: robust detection for coarse/noisy data - requires selecting 3 seed points.
- **Regular**: normal data - requires selecting 1 seed point.

Solid mode
Searches for a parallel plain in the point cloud to create a solid. For example a wall or a slab.

- **Disable**: disables the computation of solid mode data in order to save time.
- **Enable**: enables the computation of solid mode data.

Note: Solid mode is available in Model view only, if a dense area is detected in a plane that is parallel to the detected surface.

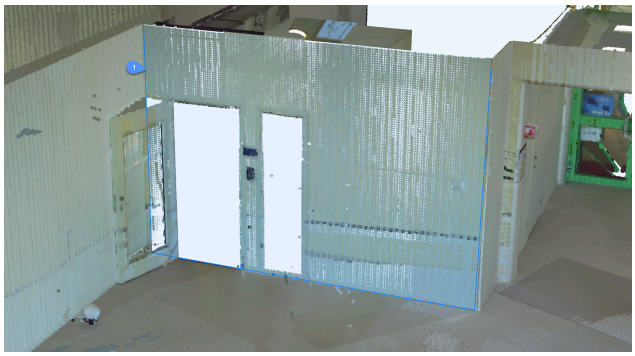
Select seed point in model view
Specifies the seed point for creating a planar surface or solid.

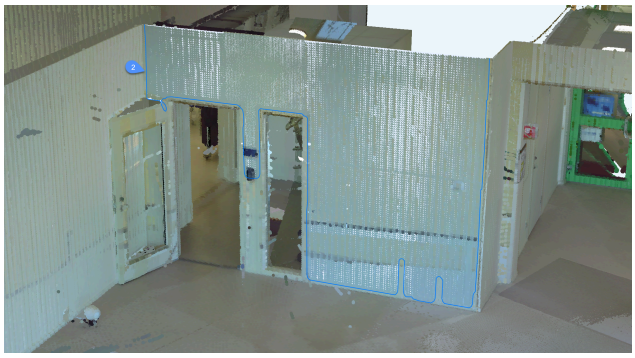
Mode
Switches between:

- **rectangle** (1): creates a single plane around the chosen point, rectangle outer border;
- **contour** (2): creates a single plane around the chosen point, contoured outer border;

- **solid** (3): creates a cuboid solid with a face that contains the specified seed point and an automatically detected second plane nearby.

Note: Press Ctrl to switch between the available modes.







adJust borders
Defines a new border for the selected plane by specifying the vertices (Select vertices on border).

Stitch surfaces
Stitches adjacent surfaces selected in the rapid selection mode.

Note: Rapid selection is activated by holding down the Shift button.

Fit tolerances
Set a tolerance value for the plane fit.

Accept
Accept the current planar surface or solid.

Decline
Decline the current planar surface or solid.

Undo
Removes the most recently added surface from the current selection set. Use **Decline** to remove the complete set.

Bubble



Note: Use Shift + click for rapid selection of adjacent surfaces . If **Stitch surfaces** is on, these surfaces will be stitched together.

Switch Bubbles
Switches between bubble viewers. Selects the bubble to switch to. Only available in bubble mode.

Select Opening
Creates an opening by selecting a point on a opening in a surface. Only available in bubble mode.

21.56 POINTCLOUDGEOGRAPHICLOCATION command

Sets the geographic location based on point cloud.

Shape Lite Pro Mechanical BIM

Sets the geographic location based on point cloud; sets point cloud transform based on geographic location; relatively aligns two point clouds.

Note: Only LAS and LAZ file formats can make use of this functionality.

21.56.1 Method

There are three methods:

- Geolocation
- Transform
- Relative

21.56.2 Options within the command

Geolocation
Allows to set a geographic marker in the DWG based on a point cloud with geographic location.

Transform
Updates the point cloud transform based on a geographic location in the DWG, as well as the geographic information in the point cloud.

Relative
Relatively aligns two point clouds if both contain a geographic location.

21.57 POINTCLOUDIMPORT command

Opens the Please provide a point cloud cache dialog box.

Shape Lite Pro Mechanical BIM

21.57.1 Description

Opens the Please provide a point cloud cache dialog box to select a point cloud cache folder.

21.58 POINTCLOUDPOINTSIZE_MINUS command

Decreases the display size of points in a point cloud.

Shape Lite Pro Mechanical BIM

21.58.1 Description

Decreases the display size of points in a point cloud to see the individual scan points more clearly. This decreases the POINTCLOUDPOINTSIZE system variable by one.

This command displays no prompts and has no options.

21.59 POINTCLOUDPOINTSIZE_PLUS command

Increases the display size of points in a point cloud.

Shape Lite Pro Mechanical BIM

21.59.1 Description

Increases the display size of points in a point cloud to visually fill gaps between individual scan points. This increases the POINTCLOUDPOINTSIZE system variable by one.

This command displays no prompts and has no options.

21.60 POINTCLOUDPREPROCESS command

Attaches point cloud files to the current drawing.

Shape Lite Pro Mechanical BIM

Note: Since BricsCAD V20, this command has been replaced by the POINTCLOUDATTACH command.

21.61 -POINTCLOUDPREPROCESS command

Attaches point cloud files to the current drawing.

Shape Lite Pro Mechanical BIM

Note: Since BricsCAD V20, this command has been replaced by the -POINTCLOUDATTACH command.

21.62 POINTCLOUDPROJECTSECTION command

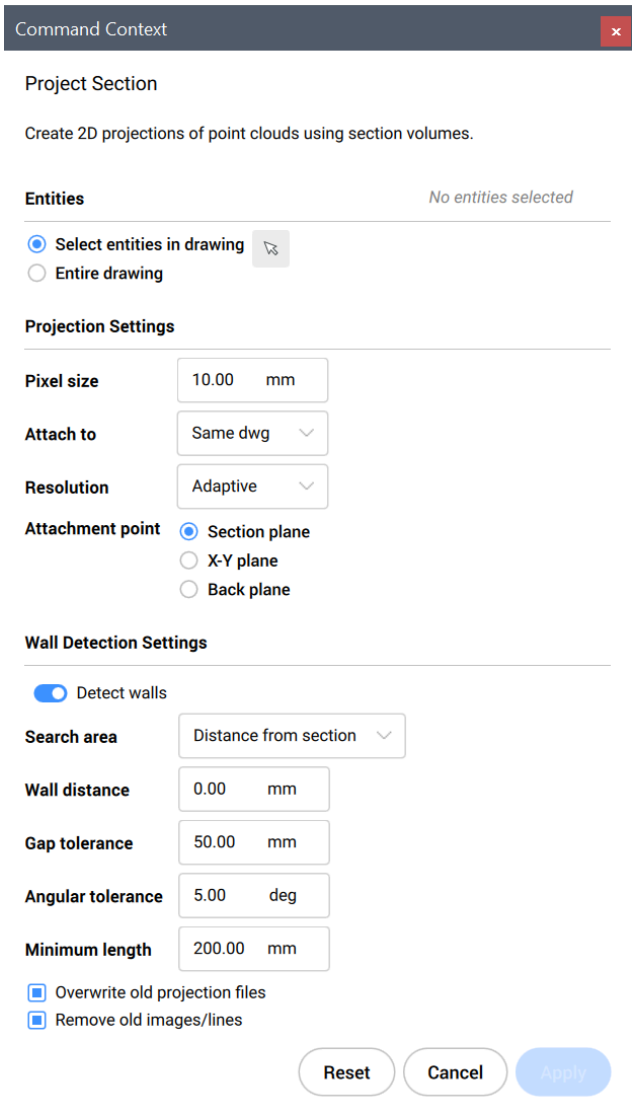
Automatically generates a 2D raster image with optional contour lines from a defined section box.

Shape Lite Pro Mechanical BIM

Icon:

21.62.1 Description

Opens the Command Context panel.



Note: The options within the POINTCLOUDPROJECTSECTION command are identical to the ones in the **Command Context** panel.

21.62.2 Options within the command

change Selection
Allows you to select section volumes to project.

change Pixel size
Controls the pixel size of the generated image.

Attach to
Determines where the resulting image is generated. Allows you to switch between **Same.dwg** and **New destination**.

Same dwg
The projection is made in the same file.

Destination file
Creates a new .dwg file to make the projection.

Note: You can overwrite a previous file or create a new one.

Resolution
Determines the resolution of the projection. Allows you to switch between **Finest** and **Adaptive**.

Finest
Produces a sharp and detailed background picture. It uses all the points in the point cloud.

Note: The projection in the finest resolution would take longer to complete.

Adaptive
Produces a less detailed and softer background image.

aTtachment point
Determines the insertion point for the generated image. Allows you to switch between **Section plane**, **X-y plane** and **Back plane**.

Section plane
Attaches the projection to the section plane.

X-y plane
Attaches the projection flat to the X-Y plane.

Back plane
Attaches the projection to the back plane of the section volume.

detect Walls
Allows you to switch between **Yes** and **No**.

Yes
Lines representing walls will be generated with the raster image.

No
Only a raster image is created.

toLerances
Allows you to modify the parameters that control 2D line drawing.

Search area
Allows you to switch between **Across volume** and **Distance from section**.

Across volume
Detects all walls in the point clouds. This option is often used to generate floor plans.

Distance from section
Detects walls in a given radius from a plane. This option is often used to generate vertical sections.

Wall distance
Controls the search area for wall detection.

Gap
Controls the search area for the algorithm to draw lines. Gaps smaller than the specified value will be closed.

Angular
Controls the angular tolerance of the generated lines. A high tolerance will result in pronounced right angles. This is useful for perpendicular plans.
Generated lines with a smaller angle than the specified value will be merged.

Minimum length
Controls the minimum length of a generated line. Lines with a smaller length than the specified value will not be generated.

Overwrite
Allows you to switch between **Yes** and **No**.
If applicable, it overwrites any old image files on disk related to this section volume.

reMove old
Allows you to switch between **Yes** and **No**.
If applicable, it removes previously added images or lines to this section volume.

21.63 POINTCLOUDREFERENCE command

Opens the Point Cloud Reference Manager dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

21.63.1 Method

If you press **Insert**, the Attach Point Cloud dialog box is opened where you can attach point cloud files from the cache to the current drawing.

21.64 POINTCLOUDSHOWBUBBLES command

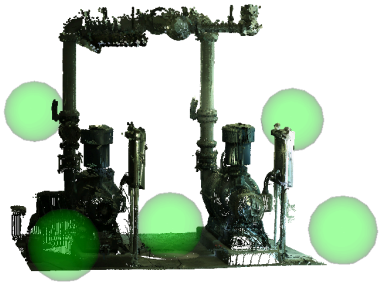
Toggles the display of point cloud bubbles.

Shape Lite Pro Mechanical BIM

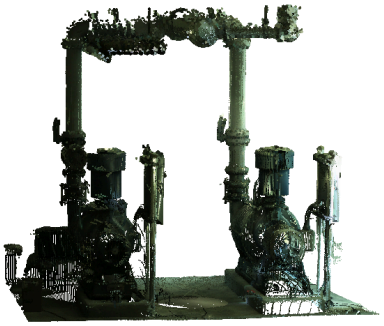
Note: You can change the bubbles size in the Properties panel.

21.64.1 Options within the command

Yes
Displays the bubbles.



No
Does not display the bubbles.



21.65 POINTCLOUDUNCROP command

Shape Lite Pro Mechanical BIM

21.66 POINTLIGHT command

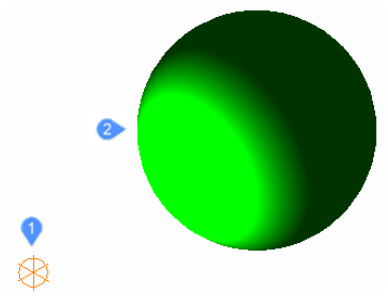
Places point lights.

Shape Lite Pro Mechanical BIM

Icon:

21.66.1 Description

Places point lights for use with renderings. A point light represents sources that illuminate in all directions, such as a bare lightbulb, and it so has no target.

	1 Glyph for a point light illuminating in all directions. 2 Sphere illuminated by the point light.

Note: Unlike all other light types, point lights do not have a target or vector associated with them. Drawings can have more than one point light.

21.66.2 Options within the command

Name
Specifies a name for the light so that it can be identified by the LightList command.

Intensity factor
Specifies the intensity of the light.

Status
Toggles the use of this light.

Photometry
Specifies the photometric properties of the light.

Intensity
Sets the intensity expressed in candela.

Flux
Sets the luminous flux expressed in lumens.

Illuminance
Sets the illuminance expressed in lux or foot-candles.

Distance
Sets a distance expressed in drawing units.

Color
Sets the color.

?
Lists the names of available colors.

Kelvin
Sets a color temperature expressed in degrees Kelvin.

shadoW
Specifies the look of shadows cast by this light.

Off
Disables the calculation of shadows for the light.

Sharp
Displays shadows with sharp edges. Use this option to increase performance.

soFt mapped
Displays realistic shadows with soft edges.

soft sAmpled
Displays realistic shadows with softer shadows based on extended light sources.

Shape
Sets the shape of the light.

Disk
Sets the radius of the disk.

Rect
Sets the length and width of the rectangle.

sAmples
Specifies the sampling size of the shadow.

Note: Larger numbers are more accurate but take longer to render.

Visible
Toggles the shape visibility.

Attenuation
Specifies how the illumination falls off with distance from the light.

attenuation Type
Specifies the type of attenuation.

None
No attenuation so that the distance to the light source has no influence.

Inverse linear
Attenuation is the inverse of the linear distance from the light.

Note: At a distance of 2 units from the light source, light is half as strong. At a distance of 4 units, light is one quarter as strong.

inverse Squared
Attenuation is the inverse of the square of the distance from the light.

Note: At a distance of 2 units, light is one quarter as strong. At a distance of 4 units, light is one sixteenth as strong.

Use limits
Toggles whether the extent of the illumination is limited.

attenuation start Limit
Defines the point where the light starts to shine, measured from the center of the light.

attenuation End limit
Defines the point where the light stops to shine, measured from the center of the light.

filterColor
Specifies the color of the light.

Index color
Specifies an Index color.

Hsl
Specifies the color using three parameters Hue, Saturation and Lightness.

Colorbook
Specifies a bookcolor name.

21.67 POLYGON command

Creates a closed polyline in the shape of a polygon.

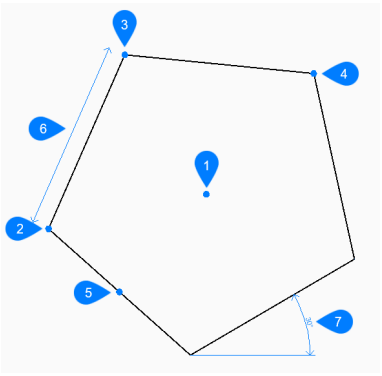
Shape Lite Pro Mechanical BIM

Icon:

Alias: POL

21.67.1 Description

Creates a closed polyline in the shape of an equal-sided polygon. Options allow you to specify the center, number of sides, edge length and angle, and the distance from the center to the vertex or midpoint of an edge.



1 Center of polygon

2 First endpoint edge

3 Second endpoint edge

4 Vertex

5 Midpoint of side

6 Edge length

7 Angle of polygon

21.67.2 Methods to begin a polygon

This command has 1 method to begin creating a polygon:

- Set number of sides

Set number of sides
Begin creating a polygon by specifying the number of sides between 3 and 1024 then:
Additional options: [Width of line/Multiple polygons]

Set center of polygon
Specify the center point of the polygon.
Additional option: [Specify by Edge]

Select midpoint of side
Specify the location of the midpoint of a polygon line segment. The location of the midpoint defines the size and angle of the polygon.
Additional option: [Specify by Vertex]

21.67.3 Additional options within the POLYGON command

After you begin creating a polygon, the following options may be available.

Width of line
Specify the width of the polygon line segments. All segments have the same width.

Multiple Polygons
Create multiple polygons with the same size and orientation when using the Set center of polygon option. Continue placing polygons until you press Enter to end the command.

Specify by Edge
Specify the endpoints of one edge of the polygon to define its size and angle.

Specify by Vertex
Specify the location of a vertex of the polygon. The location of the vertex defines the size and angle of the polygon.

21.68 POLYSOLID command

Createsa 3D solid in the shape of a wide, extruded polyline.

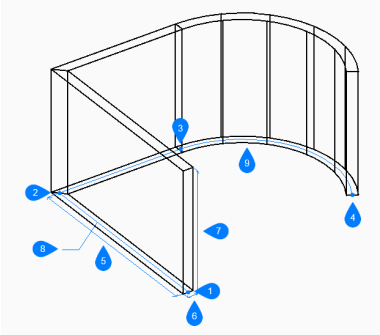
Shape Lite Pro Mechanical BIM

Icon: 

Alias: PSO

21.68.1 Description

Creates a 3D solid in the shape of a wide, extruded polyline with multiple line and arc segments. Options allow you to specify the width, height and justification.



1 Start

2 Next

3 Next

4 End

5 Length

6 Width

7 Height

8 Base path

9 Arc segment

21.68.2 Methodstocreateapolysolid

This command has 2methodsto begin creating a polysolid:

- Start point
- Entity

Start point

Begin creating a polysolid by specifying a start point for the base path then:
Additional options: [Height/Width/Justification/ Separate solids/Dynamic]

Set next point:

Specify the next vertex of the base path.
You can continue adding unlimited vertices until you press Enter to end the command.
Additional options: [draw Arcs/Close/Distance/Follow/Undo]

Height of polysolid

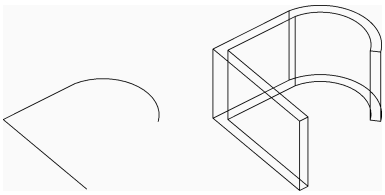
Specify the height of the polysolid.
The height is saved in the PSOLHEIGHT variable.

Entity

Begin creating a polysolid by selecting a 2D entity (line, open or closed polyline, arc, circle, ellipse, elliptical arc, or splines) as the base path then:

Height of polysolid

Specify the height of the polysolid.



The height is saved in the PSOLHEIGHT variable.

21.68.3 Additional options within the POLYSOLIDcommand

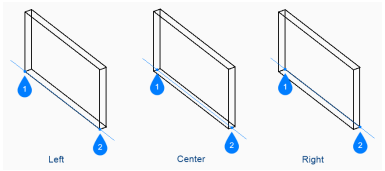
After you begin creating a polysolid, the following options may be available:

Height
Specify the height of the polysolid.
The height is saved in the PSOLHEIGHT system variable.

Width
Specify the width of the polysolid.
The width is saved in the PSOLWIDTH system variable.

Justification
Specify the position of the polysolid with respect to the base path.

- **Left**: the left edge of the polysolid follows the base path.
- **Right**: the right edge of the polysolid follows the base path.
- **Center**: the center of the polysolid follows the base path.



1 Start

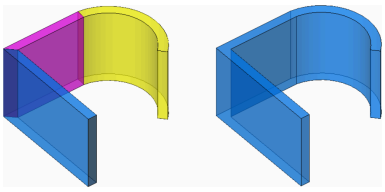
2 End

Hot Key Assistant
When Hot Key Assistant (HKA) is enabled, the following widget allows you to dynamically switch between left, center, and right justification by pressing the Ctrl key.



Separate solids
Specify whether polysolid creates an individual 3D solid for each segment or a single 3D solid for all the segments.

- **On**: create individual solids.
- **Off**: create a single solid.

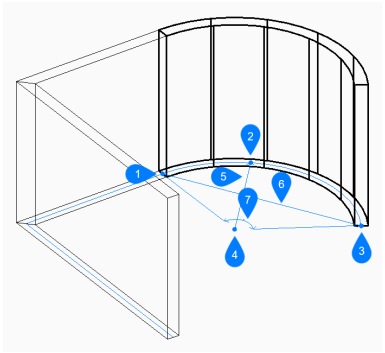


Dynamic

Specify whether you are prompted to specify the height when creating a polysolid.

- **On**: you are prompted to specify the height.
- **Off**: the height is defined by the PSOLHEIGHT system variable.

Draw arcs



1 First point

2 Second point

3 End of arc

4 Center

5 Radius

6 Length of chord

7 Angle

Specify the end point of a polysolid arc segment. The arc segment is drawn tangent to the previous segment.

Additional options: [draw Lines/CLose/Direction/Second point/Undo]Direction

Specify the direction of the polysolid arc segment then:

End of arc

Specify the next vertex of the polysolid.

Second point

Specify a point along the polysolid arc's circumference.

Draw lines

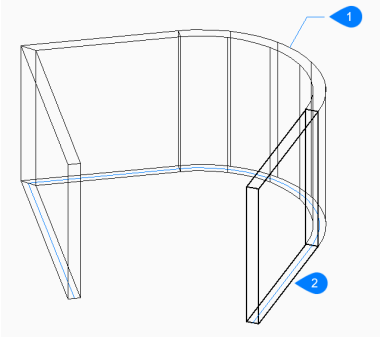
Specify the end point of a polysolid line segment. The line segment is drawn tangent to the previous segment.

Additional options: [draw Arcs/Close/Distance/Follow/Undo]

Distance

Specify the length and angle of a polysolid segment.

Follow
Begin creating a polysolid from the last drawn arc or line segment, following its angle.

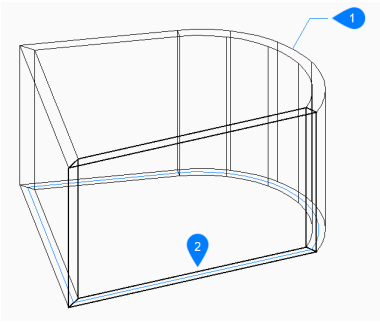


1 Last drawn segment

2 Follow

Undo
Undo the last polysolid segment and continue drawing from its previous start point.

Close
Automatically draw a polysolid segment from the end point of the last segment to the start point of the first segment. This ends the command.



1 Last drawn segment

2 Close

21.69 PREVIEW command

Displays the Preview window so that you can check how the drawing looks before plotting it.

Shape Lite Pro Mechanical BIM

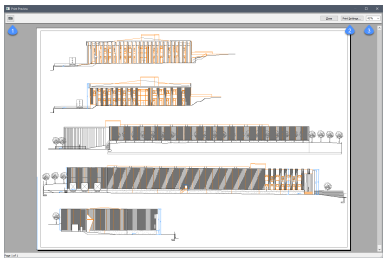
Icon:

Alias: PREVIEW, PRE

Note: This command does not work with the "None" printer; use the PRINT or PAGESETUP command to first specify a printer.

21.69.1 Description

Displays a window:



Use the mouse scroll wheel to zoom and use the scroll bars to pan.

1. Print
Prints the drawing; does not display the PRINT dialog box. See PRINT command.

2. Print Settings
Displays the PRINT dialog box. See PAGESETUP command.

3. Zoom
Changes the size of the preview.

21.70 PRINT command

Opens the Print dialog box.

Shape Lite Pro Mechanical BIM

Icon:

21.70.1 Description

Opens the Print dialog box to specify print options and preview or print the current drawing.

21.71 PROFILEMANAGER command

Opens the User profile manager dialog box.

Shape Lite Pro Mechanical BIM

21.71.1 Description

Opens the User profile manager dialog box to create and manage user profiles.

21.72 PROJECTGEOMETRY command

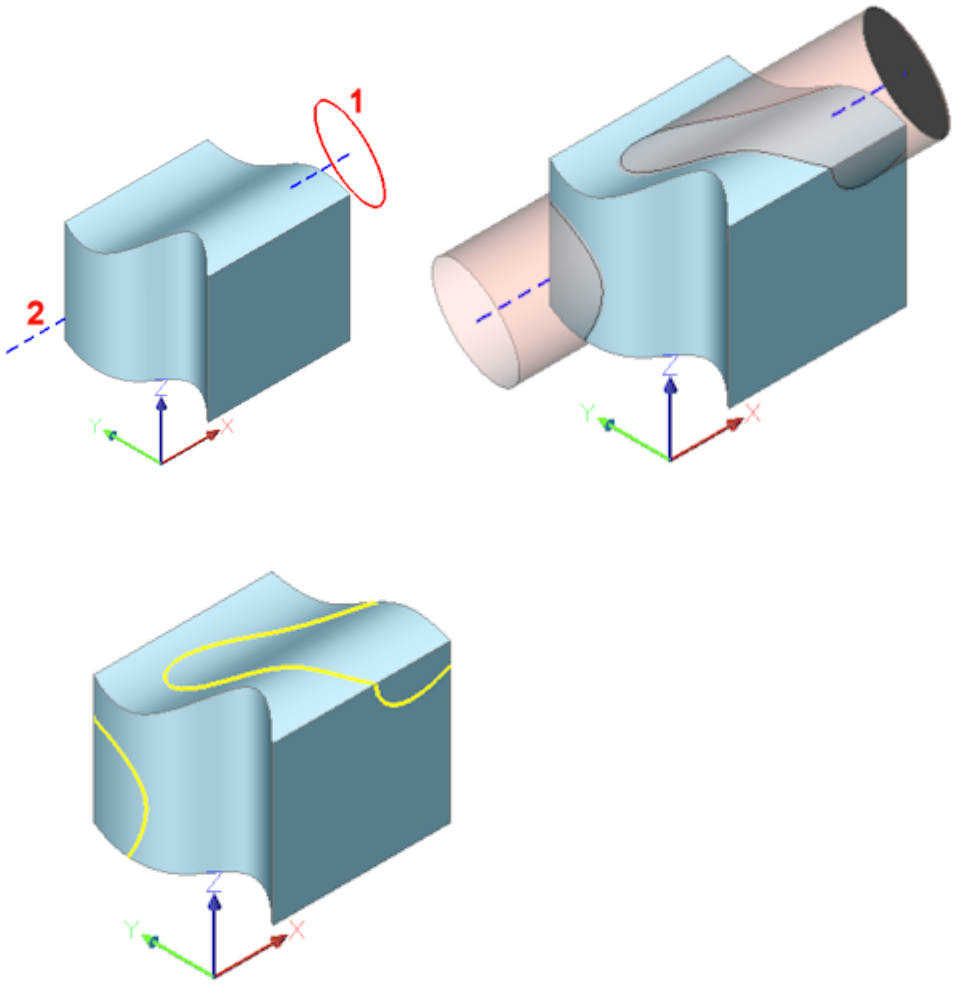
Projects 2D geometry.

Shape Lite Pro Mechanical BIM

Icon:

21.72.1 Description

Projects 2D geometry onto regions, surfaces, or 3D solids to create additional edges (line work).



(1) 2D entity; (2) 3D solid; blue line = projection direction; pink tube = projection; yellow curves = new edges

21.72.2 Options within the command

set PROjection direction
Defines the projection direction. By default, 2D geometry is projected orthogonally onto the receiving entity.

View
Sets the projection direction perpendicular to the current view orientation.

UCS
Uses the Z-axis of the current UCS.

Points
Specify the start and end point of the projection direction.

21.73 PROPERTIES command

Opens the Properties panel in Properties mode.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: CH, DDCHPROP, DDMODIFY, MO, PR, PROPS

21.73.1 Description

Opens the Properties panel to display it in the current workspace. The Properties panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Properties panel can be either floating, docked or stacked.

21.74 PROPERTIESCLOSE command

Closes the Properties panel.

Shape Lite Pro Mechanical BIM

Alias: PRC

21.74.1 Description

Closes the Properties panel to hide it from the current workspace. If the Properties panel is stacked when you close it, the Properties tab or icon is removed from the stack.

21.75 PROXYINFO command

Opens the Proxy information dialog box.

Shape Lite Pro Mechanical BIM

21.75.1 Description

Opens the Proxy information dialog box to view information about proxy entities and toggle their display in the current drawing.

21.76 PSETUPIN command

Opens the Select page setup from file dialog box.

Shape Lite Pro Mechanical BIM

21.76.1 Description

Opens the Select page setup from file dialog box to select a dwg, dwt, or dxf file from which to import page setups. After selecting the file and choosing **Open**, the Import page setups dialog box displays. It enables you to choose which page setups from the selected file you wish to import into the current drawing.

21.77 -PSETUPIN command

Imports page setup definitions.

Shape Lite Pro Mechanical BIM

21.77.1 Description

Imports page setup definitions from another drawing through the Command line.

Note: Page setups define how a drawing is printed and are used by the Print and Publish commands.

21.77.2 Method

Run the command to open the Select Page Setup From File dialog box.

21.78 PSPACE command

Switches from model space and paper space viewports.

Shape Lite Pro Mechanical BIM

Alias: PS

21.78.1 Description

Switches from model space to paper space viewport in a layout tab. This command has a shortcut menu that opens by right-clicking on **P:Layout** on the Status Bar.

Note: This command operates only when a layout tab is in paper space mode.

21.79 PUBLISH command

Opens the Publish dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

21.79.1 Description

Opens the Publish dialog box to batch print a specified set of drawings.

21.80 -PUBLISH command

Prints the content of a DSD file.

Shape Lite Pro Mechanical BIM

21.80.1 Description

Prints the content of a Drawing Set Description (*.dsd) file at the Command line.

Note: Create the .dsd file with the PUBLISH command. The .dsd file specifies the file name, layouts, page setups, plotter and printer names, orientation, plot scale, number of copies, optional plot stamp, and the order in which to plot the files.

21.80.2 Method

Select a .dsd file in the Select Sheet List dialog box.

Note: If the BACKGROUNDPLOT system variable is 2 or 3, the sheets are printed in the background.

21.81 PURGE command

Removes unused named entities from drawings.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: PU

21.81.1 Method

Specify which type of entities to purge.

21.81.2 Options within the command

BAtch all
Purges all unused named entities from the drawing without prompts.

purge All
Purges all unused named entities from the drawing following the prompts.

Yes
Purges each unused named entity after you confirm.

No
Purges all unused named entities without confirmation.

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 with no length.

Empty text entities
Removes empty text entities from the current drawing.

Orphaned data
Performs a drawing scan and removes unused DGN linestyles data.

21.82 -PURGE command

Removes unused named entities from drawings.

Shape Lite Pro Mechanical BIM

Alias: -PU

See the PURGE command.

21.83 PYRAMID command

Creates a 3D Solid in the shape of a pyramid.

Shape Lite Pro Mechanical BIM

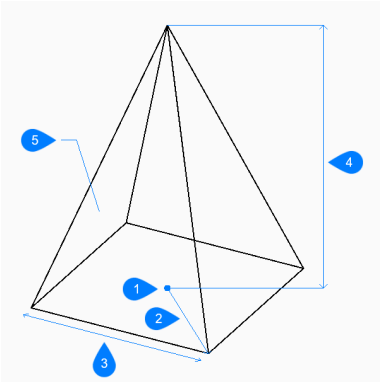
Icon: 

Alias: PYR

Note: In BricsCAD Lite, which doesn't support 3D Solids, the PYRAMID command launches the AI_PYRAMID command.

21.83.1 Description

Creates a 3D Solid in the shape of a pyramid with at least three sides. Choose from a combination of options including center, edge, radius, sides, height, and axis. The pyramid can have a pointed or flat top.



1 Center

2 Radius

3 Edge

4 Height

5 Side

21.83.2 Methods tocreatea pyramid

This command has 2 methods to begin creating a pyramid:

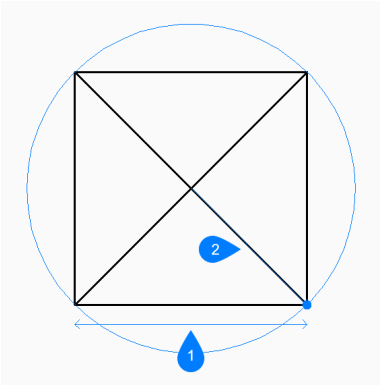
- Specify center point
- Edge

Specify center point

Begin creating a pyramid by specifying the center for the base of the pyramid then:
Alternate options: [Edge/Sides]

Base radius

Specify the distance from the center to a vertex as if the base is inscribed in a circle.



1 Center

2 Radius

Additional option: [Circumscribed]

Height
Specify the height of the pyramid.
Additional options: [2Point/Axis endpoint/Top radius]

Edge
Begin creating a pyramid by specifying the first endpoint for an edge of the base then:

Specify second point of edge
Specify the second point of the edge to define its length and angle in the xy-plane.

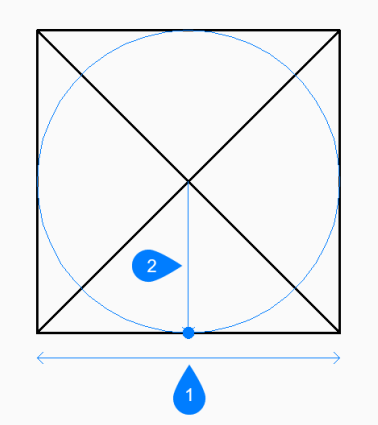
Specify height
Specify the height of the pyramid.
Additional options: [2Point/Axis endpoint/Top radius]

21.83.3 Optionswithin the PYRAMID command

After you begin creating a pyramid, the following options may be available:

Sides
Specify the number of sides or vertices for the base.

Circumscribed
Specify the distance from the center to a midpoint on an edge as if the base is circumscribed about a circle.



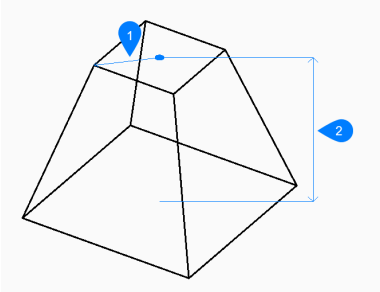
1 Edge

2 Radius

2Point
Specify the height of the pyramid as the distance between any two points.

Axis endpoint
Specify the axis endpoint to define the height and orientation of the pyramid in 3D space. The center of the base is used as the other axis endpoint.

Top radius
Specify the distance from the center of the top face to a vertex as if the top face is inscribed in a circle. Any radius larger than zero (0) creates a pyramid with a flat top.



1 Top radius

2 Height

22. Q

22.1 QLEADER command

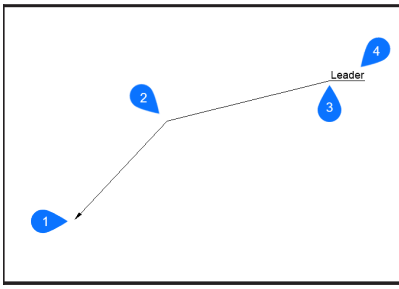
Draws leaders through a dialog box.

Shape Lite Pro Mechanical BIM

Icon:

22.1.1 Description

Draws leaders with a variety of annotation types and allows you to specify the properties through a dialog box.



1 Start point
2 Next point
3 To point
4 Annotation

22.1.2 Options within the command

Settings

The QLeader Settings dialog box pops up. Several settings can be specified there.

Text width

Specifies the text width for the text bounding box. Enter a value or 0 if no limit on the width needs to be set.

Note: When the length of the text exceeds the bounding box width, the text flows automatically to the next line.

22.2 QNEW command

Starts a new drawing based on the default template file.

Shape Lite Pro Mechanical BIM

Icon:

Alias: N

22.2.1 Description

Opens a new document tab based on the default template and user profile (short for "quick new").

22.3 QPRINT command

Prints the drawing using the default print configuration.

Shape Lite Pro Mechanical BIM

Icon:

22.3.1 Description

Allows printing a drawing without displaying the Print dialog box (short for "quick print").

If there is no printer added as a default printer a warning message will appear.

You will want to configure the drawing's print properties ahead of time with thePageSetup command.

22.4 QRTEXT command

Creates an MText and displays it as a QR code.

Shape Lite Pro Mechanical BIM

Icon:

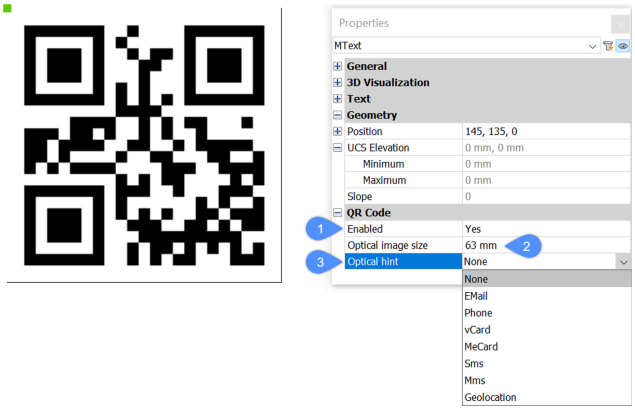
22.4.1 Description

Creates an MText which is represented as QR code by default.

Note:

- The QR code for a text is represented as a raster image.
- You can use the **Properties** panel to switch every text or mtext to QR code by setting the **Enabled** option to **Yes** (1), editing the **Optical image size** (2) and controlling the type of QR code by changing the **Optical hint** option (3).

  The **Optical hint** option allows you to specify 7 types of hints for QR code scanner: EMail, Phone, vCard, MeCard, Sms, Mms, Geolocation.



22.4.2 Options within the command

The Command line options are identical to the Command line options in the MTEXT command. See the MTEXT command article for details.

22.5 QSAVE command

Saves the drawing immediately.

Shape Lite Pro Mechanical BIM

Icon:

22.5.1 Description

Saves the drawing without displaying the Save dialog box (short for "quick save").

22.5.2 Method

If the drawing file has already been saved at least once, no dialog box pops up and the drawing is saved. To save the drawing by another name, use the Save As command.

When the drawing has never been saved before or when the drawing is opened in read-only mode, the Save Drawing As dialog box pops up.

22.6 QSELECT command

Opens the Properties panel in Quick select mode.

Shape Lite Pro Mechanical BIM

Icon:

22.6.1 Description

Opens the Properties panel in **Quick Select** mode to display it in the current workspace. The Properties panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Properties panel can be either floating, docked or stacked.

22.7 QRLEADER command

Creates a leader and displays its text as a QR code.

Shape Lite Pro Mechanical BIM

Icon:

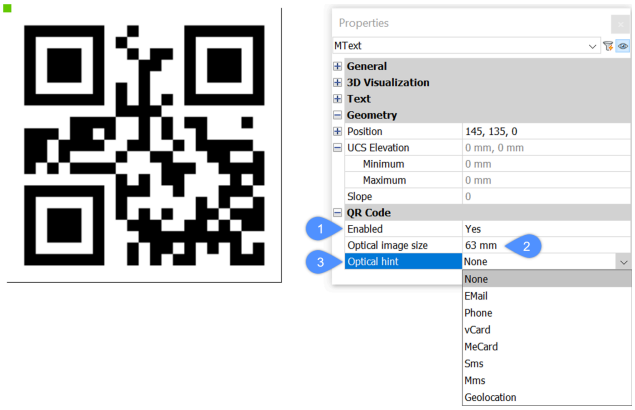
22.7.1 Description

Creates a leader with the text represented as QR code by default.

Note:

- The QR code for a text is represented as a raster image.
- You can use the **Properties** panel to switch every text or mtext to QR code by setting the **Enabled** option to **Yes** (1), editing the **Optical image size** (2) and controlling the type of QR code by changing the **Optical hint** option (3).

The **Optical hint** option allows you to specify 7 hints for QR code scanner: EMail, Phone, vCard, MeCard, Sms, Mms, Geolocation.



22.7.2 Options within the command

The Command line options are identical to the Command line options in the LEADER command. See the LEADER command article for details.

22.8 QTEXT command

Toggles the QTEXTMODE system variable.

Shape Lite Pro Mechanical BIM

Alias: QT

22.8.1 Description

Toggles the QTEXTMODE system variable to display text entities as rectangles or text. You must use REGEN or REGENALL to see the change.

- On: turns on the QTEXTMODE system variable.
- Off: turns off the QTEXTMODE system variable.
- Toggle: changes the QTEXTMODE system variable to the opposite of the current setting.

22.9 QUADRANT command

Toggles the Quadrant entity snap.

Shape Lite Pro Mechanical BIM

Icon:

22.9.1 Description

Toggles the Quadrant entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn

off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

22.10 QUIT command

Closes all open drawings and exits BricsCAD; offers the opportunity to save unsaved drawings.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: EXIT

22.10.1 Method

If all drawings have been saved before, the command exits BricsCAD without displaying a dialog box.

If there are unsaved files that are still open, you are asked if you want to save the drawings. For each unsaved drawing, a dialog box pops up and you can decide to save the drawing or not.

23. R

23.1 RAY command

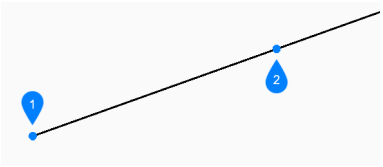
Creates rays.

Shape Lite Pro Mechanical BIM

Icon:

23.1.1 Description

Creates a ray (semi-infinite line) from a combination of options including point, direction and angle.



1 Start of ray

2 Direction

23.1.2 Methods to begin a ray

This command has 6 methods to begin creating a ray:

- Start of ray
- Horizontal
- Vertical
- Angle
- Bisect
- Parallel

You can continue adding unlimited rays until you press Enter to end the command.

Start of ray
Begin creating a ray by specifying the starting point of the ray then:

Direction
Specify the direction of the ray from the starting point.

Horizontal
Begin creating a ray horizontal to the x-axis then:

Location
Specify the starting point of the ray.

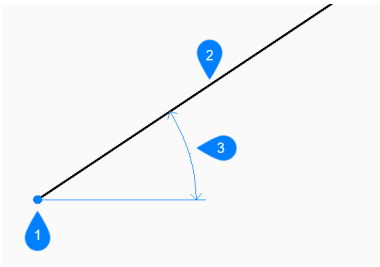
Vertical
Begin creating a ray parallel to the y-axis then:

Location
Specify the starting point of the ray.

Angle
Begin creating a ray based on an angle then:

Enter angle
Specify the angle of the ray.
Additional option: [Reference]

Location
Specify the starting point of the ray.



1 Location

2 Ray

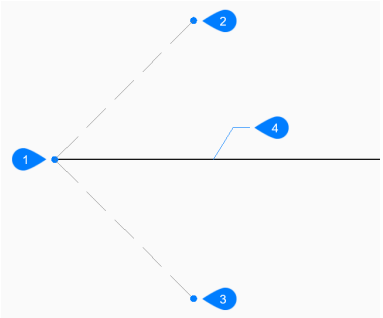
3 Angle

Bisect
Begin creating a ray which bisects the angle between two imaginary lines then:

Set vertex point
Specify the start point of the ray.
Additional option: [Entity]

Bisection angle start point
Specify a point to define the first imaginary line. The vertex is used as the other point.

Bisection angle end point
Specify a point to define the second imaginary line. The vertex is used as the other point.



1 Vertex point

2 Bisection angle start point

3 Bisection angle end point

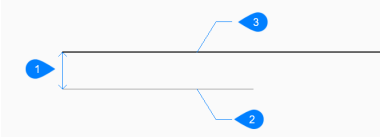
4 Ray

Parallel
Begin creating a ray parallel to a line or polyline segment then:

Set offset distance for parallel infinite ray
Specify the offset distance for the ray.
Additional option: [Through point]

Select entity for parallel infinite ray
Select a line or polyline segment from which to offset the ray.

Side for parallel infinite ray
Specify the side on which to place the ray. The ray's start point is parallel to the line's start point.



1 Offset distance

2 Line segment

3 Ray

23.1.3 Options within the RAY command

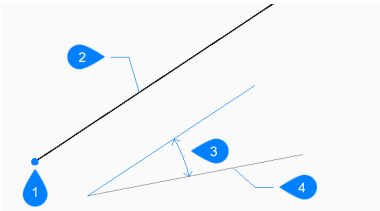
After you begin creating a ray, the following options may be available:

Reference
Select an entity to use a reference angle then:

Enter angle
Specify the angle to place the ray relative to the selected entity.

Location
Specify the starting point of the ray.

Location
The **Location** prompt repeats so that you can draw more angled rays with different starting points. Press Enter to end the command.



1 Location

2 Ray

3 Angle

4 Reference entity

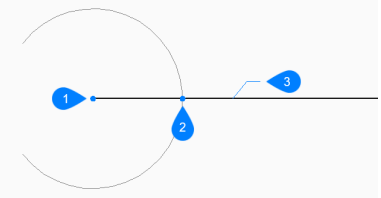
Entity
Select a line, arc, a polyline segment to bisect then:

Select side for ray bisection

Specify the side on which to place the ray.

When you select a line or polyline segment, the command draws the xline perpendicular to the segment's midpoint. When you select an arc or polyarc, the command draws the xline perpenduclar to the arc's center and midpoint.

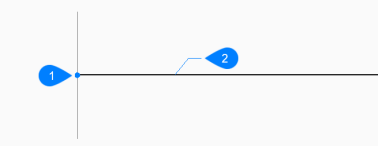
This command works with splined polylines, but not with splined entities.



1 Perpendicular to arc center point

2 Perpendicular to arc midpoint

3 Ray



1 Perpendicular to line midpoint

2 Ray

Through point

Specify a point through which to draw the ray.

Select entity for parallel infinite ray

Specify the entity to offset with the ray.

Through the point

Specify the point through which to draw the ray. This point is the ray's start point.



1 Through point

2 Line segment

3 Ray

23.2 REASSOCAPP command

Opens the Reassociate entity data dialog box.

Shape Lite Pro Mechanical BIM

Icon:

23.2.1 Description

Opens the Reassociate entity data dialog box to associate extended entity data with a specific application.

23.3 RECOVER command

Opens the Open drawing dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

23.3.1 Description

Opens the Open drawing dialog box to select a damaged dwg, dwt, or dxf file from which to recover data.

23.4 RECOVERALL command

Opens the Open drawing dialog box.

Shape Lite Pro Mechanical BIM

23.4.1 Description

Opens the Open drawing dialog box. to select a damaged dwg, dwt, or dxf file from which to recover data. In addition to the selected file, BricsCAD will attempt to recover data from all nested external references.

23.5 RECSCRIPT command

Opens the Record script dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

23.5.1 Description

Opens the Record script dialog box to save data from the current drawing to an scr file. After choosing Save in the dialog box, the commands and pick point you specify in the drawing editor are recorded in the scr file until you run the STOPSCRIPT command.

23.6 RECTANG command

Creates a polyline in the shape of a rectangle.

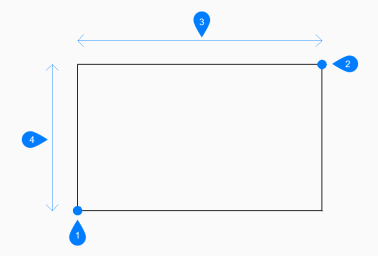
Shape Lite Pro Mechanical BIM

Icon: 

Alias: REC, RECT, RECTANGLE

23.6.1 Description

Creates a closed, four-sided rectangular polyline in the shape of a rectangle, square, and variants with chamfers or fillets.



1 First corner

2 Second corner

3 Length

4 Width

23.6.2 Methods to create a rectangle

This command has 5 methods to begin creating a rectangle:

- Select first corner of rectangle
- Rotated
- Square
- Area
- Dimensions

Additional options: [Chamfer/Fillet/Elevation/Thickness/Width of line]

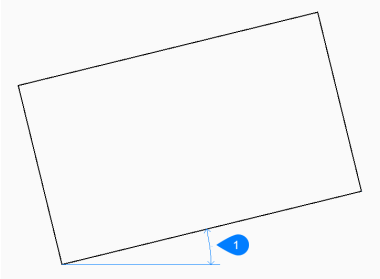
Select first corner of rectangle
Begin creating a rectangle by specifying a corner of a rectangle then:

Other corner of rectangle:
Specify the opposite corner of the rectangle. The rectangle is drawn parallel to the x and y axes.

Rotated
Begin creating a rectangle by specifying a corner of a rectangle then:
Additional options: [Chamfer/Fillet/Rotated/Square/Elevation/Thickness/Width of line/Area/Dimensions]:

Other corner of rectangle:
Specify the opposite corner of the rectangle.

Rotation angle for rectangle
Specify the rotation angle of the rectangle.



1 Rotation angle

Square
Begin creating a rectangle by specifying a corner of a square then:
Additional options [Chamfer/Fillet/Rotated/Elevation/Thickness/Width of line/Area/Dimensions]

Second corner of square
Specify a corner along the same side of the square to define its length and angle.

Area
Begin creating a rectangle by specifying its area then:

Calculate rectangle dimensions based on [Length/Width]
Choose whether to use the length or width to calculate the dimensions for the rectangle.

- **Length**: enter rectangle length
- **Width**: enter rectangle width

Dimension
Begin creating a rectangle by specifying its length then:

Width to use for rectangles
Specify the width of the rectangle.

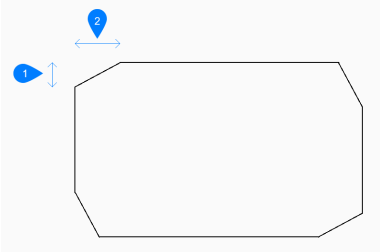
Select first corner of rectangle
Specify the first corner of the rectangle.
Additional options: [Chamfer/Fillet/Rotated/Square/Elevation/Thickness/Width of line/Area/Dimensions]:

23.6.3 Options within the RECTANG command

After you begin creating a rectangle, the following options may be available:

Chamfer
Specify the first chamfer distance to use for all rectangles then:
Additional options [turn chamfer Off, use Default setting]

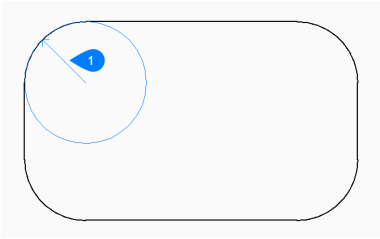
Second chamfer distance to use for all rectangles
Specify the second chamfer distance to use for all rectangles



1 First chamfer distance

2 Second chamfer distance

Fillet
Specify the fillet distance to use for all rectangles.
Additional options [turn fillet Off, use Default setting]



1 Fillet radius

Elevation
Specify the height above the xy-plane to use for all rectangles.
Additional options: [use Default]:

Width of line
Specify the width of the rectangle line segments. All segments have the same width.

Thickness
Specify the thickness of the rectangle line segments. All segments have the same thickness.

Turn chamfer off
Turn off chamfer.

Turn fillet off.
Turn off fillet.

Use default
Reset the default value.

23.7 REDEFINE command

Activates commands that were deactivated with the Undefine command.

Shape Lite Pro Mechanical BIM

23.7.1 Description

Reactivates commands that were deactivated with the Undefine command by entering the name of the undefined command.

23.8 REDO command

Reverses the action of the previous U or UNDO command.

Shape Lite Pro Mechanical BIM

Icon:

23.8.1 Description

Reverses the action of the previous U or UNDO command to restore entities to their state prior to the U or UNDO operation. The REDO command only works immediately following the U or UNDO commands.

This command displays no prompts and has no options.

23.9 REDRAW command

Redraws entities in the current viewport.

Shape Lite Pro Mechanical BIM

Icon:

Alias: R

23.9.1 Description

Redraws entities in the current viewport to remove graphic debris such as blip or drag marks.

This command displays no prompts and has no options.

23.10 REDRAWALL command

Redrawsentitiesin all viewports.

Shape Lite Pro Mechanical BIM

Icon:

Alias: RA

23.10.1 Description

Redrawentitiesin all viewports to remove graphic debris such as blip or drag marks.

This command displays no prompts and has no options.

23.11 REDSDKINFO command

Displays rendering related hardware and driver specifications in the command bar.

Shape Lite Pro Mechanical BIM

23.11.1 Description

Prints a report in the command window containing information about the graphics capabilities of your computer.

Press F2 to open the prompt history window.

23.12 REFCLOSE command

Closes the reference drawing editor.

Shape Lite Pro Mechanical BIM

Icons: 

23.12.1 Description

Closes the reference drawing editor, used to edit externally-referenced files in the current drawing session.

Note: This command can only be used after the REFEDIT command has commenced.

23.12.2 Methods

There are 2 methods to close the reference drawing editor:

- Save
- Discard

23.12.3 Options

Save
Saves the Reference drawing including the changes.

Discard
Saves the Reference drawing without saving the changes.

23.13 REFEDIT command

Edits block references and externally referenced drawings.

Shape Lite Pro Mechanical BIM

Icon: 

23.13.1 Description

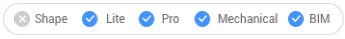
Edits the selected block reference or externally referenced drawing. The **Reference Edit** dialog box displays after you select a block. You can use the REFCLOSE command or the Ref Edit toolbar to end the reference editing session. The XFADECTL system variable sets the fading the rest of the drawing when editing an external reference or block. Values between 0 (no fading) and 90 are accepted. If the XEDIT system variable of the selected external reference is 0 or OFF, a dialog box displays the message that you cannot edit the selected xref block.

Note: Only one reference can be edited at a time.

Note: The HIDEOBJECTS, ISOLATEOBJECTS, and UNISOLATEOBJECTS commands are enabled.

23.14 -REFEDIT command

Edits block references and externally referenced drawings.



23.14.1 Description

Edits the selected block or externally-referenced drawing. The XFADECTL system variable sets the fading the rest of the drawing when editing an external reference or block. Values between 0 (no fading) and 90 are accepted. Use the REFCLOSE command or the Refedit toolbar to end reference editing session.

Note: Only one reference can be edited at a time.

23.14.2 Options

Nesting level
Specifies a nested reference to edit.

Ok
Edits the selected reference.

Next
Edits reference on a deeper nesting level.

Note: This option will keep repeating until 0 is entered.

Entity selection method
Specifies how nested references are selected.

All
Selects all nested entities automatically.

Nested
Selects specific nested entities.

inside Window
Selects all nested entities inside a rectangular window, specified by picking 2 opposite corners.

Crossing window
Selects all nested entities crossing a rectangular window, specified by picking 2 opposite corners.

Window Polygon
Selects all nested entities inside a polygonal window, specified by picking points.

Crossing Polygon
Selects all nested entities crossing a polygonal window, specified by picking points.

Add
Adds entities to the selection.

Remove
Removes entities from selection.

Undo
Undoes the last step of selecting entities.

Display attribute definitions
Toggles the inclusions of attributes on or off.

Yes
Attribute definitions can be edited, all attribute values are visible.

No
Attributes are unavailable for editing.

Note: Modified attribute definitions do not affect existing insertions; they come into effect with insertions made from now on.

23.15 REFERENCECURVES command

Creates reference geometry to automatically align a block during insertion.

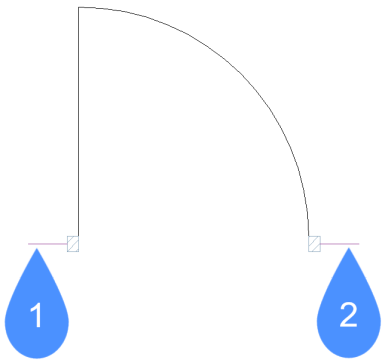
Shape Lite Pro Mechanical BIM

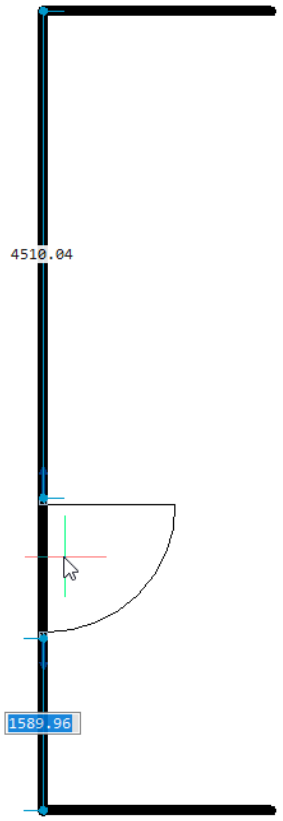
Icon:

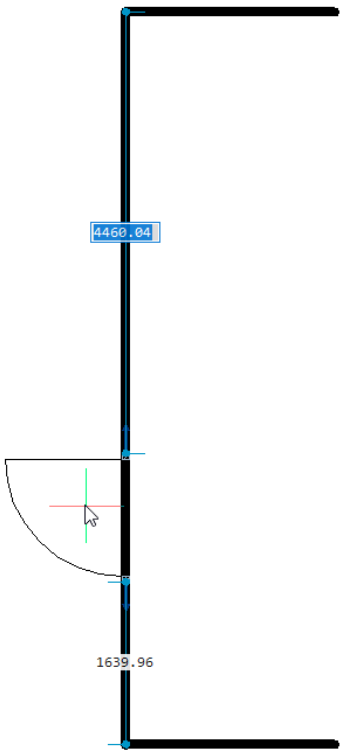
23.15.1 Method

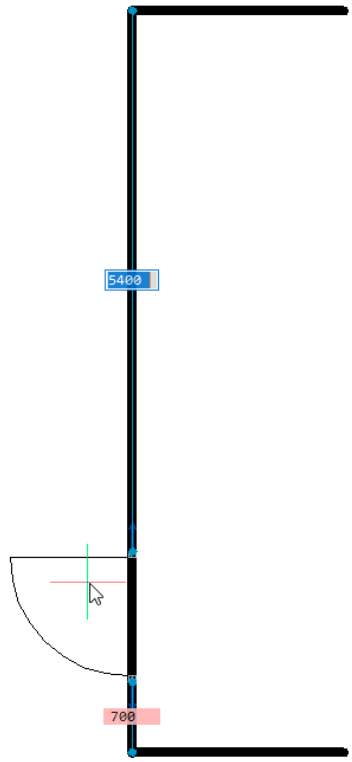
The selected entities which you want to use as a reference to align the inserted block will be moved in a new created REFERENCE_CURVES layer (if it doesn't already exist).

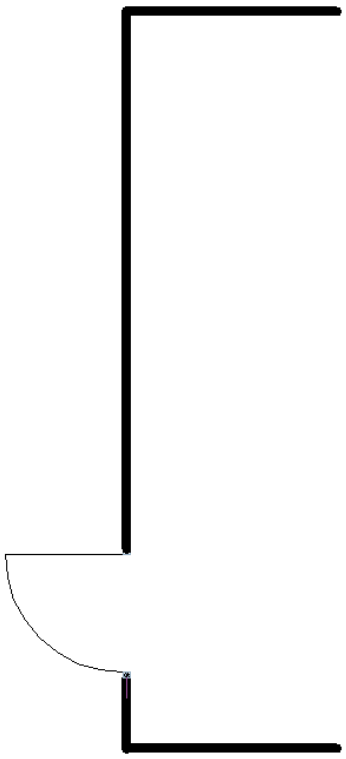
With reference curves ((1) and (2)) specified in a drawing or block definition, you can automatically align the block or drawing to relevant geometry when you insert it. The number of reference curves and the distance between them determines with which geometry it can align. As the cursor approaches relevant geometry, the block can automatically flip, offering multiple insertion options. Distances between the ends of the relevant geometry and block are displayed, enabling you to enter specific values if you wish. And, if the reference curves include gaps, relevant geometry is automatically trimmed to produce matching gaps.



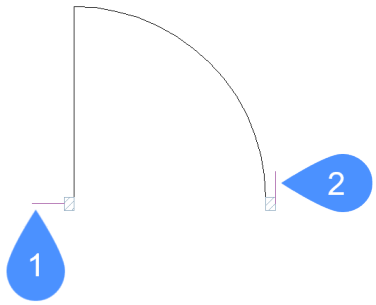

4510.04
1589.96

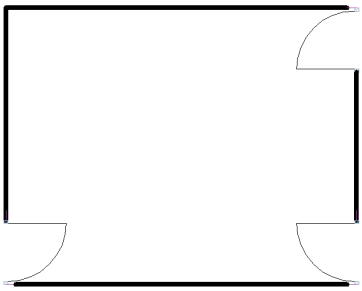

4460.04
1639.96


5400
700



You can also use reference curves to automatically align with corners. The following example includes a parallel reference curves (1) and a corner reference curve (2) enabling the door block to align with geometry that matches the parallel line and the perpendicular line.

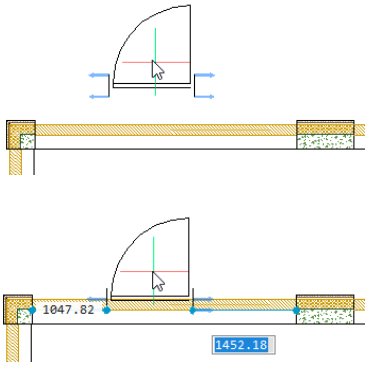




23.15.2 Options

Parametrize

Parametrizes the reference curves, so that the resulting parametric block can be inserted in a "fuzzy" mode. E.g. you can parametrize the four parallel reference curves in a simple door example. Then you can insert in a guided way the resulting parametric door block into walls with different thicknesses.



Accept

Creates reference curves without parametrizing.

23.16 REFSET command

Adds or removes entities from the reference (block or xref) being edited.

Shape Lite Pro Mechanical BIM

Icons: 

23.16.1 Description

Transfers objects between the RefEdit working set and host drawing.

Note: This command can be used only after the REFEDIT command has commenced.

23.16.2 Options

Add
Adds entities from the drawing to the reference being edited.

Remove
Removes entities from the reference being edited; removed entities are placed in the drawing. Entities not in the reference are shown in gray.

23.17 REGEN command

Regenerates entitiesin the current viewport.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: RE

23.17.1 Description

Regeneratesentitiesin the current viewportforoptimaldisplay andselectionperformance.

REGENdoes the following:

- Recalculates the location and visibilityof all entities in the current viewport.
- Reindexes the drawing database.
- Updates theareaavailableforzooming andpanning.

This command displays no prompts and has no options.

23.18 REGENALL command

Regeneratesentities in all viewports.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: REA

23.18.1 Description

Regenerates entities in all viewports for optimal display and selection performance.

REGENALL does the following:

- Recalculates the location and visibilityof all entities.
- Reindexes the drawing database.
- Updates the area available for zooming andpanning.

This command displays no prompts and has no options.

23.19 REGENAUTO command

Toggles the REGENMODE system variable.

Shape Lite Pro Mechanical BIM

23.19.1 Description

Toggles the REGENMODE system variable to specify if the display automatically regenerates when needed. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'REGENMODE.

- On: turns on the REGENMODE system variable.
- Off: turns off the REGENMODE system variable.
- Toggle: changes the REGENMODE system variable to the opposite of the current setting.

23.20 REGION command

Creates regions.

Shape Lite Pro Mechanical BIM

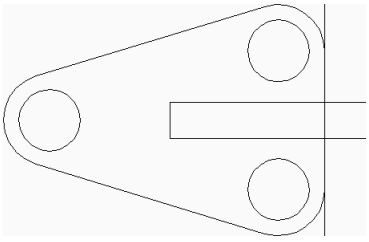
Icon:

Alias: REG

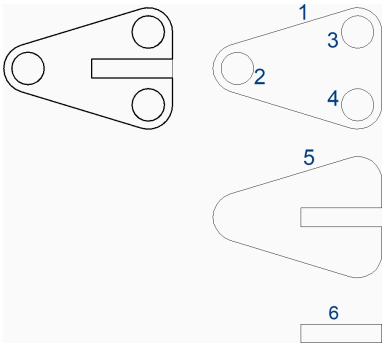
23.20.1 Description

Creates regions from closed entities or sets of entities that enclose a space.

Original entities:



Regions (6):



23.20.2 Options to create a region

This command has 1 option to begin creating a region:

- Select entities or boundaries

You can continue creating regions until you press Enter to end the command.

Select entities or boundaries
Begin creating a region by selecting closed entities or sets of entities that enclose a space. Each closed entity or set of entities produces a region. Depending on the current value of the DELOBJ system variable the original geometry is deleted or retained.
Additional options: [selection options (?)]

23.20.3 Options within the REGION command

Selection options (?)

23.21 REINIT command

Reloads the alias (PGP) file.

Shape Lite Pro Mechanical BIM

Alias: RI

23.21.1 Description

Reloads the alias (PGP) file after it has been edited externally. A BricsCAD dialog box confirms if you want to reload the file.

23.22 RENAME command

Renames named entities through the Drawing explorer dialog box.

Shape Lite Pro Mechanical BIM

Alias: DDRENAME, REN

23.22.1 Description

Opens theDrawing explorer dialog boxon the last used tab or category to manage definitions and reference content that is used in the drawing. Rename can be accessed through the context menu.

Note: To change names at the Command line, use the-RENAMEcommand.

You cannot change the names of attached files, such as images and external references, nor the following names:

- Layer "0"
- Linetype "ByBlock", "ByLayer", or "Continuous"
- Text style "Standard"
- Dimension style "Standard"
- Multiline style "Standard"
- Visual style "2DWireframe"
- Material "Global"
- Page layouts that have * as a prefix and suffix, such as *Model*
- View detail style "Metric50"
- View section style "Metric 50"

23.23 -RENAME command

Renames entities in the Command line.

Shape Lite Pro Mechanical BIM

Alias: -REN

23.23.1 Options within the command

Block
Renames blocks.

Dimension style
Renames dimension styles.

LAyer
Renames layers.

LineType
Renames linetypes.

text Style
Renames text styles.

Table style
Renames table styles.

Ucs
Renames user-defined coordinate systems.

Vlew
Renames views.

ViewPort
Renames viewport configurations.

23.24 RENDER command

Opens the Render dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: RR

23.24.1 Description

Opens the Render dialog box to create a photo-realistic rendering of the current drawing.

23.25 -RENDER command

Renders 3D models.

Shape Lite Pro Mechanical BIM

23.25.1 Description

Generates photo-realistic renderings of 3D models in the current viewport, and applies materials and lights if available; works at the Command line.

23.25.2 Options

Render preset
Choose a preset rendering style from the ones included with the program, or else choose Other to select a custom preset made by users with the RENDERPRESET command:

- Draft - renders 3D models without ray tracing; fastest rendering speed.
- Low - renders with ray tracing set to level 3 (low).
- Medium - renders with ray tracing set to level 5 (medium).
- High - renders with ray tracing set to level 7 (high).
- Presentation - renders with ray tracing set to level 9 (highest); slowest rendering speed.
- Other - specify the name of a user-defined rendering preset.

Render destination
Display the rendering in one of the following locations:

- Viewport - displays the rendering in the current viewport. If the cursors hovers the working are area the rendering result will disappear as the viewport is redrawn.
- File - saves the rendering as a BMP file in the folder defined by the DWGPREFIX variable.
- Render view - displays the rendering in a separate window. The dimensions of the rendering are specified in pixels.

23.26 RENDERPRESETS command

Creates and edits properties for renderings through Drawing explorer dialog box.

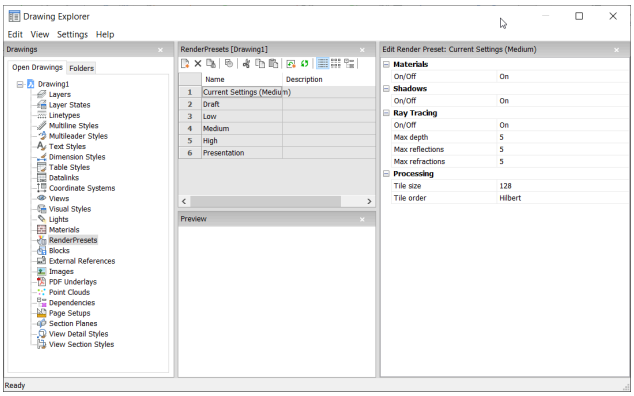
Shape Lite Pro Mechanical BIM

Icon:

Alias: ROPTIONS

23.26.1 Description

Opens the Drawing explorer dialog box with RenderPresets category selected to view and modify render presets in the current drawing.



23.26.2 Options within the command

Materials
Toggles the use of materials in a rendering

Note: Materials are applied to layers and entities, see Materials command.

On
Uses material definitions for rendering.

Note: This comes with the cost of a slower rendering process.

Off
Does not use materials.

Note: This comes with the cost of a less realistic rendering

Shadows
Toggles the use of shadows in the rendering

On
Uses shadows for rendering.

Note: This comes with the cost of a slower rendering process.

Off
Does not use shadows.

Note: This comes with the cost of a less realistic rendering.

Ray Tracing
Tracks each ray of light from each light source as it travels through the scene and bounces off entities.

On
Uses ray tracing.

Note: this comes with the cost of a slower rendering process.

Off
Does not use Ray tracing.

Note: this comes with the cost of a less realistic rendering.

Max Depth
Specifies the maximum total number of times light bounces and refracts.

Max Reflections
Specifies the maximum times that light beams bounce off entities.

Max Refractions
Specifies the maximum times that light refract through transparent entities.

Processing
Specifies how the rendered image is processed, applied to renderings sent to a separate window.

Tile Size
Sets the size of the tile, representing the area of the rendered image being processed.

Tile Order
Defines the order in which the tiles are generated

- **Hilbert**: renders with a Hilbert curve, a continuous fractal space-filling; seehttp://en.wikipedia.org/wiki/Hilbert_curvefor a description.
- **Spiral**: renders in a counterclockwise spiral from the center.
- **Left to right**: renders vertical columns, starting from the bottom left corner.
- **Right to left**: renders vertical columns, starting from the bottom right corner.
- **Top to bottom**: renders horizontal rows, starting at the top left corner.
- **Bottom to top**: renders horizontal rows, starting at the bottom left corner.

23.26.3 Context Menu Options

New
Creates additional Render Preset definitions into the drawing. Displays the New Render Preset dialog box.

Delete
Deletes Render Preset definitions from the drawing. The following Render Preset definitions cannot be deleted:

- Draft
- Low
- Medium
- High
- Presentation

Set current
Sets the selected Render Preset as current.

Note: The current render preset is used by the RENDER command or when printing using the shade plot type Rendered as defined in Shade Plot setting in the Page Setup and the Print commands. The name of the current render preset displays between brackets: e.g. Current Settings (Medium).

Note: You can override the properties of a preset in the Edit Render pane. When overrides exist, an asterisk displays in front of the current render preset name: e.g. Current Settings (*Medium).

Rename
Renames the Render Preset.

Note: FollowingRender Preset itemscannotbe renamed:

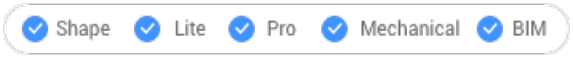
- Draft
- Low
- Medium
- High
- Presentation

Select All
Selects all Render Preset definitions.

Invert selection
Deselects the current selection and vice versa.

23.27 RENDERWINCLOSE command

Closes the Render window.

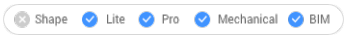


23.27.1 Description

Closes the Render window, which had displayed the result of the Render command.

23.28 REPORTPANELCLOSE command

Closes the Report panel.

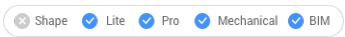


23.28.1 Description

Closes the Report panel to hide it from the current workspace. If the Report panel is stacked when you close it, the Report tab or icon is removed from the stack.

23.29 Команда REPORTPANELOPEN

Открывает панель Отчет.



23.29.1 Description (Описание)

Открывает панель Отчет для отображения его в текущем рабочем пространстве. Панель Отчета открывается в том же размере и в том же месте, что и до ее закрытия или сворачивания. Как и любая другая закрепляемая панель, панель Отчет может быть плавающей, закрепленной или вложенной.

23.30 RESETASSOCVIEWS command

Removes associativity between 2D drawings and 3D models.

Shape Lite Pro Mechanical BIM

23.30.1 Description

Removes the associativity between 2D drawings and 3D models, which had been generated by commands like VIEWBASE. This can be used to “freeze” drawing layouts at certain stages throughout the 3D model development, such as for archiving plans.

23.31 RESETBLOCK command

Resets parametric blocks to default values.

Shape Lite Pro Mechanical BIM

23.31.1 Description

Parametric blocks can be manipulated by users interactively, and so this command sets the blocks back to their original condition.

23.32 RESUME command

Resumes paused scripts.

Shape Lite Pro Mechanical BIM

23.32.1 Description

Resumes scripts after they have been stopped with Esc key.

23.33 REVCLOUD command

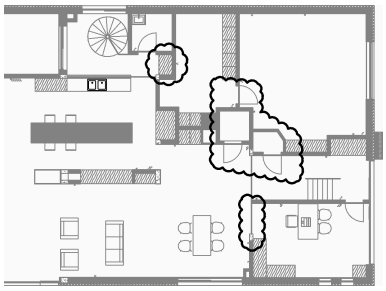
Creates a polyline in the shape of a revision cloud.

Shape Lite Pro Mechanical BIM

Icon:

23.33.1 Description

Creates a closed polyline in the shape of a revision cloud. Choose from a combination of options including rectangular, polygonal, freehand, and entity.



23.33.2 Methods to create a revision cloud

This command has 4 methods to begin creating a revision cloud:

- Rectangular
- Polygonal
- Freehand
- Entity

Rectangular
Begin creating a rectangular revision cloud by specifying a corner of the rectangle then:
Additional options: [Arc length/Entity/Rectangular/Polygonal/Freehand/Style]

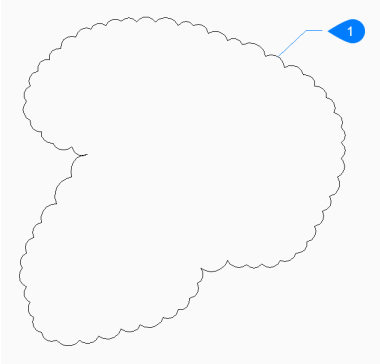
Specify opposite corner point
Specify the opposite corner of the rectangle. The revision cloud is drawn parallel to the x and y axes.

Polygonal
Begin creating a polygonal revision cloud by specifying a vertex of the polygon then:
Additional options: [Arc length/Entity/Rectangular/Polygonal/Freehand/Style]

Specify next point
Specify the next point of the polygon.
You can continue adding unlimited vertices until you press Enter to end the command.
Additional option: [Undo]

Freehand
Begin creating a freehand revision cloud by specifying a start point then:
Additional options: [Arc length/Entity/Rectangular/Polygonal/Freehand/Style]

Guide crosshairs along cloud path...
Drag the cursor to add revision cloud segments. Continue adding segments until you:
Pass the cursor over the start point to create a closed revision cloud.

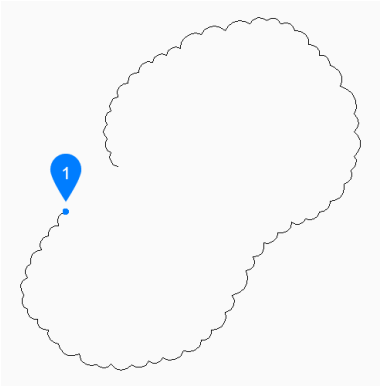


1 Closed revcloud

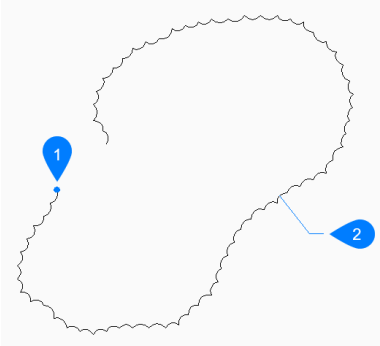
Or
Right-click to create an open revision cloud then:

Reverse direction [Yes/No]
Specify whether you want to change the direction of the revision cloud.

- **Yes**: the direction of the arc segments is reversed.
- **No**: the direction of the arc segment is retained.



1 Start point



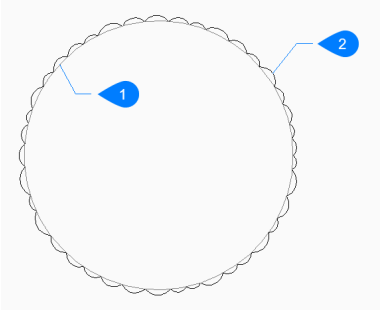
1 Start point

2 Reversed

Entity
Begin creating a revision cloud by selecting an existing entity to convert to a revision cloud then:

Reverse direction [Yes/No]
Specify whether you want to change the direction of the revision cloud.

- **Yes**: the direction of the arc segments is reversed.
- **No**: the direction of the arc segment is retained.



1 Original entity

2 Revcloud

Entities can be open or closed. The original entity is erased.

23.33.3 Options within the REVCLOUD command

After you begin creating a revision cloud, the following options may be available.

Arc length
Specify the shortest and longest length for the arcs that form the cloud.

Specify minimum length of arc
Specify the minimum length of the arcs.
The minimum arc length is stored in the REVCLOUDMINARCLENGTH system variable.

Specify maximum length of arc
Specify the maximum length of the arcs.
The minimum arc length is stored in the REVCLOUDMAXARCLENGTH system variable.

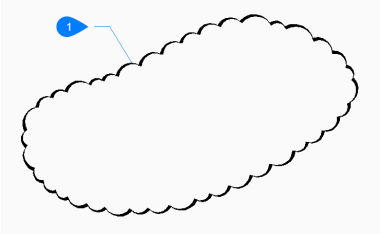


1 Minimum arc length

2 Maximum arc length

Style
Specify the arc style to use for the revision cloud.

- **Normal**: the polyline making up the cloud has a uniform width.
- **Calligraphy**: the polyline has a varying width (tapered polyarcs) to simulate a calligraphic look.



1 Calligraphic style

The arc style is stored in the REVCLOUDARCSTYLE system variable.

Undo
Undo the last segment in a polygonal revision cloud and continue drawing from the previous segment.

23.34 REVOLVE command

Creates 3D solids or surfaces by revolving 2D entities about an axis.

Shape Lite Pro Mechanical BIM

Icon:

Alias: REV

23.34.1 Description

Revolves open or closed 2D entities, solid edges, faces of 3D solids, regions or closed boundaries into 3D solids or 3D surfaces.

Note:

- The SELECTIONPREVIEW system variable must be set to 2 or 3 to highlight faces.
- Depending on the value of the DELOBJ system variable, the source entities are either retained or deleted, or else you are prompted whether you want the entities deleted or not.

23.34.2 Method

There are two methods to revolve entities:

- Create 3D solids.
- Create surfaces.

23.34.3 Options within the command

MOde
Allows you to create either solids or surfaces.

SOlid
Creates a 3D solid.

SUrface
Creates a 3D surface.

Xaxis
Uses the X axis of the current coordinate system as the axis of rotation.

Yaxis
Uses the Y axis of the current coordinate system as the axis of rotation.

Zaxis
Uses the Z axis of the current coordinate system as the axis of rotation.

2Points
Specify the start point and end point of the axis of revolution.

Note: If the resulting 3D entity is self-intersecting, an error will occur.

Object
Specifies the axis of rotation by selecting an axial entity of the object.

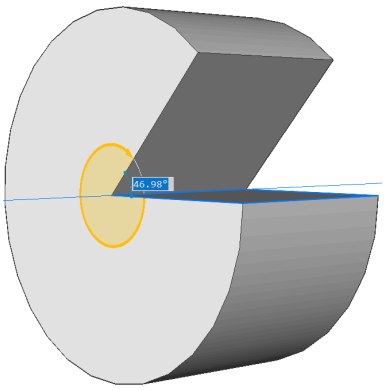
Last
Uses the last entered axis for rotating the entities.

View
Uses the current view as the axis for rotating the entities.

Note: The axis of rotation is parallel to the direction of view, passing through the picked point.

Angle of revolution
Specifies how far around the source entity is rotated about the axis.

Note: You can specify the angle of revolution dinamically, using the **Manipulator**, or typing a value.



Auto
Move the mouse to specify the angle direction.
The result depends on the extrusion direction and the value of the EXTRUDEOUTSIDE, EXTRUDEINSIDE, INTERSECTEDENTITIES and UNITESURFACES system variables.

Subtract
The 3D solid is subtracted from each interfering existing solid.

Create
Regardless of the revolving direction, a new 3D solid is created.

Unite
The new 3D solid is unified with each interfering existing solid.

Both sides
Revolves in both directions.

Note: If the HOTKEYASSISTANT system variable is ON, the **Hotkey Assistant** widget displays. Repeatedly press the **Ctrl** key during the dynamic display of the extrusion to cycle the various options:

- Solid mode:



  - : Auto
  - : Create
  - : Subtract
  - : Unite

- Surface mode:



  - : Auto
  - : Create

23.35 REVSURF command

Creates a 3D mesh surface by revolving a linear entity around a line (short for "revolved surface").

Shape Lite Pro Mechanical BIM

Icon:

Note: To create 3D surfaces and 3D solids by revolving a profile around an axis, use the REVOLVE command.

The SURFTAB1 system variable controls the number of segments of the revolution surface.

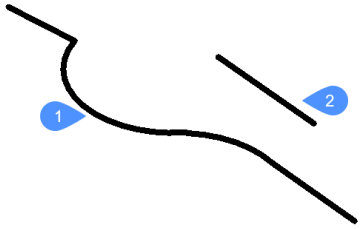
The SURFTAB2 system variable controls the number of segments of each arc segment in the revolved entity

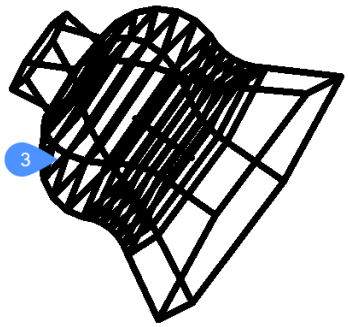
23.35.1 Method

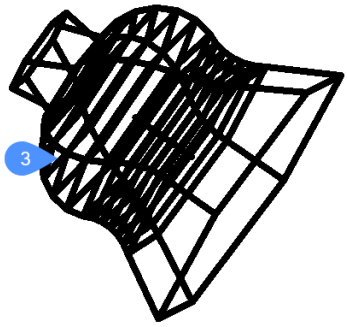
Specifies a single entity (1) to be revolved into a 3D surface mesh (3). You may choose a line, circle, arc, open or closed polyline, or open or closed spline; 3D objects do not work.

Specifies the entity (2) about which the surface is revolved; you may choose a line or polyline; curved objects do not work.

Specifies the starting angle (4) of the revolution and the angle revolved (5), the number of degrees by which the entity is revolved.







23.36 RIBBON command

Opens the Ribbon panel.

Shape Lite Pro Mechanical BIM

23.36.1 Description

Opens the Ribbon panel to display it in the current workspace. The Ribbon panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Ribbon panel can be either floating, docked or stacked.

23.37 RIBBONCLOSE command

Closes the Ribbon panel.

Shape Lite Pro Mechanical BIM

23.37.1 Description

Closes the Ribbon panel to hide it from the current workspace. If the Ribbon panel is stacked when you close it, the Ribbon tab or icon is removed from the stack.

23.38 ROTATE command

Rotates entities about a specified point.

Shape Lite Pro Mechanical BIM

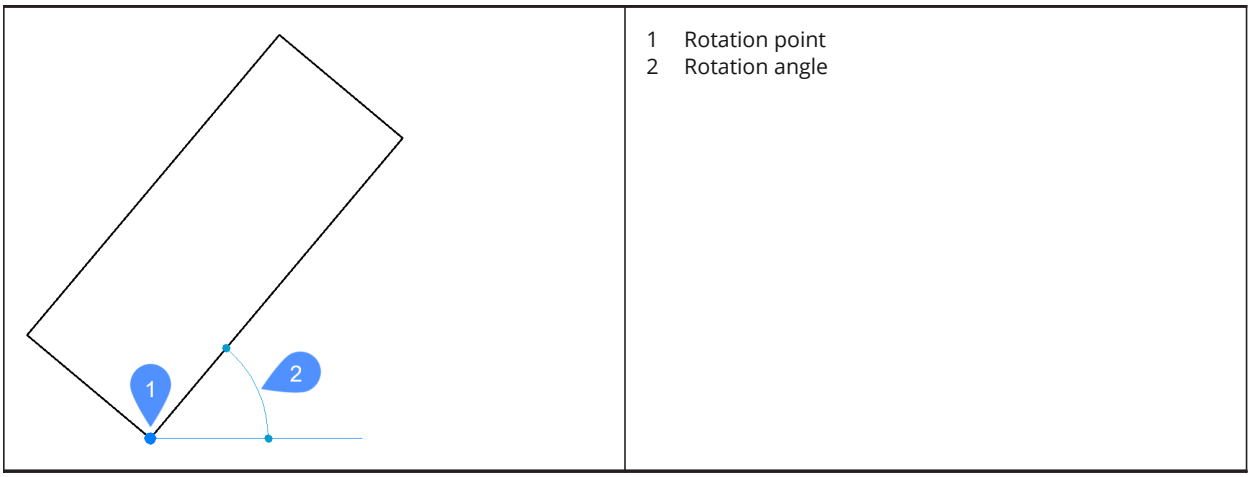
Icon:

Alias: RO

23.38.1 Description

Rotate entities about a specified point at a specified rotation angle, or by an angle referenced to a base angle.

Note: A positive number rotates the entities counter-clockwise. A negative number rotates the entities clockwise. The starting angle is 0 degrees, and the rotation angle is measured from the positive x axis.



23.38.2 Options within the command

Base angle

Specifies a new base angle, which is a starting angle other than 0. This option lets you define a different angle as the starting point.

Copy

Rotates a copy of the source entity, instead of rotating the source itself.

23.39 ROTATE3D command

Rotates entities about an axis in 3D space.

Shape Lite Pro Mechanical BIM

Icon:

Alias: 3DROTATE, 3R

23.39.1 Description

Dynamically rotates 3D solids, surfaces, 2D entities, faces (planar, cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface), edges, or vertices of a solid around an axis.

Note: When the face of a solid or surface is rotated, adjacent faces and edges are adjusted to preserve the correct solid/surface topology.

23.39.2 Method

There are two methods to rotate entities:

- Specify an axis.
- Use the geometry of an object as the axis for rotating the selected entities.
  - Hover over a line or linear polyline segment. Click when the entity highlights.
  - Hover over a 3D solid face, then move the cursor close to the edge you want to use as the rotation axis. A rotation arc and the rotation axis display dynamically. Click to accept the rotation axis.

23.39.3 Options within the command

Xaxis
Rotates the selection around the X-axis of the current coordinate system.

Yaxis
Rotates the selection around the Y-axis of the current coordinate system.

Zaxis
Rotates the selection around the Z-axis of the current coordinate system.

2points
Defines the rotation axis by two points.

Object
Uses the geometry of an object to determine the axis for rotating the selected entities.

Start
Starts the rotation of the selected entity. It automatically selects a rotation axis if the cursor is over a planar face.

Last
Uses the last entered axis for rotating the entities.

View
Uses the current view as the axis for rotating the entities.

Note: The axis of rotation is parallel to the direction of view, passing through the picked point.

Rotation angle
Specifies the angle by which to rotate the entities.

Base angle
Allows you to define a new base angle.

Copy
Rotates a copy of the selection set.

Repeat
Creates multiple copies.

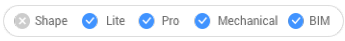
Disable connectivity mode
Removes the connectivity with adjacent entities.

Enable connectivity mode
Keeps the connectivity with adjacent entities.

Note: If the HOTKEYASSISTANT system variable is set on, press Ctrl key to cycle between disable/enable connectivity mode.

23.40 RSCRIPT command

Reruns the currently loaded SCR script file.



Note: Load and run SCR script files with the Script command.

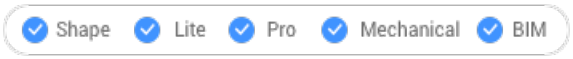
Note: This command can be entered transparently during commands (**'rscript**).

23.40.1 Method

After a script file is loaded and run, enter RSCRIPT to run it again (short for "repeat script").

23.41 RTLOOK command

Looks around in a 3D scene.



23.41.1 Method

The look-around cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to look around in real-time.

23.41.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.42 RTPAN command

Pans the drawing in real-time.

Shape Lite Pro Mechanical BIM

Icon: 

23.42.1 Method

The pan cursor is displayed during real-time panning. Hold down the mouse's left button and then move the mouse to pan the drawing in real-time.

Note: Hold down the Shift key to restrict panning to the X and Y direction of the viewport.

23.42.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.43 RTROT command

Rotates 3D drawings in a constrained manner, in real-time.

Shape Lite Pro Mechanical BIM

Icon: 

23.43.1 Method

The constrained rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

Note: The ORBITAUTOTARGET variable determines if the viewpoint rotates around your pick point or the center of the objects.

23.43.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.44 RTROTCTR command

Rotates 3D drawings about a center point that you pick in the drawing, in real-time.

Shape Lite Pro Mechanical BIM

23.44.1 Method

The constrained rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

23.44.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.45 RTROTF command

Rotates 3D drawings about a fixed point, in real-time.

Shape Lite Pro Mechanical BIM

Icon: 

23.45.1 Method

The rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to look around in real-time.

Note: The ORBITAUTOTARGET system variable determines if the viewpoint rotates around your pick point or the center of the object.

23.45.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.46 RTROTX command

Rotates 3D drawings in real-time about the x axis.

Shape Lite Pro Mechanical BIM

Icon: 

23.46.1 Method

The rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

Note: The ORBITAUTOTARGET system variable determines if the viewpoint rotates around your pick point or the center of the object.

23.46.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.47 RTROTY command

Rotates 3D drawings in real-time about the y axis.

Shape Lite Pro Mechanical BIM

Icon: 

23.47.1 Method

The rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

23.47.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.48 RTROTZ command

Rotates 3D drawings in real-time about the z axis.

Shape Lite Pro Mechanical BIM

Icon:

23.48.1 Method

The rotation cursor is displayed during real-time rotation. Hold down the mouse's left button and then move the mouse to rotate the drawing in real-time.

23.48.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.49 RTUPDOWN command

Moves the viewpoint up and down, and left and right in a 3D scene, in real time.

Shape Lite Pro Mechanical BIM

23.49.1 Method

The up-down cursor is displayed during real-time movement. Hold down the mouse's left button and then move the mouse to move up-down and left-right in real-time.

23.49.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.50 RTWALK command

Walks left-right and forward-backwards through 3D scenes, in real-time.

Shape Lite Pro Mechanical BIM

23.50.1 Method

The walk cursor is displayed during real-time movement. Hold down the mouse's left button and then move the mouse to walk left-right and forward-backward in real-time.

23.50.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.51 RTZOOM command

Zooms the viewpoint in and out of the drawing in real-time.

Shape Lite Pro Mechanical BIM

Icon: 

23.51.1 Method

The zoom cursor is displayed during real-time zooming. Hold down the mouse's left button and then move the mouse to zoom the drawing in real-time.

23.51.2 Context Menu Options

Right-click to display the context menu to switch viewing mode.

Pan
Pans the drawing in real-time. Starts the RTPAN command.

Zoom
Zooms the drawing in real-time. Starts the RTZOOM command.

Constrained sphere
Rotates the 3D drawing in real-time. Starts the RTROTCTR command.

Sphere
Rotates the 3D drawing in real-time. Starts the RTROT command.

23.52 RULESURF command

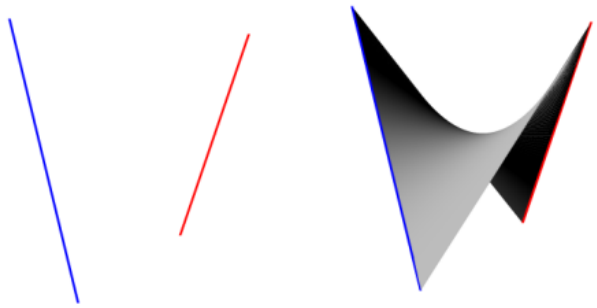
Creates 3D polygon mesh by connecting two separate linear entities (short for "ruled surfaces").

Shape Lite Pro Mechanical BIM

Icon:

23.52.1 Description

Creates a 3D polygon mesh in the shape of a ruled surface by specifying two separate entities.



23.52.2 Method

Specify the first and the second boundary entity. For the first boundary any open object, such as a line, arc, polyline or spline can be selected.

Closed entities and 3D entities are not accepted.

24. S

24.1 SAVE command

Opens the Save drawing as dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SA

24.1.1 Description

Opens the Save drawing as dialog box to save data from the current drawing to a dwg, dxf, dwt or dws file.

File types available to save are:

- **AutoCAD 2018** (*dwg)
- **AutoCAD 2018 ASCII DXF** (*dxf)
- **AutoCAD 2018 Binary DXF** (*dxf)
- **Standard File** (*dws)
- **Drawing Template** (*dwt)
- **AutoCAD 2013** (*dwg)
- **AutoCAD 2013 ASCII DXF** (*dxf)
- **AutoCAD 2013 Binary DXF** (*dxf)
- **AutoCAD 2010** (*dwg)
- **AutoCAD 2010 ASCII DXF** (*dxf)
- **AutoCAD 2010 Binary DXF** (*dxf)
- **AutoCAD 2007** (*dwg)
- **AutoCAD 2007 ASCII DXF** (*dxf)
- **AutoCAD 2007 Binary DXF** (*dxf)
- **AutoCAD 2004** (*dwg)
- **AutoCAD 2004 ASCII DXF** (*dxf)
- **AutoCAD 2004 Binary DXF** (*dxf)
- **AutoCAD 2000** (*dwg)
- **AutoCAD 2000 ASCII DXF** (*dxf)
- **AutoCAD 2000 Binary DXF** (*dxf)
- **AutoCAD Release 14** (*dwg)
- **AutoCAD Release 14 ASCII DXF** (*dxf)
- **AutoCAD Release 14 Binary DXF** (*dxf)
- **AutoCAD Release 14** (*dwg)

- **AutoCAD Release 14 ASCII DXF** (*dxf)
- **AutoCAD Release 14 Binary DXF** (*dxf)
- **AutoCAD Release 13 Drawing** (*dwg)
- **AutoCAD Release 13 ASCII DXF** (*dxf)
- **AutoCAD Release 13 Binary DXF** (*dxf)
- **AutoCAD Release 11/12 Drawing** (*dwg)
- **AutoCAD Release 11/12 ASCII DXF** (*dxf)
- **AutoCAD Release 11/12 Binary DXF** (*dxf)
- **AutoCAD Release 10 ASCII DXF** (*dxf)
- **AutoCAD Release 10 Binary DXF** (*dxf)
- **AutoCAD Release 9 ASCII DXF** (*dxf)

24.2 SAVEALL command

Opens the Save drawing as dialog box.

Shape Lite Pro Mechanical BIM

24.2.1 Method

The dialog is displayed for each open drawing that has been modified since the previous save.

24.3 SAVEAS command

Opens the Save drawing as dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

24.3.1 Description

Opens the Save drawing as dialog box to save data from the current drawing to a dwg, dxf, dwt or dws file.

Note: File types available for saving are the same as for the SAVE command.

24.4 SAVEASR12 Command

Opens the Save drawing as dialog box.

Shape Lite Pro Mechanical BIM

24.4.1 Description

Opens the Save drawing as dialog box to save data from the current drawing to an R11/12 dwg file.

24.5 SAVEFILEFOLDER command

Opens the default file explorer.

Shape Lite Pro Mechanical BIM

24.5.1 Description

Opens the default file explorer to quickly access automatic save and backup files. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

24.6 SCALE command

Resizes 2D and 3D entities.

Shape Lite Pro Mechanical BIM

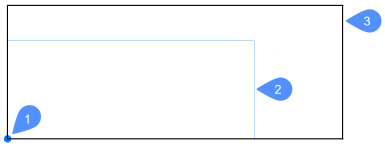
Icon:

Alias: SC

24.6.1 Method

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.

Note: A scale factor smaller than 1 shrinks the entities. A negative scale factor scales the entities in the negative direction.



1 Base point
2 Entity to scale
3 Scaled entity

24.6.2 Options within the command

Reference

Resizes entities using a base or reference scale.

Note: Use this option to scale entities relative to other entities. This command normally scales from a base factor of 1.

Copy

Scales a copy of the selection set, leaving the original entities intact.

24.7 SCALELISTEDIT command

Opens the Edit scale list dialog box.

Shape Lite Pro Mechanical BIM

24.7.1 Description

Opens the Edit scale list dialog box to edit the list of scales accessible by some commands.

24.8 -SCALELISTEDIT command

Edits preset scale factors.

Shape Lite Pro Mechanical BIM

24.8.1 Description

Adds and removes scale factors to and from the list used by commands, such as Print, and by annotative scaling; works at the Command line.

24.8.2 Options within the command

? to list
Lists existing preset scale factors in the Prompt History window.

Add
Creates a new scale factor.

Delete
Removes specified scale factor or all unused scale factors.

Reset
Removes all custom scale factors that are not used.

24.9 SCREENSHOT command

Opens the Save image file dialog box.

Shape Lite Pro Mechanical BIM

24.9.1 Description

Opens the Save image file dialog box to save data from the current drawing to a png file. The current view specifies what data to include in the image.

24.10 SCRIPT command

Opens the Run script dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SCR

24.10.1 Description

Opens the Run script dialog box to select an scr file to run. After choosing Open in the dialog box, the script begins running immediately. You can press the Esc key to stop it.

24.11 Команда SCROLLBAR

Переключает значение системной переменной WNDLSCRL.

Shape Lite Pro Mechanical BIM

24.11.1 Description (Описание)

Переключает системную переменную WNDLSCRL для показа или скрытия полос прокрутки графической области. Вы можете запустить эту команду с помощью командной строки или при выполнении другой команды, добавив перед ней апостроф: 'SCROLLBAR.

- On: включает системную переменную WNDLSCRL.
- Off: выключает системную переменную WNDLSCRL.
- Переключение: изменяет значение системной переменной WNDLSCRL на противоположное.

24.12 SECTION command

Creates section planes.

Shape Lite Pro Mechanical BIM

Icon:

Alias: SEC

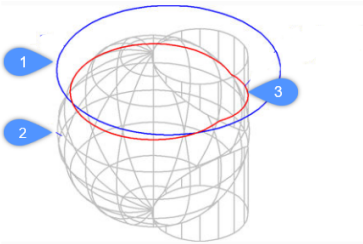
24.12.1 Description

Creates section planes of 3D solids, surfaces, polyface meshes, and 3D faces. The result is a slice made of region entities.

24.12.2 Options within the command

Object

Creates the section from entities that intersect the 3D solid.



1 Circle entity

2 3D solid

3 Section defined by the plane of the circle.

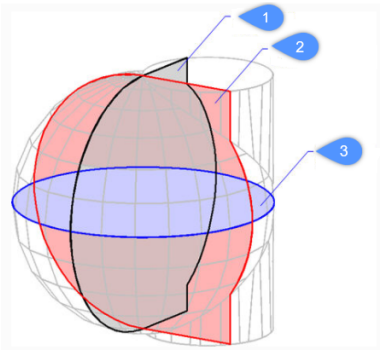
Note: The entity defines the plane through which to create the section.

Zaxis

Creates a section in the plane defined by one point on the plane, and a second on the plane's normal (z axis).

View

Creates a section in the plane of the current 3D viewport.



XY (3)
Creates a section parallel to the x, y plane.

YZ (2)
Creates a section parallel to the y, z plane.

ZX (1)
Creates a section parallel to the z, x plane.

3points
Pick points to specify the location of the plane in which the section lies.

Note: Three points define a plane.

24.13 SECTIONPLANE command

Creates section entities.

Shape Lite Pro Mechanical BIM

Icon:

24.13.1 Description

Creates section entities from 3D solids, 3D surfaces, polyface meshes and 3D faces. This command helps to see inside a 3D entity.

Note: New created section planes have their Clip Display status set to on. As section planes are entities, they can be edited and deleted.

24.13.2 Options within the command

Specify start point
Specifies the starting point of a 2D section plane.

Note: BricsCAD ghosts the section plane during the cursor's movement.

Note: Use entity snaps to make the pick points accurate.

select Face
Select the planar face of a 3D solid.

Note: BricsCAD ghosts the section plane during the cursor's movement. Use entity snaps to make the pick points accurate.

Note: The section plane coincides with the selected face. The Live Section property is turns on automatically. Live sections are sections that can be interactively edited through the Properties panel.

Draw

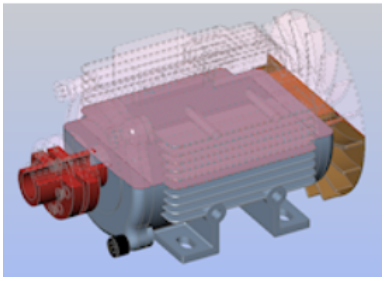
Define the section plane by two ore more points. Draws L and U-shaped section planes.

Note: The section plane is created through the specified points and perpendicular to the current UCS.

Orthographic

Places a section plane entity in one of the six standard drawing orientations, relative to the current UCS.

Note: BricsCAD draws the orthographic section plane through the center of the bounding box that contains all 3D entities in the drawing. E.g.; the result of a Top section plane:



24.14 SECTIONPLANESETTINGS command

Shape Lite Pro Mechanical BIM

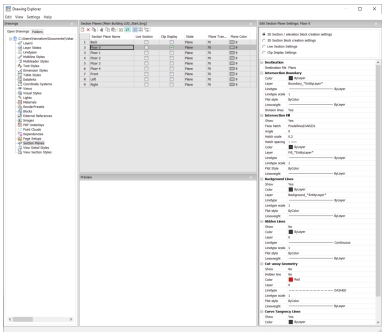
Icon:

24.14.1 Description

Opens the Drawing Explorer dialog box with Section Planes category selected.

24.14.2 Methods

Open the Drawing explorer dialog boxwithSection Planescategory selected to view and modifysection planes in the current drawing.



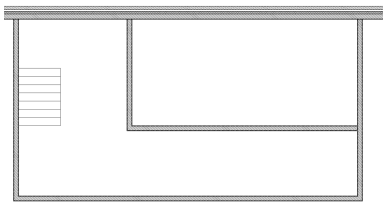
Note: The available options differ depending on the kind of section you choose to edit.

24.14.3 Options within the command

Section Types

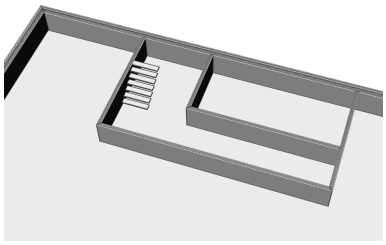
2D Section/elevation block creation settings

Specifies properties for 2D sections and elevations. These properties are assigned to the 2D section you create as a block.



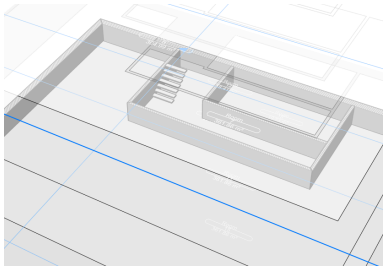
3D section block creation settings

Specifies properties for 3D sections. These properties are assigned to the 3D section you create as a block.



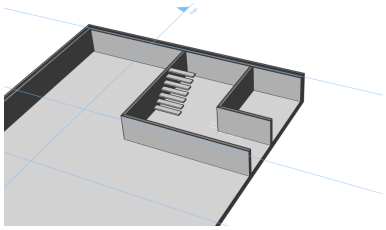
Live Section Settings

Specifies properties for live sections. With live sections, the entities in the drawing are temporarily modified and viewable in the drawing.



Clip Display Settings

When using the clip display status, the display of entities is clipped. The status can be set for multiple section entities simultaneously.



Destination

Destination file
Specifies the destination file to save the section plane to.

Intersection Boundary

Color
Defines the color of the intersection boundary. You can choose a color from the drop list or choose Select Color, the Select Color dialog box will then display.

Layer
Specifies the layer of the intersection boundary. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.
- New layer name settings: option opens the New Layer Name dialog box. Here you can edit the name settings for the option above.

Linetype
Specifies the linetype of the intersection boundary. You can choose between:

- The available linetypes in the drawing
- Load to load new linetypes. This opens the Load Linetypes dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the intersection boundary.

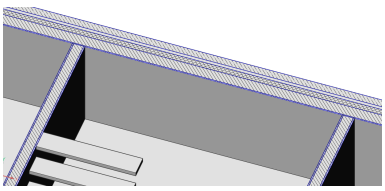
Plot style
Specifies the plotstyle of the intersection boundary.

Lineweight
Specifies the lineweight of the intersection boundary.

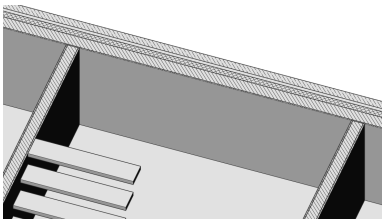
Division lines (option available for the 2D sections)
Specifies whether the division lines of the intersection boundary should be drawn.

Show (option available for the 3D sections)
Specifies whether the intersection boundary is drawn.

- Yes: the intersection boundary is displayed. The intersection boundaries are indicated in blue.



- No: no intersection boundary is displayed.



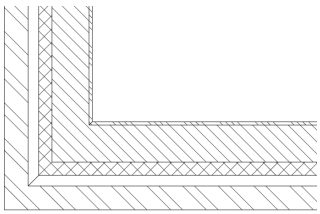
Note: The intersection boundary is still visible, since the lines are also drawn for the intersection fill. The color is now determined by the intersection fill instead of by the intersection boundary.

Intersection Fill

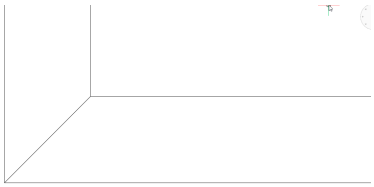
Show

Specifies whether the intersection filling is showed.

- Yes: the cut through geometry is displayed by a hatch



- No: no hatch is drawn on the cut through geometry



Face hatch

Specifies the used hatch pattern to fill the section. Opens the Hatch Pattern Type dialog box where you can choose a hatch pattern.

Angle

Specifies the angle of the hatch pattern.

Hatch scale

Specifies the scale of the hatch pattern.

Hatch spacing

Specifies the hatch spacing.

Color

Specifies the color of the hatch. You can choose a color from the drop list or choose Select Color, the Select Color dialog box will then display.

Layer

Specifies the layer of the intersection boundary. You can choose between:

- The available layers in the drawing.

- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.

- New layer name settings: option opens the New Layer Name dialog box. Here you can edit the name settings for the option above.

Linetype

Specifies the linetype of the intersection boundary. You can choose between:

- The available linetypes in the drawing
- Load to load new linetypes. This opens the Load Linetypes dialog box where you can choose new linetypes to load into the drawing.

Linetype scale

Specifies the linetype scale of the hatch pattern.

Plot Style

Specifies the plot style of the hatch pattern.

Lineweight

Specifies the lineweight of the hatch pattern.

Background Lines

Show

Specifies whether the background lines are drawn. These are the lines that show the geometry behind the section plane, which is not cut-through.

- Yes: the background lines are displayed



- No: no background lines are displayed



Color

Specifies the color of the background lines. You can choose a color from the drop list or choose Select Color, the Select Color dialog box will then display.

Layer

Specifies the layer of the background lines. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.

- New layer name settings: option opens the New Layer Name dialog box. Here you can edit the name settings for the option above.

Linetype
Specifies the linetype of the background lines. You can choose between:

- The available linetypes in the drawing
- Load to load new linetypes. This opens the Load Linetypes dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the background lines.

Plot Style
Specifies the plot style of the background lines.

Lineweight
Specifies the lineweight of the background lines.

Hidden lines

Show
Specifies whether the hidden lines are drawn.

- Yes: the hidden lines are displayed.



- No: no hidden lines are displayed.



Color
Specifies the color of the hidden lines. You can choose a color from the drop list or choose Select Color, the Select Color dialog box will then display.

Layer
Specifies the layer of the hidden lines. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.

- New layer name settings: option opens the New Layer Name dialog box. Here you can edit the name settings for the option above.

Linetype
Specifies the linetype of the hidden lines. You can choose between:

- The available linetypes in the drawing
- Load to load new linetypes. This opens the Load Linetypes dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the hidden lines.

Plot Style
Specifies the plot style of the hidden lines.

Lineweight
Specifies the lineweight of the hidden lines.

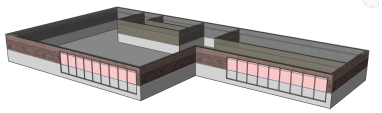
Cut-away Geometry

Show
Specifies whether the section lines of the geometry that is cut away are shown.

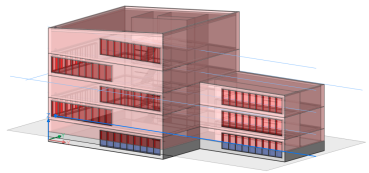
- Yes: the cut-away geometry is displayed.



Cut-away geometry of a 2D section.



Cut-away geometry of a 3D section.

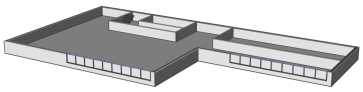


Cut-away geometry of a live section.

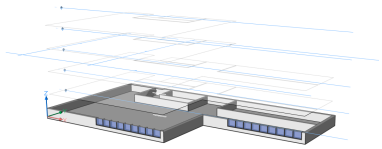
- No: no cut-away geometry lines are displayed



No cut-away geometry of a 2D section is shown.



No cut-away geometry of a 3D section is shown.



No cut-away geometry of a live section is shown.

Hidden line
Specifies whether the hidden lines of the cut-away geometry are shown.

Color
Specifies the color of the cut-away geometry (lines). You can choose a color from the drop list or choose Select Color, the Select Color dialog box will then display.

Layer
Specifies the layer of the cut-away geometry. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.
- New layer name settings: option opens the New Layer Name dialog box. Here you can edit the name settings for the option above.

Linetype
Specifies the linetype of the cut-away geometry. You can choose between:

- The available linetypes in the drawing
- Load to load new linetypes. This opens the Load Linetypes dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the cut-away geometry lines.

Plot Style
Specifies the plot style of the cut-away geometry lines.

Lineweight
Specifies the lineweight of the cut-away geometry lines.

Curve Tangency Lines

Show
Specifies whether the curve tangency lines are drawn.

Color
Specifies the color of the curve tangency lines. You can choose a color from the drop list or choose Select Color, the Select Color dialog box will then display.

Layer
Specifies the layer of the curve tangency lines. You can choose between:

- The available layers in the drawing.
- *EntityLayer*_IntersectionBoundary: the properties of layers of the entities in the section are kept, but a copy of these layers is created.

  Note: This overwrites the other specified properties.
- New layer name settings: option opens the New Layer Name dialog box. Here you can edit the name settings for the option above.

Linetype
Specifies the linetype of the curve tangency lines. You can choose between:

- The available linetypes in the drawing
- Load to load new linetypes. This opens the Load Linetypes dialog box where you can choose new linetypes to load into the drawing.

Linetype scale
Specifies the linetype scale of the curve tangency lines.

Plot Style
Specifies the plot style of the curve tangency lines.

Lineweight
Specifies the lineweight of the curve tangency lines.

24.14.4 Context Menu Options

New
Creates a new section plane definitions into the drawing. For a detailed explanation, see the SECTIONPLANE command.

Delete
Deletes section planes from the drawing.

Rename
Renames the selected section plane.

Select All
Selects all section plane definitions.

Invert selection
Deselects the current selection and vice versa.

24.15 SECTIONPLANETOBLOCK command

Opens the Create section/elevation dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

24.15.1 Description

Opens the Create section/elevation dialog box to save section geometry as a block or to an external dwg file.

24.16 SECURITY command

Opens the Security dialog box.

Shape Lite Pro Mechanical BIM

24.16.1 Description

Opens the Security dialog box to specify if VBA macros are allowed to run automatically.

24.17 SECURITYOPTIONS command

Opens the Security options dialog box.

Shape Lite Pro Mechanical BIM

24.17.1 Description

Opens the Security options dialog box to specify a password to protect your drawing from unauthorized access.

24.18 SELECT command

Selects entities.

Shape Lite Pro Mechanical BIM

Icon: 

24.18.1 Description

Selects one or more entities through a variety of selection methods.

Note: You access the selection set during subsequent commands with the Previous option.

24.18.2 Methods

There are two methods to select entities:

- Select entities separately using the command.
- Crtl+A selects all entities on non-frozen layers.

24.18.3 Options within the command

Select entities to include in set
Chooses entities to add to the selection set; use one of methods listed below. As you add entities to the selection set, the program highlights them with dashes.

select ALL entities
Selects all entities in the drawing, except for those on frozen layers.

Add
Adds additional entities to the selection set. (Used after the Remove option by typing Add in the Command line.)

add to set (+)
Adds additional entities to the selection set. (Used after the Remove option by typing + in the Command line.)

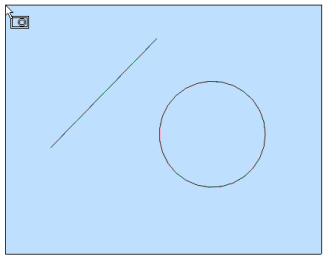
Remove
Removes entities from the selection set. As you remove entities from the selection set, the program unhighlights them.

subtract from set (-)
Removes entities from the selection set. As you remove entities from the selection set, the program unhighlights them.

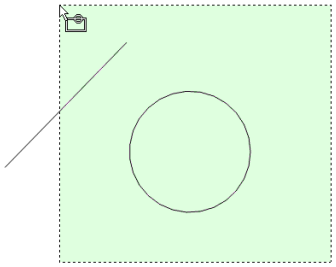
Previous selection
Add the entities from the previous selection to the current select set.

Last entity in drawing
Adds the last entity drawn to the selection set.

inside Window
Selects all entities that are entirely inside a rectangular selection window. The selection window outline is drawn, filled with transparent blue.

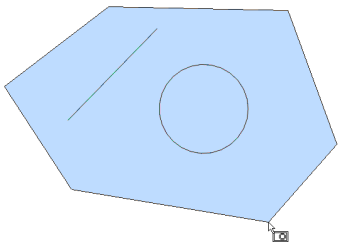


Crossing window
Selects all entities that are inside of and cross a rectangular selection window. The selection window outline is drawn in a dashed line, filled with transparent blue.

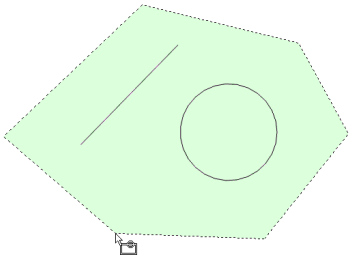


Outside window
Selects all entities fully outside a rectangular selection window.

Window Polygon
Selects all entities fully inside a polygonal selection window. The selection polygon outline is drawn, filled with transparent blue.

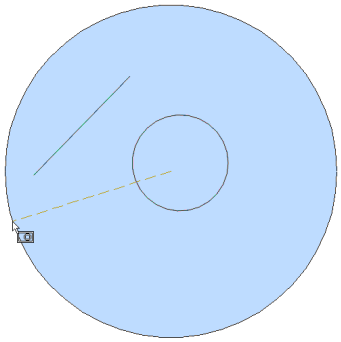


Crossing Polygon
Selects all entities inside and crossing a polygonal selection window. The selection polygon outline is drawn with a dashed line, filled with transparent green.



Outside Polygon
Selects all entities fully outside a polygonal selection window.

Window Circle
Selects all entities fully inside a circular selection window. The selection circle outline is drawn, filled with transparent blue.



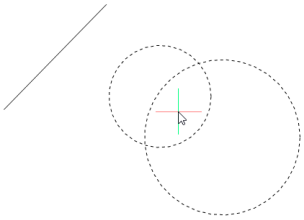
Crossing Circle
Selects all entities inside and crossing a circular selection window. The selection circle outline is drawn with a dashed line, filled with transparent green.

Outside Circle
Selects all entities fully outside a circular selection window.

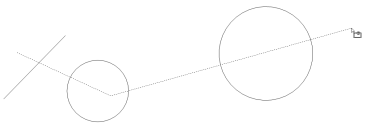
Box
Selects entities depending on the direction you move the cursor:

- Right to left - Box acts like Crossing mode.
- Left to right - Box acts like Window mode.

POint
Selects all closed entities surrounding the pick point, such as all walls of a room.



Fence
Selects all entities that cross the fence selection lines.



AUto
Selects entities depending on how you use the cursor:

- Pick an entity - the entity is selected
- Pick two points right to left - Auto acts like Crossing mode.
- Pick two points left to right - Auto acts like Window mode.

Multiple
All entities selected during this mode are added to the selection set.

Single
After you select a single entity, the command ends.

select by PROperties…
Selects entities based on their properties.

Note: For the panel version of this option, see the QSELECT command.

Color
Selects entities based on their color.

Note: :If there are entities colored in a specific color because the color properties of the layer or the block are on that specific color, these entities will not be selected.

COlorbook
Selects entities with a colorbook color.

Note: If there are entities colored in a colorbook color because the color properties of the layer or the block, these entities will not be selected.

Layer
Selects entities based on their layer. Enter * to select all layers.

LineType
Selects entities based on their linetype. Enter * to select all linetypes.

Name
Selects entities based on their entity name. Enter * to select all entity names.

Thickness
Selects entities based on their thickness.

Type
Selects entities based on their entity type, such as line, circle, arc... Enter * to select all entity types.

Value
Selects entities based on their value. Enter * to select all values.

Width
Selects entities based on the width of the entities.

Handle
Selects entities based on the value of a handle. This is a hexadecimal number that uniquely identifies each entity in the drawings; the List command reports handle numbers.

Location
Returns to the 'Select entities to include in set' prompt.

selection methoDs…
Displays the Entity Selection section of the Settings dialog box.

Undo
Undoes the last action during this command.

Group
Adds a group to the selection set. See the GROUP command.

Note: In macros, the ^S metacharacter selects the entity under the cursor automatically.

24.19 SELECTALIGNEDFACES command

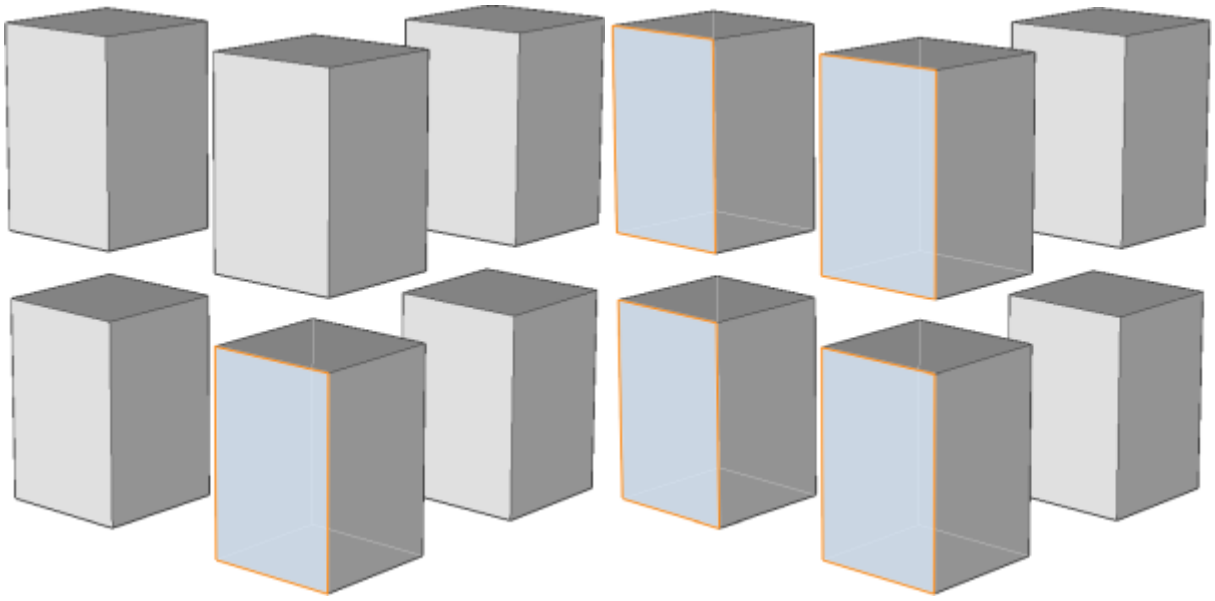
Selects all faces in a model that are coplanar with a selected face.

Shape Lite Pro Mechanical BIM

Icon: 

24.19.1 Method

Select a single face. The program selects all other faces that are in the same plane, shown outlined.



24.20 SELECTALIGNEDSOLIDS command

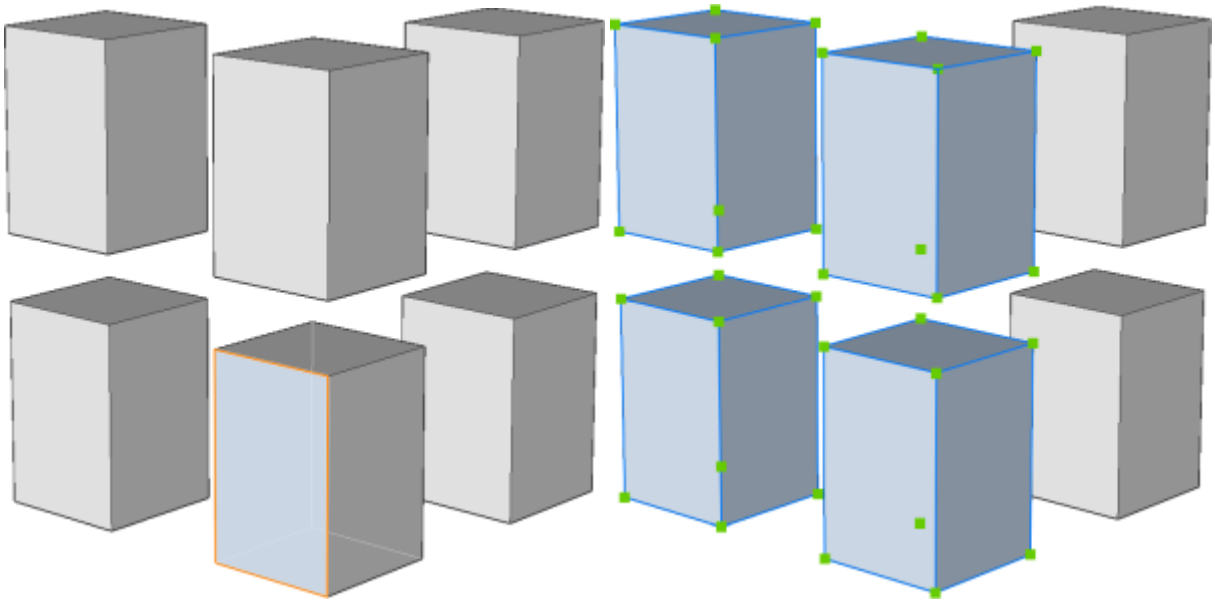
Selects all solids in a model that have faces coplanar with the selected face.

Shape Lite Pro Mechanical BIM

Icon:

24.20.1 Method

Select a single face. The program selects all solids that have faces in the same plane, shown outlined.



24.21 SELECTCONNECTEDFACES command

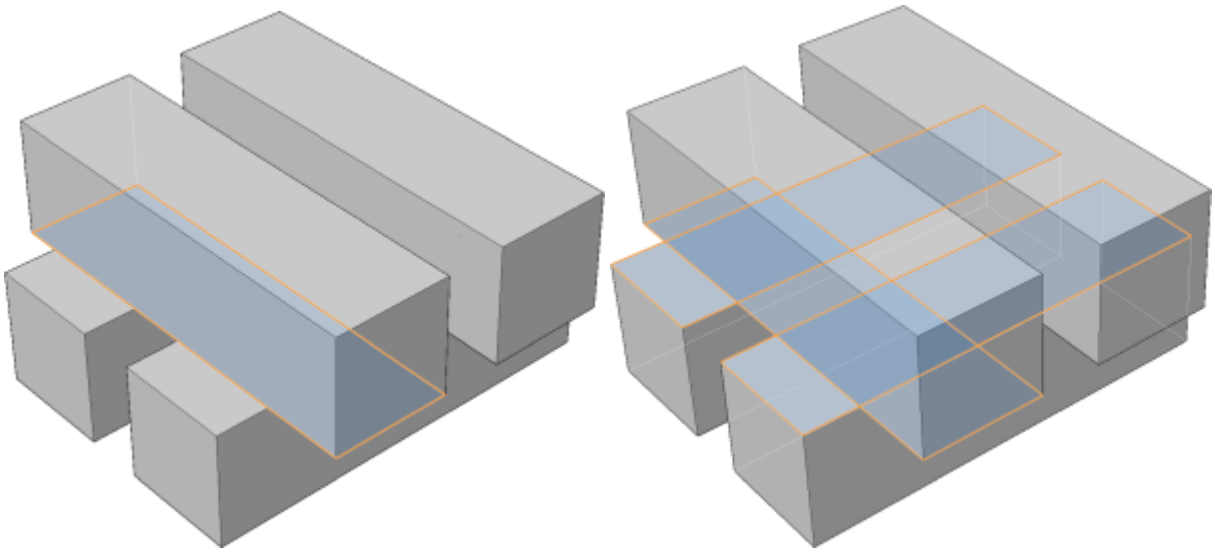
Selects all faces in a model that are connected to a selected face.

Shape Lite Pro Mechanical BIM

Icon:

24.21.1 Method

Select a face. The program selects all other faces that are connected to the selected face, shown outlined.



24.22 SELECTCONNECTEDSOLIDS command

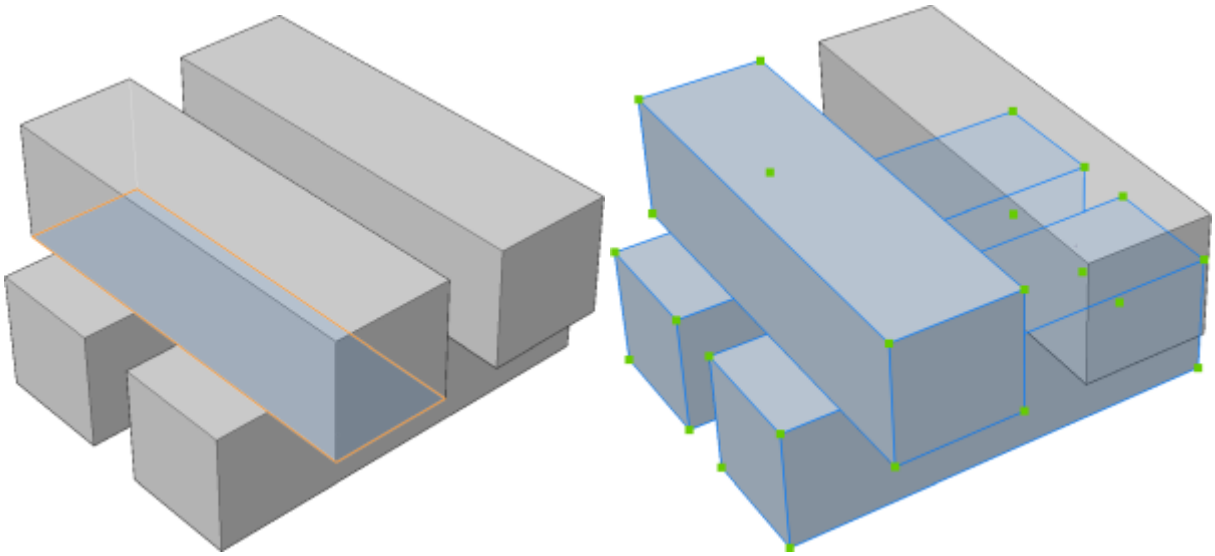
Selects all solids in a model that are connected to a selected face.

Shape Lite Pro Mechanical BIM

Icon:

24.22.1 Method

Select a face. The program selects all solids that are connected to the selected face, shown outlined.



24.23 SELECTSIMILAR command

Selects similar entities.

Shape Lite Pro Mechanical BIM

Icon:

24.23.1 Description

Selects all entities with the same properties as the currently selected entity.

24.23.2 Method

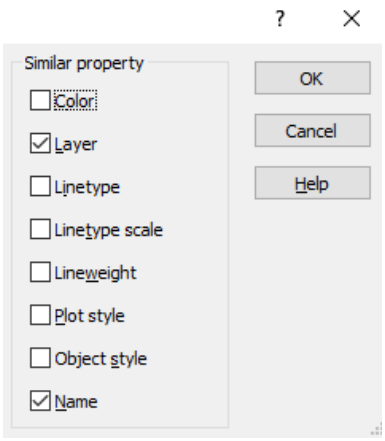
Select one or more entities and choose the properties based on which the selection will be made. When you exit the command, the entities that match the properties are shown as dashed and with entity grips.

Note: For this command to operate as intended, at least one property must be selected.

24.23.3 Options within the command

SEttings

Displays a dialog box to select the properties.



Note: Changing the settings in this dialog box changes the value of the SELECTSIMILARMODE system variable, which is accessed through the Settings command.

24.24 SELGRIPS command

Displays entities grips.

Shape Lite Pro Mechanical BIM

24.24.1 Description

Selects entities and then displays their grips.

BricsCAD highlights selected entities and displays their grips.

Edit the entities by dragging the grips. To select all entities in the drawing, press Ctrl+A.

24.25 SETBYLAYER command

Resets overridden properties to BYLAYER.

Shape Lite Pro Mechanical BIM

24.25.1 Description

Changes the value of overridden properties (color, linetype, lineweight, material, plot style, and transparency) back to the default value of BYLAYER.

24.25.2 Methods

Select the entities whose properties will be set as BYLAYER according to the value of the SETBYLAYERMODE system variable.

Enter ‘all’ to select all non-frozen entities in the current viewport.

When entities contain blocks the BricsCAD asks whether blocks should have their properties changed.

24.25.3 Options

Settings

Allows to select the entity properties to be changed from the SetByLayer settings dialog box:

- Color - resets the color of the selected entities to BYLAYER.
- Linetype - resets the linetype of the selected entities to BYLAYER.
- Lineweight - resets the lineweight of the selected entities to BYLAYER.
- Material - resets the material of the selected entities to BYLAYER.
- Plot Style - (only in drawings with plot styles) resets the plot style of the selected entities to BYLAYER.
- Transparency - resets the transparency value of the selected entities to BYLAYER.

Note: The settings are saved through the SETBYLAYERMODE system variable.

Note: The Plotstyle option is available for STB drawings only. See the CONVERTCTB and CONVERTPSTYLES command.

24.26 -SETBYLAYER command

Sets the property overrides of selected entities to ByLayer.

Shape Lite Pro Mechanical BIM

24.26.1 Description

Changes the value of overridden properties (color, linetype, lineweight, material, plot style, and transparency) back to the default value of BYLAYER.

24.26.2 Methods

Select the entities whose properties will be set as BYLAYER according to the value of the SETBYLAYERMODE system variable.

Enter ‘all’ to select all non-frozen entities in the current viewport.

When entities contain blocks the BricsCAD asks whether blocks should have their properties changed.

24.26.3 Options

Settings

Allows to select the entity properties to be changed from the command line:

- Color - resets the color of the selected entities to BYLAYER.
- Linetype - resets the linetype of the selected entities to BYLAYER.
- Lineweight - resets the lineweight of the selected entities to BYLAYER.
- Material - resets the material of the selected entities to BYLAYER.
- Plot Style - (only in drawings with plot styles) resets the plot style of the selected entities to BYLAYER.
- Transparency - resets the transparency value of the selected entities to BYLAYER.

Note: The settings are saved through the SETBYLAYERMODE system variable.

Note: The Plotstyle option is available for STB drawings only. See the CONVERTCTB and CONVERTPSTYLES command.

24.27 SETLICENSELEVEL command

Changes the available commands by setting the license type.

Shape Lite Pro Mechanical BIM

24.27.1 Description

Runs BricsCAD and/or Communicator at different license levels, which affects the commands that are available to the user. For the new license level to take effect, users must exit BricsCAD and then start it again.

24.27.2 Methods

There are two methods to change the license level:

- Change the license level of BricsCAD.
- Change the license level of Communicator.

24.27.3 Options

Run BricsCAD at another license level

- Run BricsCAD as: Lite - runs BricsCAD without 3D functionalities, except API and LISP.
- Pro - runs BricsCAD without BIM and Mechanical commands.
- Bim - runs BricsCAD with Pro and BIM commands.
- Mechanical - runs BricsCAD with Pro and Mechanical commands.
- Ultimate - runs BricsCAD with full functionalities, including BIM and Mechanical commands.

Run Communicator at another license level

- Run Communicator with: No license - disables Communicator.
- Trial - stops running Communicator after 30 days.
- Full - runs Communicator normally.

24.28 SETTINGS command

Opens the Settings dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

24.28.1 Description

Opens the Settings dialog box to view and modify system variables. Most, but not all, system variables are available in the Settings dialog box. You can modify all system variables using the SETVAR command.

24.29 SETTINGSSEARCH command

Opens the Settings Dialog box at the word specified by the user.

Shape Lite Pro Mechanical BIM

24.29.1 Description

Searches the in the Settings dialog box for the word specified by the user.

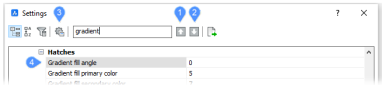
24.29.2 Methods

1 Enter a single word. The search word that will be matched to the following items:
   - Category titles
   - Variable titles, names, and values
   - Variable help text

2 The program displays the Settings dialog box and goes to the first entry that matches the search word.

3 Click the Find Previous and Find Next buttons to find other instances of the search word.

4 Click the Configuration button to change how the program searches Settings.



1 Find Previous

2 Find Next

3 Configuration

4 Search phrase found

24.30 SETUCS command

Opens the User coordinate system dialog box.

Shape Lite Pro Mechanical BIM

Icon:

Alias: DDUCSP, UCP

24.30.1 Description

Opens the User coordinate system dialog box to specify a viewpoint to use for the current UCS (User Coordinate System).

24.31 SETVAR command

Displays and changes the values of system variables.

Shape Lite Pro Mechanical BIM

Alias: SET

24.31.1 Description

Displays and changes the values of system variables at the Command line.

Note: This command can be entered transparently: ‘SETVAR.

Note: The names of system variables can also be entered directly at the Command line, without using this command.

24.31.2 Methods

There are two methods to use SETVAR command:

- Enter the name of a system variable to see its state.
- Choose to display a list that contains all the system variable.

24.31.3 Options

Variable name
Specify the name of a system variable.

?
Lists the names of system variables.

*****
Lists all system variables.

Name*
Lists all system variables starting with Name.

24.32 SH command

24.33 SHADE command

Generates shaded images of 3D drawings.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SHA

24.33.1 Description

Generates shaded images of the current 3D drawing, based on the current setting of the SHADEMODE command. Shaded images are simpler versions of visual styles and rendered images.

24.34 SHADEMODE command

Specifies the shading style for the current drawing.

Shape Lite Pro Mechanical BIM

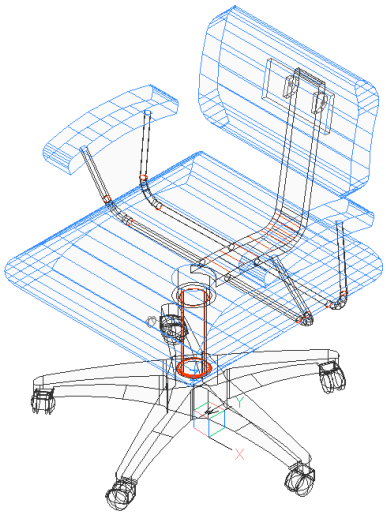
24.34.1 Description

Specifies the shading style for the current drawing to be used by the SHADE command.

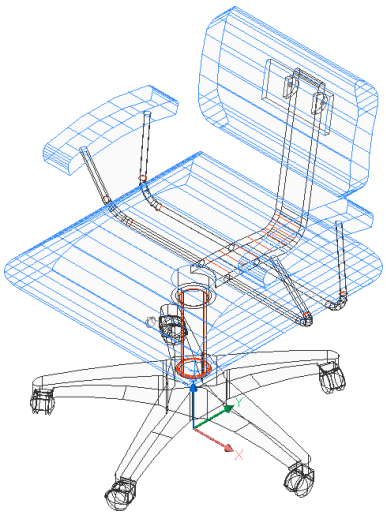
24.34.2 Options

Enter visual style
Choose the name of a preset visual style.

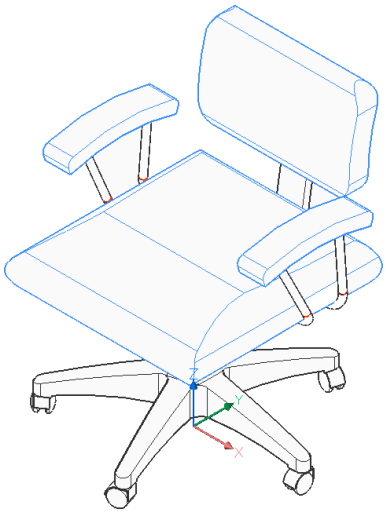
2dwireframe
Default display mode.



Wireframe
Wireframe, which shows all edges.



Hidden
Wireframe with hidden lines removed.



Realistic
Rendered with materials, if available.



Conceptual
Rendered with psychographic color range.



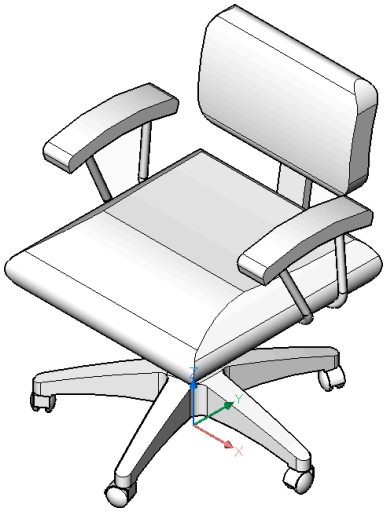
Shaded
Rendered without materials.



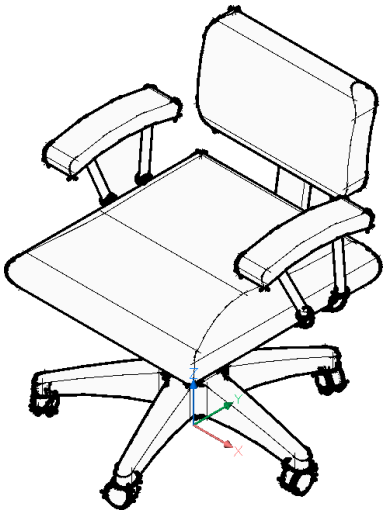
shaded with Edges
Rendered with contrasting edges, without materials.



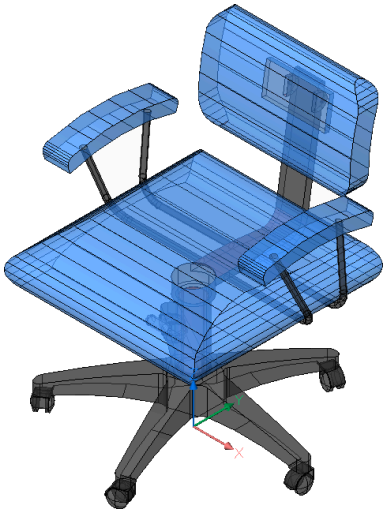
shades of Grey
Rendered in shades of gray, without colors.



Sketchy
Rendered with rough edge lines.



X-ray
Rendered with semi-transparent faces.



Other
Create a new visual style.

cUrrent
keep the current visual style.

24.35 -SHADEMODE command

Specifies the shading style for the SHADE command.

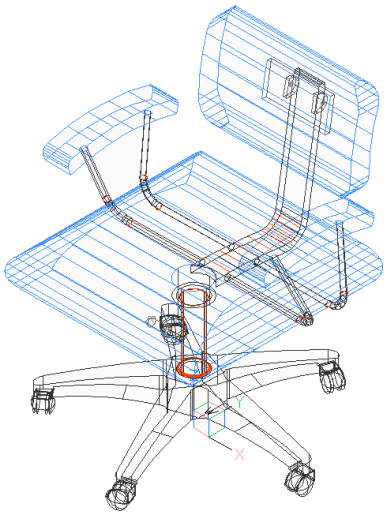
Shape Lite Pro Mechanical BIM

24.35.1 Description

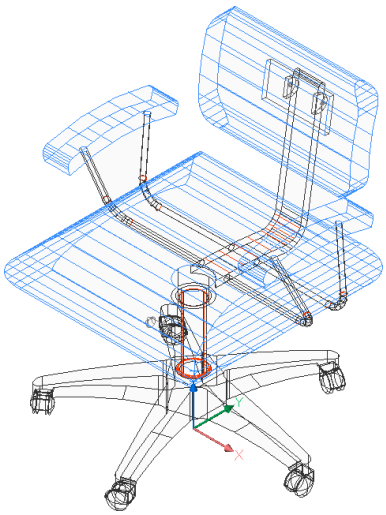
Specifies the shading style for the current drawing to be used by the SHADE command.

24.35.2 Options

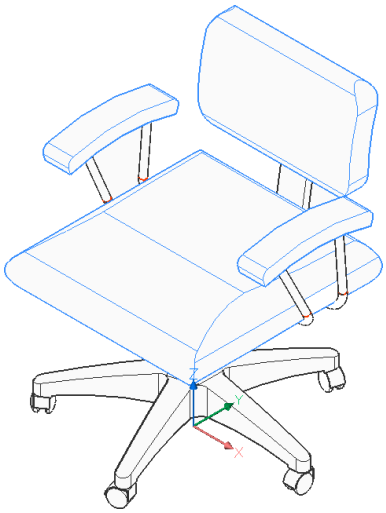
2dwireframe
Wireframe display with no shading.



3d wireframe
Wireframe display with no shading.



Hidden
Wireframe with hidden lines removed.



Flat
Flat shading.



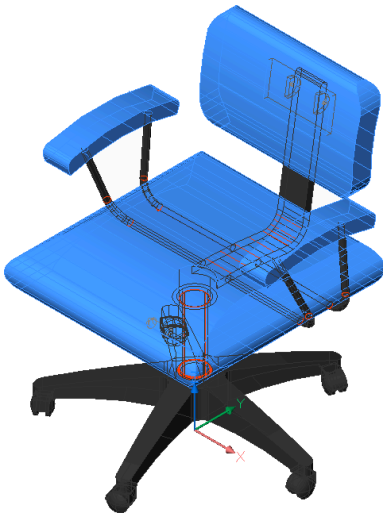
Gouraud
Smooth shading.



fLat+edges
Flat shading with edges highlighted.



gOuraud+edges
Smooth shading with edges highlighted.



24.36 SHAPE command

Places shapes in drawings.

Shape Lite Pro Mechanical BIM

Icon: 

Note: Before you can place shapes in drawings, you must first load a SHX shape file with the Load command.

24.36.1 Method

Enter the name of the shape and specifies the insertion point, the scale and rotation angle to place it in the drawing.

Shapes are an early version of blocks that were highly efficient but difficult to code; shapes are rarely used anymore. The SHP format is also used by certain fonts; it contains a flag to distinguish between shapes and fonts.

24.36.2 Options within the command

? to list

Lists the names of shapes loaded in the current drawing. Reports the file name and names of shapes:
Shape file: C:\temp\611.shx
HAIE BOIS
HOTALU BATALU
CLOTUR FROST
RAILS PGA
PGAE PGBR
If no shapes are loaded, reports: “No shapes loaded.”

Note: Shapes can be edited through grips.

24.37 SHEETSET command

Opens the Sheet sets panel.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SSM

24.37.1 Description

Opens the Sheet sets panel to display it in the current workspace. The Sheet sets panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Sheet sets panel can be either floating, docked or stacked.

24.38 SHEETSETHIDE command

Closes the Sheet sets panel.

Shape Lite Pro Mechanical BIM

24.38.1 Description

Closes the Sheet sets panel to hide it from the current workspace. If the Sheet sets panel is stacked when you close it, the Sheet sets tab or icon is removed from the stack.

24.39 SHELL command

Opens command prompt window.

Shape Lite Pro Mechanical BIM

24.39.1 Description

Opens command prompt window and runs other applications.

Note: This is a Windows only command.

24.39.2 Method

There are two methods to use the SHELL command:

- Specify the name of a program to run.
- Press Enter to open the command prompt window.

24.40 SIGVALIDATE command

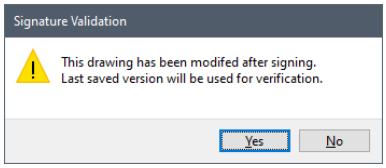
Displays the digital signatures for the drawing and its Xrefs.

Shape Lite Pro Mechanical BIM

24.40.1 Method

Opens the Validate Digital Signature dialog box.

Note: If the current drawing was modified after the last signing, the warning message below will be displayed:



Note: The SIGWARN system variable controls the display of a dialog with the signature content when a drawing with a digital signature is opened.

24.41 SIMPLIFY command

Simplifies entities.

Shape Lite Pro Mechanical BIM

Icon:

24.41.1 Description

Reduces the number of vertices of polylines, splines and non-associative hatches with geometry boundaries represented by polylines, without changing their general shape.

Note: Simplified entities are easier to manipulate and can significantly reduce file size.

24.41.2 Options within the command

Simplify
Reduces the number of vertices.

max Angle and distance
Specifies the maximum distance between non-collinear vertices to straighten and the maximum change (angle) in a direction between each two consecutive segments to straighten. This method is available for lw/2d polylines, hatches and 3d polylines.

Straighten polylines arc segments?
Decides if straightens polyline arc segments (Yes/No).

Smoothen lw/2d polylines segments?
Decides if smoothens lw/2d polyline segments (Yes/No).

max Deviation
Specifies the maximum arc's length between each two adjacent vertices and maximum arc's angle between each two adjacent vertices. This method is available for lw/2d polylines and splines.

sMoothen
Controls whether consecutive straight segments (at least three), of which the endpoints lie on an arc, are replaced by a bulged polyline segment.

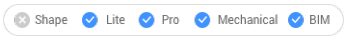
Maximum arc's length between each two adjacent vertices
Specifies the maximum arc's length between each two adjacent vertices.

Maximum arc's angle between each two adjacent vertices
Specifies the maximum arc's angle between each two adjacent vertices.

Minimum number of vertices per arc
Specifies the minimum number of vertices per arc.

24.42 SINGLETON command

Determines whether (or not) multiple copies of BricsCAD can run at the same time.



Note: This command can be entered transparently during commands ('singleton).

24.42.1 Options within the command

Yes
Allows only one copy of BricsCAD to run.

No
Allows two or more copies of BricsCAD to run.

Note: Some third-party applications run only in a single instance of BricsCAD.

24.43 SKETCH command

Creates lines or polylines from a freehand sketch.

Shape Lite Pro Mechanical BIM

Icon:

Alias: FREEHAND

24.43.1 Description

Creates a series of individual lines or a polyline by sketching freehand.



24.43.2 Methods to begin a sketch

This command has 1 method to begin creating a sketch:

- Length of sketch line segments

Length of sketch line segments

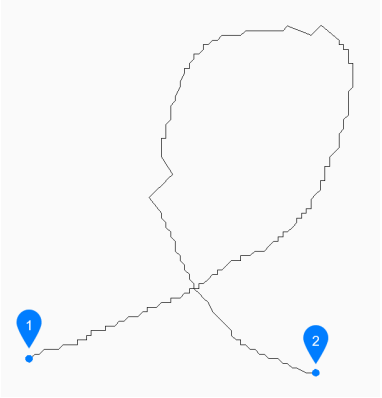
Begin creating a sketch by specifying the length of the sketch line segments. A small segment length creates a smoother sketch but increases the size of the drawing. The length is saved in the SKETCHINC variable.

Press ENTER or click to toggle

Press the Enter key or click the left mouse button to lower the pen. The cursor changes to a pen, automatically sketching as you move it.

Press ENTER or click to toggle

Press the Enter key or click the left mouse button to raise the pen. You can move the cursor without drawing.



1 Pen up

2 Pen down

You can continue raising and lowering the pen to create unlimited sketches until you press X or Q to end the command.

Additional options: [Erase/Connect/save and eXit/Quit no save/Record (save)]

24.43.3 Options within the SKETCH command

After you begin creating a sketch, the following options may be available:

Erase
Erase sketch lines sequentially by dragging the cursor along the sketch from either endpoint. Click the left mouse button to stop erasing and start sketching.

Connect
Continue to the last sketch point, after raising the pen, by passing the cursor over the end of the previous segment.

Save and exit
Save the sketch and end the command. Each continuous sketch is saved as a series of individual lines or a polyline based on the SKPOLY variable.

Quit no save
Exit the command without saving the sketch.

Record (save)
Save the current sketch without ending the command, allowing you to continue sketching.

24.44 SLICE command

Creates 3D solids and surfaces by cutting existing entities.

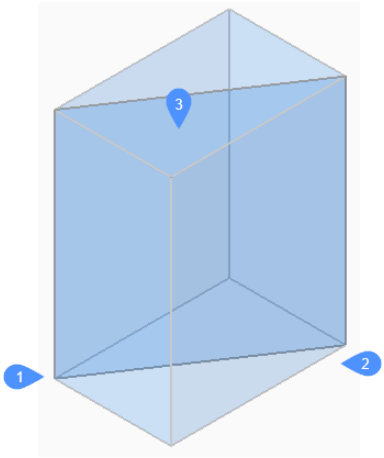
Shape Lite Pro Mechanical BIM

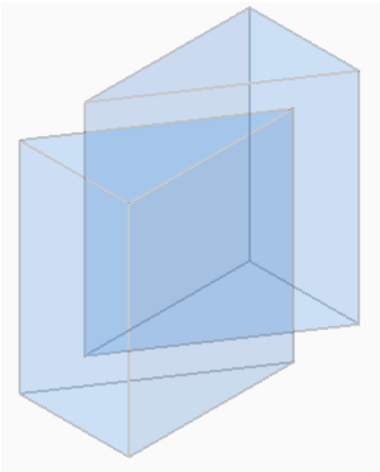
Icon: 

Alias: SL

Note: Ensure that the Select Faces option of the SELECTIONMODES system variable is on to be able to use a planar 3D solid as the slicing plane.

24.44.1 Method





Specifies the entities that will be sliced. Choose one or more 3D solids or surfaces.

Note: This command does not work with 2D regions.

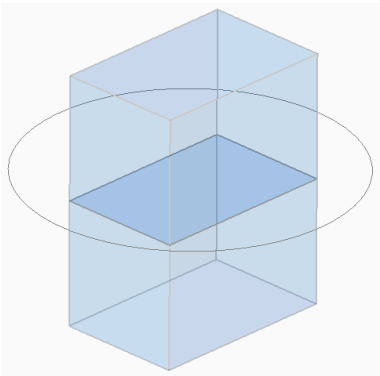
Specify the first (1) and the second (2) point on slicing plane and choose to keep only one of the parts (3) or both. The section plane is perpendicular to the XY-plane of the World coordinate system (WCS):

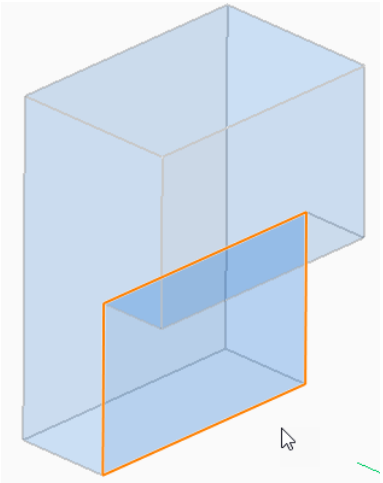
The sliced entity can be manipulated as independent entities.

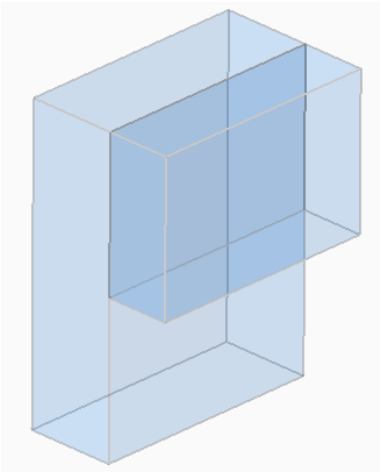
24.44.2 Options within the command

Object

Slices solids and surfaces with an intersecting entity (circle, ellipse, arc, 2D-spline, 2D-polyline, planar 3d Solid face, planar Surface face, or planar region). The entity defines the plane that slices the 3D selected entities.





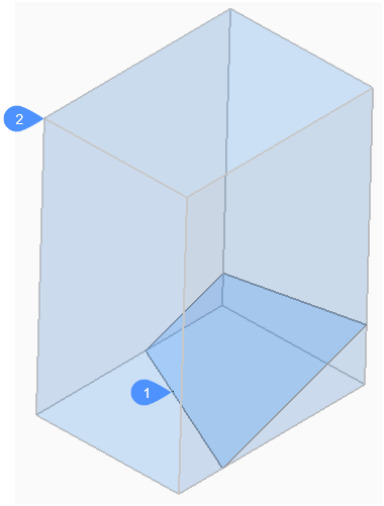


Surface

Slices solids and surfaces with a surface.

Zaxis

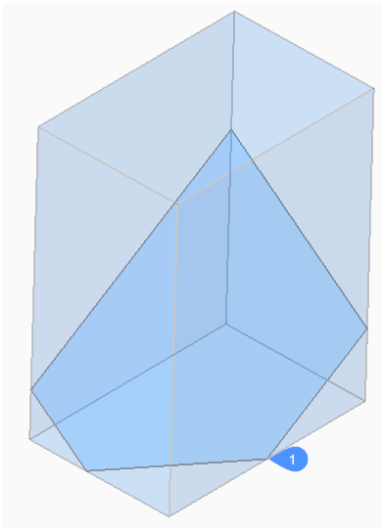
Slices solids and surfaces with a plane perpendicular with a line defined by one point on the plane (1), and a second point on the plane's normal axis (z axis) (2).



View

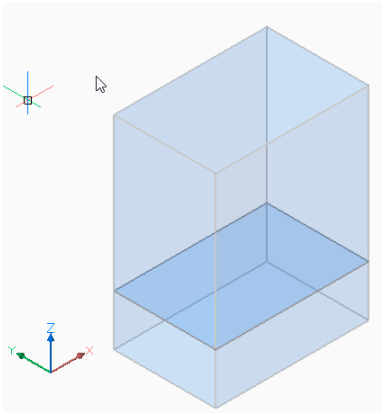
Slices solids and surfaces with a plane defined by the current 3D viewpoint.

The section plane is perpendicular to the view direction and passes through the picked point.



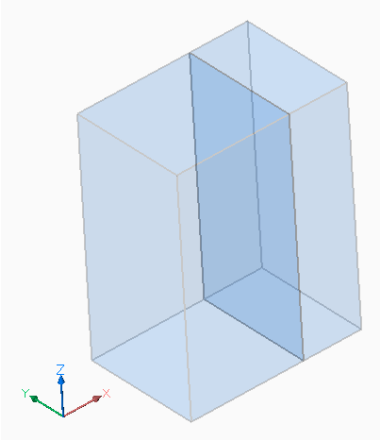
XY

Slices solids and surfaces with a plane parallel to the XY-plane of the current coordinate system.



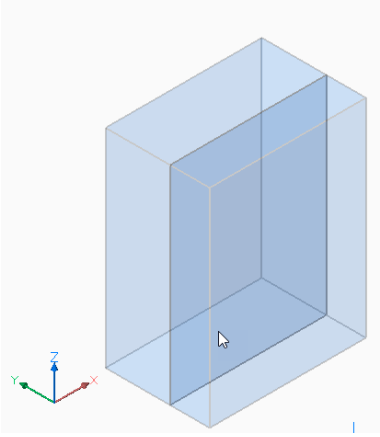
YZ

Slices solids and surfaces with a plane parallel to the YZ-plane of the current coordinate system.



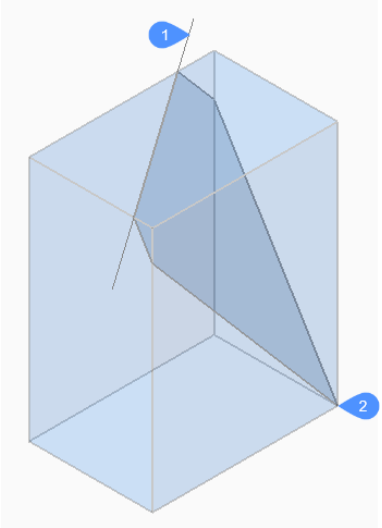
ZX

Slices solids and surfaces with a plane parallel to the ZX-plane of the current coordinate system.

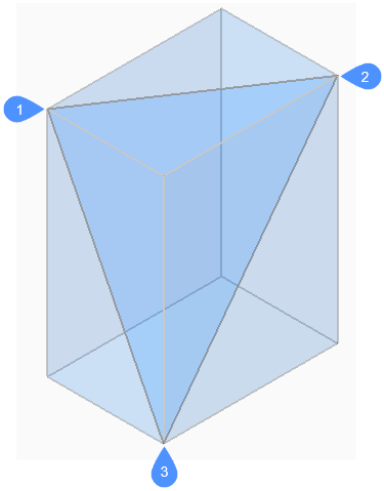


Line-point

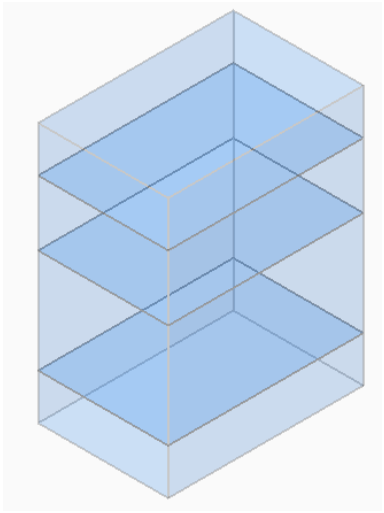
Slices solids and surfaces with a plane defined by a linear entity (1) and a point (2).



3points
Slices solids and surfaces with a plane defined by three points.



Multislice
Slices solids and surfaces in more than one piece using a face and specifying the distance between slices.



Both
Retains both parts.

24.45 SMASSEMBLYEXPORT command

Searches the assembly structure, of the current drawing, for 3D solids within a component and applies the SMCONVERT and SMEXPORT2D commands.

Shape Lite Pro Mechanical BIM

Icon:

24.45.1 Method

The result of this command is a set of .dxf files with unfold information saved in a user-defined output folder and sorted by thickness. Parts processed with command warning or errors are placed in a special folder. An HTML report listing all the solids in the assembly is generated. This indicates their status and links to the corresponding .dwg and .dxf files.

Note: SMASSEMBLYEXPORT can be applied to mixed assemblies containing both sheet metal and non-sheet metal parts. It quickly sorts out non-sheet metal parts from sheet metal parts.

The solid classification is as follows:

- Sheet Metal – solid is a sheet metal part
- Poor Sheet Metal – solid looks like a sheet metal design, user assistance and rework are needed
- Not Sheet Metal – solid is not recognized as a sheet metal design

Note: SMASSEMBLYEXPORT can be applied to simple drawings with solids, to assemblies created in BricsCAD and to drawings imported with the Communicator for BricsCAD.

Note: For optimal processing set IMPORTPRODUCTSTRUCTURE system variable to 2, otherwise mechanical components will be exploded to solids, increasing processing time.

Note: For optimal performance set the Visual Style to 2dWireframe.

24.45.2 Options within the command

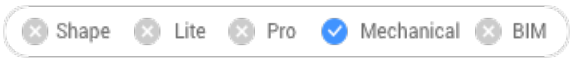
Output folder
Specifies the output folder.

bend Table
Assigns a bend table used during SMEXPORT2D call in the output process.

Dxf version
Specifies the DXF version.

24.46 SMBEAD command

Creates associative Rib form features on sheet metal.



Icon: 

24.46.1 Description

Creates associative Rib form features on sheet metal parts from 2D profiles. The feature is updated automatically when its defining profile is changed.

The difference with linear rib features inserted from the library byBMINSERTcommand is that rib features created by theSMBEADcommand can have an arbitrary trajectory.

24.46.2 Method

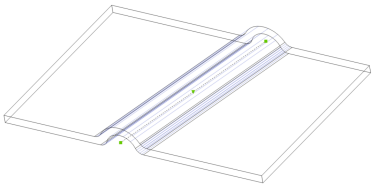
There are two methods to select a control curve along which the rib will be created:

- Specify a profile radius.
- Specify a round radius.

24.46.3 Options within the command

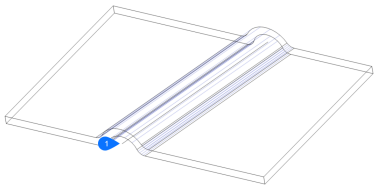
Select control curve

Selects a control curve along which the rib will be created. The selected curve can be a line, polyline, circle, arc, ellipse, elliptical arc or a spline. The 2D curve can be open or closed but it cannot intersect itself or other form features.



Profile radius

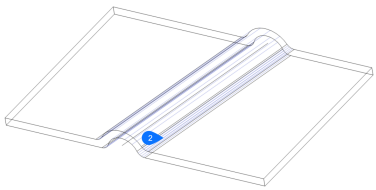
Sets the radius of the rib profile (1).



Note: The initial value of the Profile Radius is taken from the **Mechanical Browser for Sheet Metal** or from the **Bead feature** in **Settings** > **Initial Parameters**.

Round radius

Sets the radius of the smoothing fillets (2).



Note: The initial value of the Round Radius is taken from the **Mechanical Browser for Sheet Metal** or from the **Bead feature** in **Settings** > **Initial Parameters**.

24.47 SMBEND command

Converts hard edges (sharp edges between flange faces) or junctions into bends.

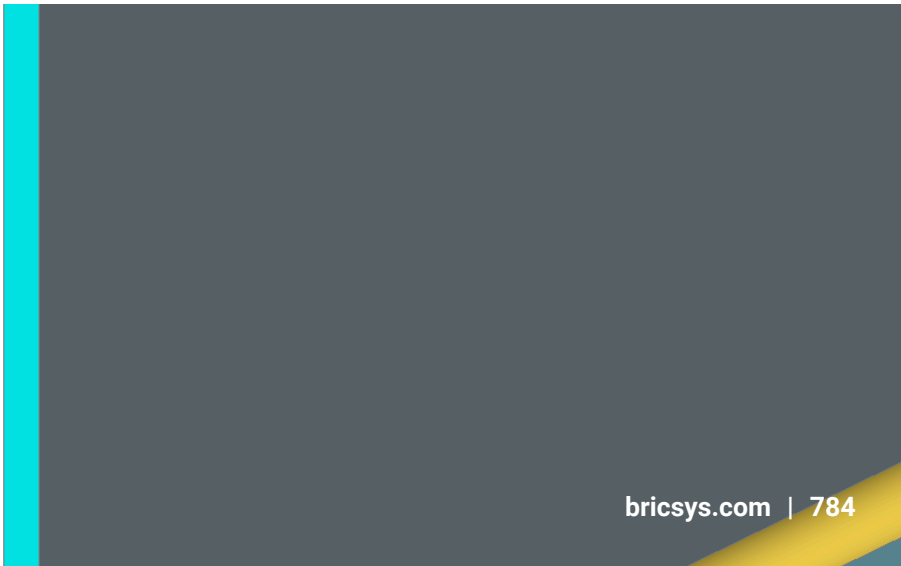
Shape Lite Pro Mechanical BIM

Icon:

24.47.1 Method

Select hard edges, junctions, flanges or 3D solids.





24.47.2 Options within the command

Entire Model
Detects and converts hard edges or junctions in the entire model.

24.48 SMBENDCREATE command

Converts hard edges (sharp edges between flange faces) or junctions into bends.

Shape Lite Pro Mechanical BIM

24.48.1 Description

This command is obsolete. Use the *SMBEND* command instead.

24.49 SMBENDSWITCH command

Converts bends to lofted bends.

Shape Lite Pro Mechanical BIM

Icon: 

24.49.1 Description

Allows selecting bend faces which will be transformed into lofted bends.

24.50 SMCONVERT command

Automatically recognizes sheet metal features.

Shape Lite Pro Mechanical BIM

Icon: 

24.50.1 Description

Automatically recognizes flanges, bends, lofted bends, ribs, form features and holes for selected 3D solids.

24.50.2 Options within the command

Entire Model
Applies the command to the entire model.

Note: All the solids in the model are analyzed and if possible, converted to Sheet Metal bodies. The command recognizes sheet metal features (such as flanges and bends) in the input geometry. The body thickness of the sheet metal drawing is the criteria in case multiple solids are selected. Therefore, selected bodies must have the same thickness, otherwise an error message displays in the command window.

24.51 SMDELETE command

Removes features from sheet metal elements.

Shape Lite Pro Mechanical BIM

Icon: 

24.51.1 Description

Removes a bend or a junction by restoring a hard edge between two flanges. This command can also remove a flange with all the bends adjacent to it and it can remove miter by restoring geometry being cut by the feature.

The adjacent flanges are extended up to a junction configuration with the flange being deleted.

24.51.2 Method

There are two methods to replace features by hard edges:

- Select feature faces.
- Select thickness faces.

24.52 SMDISSOLVE command

Removes the sheet metal data from the selected features.

Shape Lite Pro Mechanical BIM

Icon: 

24.52.1 Description

Dissolves sheet metal feature on 3D solid faces. Dissolved features are no longer listed in the Mechanical Browser. Faces of dissolved features can be moved, rotated or push-pulled freely.

24.52.2 Method

There are two methods to remove sheet metal data from features:

- Select faces on model.
- Remove from the entire drawing.

24.53 SMEXPLODE command

Converts bends, forms, hems, jogs, junctions and tabs into their possibly primitive features and removes the original feature.

Shape Lite Pro Mechanical BIM

Icon: 

24.53.1 Method

The feature is either decomposed into some primitives (i.e. a form feature into bends and flanges), or separated into several ones of the same type (i.e. a bend with hole into two bends):

- Bend containing several connectivity components into a list of bend features;
- Junction with several components into several junction;
- Form into flanges and bends;
- Jog into flanges and bends;
- Hem into flanges and bends;
- Tab into multiple junctions;

24.54 SMEXPORT2D command

Exports the unfolded representation of a sheet metal body.

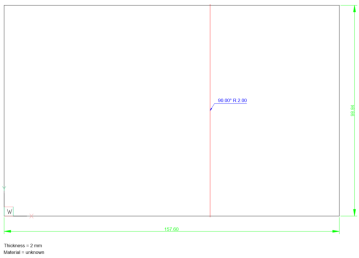
Shape Lite Pro Mechanical BIM

Icon: DXF

24.54.1 Description

Exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.

Displays the Save unfolded 2D entities dialog box. Here you can save the 2D profile file.



24.55 SMEXPORTOSM command

Exports a sheet metal solid to an Open Sheet Metal (.osm) file format.

Shape Lite Pro Mechanical BIM

Icon: OSM

24.55.1 Description

Displays the Save OSM file dialog box, which allows you to select a folder for the exported file.

24.55.2 Method

The .osm part is saved as mm- or inch-based, depending on the value of the document MEASUREMENT system variable. Document INSUNITS are considered for proper scaling.

Note: You can set the accuracy of the approximation and the minimal length of the document.

24.55.3 Options within the command

add Decorations
Allows you to export adjacent non-sheet metal parts in same .osm part.

24.56 SMEXTRUDE command

Creates sheet metal part by extruding a planar polyline.

Shape Lite Pro Mechanical BIM

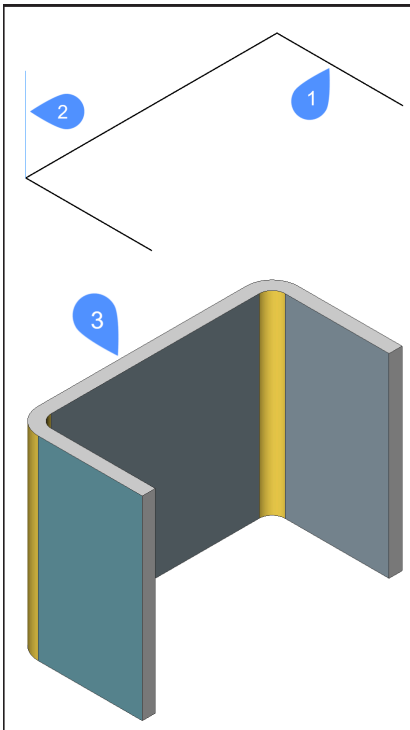
Icon:

24.56.1 Description

Creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.

24.56.2 Method

Select a planar polyline and specify the height of extrusion.



1 Planar polyline
2 Extrusion direction
3 Extruded polyline

The thickening direction of the extruded surface is applied for each flange separately, to preserve the overall dimensions of polyline.

The overall dimensions of the Sheet Metal part match the dimensions of the extruded polyline.

24.57 SMFLANGE command

Creates a base (initial) flange.

Shape Lite Pro Mechanical BIM

Icon: 

24.57.1 Description

Creates a base (initial) flange of a sheet metal part from a closed planar profile.

Note: A base flange is created for each selected closed polyline or region. In the **Mechanical Browser** a body and a flange are added for each entity.

24.57.2 Options within the command

Up
The flange is extruded on top of the base entity.

Middle
The flange is extruded on both sides of the base entity.

Down
The flange is extruded below the base entity.

Note: If the **Hot Key Assistant** is toggled ON, you can cycle the various option by pressing the Ctrl key.

Accept model
Accept the current extrusion.

24.58 SMFLANGEBASE command

Creates a base (initial) flange.

Shape Lite Pro Mechanical BIM

24.58.1 Description

This command is obsolete. Use the *SMFLANGE* command instead.

24.59 SMFLANGEBEND command

Allows to bend an existing flange along a line, obeying the k-factor for given bend radius.

Shape Lite Pro Mechanical BIM

Icon: 

24.59.1 Options within the command

New line
Allows to draw a line to define the bend location.

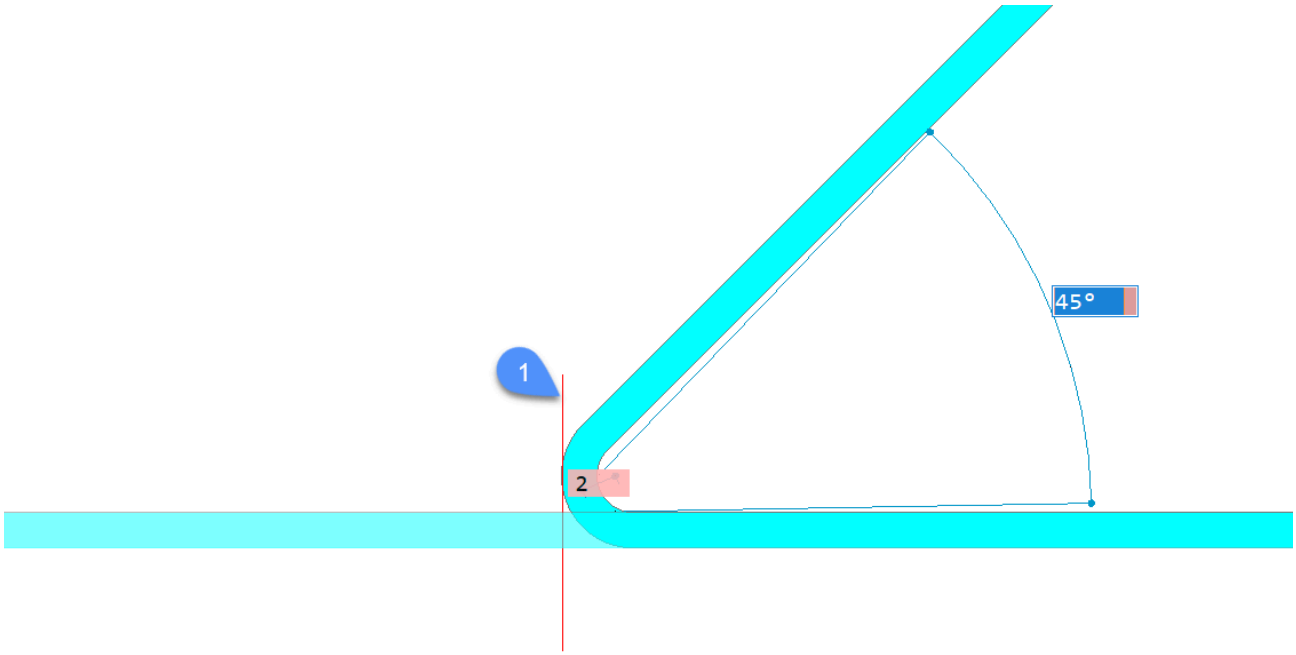
Angle
Specifies the bend angle.

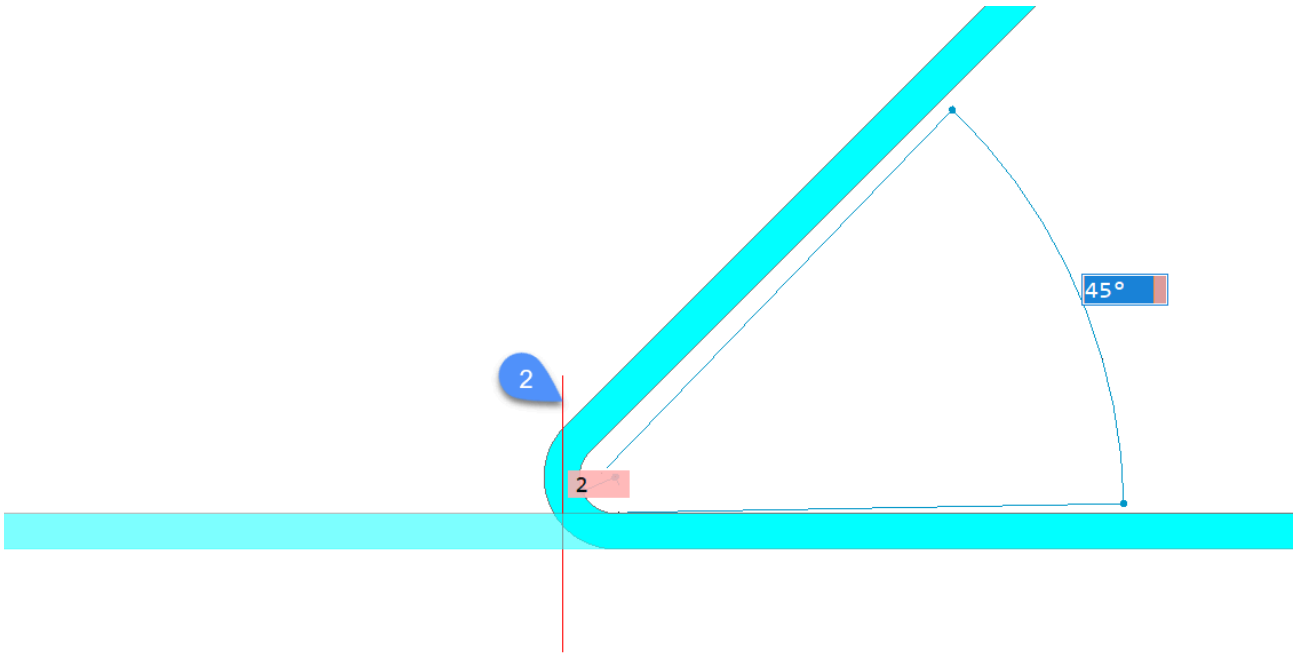
Radius
Specifies the bend radius.

Note: The default bend radius is defined by the Modeling/Bend Radius property in the Mechanical Browser for Sheet Metal.

Switch sides
Allows to move the part of the flange on the other side of the line or edge.

Toggle bend extents
Toggles between keeping the bend extents (1) or not (2). Default is not.


45°
1
2


45°
2
2

ACcept
Creates the bend at the current angle and radius.

24.60 SMFLANGECONNECT command

Shape Lite Pro Mechanical BIM

Icon: 

24.61 SMFLANGECONTOUR command

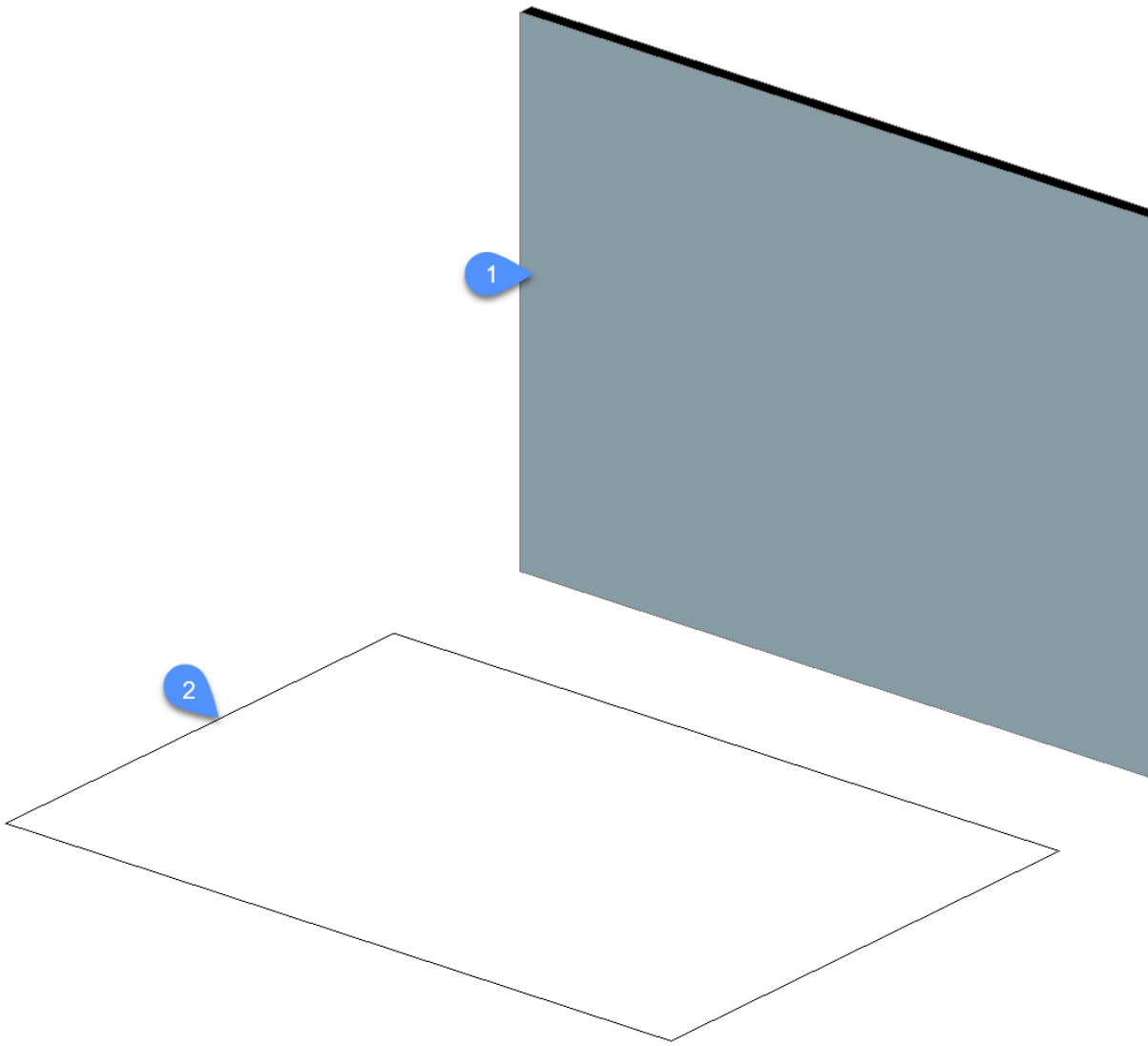
Creates a new flange from a closed contour and attaches it to an existing sheet metal part.

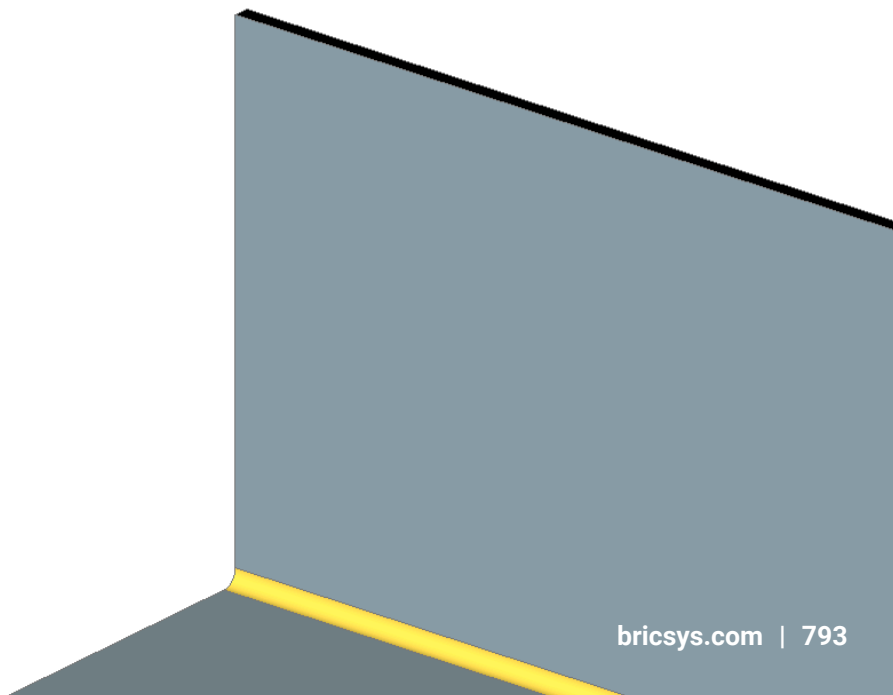
Shape Lite Pro Mechanical BIM

Icon: 

24.61.1 Description

Extrudes a flange from a contour and connects it with selected flanges via bends.


1
2



24.62 SMFLANGEEDGE command

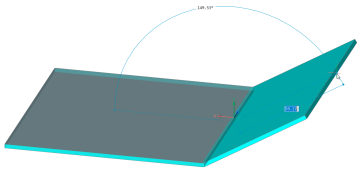
Creates flanges to a sheet metal part.

Shape Lite Pro Mechanical BIM

Icon:

24.62.1 Description

Creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange or lofted bend.



24.62.2 Options within the command

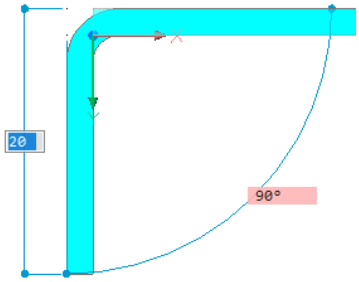
Dynamic dimensions

Creates a new flange using the dynamic dimensions of the height and angle of the new flange or by positioning the location.

Note: Use the Tab key to switch between the height and angle field.

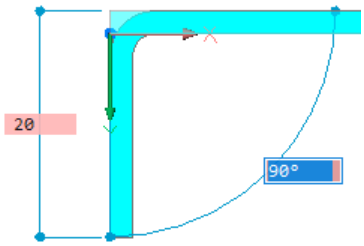
Material Outside

Creates a flange shifted by one material thickness to the outside.



material Inside

Creates a flange preserving the external dimensions of the parent flange.

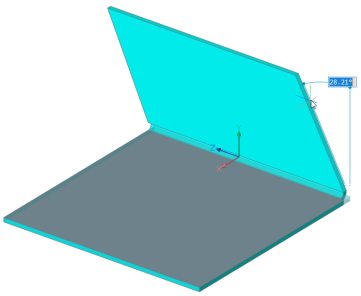


Angle
Allows you to first define the angle of the flange, then the height.

Length
Allows you to first define the length of the flange, then the angle.

Radius
Defines the radius of the bend connected to the new flange.

Taper Angle
Creates a flange with one or two tapered side faces.

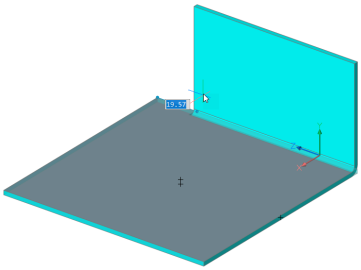


Note: This option is not available if multiple edges are selected.

Back
Goes back to the previous prompt.

Skip
Keeps the flange side orthogonal instead of tapering it.

Width
Allows to define a different width than the length of the selected edge. By default, the flange width equals the length of the selected edge.



Note: This option is not available if multiple edges are selected.

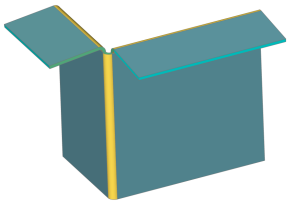
Back
Undoes the current width procedure.

Skip
Keeps the side edge of the flange at the end point of the edge.

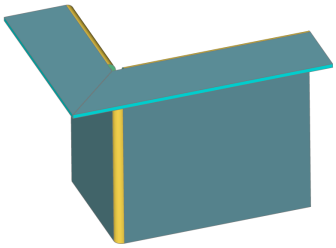
Toggle connection
Toggles between different connection options when creating several flanges simultaneously.

Note: The toggle connection is only available when at least two adjacent edges are selected.

No connection



With connection



Note: Use the Ctrl key to cycle through the options. Make sure that the Hot Key Assistant (HKA field in the Status bar) is ON.

Note: If TIPS = ON, the currently selected option is indicated in the Tips widget at the bottom of the screen.

set Direction
Defines the angle of the flange by specifying a direction to be parallel to.

2Points
Defines the direction by specifying two points.

Xaxis
Uses the X axis as direction.

Yaxis
Uses the Y axis as direction.

Zaxis
Uses the Z axis direction., if suitable.

Entity
Defines the direction by selecting an axial entity (e.g. linear edges or planar faces).

Last
Uses the last used direction.

View
Uses the Z direction of the view.

24.63 SMFLANGEROTATE command

Rotates a selected flange of a sheet metal part.

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24.63.1 Description

This command is obsolete. Use the *SMROTATE* command instead.

24.64 SMFLIP command

Swaps sides of a selected flange.

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Icon:

Swaps sides of a selected flange so that reference faces are on the other geometric side of the flange. Optionally shifts the flange over the thickness of the sheet metal part.

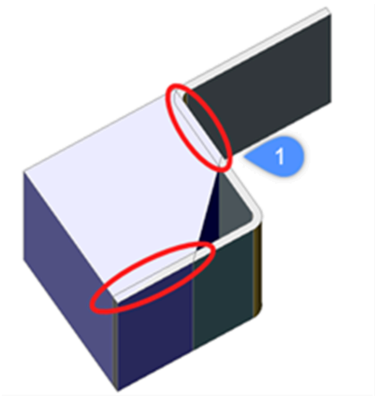
24.64.1 Method

Select a 3D solid or a flange face or press Enter to flip sides for all flanges.

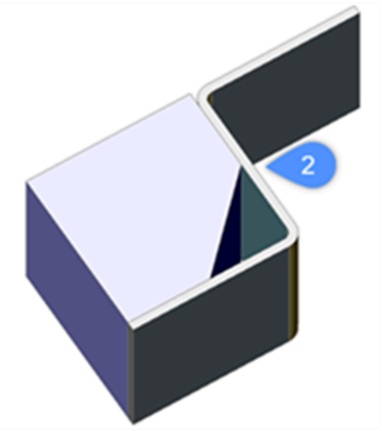
24.64.2 Options within the command

Flip reference side only
Turns OFF the shift of flanges by thickness, in this mode reference sides for selected flanges are just swapped.

Note: Changing reference faces with shift by thickness might be necessary if SMEXTRUDE command creates flanges which collide with another solid.



1. Collisions between a solid and sheet metal flanges.



2. Collisions solved after applying SMFLIP.

Entire Model
Applies flip operation to all flanges in the drawing.

24.65 SMFORM command

Converts a selected set of faces to a form feature.

Shape Lite Pro Mechanical BIM

Icon:

24.65.1 Method

This command converts a selected set of faces to a form feature or explodes form features into flanges and bends:

In Create mode, the command allows you to define a form feature by selecting the faces manually.

> ***Note:*** This can be necessary if automatic recognition by SMCONVERT does not produce the expected result.

For example: if you insert and dissolve the **Cardguide** form feature from the library, SMCONVERT recognizes two forms features because their geometry is separated. A manual selection of the form feature faces when executing SMFORM solves the problem.

> ***Note:*** Form features created by SMFORM have the same set of basic operations as if they were created by BMINSERT or recognized by SMCONVERT.

In Explode mode, the command dissolves the form feature and tries to recognize its geometry as flanges and bends. This can be helpful if SMCONVERT incorrectly recognizes the design as a form feature.

24.65.2 Options within the command

Create new form feature
Creates new form features.

Explode form feature
Explodes existing form features into flanges and bends.

Note: At least one face of each form feature to be exploded must be selected.

COnvert to form
Allows converting bead and bevel features in forms.

24.66 SMHEM command

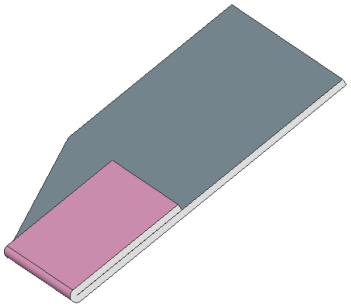
Creates one or more hem features on a sheet metal part by pulling one or more edges of an existing flange.

Shape Lite Pro Mechanical BIM

Icons:

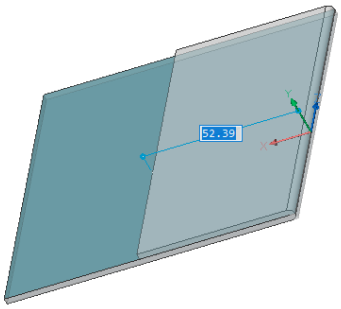
24.66.1 Method

By default, the command creates closed hem.



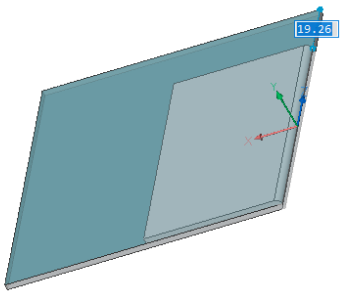
24.66.2 Option within the command

Length
Input the length of the hem.

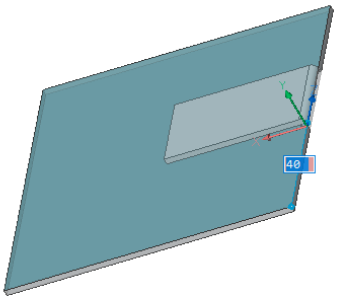


Width
By default, the hem width is equal to the length of the selected edge. The **Width** option defines a different width.

Note: The **Width** option is not available if multiple edges are selected.

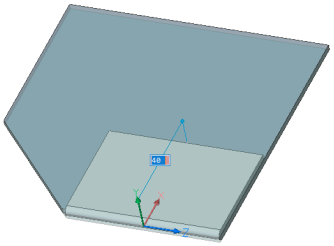


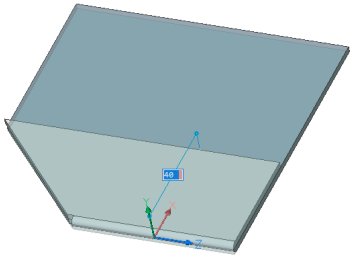
Skip
Keeps the distance value for the first side edge of the flange at the start point of the edge and switch the dynamic field to define the offset from the end point of the edge.



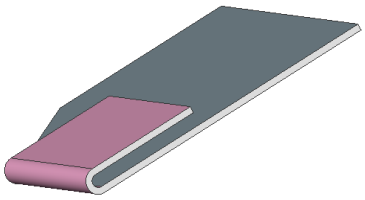
Back
Undoes the current **Width** procedure.

Toggle tapering
Toggle how a new hem fits the border of an existing flange. Two variants are possible:
The option only works if the edges of a flange at the corner are not at 90°.



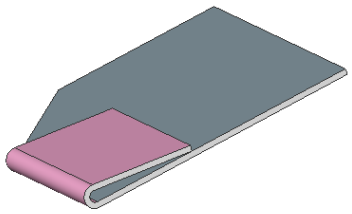


open Hem
Creates an open hem.

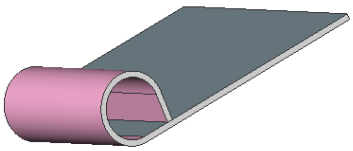


Radius
Defines the radius of the hem.

Teardrop
Creates a teardrop shape hem.



Round
Creates a round shape hem.



24.67 SMHEMCREATE command

Creates one or more hem features on a sheet metal part by pulling one or more edges of an existing flange.

Shape Lite Pro Mechanical BIM

24.67.1 Method

This command is obsolete. Use the *SMHEM* command instead.

24.68 SMIMPRINT command

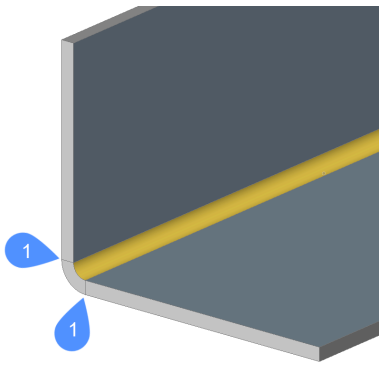
Splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.

Shape Lite Pro Mechanical BIM

Icon:

24.68.1 Description

Imprints edges on thickness faces to separate them into areas, which can be reworked automatically.

	1 Imprinted edges

24.69 SMJOG command

Creates a jog on a flange or on cylindrical lofted bends that avoid the collision with the other extent and allowing to do the smallest cuts.

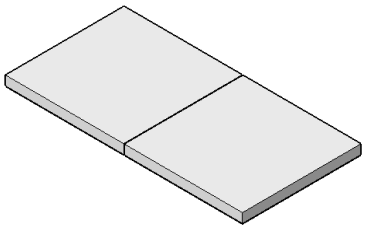
Shape Lite Pro Mechanical BIM

Icon:

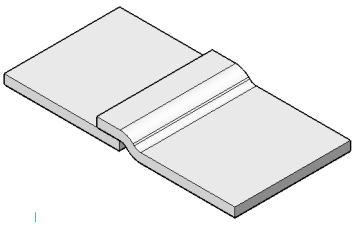
24.69.1 Method

Jog on a flange

Select an edge of a flange:



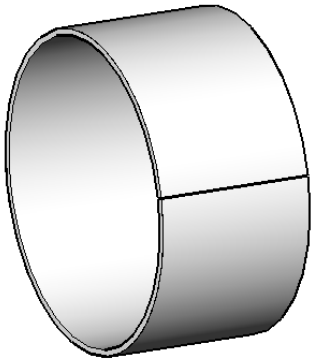
When Dynamic Input Mode (DYN) is active, type the length in the dynamic input fields.



Note: There is no collision on the other part.

Jog on cylindrical lofted bends

Select an edge of the cylindrical lofted bend.

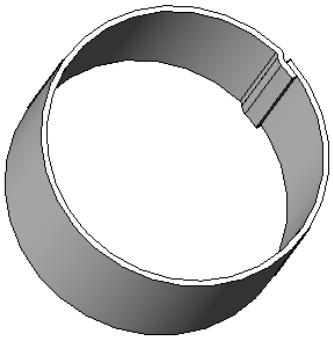


When Dynamic Input Mode (DYN) is active, type the length in the dynamic input fields.



Note: There is no collision on the other part.

Optionally, jog can be built on internal edges:



Note: If the FEATURECOLORS system variable is ON, the jogs will be displayed in the color set by SMCOLORJOG system variable.

24.69.2 Option within the command

Angle
Sets the jog angle; the default angle is 45 degrees.

Length
Sets the length of the jog. When Dynamic Input Mode (DYN) is active, type the length in the dynamic input fields.

Height
Sets the height of the jog.

Radius
Sets the radius for the jog.

Note: The starting point to measure the above dimensions is on the selected edge.

24.70 SMJUNCTION command

Creates a junction feature on hard edges (sharp edges between flange faces) and bends or curved hard edges between two bends (or lofts).

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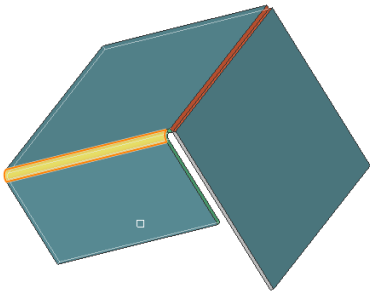
Icon:

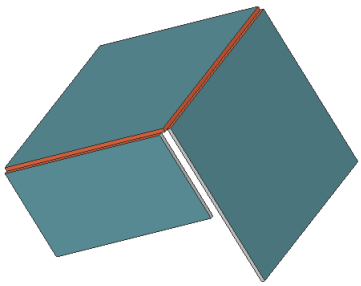
24.70.1 Description

The "Heal coincident junction" setting (*SMJUNCTIONCREATEHEALCOINCIDENT* system variable) controls how BricsCAD recognizes junction designs with coincident faces and how they are converted to regular junctions.

24.70.2 Method

Select flanges, hard edges, bends or 3D solids. You can also select "curved" edges between flanges and bends, between flanges and lofted bends, between two bends, or between two lofts.





24.70.3 Option within the command

Entire model
Analyzes the entire model to detect hard edges. A curved hard edge into a curved tabbed connection of a flange and a bend (or lofted bend).

24.71 SMJUNCTIONCREATE command

Creates a junction feature on hard edges (sharp edges between flange faces) and bends or curved hard edges between two bends (or lofts).

Shape Lite Pro Mechanical BIM

24.71.1 Description

This command is obsolete. Use the *SMJUNCTION* command instead.

24.72 SMJUNCTIONSWITCH command

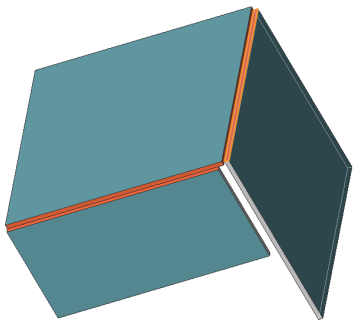
Allows changing a symmetrical junction feature to one with overlapping faces.

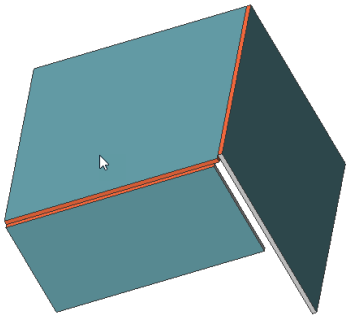
Shape Lite Pro Mechanical BIM

Icon:

24.72.1 Method

Select the junction face (left side picture) and see how the junction is transformed (right side picture).





24.73 SMLOFT command

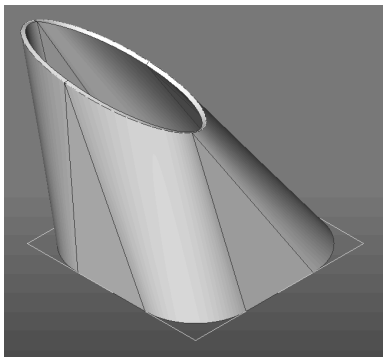
Creates sheet metal part.

Shape Lite Pro Mechanical BIM

Icon:

24.73.1 Description

Creates sheet metal part with lofted bends and flanges from two non-coplanar curves.



24.73.2 Options within the command

fillets Radius

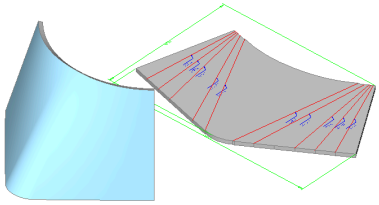
Sets radius of fillet which will be created on straight lines connections. A radius can be specified or the minimum radius can be calculated regarding the thickness of the model by pressing enter.

Thickness

Sets model thickness.

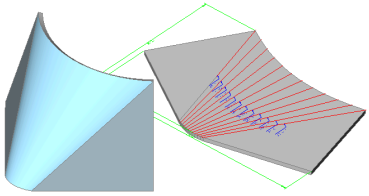
Single bend

Creates only lofted bend feature without any flanges.



Flanges with bends

Creates a flange feature for each straight segment of the selected 2D entities or a lofted bend feature for curved segments.



Alignment
After setting all options, the alignment of the metal sheet loft can be chosen.

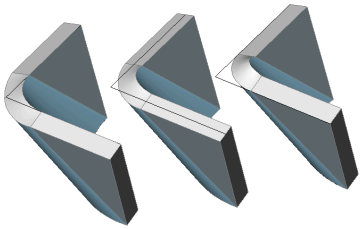
Note: Hit the CTRL key to adjust the alignment: Inside, Both Sides or Outside.



thicken Inside
Thickening direction is selected that way that 3D part is inside the selected 2D entities.

Both sides
Thickens at both sides of the selected 2D entities.

thicken Outside
Thickening direction is selected that way that 3D part is outside the selected 2D entities.



24.74 SMPARAMETRIZE command

Creates a consistent set of 3D constraint for a selected sheet metal part.

Shape Lite Pro Mechanical BIM

Icon:

24.74.1 Description

Creates a consistent set of 3D constraints for a selected sheet metal part and recognizes a rectangular array of holes on flanges. The command takes into account implicit constraints from sheet metal features and ensures that the constraint system is not overdefined.

For the best results, the sheet metal part should have a complete set of features including flanges, bends, junctions and bend & corner reliefs.

Constraints can be edited in the Mechanical Browser panel.

24.75 SMQUICK command

Suggests the optimal set of splits/junctions/bends for a shelled solid.

Shape Lite Pro Mechanical BIM

Icon:

24.75.1 Method

This command can be used on 3D Solids and on Sheet Metal solids.

If a 3D Solid is selected, this command can be used to:

- Create shelled solids.
- Create reliefs.
- Convert hard edges into junctions and bends.

If a Sheet Metal Solid is selected, this command recognizes its features and creates reliefs, junctions, and bends.

24.75.2 Options within the command

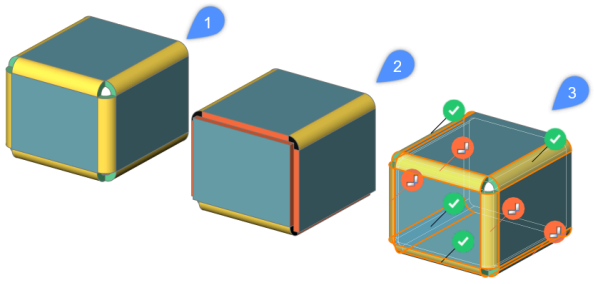
Enter shell offset distance
Specifies a distance.

Note: If a positive distance is entered, the body shells outwards. If a negative distance is entered, the body shells inwards.

Note: If the shell offset distance is larger than 3D solid, the command fails.

Skip
Skips the creation of shelled solid and acts similar to SMCONVERT command.

Select optimization mode
Select a mode to optimize the model.



Off (1)
Creates bends for all hard edges.

Auto (2)
Creates bends, junctions and split.

Interactive (3)
Gives suggestions for bend, junctions and split using a widget interface. Choose to apply the suggested configuration or change the suggestions.

24.76 SMRELIEF command

Creates proper corner and bend reliefs.

Shape | Lite | Pro | Mechanical | BIM

Icon:

24.76.1 Description

Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built at the start and end of a flange edge. Command supports the creation of reliefs on wrong bends.

The command automatically decides whether a relief must be created for a given bend. The size is either automatically chosen by default from the **Mechanical Browser for Sheet Metal** in the drawing or can be entered in the Command line.

In order to force the creation of bend relief for given bend, select the face of desired bend. In this case a bend will be created, even if the command decides a bend relief is not needed (i.e., it detects there are no clashes or stretches of the material).

24.76.2 Options within the command

Entire model
Creates reliefs on all hard edges, bends and corners in the entire model.

force Bend reliefs
Forces bend relief creation on bends.

Auto
The command automatically detects bend radii and adjusts the relief size accordingly.

24.77 SMRELIEFCREATE command

Creates proper corner and bend reliefs.

Shape Lite Pro Mechanical BIM

24.77.1 Description

This command is obsolete. Use the *SMRELIEF* command instead.

24.78 SMRELIEFSWITCH command

Converts corner reliefs.

Shape Lite Pro Mechanical BIM

Icons: 

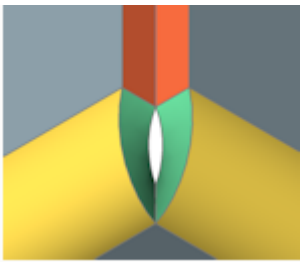
24.78.1 Description

Converts corner reliefs to circular, rectangular or V-shaped reliefs. Converts bend reliefs to smooth, round, rip or rectangular bend reliefs. This command also allows to change the parameters of existing corner reliefs.

24.78.2 Options within the command

V-type
Switches the selected corner reliefs to V-type.

Note: Only corner reliefs which are related to exactly two bends can be switched to v-type.

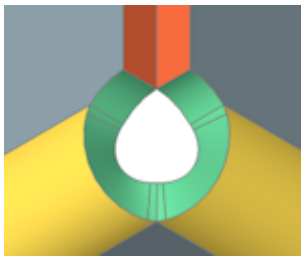


RECtangular
Switches the selected bend reliefs/ corner reliefs to rectangular.

Bend	Corner
	

CIrcular
Switches corner reliefs to circular.

Note: Only corner reliefs which are related to exactly two bends can be switched to circular reliefs.



relief Diameter
Specifies the relief diameter.

Auto
Uses the minimal diameter.

REference point
Specifies the reference point for the relief.

Middle
The reference point is at the intersection of the bend lines.

Corner
The reference point is at the circular arc through the base points of the corresponding rectangular reliefs.

from Global settings
Uses the value from global sheet metal settings of document.

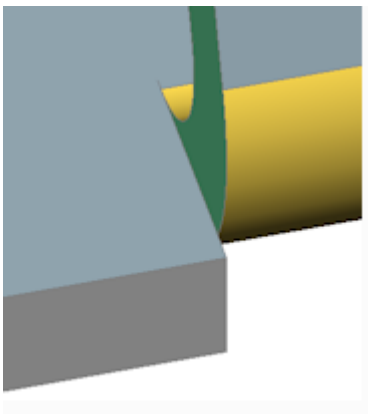
SMooth
Switches the selected bend reliefs to smooth.

Note: Smooth bend reliefs can be applied only if the current relief type is Rectangular. If not, BricsCAD displays the message: Unable to build relief geometry for this configuration.



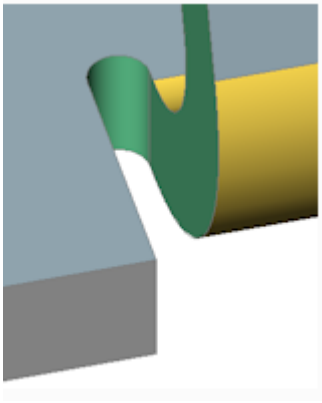
RIp
Switches the selected bend reliefs to rip.

Note: Rip bend reliefs can be applied only if the current relief type is Rectangular. If not, BricsCAD displays the message: Unable to build relief geometry for this configuration.



ROund
Switches the selected bend reliefs to round.

Note: Round bend reliefs can be applied only if the current relief type is Rectangular. If not, BricsCAD displays the message: Unable to build relief geometry for this configuration.



relief EXtension
Specifies the relief extension.

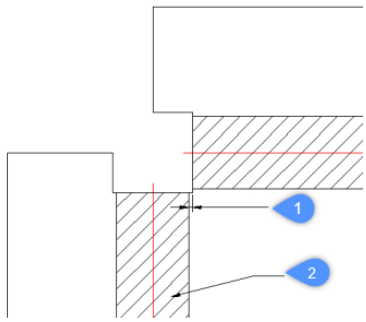
Auto
Keep the current extension.

Entire model
All corner or bend reliefs in all sheet metal parts in the model will be switched to the specified type using the specified parameters.

Note: If this option is chosen as the response to the initial prompt, all corner reliefs switch to V-type.

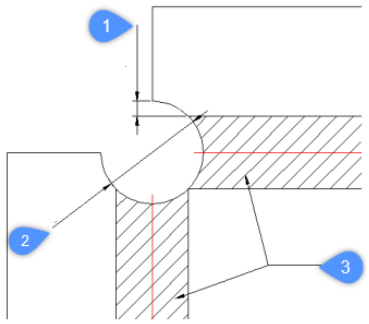
24.78.3 Relief extension illustration

Rectangular and V-type reliefs:

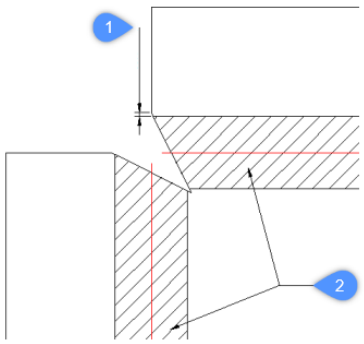


1 Extension
2 Bend

Circular reliefs:

	1 Minimal extension 2 Diameter of the relief in the unfolded representation ***Note:*** Reference to determine the center of the relief. 3 Bend

V-type reliefs:

	1 Extension 2 Bend

24.79 SMREPAIR command

SmRepair restores the 3D solid model of a sheet metal part.

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Icon:

24.79.1 Method

There are three methods:

- Repair by thickening one of the sides

  Note: SmRepair restores the 3D solid model of a sheet metal part by thickening one of its sides. All thickness faces become perpendicular to the flange faces.

- Repair of lofted bend features

  Note: Fixes problems specific to parts with lofted bend features. It merges adjacent bends and provides tangential connections with flanges.

- Repair by converting bends

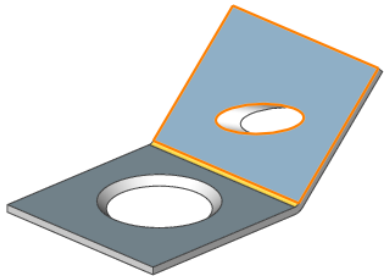
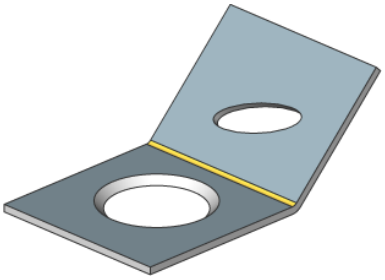
  Note: Converts incorrect bends to regular bends. If a single, incorrect bend is selected, it will be repaired. If a 3D solid is selected, then all incorrect bends will be converted to regular bends.

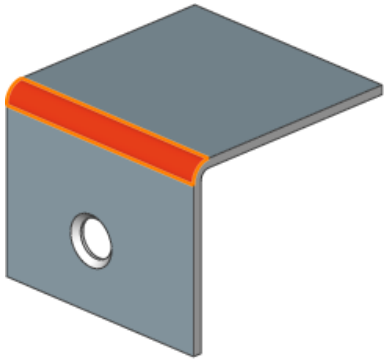
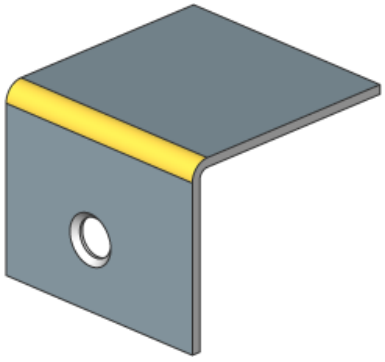
24.79.2 Options within the command

switch on Local repairing

Repairs one selected flange face, lofted bend face, thickness face or side edge. A body with thickness faces perpendicular to the selected face and without adjacent lofted bends will be created.

Note: The size and position of the opening on the selected face is preserved and the size and position of the opening on the opposite face is adjusted.

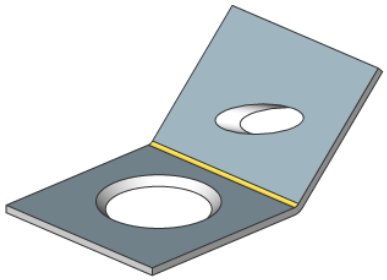
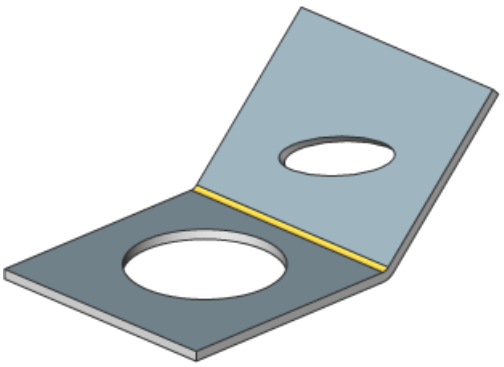
Before (flange face is selected)	After
	

Before (incorrect bend face is selected)	After
	

switch on Global repairing

Repairs all the incorrect flange faces, lofted bend faces, thickness faces or side edges from the selected solid. A body with thickness faces perpendicular to the selected face and without adjacent lofted bends will be created.

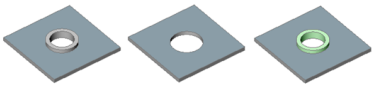
Note: The size and position of the opening on the selected face is preserved and the size and position of the opening on the opposite face is adjusted.

Before	After
	

Scan for lost details

Searches and retains model parts that remain as separated bodies after repair.

Note: The scan tolerance is a fraction of the initial model volume. This fraction will be ignored and only larger bodies will be retained. For example, if the scan tolerance = 0.25, only parts with a volume that is larger than 25% of the initial body will be retained.



From left to right:

- Initial model with an unrecognized form feature.
- Model after SmRepair without scanning for lost details: the circular feature is removed.
- Model after SmRepair with scanning for lost details. Two bodies are created: a horizontal flange with a hole and a separate body (highlighted in green color). The bodies can be unified with the Union command if required.

24.80 SMREPLACE command

Replaces form features in sheet metal parts.

Shape Lite Pro Mechanical BIM

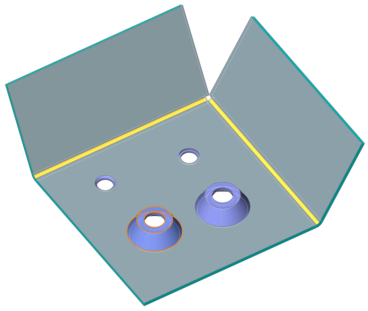
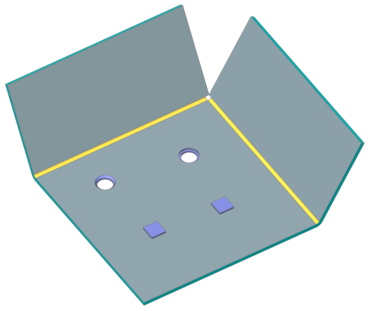
Icon: 

24.80.1 Description

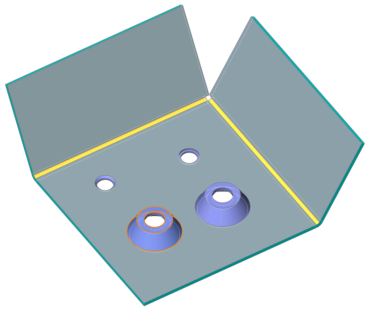
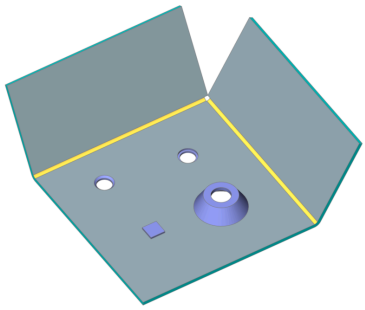
Replaces form features (including recognized ones) in sheet metal parts with form features from the built-in or user library. Displays the Select file for form feature dialog box where you can select a form feature file.

24.80.2 Options within the command

Yes
Replaces all form features of the same type as selected feature(s).

Before	After
	

No
Replaces the selected form feature(s) only.

Before	After
	

parameters Mode
Controls whether the parameter values of the form features that are replaced will be inherited or not. It is defined by the selected mode of replacement.

component Defaults
Parameter values of the new form feature are used.

Inherit from replaced features
If possible, parameter values are copied from the form feature(s) being replaced.

Edit parameters
Allows to edit all parameters.

24.81 SMRIBCREATE command

Creates associative Rib form features on sheet metal.

Shape Lite Pro Mechanical BIM

24.81.1 Description

This command is obsolete. Use the SMBEAD command instead.

24.82 SMROLLEDEDGE command

Creates rolled edges on curved flanges.

Shape Lite Pro Mechanical BIM

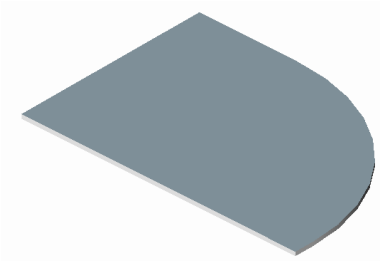
Icon:

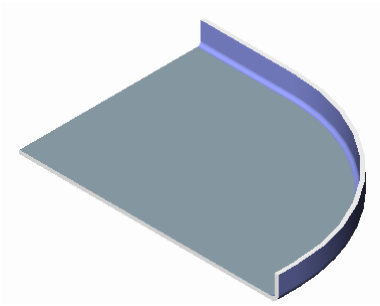
24.82.1 Method

Select the edges on flanges or the flange face (entire flange's side) and specify the length of the rolled edge.

Note: The RolledEdge is not created on edges adjacent to Bend, Jog, BendRelief, CornerRelief.

When Dynamic Input Mode (DYN) is active, type the length in the dynamic input fields.

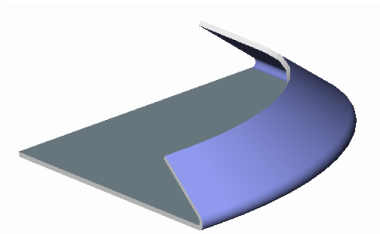




24.82.2 Option within the command

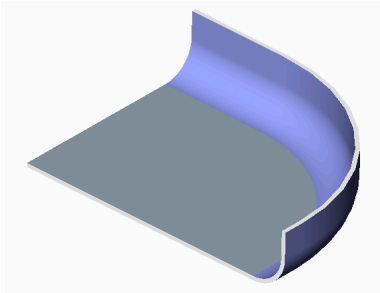
Angle

Specifies the angle of the rolled edge. The default value is 90 degrees.



Note: If the FEATURECOLORS system variable is ON, the rolled edges will be displayed in the color set by SMCOLORROLLEDEDGE system variable.

Radius
Specifies the radius of the rolled edge. The default value is the thickness of the flange.



24.83 SMROTATE command

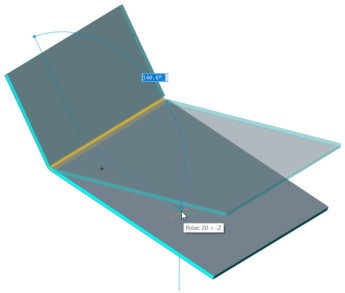
Rotates a selected flange of a sheet metal part.

Shape Lite Pro Mechanical BIM

Icon:

24.83.1 Description

Rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent.



Note: A dynamic field angle displays the angle between the selected flange and the base flange. You can press the TAB key to display the absolute rotation angle.

24.84 SMSELECT command

Selects all identical form features in the sheet metal model.

Shape Lite Pro Mechanical BIM

Icons:

24.84.1 Description

Selects hard edges, same form features, non-orthogonal thickness edges, flat edges, and the side of a sheet metal part.

24.84.2 Options within the command

Hard edges
Selects all hard edges in the model.

Same form features
Selects all form features identical to the selected feature.

SImilar form features
Selects all form features similar (the same type, but with different parameter values) to the selected feature.

Non-orthogonal thickness faces
Selects all edges of a specified flange common with non-orthogonal thickness faces.

SIDe of sheet metal part
Selects faces on the same side of a sheet metal part within a selected face.

Note: Requires flange, regular or lofted bend face to start.

Flat edges
Selects all flat edges on a specified solid.

24.85 SMSPLIT command

Splits a flange.

Shape Lite Pro Mechanical BIM

Icons: 

24.85.1 Description

Splits a flange along a polyline drawn along a face. It splits lofted bends along the line on the ruled surface. (Short for Sheet Metal Split).

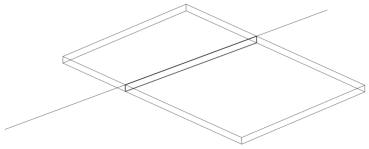
24.85.2 Method

There are three methods to split sheet metal:

- Split
- Smart Split
- Smart Split Propagate

24.85.3 Options within the command

Select lines or edges to split the flange
Splits the flange according to a selected line or edge.



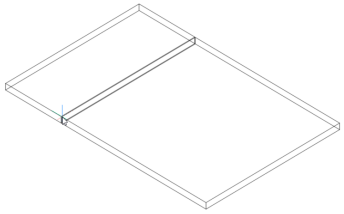
Center
Splits along the center of the polyline.

Left
Splits to the left side of the polyline.

Right
Splits to the right side of the polyline.

Accept
Accepts the suggested way to split.

SMart split
Splits the flange at a specified point.



Note: Make sure the 3D entity snaps are toggled on to be able to pick points on the flange.

Propagate
Automatically suggests multiple splits of a similar type.

 : Selected suggestion. Click to switch to not selected.

 : Not selected suggestion. Click to switch to selected.

draw a New line
Gives the option to draw a line on a selected flange to split the flange.

Note: Make sure the 3D entity snaps are toggled on to be able to pick points on the flange.

24.86 SMTAB command

Creates tab features on sheet metal parts.

Shape Lite Pro Mechanical BIM

Icons: 

24.86.1 Method

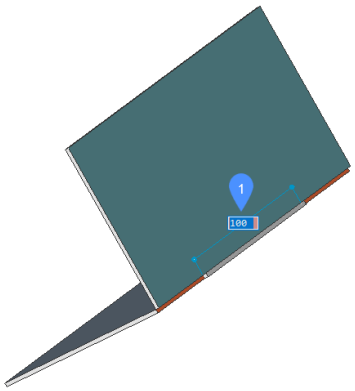
There are two main workflows:

- Convert a junction into tabbed junction (in either Single tab or Array tab mode).
- Convert a curved hard edge into a curved tabbed connection of a flange and a bend (or lofted bend).

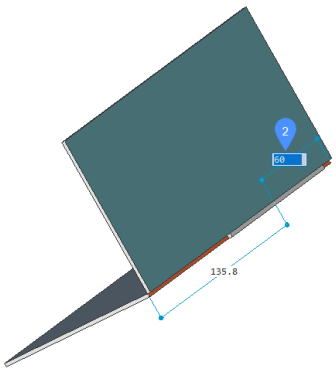
24.86.2 Option within the command

Single tab
Creates a single tab.

Length
Defines the length of the tab (1). The tab is placed symmetrically with respect to the center point.



Central point
Positions the tab along the junction. The dynamic dimension fields show the distance to the end points of the junction (2). Press the TAB key to switch between the dynamic fields.



Array tab
Creates multiple tabs.

Flip tab
Allows to toggle between two possible configurations when the tab feature is not symmetrical with respect to the junction sides.

slot Number
Defines the number of slots.

Distance
Defines the distance between the tabs.

Length
Defines the length of a tab.

Round radius
Creates a filleted tab with a given fillet radius.

CHamfer distance
Creates a chamfered tab with a given chamfer parameter.

24.87 SMTABCREATE command

Creates tab features on sheet metal parts.

Shape Lite Pro Mechanical BIM

24.87.1 Description

This command is obsolete. Use the *SMTAB* command instead.

24.88 SMUNFOLD command

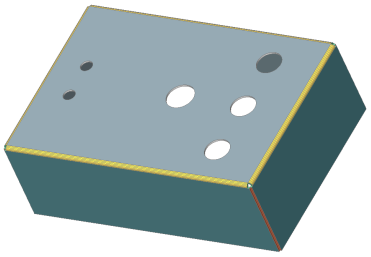
Creates an unfolded 2D or 3D representation of a sheet metal part.

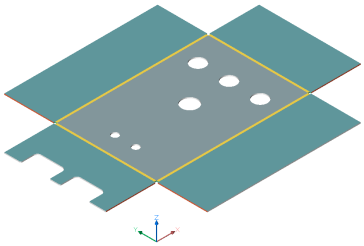
Shape Lite Pro Mechanical BIM

Icons: 

24.88.1 Method

Select the base flange of the sheet metal part, specify a point in the drawing to place the unfolded body and press Enter to keep the unfolded body in the drawing or choose an option.





24.88.2 Option within the command

Associative
Opens an additional window with model's unfolded representation. Both folded (left) and unfolded (right) representations have bidirectional associativity which allows transferring modifications to neighbor model automatically when changing window focus.

save 2D geometry
Saves the 2D geometry of the unfolded metal part as a separate drawing file.
From the Standard Save File dialog box displayed you have to choose to save the drawing either in Drawing Exchange Format (*.dxf) or Standard Drawing File (*.dwg).

save 3D geometry
Saves the 3D solid of the unfolded metal part as a separate drawing file.
From the Standard Save File dialog box displayed you have to choose to save the drawing either in Drawing Exchange Format (*.dxf) or Standard Drawing File (*.dwg).

Place view to layout
Places the view in a new or an existing layout.

Optimize bend annotations
Optimizes the bend annotations in the exported geometry.

Keep
Places the unfolded 3D solid in the current drawing.

24.89 SNAP command

Sets the cursor snapping properties.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SN

Sets the cursor snapping increments either by specifying the x and y values or between two points, the rotation and the style.

24.89.1 Method

There are three methods to set snapping increments:

- By specifying a single value for both the X and Y axis.
- By specifying a distance between two points as a value for both the X and Y axis.
- By specifying the spacing independently for the X and Y axis.

Note: Independent X and Y spacing work only in the Standard snap style.

24.89.2 Options within the command

snap ON/OFf
Turns on/off snap.

Note: Snap must be turned on in each viewport separately.

Note: The snap can be controlled by SNAPTYPE, POLARDIST, ADAPTIVEGRIDSTEPSIZE and SNAPSTYL system variables.

Rotate
Specify the base point and rotation angle for the snap grid.

snap Style
Toggle between Standard and Isometric snap distances.

Standard
Rectangular snap distance.

Aspect
Specifies the grid spacing independently for the horizontal(X) and vertical(Y) directions.

Isometric
Sets the grid, snap distance and crosshair to the current isometric plane.

24.90 SOLID command

Draws solid-filled polygons.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: PLANE, SO

Draws 3 and 4-sided solid-filled regular and irregular 2D polygons.

Note: Does not draw 3D solids.

24.90.1 Method

There are four methods to draw solid-filled polygons:

- By picking points.

  Note: After picking the first point, BricsCAD draws a ghost line to the next point. After specifying the second corner of the polygon, its shape and size can be previewed. If a fourth point is specified, an adjacent polygon is created (the third and fourth points of the first polygon become the first and second points of the second polygon).
- Rectangular.
- Square.
- Triangle.

24.90.2 Options within the command

Rectangle
Draws one or more connected solid-filled rectangles.

Square
Draws one or more connected solid-filled squares.

Triangle
Draws one or more connected equilateral solid-filled triangles.

Note: 2D solids can be edited directly through grips.

24.91 SOLIDEDIT command

Edits 3D solids and 2D regions.

Shape Lite Pro Mechanical BIM

Icons: 

24.91.1 Description

Edits the faces, edges, and bodies of 3D solids and 2D regions.

24.91.2 Options within the command

Face
Edit a face.

Extrude
Extrudes the selected face or region.

Specify height of extrusion
Specify a distance.

Note: Positive distance extrudes outwards from the face (creates a post). Negative distance extrudes into the body (creates a hole).

Path
Specifies the entity for defining the extrusion shape and direction.

Specify angle of taper for extrusion
Specify an angle.

Note: Positive angle tapers inwards (get narrower). Negative angle tapers outwards (gets wider).

Move
Moves the selected face.

Vector
Specifies the displacement through a vector measured from the origin (0,0,0) to the cursor's pick point.

Rotate
Slants the selected face about an axis.

Specify the rotation angle
Specify an angle.

Note: Positive angle rotates the face counterclockwise around the axis. Negative angle rotates the face clockwise.

Offset
Extends the selected face.

Specify offset distance
Specify a distance.

Note: When entering a positive distance, the face is offset outwards (3D body becomes larger). When entering a negative distance, the face is offset inwards (3D body becomes smaller).

Taper
Tapers (slants) the selected face.

Delete
Deletes the selected face and then repairs the model.

Note: The selected faces are deleted and BricsCAD repairs the 3D solid to keep it solid. This option cannot remove innate faces.

Copy
Copies faces. Selected faces turn into regions.

coLor
Changes the color of the selected face.

Undo
Undoes the last editing operation.

eXit
Returns to the previous prompt.

Edge
Edits an edge.

Copy
Copies the selected edge and turns it into a line.

Note: The copies edges are turned into entities such as lines and splines.

Body
Edits the body of a 3D solid.

Imprint
Imprints 2D onto 3D solids. The 2D entity must lie on, or intersect, one of the 3D solid's faces.

Note: The imprint can be extruded.

seParate
Separates disjointed 3D solids.

Note: This option does not work on bodies created through Boolean operations (Union, Intersect, Subtract).

Shell
Turns the 3D solid into a hollow entity (shelled).

Enter shell offset distance
Specify a distance.

Note: If a positive distance is entered, the body shells outwards. If a negative distance is entered, the body shells inwards.

Note: If the shell offset distance is larger than 3D solid, the command fails.

cLean
cleans the 3D solid.

Note: The cleaning operation checks for errors in the 3D solid.

Check
Reports on the 3D solid.

24.92 SOLPROF command

Creates hidden line representations of 3D solids in layout viewports.

Shape Lite Pro Mechanical BIM

Icon:

24.92.1 Method

Select one or more 3D solids and create hidden line representations of them. The representations are blocks.

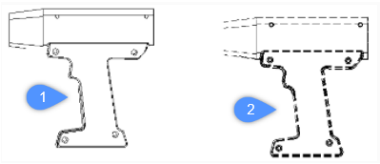
Note: This command operates only in model space of a layout tab viewport. Use the MSPACE command to activate the model space viewport.

Note: This command does not work with any other kind of 3D entity or with 3D solids referenced externally (xrefs). BricsCAD can filter out the ineligible entities.

24.92.2 Options within the command

Use separate layers for hidden lines?
Determines id hidden lines are placed on a separate layer from the visible lines.

Yes
Creates two 2D blocks, one for the visible lines placed on layer PV (1) and another block for hidden lines on layer PH (2).



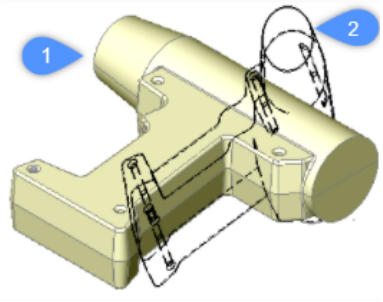
No
Creates one block of both hidden and visible lines on layer PV.

Note: The blocks and layers are named by the program, such as block *U3 and layers PH-88 and PV_88:

- *U identifies blocks created by the program
- PH is short for “profile hidden”
- PV is short for “profile visible”

Project result onto plan?
Determines if the profiles are projected (if the resulting blocks are 2D).

Yes
Profile lines are projected onto the drawing. The 3D solids (1) are projected onto a plane parallel with the layout's viewport, creating a block in 2D (2).



Note: In this image, the viewpoint has been rotated to show that the block(s) are indeed 2D.

No
No projection, the blocks are made of 3D entities.

Hide tangential edges?
Toggles the display of tangential lines.

Note: Tangential lines are used to show the transition between two faces that are curved. In a 3D shaded model, they are not needed as highlights and shadows indicate the transition. In 2D wireframe mode it can be useful to show them.

24.93 SPELL command

Opens the Spell checking dialog box.

Shape Lite Pro Mechanical BIM

Icon:

Alias: SP

24.93.1 Description

Opens the Spell checking dialog box to find and fix potential spelling errors in the current drawing.

24.94 SPHERE command

Creates a 3D Solid in the shape of a sphere.

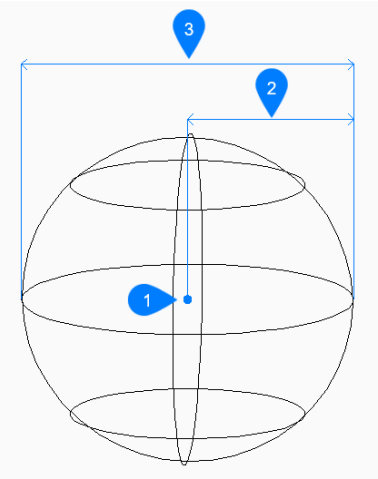
Shape Lite Pro Mechanical BIM

Icon:

Note: In BricsCAD Lite, which doesn't support 3D Solids, the SPHERE command launches the AI_SPHERE command.

24.94.1 Description

Creates a 3D Solid in the shape of a sphere. Choose from a combination of options including center, radius and diameter.



1 Center

2 Radius

3 Diameter

24.94.2 Methods to create a sphere

This command has 1 method to begin creating a sphere:

- Center of sphere

Center of sphere
Begin creating a sphere by specifying the center then:

Set radius of sphere
Specify the radius of the sphere.
Additional option: [Diameter]

24.94.3 Option within the SPHERE command

After you begin creating a sphere, the following option may be available:

Diameter
Specify the diameter of the sphere.

24.95 SPLINE command

Creates a spline.

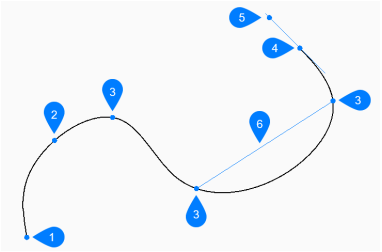
Shape Lite Pro Mechanical BIM

Icon:

Alias: SPL

24.95.1 Description

Creates a 2D or 3D spline that is open or closed. Options allow you to specify start and end points, fit points, and tangencies.



1 First point of spline (no tangency)

2 Second point

3 Next point

4 End point

5 Tangent point to end point

6 Distance between two drag lines

24.95.2 Methods to create a spline

This command has 1 method to begin creating a spline:

- First point for spline

You can continue adding unlimited points until you press Enter to end the command.

First point for spline
Begin creating a spline by specifying the start point then:

Second point
Specify the first fit point.

Set next point
Specify the next fit point.
Additional options: [Close/Fit tolerance/Undo]

Select starting tangent point
Specify a tangency for the starting point or press Enter to ignore this option.

Enter tangent for ending point
Specify a tangency for the ending point or press Enter to ignore this option.

24.95.3 Additional options within the SPLINE command

After you begin creating a spline, the following options may be available:

Close
Automatically draw a spline segment between the start and end points to create a closed spline.

Fit tolerance
Specify (in drawing units) how closely the spline matches its fit points. If the fit tolerance is 0, the spline passes through its fit points.

Undo
Undo the last fit point and continue drawing from the previous fit point.

24.96 SPLINEDIT command

Modifies the properties of a spline. Converts a spline to a polyline.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: SPE

24.96.1 Description

Allows the modification of the fit data (fit points, fit tolerance, tangents at the endpoints and knot parametrization), vertices, the polynomial degree of the spline and the weights of each control vertex. Also allows the joining of the spline with an open 2D entity and reversing the direction of the spline.

24.96.2 Options within the command

Close/Open
Opens or closes the spline by adding or removing a segment between the start and endpoint.

Join
Joins the spline with an open 2D entity that shares an (coincident) endpoint with the spline.

Fit Data
Modifies the fit point data.

Add
Adds a fit point between two existing ones.

Delete
Removes fit points.

Kink
Adds a knot and fit point at the specified location on the spline which does not maintain tangent or curvature continuity at that point.

Move
Moves a fit point.

Purge
Replaces the fit data from the spline with control vertices.

Tangents
Specifies the start and end tangents.

toLerance
Refits the selected spline to the existing fit points using a new tolerance value.

Edit vertex
Edits control frame data.

Add
Adds a new control vertex at the specified point.

Delete
Removes a control vertex.

Elevate order
Increases the polynomial order of the spline (degree plus one) to increase the number of control vertices across the spline. Maximum value is 26.

Move
Relocates control vertices one by one.

Weight
Changes the weight of a specified control vertex then recalculates the spline based on the new value. Larger weights pull the spline closer to the control vertex.

convert to Polyline
Converts the spline to a polyline. The precision determines how closely the resulting polyline matches the spline.

Note: The PLINECONVERTMODE variable specifies the kind of polyline to convert to (0 – with line segments, 1 – with arc segments).

Reverse
Reverses the direction of the spline; a triangle icon appears on what was formerly the spline's endpoint.

Undo
Undoes the previous modification.

eXit
Concludes the editing of the spline or returns to the previous prompt.

24.97 SPOTLIGHT command

Creates a spotlight.

Shape Lite Pro Mechanical BIM

Icon: 

24.97.1 Description

Creates a spotlight, which casts a narrow beam towards a target point.

Note: The options of this command are influenced by the value of the LIGHTUNITS system variable.

24.97.2 Options within the command

Specify source position
Specifies the location of the light in 3D space.

Specify target position
Specifies the location of the target.

Note: The target is the point at which the spotlight is focused.

Name
Specifies the name for the spotlight.

Intensity factor
Specifies the intensity of the light.

Note: Enter an intensity value between 0.00 and the maximum value supported by your system (max float = maximum value of a floating-point number).

Note: When the LIGHTINGUNITS system variable value is 0, this option is named Intensity.

Status
Toggles the use of light.

ON
Light is included when calculating rendering.

OFf
Light is excluded.

Photometry
Specifies the photometric properties of the light.

Note: This option is only available when the LIGHTUNITS value is 1 or 2.

Intensity
Enter the intensity expressed in candela (Cd).

Flux
Enter the luminous flux in lumens (Lm).

Illuminance
Enter the illuminance expressed in lux (Lx) or foot-candles (Fc).

Distance
Enter a distance expressed in drawing units.

Color
Enter a color name.

?
Lists the names of available colors.

Kelvin
Enter a Kelvin temperature expressed in degrees Kelvin.

Hotspot
Specifies the angle of the hotspot, the area of light concentration.

Note: The Hotspot angle represents the angle of the central light cone.

Note: The angle determines the size of the hotspot, regardless the distance of target.

Falloff
Specifies the angle of weaker illuminations surrounding the hotspot.

Note: The Falloff angle represents the angle of the full light cone.

Note: Outside of the falloff, there is no light.

Note: When a Falloff angle is entered that is smaller than the Hotspot angle, the latter is set equal to the Falloff angle. When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot) that is surrounded by an area of lesser intensity (falloff).

Note: 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 the light cone is rather sharp. The greater the difference between both angles, the softer the edge of the light cone.

shadoW
Specifies the look of shadows cast by this light.

Off
Disables the calculation of shadows for the light.

Sharp
Displays shadows with sharp edges.

Note: Use this option to increase performance.

soFt mapped
Displays realistic shadows with soft edges.

soft sAmpled
Displays realistic shadows with softer shadows based on extended light sources.

Shape

Disk
Defines a circular shadow shape that behaves as an area light.

Rect
Defines a rectangular shadow shape that behaves as an area light.

sAmples
Specifies the sampling size of the shadow.

Note: Larger numbers are more accurate but take longer to render.

Visible
Casts a shadow that represents shapes (more accurate) or that is rectangular (faster to render).

Attenuation
Specifies how the illumination falls off with distance from the light.

attenuation Type
Specifies the type of attenuation.

None
No attenuation so that the distance to the light source has no influence.

Inverse linear
Attenuation is the inverse of the linear distance from the light.

Note: 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 Squared
Attenuation is the inverse of the square of the distance from the light.

Note: 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
Toggles whether the extent of the illumination is limited.

attenuation start Limit
Defines the point where the light starts to shine, measured from the center of the light.

attenuation End limit
Defines the point where the light stops to shine, measured from the center of the light.

filterColor
Specifies the color of the light.

Note: When the LIGHTUNITS system variable's value is 0, this option is named Color.

Enter true color
The RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values).

Index color
Defines the color name by the index color.

Hsl
Specifies the color using three parameters Hue, Saturation and Lightness.

Colorbook
Enter a name from a colorbook.

eXit
Concludes the command and creates the light.

24.98 STANDARDPARTSPANELCLOSE command

This command is obsolete.

Shape Lite Pro Mechanical BIM

24.98.1 Description

Use LIBRARYPANELCLOSE instead.

24.99 STANDARDPARTSPANELOPEN command

This command is obsolete.

Shape Lite Pro Mechanical BIM

24.99.1 Description

Use LIBRARYPANELOPEN instead.

24.100 STANDARDS command

Allows to add and edit CAD standards for the current drawing.

Shape Lite Pro Mechanical BIM

Icon: 

24.100.1 Methods

This command associates one or more standard (DWS) files to the current drawing.

A standards file defines common properties for many drawing files to maintain consistency throughout those files. This consistency might be important in collaborative environments, where many people are involved in creating sets of drawings that should comply with some predefined rules.

Standards are created for the following named objects:

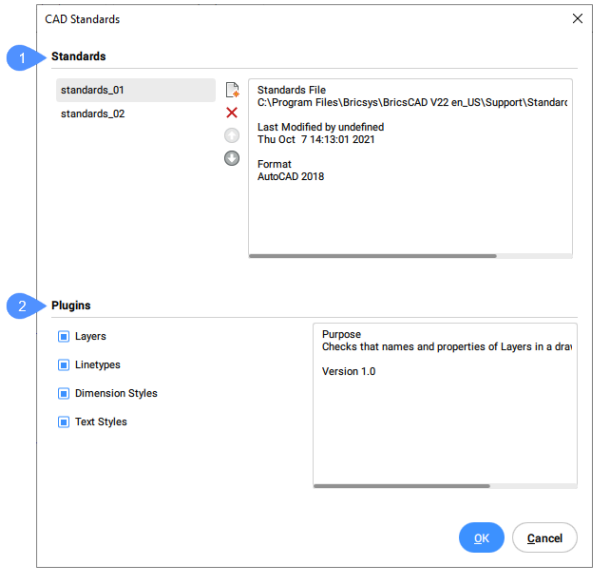
- Layers
- Text styles
- Line types
- Dimension styles

Standard (DWS) files can be created by the SAVEAS command, when you choose Standards File (*.dws) from the Save as type list.

This command has an associated dialog box that manages all the operations associated with this command.

24.100.2 STANDARDS dialog box

The STANDARDS dialog box allows to select the options for the STANDARDS command.



1 Standards

2 Plugins

Standards

This section is split into two sides.

The left side contains a list with the names of the standards already loaded into the drawing and a set of buttons for managing the list.

The names of the standards are the same as the names of the files selected as standards.

The buttons for managing the list are from top to bottom:

- **Adds Standards file**: adds the selected files to the list of standards.
- **Removes Standards file**: removes the selected file from the list of standards.
- **Moves Standards file one position up**: moves the name selected in the list one position up.
- **Moves Standards file one position down**: moves the name selected in the list one position down.

The right side of the **Standards** section displays some properties of the standard selected in the list. These properties include the path of the file containing the standard, the date of creation and the file format.

Standard (DWS) files can be created by the SAVEAS command, when you choose Standards File (*.dws) from the Save as type list.

Plug-ins

This section contains a list of plug-ins to be used by a checking operation.

A plug-in is a property or a set of properties of the entities in the standards files. When the corresponding checkbox of a plug-in is checked, the checking operation will include that plug-in.

The available plug-ins are as the following:

Layers

The layers defined in the standard files.

Linetypes

The line types defined in the standard files.

Dimension Styles
The dimension styles defined in the standard files.

Text Styles
The text styles defined in the standard files.

24.101 START command

24.102 STATBAR command

Toggles the STATUSBAR system variable.

Shape Lite Pro Mechanical BIM

24.102.1 Description

Toggles the STATUSBAR system variable to show or hide the status bar. You can launch this command at the Command prompt or within another command by preceding it with an apostrophe: 'STATBAR.

- On: turns on the STATUSBAR system variable.
- Off: turns off the STATUSBAR system variable.
- Toggle: changes the STATUSBAR system variable to the opposite of the current setting.

24.103 STATUS command

Reports the status of the drawing.

Shape Lite Pro Mechanical BIM

Icon: 

24.103.1 Description

Displays the status report of the drawing in the Command line.

Note: To see the full report press F2 key.

24.104 STLOUT command

Export 3D solids and watertight meshes.

Shape Lite Pro Mechanical BIM

24.104.1 Description

Exports 3D solids and watertight meshes in a format that can be used for stereolithography (3D printing).

24.104.2 Method

There are two methods to use STLOUT command:

- Export 3D solids.
- Export meshes.

24.104.3 Options within the command

Create binary STL file
Determines whether the output file is in binary or ASCII format.

Note: Binary STL file are more compact, while an ASCII text STL file is human-readable.

Select smoothness of object exported to STL file
Adjusts the quality of the exported entities.

Note: When exporting 3D solids to STL, curved faces become faceted. Use the FACETRES variable to determine the size of the facets:

- Minimum: 0.01 for coarser curves but smaller file size.
- Default: 0.5.
- Maximum: 10 for very fine curves but larger file size.

Low
Exports with FACETRES = 0.5.

Medium
Exports with FACETRES = 2.0.

High
Exports with FACETRES = 10.

Default
The modeler properties from Settings are used.

Note: The modeler properties override FACETRES to control smoothness by using different parameters.

24.105 STOPSCRIPT command

Stops the recording of scripts.

Shape Lite Pro Mechanical BIM

Icon: 

24.105.1 Description

Stops the recording of scripts begun by theRECSCRIPTcommand.

24.106 STRETCH command

Stretches 2D entities.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: S

Note: This command does not stretch texts, shapes, and 3D solids.

24.106.1 Method

Select an area in your drawing using either a rectangular window or a polygon, then specify a base point and a displacement point. All points and nodes inside the selected area will be moved over the specified distance.

24.107 STRUCTUREPANEL command

Opens the Structure panel.

Shape Lite Pro Mechanical BIM

24.107.1 Description

Opens the Structure panel to display it in the current workspace. The Structure panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Structure panel can be either floating, docked or stacked.

24.108 STRUCTUREPANELCLOSE command

Closes the Structure panel.

Shape Lite Pro Mechanical BIM

24.108.1 Description

Closes the Structure panel to hide it from the current workspace. If the Structure panel is stacked when you close it, the Structure tab or icon is removed from the stack.

24.109 STYLE command

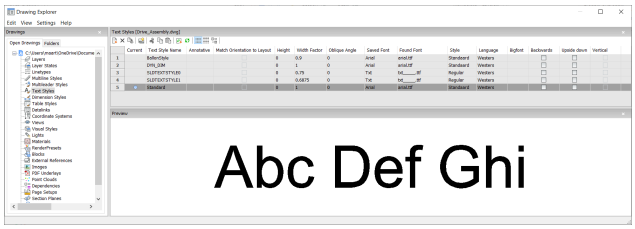
Opens the Drawing explorer dialog box with **Text Styles** selected.

Shape Lite Pro Mechanical BIM

Alias: DDSTYLE, EXPFONTS, EXPSTYLE, EXPSTYLES, ST

24.109.1 Description

Opens theDrawing explorer dialog boxwithText Stylescategory selected to view and modifystylesinthe current drawing.



24.109.2 Options within the command

Current

Designates the current text style. This is the style used by default by the TEXT and MTEXT commands.

Text Style Name
Specifies the style name.

Annotative
Sets the Annotative property of the text entities created with this style.

Note: When the text style is an annotative one, the Height property specifies the height in paper space, while in model space the height depends on the current annotation scale (specified by the CANNOSCALE and CANNOSCALEVALUE variables).

Match Orientation to Layout
Matches the orientation of the text to the orientation of the paperspace layout.

Note: This option is for annotative text only.

Height
Specifies the height of the text. Enter 0 or a number larger than 0:

- 0: does not set the height; TEXT command prompts for height.
- Not 0: specifies the height of text; TEXT command does not ask for height.

Width Factor
Specifies the width factor of the text. For example, entering 2 makes the text twice as wide.
Enter a number larger or smaller than 1:

- Less than 1: text is squeezed narrower.
- More than 1: text is spread wider.

Oblique Angle
Specifies the obliquing angle for the text. For example, entering 15 slants the text forward by 15 degrees.
Enter a positive or negative number:

- Negative number: text slants to the left.
- 0: text is upright.
- Positive number: text slants to the right.

Saved Font
Specifies the SHX or TTF font file to use; choose a name from the drop-down list:

- SHX: compiled shape files created for AutoCAD and BricsCAD
- TTF: TrueType font files created for Windows and Macintosh

Note: The program can use any SHX and TTF font found on your computer.

Found Font
Shows the name of the font file found. If the font defined in by Saved Font is not found .shx fonts are replaced by the font defined by the FONTALT (Alternative font) system variable.

Note: TTF fonts are replaced according to the replacement table defined by the FONTMAP system variable. By default, this is the Default.fmp file stored in the Support subfolder of the roamable root folder (e.g. *C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V21x64\en_US\Support*).

Note: Fonts that have been replaced display in Red.

Style
Specifies style for TTF fonts; choose a style from the drop-down list:

- Regular: applies neither bold nor italics
- Bold: applies boldface
- Italic: applies italics
- Bold Italic: applies bold and italics

Note: This property is not available for SHX fonts.

Language
Specifies the language for TTF fonts. Choose one from the drop-down list.

Note: Available languages may vary for each font. This property affects accents and other attributes specific to languages.

Note: This property is not available for SHX fonts. Use Bigfont property instead.

Bigfont
Lists the names of SHX bigfont files. Choose one from the drop-down list.

Note: "Bigfonts" is the name given to large SHX files that contain characters for multiple languages.

Note: This property is not available for TTF fonts. Use Language property instead.

Backwards
Toggles backwards text. Backwards text is useful when text is plotted on the back of mylar or other clear media.

Upside down
Toggles upside-down text.

Vertical
Toggles vertical text. Only some SHX fonts support vertical text; this option is not available for TTF fonts.

Preview
Shows what the current style looks like: the font with the properties applied.

24.109.3 Context Menu Options

New
Loads additional Text Style definitions into the drawing.

Delete
Deletes Text Style definitions from the drawing. The current style and the Standard style cannot be deleted.

Rename
Renames the selected style.

Select All
Selects all styles.

Invert selection
Deselects the current selection and vice versa.

Set current
Specifies the current working style. Only one style can be current at a time.

24.110 -STYLE command

Creates and modifies text styles.

Shape Lite Pro Mechanical BIM

Alias: FONT

24.110.1 Description

Creates and modifies text styles that set the look of text, such as its font and height. It is meant for macros. Styles determine the look of text in single-line text, mtext, attributes, tables, and dimensions and leaders, but not in dimensional constraints.

24.110.2 Options within the command

?
Lists existing styles and some of their properties including style name, height, generation.

Text style to create or modify
Specifies the style name.

Font type
SHX - font files created for DWG-based programs like BricsCAD.
TTF - font files created for Windows and macOS, and can be used by BricsCAD.

Height
Indicate the height of the text, measured from the baseline to the top of the ascenders.

Width
Makes characters wider or narrower.

Obliquing angle
Slants text to the left for a negative number, to the right for a positive number and keep text upright for value 0.

Draw text backwards
Toggles backwards text.

Draw upside down draw
Toggles upside-down text.

Draw text vertically
Toggles vertical text. This option is available only with certain SHX font files and not at all with TTF font file.

Annotative
An annotatively-scaled style adjusts its height in paper space automatically, according to the plot scale.
You can toggle the creation of an annotative text style, match the text orientation to layout.
When the text style is an annotative one, the text height property specifies the height in paper space, while the annotation scale sets the height in model space.

24.111 STYLESMANAGER command

Creates and edits plot style files.

Shape Lite Pro Mechanical BIM

24.111.1 Method

This command displays a file manager dialog box from which a CTB or STB file can be selected. Double click on a file to display the Plot Style Table Editor dialog box or on Create Plot Style Table to create a plot style from scratch or from an existing table.

Note: Plot styles fine-tune how drawings are printed but are not necessary for printing. BricsCAD supports two groups of plot styles to control the printed output:

- CTB color tables simply use colors.
- STB style tables use an extensive range of properties.

24.112 SUBTRACT command

Performs Boolean subtraction operation.

Shape Lite Pro Mechanical BIM

Icon:

Alias: SU

Note: In the BricsCAD® Lite license level, the command applies to region entities only.

24.112.1 Description

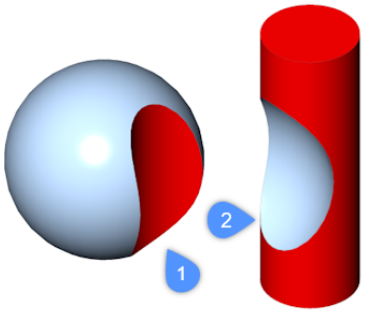
Performs Boolean subtraction operations on 3D solids and 2D regions by removing the overlapping portions of one from another.

24.112.2 Method

Select the object to subtract from and the objects to subtract. The DELETETOOL system variable determines whether the source entities are deleted or retained.

Note: The result of this command is sensitive to the order in which you select entities.

- A cylinder was subtracted from a sphere (1) and vice versa (2).



- The order in which you select entities also applies to regions.

24.113 SUNPROPERTIES command

Opens the Drawing explorer dialog box with **Lights** selected.

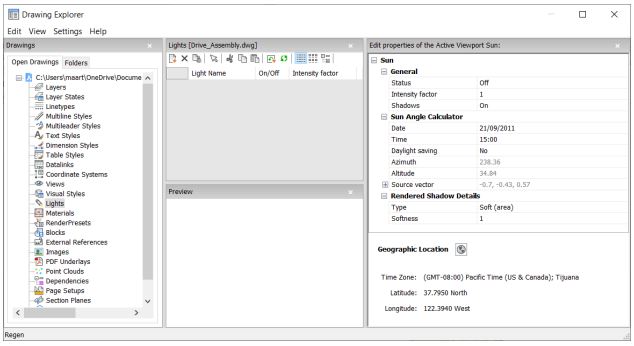
Shape Lite Pro Mechanical BIM

Icon:

Alias: SUN

24.113.1 Description

Opens the with **Lights** categoryselected to view and modify sun propertiesinthe current drawing.



24.113.2 Options within the command

General

Status

Toggles the sun on and off.

Intensity factor

Specifies the intensity or brightness of the sun; the higher the number, the brighter the light.

Shadows

Toggles the display of shadows by the sunlight.

Note: The color of the sunlight is hard-coded.

Sun Angle Calculator

Date
Displays the current date. Edit the date to show the sun at a different time of year.

Time
Displays the current time. Edit the time to show the sun at a different time of day.

Daylight saving
Toggles the effect of daylight saving time on the time property.

Azimuth
Reports the horizontal angle of the sun measured clockwise from the north.

Note: This setting cannot be changed as the program calculates it from the other settings in this section.

Altitude
Reports the vertical angle of the sun, measured from the horizon.

Note: This setting cannot be changed as the program calculates it from the other settings in this section.

Source vector
Reports the direction of the sunlight. This setting cannot be changed as the program calculates it from the other settings in this section.

Rendered Shadow Details

Type
Displays realistic shadows with softer shadows based on extended light sources, according to the Softness property.

Softness
Determines the sampling size of shadows. Larger numbers are more accurate but take longer to render.

Geographic Location
Reports location of the drawing on earth. To change the location, click the Edit button to access the Geographic Location dialog box; see GEOGRAPHICLOCATION command.

24.113.3 Context menu options

New
Loads additional light definitions into the drawing.

Delete
Deletes light definitions from the drawing.

Rename
Renames the light definition.

Select All
Selects all light definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected light definition as current.

24.114 SUPPORTFOLDER command

Opens the operating system's file manager at the Support folder.

Shape Lite Pro Mechanical BIM

24.114.1 Description

Displays a file manager dialog box showing the contents of the support folder.

Note: The support folder holds many files that drawings and commands need, such as linetype patterns and tool palette libraries.

Note: Support folder paths are stored in theSRCHPATH systemvariable, and the default path is: *C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\Support*.

24.115 SUPPRESS command

Suppresses the display of entities.

Shape Lite Pro Mechanical BIM

Icon: 

24.115.1 Description

Creates named groups of entities whose display can be turned off. Use the STRUCTUREPANEL command to turn their display back on.

24.115.2 Options within the command

New
Create a new named group of entities.

Edit
Change the entities included with a named group.

Dissolve
Turn the display of suppressed entities back on and remove the name of the group.

24.116 SVGOPTIONS command

Opens the Settings dialog box with the **SVG export** category expanded.

Shape Lite Pro Mechanical BIM

24.116.1 Description

Opens the Settings dialog box with the **SVG export** category expanded to view and modify relevant system variables.

24.117 SWEEP command

Creates 3D solids or surfaces by sweeping profiles along a path.

Shape Lite Pro Mechanical BIM

Icon:

24.117.1 Description

Creates 3D solids or surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path.

Entities accepted as profile entities: 2D splines, 3D splines, 2D polylines, 2D solids, 3D solid faces, 3D solid edges, arcs, circles, ellipses, elliptical arcs, lines, regions, surface and mesh edges, traces.

Entities accepted as a path entity: 2D splines, 3D splines, 2D polylines, 3D polylines, edges of 3D solids, surfaces or meshes, helices, arcs, circles, ellipses, elliptical arcs, lines.

Note: Depending on the value of the DELOBJ system variable, the defining entities are either retained or deleted.

24.117.2 Methods

There are two methods to sweep entities:

- Create 3D solids
- Create surfaces

24.117.3 Options within the command

MOde
Sets the 3D entity type this command will create.

- **Solid**: creates 3D solids.
- **Surface**: creates surfaces.

Note: Open entities always construct 3D surfaces.

Select sweep path
Select an entity to define the path.

Alignment
Determines the angle at which the profile 2D entities are swept along the paths.

- **Yes**: aligns the sweep entity perpendicularly to the sweep path entity
- **No**: the sweep entity maintains its original orientation, and so it's not rotated along the path.

Base point
Defines the point on the sweep entity that follows the path. This option allows offset sweep paths.

Twist
Specifies a twist angle for the swept entity. The twist angle is the amount of rotation along the entire length of the sweep path.

- **Banking**: determines whether the swept entities will rotate naturally along a 3D sweep path, such as those made from a 3D polyline, spline, or helix.

Interactive
Allows to dynamically sweep profiles along a sequence of interconnected linear entities or along a portion of a path.

- **Transform profile**: use the Manipulator to move, rotate, mirror or scale the profile.

- **Accept**: accepts the current position of the profile.
- **Move cursor along selected edge**: the profile is swept dynamically along the path or path segment.
- **Select adjacent edge**: click when the next segment highlights to continue.
- **Undo**: removes the last segment.

24.118 SYSWINDOWS command

Arranges windows.

Shape Lite Pro Mechanical BIM

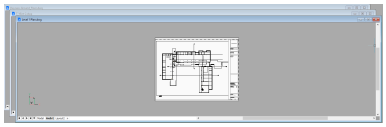
24.118.1 Description

Arranges the program's windows in tiled or cascaded formations (short for "system windows").

24.118.2 Options within the command

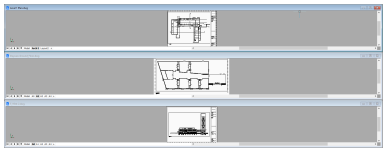
Cascade

Overlaps windows in a cascading arrangement like the WCASCADE command.



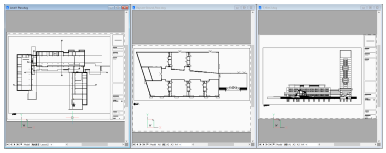
tile Horizontally

Arranges windows horizontally like the WHTILE command.



tile Vertically

Arranges windows vertically like the WVTILE command.



Arrange icons

Arranges icons of minimized windows at the bottom of the main window like the WIARRANGE command.



25. T

25.1 TABLE command

Opens the Insert table dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

25.1.1 Description

Opens the Insert table dialog box to create a table entity in the current drawing.

25.2 -TABLE command

Creates a table entity.

Shape Lite Pro Mechanical BIM

25.2.1 Description

Creates a table using specified number of columns and rows or automatically by specifying a point and dragging.

25.2.2 Method

There are 2 methods:

- By specifying the number of columns and rows.
- Automatically by dragging.

25.2.3 Option within the command

Style
Specifies the table style.

Width
Specifies the column width.

Height
Specifies the minimum row height.

Auto
Specify for the columns, rows or both to be created automatically by dragging.

Note: The Text Formatting window is displayed after creating the table.

25.3 TABLEDIT command

Edits text in a table cell.

Shape Lite Pro Mechanical BIM

25.3.1 Method

Edits text in a table cell through the Text Formatting window.

Note: To edit the format of cells, use the TABLEMOD command.

Note: The Text Formatting window operates exactly like the one from the MTEXT command.

25.4 TABLEEXPORT command

Exports the contents of a table.

Shape Lite Pro Mechanical BIM

25.4.1 Method

Exports the contents of a table to a CSV or XML file, through the Export Data dialog box.

Note: The CSV file can be imported into spreadsheet and database programs to format and further process the data.

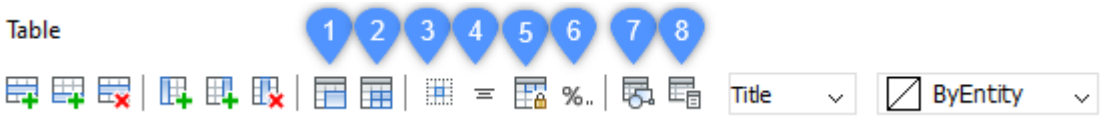
25.5 TABLEMOD command

Formats the properties of table cells and inserts blocks.

Shape Lite Pro Mechanical BIM

25.5.1 Method

Select a table cell to edit and the Table toolbar will be displayed.



Note: Hold the left mouse button and define a rectangle to select multiple cells.

Note: To edit format text in cells, use the TABLEDIT command.

25.5.2 Options within the command

Insert row above
Inserts an empty row above the selected cell.

Insert row below
Inserts an empty row below the selected cell.

Delete row
Deletes the row(s) of the selected cell(s) without warning.

Note: Use the U command to reverse the change.

Insert column left
Inserts an empty column at the left of the selected cell.

Insert column right
Inserts an empty column at the right of the selected cell.

Delete column
Deletes the column(s) of the selected cell(s) without warning.

Note: Use the U command to reverse the change.

Merge cells (1)
Merges the selected cells. Click the icon and choose an option.

- **Merge All**: merges all selected cells into a single cell.
- **Merge by Row**: merges the selected cells into rows, each with one cell.
- **Merge by Column**: merges the selected cells into columns, each with one cell.

Note: When cells are merged, all their content is erased, except the content of the first cell.

Unmerge cells (2)
Splits previously merged cells by rows and columns. BricsCAD® uses the row numbers (1, 2, 3, etc) and column letters numbers (A, B, C, etc) to know how to split the merged cells into the original set.

Note: When cells are unmerged, the content erased by the Merge Cells option is not restored.

Border style (3)
Sets the border style of the selection. Displays the **Cell Border Properties** dialog box.

Text alignment (4)
Sets the text alignment of text in the selected cells. Click the icon and choose an option.

Note: The content of each selected cell is aligned inside the cell. For example, if you select four cells and then choose Bottom Right, the content in each of the four cells is justified bottom-right.

Lock/unlock (5)
Locks and unlocks the content and/or the format of selected cells. Click the icon and choose an option:

- **Unlocked**: the selected cells are unlocked, which means that they can be edited.
- **Locked Content**: the content of the selected cells is locked, but the format of the cells can still be modified.
- **Locked Format**: the format of the selected cells is locked, but the content of the cells can still be edited.
- **Locked Format and Content**: the content and the format of the selected cells are locked.

Note: It is common to lock header cells.

Cell format (6)
Sets the cell format of the selection from the **Table Cell Format** dialog box.

Insert block reference (7)
Inserts a block or a drawing in the selected cell. Click the icon to display the **Insert Block in Cell** dialog box.

Note: Placing a large drawing in a small cell can lead to program instability.

Note: You can have both text and blocks in a single cell.

Note: Use the **Manage Cell Content** dialog box to arrange the text and blocks.

Manage cell content (8)
Manages blocks and drawings in cells; click the icon to display the **Manage Cell Content** dialog box.

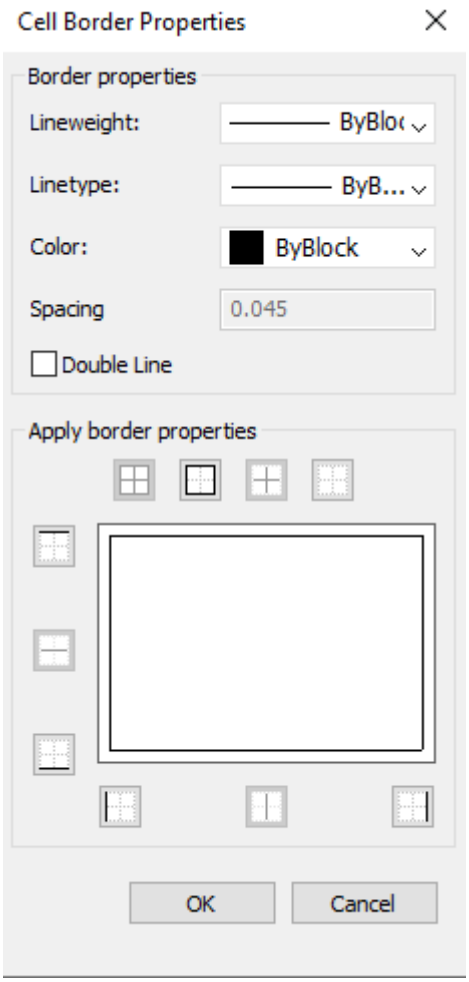
Cell style
Sets the sub-style of the selected cells. Click on the drop list and choose a sub-style name:

- **Title**: the title style is usually applied to the topmost row of the table
- **Header**: the Header style is usually applied to the first row of a column
- **Data**: the Data style is usually applied to all other rows in the table

Note: The properties of the Header, Border, and Cell sub-styles are defined by the table style. See the TABLESTYLE command.

Cell background
Sets the background color of the selected cell(s). Click on the drop-down list and choose a color.

Cell border properties dialog box
The **Cell Border Properties** dialog box controls the border properties of a table cell.



Border properties
Determines the look of the borders that surround each cell. More information about the Lineweight, Linetype and Color properties can be found in Properties panel.

Note: The Spacing option is not implemented yet.

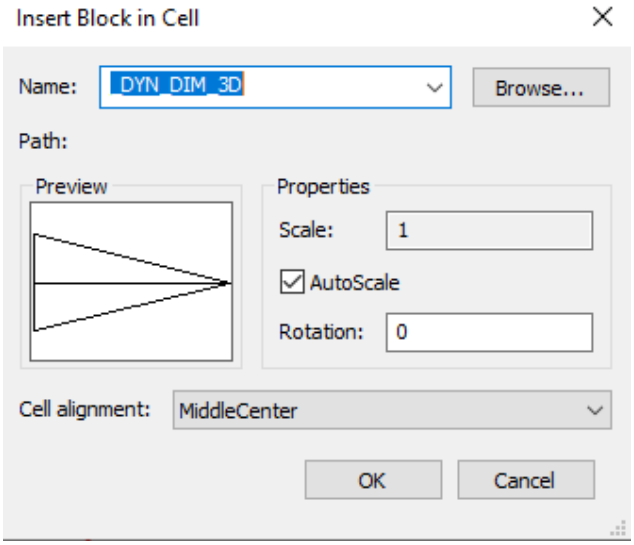
Apply border properties
Applies properties to only certain borders.

Note: Select more than one cell for the Inside borders option to work.

Note: Although the preview window only shows the double line effect, the other properties are applied to the specified cell border line.

OK
Exits the dialog box, applies the properties and returns to the Table toolbar.

Insert block in cell dialog box
The **Insert Block in Cell** dialog box manages the insertion of a block in a table cell.



Name
Specifies the name of the block or drawing to insert or click on **Browse to display the Open drawing file** dialog box and search for a block to insert.

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.

Properties
Determines the size of the block in the cell.

Scale
Specifies the size of the block:

- Numbers larger than 1.0 make the block larger.
- 1 - inserts the block at its actual size.
- Numbers smaller than 1.0 make the block smaller.
- Negative numbers flip the block upside down.

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

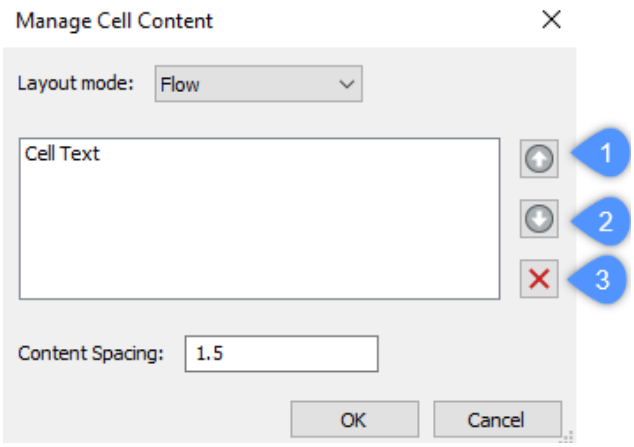
Rotation
Rotates the block around its center point (and not its insertion point). Enter an angle:

- Positive numbers rotate the block counter clockwise.
- 0 - does not rotate the block.
- Negative numbers rotate the block clockwise.

Cell Alignment
Aligns the block in the cell.

OK
Exits the dialog box, inserts the block and returns to the Table toolbar.

Manage cell content dialog box
The **Manage cell Content** dialog box manages the content of a table cell.



Layout mode
When a cell contains more than one element, this option determines how they relate to each other. Click the drop list and then choose an option:

- **Flow**: arranges the elements to fit the cell
- **Stacked horizontal**: forces the elements to be arranged horizontally
- **Stacked vertical**: forces the elements to be arranged vertically

Move Up (1)
Moves the selected element up the list so that it appears above other elements in the cell.

Move Down (2)
Moves the selected element down the list so that it's below other elements in the cell.

Delete (3)
Deletes the selected element from the cell.

Content Spacing
Specifies the distance between elements in the cell. Enter a number in drawing units.

Note: This option is inoperable.

OK
Exits the dialog box, changes the arrangement of the cell's contents, and then returns to the Table toolbar.

25.5.3 Context Menu Options

When the TABLEMOD command is running, right-click a cell to see its context menu.

Clear Cell Overrides
Resets the properties of cells to the Cell sub-style settings defined in the table style.

Clear Table Overrides
Resets the table properties to those defined in the table style.

Insert Block Reference
Inserts blocks and drawings.

Manage cell Content
Manages the text and blocks in cells.

Copy
Copies the content of the cell(s) to the Clipboard for pasting into other cell(s).

Cut
Cuts the content of the cell(s) to the Clipboard for pasting into other cell(s).

Paste
Pastes cell content from the Clipboard into cell(s), with these caveats:

- All content in the cell is replaced by the pasted item.
- When more than one cell is selected for pasting, content is pasted only in the first cell.
- Non-cell data is not pasted, such as entities from the drawing or text from another application.

Insert DataLink
Adds data from an external file to the end of the table. Acceptable file formats are:

- CVS: ASCII files in which fields are separated by a delimiter character, such as comma or tab.
- XLS: older Excel spreadsheet files.
- XLSX: new Excel spreadsheet files.

Note: For more details, see the DATALINK command.

Remove DataLink
Removes the link to the table, but the data remains in the table.

Note: Only available when the table has a datalink.

Update DataLink
Updates the data in the table from the external file.

Note: Available only when the table has a datalink.

Size rows equally
Sizes the selected rows to match the size of the tallest cell in the selection set.

Size columns equally
Sizes the selected columns to match the size of the widest cell in the selection set.

Size all rows equally
Sizes all rows in the table to match the tallest row.

Size all columns equally
Sizes all column in the table to match the widest column.

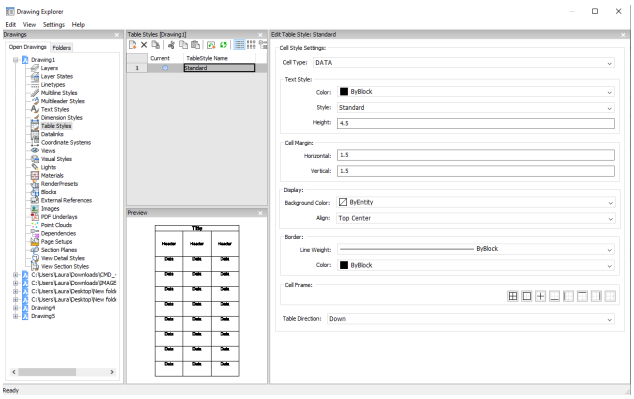
25.6 TABLESTYLE command

Opens the Drawing explorer dialog box with **Table Styles** selected.

Shape Lite Pro Mechanical BIM

25.6.1 Description

Opens the Drawing explorer dialog box with **Table Styles** section selected to view and modify table styles in the current drawing.



25.6.2 Options within the command

Cell style settings
Defines the cell style settings for each cell style: Title, Header, and Data. Click the drop-down list and choose a cell style.

Text Style
Sets the Color , Style and Height for the selected cell style.

Cell Margin
Defines the spacing between the cell border and the content of the cell.

Horizontal
Specifies the distance between the cell content and the vertical cell borders.

Vertical
Specifies the distance between the cell content and the horizontal cell borders.

Display
Defines the background color and the text alignment.

Background color
Specifies background color This can be done from the drop-down list or through the Color dialog box.

Align
Specifies an alignment through a drop-down list

Border
Sets the lineweight and color of the cell borders.

Lineweight
Sets lineweight through a drop-down list.

Color
Pick a border color from the drop-down list, or else choose Select Color to display the Color dialog box.

Cell Frame
Applies the border properties to specific lines that make up the cell frames.

Table Direction
Determines the direction in which the table grows when you add and remove rows.

Down
The title and header cells at the top, and the table grows downwards.

Up
The title and header cells at the bottom, and the table grows upwards.

25.6.3 Context Menu Options

New
Creates a new table style.

Delete
Deletes table style definitions from the drawing. The following table style definitions cannot be deleted

- Table styles in use
- 'Standard' table style

Rename
Renames the select table style.

Select All
Selects all table style definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected table style as current.

25.7 TABLET command

Configures digitizing tablets.

Shape Lite Pro Mechanical BIM

Alias: TA

25.7.1 Description

Works only with Windows, and requires that Wintab32.dll be installed in folder *C:\Windows\System32*.

25.7.2 Method

Toggles tablet mode. Configures and calibrates digitizing tablets.

25.7.3 Options within the command

ON
Turns on digitizing mode so that the entire surface of the tablet can be used for picking points and choosing entities, like a mouse.

OFF
Turns off digitizing mode to make available the menu areas for choosing commands.

CALibrate
Calibrates the corners of the tablet.

ConFiGure
Configures the tablet's menu areas.

25.8 TABSURF command

Creates a 3D mesh surface by extruding an 2D entity along a straight path entity.

Shape Lite Pro Mechanical BIM

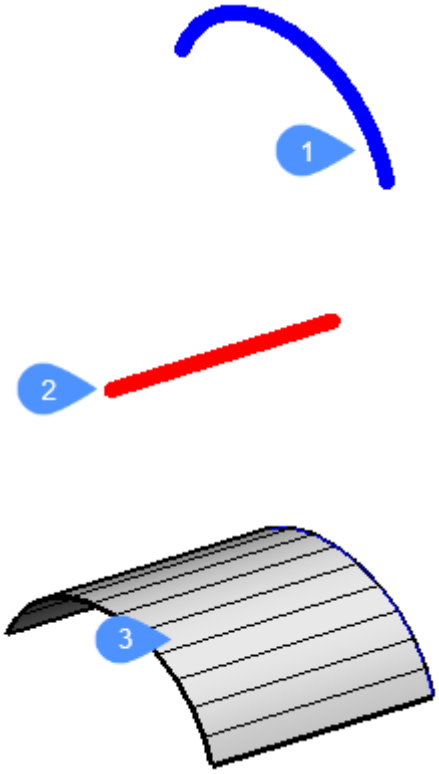
Icon: 

25.8.1 Method

Specify the cross-section shape (1) of the extrusion. The entity to be extruded must be open and must be a single entity.

Specify the extrusion path (2), which can be a line or open polyline. The polyline need not be a line segment. If it is a polyarc or multisegment polyline, the extrusion (3), however, is always "straight," going from the start to the endpoint of the polyline.

The path is a vector, which means its length specifies the length of the surface. Its direction specifies the direction of extrusion. The extrusion starts at the entity to be extruded, but goes in the direction indicated by the path. The end selected on the polyline or line defines the direction of the extrusion.



25.9 TANGENT command

Toggles the Tangent entity snap.

Shape Lite Pro Mechanical BIM

Icon:

25.9.1 Description

Toggles the Tangent entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

25.10 TCONNECT command

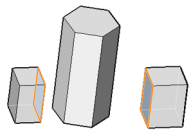
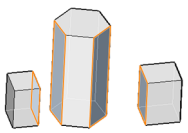
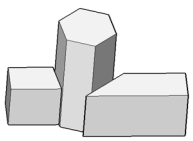
Connects solid faces and edges of planar surfaces to other solids, planar surfaces or regions.

Shape Lite Pro Mechanical BIM

Icons:

25.10.1 Description

Connects the selected solid faces or edges of the planar surfaces to other solid faces, planar surfaces or regions.

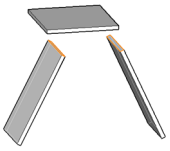
Step1	Step2	Step3
		

Note: If the DELETEINTERFERENCE system variable is ON, the connected solid is subtracted from interfering solids.

25.10.2 Options within the command

Connect to nearest

Connects the selected face(s) or edges of planar surfaces to the nearest regions, planar surfaces or face(s) of other solid(s).

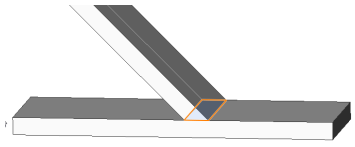
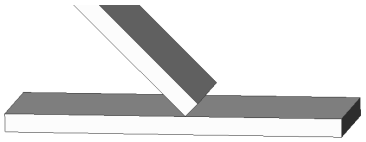




Disconnect

This option applies to solid faces only. The face is disconnected and made perpendicular to the adjacent faces, which results in a single edge or vertex connection.

Select a 3D solid face, planar surface or region to disconnect from the nearest face in the model (or the currently connected face).

Before	Disconnect result
	

Connect

Switch to the connect face mode.

25.11 TEMPLATEFOLDER command

Accesses the folder holding template files.

Shape Lite Pro Mechanical BIM

25.11.1 Description

Opens the Templates folder in which the program stores DWT template files; the folder is displayed by the operating system's file manager. To open the template file as a new drawing, double-click a file name.

By default, the template files are found here: *C:\Users\<user_name>\AppData\Local\Bricsys\BricsCAD\Vxx\en_US\Templates*.

25.12 TEXT command

Creates a single-line text entity.

Shape Lite Pro Mechanical BIM

Icon: A

Alias: TX

Note: When the TextEval system variable is set to 1, this command evaluates LISP expressions.

25.12.1 Method

There are two methods:

- Places single lines of text in the drawing by specifying height of text and rotation angle.
- Evaluates LISP expressions.

Note: If the text style is annotative, the height value 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.

25.12.2 Options within the command

use defined Style
Changes the text style to use for the text.

?
Lists all styles defined in the current drawing.

Align on line
Aligns text between two points.

Note: This option suppresses the height and angle prompts, because the height and angle are determined by the two pick points.

Fit between points
Fits text between two points and prompts for the height.

Note: This option suppresses the angle prompt, because the angle is determined by the two pick points. Entering certain heights can cause the text to be excessively elongated or squeezed.

Center horizontally
Centers text in the middle of the baseline.

Middle (hor/vert)
Centers text on the middle of the text.

Right justify
Right-justifies each line of text.

Justification options...
Displays all justification options. Choose between: Top Left, Top Center, Top Right, Middle Left, Middle Center, Middle Right, Bottom Left, Bottom Center, Bottom Right.

Note: Text can be moved or stretched through grips.

25.13 -TEXT command

Places single-line text in drawings.

Shape Lite Pro Mechanical BIM

Alias: -T

25.13.1 Description

Places single lines of text in the drawing, unformatted; works at the Command line.

See the TEXT command.

25.14 TEXTSCR command

Switches to the text screen.

Shape Lite Pro Mechanical BIM

25.14.1 Description

Displays the Prompt History window (short for "text screen"). This window displays the last 256 lines of command and prompt text. To display greater or fewer lines of history, change the value of the SCRLHIST variable. To return to the graphics screen, enter the GRAPHSCR command or press F2.

Note: A context menu displays when you right click on the command history. See the full description in the Command line panel article.

25.15 TEXTTOFRONT command

Displays text and/or dimensions on front of all other drawing entities.

Shape Lite Pro Mechanical BIM

Icon: 

25.15.1 Description

Brings the display of all text and/or all dimensions to the front of the display order, so that no other drawing entities can overlap text and dimensions.

25.15.2 Options within the command

Text
Brings all text to the front of the display order.

Dimensions
Brings all dimensions to the front.

Both
Brings both text and dimensions to the front.

25.16 TFLOAD command

25.17 TFSAVE command

25.18 TIME command

Reports the use of time in the drawing.

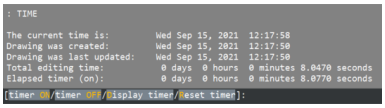
Shape Lite Pro Mechanical BIM

Icon:

Alias: TI

25.18.1 Description

Reports the date and time the drawing was created, last updated, and spent editing; works at the Command line.



25.18.2 Options within the command

timer ON
Turns on the independent (“elapsed”) timer. The program reports, “The drawing timer is ON.”.

timer OFF
Turns off the independent timer. The program reports, “The drawing timer is OFF.”.

Display timer
Reports the date and time the drawing was created, last updated, and the amount of time spent editing (that the drawing was open).

Reset timer
Resets the independent timer.

25.19 TIN command

Creates a TIN Surface.

Shape Lite Pro Mechanical BIM

Icon:

25.19.1 Description

Creates a TIN Surface from an imported point files or by placing new points.

A TIN Surface is a three-dimensional geometric representation of a terrain or a civil feature, which also includes its representation with triangles, contours, slopes and elevation ranges.

The algorithm connects the nearest lying points into triangles that form a Triangular Irregular Network (TIN), using the Delaunay triangulation method.

The user can affect the way points are connected by adding boundaries and breaklines to the TIN Surface. Triangles always follow specified boundaries and breaklines and never intersect them. The TIN Surface can also be edited with other options available in the TINEDIT command.

A TIN Surface is dynamically linked to its input data, while other Civil entities, such as Gradings, Alignments and Corridors are dynamically linked to the TIN Surface.

25.19.2 Method

A TIN surface can be created from different type of input data:

- from CAD elements (points and linear entities) and Civil points,
- from imported point files,
- from Civil points, for which we specify the position and elevation in the drawing,
- from 3D Face elements,
- from Point Cloud element,
- from a specified point group.

Select a set of points, polylines, 3D faces or a point cloud in the drawing to create a TIN surface. Points are added to the surface as TIN points, 3D polylines are added to the surface as breaklines. Linear entities can be added to the TIN surface as breaklines, edges or points.

25.19.3 Options within the command

Import from file
Creates a TIN surface from an imported points file in text file format (.TXT), comma-delimited file format (.CSV) and other text formats where the XYZ coordinates in the input file are separated by any delimiter. Multiple point files could be selected to create a TIN surface.

Enter point file delimiter
Defines another point file delimiter.

place Points
Creates a TIN surface by placing at least three points.

create from Faces
Creates a TIN surface by selecting 3D faces. You may choose to add edges of 3D faces as breaklines (Y) or to create TIN Surface only with points of 3D Faces (N).

Create from Point Cloud
Creates a TIN surface from a Point Cloud.

Simplify the TIN surface?
Yes: simplifies the TIN surface created from an imported points file, and from a Point Cloud by reducing the points density. The point filtration is defined by setting values for the radius or simplification step, and for elevation difference.
No: A TIN Surface is created from all points in the input file or in Point Cloud.

> ***Note:*** After the first iteration of the simplification, a report is displayed on the command line stating the number of points before simplification, the number of removed points and the number of points after simplification. In the next step, the user can choose between options “Simplify again” or "Create TIN Surface”.

Create TIN surface
Creates the TIN surface.

cLip polygon
Creates TIN Surface only in the area defined by the CLIP polygon (a closed polyline).

> ***Note:*** There is the option to remove the crossed breaklines, if any.

25.20 TINASSIGNIMAGE command

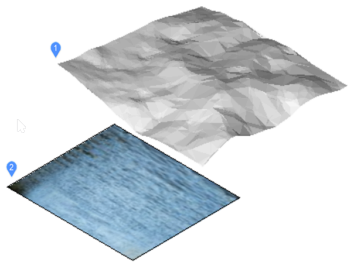
Assigns a raster image as a TIN surface material.

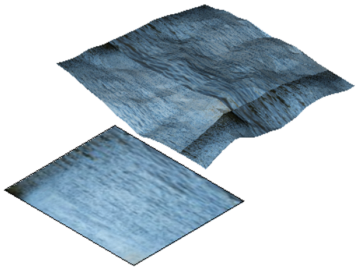
Shape Lite Pro Mechanical BIM

Icon:

25.20.1 Method

Select the TIN surface (1) and select an attached raster image (2) to assign to the surface.





25.21 TINEDIT

Allows editing a topographical surface (TIN surface).

Shape Lite Pro Mechanical BIM

Icons: 

25.21.1 Description

Allows you to add or remove points, breaklines and boundaries on a TIN (triangulated irregular network) surface.

Pick points and specify their elevation value to add them to an existing TIN surface or choose another option in the Command line.

25.21.2 Options within the command

Add Points
Sets points to add to a TIN surface.

Add Breaklines
Sets linear entities to be added as breaklines to a TIN surface.

Standard
Adds standard breaklines.

Projected
Adds projected (non-destructive) breaklines.

Add bOundaries
Sets a closed polyline as boundaries to add to a TIN surface.
You can choose the boundary type:

Outer
Creates an outer surface boundary.

Show
Shows the surface boundary inside a hidden surface boundary.

Hide
Hides the surface boundary.

Remove Points
Deletes points from a surface.

Remove Breaklines
Deletes breaklines from a surface.

Remove bOundaries
Deletes boundaries from a surface.

CLip
Clips an existing TIN Surface in the drawing with a selected polygon as the clipping boundary for the surface.
You can choose to remove the polygonal breaklines or not.

Point Location
Allows you to modify a point location by specifying the point to edit location and the new location.

Point Elevation
Allows you to modify a point elevation by specifying a point and entering a new elevation value.

Multiple
Allows you to select multiple points to change their location or elevation.

Delta elevation
Allows you to modify the elevation for multiple points.

Add Line
Allows you to add a new line by specifying the first point and the second point.

Delete Line
Deletes lines covered by a selection window.

Note: The lines to be deleted are displayed in red.

25.22 TINEXTRACT command

Extract entities from TIN surface to which creates a mesh or 3D solid between TIN surfaces or between a TIN surface and elevation or vertical offset.

Shape Lite Pro Mechanical BIM

Icon: 

25.22.1 Description

Creates a mesh or 3D solid between two TIN surfaces or between a TIN surface and elevation or vertical offset.

Select a TIN surface and choose which entity want to extract or create.

25.22.2 Options within the command

Mesh
Creates a mesh as an offset, between surfaces or as an elevation:

vertical offset
Creates a mesh as a vertical offset above the TIN surface.

between Surfaces
Creates a mesh between two TIN surfaces.

Elevation
Creates a mesh vertically from a TIN surface to a fixed elevation height.

Solid
Creates a solid as an offset, between surfaces or as an elevation.

vertical offset
Creates a solid as a vertical offset above the TIN surface.

between Surfaces
Creates a solid between the TIN surfaces.

Elevation
Creates a solid vertically from a TIN surface to a fixed elevation height.

Points
Extracts all points on the surface.

Faces
Extracts all triangular faces of the surface.

Contours
Extracts the contours of the surfaces for an elevation.

Minor
Extracts the contours of the surfaces for elevation according to minor contours interval set in TIN surface properties panel .

maJor
Extracts the contours of the surfaces for elevation according to major contours interval set in TIN surface properties panel.

All
Creates contours for all elevations.

Border
Extracts the outer boundary of the surface.

25.23 TINMERGE command

Combines two or more TIN surfaces (or gradings) into a new TIN surface.

Shape Lite Pro Mechanical BIM

Icon: 

25.23.1 Description

Combines a base surface and two or more other TIN surfaces to merge with, that will replace the part of the base surface that it covers.

Select a base TIN surface and one or more TIN surfaces to merge with and specify if you want to keep or delete the original surfaces.

25.24 TINMODIFY command

Deforms or smoothens a selected TIN surface.

Shape Lite Pro Mechanical BIM

Icon: 

25.24.1 Description

Modifies a selected TIN surface by deforming or smoothening it or creates a new modified TIN surface.

25.24.2 Options within the command

Deform
Modifies the selected TIN surface or creates the deformable part as a new TIN surface.

specify Contour
Creates a deformation with a contour.

select Entity
Creates a deformation with an elevated chosen closed entity.

Note: Net volume = fill-cut.

draw Polygon
Creates a deformation with an elevated polygon.

Smoothen
Modifies the selected TIN surface by smoothening it between the indicated boundaries.
Creates a circular smoothening boundary by indicating a radius value and a location where you want the smoothening.

select Entity
Smoothens the surface part between the boundaries of an entity.
Select a closed entity for the smoothening and choose a location where you want the smoothening.

draw Polygon
Smoothens the surface part between the boundaries of a polygon.
Pick points for a polygon and press ENTER to Close the polygon and choose a location where you want the smoothening.

25.25 TINPROJECT command

Projects a point-based entities or linear entities to a TIN surface.

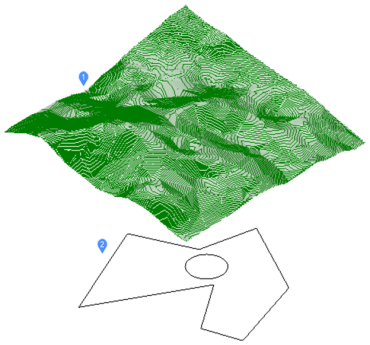
Shape Lite Pro Mechanical BIM

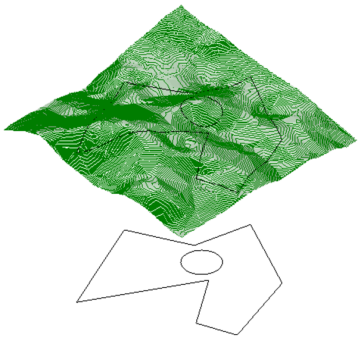
Icon: 

25.25.1 Description

Projects point entities (Point, Block, Text) or linear entities (Line, Polyline, Circle) to a TIN Surface.

Select the TIN surface (1), select the entities to project (2) and choose to keep or delete the projected entities.





25.26 TINSERT command

Inserts blocks into the cell of a table.

Shape Lite Pro Mechanical BIM

25.26.1 Method

Pick a table cell to open the Insert Block in Cell dialog box.

25.27 TINVOLUME command

Creates a TIN volume surface between a base and comparison TIN surfaces or an elevation.

Shape Lite Pro Mechanical BIM

Icon:

25.27.1 Method

When the TIN Volume Surface associativity option of the Civil / Associativity user preference is set, TIN volume surfaces automatically rebuild when their source surfaces change.

Select the base and the comparison TIN surfaces between which a TIN volume surface will be created and select a bounding area for the TIN volume surface.

25.27.2 Options within the command

Elevation
Creates a TIN volume surface between a base and an elevation.
Select the base TIN surface, enter the desired elevation value and select a bounding area for the TIN volume surface.

25.28 TINWATERDROP command

Creates water drop paths in real-time by moving the cursor over a TIN Surface. Water path is created as a 3D polyline in the current position if you click the left mouse button.

Shape Lite Pro Mechanical BIM

Icon: 

25.28.1 Method

Select the TIN surfaces, move the cursor over the surface and a line representing the drop path is shown in real time. Click the left mouse button to create the 3D polyline of the water drop path.

25.29 TIPS command

Displays the Tips panel.

Shape Lite Pro Mechanical BIM

25.29.1 Description

Displays the Tips panel, which provides helpful advice on using certain commands.

25.29.2 Method

There is no command named Tips. To display this panel, right-click a toolbar or the ribbon, and then choose Tips from the context menu.

Note: The Tips panel is always open in Shape.

25.30 TOLERANCE command

Opens the Geometric tolerance dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

25.30.1 Description

Opens the Geometric tolerance dialog box to add tolerance symbols to the current drawing.

25.31 TOOLBAR command

Toggles the display of toolbars at the Command line.

Shape Lite Pro Mechanical BIM

25.31.1 Method

Enter the name of the toolbar or choose the All option to turn all toolbars on or off.

25.31.2 Options within the command

Show
Displays the toolbar(s).

Hide
Hides the toolbar(s).

Left
Docks the toolbar(s) on the left.

Right
Docks the toolbar(s) on the right.

Top
Docks the toolbar(s) to the top.

Bottom
Docks the toolbar(s) to the bottom.

Float
Show the floating toolbar(s).

25.32 -TOOLBAR command

Toggles the display of toolbars at Command line.

Shape Lite Pro Mechanical BIM

25.32.1 Description

For more information see TOOLBAR command.

25.33 TOOLPALETTES command

Opens the Tool palettes panel.

Shape Lite Pro Mechanical BIM

25.33.1 Description

Opens the Tool palettes panel to display it in the current workspace. The Tool palettes panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Tool palettes panel can be either floating, docked or stacked.

25.34 TOOLPALETTESCLOSE command

Closes the Tool palettes panel.

Shape Lite Pro Mechanical BIM

25.34.1 Description

Closes the Tool palettes panel to hide it from the current workspace. If the Tool palettes panel is stacked when you close it, the Tool palettes tab or icon is removed from the stack.

25.35 -TOOLPANEL command

Toggles the display of panels.

Shape Lite Pro Mechanical BIM

25.35.1 Description

Turns the display of panels on and off, such as the Properties and Tips panels.

25.35.2 Method

There are two methods to toggle panels:

- Enter the name of the panel, and then choose a display option.
- Enter ? to list the names of all panels in the program.

25.35.3 Options within the command

Show
Displays the tool panel.

Hide
Hides the tool panel.

Toggle
Toggles the display of the tool panel, On and Off.

25.36 TORUS command

Creates a 3D Solid in the shape of a torus.

Shape Lite Pro Mechanical BIM

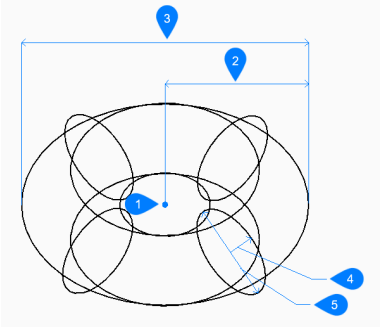
Icon: 

Alias: TOR

Note: In BricsCAD Lite, which doesn't support 3D Solids, the TORUS command launches the AI_TORUS command.

25.36.1 Description

Creates a 3D Solid in the shape of a torus. Choose from a combination of options including center and the radius or diameter of both the whole torus and the body of the torus.



1 Center

2 Radius of the whole torus

3 Diameter of the whole torus

4 Radius of the body

5 Diameter of the body

25.36.2 Methods to create a torus

This command has 1 method to begin creating a torus:

- Center of whole torus

Center of whole torus
Begin creating a torus by specifying its center then:

Set radius of whole torus
Specify the overall radius of the torus. The radius is measured from the center of the whole torus to the center of the body (tube) of the torus.
Additional option: [Diameter]

Set radius of body of torus
Specify the radius of the body (tube) of the torus.
Additional option: [Diameter]

25.36.3 Options within the TORUS command

After you begin creating a torus, the following options may be available:

Diameter (of whole torus)
Specify the diameter of the entire torus. The diameter is twice the distance from the center of the whole torus to the center of the body (tube) of the torus.

Diameter (of body of torus)
Specify the diameter of the body of the torus.

25.37 TPNAVIGATE command

Loads tool palettes at the Command line.

Shape Lite Pro Mechanical BIM

25.37.1 Description

Loads a tool palette or palette group by name. If the Tool Palettes panel is not already open, then it is displayed (short for "tool palette navigation"). This command is meant for use by macros.

25.37.2 Method

There are two methods to load tool palettes:

- Specify the tool palette to display by entering the name of a palette.
- Specify the palette group to display by entering the name of a group.

25.38 TRACE command

Draws traces.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command is rarely used as the PLINE command is more convenient.

25.38.1 Description

Draws traces, like wide lines, with mitered vertices.

The trace is not drawn until the second pick point. The delay allows BricsCAD to determine the mitering needed at vertices, which this command does automatically.

Note: The FILLMODE system variable affects the look of traces.

25.39 TRANSPARENCY command

Toggles the transparency of monotone images.

Shape Lite Pro Mechanical BIM

Icon:

25.39.1 Methods

This command allows to set the transparency for a certain type of images. When transparency is on, the background color of the image is transparent.

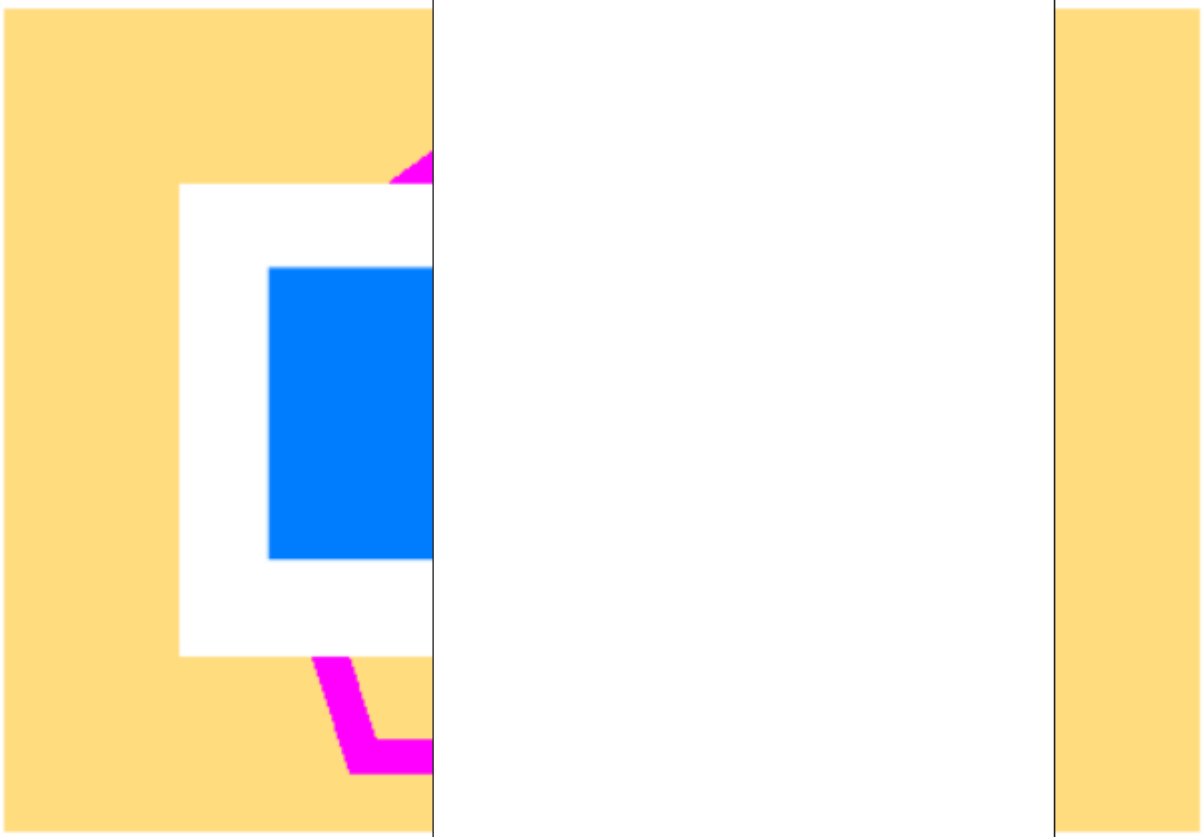
Note: To change the transparency of entities, use the Transparency property in the LAYER and PROPERTIES commands.

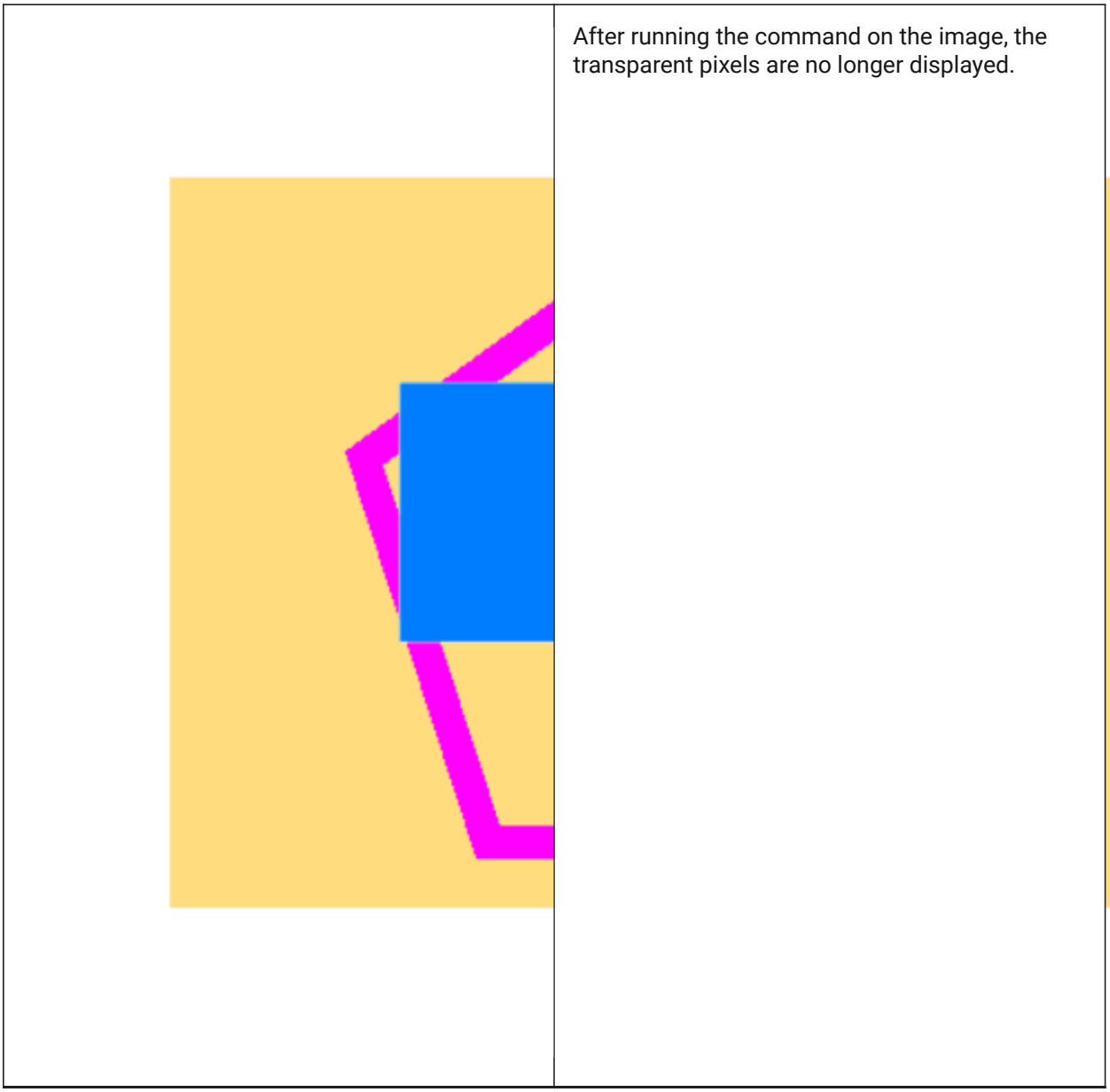
There are two main categories of images that can be processed by this command:

- Images that contain transparency in form of an alpha channel, usually PNG and TIFF formats.
- Images that do not contain an alpha channel.

Example of an image that contains an alpha channel

The images with an alpha channel can be processed by the TRANSPARENCY command. Before running the command on the image, its transparent pixels will be displayed as white. After the command has been run on the image, its transparent pixels will no longer be displayed.

	When the image with an alpha channel, placed of top of other entities, the transparent pixels are displayed as white.



After running the command on the image, the transparent pixels are no longer displayed.

Example of an image that does not contain an alpha channel

Before processing this kind of images, they must be converted to a monochrome palette using some photo editor.

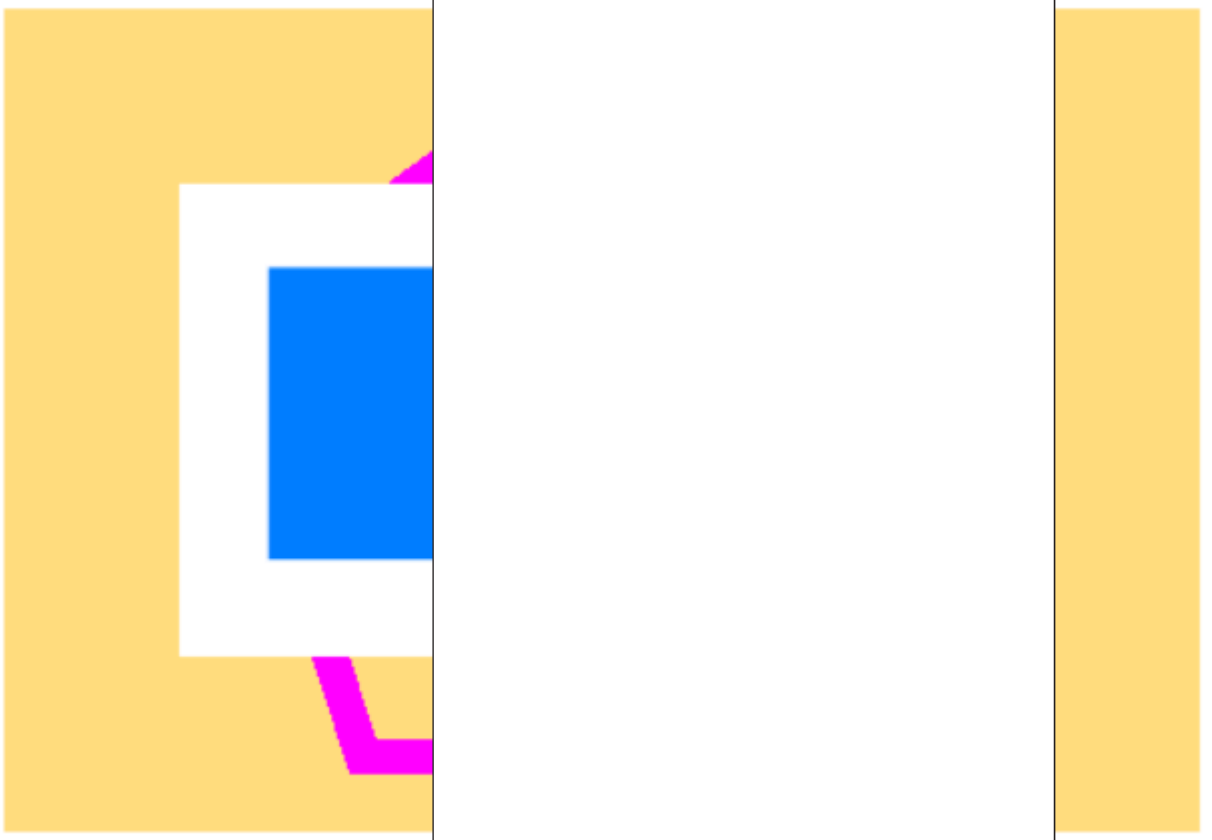
There are some conditions for an image to be affected by this command and to obtain a satisfactory result:

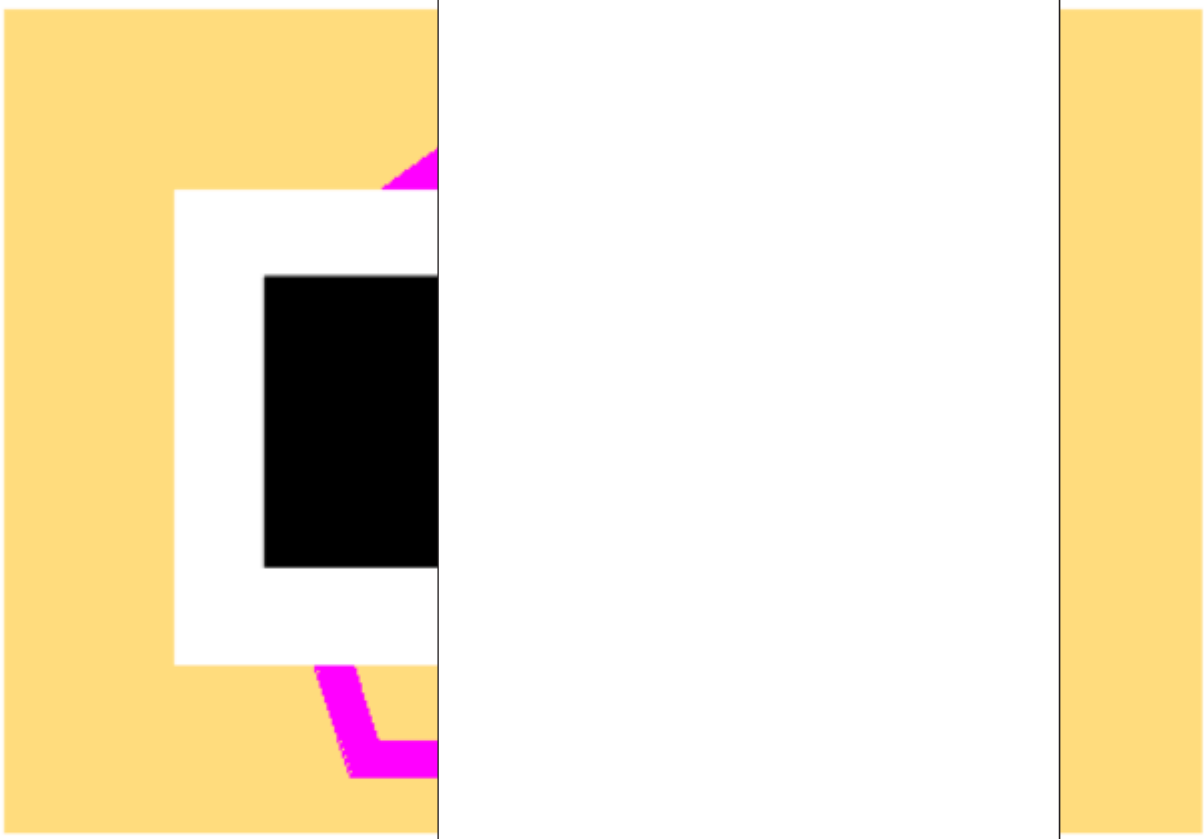
1. The image must be in TIFF, BMP or PNG graphical format.
2. The background color of the image must be white or very close to white.
3. The image must have a good contrast between the objects in focus and the background.
4. In a photo editor, there should be applied a Threshold adjustment. This way, there would be only two colors in the image: black and white.

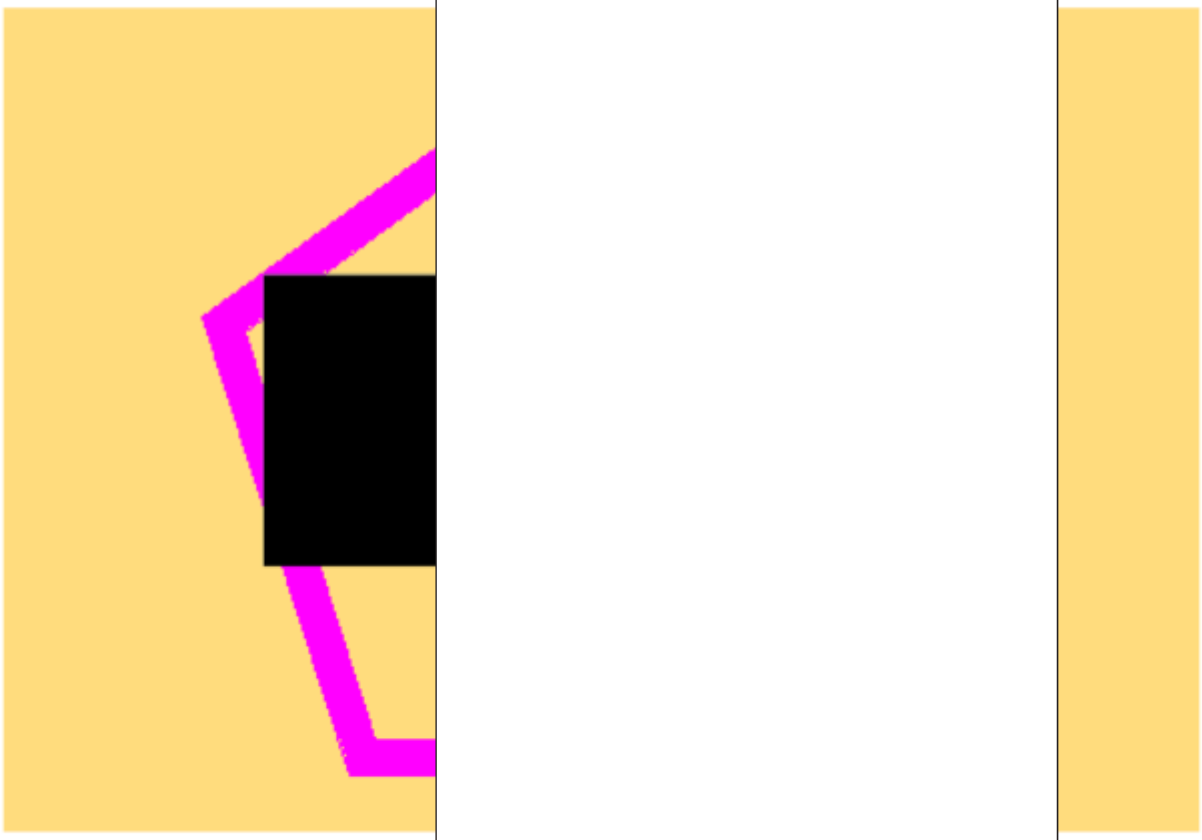
5. The image must be saved in a graphical format with a color bit depth of 1.

Note: Saving with a color bit depth of 1 is an important step. Even if there are only black and white colors in the image, the graphical format could be different than this. Check the graphical format before attaching the image to the drawing.

After running the command on the image, the white pixels will become transparent.

	The image with white background placed on top of other entities looks like this. This is just for visualizing purposes. The original image should not be attached to the drawing.

	The image has been applied to a Threshold adjustment and converted to a color bit depth of 1, and then attached to the drawing.

	After running the command on the image, the transparent pixels are no longer displayed.

Other graphical formats, as JPEG, are not affected by this command. The files that have one of these formats must be converted using a photo editor that can output according to the specifications above.

25.39.2 Options within the command

selection options(?)
Allows to choose a selection method. See the SELECT command.

Enter transparency mode [ON/Off]
Toggles the transparency of the selected images.

25.40 TRIM command

Trims entities by a cutting line.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: TR

25.40.1 Method

There are two methods:

- Trims entities.
- Extends entities while holding down the Shift key.

Note: The following entities can be trimmed: lines, two and three dimensional polylines, arcs, circles, ellipses, elliptical arcs, splines, rays and infinite lines.

Note: Cutting entities can be: lines, splines, polylines, arcs, circles, elliptical arcs, ellipses, rays, infinite lines, layout viewports.

25.40.2 Options within the command

Edge mode
Toggles edge mode.

Extend
Trims entities that do not physically intersect the cutting entity.

No extend
Trim only entities that intersect.

Projection
Defines how intersections are projected.

Note: This option affects the operation of the Edge mode option.

No projection
Entities are not projected.

Ucs
Entities are projected according to the current UCS.

View
Entities are projected according to the current view plane.

eRase
Undoes the last trim action.

25.41 TXT2MTXT command

Combines a selection of TEXT and MTEXT entities into a single MTEXT entity.

Shape Lite Pro Mechanical BIM

Alias: COMBINETEXT

25.41.1 Method

Select the TEXT and MTEXT objects to combine. The combination is made depending on the value of the COMBINETEXTMODE system variable.

25.41.2 Options within the command

SEttings
Displays the options of the COMBINETEXTMODE system variable in a dialog box.

Combine into a single mtext
Combines the selected TEXT entities into a single MTEXT entity.

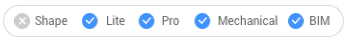
Sort top-down
Specifies the order of the selected text entities by descending vertical position.

Word-wrap text
Combines all selected TEXT entities into a single line and then wraps any text that exceeds the width of the MTEXT to the next line. The MTEXT width matches the width of the largest text entity in the selection. Paragraphs in the selected MTEXT entities are preserved.

Uniform linespacing
Applies consistent interline spacing.

25.42 TXTEXP command

Explodes text into polylines.



25.42.1 Method

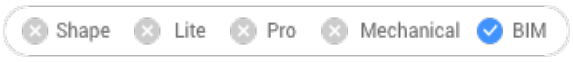
Select the TEXT entities to be exploded.

Note: SHP and TTF fonts are exploded to individual polylines and polyarcs.

Note: The command does not explode attributes in blocks or text in tables, or text in another space (Model vs Paper). Standalone attributes are, however, exploded.

25.43 TYPEDPLANS command

Opens the BIM Typed Plans dialog box.



Icon: 

25.43.1 Description

Opens the BIM Typed Plans dialog box, listing all Typed Plans in the support folder. You will be able to customize drawing plans as well as the BIM Tags.

26. U

26.1 U command

Reverses the action of the previous command.

Shape Lite Pro Mechanical BIM

Icon: 

26.1.1 Description

Reverses the action of the previous command to restore entities to their prior state.

This command displays no prompts and has no options.

26.2 UCS command

Creates and displays user-defined coordinate systems (UCS) through the command line.

Shape Lite Pro Mechanical BIM

Icon: 

26.2.1 Method

Specify a new UCS origin through one, two or three points in the drawing or choose one of the command's options.

26.2.2 Options within the command

Face
Aligns the UCS to a face of a 3D solid.

Note: This option works only with flat 3D faces and not curved ones, such as on a sphere.

Flip
Reverses the direction of the Z-axis.

Xrotate
Rotates the UCS 180˚about the X-axis.

Yrotate
Rotates the UCS 180˚about the Y-axis.

NAmed
Creates, restores and deletes names UCSs.

?
Lists the names of UCSs in the drawing.

Note: Type:

- *to list all UCS names.

- name* to list name of UCSs that begin with name.
- name to list the UCS specified by name.

Entity
Aligns the UCS to a selected entity.

UCS definition
Represents the orientation and type of entity defining the orientation of the UCS.

Note: The UCS origin is located to the end point, center point, or vertex closest to the pick point. The x-axis is aligned with the entity or an edge. The x,y-plane is aligned to the entity's plane. For ambiguous objects with obvious orientation, like a circle, the orientation is maintained.

Previous
Changes the UCS to the previous one.

View
Sets the UCS to the current viewpoint.

Note: The x axis and y axis are parallel to the view edges. The z axis is perpendicular to the view, with positive Z-axis pointing to the viewer. The origin is copied from the previous coordinate system.

X
Rotates the current UCS about the X-axis.

Y
Rotates the current UCS about the Y-axis.

Z
Rotates the current UCS about the Z-axis.

Z Axis
Sets the UCS relative to the Z-axis.

Note: Y-plane is perpendicular to the Z-axis with the X-axis horizontal and the Y-axis pointing upwards.

Move
Moves the origin of the current UCS, keeping the orientation of the axes.

World
Switches to the world coordinate system (WCS).

26.3 UCSICON command

Controls the display of the UCS icon.

Shape Lite Pro Mechanical BIM

26.3.1 Description

The UCS icon is displayed differently, depending on the visual style and workspace.

The colors denote the directions of the axes. It can be controlled by the COLORX, COLORY, COLORZ system variables.

26.3.2 Options within the command

display in All views
Applies the changes to all viewports.

display at ORigin
When the origin is outside of the viewport, the UCS icon is displayed at the corner defined by the UCSICONPOS system variable.

display in Corner
Displays the UCS icon in the viewport corner defined by the UCSICONPOS system variable.

Note: The values for UCSICONPOS system variable are displayed in the Settings dialog box.

26.4 UNDEFINE command

Removes access to commands temporarily.

Shape Lite Pro Mechanical BIM

26.4.1 Method

Enter the name of the command to undefine.

Undefined commands are accessed:

- By prefixing the command name with a dot, such as: .CommandName.
- By using the REDEFINE command to recover the command name.

Note: Commands are undefined by developers who want to replace the command with a more extensive version they're written, or to prevent neophytes from using destructive commands, like ERASE and EXPLODE.

26.5 UNDO command

Undoes one or more commands.

Shape Lite Pro Mechanical BIM

Icon: 

26.5.1 Description

Undoes one or more commands. BricsCAD reports the commands being undone.

Note: Some commands cannot be undone.

26.5.2 Options within the command

Number of steps to undo
Specifies the number of commands to undo by entering a number.

Mark
Sets a marker.

Back to mark
Undoes all commands back to when the marker was set.

BEgin set
Sets the start to a group that groups the following commands. The UNDO command treats the commands in the set as a single undo.

End set
Ends the group of commands.

Control
Specifies several options for the command.

None
Turns off the undo mechanism.

Note: This is useful when disk space is running low, because the undo mechanism uses disk space.

One
Limits this command to a single undo. The UNDO command is changed into the U command.

All
Turns on the Undo mechanism.

Layer
Specifies whether the UNDO command combines Layer dialog operations.

Auto
Considers all commands executed by a macro as a single undo.

Note: When Control is set to **None** or **One**, the **Auto**, **Begin** and **Mark** options are not available.

26.6 UNDOENT command

Undoes revisions to entities.

Shape Lite Pro Mechanical BIM

26.6.1 Description

Reverses editing changes made to individual entities. Acts like the UNDO command, but is specific to each entity.

This command treats a block, xref, or 3D solid as a single entity, but a group as individual entities.

26.6.2 Options within the command

Number of steps to undo
Specify the number of editing steps to reverse. After the initial revision is undone, the entity is erased from the drawing.

rEvisions
Report the number of revisions the selected entity has undergone.

eXit
Exit the command.

Redo
Reverses the undo action.

Note: When an entity is connected to other ones, such as an edge connected to a face, BricsCAD asks if the other entities should be reverted. Alternatively, use the History drop-down list in the Properties panel to undo revisions to entities.

26.7 UNGROUP command

Explodes groups of entities.

Shape Lite Pro Mechanical BIM

Icon: 

26.7.1 Method

There are two methods to explode a group of entities:

- By selecting the group.
- By entering its name in the Command line.

26.7.2 Options within the command

Name
Enter the name of a group to explode.

?
Lists the groups existing in the drawing.

26.8 UNION command

Performs Boolean union operations on 3D solids and 2D regions.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: UNI

Note: In the BricsCAD® Lite license level, the command applies to region entities only.

26.8.1 Description

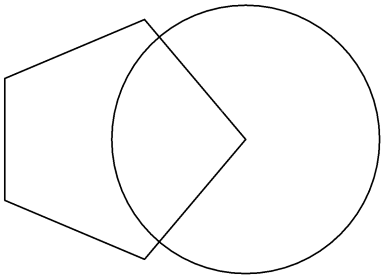
Performs Boolean union operations on 3D solids and 2D regions by adding set of entities to another set to form a single entity.

Note: The resulting entity takes on the properties of the ACIS entity selected first.

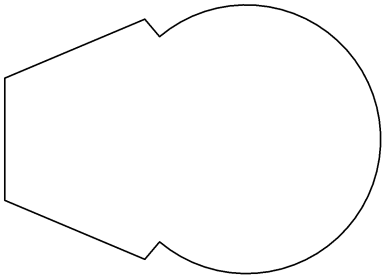
26.8.2 Method

Specify the ACIS entities to join into a single entity. The program joins regions and 3D solids into a single entity.

The entities selected:



Result:



26.9 UNISOLATEOBJECTS command

Unhides entities.

Shape Lite Pro Mechanical BIM

Icon:

Alias: UNHIDE, INHIDOBJECTS, UNISOLATE

26.9.1 Description

Unhides entities that were hidden by the HIDEOBJECTS and ISOLATEOBJECTS commands.

Note: The UNISOLATEOBJECTS command is enabled under BEDIT and REFEDIT sessions.

26.10 UNITS command

Opens the Settings dialog box with the **Drawing Units** category expanded.

Shape Lite Pro Mechanical BIM

Icon: 1.2 3'4"

Alias: DDUNITS, UN

26.10.1 Description

Opens the Settings dialog box with the **Drawing Units** category expanded to view and modify relevant system variables.

26.11 -UNITS command

Sets units of linear and angular measurements in the Command line.

Shape Lite Pro Mechanical BIM

Alias: -UN

transparent: **'-units**

26.11.1 Options

Unit mode (LUNITS)

1 Scientific 4.225E+01

2 Decimal 42.25

3 Engineering 3'-6.25"

4 Architectural 3'-6 1/4"

5 Fractional 42 1/4

Number of decimal places for LUNITS

Specifies the display precision for decimal units. Enter a number between 0 and 8:

- **0** - 0 decimal places, such as 0.
- **1** - 1 decimal places, such as 0.0
- **2** - 2 decimal places, such as 0.00
- **3** - 3 decimal places, such as 0.000
- **4** - 4 decimal places, such as 0.0000
- **5** - 5 decimal places, such as 0.00000
- **6** - 6 decimal places, such as 0.000000
- **7** - 7 decimal places, such as 0.0000000
- **8** - 8 decimal places, such as 0.00000000

This option also affects the display precision of fractions used by architectural and fractional units:

- **0** - 0 fractional precision, such as 1
- **1** - 1/2
- **2** - 1/4
- **3** - 1/8
- **4** - 1/16
- **5** - 1/32
- **6** - 1/64
- **7** - 1/128
- **8** - 1/256

Angular unit mode (AUNITS)

Specifies the display style of angular units. Enter a number:

1 Decimal degrees 90.0

2 Degrees/minutes/seconds 90d0'0

3 Grads 100.00g

4 Radians 1.57r

5 Surveyor's units N 00d0'0"E

There are 400 grads in a circle. There are 2*pi radians (approximately 6.282) in a circle. The N and E in Surveyor's units refers to North and east.

AUNITS variables stores the style of angular units.

Number of decimal places for angular units

Specifies the number of decimal places. Enter a number between 0 and 8.

Angle 0 direction

Specifies the direction for 0 degrees. Enter an angle, or pick two points in the drawing. The default is the positive x axis. You can also rotate the drawing through the Snap command's Rotate option.

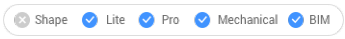
Do you want angles measured clockwise?

Specifies the direction in which to measure angles:

- **Yes** - measures angles clockwise
- **No** - measures angles counterclockwise (default)

26.12 UPDATEFIELD command

Updates the display values of fields.



Icon: 

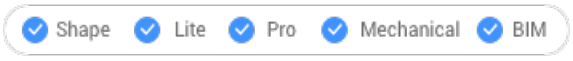
26.12.1 Description

Fields are lines of text that change as conditions in drawings change.

UPDATEFIELD forces selected field text to update, to reflect the changes.

26.13 URL command

Opens the default Web browser.



Icon: 

26.13.1 Description

Opens the default Web browser to browse the internet starting from a specified URL. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

27. V

27.1 VBAIDE command

Opens the Microsoft Visual Basic editing window for writing and debugging VBA code (short for "visual basic for applications integrated development environment").

Shape Lite Pro Mechanical BIM

Icon:

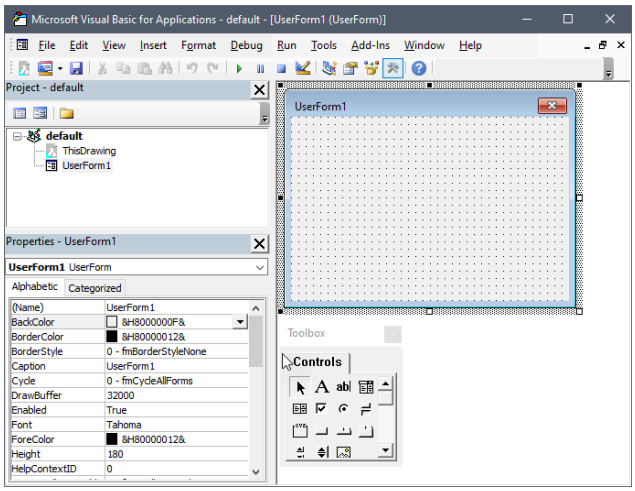
Alias: VBA

Keyboard Shortcuts: **Alt+F11**

Note: Command only available on the Windows platform.

27.1.1 Description

Displays a window:



27.1.2 Option within the command

See the help files provided by Microsoft for Visual Basic.

27.2 VBALOAD command

Opens the Open dialog box.

Shape Lite Pro Mechanical BIM

27.2.1 Description

Opens the Open dialog box to select a dvb or vbi file to load. After the project is loaded, use the VBARUN command to run macros within the project.

27.3 -VBALOAD command

Loads VBA projects.

Shape Lite Pro Mechanical BIM

27.3.1 Description

Loads a VBA project; works in the Command line (short for “Visual Basic for Applications”).

27.4 VBAMAN command

Opens the VBA manager dialog box.

Shape Lite Pro Mechanical BIM

27.4.1 Description

Opens the VBA manager dialog box to manage VBA projects.

27.5 VBANEW command

Begins a new VBA project.

Shape Lite Pro Mechanical BIM

27.5.1 Description

Starts a new VBA project (short for “Visual Basic for Applications”).

27.6 VBARUN command

Opens the Run BricsCAD VBA macro dialog box.

Shape Lite Pro Mechanical BIM

27.6.1 Description

Opens the Run BricsCAD VBA macro dialog box to run and manage VBA macros.

27.7 -VBARUN command

Runs VBA macros.

Shape Lite Pro Mechanical BIM

27.7.1 Description

Runs a VBA macro defined in a DVB (drawing visual basic) file (short for ”Visual Basic for Applications”). This command is meant for use by macros.

27.8 VBASECURITY command

Opens the Security dialog box.

Shape Lite Pro Mechanical BIM

27.8.1 Description

Opens the Security dialog box to specify if VBA macros are allowed to run automatically.

27.9 VBAUNLOAD command

Unloads VBA projects.

Shape Lite Pro Mechanical BIM

27.9.1 Description

Unloads DVB (drawing visual basic) project files from the program (short for “Visual Basic for Applications”).

27.10 VERSIONCONTROL command

Enables multi-user collaboration.

Shape Lite Pro Mechanical BIM

27.10.1 Description

Enables multi-user collaboration bystoring and managing projects in the cloud.Drawings are stored in the cloud usingBricsys24/7 and checked in and out from each user's local machine.

Note: This is Beta functionality and you must request access by submitting a support request following this procedure:

1 Log into your Bricsys account.

2 Choose New Support Request.

3 Open the BricsCAD menu and choose **BricsCAD** > **Version Control**.

4 In the Subject field enter: Access to VERSIONCONTROL Beta.

5 Fill in the rest of the fields as needed.

6 Choose Send Support Request.

27.10.2 Method

The first time you launch VERSIONCONTROL in a new BricsCAD session and choose one of the options, it prompts you to log into your Bricsys account.

If your account has not been granted access to the beta functionality, it directs you to a web page with instructions to request access.

If your account has access to the beta functionality, your successful login is indicated in the Command window and it prompts you to enter the name of the project on which you want to work.

27.10.3 Options within the command

Init
Creates a new multi-user project in Bricsys 24/7. If you are not already logged into your Bricsys account, the Login dialog box displays.

CheckOut
Checks out a drawing from your Bricsys 24/7 projects.

Project name
Specify the name for the new project/for the project from which you want to check out a drawing. If the project name already exists, you are prompted to specify a different project name.

Note: Keep in mind that project names are case-sensitive.

Version control folder
Specify the folder on your local machine to set under version control. The default location is *drive:\Users\username\Documents\Bricsys247\projectname*. However, you can specify any location.

- If the location doesn't exist, it's created and set under version control.
- If the location already exists, it's set under version control.
- If the location is already under version control, you are prompted to specify a different location.

CheckIn
Checks in drawings from your local version control folder to the associated Bricsys 24/7 project.

File to Commit
Specify if you want to check in all files in the version control folder or you commit modified and untracked files.

Check in Message
Enter a relevant Check In message.

Update
Syncs project drawings from your local folder with the associated Bricsys 24/7 project.

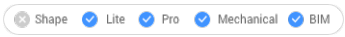
ReName
Renames a drawing that is under version control.

History
Enables you to go to older versions.

ReVert
Reverts to previous state.

27.11 VIEW command

Opens the Drawing explorer dialog box with **Views** selected.

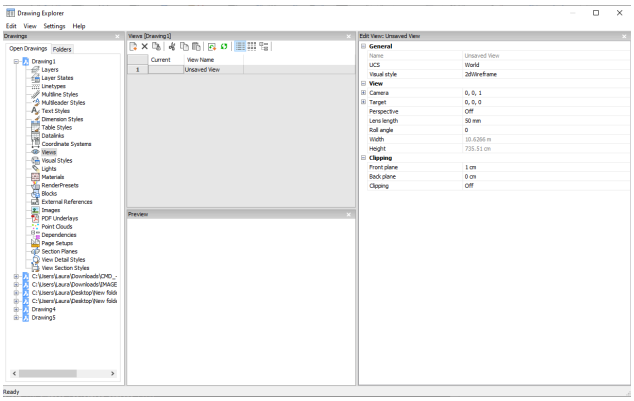


Icon: 

Alias: DDVIEW, EXPVIEWS, V

27.11.1 Description

Opens the Drawing explorer dialog box with **Views** section selected to view and modify views in the current drawing.



27.11.2 Options within the command

General

Name
Sets the name of the current view.

UCS
Assigns a UCS. When the drawing contains no saved UCS's, the droplist shows <None>. See the UCS and EXPUCS commands.

Background
Sets the background for the view. See the BACKGROUND command.

Clip Display
Assigns a clip display.

Layer Snapshot
Toggles whether to assigned the current on and off settings of layers to the view.

- Yes: assigns the current visual state of layers, so that the next time you make this view current, the layers turn themselves on or off.
- No: does not assigns the current visual state of layers, so that the next time you make this view current, the layers do not turn themselves on or off.

Visual Style
Sets the visual style. See the VISUALSTYLES command.

View

Camera
Specifies the camera point of the view in x,y,z coordinates in visual perspective mode.

Target
Specifies the target point of the view in x,y,z coordinates in visual perspective mode.

Perspective
Toggles perspective view:

- On: view is displayed in visual perspective mode
- Off: view displays in parallel perspective mode

A camera glyph is displayed in the drawing for perspective views (see the CAMERA command).

Lens Length
Sets the lens length of the camera.

Roll Angle
Rotates the camera around the view axis.

Width
Specifies the width of the view in current units.

Height
Specifies the height of the view in current units.

Clipping

Front Plane
Sets the distance between the target point and the front clipping plane.

Back Plane
Sets the distance between the target point and the back clipping plane.

Clipping
Toggles the clipping planes, which remove from any entities that are outside the planes.

27.11.3 Context Menu Options

New
Creates new named views.

Delete
Erases the named view from the drawing without warning.

Rename
Renames the view.

Select All
Selects all view definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected view as current.

27.12 -VIEW command

Controls named views.

Shape Lite Pro Mechanical BIM

Alias: -V

27.12.1 Description

Creates and sets named views in the current viewport, and deletes named views from the drawing.

Every viewport can show a different view in model and paper space.

27.12.2 Options within the command

Enter ? to list the existing views in the current drawing.

Enter Delete, Save and Restore options to perform the appropriate action.

Orthographic
Set a standard orthographic view point in the current viewport -- either the Top, Bottom, Front, BAck, Left, or Right view.

TWist
Rotate the current view about the Z axis in the direction set by the ANGDIR variable.

Window
Save a windowed area as a named view.

27.13 VIEWBASE command

Generates associative orthographic and isometric views of 3D models in paper space layouts.

Shape Lite Pro Mechanical BIM

Icon: 

Note:

- This command is valid in Model Space only.
- Use the Tab key to select obscured entities.
- When the GENERATEASSOCVIEWS (Generate associative drawings) system variable is ON, associative dimensions for generated views are updated automatically when the 3D model is modified.
- Setting the GENERATEASSOCATTRS (Generate associative attributes) system variables ON, allows VIEWBASE to generate drawings in which dimensions and tags are updated automatically when the 3D model has been modified.
- DRAWINGVIEWQUALITY variable defines the quality of drawing views.
- The DRAWINGVIEWFLAGS system variable allows to enable parallel creation or updating of drawing views. This can reduce view processing time, but uses more resources.
- This command can be entered transparently during commands ('viewbase).

27.13.1 Method

Select one or more entities (3D solids, blocks, components) or press Enter to select all 3D entities in the Model Space from which to generate the drawing views in a layout tab or choose an option. Hit the Tab key to select nested entities.

Enter the name of a new or existing layout or press Enter to accept the current layout.

The command switches to the layout tab where a point must be picked to define a position for the base view or enter an option.

Select the position for each projected view by moving the cursor. Depending on the position of the cursor with respect to the base view one of five orthogonal views (top, left, right, back and bottom) and four isometric views can be inserted. The views are aligned automatically depending on the selected projection type (see above).

Tap the Ctrl key to toggle alignment ON and OFF. When it is off, you can place the current view in any location.

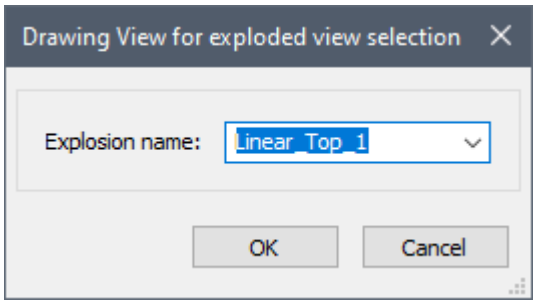
27.13.2 Options within the command

Entire model
Select all the 3D entities in model space.

preseTs
Specifies the types of generated drawings and their placement in the layout; displays the **Drawing View Presets** dialog box .
The selected preset is saved through the DRAWINGVIEWPRESET system variable.

Special views
Select a style of exploded representations, if any exist in the drawing.

Exploded view
Creates an exploded drawing view if an exploded representations exists in the drawing.
From the drop-down list, choose a style for the exploded representation.



Unfolded view
Creates an unfolded drawing view for solids with associated unfolded view.

Back
Returns to the previous prompt.

Scale
Sets the Scale property of the paper space viewports for the various views:

fit 4 views
The scale is adjusted to fit the four standard orthographic views: Front, Top, Left, Right. The Front view (base view) is defined by the Orientation option.

fit 9 views
Adjusts the scale to fit five orthographic views and four isometric views.

fit 5 views
Adjusts the scale to fit five orthographic views: Front, Top, Left, Right, Back.

fit 10 views
Adjusts the scale to fit six orthographic views and four isometric views.

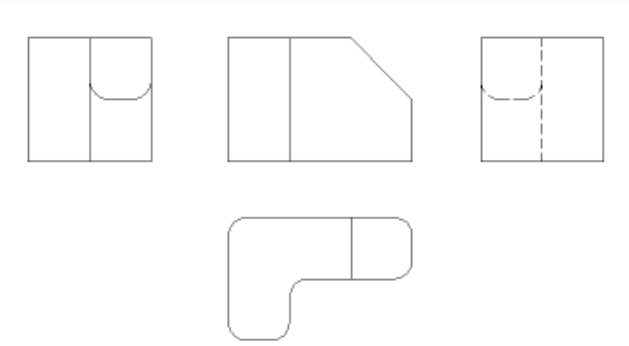
Standard scales
Displays the scale list as maintained by the SCALELISTEDIT command; select a scale from the list.

Custom scales
Prompts you to type a scale in the Command line.

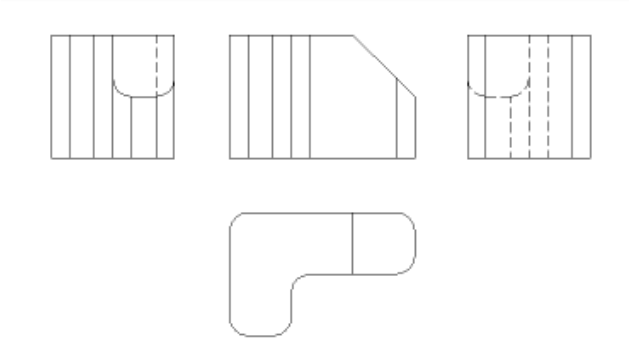
Hidden Lines
Controls the visibility of hidden lines.
When the BM_Ortho_Hidden and BM_Isometric_Hidden layers are off or frozen, hidden lines will not display.

Tangent Lines
Toggles whether tangent edges between tangent faces are created. Visible tangent edges are created on the BM_Tangent_Visible layer. Hidden tangent edges are created on the BM_Tangent_Hidden layer.

No
Does not display tangent lines.



Yes
Displays the tangent lines.



Note: Tangent edges are always created for isometric views. Freeze or switch off the display of the BM_Tangent_Visible layer to hide tangent edges in isometric views.



INterferance edges
Toggles between ON and OFF the visibility of interference edges between solids which intersect with each other. When it is ON, a line is drawn where the solids are met.

trailiNg lines
Controls the visibility of trailing lines.
To create exploded views, use the BMEXPLODE command; the model must have inserts of mechanical components to use this command.

Orientation
Defines the base view orientation. Rotates the 3D model so that the main view is projected on the vertical projection plane (V.P.).

Projection type
Defines the layout of the views.

First angle
It is also known as European projection.

Third angle
It is also known as American projection.
The default projection type depends on the value of the MEASUREMENT system variable.

Isometric Geometry
Defines the style for the isometric views: rendered 3D view or 2D drawing:

2D views
Draws isometric views as 2D drawings.

3D views
Draws isometric views as 3D solids with the Conceptual visual style applied.

sElect
Selects additional entities to include or exclude.

Remove
Removes entities from drawing views.

Entire model
Includes all entities from model space in the drawing views.

Layout
Opens the previous layout with updated drawing views.

27.14 VIEWBREAK command

Creates a broken view on drawing views generated by the VIEWBASE command in a paper space layout.

Shape Lite Pro Mechanical BIM

Icon:

Note: This command operates only in Paper Space.

Note: This command can be entered transparently during commands (‘viewbreak)

27.14.1 Method

Select the drawing view from which to generate the broken view by clicking inside the drawing view. Select the first and the second points which specifies the first and the second planes of the cuts.

The default direction of the symbol is based on the size of the viewport:

- Vertical if the viewport is wider than high.
- Horizontal if the viewport is higher than wide.

27.14.2 Options within the command

Type
Allows to set the broken symbol type:

sTraight
Line geometry. Supports the Gap distance property.



Spline
Spline geometry. Supports the Gap distance, Width and Height.



Zigzag
Spline geometry. Supports the Gap distance, Width and Height properties.



sMall zigzag
Supports the Gap distance, Width, Height and Step properties.



Current
Use the last used broken symbol type.

Note: The properties of the Broken Symbol can be edited in the Properties panel:

Broken Symbol	
Style	Small Zigzag
Gap distance	5 mm
Type	Small Zigzag
Overshoot	3 mm
Width	5 mm
Height	5 mm
Step	20 mm

Style/Type
Select the style / type in the drop-down list.

Gap distance
Defines the distance* between the two parts of the broken symbol.

Overshoot
Defines the length of the extends for the break lines outside of the 2D view.

Width
Defines the width* of the shape symbol in the symbol direction.

Height
Defines the height* of the shape symbol in the direction orthogonal to the symbol direction.

Step
Defines the distance* between the small zigzag symbols.

* Expressed in layout units.

Vertical
Aligns the symbol along the Y axis.



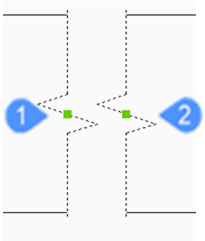
Horizontal
Aligns the symbol along the X axis.



27.14.3 Edit Grips

You can edit break symbol grips.

Select the break symbol and 2 Grips display:



1 Controls the position of the first break plane in the model space

2 Controls the position of the second break plane in the model space

27.15 VIEWDETAIL command

Draws view details of drawing views made with the VIEWBASE command.

Shape Lite Pro Mechanical BIM

Icon:

Note:

- This command operates only in Paper Space.
- When the GENERATEASSOCVIEWS (Generate associative drawings) system variable is ON, associative dimensions for detailed views are updated automatically when the 3D model is modified.
- DRAWINGVIEWQUALITY variable defines the quality of drawing views.
- The AUTOVPFITTING variable controls whether the size of the viewport is adjusted automatically to fit the current extents of the 3D geometry. By default AUTOVPFITTING = ON.

27.15.1 Method

Select the drawing view from which to extract the detail by clicking inside a drawing view. The program highlights the selected view.

After that, specify the center point of the detail view by picking a point inside the parent view and the style of the view boundary.

There are two boundary types from which you can choose:

- Circular - draws a circular boundary.
- Rectangular - draws a rectangular boundary.

Note: You may always switch the boundary type between circular and rectangular using the Boundary option.

Specify the position for detail view by picking a point in the layout, away from the parent view.

27.15.2 Options within the command

Scale
Specifies the scale of the detail view, which by default is twice the parent viewport scale:

Standard scales
Choose a standard scale in the list; the list can be edited by the SCALELISTEDIT command.

Custom
Specify a custom scale factor.

Relative custom scale
Calculates the scale factor relative to parent view by multiplying the scale factor of the source view with this number.

from Parent
Sets the scale of the section view equal to the scale of the parent view.

Hidden lines
Control the visibility of hidden lines, or uses the same hidden lines setting as the parent view.

Note: When the BM_Ortho_Hidden and BM_Isometric_Hidden layers are off or frozen, hidden lines will not display.

Tangent lines
Controls the display of tangent edges which appear in the transition from a flat face and a curved face, such as with fillets:

Note: When the BM_Tangent_Visible layer is off or frozen, tangent lines are not display.

anChor
Determines if the center of the viewport is anchored so that the viewport grows and shrinks around its center point or not.

Geometry
Selects the visual style for the section view:

2D
Section view uses the 2dWireframe visual style.

3D
Section view uses a rendered visual style. This is Conceptual by default. Use the Properties panel to choose a different visual style.

Annotation
Determines the annotations to use.

Identifier
Specifies the view detail identifier by entering a name for the view detail.

Label
Toggles the display of the view detail label.

Boundary
Toggles the style the boundary between circular and rectangular.

Circular
Changes the boundary to a circle.

Rectangular
Changes the boundary to a rectangle.

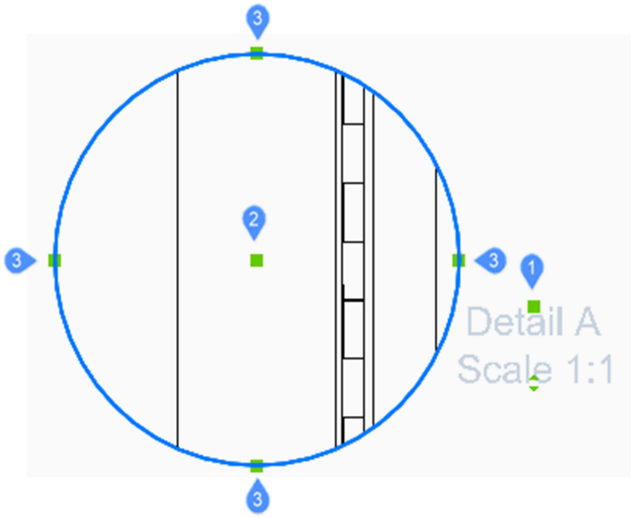
modeL edge
Controls whether a connection line is drawn between the detail view and the detail boundary in the parent view.

smooth with Border
Does not draw a line.

smooth with Connection line
Draws a connecting line from the detail view to the parent view.

27.15.3 Grips Editing

Detail views can be edited through grips: select the view and six grips display:



1 Controls the position of the identifier.

2 Controls the position of the section detail.

3 4 grips control the size of the detail boundary.

27.16 VIEWDETAILSTYLE command

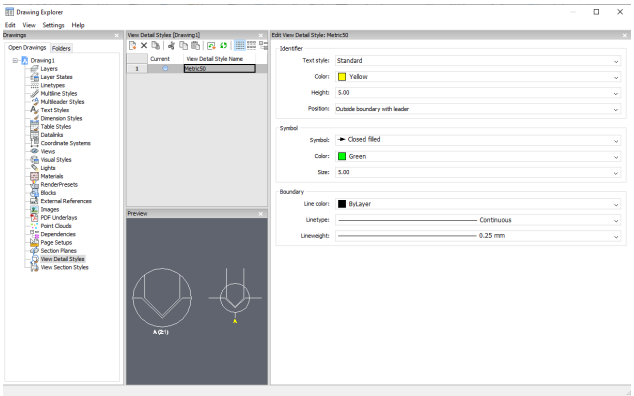
Opens the Drawing explorer dialog box with **View Detail Styles** selected.

Shape Lite Pro Mechanical BIM

Icon:

27.16.1 Description

Displays the Drawing explorer dialog box with the View Detail Style category selected to create and modify view detail styles.



27.16.2 Options within the command

Identifier

Defines the properties of the view detail identifier.

Text style
Specifies the text style used by the identifier text.

Color
Specifies the color of the identifier.

Height
Specifies the height of the identifier.

Position
Determines where the identifier is located.

Outside boundary
Places the identifier outside the detail's boundary. No symbol is used.

Outside boundary with leader
Places the identifier outside the detail's boundary and draws a leader from the parent view to the detail. No symbol is used

On boundary
Places the identifier on the detail's boundary.

On boundary with leader
Places the identifier on the detail's boundary and draws a leader from the parent view to the detail.

Symbol
Defines the properties of the symbol.

Symbol
Specifies the look of the symbol.

Color
Specifies the color of the symbol.

Size
Specifies the size of the symbol.

Boundary
Defines the properties of the detail viewport boundary.

Line color
Specifies the color of the boundary line

Linetype
Specifies the line type of the boundary.

Lineweight
Specifies the weight of the boundary line.

27.16.3 Context Menu Options

New
Creates new named view detail styles.

Delete
Erases the named view detail style from the drawing without warning.

Rename
Renames the view detail style.

Select All
Selects all view detail styles definitions.

Invert selection
Deselects the current selection and vice versa.

Set current
Sets the selected view detail style as current.

27.17 VIEWEDIT command

Changes the scale and hidden line visibility of a drawing view(s) if the view(s) was created with the VIEWBASE command.

Shape Lite Pro Mechanical BIM

Icon: 

Note: This command operates only in Paper Space.

27.17.1 Method

Select the drawing view(s) for modifying by clicking inside one or more drafting viewports or press Enter to select all drawing views in the current layout.

The next prompt will vary depending on the selected view type. When different view types are selected, only the shared options are available.

27.17.2 Options within the command

Standard view options: see the VIEWBASE command

Section view options: see the VIEWSECTION command

Detail view options: see the VIEWDETAIL command

27.18 VIEWEXPORT command

Exports drawing views.

Shape Lite Pro Mechanical BIM

Icon: 

27.18.1 Description

Exports drawing views generated by the VIEWBASE , VIEWSECTION or VIEWDETAIL commands to the model space of the current drawing, via the Clipboard to a new drawing or as a DWG/DXF file on disc.

27.18.2 Method

Click inside one or more viewports or select All views and choose the destination of the views (see options).

27.18.3 Options within the command

Clipboard
Exports the selected view(s) to the clipboard, which can then be pasted into any drawing.

File
Exports the selected views to a DWG or DXF file; displays the Save Exported Views dialog box.

Model space
Exports the view to the current drawing's model space.

Note: Specify if remove exported views:

- Yes - the viewports of exported views are removed from the layout.
- No - viewports of exported views are kept in the layout.

27.19 VIEWHORIZONTAL command

Rotates viewpoint to the horizontal.

Shape Lite Pro Mechanical BIM

27.19.1 Description

Sets the Z component of the 3D viewpoint to 0 in the current UCS, so that the viewpoint rotates to the horizontal.

This command works like setting Z to 0 with the VIEWPOINT command.

27.20 VIEWLABEL command

Shape Lite Pro Mechanical BIM

27.20.1 Description

This is a service command which is not supposed to be entered directly by the user. It is used by the program when a context menu item is selected.

27.21 VIEWPOINT command

27.22 VIEWPROJ command

Generates additional projected views from an existing drawing view previously created by the VIEWBASE command (short for "view projection").

Shape Lite Pro Mechanical BIM

Icon: 

Note:

- This command operates only in Paper Space.
- The DRAWINGVIEWFLAGS system variable allows to enable parallel creation or updating of drawing views. This can reduce view processing time, but uses more resources.

27.22.1 Method

Select the view from which to generate new projected views and select the position of the new projected views.

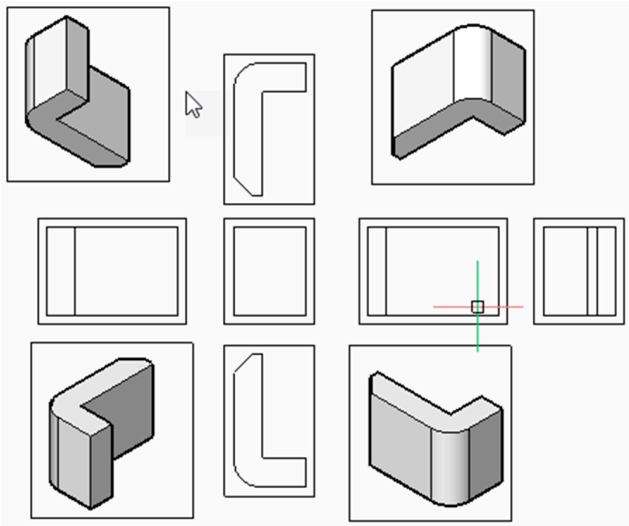
Depending on where you move the cursor, you can place up to five orthogonal and four isometric views. The views are aligned automatically. Press Ctrl key to toggle alignment on and off to place the view anywhere on the sheet.

27.22.2 Options within the command

Isometric geometry
Sets the style for isometric views.

2D views
Draws isometric views as 2D projections of the 3D geometry and turns on the BM_Hidden layer so that hidden lines are shown as dashed lines.

3D views
Draws isometric views as 3D geometry and sets the visual style of the viewport.



27.23 VIEWRES command

Sets viewing resolution.

Shape Lite Pro Mechanical BIM

27.23.1 Description

Sets the view resolution for curved entities; also toggles fast-zoom mode (short for "view resolution").

27.23.2 Options within the command

Choose whether to use redraws or regenerations for view changes, such as zooms.

Choose the level of quality for displaying 2D curves, such as circles and arcs (range 1-20000)

- 1 - curves like circles and arcs may look like polygons, but display at a fast speed.

- 100 - default value.
- 20000 - curves almost always look round, but at a slower display speed.

27.24 VIEWSECTION command

Creates cross-section views of drawing views generated with the VIEWBASE command in a paper space layout.

Shape Lite Pro Mechanical BIM

Icons: 

Note:

- This command operates only in Paper Space.
- When the GENERATEASSOCVIEWS (Generate associative drawings) system variable is ON, associative dimensions for section views are updated automatically when the 3D model is modified.
- If the SECTIONABLE property of a mechanical component is OFF, the component appears non-sectioned in the section views of the Full section type.
- DRAWINGVIEWQUALITY variable defines the quality of drawing views.
- The AUTOVPFITTING variable controls whether the size of the viewport is adjusted automatically to fit the current extents of the 3D geometry. By default AUTOVPFITTING = ON.
- This command can be entered transparently during commands ('viewsection)

27.24.1 Method

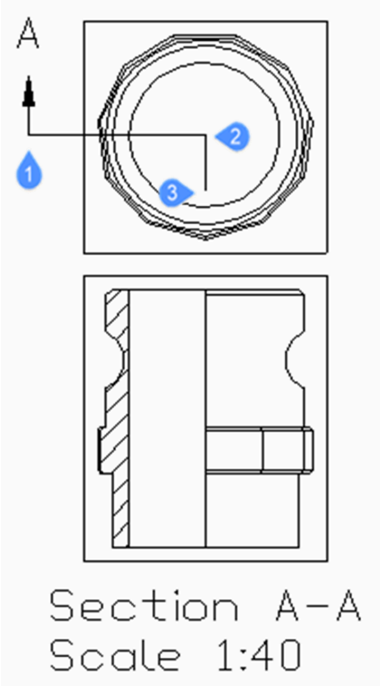
Select the drawing view to generate the section by clicking inside a drawing view. The program highlights the selected view. Create a section and choose a location for the result view.

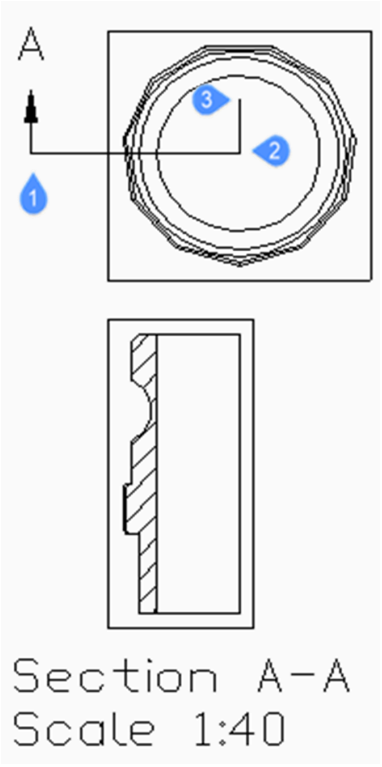
27.24.2 Options within the command

Select type
Control the shape of the section plane:

Full
The section line defines an infinite plane that cuts through the entire model.

Half
The section line defines a half-plane that cuts part of the model. You must specify by clicking the start point (1) of section line, the second point (2) to define the limit of the half plane and the third point (3) to define the view direction.
The position of the third point determines whether the part of the model that is not cut displays (left) or not (right).





Offset

The section line defines a series of cutting regions located on different offsets from each other. Enter Done to complete the section line.

Aligned
The section line defines a polyline, where every segment defines a cutting region. The resulting section will have the length equal to the sum of the lengths of cutting regions. Enter Done to complete the section line.

Scale
Specifies the scale of the detail view, which by default is twice the parent viewport scale.

Standard scales
Choose a standard scale from the list; the list can be edited by the SCALELISTEDIT command.

Custom
Specify a custom scale factor.

from Parent
Sets the scale of the section view equal to the scale of the parent view.

Hidden lines
Control the visibility of hidden lines or : uses the same hidden lines setting as the parent view.

Note: When the BM_Ortho_Hidden and BM_Isometric_Hidden layers are off or frozen, hidden lines will not display.

Tangent lines
Controls the display of tangent edges which appear in the transition from a flat face and a curved face, such as with fillets.

Note: When the BM_Tangent_Visible layer is off or frozen, tangent lines are not display.

anChor
Determines if the center of the viewport is anchored so that the viewport grows and shrinks around its center point or if the geometry is fixed.

Geometry
Selects the visual style for the section view.

2D
The section view uses the 2dWireframe visual style.

3D
The section view uses a rendered visual style. This is Conceptual by default. Use the Properties panel to choose a different visual style.

Annotation
Determines the annotations to use.

Identifier
Specifies the view detail identifier by entering a name for the view detail.

Label
Toggles the display of the view detail label.

Depth
Specifies the view depth of a section view.

Full
Sets the depth to the extents of the model (maximum view depth).

Custom
Limits the view depth by entering the depth distance (type a positive value or move the cursor to define the view depth dynamically).

Projection
Determines how the section is projected.

Normal
Draws the section as a union of projections from every cutting region (each segment of the section polyline) in its normal direction.

Orthogonal
Draws the section in the normal direction of the first section region (the first segment of the section polyline).

Rotate view
Allows you to rotate the drawing view viewport but does not rotate the frame.

Horizontal
Rotates a segment horizontally.

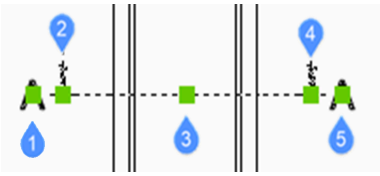
Vertical
Rotates a segment vertically.

Custom angle
Allows you to specify the angle of alignment.

27.24.3 Grips Editing

You can edit section lines with grips.

Selecting either the section line, one of the identifiers or an arrow, 5 Grips display:



1 Controls the position of the first identifier.

2 Defines the start point of the section line.

3 Allows you to move the section line.

4 Defines the endpoint of the section line.

5 Controls the position of the second identifier.

27.25 VIEWSECTIONSTYLE command

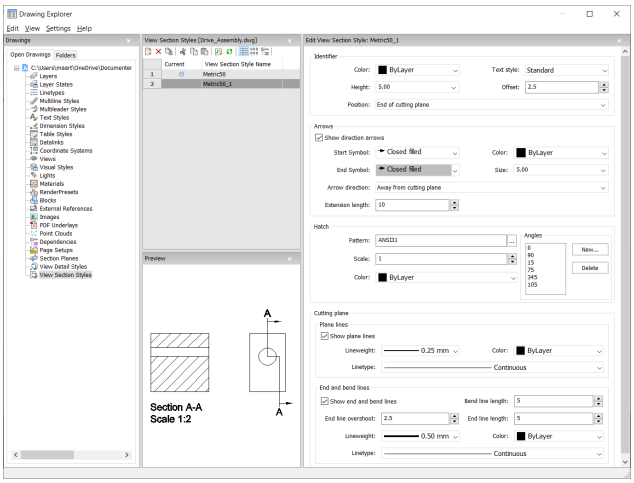
Opens the Drawing explorer dialog box with **View Section Styles** selected.

Shape Lite Pro Mechanical BIM

Icon:

27.25.1 Description

Opens the Drawing explorer dialog box with **View Section Styles** category selected to view and modify view section styles in the current drawing.



27.25.2 Options within the command

Identifier

Sets the properties of the section identifier.

Color

Specifies the color of the identifier.

Text Style

Specifies the text style used by the identifier text. To employ a different text style, use the STYLE command to create it.

Height

Specifies the height of the identifier.

Offset

Specifies the distance from the end of the cut lines to the arrow.

Position

Determines where the identifiers are located.

- End of cutting plane: places the identifiers at the ends of the cutting lines.
- Above direction line: places the identifiers outside the detail's boundary and draws a leader from the parent view to the detail. No symbol is used.
- On boundary: places the identifiers on the detail's boundary.
- On boundary with leader: places the identifier on the detail's boundary and draws a leader from the parent view to the detail.

Arrows

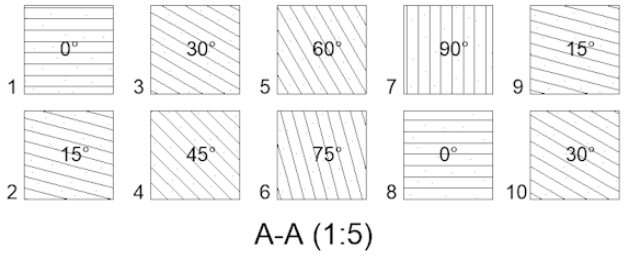
Sets the properties of the section arrows.

- Show direction arrows: toggles the display of the arrows
- Start Symbol: specifies the look of the start symbol
- Color: specifies the color of the arrow
- End Symbol: specifies the look of the end symbol
- Size: specifies the size of the arrow
- Arrow direction: points the arrow towards or away from the cutting line
- Extension length: determines the length of the "dimension" line on the arrows

Hatch

Sets the properties of the section hatching.

- Pattern: choose a pattern name from the drop-down list.
- Browse: displays the Hatch Pattern Palette dialog box, from which you choose a pattern visually.
- Angles: sets the angle for the pattern, one angle for subsequent cuts. The first angle in the list is applied to the first solid, the second angle to the second, and so on. When there are more solids to cut than angles listed, then the program starts again from the first angle. See figure below.



- New: adds angles to the list. Displays the New Hatch Angle dialog box, in which you enter an angle.
- Delete: deletes the selected angle from the list.
- Scale: specifies the scale of the pattern.
- Color: specifies the color of the pattern.

Cutting Plane | Plane Lines

Sets the properties of the cutting plane lines.

- Show plane lines: toggles the display of the lines.
- Lineweight: specifies the weight of the lines.
- Color: specifies the color of the lines.
- Linetype: specifies the pattern of the lines.

Cutting Plane | End and Bend Lines

Sets the properties of the end and bend lines.

- Show end and bend lines: toggles the display of the lines.
- Bend line length: specifies the distance the bend line offsets from one cut line to another.
- End line overshoot: specifies the distance from the section to the end of the plane line.
- End line length: specifies the length of the end lines.

- Lineweight: specifies the weight of the lines.
- Color: specifies the color of the lines.
- Linetype: specifies the pattern of the lines.

27.25.3 Context Menu Options

New
Creates a new view detail style as a copy of the currently selected style.

Delete
Removes the selected style from the drawing.

Rename
Renames the selected view section style.

Select All
Selects all view section style definitions.

Invert selection
Deselects the current selection and vice versa.

27.26 VIEWUPDATE command

Updates drawing views.

Shape Lite Pro Mechanical BIM

Icon: 

Note: The DRAWINGVIEWFLAGS system variable allows to enable parallel creation or updating of drawing views. This can reduce view processing time, but uses more resources.

27.26.1 Description

Manually updates the selected or all drawing views created by the VIEWBASE and VIEWSECTION commands when automatic updates (VIEWUPDATE) are turned off.

27.26.2 Options within the command

Select drawing views
Updates the selected views.

All views
Updates all views in the current layout.

27.27 VISIBILITYSTATES command

Creates and edits visibility states in the Command line. It also opens the **Visibility States** panel.

Shape Lite Pro Mechanical BIM

Icon: 

Note: BricsCAD® visibility states are not compatible with AutoCAD® visibility states. When you open a drawing with BricsCAD® parametric blocks in AutoCAD®, they are displayed as they were saved in BricsCAD®. However, when such parametric blocks are edited or copied to another drawing in AutoCAD®, all entities in the block become visible.

27.27.1 Options within the command

New parameter
Creates a new parameter.
Type a name for the parameter (<P>), then press Enter.

Enter new state for <P>
Type a name for the new parameter state (<S>).

Add entities
Adds entities to the specified state of a parameter.

Select entities for <P>=<S> [selection options (?)]
Select entities for the new state using any selection method.
Press Enter to stop selecting.

When the parameters and states already exist in the drawing, the existing parameters and states are added to the prompts:

New parameter or [<P> (1)/ [<P> (2) …]
Do one of the following:

- Type a name for a new parameter.
- Type the number of the parameter you want to edit.

Enter new state for [<P> or [<S> (1)/ [<S> (2) …]
Do one of the following:

- Type a name for a new state of the parameter.
- Type the number of the state you want to edit.

Remove entities
Removes entities from the specified state of a parameter.

make entities Visible
Makes entities visible to the specified state of a parameter.

make entities Invisible
Makes entities invisible to the specified state of a parameter.

Note:

<P> is a placeholder for the name of the visibility parameter entered in the previous step.

<S> is a placeholder for the name of the active visibility state of the parameter <P>. On the screen, the actual names of the parameter and the state will be displayed instead of <P> and <S>.

27.28 VISIBILITYSTATESADD command

Adds entities to the active state of the selected visibility parameter.

Shape Lite Pro Mechanical BIM

27.28.1 Methods

The VISIBILITYSTATESADD command adds entities to a selected visibility parameter. The entities added will be visible for the active state of the selected parameter.

27.28.2 Options within the command

Specify parameter name
Asks for the name of an existing visibility parameter.

Select entities to add to <P>=<S> [selection options (?)]:
Asks for a selection of entities to be added to the parameter <P>. These entities will be visible for the state <S>.
<P> is a placeholder for the name of the visibility parameter entered in the previous step. <S> is a placeholder for the name of the active visibility state of the parameter <P>. On the screen, instead of the texts <P> and <S>, the actual names of the parameter and the state will be displayed.
For more information on viewing and managing the visibility parameters and states see the article Visibility states panel.

27.29 VISIBILITYSTATESHIDE command

Hides entities for the active state of the selected visibility parameter.

Shape Lite Pro Mechanical BIM

27.29.1 Methods

The VISIBILITYSTATESHIDE command hides entities for the active state of the selected parameter.

27.29.2 Options within the command

Specify parameter name
Asks for the name of an existing visibility parameter.

Select entities to make invisible for <P>=<S> [selection options (?)]:
Asks for a selection of entities to be hidden when the state <S> is active.
<P> is a placeholder for the name of the visibility parameter entered in the previous step. <S> is a placeholder for the name of the active visibility state of the parameter <P>. On the screen, instead of the texts <P> and <S>, the actual names of the parameter and the state will be displayed.
For more information on viewing and managing the visibility parameters and states see the article Visibility states panel.

27.30 VISIBILITYSTATESPANELCLOSE command

Closes the Visibility states panel.

Shape Lite Pro Mechanical BIM

Icon: 

27.30.1 Methods

The VISIBILITYSTATESPANELCLOSE command closes the Visibility states panel.

For more information on viewing and managing the visibility parameters and states see the article Visibility states panel.

27.31 VISIBILITYSTATESPANELOPEN command

Opens the Visibility states panel.

Shape Lite Pro Mechanical BIM

Icon: 

27.31.1 Methods

The VISIBILITYSTATESPANELOPEN command opens the Visibility states panel.

For more information on viewing and managing the visibility parameters and states see the article Visibility states panel.

27.32 VISIBILITYSTATESREMOVE command

Removes entities from the selected visibility parameter.

Shape Lite Pro Mechanical BIM

27.32.1 Methods

The VISIBILITYSTATESREMOVE command removes entities from the selected visibility parameter.

27.32.2 Options within the command

Specify parameter name

Asks for the name of an existing visibility parameter.

Select entities to remove from <P>=<S> [selection options (?)]:

Asks for a selection of entities to be removed from the parameter <P>.

<P> is a placeholder for the name of the visibility parameter entered in the previous step. <S> is a placeholder for the name of the active visibility state of the parameter <P>. On the screen, instead of the texts <P> and <S>, the actual names of the parameter and the state will be displayed.

For more information on viewing and managing the visibility parameters and states see the article Visibility states panel.

27.33 VISIBILITYSTATESSHOW command

Shows entities for the active state of the selected visibility parameter.

Shape Lite Pro Mechanical BIM

27.33.1 Methods

The VISIBILITYSTATESSHOW command shows entities for the active state of the selected parameter.

27.33.2 Options within the command

Specify parameter name

Asks for the name of an existing visibility parameter.

Select entities to make visible for <P>=<S> [selection options (?)]:
Asks for a selection of entities to be visible when the state <S> is active.
<P> is a placeholder for the name of the visibility parameter entered in the previous step. <S> is a placeholder for the name of the active visibility state of the parameter <P>. On the screen, instead of the texts <P> and <S>, the actual names of the parameter and the state will be displayed.
For more information on viewing and managing the visibility parameters and states see the article Visibility states panel.

27.34 VISIBLE command

Shape Lite Pro Mechanical BIM

27.34.1 Description

This is a service command which is not supposed to be entered directly by the user. It is used by the program when a context menu item is selected.

27.35 VISUALSTYLES command

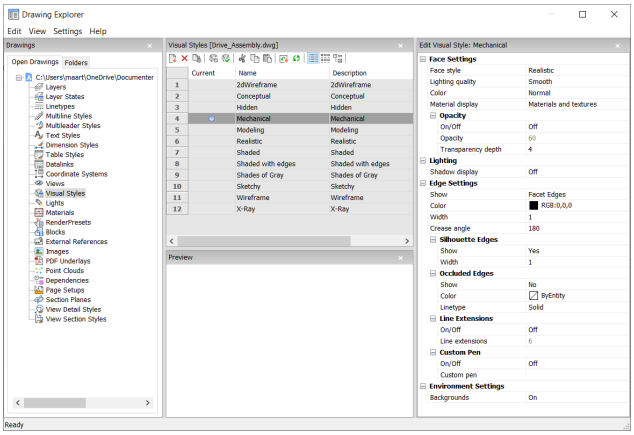
Opens the Drawing explorer dialog box with **Visual Styles** selected.

Shape Lite Pro Mechanical BIM

Icon:

27.35.1 Description

Opens the Drawing explorer dialog box with **Visual Styles** category selected to view and modify visual styles in the current drawing.



27.35.2 Options within the command

Apply visual style
Applies the selected visual style to the current viewport.

Edit visual style
Displays the settings of the selected visual styles. To edit a setting: click the settings field, then type a new value or select an option in the drop down list.

Face Settings
Specifies the look of faces of 3D models.

Face Style
Selects an overall color scheme for the visual style.

Lighting Quality
Specifies the quality of curved faces.

Color
Selects the color mode.

Material Display
Determines whether materials are displayed. Materials are applied with the MATERIALASSIGN command. When Material Display is not off, then highlights and opacity are ignored, as materials provide their own values for them.

Opacity
Specifies the level of transparency of faces.

On/Off
Toggles the opacity on and off.

Opacity
Specifies the level of transparency (not available when materials are on).

Transparency depth
Specifies how far transparency reaches in terms of the number of overlapping entities.

Lighting
Determines whether shadows are displayed.

Shadow display
Specifies how shadows are displayed:

- Off: objects cast no shadows, the preferred option as shadow-casting slows down the program's display speed, so leave it turned off unless you are saving images to file or plotting them.
- Ground shadows: objects cast shadows on the shadow plane, but not on each other
- Full shadows: objects cast shadows on the ground and on each other

Edge Settings
Specifies the style of edge to display.

Show
Specifies how edges are displayed.

- None: neither facets nor isolines nor edges are displayed; this setting turns off many other settings.
- Isolines: isolines and edges are displayed; isolines are the curved lines that simulate the surfaces of curved faces.
- Facet Edges: facets and edges are displayed on entities.

Color
Specifies the color for all edges.

Width
Specifies the width of edge lines.

Number of lines
Specifies the number of isolines drawn on curved surfaces, from 0 to 2047.

Always on top
Determines if all isolines are displayed, or only those that are "on top" (hidden isolines being hidden).

Silhouette Edges
Determines how fat outlines are around models.

Show
Toggles the display of silhouette edges.

Width
Specifies the width of the silhouette edges in the range of 1 to 25 pixels; applies to all entities in the viewport equally.

Occluded Edges
Specifies how to handle occluded (hidden) edges and facets; this setting allows you to display hidden lines in a different color and linetype from visible lines.

Show
Toggles the visibility of occluded edges and facets.

Color
Specifies the color for visible obscured edges and facets. Click Select color to select another color from the Select Color dialog box.

Linetype
Specifies the line type for visible obscured edges and facets. The program does not use the usual linetype patterns, but a separate set. All these settings do not apply to isolines.

Line Extensions
Extends edges beyond their boundaries, also known as "overhang".

On/Off
Turn line extensions on or off.

Line extensions
Specifies the distance over which the lines extend beyond boundaries, such as face edges.

Custom Pen
Determines the use of a custom line drawing style.

On/Off
Turn the drawing style on or off.

Custom pen
Specifies the style of line drawing.

Background Settings
Toggles the display of a background in the viewport.
Backgrounds consist of either a solid color, a gradient of two or three colors, or a raster image, and are set by the BACKGROUND command.

Note: The ANTIALIASSCREEN system variable controls the amount of anti-aliasing (edge smoothing) applied during on-screen rendered mode display. The default value is 1, maximum value is 5. High anti-alias values incur a high calculation cost.

Mechanical Drafting and Mechanical Drafting Section visual styles are used by the BMGENDRAFT and BMGENSECTION commands.

27.35.3 Context Menu Options

New
Creates a new visual style.

Delete
Deletes the selected visual style, except for predefined visual styles such as 2dWireframe, 3D Hidden, 3dWireframe, Conceptual, Realistic… .

Reset to default
Restores the selected visual style(s) to default.

Apply to current viewport
Applies the selected visual style to the current viewport.

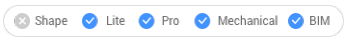
Rename
Renames the selected visual style.

Select All
Selects all visual style definitions.

Invert selection
Deselects the current selection and vice versa.

27.36 -VISUALSTYLES command

Manages visual styles.



27.36.1 Description

The command sets and manages visual styles at the Command line.

27.36.2 Options within the command

set Current
Sets a visual style for the current viewport. The options are 2dwireframe/Wireframe/Hidden/Realistic/ Conceptual/Shaded/shaded with Edges/shades of Grey/SKetchy/X-ray/Other/cUrrent/

Other
Selects a visual style customized by users.

cUrrent
Sets the current visual style.

Save as
Saves the current visual style by a new name when users make changes to the properties of a visual style.

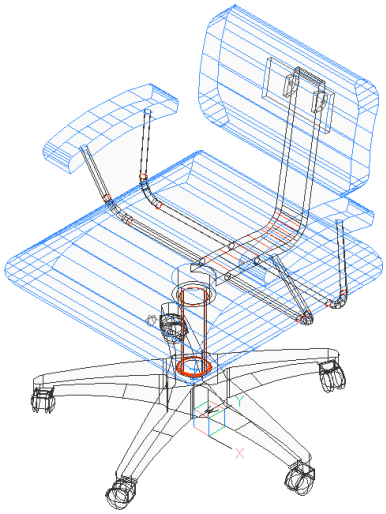
Delete

Deletes a visual style by name. The currently-used style(s) and those defined by the program cannot be deleted.

?

List the names of visual styles available in the drawing:

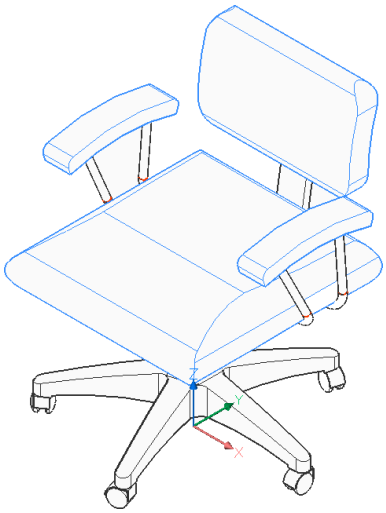
- 2dWireframe



- Conceptual



- Hidden



- Realistic



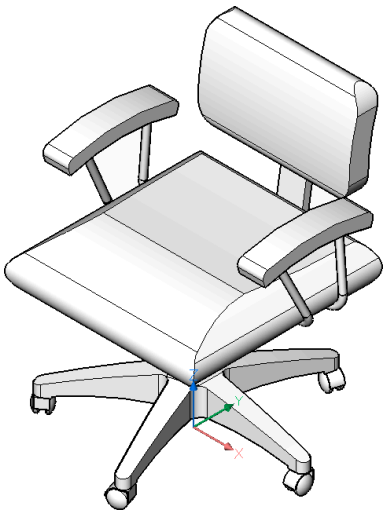
- Shaded



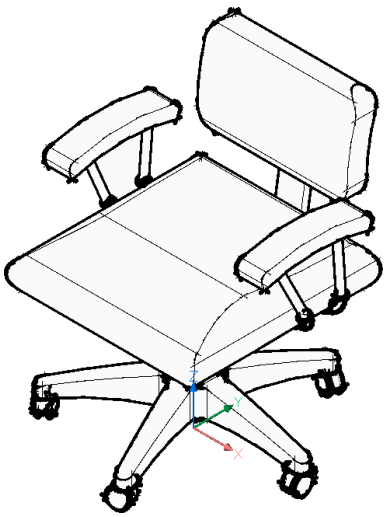
- Shaded with edges



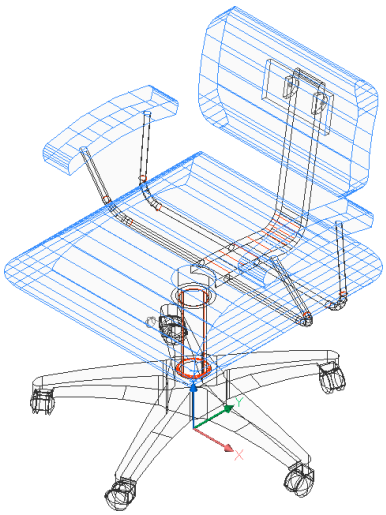
- Shades of Gray



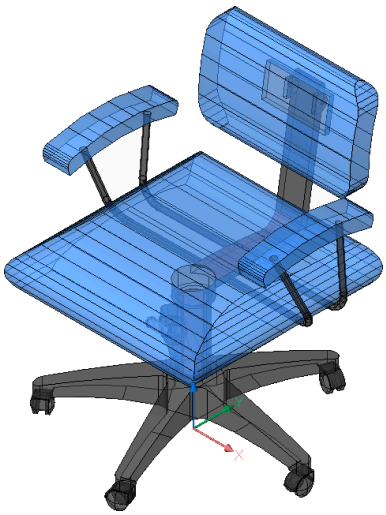
- Sketchy



- Wireframe



- X-Ray



27.37 VLIDE command

Opens the BricsCAD LISP Advanced Development Environment (BLADE).

Shape Lite Pro Mechanical BIM

27.37.1 Description

Opens the BricsCAD LISP Advanced Development Environment (BLADE) to interactively edit and debug LISP applications. It opens in an external application window, allowing it to remain open while you work on your drawings in BricsCAD. You can move and resize it with standard application window controls.

27.38 VMLOUT command

Exports drawings in VML format, embedded in an HTML file (short for "vector markup language").

Shape Lite Pro Mechanical BIM

27.38.1 Description

Opens the Create HTML file dialog box to save data from the current drawing to an HTM file. The data is stored in VML format and embedded in an HTML file. You can view the file in a Web browser. However, you may need to install a VML plug-in.

The output is in HTML format and looks something like this:

```
<html xmlns:v="urn:schemas-microsoft-com:vml">
<head>
  <object id="VMLRender" classid="CLSID:10072CEC-8CC1-11D1-986E-00A0C955B42E"></
object>
  <style>
    v\:* { behavior: url(#VMLRender); }
  </style>
</head>
<body>
 <v:group id="AN00001_" style="width:8in;height:8in;" coordSize="1600,1600">
 <v:shape style="width:1600;height:1600" path="nf m 214,42 l 213,41 214,41 e"/>
```

27.39 VPCLIP command

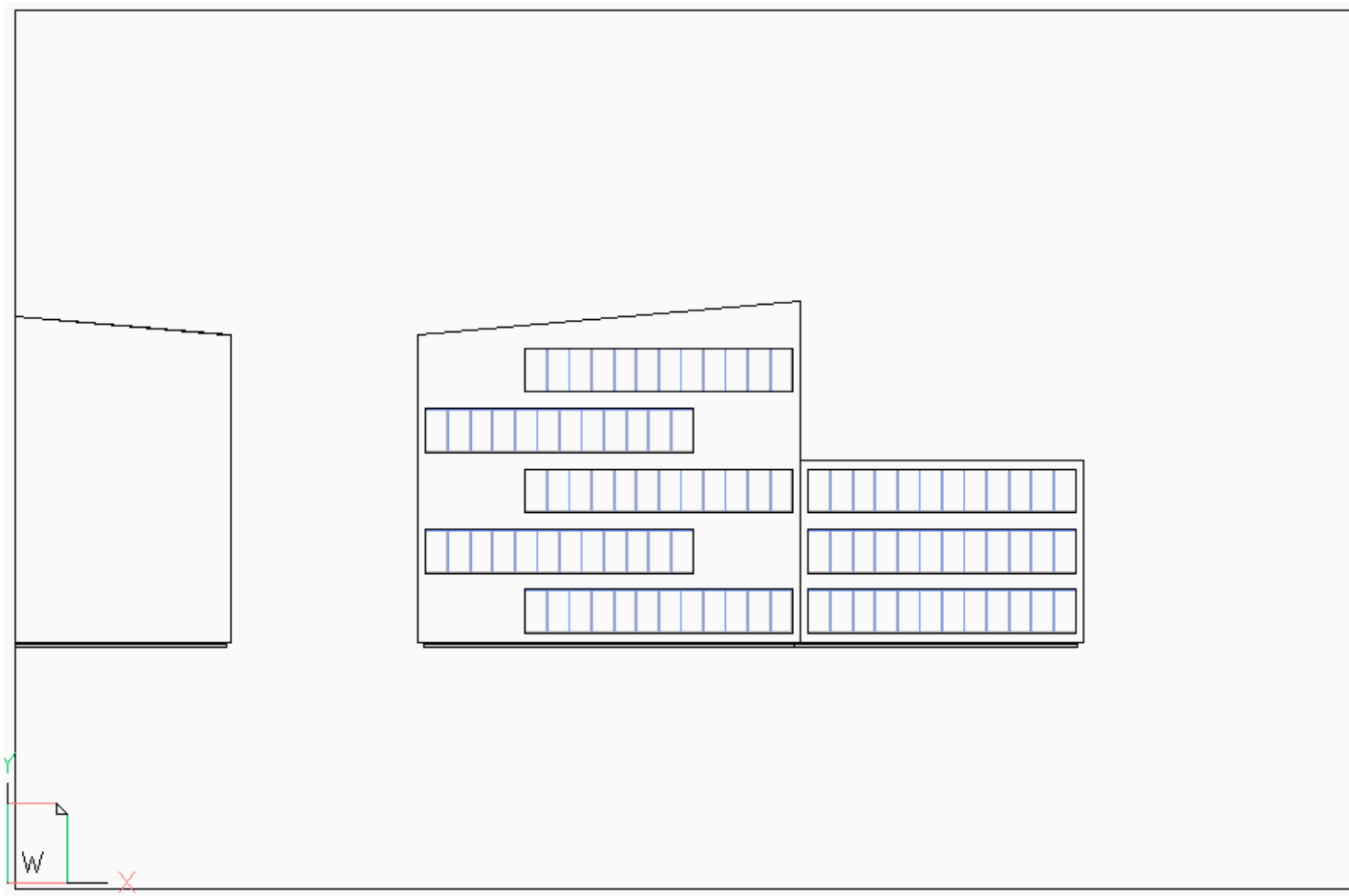
Makes viewports from entities.

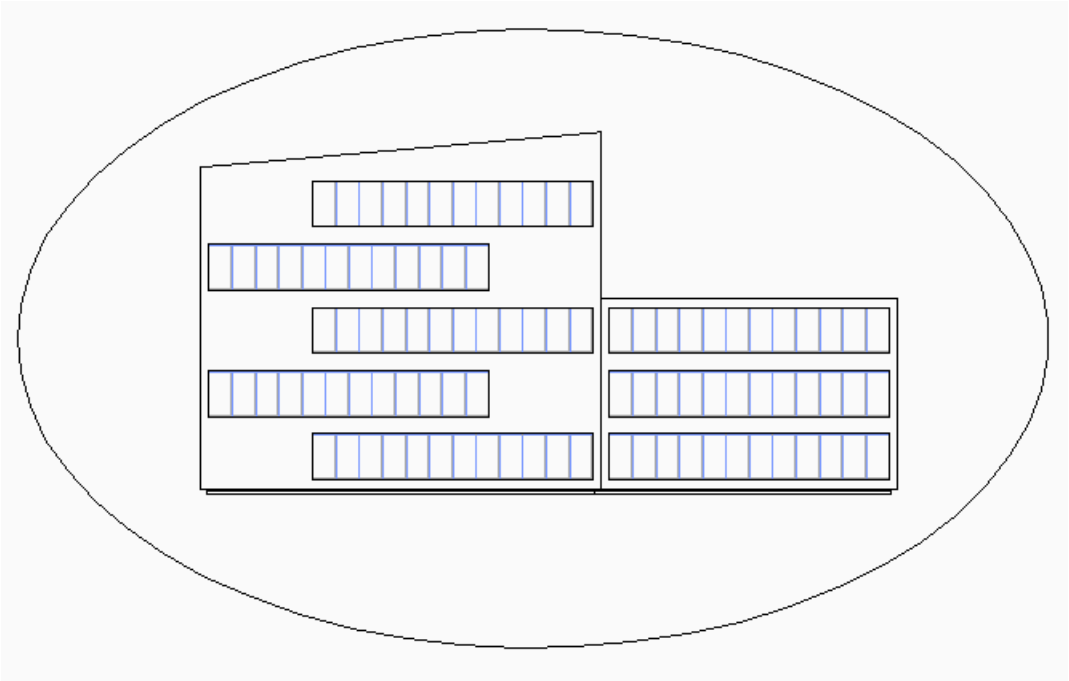
Shape Lite Pro Mechanical BIM

27.39.1 Description

Makes viewports from closed 2D entities (short for "viewport clipping") and can be easily used to show less of a drawing.

Note: This command only operates in the paper space of a layout tab and only works with already existing viewports. use the MVIEW command to make additional viewports.


Y
W
X



27.39.2 Options within the command

Select viewport to clip
Select a viewport border of the viewport to clip.

Select clipping entity
Converts a close entity such as a circle or closed polyline into a viewport boundary.

Note: The entity needs to be drawn in paper space.

Polygonal
Draws a polygonal viewport border made of straight and arc segments. Press Enter when done.

Draw Arcs
Draws an arc segment in the polygonal viewport. See the ARC command to explore the options within this option.

Distance
Specifies the distance and angle for the next polygonal segment.

Follow
Draws the next segment at the same angle as the previous polygonal segment. The length of the segment needs to be specified.

Undo
Undoes the last polygonal segment.

Close
Closes the polygon. The model space drawing is shown.

Delete
Deletes the clipping boundary and returns the original rectangular viewport.

Note: This option appears only when a clipped viewport is selected.

27.40 VPLAYER command

Changes the properties of layers in the current paper space viewport.

Shape Lite Pro Mechanical BIM

27.40.1 Description

Allows every viewport to show a different set of layers.

Note: This command does not operate in model space.

27.40.2 Options within the command

List frozen layers
Reports whether layers are frozen in the selected viewport. In the Command line, the frozen layers are shown after selecting a viewport.

Color
Overrides the color of specific layers in selected viewports. A new color is specified by entering the RGB code and applied to the layers in the selected viewports.

Red, Yellow, Green, Cyan, Blue, Magenta, White
Specifies the color for the layers in selected viewports.

Truecolor
Allows you to specify the true color for the layers in selected viewports, by entering the values for Red, Green, and Blue.

COlorbook
Opens a color book by entering its name and allows you to specify a color name from the loaded color book.

Note: The COLORBOOKPATH system variable specifies the folder(s) in which BricsCAD® should look for color book files.

Line type
Overrides the linetype of specific layers in selected viewports.

Line Weight
Overrides the lineweight of specific layers in selected viewports.

Transparency
Overrides the transparency of specific layers in selected viewports.

Freeze layers
Freezes specific layers in the current viewport.

Thaw Layers
Thaws specific layers in the current viewport.

Reset Layers
Resets frozen and thawed layers to their original settings.

New frozen layers
Creates new layers that are initially frozen when a new viewport is created.

Default visibility settings
Changes the default freeze/thaw setting for layers.

Frozen
Changes the layer's default property to frozen.

Thawed
Changes the layer's default property to thawed.

Select viewport
Specifies viewports to apply changes to.

All
Selects all viewports.

Select
Specifies a selection of viewports.

Current
Adapts changes to the current viewport.

EXcept current
Adapts changes to all viewports except the current viewport.

27.41 VPMAX command

Maximizes the current viewport.

Shape Lite Pro Mechanical BIM

27.41.1 Method

This command can be used both in Model space and Paper space to expand the selected viewport.

Note: Before using the VPMAX command in Model space, create viewports using VPORTS command.

Note: To minimize the viewport use VPMIN command.

27.42 VPMIN command

Minimizes the current viewport.

Shape Lite Pro Mechanical BIM

27.42.1 Method

This command can be used both in Model space and Paper space. This command restores the viewport to the form it was before it was maximized.

Note: To maximize the viewport use VPMAX command.

27.43 VPOINT command

Changes the 3D viewpoint.

Shape Lite Pro Mechanical BIM

Alias: -VIEWPOINT, -VP, -VPOINT, VIEWPOINT

27.43.1 Description

Changes the 3D viewpoint to change the view of the model through specifying several options.

Note: It is faster, and easier, to use the**LookFrom** widget to change the 3D viewpoint.

27.43.2 Options within the command

Set view point
Specifies the viewpoint by entering coordinates in the Command line or by specifying a point in the drawing.

Rotate
Changes the viewpoint by specifying angles.

Planview
Displays the plan view of the current UCS. See the PLAN Command.

Perspective
Sets the perspective property of the viewport.

On
Turns on perspective viewing mode. Objects further away look smaller.

Off
Turns off perspective viewing mode, returning back to orthogonal viewing mode.

27.44 VPORTS command

Creates one or more viewports in model space.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: VIEWPORTS, VPORT, VW

27.44.1 Description

Creates one or more viewports in model space and allows you to see more than one view of the same drawing.

Note: To create viewports in paper space, use the MVIEW command.

27.44.2 Options within the command

?
Lists the names and x,y coordinates of saved viewports.

Note: Press F2 key to open the prompt history window.

Save
Saves the current viewport arrangement by name.

Note: If the name already exists, you are asked to overwrite the existing configuration or not.

Yes
Replaces the viewport configuration with a new one.

No
Saves the configuration with a different specified name.

Restore
Restores a named viewport configuration after entering the name of the configuration to restore.

Delete
Erases a named viewport configuration from the drawing.

Note: Only one configuration at the time can be deleted.

Single
Creates a single viewport of the current viewport, removing all others. This option can be used to return the drawing to its original single-viewport state.

Join
Joins two or more viewports to a single viewport after specifying the dominant viewport and the ones to join.

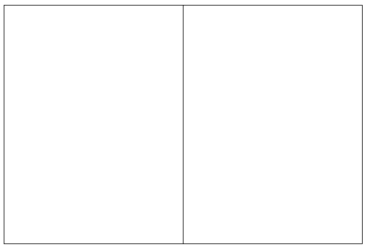
Note: If the two viewports were to join to form a non-rectangular shape, such as an L or a T shape, BricsCAD displays the message” "The selected viewports do not form a rectangle." The 'Select viewport to join' prompts you to try again.

Create 2 viewports
Splits the current viewport into two rectangular viewports.

Horizontal
Creates two horizontal viewports, one above another.



Vertical
Creates two vertical viewports, one beside another.

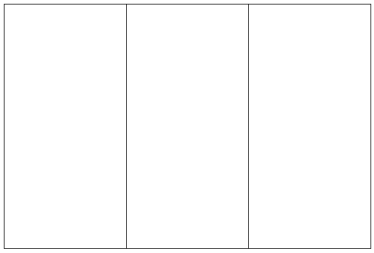


Create 3 viewports
Splits the current viewport into three rectangular viewports.

Horizontal
Creates three horizontal viewports, one above another.



Vertical
Creates three vertical viewports, one beside another.



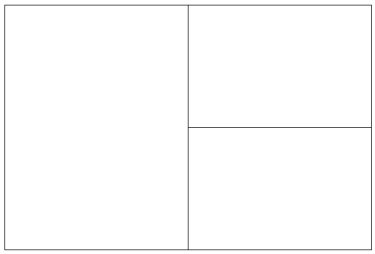
Above
Creates a single wide viewport above two side-by-side viewports.



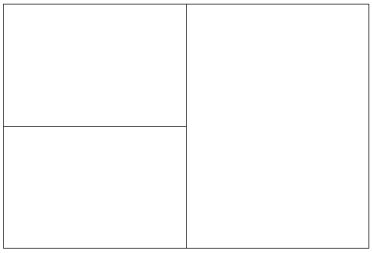
Below
Creates two side-by-side viewports above one wide viewport.



Left
Creates one tall viewport to the left of two stacked viewports.



Right
Creates one tall viewport to the right of two stacked viewports.



Create 4 viewports
Splits the current viewport into four rectangular viewports.



Note: This style is sometimes called "engineering" viewports, as it allows you to see the top, front, side, and isometric views at the same time. Once you adjust the viewpoint in each viewport, which can be done with the MVSETUP command.

27.45 -VPORTS command

Creates one or more viewports in model space.

Shape Lite Pro Mechanical BIM

27.45.1 Description

Creates one or more viewports in model space andallows you to seemorethanone view of the same drawing.

For more information see VPORTS command.

27.46 VSCURRENT command

Sets the visual style for the current viewport.

Shape Lite Pro Mechanical BIM

27.46.1 Description

Sets the visual style for the current viewport in the Command line.

27.46.2 Options within the command

cUrrent
Specifies the current visual style and gives the opportunity to keep it.

Note: The styles named in this list are the ones included with the program.

Other
Selects a user-defined visual style that was created with the VisualStyles command.

?
Displays the list of all available visual styles.

Note: Press F2 to open the Prompt History window.

27.47 VSLIDE command

Opens the View snapshot dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: VS, VSNAPSHOT

27.47.1 Description

Opens the View snapshot dialog box to select an sld, slb, emf, or wmf file to view in the current viewport. You can use the REDRAW command to remove the slide.

27.48 VTOPTIONS command

Opens the Settings dialog box with the **View Transition Options** category expanded.

Shape Lite Pro Mechanical BIM

27.48.1 Description

Opens the Settings dialog box with the **View Transition Options** category expanded to view and modify relevant system variables.

28. W

28.1 WBLOCK command

Opens the Write block to file dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: W

28.1.1 Description

Opens the Write block to file dialog box to create a block definition as an external dwg file.

28.2 -WBLOCK command

Saves blocks and other drawing entities to DWG/DXF files.

Shape Lite Pro Mechanical BIM

28.2.1 Description

Saves blocks and other drawing entities through the Save Block dialog box.

28.2.2 Options within the command

Block to save as WBlock
Specifies name of the block to write to a file.

Select entities
Selects entities to write to the file on disk.

Insertion point
Specifies the origin point of the drawing.

Select entities
Specifies the entities to create a block.

Note: The selected entities are deleted from the current drawing. To restore them, use the Oops command.

Annotative
Saves the block as an annotative block.

& to include multiple blocks
Saves multiple blocks from this drawing to the file on disk.

Note: The definitions are saved to the new drawing.

*** for whole drawing**
Saves the whole drawing, equivalent to SaveAs.

28.3 WCASCADE command

Arranges windows.

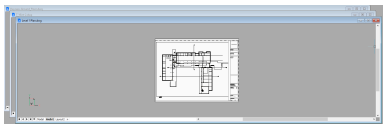
Shape Lite Pro Mechanical BIM

Icon:

Note: This is a Windows only command.

28.3.1 Description

Overlaps all windows in a cascading fashion, with the current drawing the topmost one (short for “window cascade”).



28.4 WCLOSE command

Closes the current drawing.

Shape Lite Pro Mechanical BIM

28.4.1 Description

Closes the current drawing after it has been saved. If changes have been made since the last save, a BricsCAD® dialog box offers you the opportunity to save the drawing before closing it.

28.5 WCLOSEALL command

Closes all drawings.

Shape Lite Pro Mechanical BIM

Alias: CLOSEALL

28.5.1 Description

Closes all drawings after they have been saved. If changes have been made to any drawings since their last save, a BricsCAD® dialog box offers you the opportunity to save each of those drawings before closing them.

28.6 WEBLIGHT command

Creates web lights.

Shape Lite Pro Mechanical BIM

Icon:

28.6.1 Description

Creates web lights, which illuminate scenes with a 3D representation of the light's intensity distribution.

Note: Web lights are defined by IES files, which are provided by lighting manufacturers.

Note: The command is not allowed when LIGHTINGUNITS system variable = 0.

Note: If the LIGHTINGUNITS system variable is 1 (American lighting units) or 2 (International lighting units), you need to specify the location of the light in 3D space and of the point at which the light is focused.

28.6.2 Options within the command

Name
Specifies a name for the light.

Intensity factor
Specifies the intensity of the light. Enter an intensity value between 0.00 and the maximum value supported by your system (max float = maximum value of a floating point number).

Status
Toggles the status of the light.

ON
Light is included when calculating renderings.

OFf
Light is excluded when calculating renderings.

Photometry
Specifies the photometric properties of the light:

Intensity (Cd)
Indicates the intensity expressed in candela (Cd).

Flux (Lm)
Indicates the luminous flux expressed in lumens (Lm).

Illuminance (Lx)
Indicates the illuminance expressed in lux (Lx) or in **Distance**, expressed in drawing units.

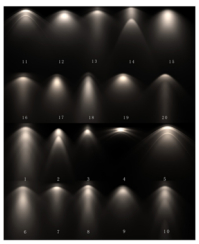
Color
Specifies a color name or choose an option.

?
Allows you to enter a color name.

*
Displays all the available color names.

Kelvin
Specifies a color temperature expressed in Kelvin degrees.

weB
Specifies the light distribution map, called **web** as an uneven light distribution can look like a spider's web:



File
Specifies a Web file name (.IES file). This is a text file that describes the intensity of a light source at many points on a spherical grid (image credit Vertheim).

X
Defines the X rotation for the web.

Y
Defines the Y rotation for the web.

Z
Defines the Z rotation for the web.

shadoW
Specifies the look of shadows cast by this light:

Off
Disables the calculation of shadows for the light.

Sharp
Displays shadows with sharp edges. Use this option to increase performance.

soFt mapped
Displays realistic shadows with soft edges.

soft sAmpled
Displays realistic shadows with softer shadows based on extended light sources:

Shape
Defines the shape of the shadow circular (Disk) or rectangular (Rect) and their dimensions.

sAmples
Defines the sampling size of the shadow; larger numbers are more accurate but take longer to render.

Visible
Defines the shape visibility (type Yes or No to cast a shadow that represents shapes (more accurate) or that is rectangular (faster to render).)

filterColor
Specifies the color of the light by entering the true color (R, G, B) or entering an option.

True color (R, G, B)
The RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The name of the model comes from the initials of the three additive primary colors, red, green, and blue. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values).

Index color
Specifies a color name or number.

Hsl
Specifies the color using three parameters Hue, Saturation and Lightness

Colorbook
Specifies the colorbook name and the color name inside of it. Verify if the colorbook (.acb) file is located in the folder indicated by the COLORBOOKPATH command.

28.7 WEDGE command

Creates a 3D Solid in the shape of a wedge.

Shape Lite Pro Mechanical BIM

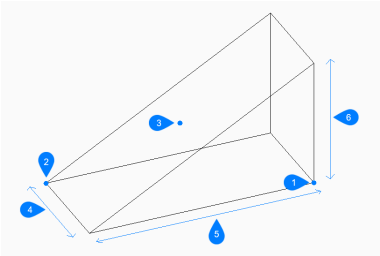
Icon:

Alias: WE

Note: In BricsCAD Lite, which doesn't support 3D Solids, the WEDGE command launches the AI_WEDGE command.

28.7.1 Description

Creates a 3D solid in the shape of a rectangular or square wedge. Choose from a combination of options including corner, center, length, height, and cube.



1 First corner

2 Opposite corner

3 Center of wedge

4 Width

5 Length

6 Height

28.7.2 Methods to create a wedge

This command has 2 methods to begin creating a wedge:

- Set corner of wedge
- Center

Set corner of wedge
Begin creating a wedge by specifying a corner for the base of the wedge then:

Set opposite corner
Specify the opposite corner for the base of the wedge to apply both the length and width. The wedge is created parallel to the x and y axes.
Additional option: [Cube/Length of side]

Height of wedge
Specify the height of the wedge. The tall end of the wedge is drawn from the first corner that you specified.
Additional option: [2Point]

Center
Begin creating a wedge by specifying the center of the wedge then:

Set corner of wedge
Specify a corner on the midplane of the wedge to apply both the length and width of the wedge's base. The wedge is created parallel to the x and y axes.
Additional option: [Cube/Length of side]

Height of wedge
Specify the height of the wedge. The tall end of the wedge is drawn from the first corner that you specified.
Additional option: [2Point]

28.7.3 Options within the WEDGE command

After you begin creating a wedge, the following options may be available:

Cube
Specify a single distance to use for the length, width, and height of the wedge.

Length of side
Specify the length and angle for one side of the wedge then:

Width of wedge
Specify the width of the wedge.

Height of wedge
Specify the height of the wedge. The tall end of the wedge is drawn from the first corner that you specified.

2Point
Specify the height of the wedge as the distance between any two points.

28.8 WHOHAS command

Opens the Open drawing dialog box.

Shape Lite Pro Mechanical BIM

28.8.1 Description

Opens the Open drawing dialog box to select a dwg file to find out who has it open. After selecting the file and choosing Open, a BricsCAD dialog box displays. It indicates who has the file open and when they opened it.

28.9 WHTILE command

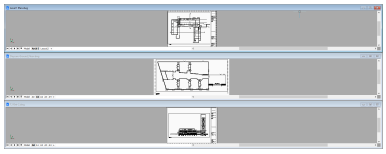
Tiles windows horizontally.

Shape Lite Pro Mechanical BIM

Icon:

28.9.1 Description

Tiles the drawing windows horizontally to maximize their width (short for "window horizontal tile").



28.10 WIARRANGE command

Lines up iconized drawing windows.

Shape Lite Pro Mechanical BIM

28.10.1 Description

Arranges the icons of minimized windows at the bottom of the graphical screen.

Note: This command works only when windows are minimized as icons.

- Command only available on the Windows platform.

28.11 WIPEOUT command

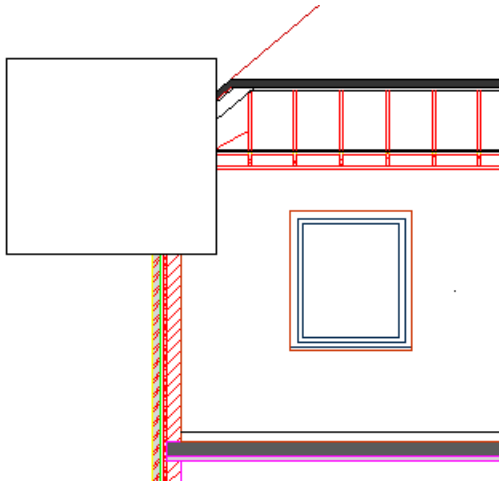
Creates a wipeout.

Shape Lite Pro Mechanical BIM

Icon:

28.11.1 Description

Creates a wipeout to mask parts a drawing. Options allow you to convert a polyline to a wipeout and control the display of wipeout frames.



28.11.2 Methods to create a wipeout

This command has 2 methods to begin creating a wipeout:

- Specify start point
- Polyline

Specify start point

Begin creating a wipeout by specifying a start point then:

Specify next point

Specify the next vertex of the wipeout. You can continue adding vertices until you press Enter to end the command.

Additional options: [Undo/Close]

Polyline

Begin creating a wipeout by selecting an existing closed polyline then:

Delete selected polyline? [Yes/No]

Choose whether to erase or retain the original polyline.

- **Yes**: erases the polyline.
- **No**: retains the polyline in addition to the wipeout.

28.11.3 Options within the WIPEOUT command

After you begin creating a wipeout, the following options may be available:

Frames

Specify whether to display wipeout frames. This is saved in the WIPEOUTFRAME system variable and applies to all wipeout entities in the drawing.

- **On**: display and plot wipeout frames.
- **Off**: hide wipeout frames.
- **Display but not plot**: display but do not plot wipeout frames.

Undo

Undo the last wipeout vertex and continue drawing from the previous one.

Close
Automatically draw a wipeout segment from the last vertex to the first one. This ends the command.

28.12 WMFOUT command

Opens the Create WMF file dialog box.

Shape Lite Pro Mechanical BIM

Alias: WO

28.12.1 Description

Opens the Create WMF file dialog box to save data from the current drawing to a wmf file.

28.13 WNEXT command

Switches to the next window.

Shape Lite Pro Mechanical BIM

28.13.1 Description

Brings another drawing window to the foreground (short for "window next"). It is useful when multiple drawings are open. Meant for use by macros.

The program changes focus to the next window. Windows are switched to in the order that they were created.

28.14 WORKSETS command

Opens the Working sets dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

28.14.1 Description

Opens the Working sets dialog box to create and load sets of drawing files. This makes it easy to load an entire group of drawings at once.

28.15 WORKSPACE command

Sets the current workspace.

Shape Lite Pro Mechanical BIM

Icon: 

28.15.1 Description

Sets the current workspace and lets you create, modify and save workspaces.

28.15.2 Options within the command

set Current
Sets the current workspace after entering a name.

?
Lists all the available workspaces.

Save As
Saves the current settings to a new workspace.

Rename
Renames a workspace.

Delete
Deletes a workspace.

SEttings
Opens the Workspaces tab of the Customize dialog box. Make adjustments to the workspace of your choice.

28.16 WPREV command

Displays the previous drawing window.

Shape Lite Pro Mechanical BIM

28.16.1 Description

Displays the previous drawing window when multiple drawings are open.

28.17 WSSAVE command

Opens the Save workspace dialog box.

Shape Lite Pro Mechanical BIM

28.17.1 Description

Opens the Save workspace dialog box to save the current workspace configuration to a new workspace name.

28.18 WSSETTINGS command

Opens the Customize dialog box.

Shape Lite Pro Mechanical BIM

28.18.1 Description

Opens the Customize dialog box to customize the user interface.

28.19 WVTILE command

Tiles windows vertically.

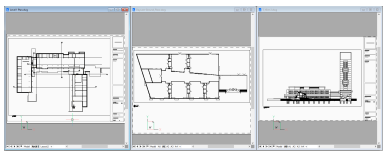
Shape Lite Pro Mechanical BIM

Icon:

28.19.1 Description

Tiles drawing windows vertically to maximize their height.

BricsCAD tiles all windows vertically, with the most recent drawing window highlighted.



29. X

29.1 XATTACH command

Opens the Select reference file dialog box.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: XA

29.1.1 Description

Opens the Select reference file dialog box to select a dwg file to reference into the current drawing. After selecting the file and choosing **Open**, the Attach external reference dialog box displays. It enables you to specify where and how to attach the dwg file.

29.2 XCLIP command

Clips externally-referenced drawings with polygons to hide parts of them, and adjusts the front and rear clipping planes (short for "xref clip").

Shape Lite Pro Mechanical BIM

Icon: 

Alias: CLIP

29.2.1 Method

Choose one or more xrefs to be clipped or enter ALL to choose all xrefs in the current drawing and create the desired clipping boundary.

29.2.2 Options within the command

ON
Turns on clipping boundaries that were turned off by the OFF option.

OFF
Turns off clipping so that all of the xref is again visible. The clipping is saved and is turned on again with the ON option.

Clipdepth
Sets 3D clipping to the front and rear, to hide parts of 3D models that might obscure the front and confuse the back.

Specify front or back clip point
Specifies the position of the front and back clipping plane by picking the two points or enter coordinates. The cutting plane is parallel to the current viewpoint.

Distance
Specifies the distance of the clipping point by entering the distance from the clipping boundary to the front or rear planes.

Remove
Removes the front or rear clipping point.

This option does not operate until you create at least one clipping boundary. It is no useful for 2D xrefs.

Invert
Inverts the clipping mode so that the interior of the clipping boundary is hidden from view (the outer area is visible); repeat this option to invert again, making the inner portion visible.
It is much easier to toggle the Invert status through the Properties panel.

Delete
Erases all clipping boundaries; does not prompt or warn you.

generate Polyline
Draws a polyline over a selected clipping boundary: this allows you to edit the clipping boundary, but in an indirect manner. This new generated polyline could be edited with PEDIT command and reused it as a New boundary option.

New boundary
If an existing boundary is detected you may choose to delete it or not.

Caution! This option replaces all existing clipping boundaries with the new one. The command continues only when all previous boundaries are deleted.
If no previous boundary exists or just deleted an existing one:

Select polyline
Choose any open or closed entity made from polylines, such as splined polylines, rectangles, donuts, and polygons. While the polyline does not need to be closed, it cannot cross over itself.

Polygonal
Creates non-rectangular clipping boundaries, with a minimum of three sizes.

Rectangular
Creates rectangular clipping boundaries.

29.3 XEDGES command

Extracts edges from 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

29.3.1 Description

Makes entities out of one or more edges found on 3D solids, 3D surfaces, and 2D regions; extracted edges become 2D entities in 3D space, such as arcs and lines.

Newly created entities are placed in the same location as the source edges, and on the current layer.

29.4 XFACES command

Extracts faces from 3D entities.

Shape Lite Pro Mechanical BIM

Icon: 

29.4.1 Description

Makes copies of one or more faces from one or more 3D solids and 3D surfaces.

29.4.2 Options within the command

Move
Move the extracted face/s by the distance indicated by two points or by a displacement vector.

29.5 XLINE command

Creates xlines.

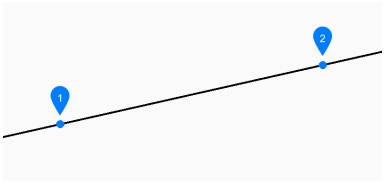
Shape Lite Pro Mechanical BIM

Icon:

Alias: INFLINE, XL

29.5.1 Description

Creates an xline (infinitely long line) from a combination of options point, direction, and angle.



1 Point along line

2 Direction

29.5.2 Methods to begin an xline

This command has 6 methods to begin creating an xline:

- Set point along line
- Horizontal
- Vertical
- Angle
- Bisect
- Parallel

You can continue adding unlimited xlines until you press Enter to end thecommand.

Set point along line
Begin creating an xline by specifying a point on the xline then:

Direction
Specify the direction of the xline from the starting point.

Horizontal
Begin creating an xline parallel to the x-axis then:

Location
Specify a point on the xline.

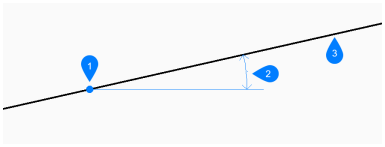
Vertical
Begin creating an xline parallel to the y-axis then:

Location
Specify a point on the xline.

Angle
Begin creating an xline based on an angle then:

Enter angle
Specify the angle of the xline.
Additional option: [Reference]

Location
Specify a point on the xline.



1 Location

2 Angle

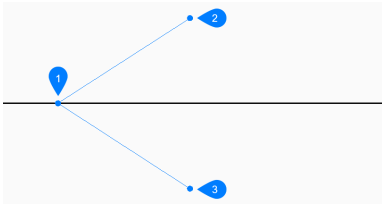
3 Xline

Bisect
Begin creating an xline which bisects two imaginary lines then:

Set vertex point
Specify a point at the vertex of the two imaginary lines.
Additional option: [Entity]

Bisection angle start point
Specify a point to define the first imaginary line. The vertex is used as the other point.

Bisection angle end point
Specify a point to define the second imaginary line. The vertex is used as the other point.



1 Vertex point

2 Bisection angle start point

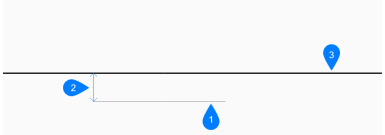
3 Bisection angle end point

Parallel
Begin creating an xline parallel to a line or polyline segment then:

Set offset distance for parallel infinite line
Specify the offset distance for the xline.
Additional option: [Through point]

Select entity for parallel infinite line
Select a line or polyline segment from which to offset the xline.

Side for parallel infinite line
Specify the side on which to place the xline.



1 Line segment

2 Offset distance

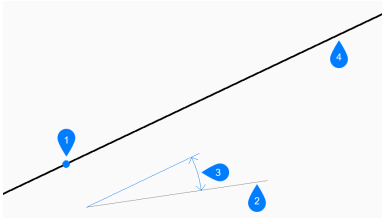
3 Xline

29.5.3 Options within the XLINE command

After you begin creating an xline, the following option may be available:

Reference
Select an entity to use as a reference angle then:

Enter angle
Specify the angle to place the xline relative to the selected entity.



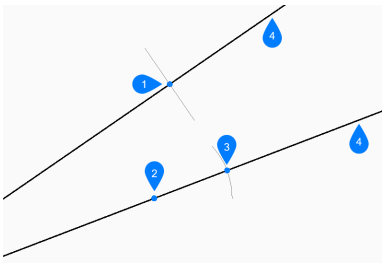
1 Location

2 Reference entity

3 Angle

4 Xline

Entity
Select a line, arc or polyline segment to bisect. When you select a line or polyline segment, the command draws the xline perpendicular to the segment's midpoint. When you select an arc or polyarc, the command draws the xline perpendicular to the arc's center and point.
This option works with splined polylines, but not with spline entities.



1 Perpendicular to line's midpoint

2 Perpendicular to arc's centerpoint

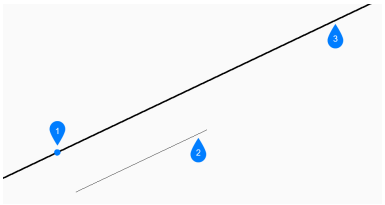
3 Perependicular to arc' midpoint

4 Xline

Through point
Specify a point through which to draw the xline

Select entity for parallel infinite line
Select an entity to which the xline should be parallel.

Through the point
Specify a point through which to draw the xline.



1 Through point

2 Line segment

3 Xline

29.6 XMLSAVE command

29.7 XOPEN command

Opens referenced drawings for editing.

Shape Lite Pro Mechanical BIM

Icon:

29.7.1 Description

Opens an externally-referenced drawing (xref, attached to the current drawing) for editing in a separate drawing tab. When done editing, close the tab.

29.8 XPLODE command

Explodes entities.

Shape Lite Pro Mechanical BIM

Icon: 

29.8.1 Description

Explodes entities providing control over the resulting entities.

Note: This command does not explode attached references, entities on frozen layers or basic entities such as lines, circles and arcs.

29.8.2 Options within the command

Entities to explode
Selects the entities to explode.

Note: Entities on frozen layers are not selected and not exploded.

Separately
Explodes one entity at a time.

All
Explodes all selected entities at once.

All listed
Allows to specify all properties for the selected entities.

Multiple
Specifies more than one property for the exploded entities by choosing properties one by one.

LAyer
Specifies the layer for the exploded entities

Note: The layer must already exist in the drawing.

Color
Specifies the color for the exploded entities.

LType
Specifies the linetype for the exploded entities.

ltScale
Specifies the linetype scale for the exploded entities.

Inherit all from parent
Inherits the color, layer and linetype scale from source entities.

29.9 XREF command

Opens the Attachments panel.

Shape Lite Pro Mechanical BIM

Icon: 

Alias: XR

29.9.1 Description

Opens the Attachments panel to display it in the current workspace. The Attachments panel appears at the same size and location that it did before it was closed or collapsed. Like any other dockable panel, the Attachments panel can be either floating, docked or stacked.

29.10 -XREF command

Attaches and detaches DWG files.

Shape Lite Pro Mechanical BIM

Alias: -XR

29.10.1 Description

Controls DWG drawing files that are attached to the current drawing (short for **external reference**).

29.10.2 Options within the command

? to list external references
Lists the names and paths of attached drawings, if any, and their statuses.

Attach
Opens the **Select File to Attach** dialog box to select a DWG file to attach.

Overlay
Opens the **Select File to Overlay** dialog box to select a DWG file to overlay.

Bind
Copies the contents of the attached drawing to the current drawing as a block reference.

Note: The BINDTYPE system variable defines how the Xref layers/objects will be brought in the current drawing.

Detach
Removes attached drawings.

Path
Changes the path (drive and folder) to the source DWG file, in case the original path no longer points to the file.

Reload
Reloads the attached drawing, after its content have changed.

Unload
Hides the attached drawing, but keeps it attached.

30. Y

31. Z

31.1 ZCENTER command

Toggles the Center of a Face 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.1.1 Description

Toggles the Center of a Face 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.2 ZINTERSECTION command

Toggles the Intersection 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.2.1 Description

Toggles the Intersection 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.3 ZKNOT command

Toggles the Knot 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon:

31.3.1 Description

Toggles the Knot 3D entity snap to enable or disable snapping to spline knot. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.4 ZMIDPOINT command

Toggles the Midpoint 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.4.1 Description

Toggles the Midpoint 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.5 ZNEAREST command

Toggles the Nearest 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.5.1 Description

Toggles the Nearest 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.6 ZNONE command

Disables all 3D entity snaps.

Shape Lite Pro Mechanical BIM

Icon: 

31.6.1 Description

Disables all 3D entity snaps to prevent the cursor from snapping to 3D entities. You can launch this command at the Command prompt to turn off running entity snaps. Doing so changes the value of the 3DOSMODE system variable to zero. You can also launch this command within another command to turn off entity snaps only for the current operation. This does not change the value of the 3DOSMODE system variable.

This command displays no prompts and has no options.

31.7 ZOOM command

Visually changes the size of the drawing within the current viewport.

Shape Lite Pro Mechanical BIM

Icon: 🔎

Alias: Z

31.7.1 Description

You can change the magnification of your drawing at any time by zooming.

31.7.2 Options within the command

In
Zooms in by 50%. It is equivalent to Zoom 2.

Out
Zooms out by 50%. It is equivalent to Zoom 0.5.

All
Displays the entire drawing.

Center
Zooms in or out about a specified center point.

Dynamic
Pans and zooms using a rectangular box which represents the viewport. This option first executes a zoom extents or zoom limits to show the original size of the viewport as a dashed rectangle.
Adjust the size of the view box dynamically, then pan it to the part of the drawing you want to view.

Extents
Displays the drawing to the extents of entities.

Note: Limits and frozen entities are ignored.

Left
Zooms in or out with the lower left corner specified by a point.

Previous
Displays the previous view, whether set by the Zoom, Pan, or View commands.

Right
Zooms in or out with the upper right corner specified by a point.

Scale
Specifies the absolute zoom scale factor.

Note: Less than 1 makes the drawing appear smaller. More than 1 makes the drawing appear larger.

nx
Specifies the zoom factor relative to the current zoom. For example, 2x makes the current view twice as large.

Nxp
Specifies the zoom factor in paper space relative to model space. For example, 2xp makes the model space view twice as large as the surrounding paper space.

Window
Zooms in to a rectangular area.

Object
Zooms in on a single entity or a selection set.

31.8 ZPERPENDICULAR command

Toggles the Perpendicular on a face 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon:

31.8.1 Description

Toggles the Perpendicular on a face 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.

31.9 ZVERTEX command

Toggles the Vertex 3D entity snap.

Shape Lite Pro Mechanical BIM

Icon: 

31.9.1 Description

Toggles the Vertex 3D entity snap to enable or disable snapping to extension. You can launch this command at the Command prompt to toggle a running entity snap. Doing so changes the value of the OSMODE system variable accordingly. You can also launch this command within another command to turn off the entity snap only for the current operation. This does not change the value of the OSMODE system variable.