BRICSCAD®

USER GUIDE

Updated for V18

Table of Contents

Get Started 21
The Get Started Dialog 21
To start a BricsCAD session 22
To select the startup user profile 22
Using a Profile Preset 23
To Start a BricsCAD Session 24
To Set the Startup User Profile 24
Using a Profile Preset 25

User Interface 27
Keyboard Shortcuts 27
The BricsCAD Application Window 29
Overview 29
To Display the Ribbon 30
To Hide the Ribbon 30
Document Tabs 31
Open/Close the Command Bar 32
To Set the Command Line Properties 33
Open/Close the Status bar 33
Working with the Status Bar 34
Display/Hide Scroll Bars 40
Hide / show a selection of user interface elements 42
Toolbars 43
Main Toolbars 43
Toolbar Flyouts 47
Opening a Toolbar 50
To set the Icon Size 51
To Set the Position and Visible Property 51
To Tear off a Flyout Toolbar 52
Closing a Toolbar 52
Properties Panel 53
Overview 53
Open the Properties Panel 53
Adjust the Size of the Properties Panel 56
Setting the current properties in the Properties panel 56
Edit a Single Entity 56
Edit a Selection Set 58
Edit Endpoints and Nodes 58
To edit the vertices of a polyline 59
The Quad Cursor Menu 59
To Activate the Quad 60
Editing the Quad Settings 60
The Quad Layout 62
Using the Quad 64
Selecting Multiple Entities 64
Selecting Edges of Faces or Solids 65
Using the Quad to Create Entities 65
Customizing the Quad 65
To Edit the Workspace Command Groups 66
Prompt Menus 68
To control the display of prompt menus 69
Dockable Panels 69
To open or close a panel 70
To dock a panel 70
To dock a panel with respect to another docked panel 71
To stack panels in a tabbed panel 71
Using tabbed panels 73

Customizing BricsCAD 75
Startup Options 75
What are BricsCAD User Files 76
Using the BricsCAD User File Manager 77
What are the CUI Files 78
Opening the Customize Dialog 78
Working with CUI files 79
To Load a Main CUI File 80
To Load a Partial CUI File 80
To Unload a Partial CUI File 81
Managing customization groups 82
Creating Custom Tools 82
Overview 82
Create a Partial CUI File 82
Add a Toolbox to a Menu Group 83
Add Tools to a Toolbox 84
Add a Main Menu 85
Add a Submenu to a Menu 86
Add a Context Menu 86
Add a Toolbar 89
Add a Flyout to a Toolbar 90
Add a Control to a Toolbar 91
Add a Keyboard Shortcut 92
To add an existing tool to a menu, toolbar or shortcut 93
To change the order of the tools in a menu or toolbar 93
Create a New Tool 93
Ribbon 95
Ribbon Panels 95
Ribbon Buttons 96
Creating a Ribbon Tab 97
Editing a ribbon tab 97
To add a panel to a ribbon tab 97
To remove a panel from a ribbon tab 98
To add a row panel to a row 98
To add a command button to a row 98
To add a split button to a row 99
To add a toggle button to a row 100
To add a control to a row 101
Creating a Ribbon Panel 101
Editing a Ribbon Panel 101
To add a row to a ribbon panel 101
To add a launcher to a ribbon panel 101
Editing a Panel Row 102
To add a row panel to a row 102
To add a command button to a row 102
To add a split button to a row 103
To add a toggle button to a row 104
To add a control to a row 104
Tool Palettes 105
To Open the Tool Palettes Bar 105
To Set the View Options 106
Adding a New Tool Palette 106
To open the customize dialog box 107
Importing a Tool Palette 107
Working with Palette Groups 108
To Rename a Tool Palette 109
To Delete a Tool Palette 109
Adding a Hatch Pattern 109
Adding blocks 110
Adding Blocks from the Current Drawing 110
Adding Blocks from Any Drawing 110
Adding a Drawing as a Block 111

Adding All Blocks from Multiple Drawings ... 111
Adding a Component ... 112
Adding a Command ... 113
Editing a Tool ... 114
Managing tools ... 117
To Specify a Tool Image ... 117
To Move a Tool ... 118
To Copy a Tool ... 118
Command Aliases ... 119
To Edit the Alias File in a Text Editor ... 119
Using the Customize Dialog to Edit the Alias File ... 120
User Profile Manager ... 122
Content of a User Profile ... 122
Launch the User Profile Manager ... 122
Creating User Profiles ... 123
Loading a User Profile ... 124
Restarting BricsCAD Using a Different User Profile ... 125
Managing User Profiles ... 125
Copy a User Profile ... 125
Rename a User Profile ... 126
Delete a User Profile ... 126
Export a User Profile ... 126
Edit the User Profile Description ... 126
Import a User Profile ... 126
Import a Profile from Another User ... 126
Workspaces ... 128
Definition ... 128
Managing Workspaces ... 128
Setting the Current Workspace ... 130
Creating a Workspace ... 130
Editing a Workspace ... 131
Deleting a Workspace ... 133
Importing a Workspace ... 134
Managing Drawings ... 135
Creating a Drawing ... 135
Using the Welcome Dialog Box ... 135
To create a drawing using a default template drawing ... 136
First method: ... 136
Second method: ... 136
Setting the Default Template Drawing ... 137
Creating a Drawing from Scratch ... 137
Setting the Plot Style Policy ... 138
Creating a Drawing Using a Template Drawing ... 138
Creating a Drawing Using the New Command ... 138
Setting the Template Path Variable ... 139
Creating a Drawing Using a Wizard ... 139
Drawing Explorer ... 141
To Open the Drawing Explorer ... 142
To Control the Drawing Explorer Layout ... 143
Opening a Drawing ... 144
Adding a Drawing Folder ... 145
To Insert a Block from an Unopened Drawing ... 145
Copy Definitions Between Drawings ... 145
Merging Layers ... 145
Drawing Properties ... 146
To Open the Drawing Properties Dialog ... 146
Defining Default Properties ... 147
Adding Custom Properties ... 147
Drawing Security Options ... 148
Command ... 148
To Set a Password ... 148

To Remove a Password ... 149
Opening and Saving Drawings ... 149
Using the Drag-and-Drop option menu ... 152
Getting Drawing Information ... 154
General drawing information ... 154
Information about Entities ... 154
Comparing Drawings ... 155
Comparison System Variables ... 155
Executing a Comparison Session ... 155
Analyzing the Comparison in the Structure Panel ... 156
Merging the Compared Drawings ... 157
Repairing a Drawing ... 158
To Analyze the Integrity of the Current Drawing ... 158
To Repair a Drawing ... 158
Exporting Drawings ... 159
eTransmit ... 159
Export to PDF ... 159
Export a Drawing To PDF ... 160
Projects ... 161
Opening the Projects Settings ... 161
Creating Projects ... 161
Adding Search Paths to a Project ... 162
Changing the Order of the Search Paths ... 162
Assigning a Project to the Current Drawing ... 162
Drawing Accurately ... 163
Orthogonal Mode ... 163
To toggle Orthogonal Mode ... 163
Direct Distance Entry ... 164
To define the Coordinates setting ... 164
Unit Settings ... 164
Entering angles ... 165
Coordinate Reference System ... 166
Parametric Constraints ... 172
Commands and toolbars ... 174
Using Dimensional Constraints ... 176
To Edit Dimensional Constraints ... 176
To Set the Visibility of Dimensional Constraints ... 179
Using expressions to define dimensional constraints ... 180
Creating a New Parameter ... 180
Using Operators in Expressions ... 181
Using Functions in Expressions ... 182
Using Geometric Constraints ... 184
To Set the Display of Constraints Bars ... 185
To Control the Position of a Constraints Bars ... 186
Working with the Constraints Bar ... 187
Deleting Constraints ... 187
Using Fix Constraints ... 188
Using Coincident Constraints ... 188
Using Horizontal and Vertical Constraints ... 190
To Constrain a Line to a Fixed Angle ... 190
Dynamic Dimensions ... 191
Defining Dynamic Dimensions ... 191
Create Entities ... 194
Grip-edit Entities ... 194
Measure Entities ... 195
Using Dynamic Dimensions with Tracking Points ... 195
Coordinate Input ... 196
Relative Coordinates ... 196
Absolute Coordinates ... 196
Cartesian Coordinates ... 196
Using relative Cartesian coordinates to draw a rectangle ... 197

Cylindrical Coordinates ... 197
Spherical Coordinates ... 198
Snap and Grid ... 199
Snap and Grid Settings ... 200
To Synchronize Snap and Grid Spacing ... 200
To set the Drawing Limits ... 201
Using Isometric Snap ... 201
Setting the Snap Angle ... 202
Entity Snaps ... 203
To Define the Entity Snap Settings ... 204
To Set the Entity Snaps ... 206
Working with Multiple Entity Snap Modes ... 208
To Snap to the Extension of Two Entities ... 209
Using Temporary Tracking Points ... 209
Using 'Middle of 2 Points' Snap ... 210
Using Parallel Entity Snap ... 210
Polar Tracking ... 211
To Define the Polar Tracking Settings ... 211
Using Polar Tracking to Draw a Line ... 212
Using Snap Track Lock ... 212
Using Temporary Tracking Points ... 212
Snap Tracking ... 213
Using Entity Snap Tracking ... 214
User Coordinate Systems ... 216
To Move the Origin of the Current Coordinate System ... 216
To Define a User Coordinate System ... 216
To Restore the WCS ... 217
To Restore a UCS ... 217
To Set a Relative UCS ... 217
Dynamic UCS ... 219
Measuring ... 221
Measuring Distances ... 221
Measuring Lengths ... 222
Measuring Areas ... 223

Viewing your Drawing ... 224

Layer Utilities ... 227
Hide and Show Entities ... 228
Redrawing and Regenerating a Drawing ... 229
- Redraw All ... 229
- Regenerate All ... 229
- Regenerate ... 229
- Redraw ... 229

Panning ... 230
- Real Time Pan Command ... 230
- Pan Command ... 230

Zooming ... 231
- Zoom Command ... 231
- Real Time Zooming ... 232
- Dynamic zooming ... 232

View Manipulation Using the Mouse and Keyboard ... 234
Walk Through Navigation ... 235
Zoom Factor Variable ... 236
Middle Button Pan Variable ... 236
View Rotation ... 237
View Transition Options ... 237
Restoring Orthographic and Isometric Views ... 237
Real Time Sphere ... 240
Real Time X ... 241
Real Time Y ... 241
Real Time Z ... 242
Preset Viewpoints ... 243

Plan View 245
Layer States 246
Opening the Layer States Explorer 247
Save the Current Layer State 247
Edit a Layer State 248
Restore a Layer State 249
Copy Layer States to Another Drawing 249
Export a Layer State 249
Import a Layer State 250
Named Views 250
Exploring Views 250
Exploring views 250
Saving a View 251
Restoring a View 251
Defining the View Background 252
To Edit the View Background 254
Perspective Views 255
Commands 255
Defining a Camera View 256
Toggle the Perspective Property of a View 257
Model Space and Paper Space 258
Switching between Model Space and Paper Space 258
Using the Model and Layout Tabs 258
Toggle between Model Space and Paper Space 259
Model Space Viewports 260
Creating Viewports in Model Space 260
Drawing in Multiple Viewports 262
Joining Viewports 262
Paper Space Viewports 263
Displaying Paper Sheet and Printable Area 264
Creating Viewports in a Layout 265
Creating an Array of Layout Viewports 268
To Clip a Viewport 269
To Remove the Clipping Boundary of a Viewport 269
Defining Viewport Properties 270
Navigating in a Paper Space Layout 272
Setting the Layer Visibility in a Viewport 272
Setting the Scale of a Viewport 273
Setting the Scale of a Selection of Viewports 274
Rotating the Display in a Viewport 274
To Align Viewports 275
To Align Model Space and Paper Space Points 276
Layouts 277
Creating a new Layout 277
Copying Layouts 277
Importing Layouts 278
Renaming Layouts 278
Arranging Layout Tabs 278
Deleting a Layout 279
Deleting Multiple Layouts 279
Generated Drawing Views 280
Commands and Toolbars 281
Standard Drawing Views 282
Generating Cross Section Views 284
Updating Drawing Views 285
Exporting Views to Model Space 286
Customizing drawing Views 286

Creating Entities 287
Entity Creation Settings Overview 287
Entity Creation Settings 289
Fill Mode 289
Setting Fill Mode 290
Entity Color 290
Index color 290
Index color dialog 291
True Colors 291
True color dialog 292
To Set the Current Entity Color 292
Lineweight 293
Lineweight BYLAYER: 293
Lineweight BYBLOCK: 293
DEFAULT lineweight: 293
Setting Current Lineweight 293
Defining the Lineweight Settings 294
Linetype 295
Linetype BYLAYER: 295
Linetype BYBLOCK: 295
Linetype scale (LTSCALE) 295
Current Linetype 296
Add a New Linetype 297
Entity Linetype Scale 298
Global Linetype Scale 298
Current Layer 299
Setting the current layer 299
Transparency 300
Drawing 2D Entities 301
Drawing 2D Overview 301
Lines 303
Rays 304
Infinite Lines 305
Polylines 306
Drawing Polylines 307
Boundary Polylines 308
Splines 310
Sketch 312
Setting the Sketch Poly (SKPOLY) system variable 312
Creating a freehand sketch 312
Circles 313
General Procedure to Draw a Circle 313
Drawing a Circle Tangent to Three Entities 314
Editing a Circle 314
Arcs 315
General Procedure to Draw an Arc 315
Editing an Arc 316
Ellipses 316
Setting the Polyline Ellipse Variable 317
General Procedure to Draw an Ellipse 317
Editing an Ellipse 318
Elliptical Arcs 319
General Procedure to Draw an Elliptical Arc 319
Editing an Elliptical Arc 320
Rectangles 321
General Procedure to draw a rectangle 321
Polygons 322
Traces 323
Donuts 324
Solids 325
Revision Clouds 326
Points 328

Defining the Point Display Settings 328
Drawing Points 329
Wipeouts 330
To Create a Wipeout Using Points 330
To Create a Wipeout from a Polyline 330
To Control the Display of the Edges of Wipeouts 331

Direct Modeling 332
3D Constraints 332
Variational Direct Modeling 333
Mechanical Browser 333
3D Constraints types 333
3D Constraints and Direct Modeling 333
Design Intent 334
Direct Modeling Operations 338
Selection Preview 341
Selecting Geometry 341
To Control Dimensions 342
Making Holes 343
Deleting Features 346
To delete a circular hole 346
To delete a rectangular hole 346
To delete a niche or recess 347
To delete a hole in a surface 347
To delete a depression (hole or pocket) or a protrusion feature (extrusion or rib) 347
Removing Edges Between Coplanar Faces 348
Copying Features 349
Deformable Modeling 350

Working with Sections 351
SectionPlane command 351
Align Section Plane to a Face 351
To Create a Vertical Section Plane 352
To Create a Vertical Jogged Section Plane 352
To Create a Horizontal Section Plane 353
To Create a Horizontal Jogged Section Plane 353
To Create a Orthographic Section Plane 354
To open the Drawing Explorer - Section Planes dialog 355
To edit the properties of a section plane in the Drawing Explorer 356
To Modify the Size and Position of a Section Plane 357
Using Live Section 360
To set the *Live Section* property of a section plane: 360
To define the live section settings 360
To Save a Section 362
To launch the SectionPlaneToBlock command 362
To Insert a Section as a 2D Block 363
To Insert a Section as a 3D Block 363
To Replace an Existing block 363
To Export a Section to a File 364

Hatches and Gradient Fills 365
Defining the MEASUREMENT setting 365
Defining the MAXHATCH setting 366
General procedure to create hatches and gradient fills 366
To Define Hatch Properties 367
To Define Gradient Fill Properties 370
To define the boundary for a hatch or gradient fill 371
Set the Hatch and Gradient options 372
Editing Hatches and Gradient Fills 373
To edit a hatch or gradient fill in the properties bar 373
Editing Hatches and Gradients in the Properties Bar 375
To edit a hatch in a dialog box 377
To edit a gradient fill in a dialog box 378
To edit a hatch or gradient fill using the Quad 379

Dimensions **380**
Associative Dimensions 382
Dimension Styles Explorer 382
Exploring dimension styles 383
Comparing Dimstyles 384
Exploring Dimension Styles 385
Dimension Settings 386
To list the current status of the dimension variables 386
Dimension Styles 386
To Create a New Dimension Style 386
To create a dimension child style 387
To Edit a Dimension Style 387
To Define Dimension Style Overrides 387
To Rename a Dimension Style 388
To Set a Dimension Style Current 388
Using the Status Bar to Set a Dimension Style Current 388
To Copy Dimension Styles Between Drawings 388
To Move a Dimension Style Between Drawings 389
To Delete a Dimension Style 389
Dimension Style Tools 390
To apply the current dimension style 390
Creating Linear Dimensions **390**
To Create a Horizontal or Vertical Linear Dimension 390
To Create an Aligned Linear Dimension 391
To Create a Rotated Linear Dimension 391
To Create Baseline Dimensions 392
To Create Continued Dimensions 392
To Create an Arc Length Dimension 393
Creating Angular Dimensions 393
To Dimension an Angle Between Two Lines 394
To Dimension an Angle Encompassed by an Arc 394
Creating an angular dimension defined by a vertex and two endpoints 395
Creating Diametrical and Radial Dimensions 395
To Create a Diametrical Dimension 395
To Create a Radial Dimension 395
Creating Center Marks and Center Lines 396
Centerlines and Center Marks 396
System variables for center entities 397
Ordinate Dimensions 399
To create an ordinate dimension 399
Leaders and Annotations 399
To Create a Leader and an Annotation 399
To Create a Leader only 400
Multileaders 401
To create a multileader 401
Adding vertices to a leader line of a multileader 403
Removing vertices from a leader line of a multileader 403
Adding leader lines to a multileader 403
Removing leader lines from a multileader 403
Editing dimensions **404**
Using Grips to Edit a Linear Dimension 404
Editing the Dimension Text 404
To edit the dimension text in the MText editor 405
To edit the dimension text in the Properties Bar 405
To edit the dimension text using the Edit Dimension Text tool 405
To Rotate the Dimension Text 405
To Reposition the Dimension Text 406
To Restore the Text Position 406
To Make the Extension Lines Oblique 406

Geometric Tolerances **407**
Geometric Tolerance Symbols 407
Geometric Tolerance Dialog 407
To Create a Geometrical Tolerance Frame 407
Working with Text **409**
Text Variables 409
Working With Text Styles 410
Open the Text Style Explorer Dialog 410
Create a Text Style 410
Edit a Text Style 410
Make a Text Style Current 410
Delete a Text Style 411
Creating text 412
To Create Single Line Text 412
To Create Multiline Text 413
Text formatting toolbar tools and settings 414
Multiline Text Justification 415
Special Characters and Symbols 415
To Add a Background Mask 416
Editing text 417
To edit a text entity 417
To edit a selection of Mtext entities 418
To select a different multiline text editor 418
Exploding text 418
Finding and Replacing Text 419
To Open the Find and Replace Dialog 419
To Set the Find and Replace Options 419
To Find Text 420
To Replace Text 420
Checking Spelling 421
Dictionary Settings 421
To Open the Spell Checking Dialog 421
To Check the Spelling 422
To Change Dictionaries 423
To Add a New Main Dictionary 424
To Add a New Custom Dictionary 426
Working with Fields **427**
General Procedure to Create Fields 427
To Display an Entity Property in a Field 429
To Edit a Field 431
To Update a Field 431
Convert Field To Text 431
Working with Tables **432**
Table Styles 432
To open the Table Styles Explorer 432
To Create a Table Style 432
To Edit a Table Style 433
To Apply a Table Style 433
To Delete a Table Style 434
To Create an Empty Table 434
To Create a Table from Data 435
To Create a Table from an Excel Spreadsheet 435
Editing a Table Using Grips 436
Editing a Table Using the Table Toolbar 436
Editing a Table in the Properties Bar 439
Editing a Cell in the Properties Bar 440
Evaluating Formulas 441
To Edit the Table Content 441
To Export a Table 441
Defining the list separator character 441

Annotative Entities **443**
Commands 443
System Variables and User Preferences 443
Setting the Current Annotation Scale 444
Making an Annotation Entity Annotative 444
Annotation Scales 444
To add or remove annotation scales to annotative entities 445
Adding or Removing Annotation Scales to Annotative Entities 446
Annotative Texts 446
To make a text entity annotative 446
To remove the annotative property 446
Annotative Hatches 447
To set the HPANNOTATIVE system variable 447
To make a hatch or gradient entity annotative 447
To remove the annotative property 447
Annotative Blocks 447
To make a block definition annotative 447
To match the orientation of a block to a layout 447
Annotative Dimensions 448
To make a dimension entity annotative 448
To remove the annotative property 448

Modifying Entities **449**
Entity Modification Commands 449
Entity Modification Settings 451
Match Properties 451
To copy properties between entities 451
Grips editing **453**
Selecting Multiple Grips 453
The grips editing context menu 454
To Stretch Entities Using Grips 454
To Move Entities Using Grips 455
To copy entities using grips 455
To Mirror Entities Using Grips 456
To Rotate Entities Using Grips 457
To Scale Entities Using Grips 457
Selecting Entities **458**
Grips Settings 462
Selection Methods 463
Composing a Selection Set Before (pre- pick) 463
Composing a Selection Set Afterwards (post-pick) 464
Selecting Overlapping Entities 465
Using Quick Select 465
To create a selection set 465
The Structure Panel 470
To open the Structure panel 470
To configure the tree-structure on the Structure panel 472
Rearranging Entities **476**
Draw Order 476
Using Draw Order 476
Using Draw Order tools 476
Manipulator 477
The manipulator settings 477
To control the location and orientation of the manipulator 478
To display the manipulator when multiple entities are selected 479
To move the manipulator 479
To rotate the manipulator 479
To reorient the manipulator 480
To align the manipulator with a coordinate system 480
To align the manipulator with the face of a solid 480
Moving Entities 481
Moving entities in a drawing 481

Moving Entities Using Grips ... 481
Moving entities between drawings ... 482
Move a Selection as a Block ... 483
Rotating Entities ... 483
Rotating a Selection Set ... 483
Rotating a Selection Set in Reference to a Base Angle ... 484
Rotating in 3D ... 485
Mirroring Entities ... 485
Mirroring Entities about a Line ... 485
Mirroring Entities about a Plane ... 486
Aligning Entities ... 486
Aligning an Entity in 2D ... 486
Aligning an Entity in 3D ... 487
Changing Entities ... 489
To Change Text Entities ... 489
To Relocate Blocks ... 489
Copying Entities ... 490
Copying Entities in a Drawing ... 490
To Copy a Selection Set ... 490
To Make Multiple Copies ... 490
To Paste a Selection Set as a Block ... 491
Copying Entities Between Drawings ... 492
Using Copy and Paste ... 492
Using Copy and Pasteorig ... 493
Using Copybase and Paste ... 493
Paste as a Block ... 494
Making Parallel Copies ... 494
Making a parallel copy at a specified distance ... 494
Making a parallel copy through a point ... 495
Making a parallel copy of a 3D solid edge ... 496
Making a parallel copy a 3D solid face ... 496
Making a parallel copy of a detected boundary ... 496
Arraying Entities ... 496
Creating a Rectangular Array ... 497
Creating a Polar Array ... 499
Creating a 3D Rectangular Array ... 500
Creating a 3D Polar Array ... 501
Resizing Entities ... 503
Extending Entities ... 503
To extend entities ... 503
Trimming Entities ... 504
To trim entities ... 504
Changing the Length of an Entity ... 505
To Modify the Included Angle of an Arc ... 505
Stretching Entities ... 506
Stretching Entities using Grips ... 507
Scaling Entities ... 508
Scaling an Entity Using a Base Scale ... 508
To Scale a Selection Set ... 509
Breaking Entities ... 509
Joining Entities ... 509
To join colinear lines ... 510
To join two arcs ... 510
Chamfering and Filleting ... 511
Chamfering Using the Distance Distance Method ... 511
Chamfering Using the Length Angle Method ... 512
Chamfering All Vertices of a Polyline ... 512
Filleting Entities ... 513
Filleting Two Entities or Polyline Segments ... 513
Filleting all Vertices of a Polyline ... 514
Filleting two Parallel Lines ... 515

Editing Polylines **516**
Converting an entity into a polyline 516
Opening and Closing Polylines 516
To close or open a polyline 517
Opening and closing a polyline using the BricsCAD Properties bar 517
Joining Polylines 518
To join an arc, line, or polyline to an existing polyline 518
To Apply a Uniform Width to Polyline 518
To Taper a Polyline Uniformly along its Length 519
Editing a Polyline Segment 520
Editing a Polyline Using the Quad 520
Starting the Polyline Vertex Editing Mode 521
Converting a Straight Polyline Segment into a Curve 521
Breaking a Polyline into two Separate Polylines 522
Inserting a New Vertex 522
Moving a Vertex 523
Deleting Vertices 523
Changing the Width of a Polyline Segment 523
Curving and Decurving Polylines 524
To fit a curve to a polyline 524
Setting the Linetype Mode 525
Editing Polylines in the Properties Bar 526
To move polyline vertices 527
To change the width of polyline segments 528
To convert straight segments into a curve 528
Converting Entities **529**
Exploding Entities 529
To explode entities 529
Creating Regions 529
Measuring and Dividing Entities **530**
Measuring Entities 530
To measure an entity using points 530
To place blocks a specified interval along an entity 530
Dividing Entities 531
To divide an entity using blocks 532
Blocks, Attributes and External References **534**
Working with Blocks **534**
General procedure to create a block 534
To Create a Block in a Drawing 534
To Create a Block Using the Blocks Explorer 536
To Save a Block to a Separate File 536
Inserting Blocks 538
General procedure to insert a block: 538
To Insert a Block 539
To Insert a Block Using the Blocks Explorer 540
To Insert a Block Aligned with an Entity 543
To Insert Multiple Instances of a Block in a Rectangular Array 543
Redefining Blocks 545
See also: Editing blocks and external references 545
To Redefine an Internal Block Definition 545
To Reload an External Drawing as a Block 545
Exploring Blocks 546
Open the Blocks Explorer 546
The Blocks Explorer display options 546
Open the Detail View of the Blocks Explorer 546
Open the Icon View of the Blocks Explorer 547
To rename a block 548
Exploding Blocks 548
Setting the Explodable property of a block 548
Dynamic Blocks 549
Edit a Dynamic Block 549
Grip Editing Dynamic Blocks 550

Working with attributes 551
General procedure to define attributes: 551
To Define an Attribute..... 551
Sheet Set Properties in a Title Block..... 553
Editing Attribute Definitions 555
To edit an attribute definition 555
Using the Attribute Editor 556
Editing attributes in the Properties bar 558
Placing Blocks with Attributes..... 558
To place a block with attributes 558
Extracting Attribute Information 559

Working with external references 562
Command: XATTACH.....563
To attach an external reference 563
Managing Xrefs 565
To detach external references 566
To unload external references 566
To reload external references 566
Editing Blocks and External References.....566
To edit Xrefs or blocks..... 566
Working with Groups.....569
To Create a Group 569
To Modify a Group 570
To Ungroup Entities 570
To Change the Order of Entities..... 570
Working with Underlays 572
Inserting a PDF Underlay 573
Controlling the Display of a PDF Underlay 573
Setting the Layer Display in a PDF Underlay..... 573
Clipping a PDF Underlay 575
To Toggle the Display of the Clipped Part..... 575
Deleting a Clipping Boundary 575
To adjust the display settings of a PDF underlay 576

Managing Images 577
Open the Images Explorer 577
Open the Detail View of the Images Explorer 577
Open the Icon View of the Images Explorer 578
Open the Tree View of the Images Explorer 578
Placing Images in a Drawing 579
Load/Unload Images 580
Removing images 582
Detach Images..... 582
Image Transparency 582
Image Frame Setting 583
To Clip an Image 583
To Toggle the Clipping Boundary of an Image 584

Printing and Plotting 584
To Open the Print Dialog..... 585
To Choose the Paper Size 586
To Choose the Paper Units 586
To Choose the Paper Orientation 586
To Define the Plot Offset..... 586
To Specify the Area to Plot..... 587
To Define the Plot Scale..... 587
To Set the Plot Options..... 588
To Set the Shade Plot Mode 588
To Define the Plot Stamp 589
Variables 590
To Specify the Number of Copies 590

To See a Preview of the Plot Output ... 590
Plotter Configurations ... 591
To Configure the Built-in PDF Printer ... 591
To Create a Plotter Configuration File ... 595
To Edit a Plotter Configuration ... 598
To Assign a Plotter Configuration ... 599
Plot Style Tables ... 599
To open the Plot Style Table editor ... 599
The General tab ... 600
The Form View tab ... 601
Assigning Plot Style Tables ... 604
Missing plot style table ... 604
To Assign a Plot Style Table ... 604
To Toggle the Plot Style Table on/off ... 605
The Plot Styles Manager ... 605
To Open the Plot Styles Manager ... 605
To Create a New Plot Style Table ... 605
To Create a New Plot Style Table in the Print Dialog ... 608
To Edit a Plot Style Table ... 608
To Edit a Plot Style Table in the Print Dialog ... 608
Batch Plotting ... 608
Rendering ... 609
Setting the Render Preferences ... 609
Render Tools ... 610
To Render a Viewport ... 611
To Render in a Window ... 612
To Save a Rendering to a File ... 613
Render Materials ... 613
Glossary ... 614
Defining Materials ... 616
Assigning Materials ... 616
To assign a material to entities ... 616
Lights ... 617
Glossary ... 617
System Variables ... 617
General Properties ... 618
Photometric Properties ... 618
Geometry Properties ... 619
Attenuation Properties ... 619
Rendered Shadow Details ... 620
Spotlight Properties ... 620
Open the Light List ... 622
To Define a Light ... 623
To Edit a Light ... 624
To Define the Sun Properties ... 626
To Define the Geographic Location ... 626
Digitizing Tablet ... 629
Using a digitizing tablet in BricsCAD ... 629
To download the tablet overlay and CUI files ... 629
To load the overlay CUI file ... 629
To switch the tablet ON / OFF ... 630
To configure the tablet ... 630
To calibrate the tablet ... 631

Mechanical 633
Commands and Toolbars 633
Components and Inserts 635
Definitions 635
Converting Components 636
Parametric Components 636
To edit the parameters of a parametric component insert 638
To edit a parameter in the Mechanical Browser 639
To edit a parameter in the Properties Bar 639
Replacing Components 639
Components Blocks and Xrefs 640
Assigning a Physical Material 641
Inserting a component with a physical material 641
Organizing Geometry 642
Changing Mechanical Structure 642
Using Standard Hardware 642
Assembly Constraints 642
Component Visibility 642
Mechanical Browser 643
To open the Mechanical Browser: 643
Bill of Materials 644
3D Comparison 645
Exploded Representations 648
Levels of the Exploded Representation 648
Algorithms 648
Creating an Exploded Representation 650
Generating Drawing Views 651
Updating an Exploded Representation 651
Assembly Mass Properties 652
To Set the Mass Units 652
To Compute the Mass Properties 653
Parametric Modeling 654
Global and local parameters 654
Setting an expression to a parameter 654
Driving and driven parameters 655
Parameters of Associative Arrays 656
Parameters of Mechanical Components 657
Design Tables 658
Creating a Design Table from a .csv file 658
Direct Lookup in a Design Table 658
Reverse Lookup in a Design Table 659
Adding a New Configuration to a Design Table 659
Global and Local Parameters 660
Setting an Expression to a Parameter 660
Driving and Driven Parameters 661
Parameters and Associative Arrays 662
MCAD Tutorial 664
Assembly Design and Kinematic Analysis Tutorial 664
Step 1: Preparing for the Exercise 664
Step 2: Creating the Assembly Drawing 666
Step 3: Adding the Components 667
Step 4: Positioning the Components 668
Step 5: Kinematic Analysis 675
BricsCAD Communicator 677
Supported file formats 677

BIM 679
Creating Walls 679
Creating a Single Wall 680
Creating a Chain of L-Connected Walls 680
Connecting Solids 682
Creating L-Connections 683
Creating T-Connections 683
Creating T-connections ply-by-ply 684
Modifying the Thickness 685
Modifying the Height of a single wall 686
Modifying the height of multiple walls 686
Setting the Distance between Parallel Solids 688
Creating a Parallel Copy of a Solid 688
Creating a Partial Copy of a Solid 689
Creating Slabs 689
Creating Floor Slabs 689
Creating Pitch Roof Slabs 689
Windows and Doors 692
Inserting Windows and Doors 692
Method 1: Use dynamic dimensions fields: 694
Method 2: Specify a point 695
Editing Parameters 695
Repositioning an Insert 697
Moving an Insert Vertically 697
Moving an Insert Horizontally 698
Copying Inserts 698
Replacing Inserts 698
Creating Windows and Doors 699
Creating the 3D Geometry 699
Adding 2D Entities 704
Adding 3D Constraints 704
To apply Fix constraints 705
Using Parameters 706
To create a parameter 706
Using parameters 707
Creating Standard Windows 707
Editing Solids 709
Splitting Solids 709
To split a solid automatically 709
Using the Quad 709
To split a solid using cutting faces 709
BIM Database 711
Project Database and Library Database 711
Materials and Compositions 711
To open the BIM Libraries panel 712
To Open the BIM Project Info Dialog 715
To Set the Library Database 716
To Set the Project Database 717
To Set the Project Filter 717
Physical Materials 719
To Open the Physical Materials Editor 719
Adding Physical Materials 719
Adding a physical material from scratch 719
Adding a physical material as a copy of an existing material 719
Editing Physical Materials 720
To copy materials between the Project and Library databases. 724
Deleting Physical Materials 725
Compositions 726
To Open the Composition Editor 726
The Composition Thickness 727

Adding Compositions 727
Adding a composition from scratch 727
Adding a composition as a copy of an existing composition 727
Creating single-ply compositions 727
Editing a Composition 728
To modify the thickness of a ply. 728
To change the material of a ply. 729
To add a ply. 729
To modify the order of the plies. 729
To delete a ply. 729
Deleting Compositions 729
Attaching a Composition to Solids 730
Dragging a composition onto a single solid 730
Dragging a composition onto a selection of solids 730
Displaying the composition plies 731
BIM Sections 732
Defining BIM Sections 732
To launch the BIM Section command 732
General Procedure to Create a BIM Section 733
Defining the BIM Section Entity Properties 734
Defining the section entity properties 738
To Define a Plan Section 741
To Define a Cross Section 742
To Define an Elevation 743
To Define a Detail Section 744
To define a detail section from scratch 744
To define a detail section based on an existing section 744
To Create a 2D Isometric Drawing 745
Turning a Section Plane into a BIM Section 745
Creating a Jogged BIM Section 746
Using Clip Display 746
To toggle Clip Display 746
To edit the display properties of a section entity in the Properties Bar 747
Generating Drawings 749
To Generate or Update Sections 749
To Open a Generated Drawing 751
To Update a BIM Section 751
Update a BIM section in Model Space 751
Update a BIM Section in a layout 751
Editing Drawings 752
Inserting the Title Block 752
Inserting a View Label for a Sheet View 753
Inserting a Number Bubble for a Sheet View 754
Inserting a Sheet List Table 755
Customizing Callout Blocks 755
Spatial Locations 756
Assigning a Building and Story to a Building Element 756
Assigning a building and story to a building element 756
To examine the model in the Structure panel 756

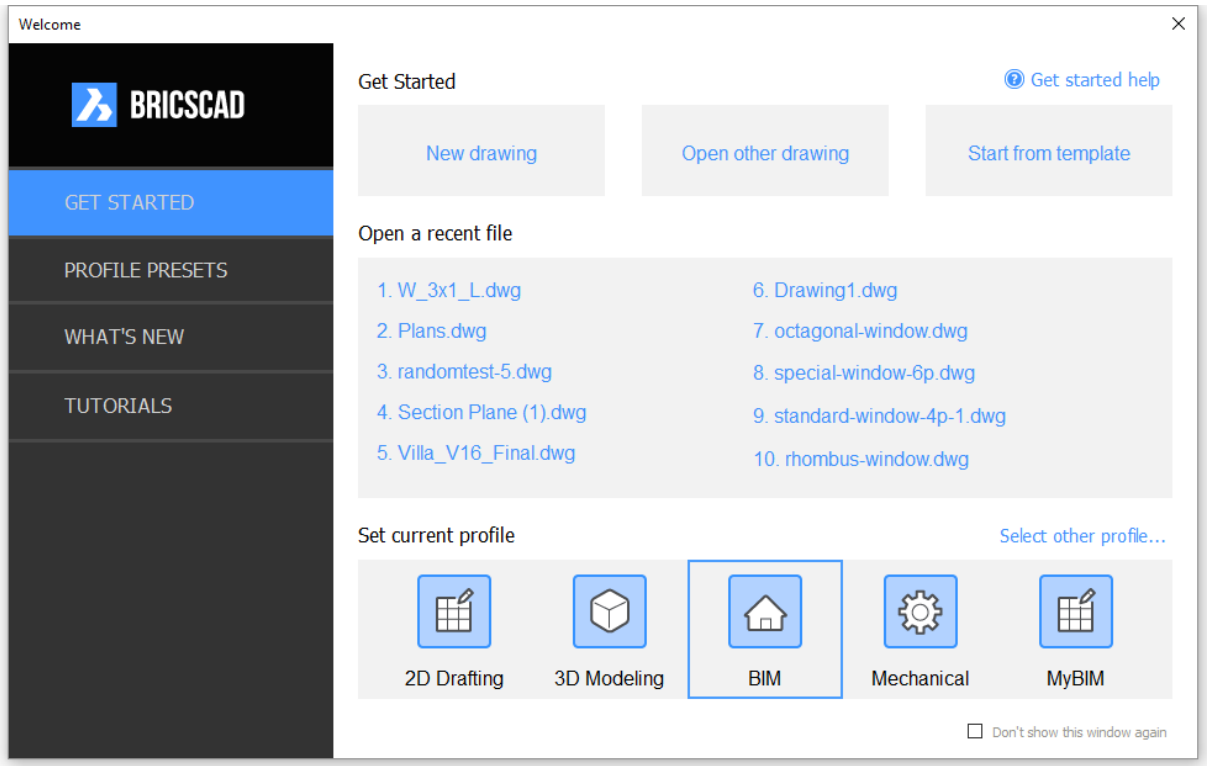
Sheet Metal 757
Commands and Toolbars 757
SheetMetal Tutorials 761
Definitions and Settings 763
Sheet Metal Features 763
K-Factor 766
To change the K-Factor for a sheet metal part: 766
Bend Table 768
The Mechanical Browser for Sheet Metal 773
Feature Colors 774
Layer Properties of 2D Geometry 775
Managing Initial Parameters 776
Sheet Metal Operations 777
Creating the Base Flange 777
Creating Edge Flanges 778
Creating Tapered Edge Flanges 781
Creating Multiple Flanges 782
To Bend a Sheet Metal Body 785
Corner Relief and Junctions 787
To Rotate a Flange 787
Creating Lofted Bends 788
Converting Geometry to Sheet Metal 790
To Connect Flanges 791
To Create Missing Reliefs 791
Switching the Bend Relief Type 792
Using the ribbon, menu or toolbar 792
Using the Quad 793
Switching the Corner Relief Type 793
Using the ribbon, menu or toolbar: 793
Using the Quad: 794
Converting hard edges to bends or junctions 794
Converting Hard Edges to Bends 794
Converting Hard Edges to Junctions 795
To Split a Flange 795
Smart Tools to Split a Flange 796
To Split a Thickness Face 798
To Repair a Sheet Metal Part 800
To Repair a Sheet Metal Body 800
To Repair a Sheet Metal Body with Lofted Bends 802
Changing the Thickness 802
To change the overall bend radius: 803
To change the bend radius for a particular bend: 803
Changing the Junction Gap 804
To change the overall junction gap: 804
To change the junction gap for a particular bend: 804
Changing the Junction Type 804
To change the junction type using the Quad: 805
To change the junction type in the Mechanical Browser 805
To Unfold the Sheet Metal Body 806
To Export the Sheet Metal Body 809
To Export an Sheet Metal Assembly to DXF 809
Working with Form Features 810
To Insert a Form Feature 810
Editing the Parameters of a Library Form Feature 812
Editing parameters in the Properties Bar 813
Editing parameters in the Mechanical Browser 814
Creating a Custom Form Feature 815
Creating a Custom Parametric form Feature 817
Creating a Rib Feature 827
Modifying a Rib Feature 828
Edit the properties in the Mechanical Browser 829

Get Started

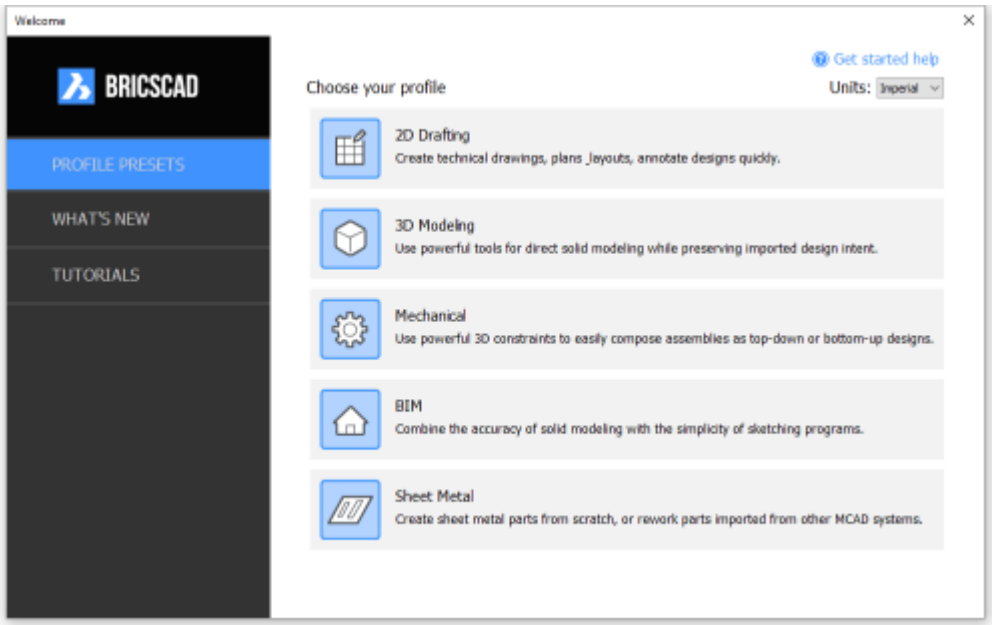
The Get Started Dialog

If the GETSTARTED system variable is *ON*, the *Welcome* dialog box displays when you launch BricsCAD. Here you can choose the User Profile for the current session. You can choose between a number of *Presets* or user defined profiles.

The most recently used profile is indicated by a blue frame.



NOTE The very first time BricsCAD is launched the *Get Started* option is missing on the *Welcome* dialog. The *Profile Presets* page is shown instead. After selecting a profile preset the *Get Started* dialog displays with the new profile selected in the *Set current profile* list.



To start a BricsCAD session

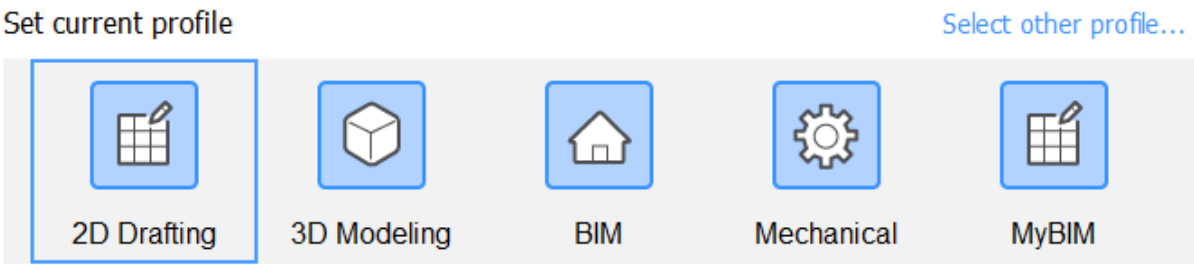
1. Accept the currently selected profile or select a profile in the *Set current profile* list.
2. Do one of the following:

- Click the *New Drawing* button to create a new drawing using the default template of the current profile.
- Click the *Start from template* button to create a new drawing using a template of your choice.
- Click the *Open other drawing* button to open an existing drawing.
- Click a drawing in the *Open a recent file* list.
  The number of files in this list is controlled through the *Recent file list* (NFILELIST) system variable. Up to 20 files can be listed here.

To select the startup user profile

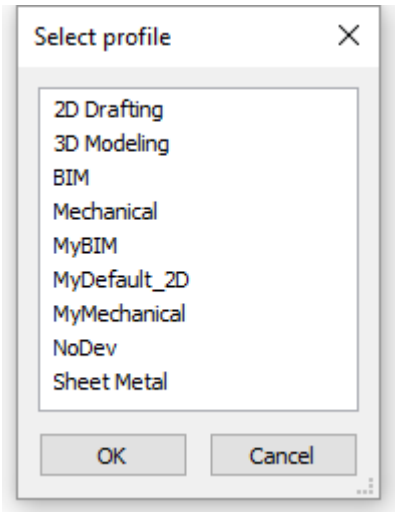
Do one of the following:

- Click a profile in the *Set current profile* list.



  By default the five most recently used profiles display. If necessary, drag the right hand edge of the *Welcome* dialog box to see more profiles.

- Click *Select other profile...*, then select a profile in the *All profiles* dialog and click the *OK* button.

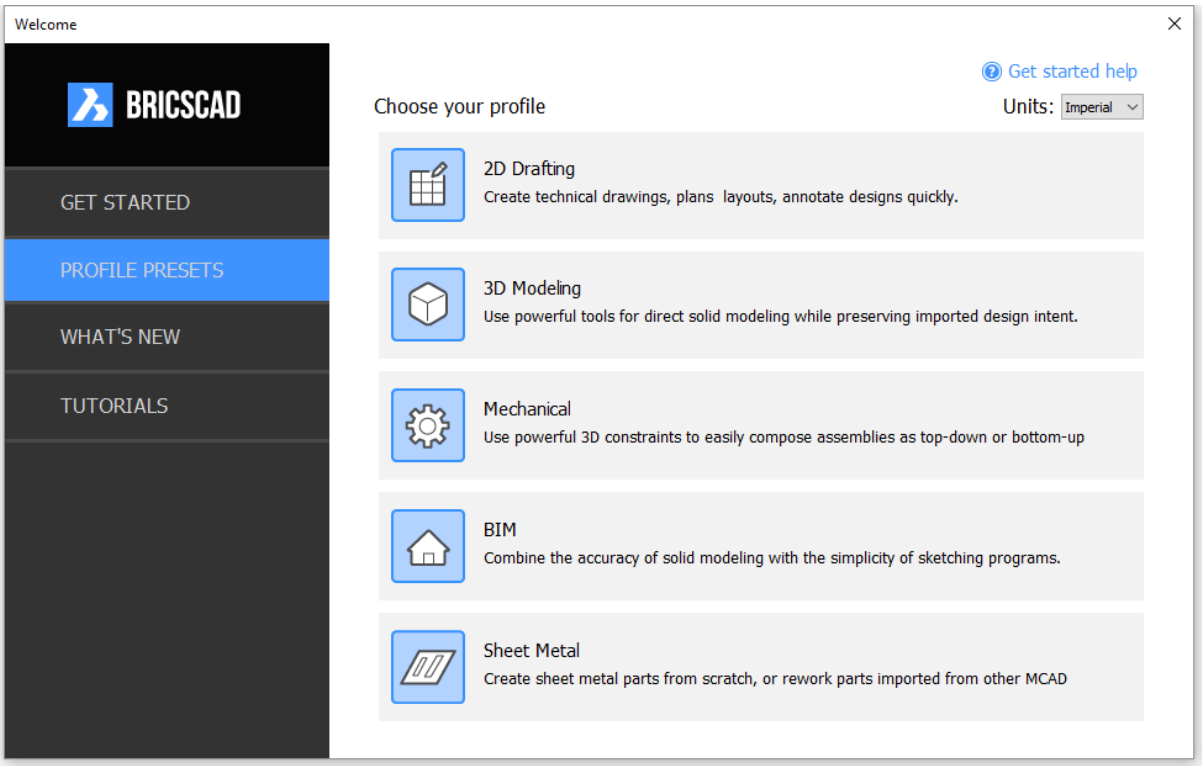


  The selected profile is added to the *Set current profile* list.

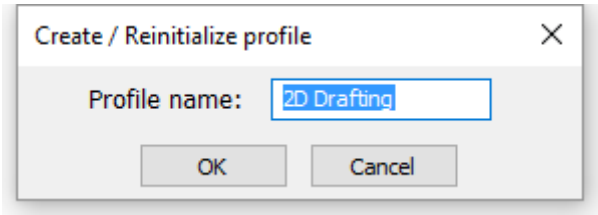
Using a Profile Preset

BricsCAD comes with a set of built-in hard coded profiles. You can use these Preset Profiles as templates to create your own set of user profiles. Each of the profile presets has the name of the Workspace they use.

1. Click *PROFILE PRESETS* on the *Welcome* dialog box.

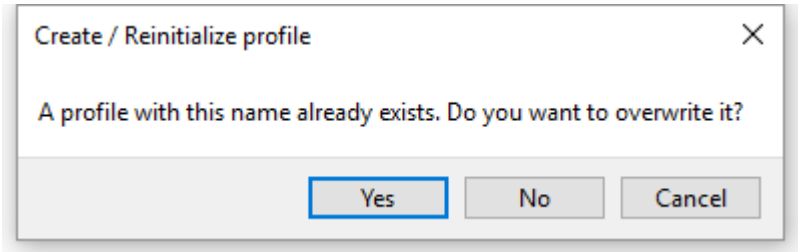


2. Choose either *Imperial* or *Metric* in the Units list button to set the MEASUREMENT system variable in the profile.
3. Select a *Profile Preset*, e.g. *2D Drafting*.
   A dialog box displays.



4. Type a new name in the *Profile name* field to create a new profile using the hard-coded settings of the selected Profile Preset.

   If you type the name of an existing profile a dialog box prompts you to confirm to overwrite the existing profile using the hard-coded settings of the selected Profile Preset.



The *Get Started* dialog displays with the new profile selected in the *Set current profile* list.

To Start a BricsCAD Session

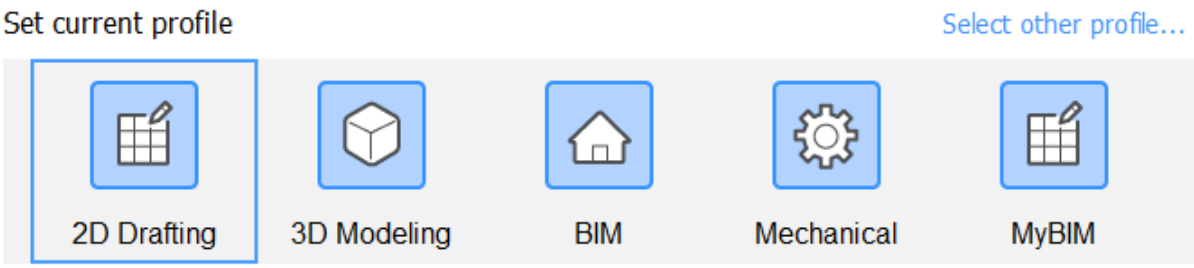
1. Accept the currently selected profile or select a profile in the *Set current profile* list.
2. Do one of the following:

- Click the *New Drawing* button to create a new drawing using the default template of the current profile.
- Click the *Start from template* button to create a new drawing using a template of your choice.
- Click the *Open other drawing* button to open an existing drawing.
- Click a drawing in the *Open a recent file* list.
  The number of files in this list is controlled through the *Recent file list* (NFILELIST) system variable. Up to 20 files can be listed here.

To Set the Startup User Profile

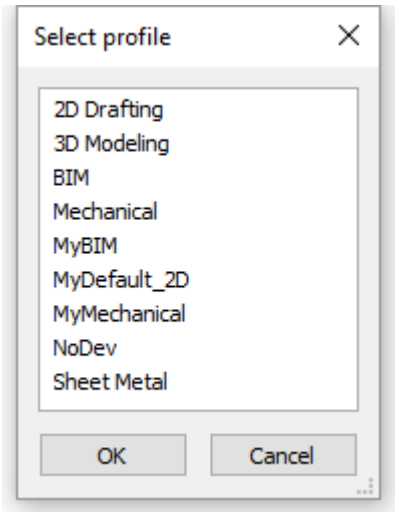
Do one of the following:

- Click a profile in the *Set current profile* list.



  By default the five most recently used profiles display. If necessary, drag the right hand edge of the *Welcome* dialog box to see more profiles.

- Click *Select other profile...*, then select a profile in the *All profiles* dialog and click the *OK* button.

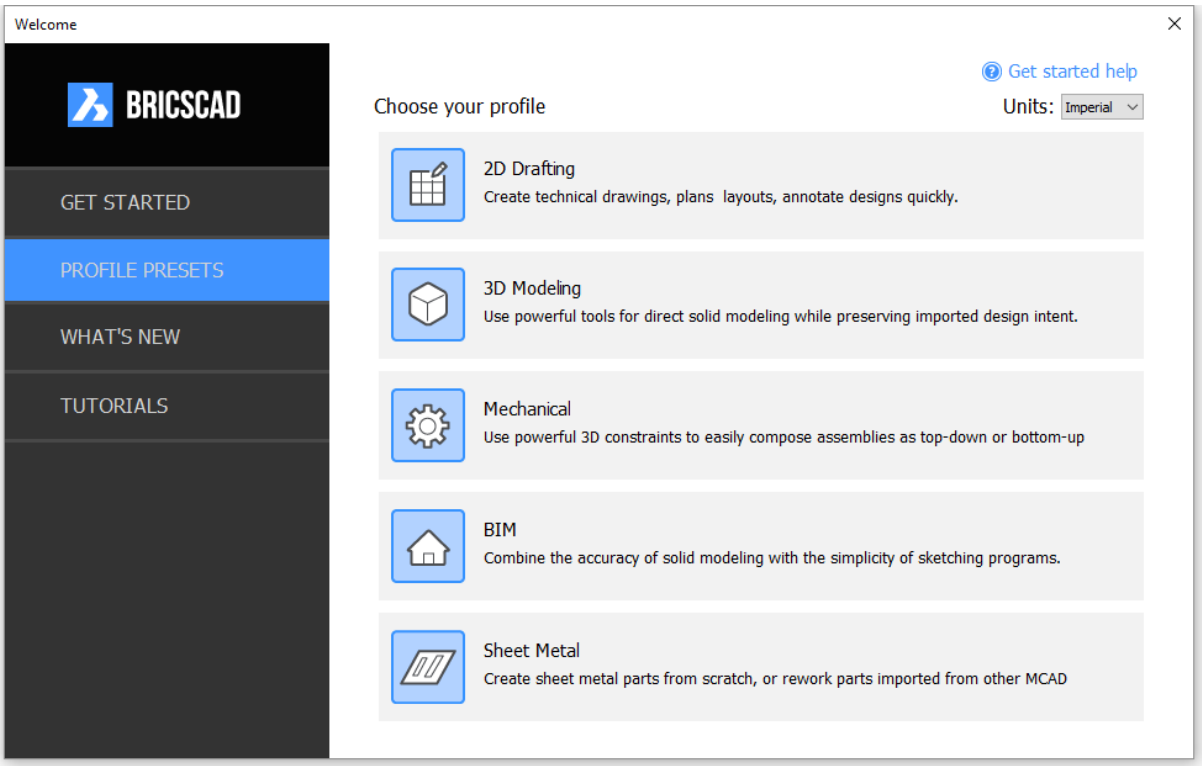


  The selected profile is added to the *Set current profile* list.

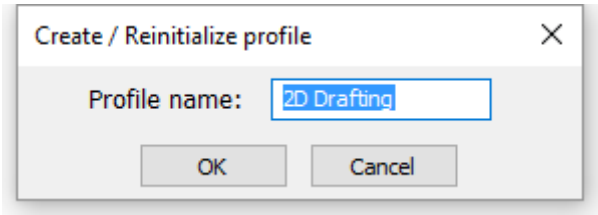
Using a Profile Preset

BricsCAD comes with a set of built-in hard coded profiles. You can use these Preset Profiles as templates to create your own set of user profiles. Each of the profile presets has the name of the Workspace they use.

1. Click *PROFILE PRESETS* on the *Welcome* dialog box.

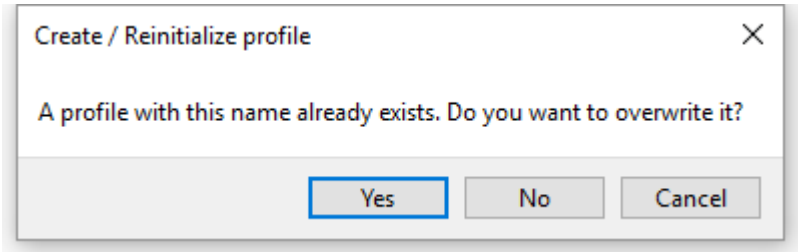


2. Choose either *Imperial* or *Metric* in the Units list button to set the MEASUREMENT system variable in the profile.
3. Select a *Profile Preset*, e.g. *2D Drafting*.
   A dialog box displays.



4. Type a new name in the *Profile name* field to create a new profile using the hard-coded settings of the selected Profile Preset.

   If you type the name of an existing profile a dialog box prompts you to confirm to overwrite the existing profile using the hard-coded settings of the selected Profile Preset.



The *Get Started* dialog displays with the new profile selected in the *Set current profile* list.

User Interface

Keyboard Shortcuts

The following keyboard shortcuts are available:

Keyboard	Description
F1	Opens the BricsCAD Help.
F2	Toggles the display of the Prompt History window.
F3	Toggles Entity Snaps.
F4	Toggles the tablet mode.
F5	Sets the isoplane (see Using Isometric snap).
F6	Toggles the Dynamic UCS feature.
F7	Toggles the display of the grid.
F8	Toggles the Ortho setting. Disables Polar Tracking, if on.
F9	Toggles the Snap setting.
F10	Toggles the Polar Tracking setting. Disables Ortho, if on.
F11	Toggles the Entity Snap Tracking setting.
F12	Toggles the Quad display.
Alt+F8	Launches the VBARUN command.
Alt+F11	Launches the VBAIDE command.
Ctrl+1*	Toggles the properties bar.
Ctrl+2*	Opens the Drawing Explorer.
Ctrl+9*	Toggles the display of the command bar.
Ctrl+0*	Toggles the display of user interface entities as specified by the SCREENCLEANOPTIONS system variable. Executes the CLEANSCREENON and CLEANSCREENOFF commands.
Ctrl+A	Selects all entities.
Ctrl+B	Toggles the Snap setting.
Ctrl+C	Copies the selection to the clipboard (COPYCLIP command).
Ctrl+Shift+C	Copies the selection along with a base point (COPYBASE command).
Ctrl+E	Sets the isoplane (see Using Isometric snap).
Ctrl+F	Launches the FIND command.
Ctrl+G	Toggles the display of the grid.
Ctrl+H	Toggles the value of the PICKSTYLE system variable, which controls the selection of groups and associative hatches.
Ctrl+I	Sets the readout of the coordinate field in the status bar.
Ctrl+J	Repeats the previously issued command.
Ctrl+K	Attach a hyperlink to an object or modify an existing hyperlink (HYPERLINK command).
Ctrl+L	Toggles the Ortho setting.
Ctrl+M	Repeats the previously issued command.
Ctrl+N	Creates a new drawing (NEW command).
Ctrl+O	Opens an existing drawing (OPEN command).
Ctrl+P	Prints the current drawing (PRINT command).

Ctrl+Shift+P	Toggles the properties bar.
Ctrl+Q	Quits the application; prompts to save changes (QUIT command).
Ctrl+R	Iterates through viewports.
Ctrl+S	Saves the current drawing (QSAVE command).
Ctrl+T	Turns the tablet on/off.
Ctrl+V	Pastes the contents of the Clipboard into the drawing (PASTECLIP command).
Ctrl+Alt+V	Pastes the contents of the Clipboard in a specified format (PASTESPEC command).
Ctrl+Shift+V	Pastes the contents of the Clipboard as a block (PASTEBLOCK command).
Ctrl+X	Cuts the selection and puts it on the clipboard (CUTCLIP command).
Ctrl+Y	Redoes the last action undone (REDO command).
Ctrl+Z	Undoes the last action (UNDO command).
Ctrl+[	Cancels the running command.
Ctrl+\	Cancels the running command.
Shift+F2	Toggles the display of the command bar.
Shift+F3	Turns the Status Bar on/off (STATBAR command).
Shift+F4	Turns the Scroll Bars on/off (SCROLLBAR command).
Shift+F8	Launches the Visual Basic Project Manager (VBAMAN command).
Shift+F11	Launches the VBA COM Add-In Manager (ADDINMAN command).
PgUp	Moves the view up.
PgDn	Moves the view down.
Shift+Left Arrow	Moves the view to the left.
Shift+Right Arrow	Moves the view to the right.
Shift+Up Arrow	Moves the view up.
Shift+Down Arrow	Moves the view down.

* Do not use the numeric keypad.
On AZERTY-keyboards: do not press the Shift-key when hitting the number key.

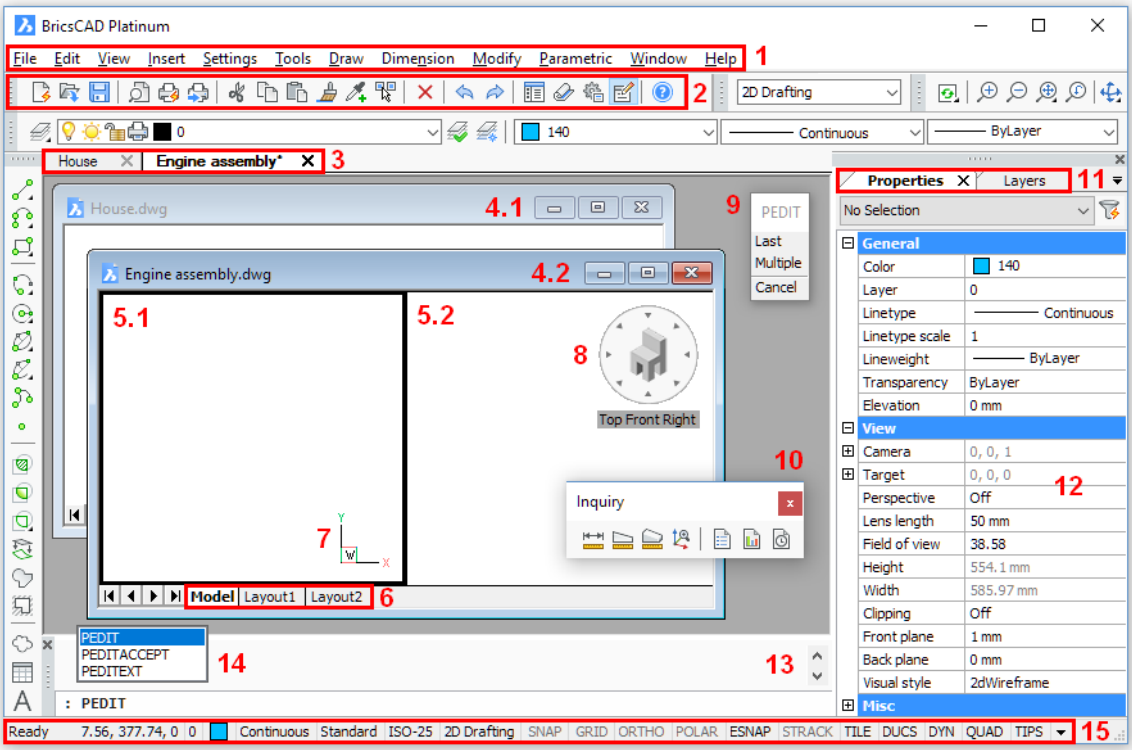
The BricsCAD Application Window

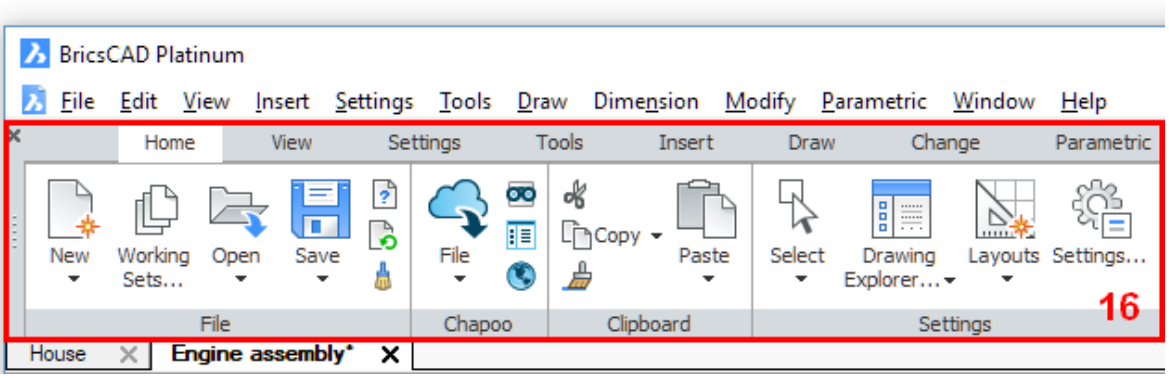
Overview

Customizing the Command Prompt

The layout of the BricsCAD application window can be fully customized.
You can:

- open / close the command window
- display / hide the ribbon
- control the display and position of document tabs
- open / close the status bar
- customize the status bar
- modify drawing settings in the status bar
- display / hide scroll bars
- display / hide panels
- hide / show a selection of user interface elements



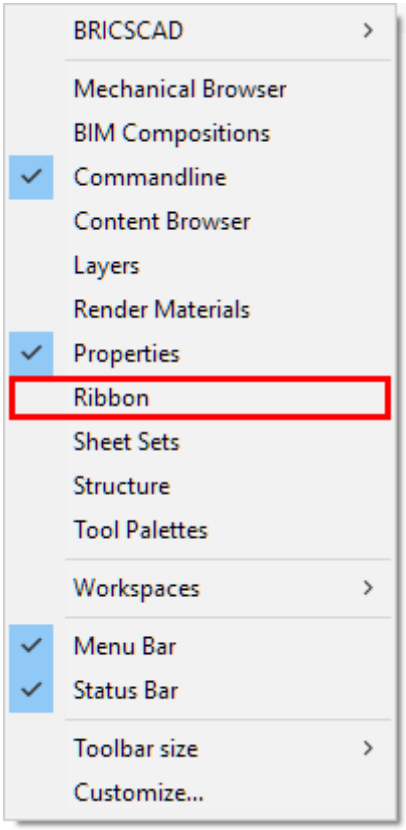


Components of the BricsCAD application window:

1. Menu Bar
2. Toolbar (docked)
3. Document tabs
4. Drawing windows
5. Drawing viewports
6. Window tabs
7. UCS icon
8. Look From widget
9. Prompt Menu
10. Toolbar (floating)
11. Panel tabs
12. Properties panel
13. Command bar
14. Suggestion list
15. Status Bar
16. Ribbon

To Display the Ribbon

- Execute the Ribbon command.
- Right click an empty space in the Layout or Drawing tab area, then choose *Ribbon* in the context menu.



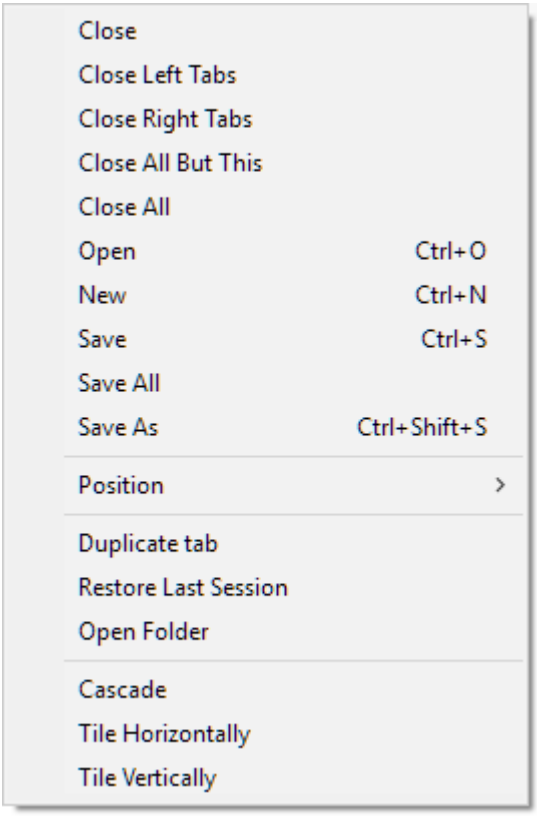
To Hide the Ribbon

- Execute the RibbonClose command.
- Right click an empty space in the Layout or Drawing tab area, then choose *Ribbon* in the context menu.

Document Tabs

Document tabs allow to:

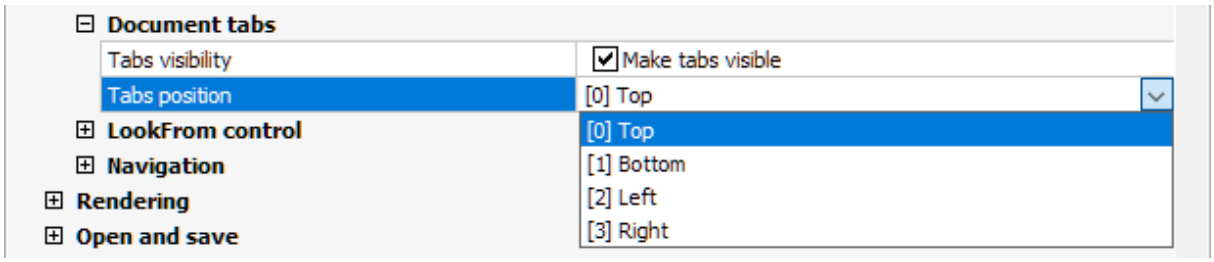
- Easily switch between all open drawings.
- Drag and drop the tabs to change their order.
- Click the *X* at the right-hand side of tab to close it.
- Right click on a document tab to display a context menu:



Item	Description
Close	Closes the current drawing (CLOSE command).
Close Left Tabs	Closes the drawings at the left of the current tab.
Close All But This	Closes all drawings, but the current drawing.
Close All	Closes all drawings (WCLOSEALL command).
Open	Displays the *Open Drawing* dialog (OPEN command).
New	Displays the *Create New Drawing* dialog NEW command.
Save	Saves the current drawing (SAVE command).
Save All	Saves all open drawings (SAVEALL command).
Save As	Launches the SAVEAS command.
Position	Lets you choose the position of the document tabs. Top Bottom Left Right

Duplicate Tab	Saves a copy of the current drawing in a new tab, while keeping the original drawing open. Displays the *Save As* dialog.
Restore Last Session	Opens the drawings that were open in the previous BricsCAD session.
Open Folder	Opens the folder of the current drawing.
Cascade	Cascades the windows of the current drawings (WCASCADE command).
Tile Horizontally	Tiles the windows of the current drawings horizontally (WHTILE command).
Tile Vertically	Tiles the windows of the current drawings vertically (WVTILE command).

The display and location of document tabs is controlled through user preferences.
In the Settings dialog go to *Program Options / Display / Document tabs*.



ShowDocTabs: controls the display of document tabs.
DocTabPosition: controls the location of the document tabs: top, bottom, left or right.

Open/Close the Command Bar

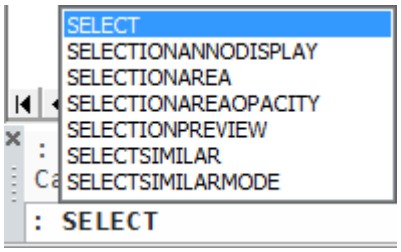
Do one of the following:

- Choose *Command Bar* in the *View* Menu.
- Move the cursor to a docked toolbar, then right click and choose *Command Bar* in the context menu.
- Double click the *Status* field at the left hand side of the *Status Bar*.

The *Command Window* closes if it was open and vice versa.

1. *Close* button
2. Press and hold the left mouse button to move the command bar
3. *Command History List* field: the content of the history list field can be copied to the command edit field.
4. *Command Edit* field
   If the Enable flag of the AUTOCOMPLETEMODE system variable is set, command entries are completed automatically as you type.
   All commands and/or system variables and/or user preferences that contain the current character string display in the *Suggestion* list.

   Do one of the following:
   - Press Enter if the currently completed command displays.
   - Choose a command in the *Suggestion* list.



Autocomplete options are:

Auto complete mode	0x002F (47)
1	☑ Enable
2	☑ Auto-Append
4	☑ Suggestion List
8	☑ Display Icons (unsupported)
16	☐ Exclude the display of System Variables
32	☑ Display preference variables
Auto complete delay	0.3

5. Command history scroll buttons

NOTES

- You can modify the height of the docked command window by dragging its top edge, when docked *Lower* or bottom edge, when docked *Upper*.
- The size of a floating command bar can be adjusted by dragging one of its edges.
- When the command bar is closed, command options and keyboard entries display in the status bar.
- Use the arrow-up and arrow-down keys of the keyboard to browse the command history.

To Set the Command Line Properties

1. Open the Settings window.
2. Go to *Program Options / User Preferences / Command Line.*

Command line	
Command line font name	Verdana
Command line font size	8
Command line list background color	(255,255,221)
Command line list foreground color	(0,0,0)
Command line edit background color	(248,248,248)
Command line edit foreground color	(0,0,0)
Command line state	Command line is visible
Prompt prefix	:
Scroll history	256
Text evaluation	All responses to prompts for text strings and attribute values are taken literally

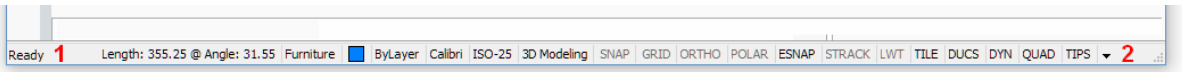
Open/Close the Status bar

Do one of the following:

- Choose *Status Bar* in the *View* Menu.
- Press the *Shift + F3* keyboard shortcut.

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

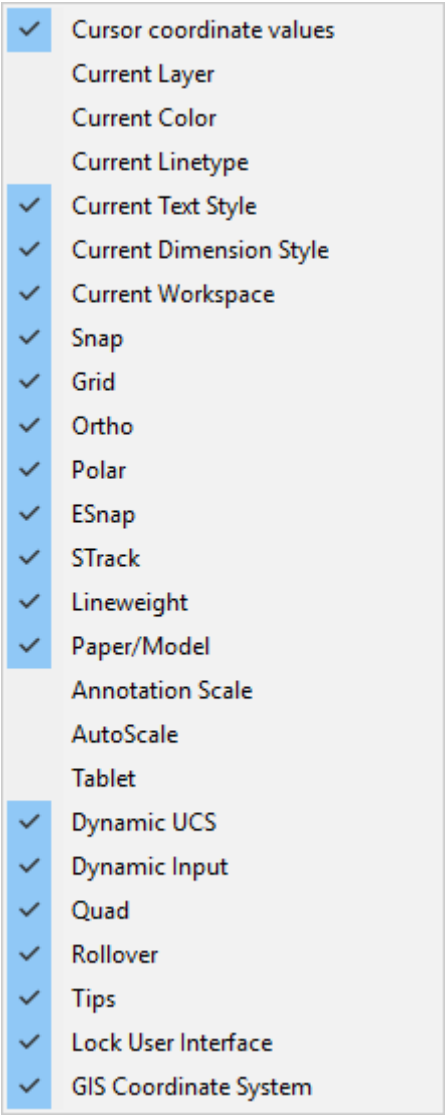
Working with the Status Bar



1. **Status** field:

- displays the status of the software.
- when the cursor is in a menu or on a toolbar: gives a brief description of the tool or menu item under the cursor.
- when the command window is closed: displays the command prompts, tool options and keyboard entry.
- double click the status field to open/close the *Command Bar*.

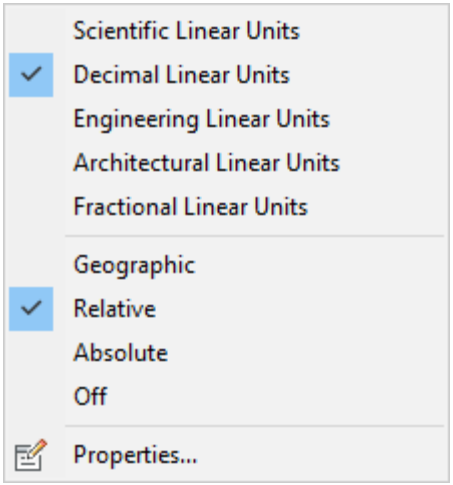
2. **Field List** button: Click the arrow button (▾) to display a list of available *Status Bar* fields. Click an item in the list to toggle the display of the corresponding field in the *Status Bar*.



Cursor coordinate values: displays the coordinates of the current cursor position (depending on the setting of the COORDS variable).

- Click to cycle the *Coordinate* setting. The sequence is *Geopgraphic*, *Relative*, *Absolute, Off.*

- Right click, then choose an option in the context menu.

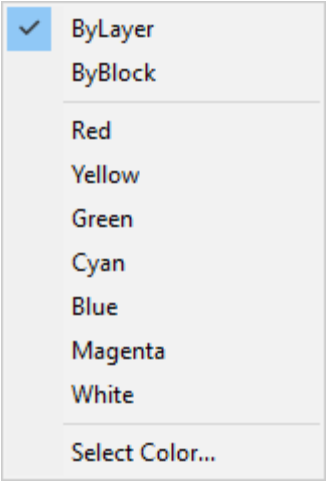


Current Layer: displays the name of the current layer.

- Click to open the *Layer Explorer*.
- Right click to select the current layer in the context menu.
- Right click and choose *Properties* in the context menu to open the Drawing Explorer - Layers.

Current Color: displays the current color.

- Click to open the *Select Color* dialog window.
- Right click to select one of the basic colors.



- Right click and choose *Select Color...* to select a color.

Current Linetype: displays the name of the current linetype.

- Click to open the *Linetype Explorer*.
- Right click to select the current linetype.
- Right click and choose *Properties* in the context menu to open the Drawing Explorer - Linetypes.

Current Text Style: displays the name of the current text style.

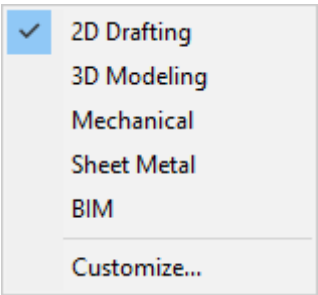
- Click to open the *Styles Explorer*.
- Right click to select the current text style.
- Right click and choose *Properties* in the context menu to open the Drawing Explorer - Text Styles.

Current Dimension Style: displays the name of the current dimension style

- Click to open the *Dimensions Settings* dialog window.
- Right click to select the current dimension style.
- Right click and choose *Properties* in the context menu to open the Drawing Explorer - Dimension Styles.

Current Work Space: Indicates the current workspace.

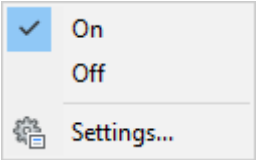
- Click to set the value of the WSCURRENT system variable in the command bar.
- Right click, then select the current workspace.



- Choose *Customize* in the context menu to open the Customize - Workspaces dialog box.

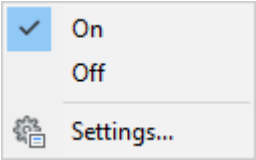
Snap: Toggles the SNAPMODE system variable.

- Click to toggle Snap *On* (SNAP) / *Off* (SNAP).
- Right click, then choose *Settings* to edit the *Snap and Grid* settings.



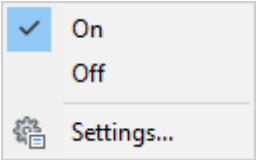
Grid: Toggles the GRIDMODE system variable.

- Click to toggle the display of the Grid *On* (GRID) / *Off* (GRID) in the current viewport.
- Right click, then choose *Settings* to edit the *Snap and Grid* settings.



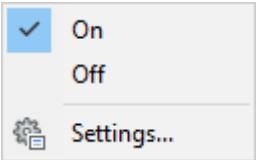
Ortho: Toggles the ORTHOMODE system variable; turning *Orthogonal mode* on automatically disables *Polar Tracking.*

- Click to toggle Orthogonal mode *On* (ORTHO) /*Off* (ORTHO).
- Right click, then choose *Settings* to edit the *Orthomode* setting.



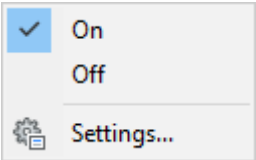
Polar: Toggles the *Polar Tracking* option of the AUTOSNAP system variable.

- Click to toggle Polar Tracking *On* (POLAR) / *Off* (POLAR).
  Turning *Polar Tracking* on automatically disables *Orthogonal mode*.
- Right click, then choose *Settings* to edit the *Polar Tracking* settings.



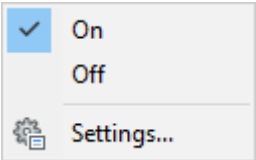
Esnap: Toggles the *Turn of all snaps* option of the OSMODE system variable.

- Click to toggle Entity Snaps *On* (ESNAP) / *Off* (ESNAP).
- Right click, then choose *Settings* to edit the *Entity Snaps* settings.



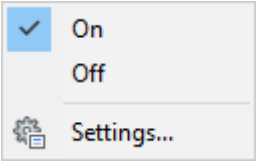
Strack: Toggles the *Entity Snap Tracking* option of the AUTOSNAP system variable.

- Click to toggle Snap Tracking *On* (STRACK) / *Off* (STRACK).
- Right click, then choose *Settings* to edit the *Snap Tracking* settings.



Lineweight (LWT): Toggles the LWDISPLAY system variable.

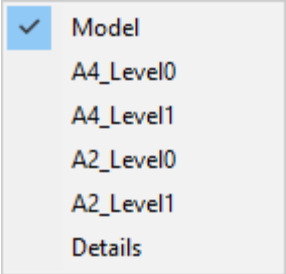
- Click to toggle the display of Line Weights *On* (LWT) or *Off* (LWT).
- Right click, then choose *Settings* to edit the *Lineweights* settings.



Paper/Model: Allows to choose between model space and paper space.

- When working in Model Space, the field reads: *Tile* (= Model space with tiled viewports).

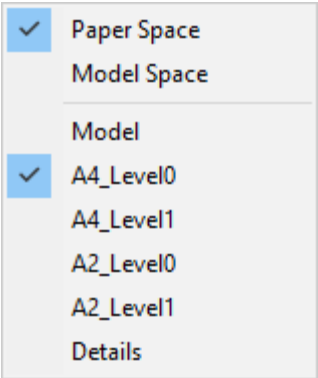
  Click to switch to the most recently opened paper space layout.
  Right click to select a paper space layout:



- When working in a layout, the field reads:
  M:Layout : Model space with floating viewports
  or
  P:Layout : Paper space

  Click to toggle between paper space and model space with floating viewports.
  Right click, then choose:

- *Paper Space*: Switch to paper space
- *Model Space*: Switch to model space with floating viewports
- *Model*: Switch to model space with tiled viewports
- *Layout*: Switch to the selected paper space layout.



Annotation Scale: Displays the value of the CANNOSCALE (= current annotation scale) system variable. Right click:

- Select an annotation scale in the list.
- Choose *Custom* to launch the SCALELISTEDIT command.
- Check the *Hide Xef scales* option to list annotation scales of the current drawing only.

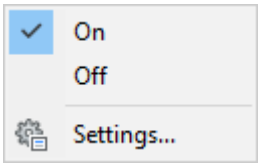
Autoscale: Click to toggle the ANNOAUTOSCALE system variable.

Tablet: Initializes the use of a drawing tablet.

Dynamic UCS (DUCS): Click to toggle the UCSDETECT system variable On (DUCS) or Off (DUCS).

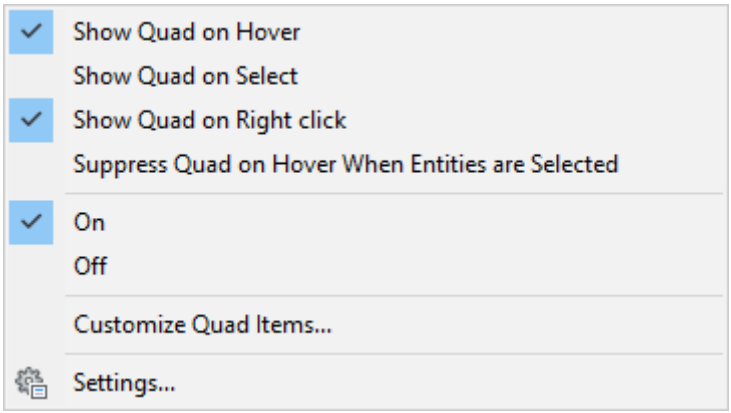
Dynamic Input (DYN): Toggles the *Switch all off temporarily* option of the DYNMODE sytem variable.

- Click to toggle the display of Dynamic Dimensions *On* (DYN) or *Off* (DYN).
- Right click, then choose *Settings* to edit the *Dynamic Dimensions* settings.



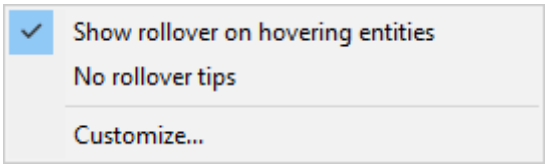
Quad: Toggles the *Switch off* option of the QUADDISPLAY system variable.

- Click to toggle the display of the Quad *On* (QUAD) or *Off* (QUAD).
- Right click, then choose an option in the context menu.
- Right click, then choose *Settings* to edit the *Quad* settings.



Rollover tips (RT): Toggles the ROLLOVERTIPS system variable.

- Click to toggle the display of rollover tips.
- Right click to select the display options for rollover tips: *On hovering entities* or *No rollover tips*.

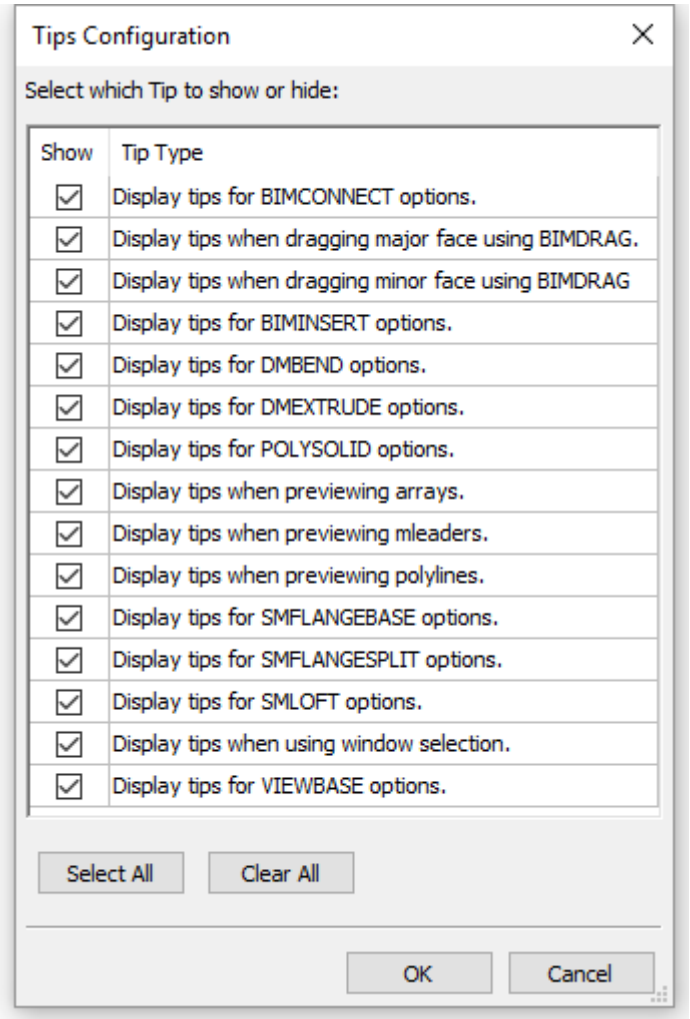


Tips: Toggles the TIPS system variable. If TIPS = ON, an assistant widget reveals options available using the Ctrl-key.

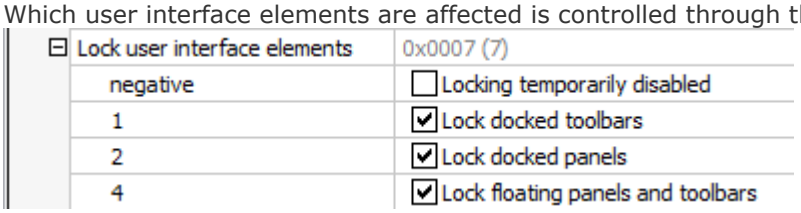
Tips when creating polysolids.

- Click to toggle the display of the TIPS widget On/off.
- Right click to display a context menu:

- Click *On* or *Off* to set the display of the the TIPS widget.
- Click *Configure* to display the Tips Configuration dialog.
  Select which tips show when TIPS = ON.



Lock User Interface: Allows to lock the location and size of toolbars and dockable panels. Press Ctrl to override.
Click the *LOCKUI* field to toggle the lock. Press Ctrl to override.
Which user interface elements are affected is controlled through the LOCKUI system variable.



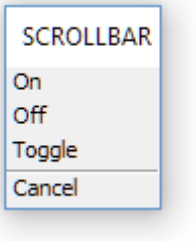
GIS Coordinate Sytem: Displays the name of the current GIS coordinate system.

Display/Hide Scroll Bars

To toggle scroll bars on / off:

1. (option) Hold down the Shift key, then press the *F4* function key.
2. (option) Choose *Scroll Bars* in the *View* menu.

3. (option) Type *scrollbar* in the command bar, then do one of the following:
   - type *off* in the command bar or choose *Off* in the context menu.
   - type *on* in the command bar or choose *On* in the context menu.
   - type *T* in the command bar or choose *Toggle* in the context menu.



NOTE Use the -TOOLPANEL command to show, hide or toggle the display of the various tool panels.

Hide / show a selection of user interface elements

The CLEANSCREENON and CLEANSCREENOFF commands allow to hide or show a selection of user interface elements simultaneously.

Which user interface elements are affected by these commands is defined by the CLEANSCREENOPTIONS system variable.

The Ctrl+0 keyboard shortcut toggles the display of user interface elements. On AZERTY-keyboards: do not use the numeric keypad; do not press the Shift-key when hitting the number key.

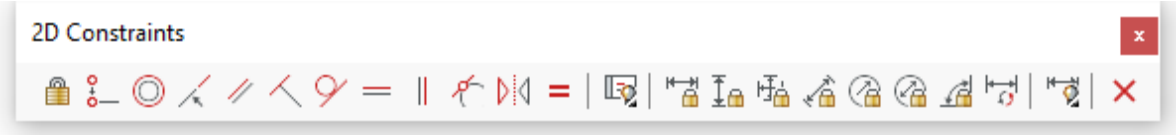
Toolbars

Main Toolbars

The following toolbars are available in BricsCAD.

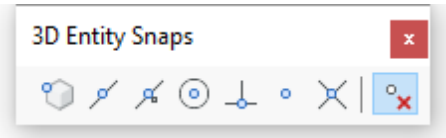
Some of the tools are available in BricsCAD Pro and/or BricsCAD Platinum only. Please see the Command Reference guide for more information.

2D Constraints



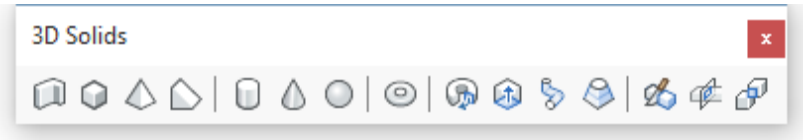
3D Constraints

3D Entity Snaps

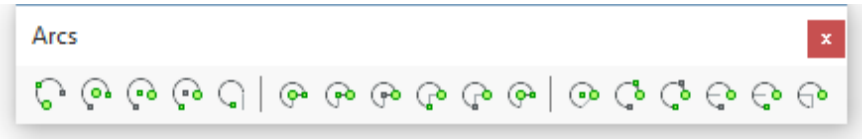


3D Solid Editing

3D Solids



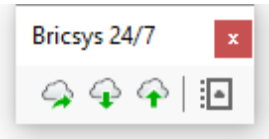
Arcs



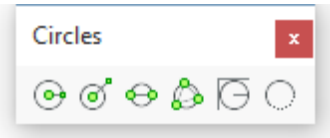
Assembly

BIM

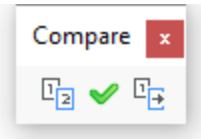
Bricsys 24/7



Circles



Compare



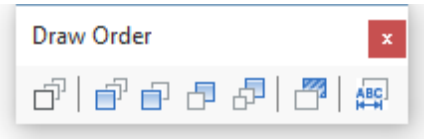
Design Intent

Dimensions

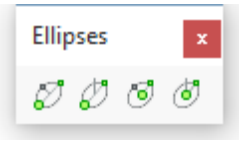
Direct Modeling

Draw

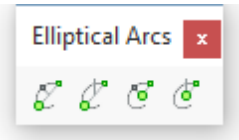
Draw Order



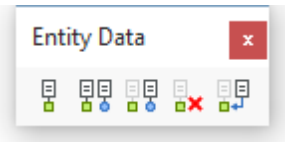
Ellipses



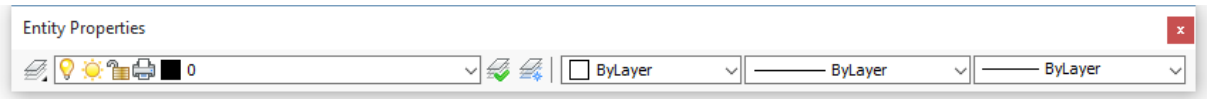
Elliptical Arcs



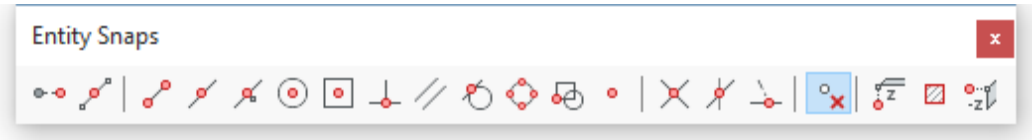
Entity Data



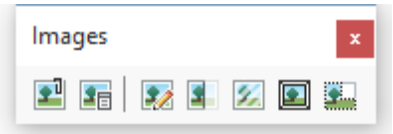
Entity Properties



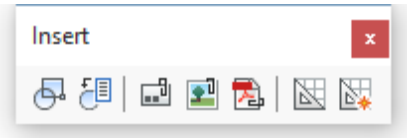
Entity Snaps



Images



Insert



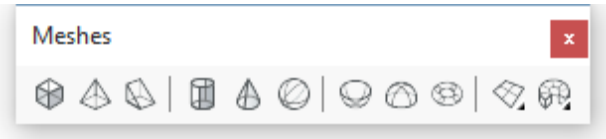
Inquiry

Look From — Replaced by the Look From control.

Layout

Mechanical — Renamed to *Assembly*

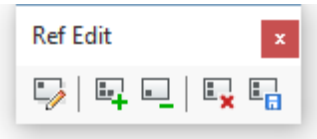
Meshes



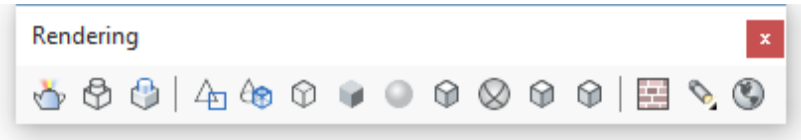
Modify

Parameters

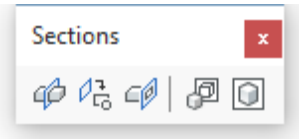
Ref Edit



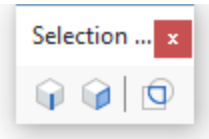
Rendering



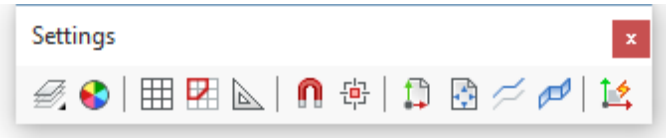
Sections



Selection Modes



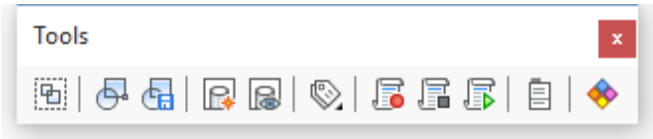
Settings



Sheet Metal

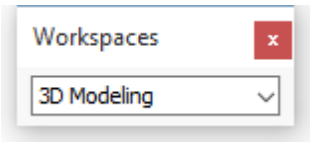
Standard

Tools



View

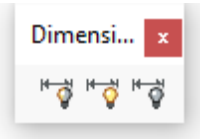
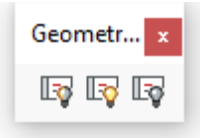
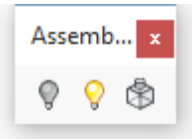
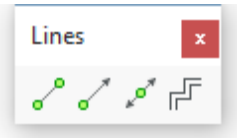
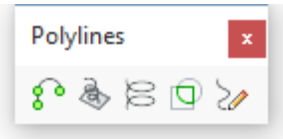
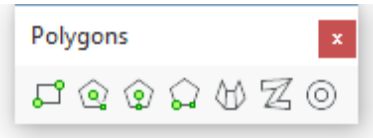
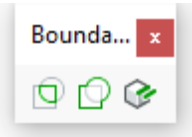
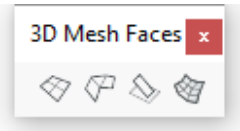
Workspaces

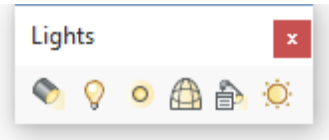
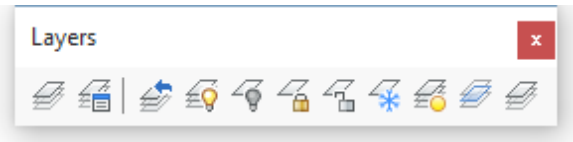
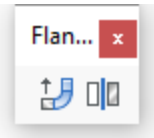
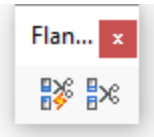
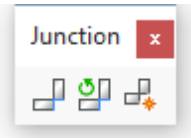
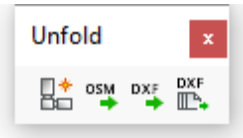
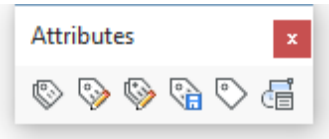
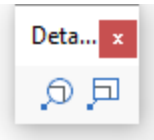
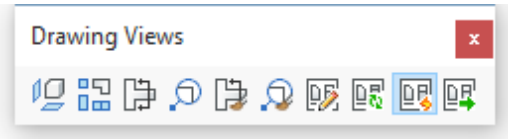


Toolbar Flyouts

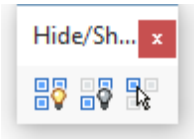
On some tool buttons a small black arrow at the bottom right corner of the icon indicates a flyout is available. A flyout holds a group of related tools. Press and hold the left mouse button to expand the flyout and choose one of the flyout tools. The tool that was last chosen remains visible in the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

Each of the flyouts can be opened as a separate toolbar. Drag the button of a toolbar flyout into the drawing area to open it as a floating toolbar.

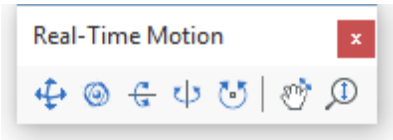
Parent	Flyout	
2D Constraints	Dimensional Constraint	
	Geometrical Constraint	
Assembly	Assembly Visualization	
Direct Modeling	Deform	
Draw	Lines	
	Polylines	
	Polygon	
	Boundaries	
Meshes	3D Faces	
	3D Surfaces	

Modify	Arrays	
Rendering	Lights	
Settings Entity Properties	Layers	
Sheet Metal	Bend	
	Flange	
	Flange Split	
	Junction	
	Relief	
	Unfold	
Tools	Attributes	
View	Detail Section Types	
	Drawing Views	

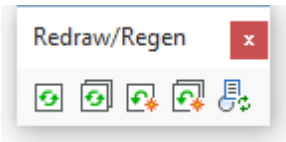
Hide/Show Entities



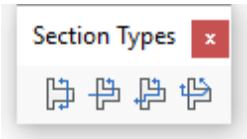
Real-Time Motion



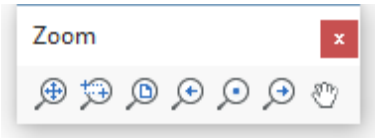
Redraw/Regen



Section Types



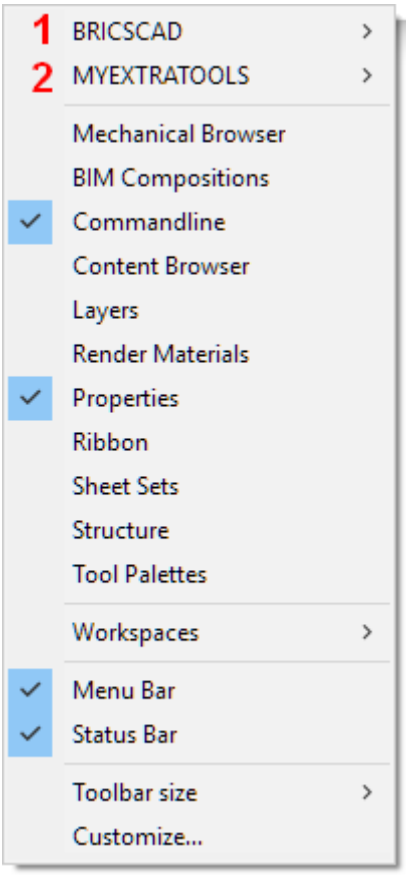
Zoom



Opening a Toolbar

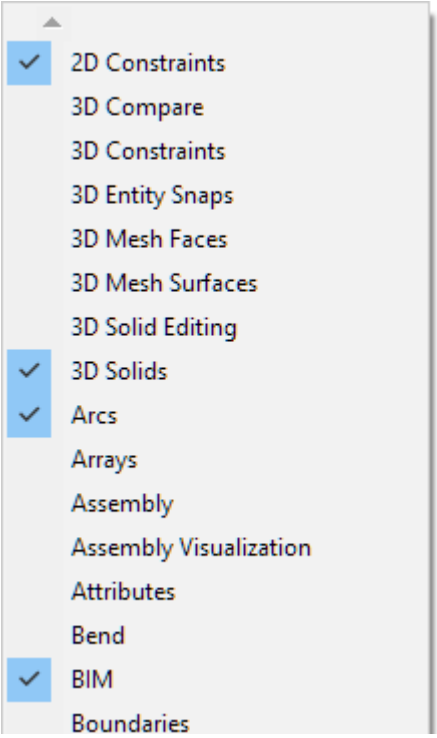
1. In *Windows*: Hover over a toolbar or a blank portion of the toolbar area, then right click.
   In *Linux*: Hover over a blank portion of the menu bar, status bar or toolbar area, then right click.

A context menu displays.



(1) Main CUI file.
(2) Partial CUI file(s)

1. In the context menu choose either the Main CUI file (e.g. BRICSCAD) or one of the Partial CUI files (if any).
   A list of available toolbars in the selected CUI file displays.
   Toolbars that are already open are checked.
   If the list does not fit on the screen, click the arrow keys on the top and the bottom to scroll.



2. Select the toolbar on the context menu.
   The toolbar opens.
3. (option) Dock the toolbar by dragging the toolbar by its title bar to one of the edges of the BricsCAD application window.

NOTES

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

To set the Icon Size

You can choose between small (16 x 16) or large (32 x 32) icons on toolbars.

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

To Set the Position and Visible Property

The *Visible* property determines whether a toolbar is visible when a cui-file is loaded.

The *Position* property determines where a toolbar displays when a cui-file is loaded.

1. In the Customize dialog click the *Toolbars* tab.
2. If necessary, expand the *BricsCAD* menu group.

3. Select the toolbar.
4. Choose *Position* in the *Properties* grid on the *Customize* dialog.
5. Click the settings field and choose the desired position in the list.
   The options are: *Floating*, *Top*, *Left*, *Bottom*, *Right*.
6. Choose *Visible* in the *Properties* grid on the *Customize* dialog.
7. Click the settings field and choose either *Show* or *Hide* in the options list.

To Tear off a Flyout Toolbar

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

Closing a Toolbar

1. In *Windows*: Place the cursor on a toolbar, then right click.
   In *Linux*: Place the cursor on the menu bar, then right click.
   A context menu displays.
2. Choose *BRICSCAD* on the context menu.
   Toolbars that are open are checked.
3. In the toolbar list, uncheck the toolbar you want to close.

NOTE Floating toolbar can be closed by clicking the *Close* button ( ).

Properties Panel

Overview

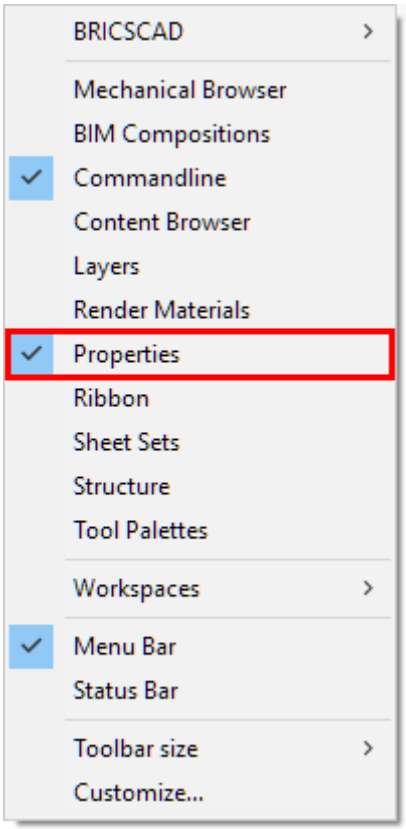
In the *Properties* panel you can:

- Set the current properties: *color*, *layer*, *linetype*, *linetype scale* and *lineweight*
- Edit the properties of a single entity
- Edit the shared properties of a selection set
- Edit endpoints of lines and vertices of polylines graphically

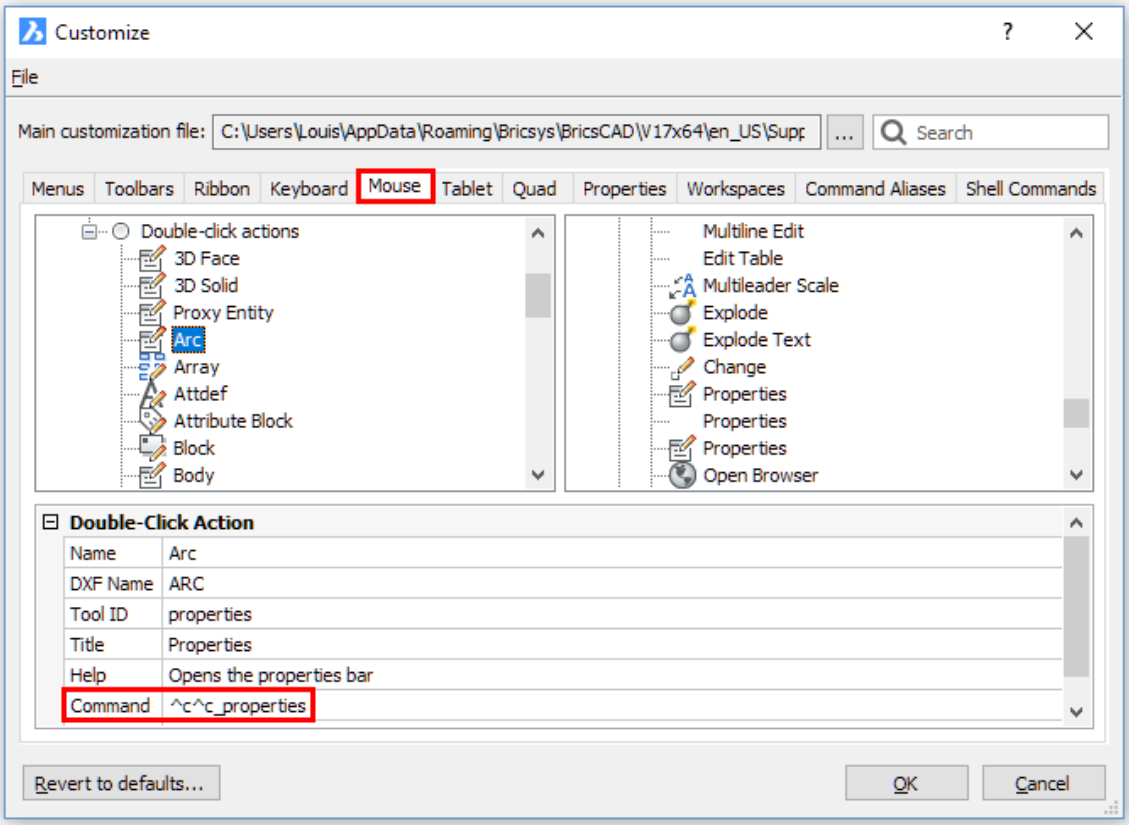
Open the Properties Panel

Do one of the following:

- Click the *Properties* tool button ( ) on the Standard toolbar.
  The pressed state of the tool button indicates that the *Properties* panel is currently open.
  Click the tool button again to close the *Properties* panel.
- Right click when the cursor is on a toolbar.
  A context menu displays.
  The marked items in the context menu are currently open.
  Select *Properties* in the context menu.

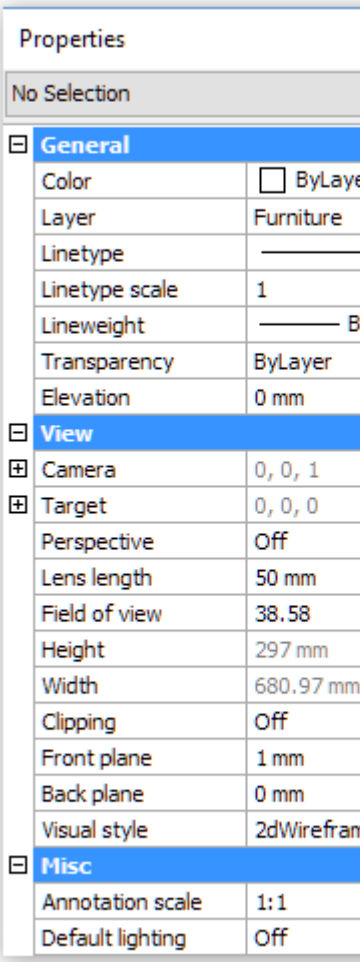


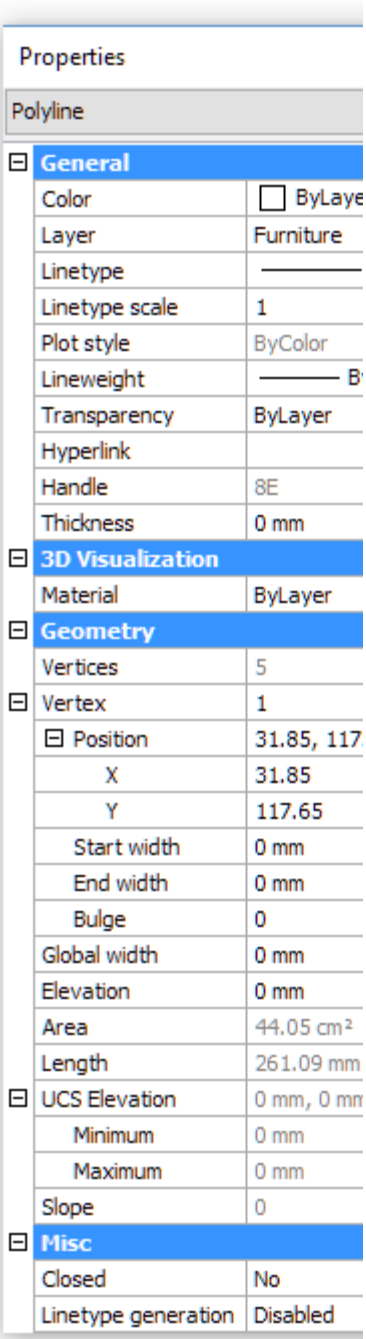
- Double click an entity, such as lines, polylines, circles, arcs, ...
  The *Properties* panel opens, showing the properties of the selected entity.
  Double clicking entities such as texts, blocks or polylines start an editing command (See the *Mouse* tab on the Customize dialog).

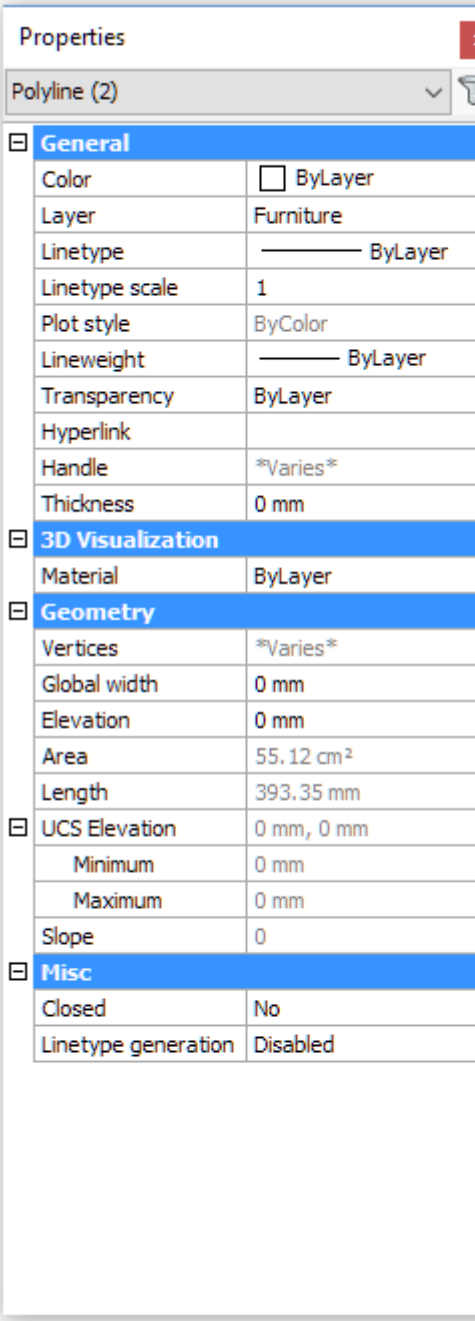


- Click the *Properties* tool button ( ) on the *Standard* toolbar.

- Choose *Properties* in the *Modify* menu.
- Type *properties* in the command window, then press Enter.







Properties - No selection

The current entity properties, current Elevation, View properties and Annotation scale display

Properties - Single entity selected

The properties of the selected entity display

Properties - Multiple entities selected

The shared properties of the selected entities display

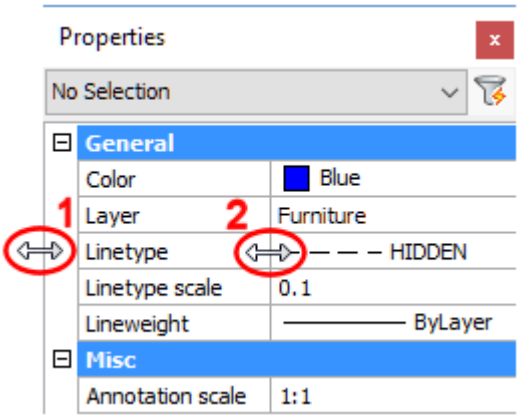
NOTE The *Properties* panel is dockable panel.
To dock the *Properties* panel, drag it by its title bar to either the left or right hand side of the BricsCAD application window or onto another dockable panel.

NOTE Read-only fields display in grey. The content of a read-only field can be copied though. To

copy a read-only field, click the field, then press Ctrl C.

Adjust the Size of the Properties Panel

1. Move the cursor over one of the edges of the *Properties* (1).
   The cursor turns into a double-headed arrow.
2. Press and hold the left mouse button to drag the edge of the *Properties* .



3. Move the cursor over the boundary between the *Setting Names* and the *Setting Fields* columns (2).
   The cursor turns into a double-headed arrow.
4. Press and hold the left mouse button to drag the boundary.

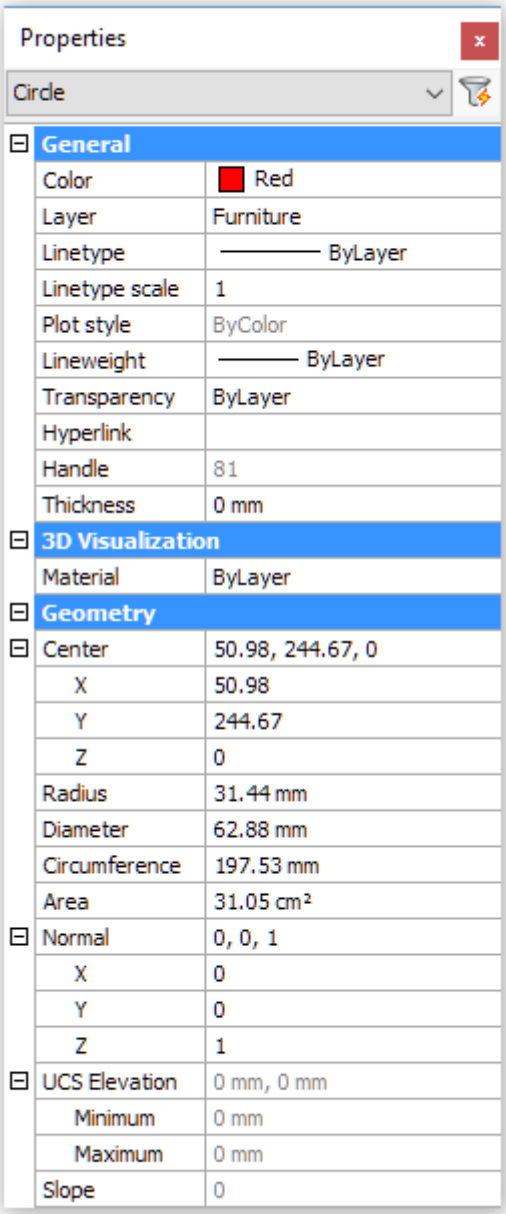
NOTE When docked, only the left (or right) edge of the *Properties* is adjustable.

Setting the current properties in the Properties panel

1. (option) Click on *Color,*
   then click the down arrow button to select a color.
2. (option) Click on *Layer,*
   then click the down arrow to select a layer.
3. (option) Click on *Linetype*.
   then click on the down arrow to select a linetype.
4. (option) Click on *Linetype scale*,
   then type the new value in the *Linetype scale* field.
5. (option) Click on Lineweight,
   then click on the down arrow to select a lineweight.

Edit a Single Entity

1. Select the entity.
   The entity's properties display in the *Properties* panel..



Properties of a circle

2. Click the property you want to modify.
   The settings field of the selected property is activated.
3. Type a new value in the settings field of the selected property or choose a setting from the list box, then press the *Enter* key or select another property.
   The entity is updated.
4. (option) repeat steps 2 and 3 to modify another property.
5. Press the *Escape* key to stop.

NOTES

- Properties of which the value displays in grey cannot be modified.
- Step 3: Properties (such as the General properties), which are chosen from a list are updated instantly.

Edit a Selection Set

1. Select the entities.
   The shared properties display in the *Properties Bar*.
2. Click the property you want to modify.
   The settings field of the selected property is activated.
3. Type a new value in settings field of the selected property or choose a setting from the list box, then press the *Enter* key or select another property.
   All selected entities are updated simultaneously.
4. (option) Repeat steps 2 and 3 to modify another property.
5. Press the *Escape* key to stop.

NOTES

- *Varies* displays for shared properties which are defined differently. If you edit such property, all entities in the selection set will be equally defined for this property.
- Step 3: Properties (such as the General properties), which are chosen from a list are updated instantly.

Edit Endpoints and Nodes

1. Select the line.
2. On the *Properties Bar*, under *Geometry*, do one of the following:
   - Select the X-, Y- or Z- field, then enter a value.
   - Select either *Start point* or *End point*.

General	
Color	ByLayer
Layer	SEWER
Linetype	ByLayer
Linetype scale	2.54
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
Thickness	0
3D Visualization	
Material	ByLayer
Geometry	
Start point	3008.47, 850.43, 0
X	3008.47
Y	850.43
Z	0
End point	3106.4, 752.5, 0
X	3106.4
Y	752.5
Z	0
Delta	97.93, -97.93, 0
X	97.93
Y	-97.93
Z	0
Length	138.5
Angle	315°0'0"

An X indicates the point being edited.

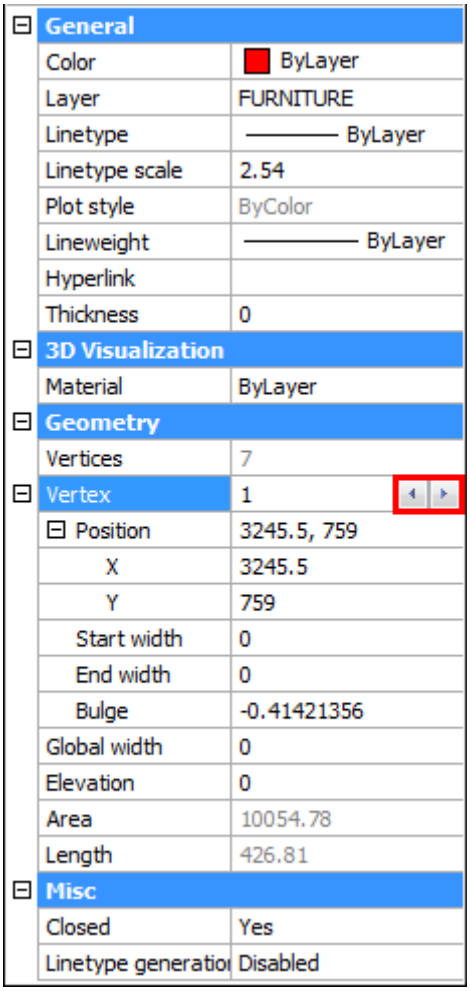
- Click the *Pick Point* button ( ).
- Specify the point in the drawing.

3. Press the *Escape* key to stop.

NOTE This procedure also applies to edit the insertion point of texts, images, blocks, Xrefs, ...

To edit the vertices of a polyline

1. Select the polyline.
2. On the *Properties Bar* under *Geometry*, select *Vertex*.
3. Click the *Next/Previous* arrow buttons to select a vertex.



General	
Color	ByLayer
Layer	FURNITURE
Linetype	ByLayer
Linetype scale	2.54
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
Thickness	0
3D Visualization	
Material	ByLayer
Geometry	
Vertices	7
Vertex	1
Position	3245.5, 759
X	3245.5
Y	759
Start width	0
End width	0
Bulge	-0.41421356
Global width	0
Elevation	0
Area	10054.78
Length	426.81
Misc	
Closed	Yes
Linetype generatio	Disabled

An X indicates the vertex being edited.

4. Do one of the following:
   - Select the X- or Y- field, then enter a value.
   - Under *Vertex* select *Position* and click the *Pick Point* button (), then specify the point in the drawing.

5. Press the *Escape* key to stop.

The Quad Cursor Menu

The Quad cursor menu, aka the Quad is an alternative to grip-editing of entities, offering a (much) richer set of editing operations while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs.

While the Quad was initially conceived for easy and fast editing of 3D solids, the Quad is extended to allow enhanced editing and creating of all entity types.

To Activate the Quad

Activation of the Quad is controlled by the QUADDISPLAY system variable.
To toggle the Quad On/Off, do one of the following:

- Click the QUAD field in the Status bar.
- Press the *F12* function key.
- Edit the *QUADDISPLAY* system variable in the Settings dialog.
- Type *quaddisplay* in the command bar, then choose an option.

Editing the Quad Settings

1. Right click the *QUAD* field in the Status bar, then choose *Settings* in the context menu.

The *Settings* dialog opens, showing the *Quad settings*:

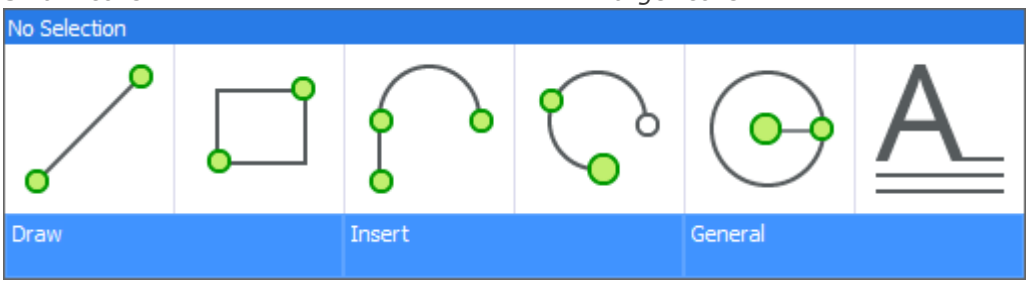
⊟ **Quad**	
⊟ Quad display	0x000D (13)
negative	☐ Switch off quad
1	☑ Display the quad when the cursor hovers on an entity
2	☐ Display the quad when entities are selected
4	☑ Display the quad on right click
8	☑ Suppress the quad on hover when entities are selected
Quad default command launch	[1] Launch default quad command on mouse right-click when quad is displayed
Quad popup corner	[1] Middle right
Quad width	6
Quad most recent items	2
Quad icon size	[32] Large icons
Quad icon space	[1] Normal
Quad show delay	150
Quad hide delay	350
Quad hide margin	50
Quad go transparent	☑ Go transparent
Quad tooltip delay	800
Quad expand delay	110
Quad expand tab delay	50
⊞ Quad tab flags	0x000C (12)
Rollover tips	[1] Show properties on hover
Rollover opacity	100
Rollover selecion set	[2] Properties shared by all selected entities

- *Quad display:* Determines when the Quad displays.
  - Negative: Switches the Quad off.
  - 1: When the cursor is over an entity. The entity under the cursor highlights.
  - 2: When entities are selected
  - 4: On right click when no entities are selected and there is no entity under the cursor.
  - 8: Suppress the quad when hovering over entities that are not in the current selection set; set Show *Quad on right click* ON to display the Quad.
- *Quad default command launch*: Sets the action to launch the default Quad command.
  - 0: Click the button.
  - 1: Right click
- *Quad popup corner*: Controls the popup position of the Quad relative to the cursor position: *Upper*, *Center* or *Middle Right*.
- *Quad width*: Defines the number of commands in a row. Default = 6 commands.

- *Quad icon size*: Small icons (16 x 16 pixels), Large icons (32 x 32 pixels) or Extra Large icons (64 x 64 pixels).

Small Icons

Large Icons

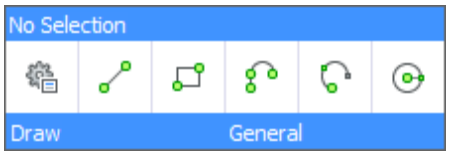


Extra Large Icons

- *Quad icon space*: Defines the size of the Quad tool buttons: narrow, normal or large.

Narrow

Normal



Wide

- *Quad show delay*: Sets the time lap in milliseconds between the highlighting of the entity and the Quad display.
- *Quad hide delay*: Sets the time lap in milliseconds to hide the Quad when the cursor is outside the *Quad hide margin* limit.
- *Quad hide margin*: Sets the distance in pixels the cursor can move away from the Quad. If the cursor moves outside this limit, the Quad is hidden.
- *Quad go transparent*: Defines whether the Quad becomes transparent when the mouse moves outside the Quad, but within the *Quad hide margin*.
- *Quad tooltip delay*: Sets the delay time in milliseconds to display tooltips. If the value is negative, no tooltips display.
- *Quad expand delay*: Sets the time lap in milliseconds to expand the Quad after mouse-enter.
- *Quad expand tab delay*: Sets the time lap in milliseconds to expand a group tab after mouse-enter.
- *Rollover tips*: Specifies whether entity properties display in the Quad while hovering.
- *Rollover opacity*: Specifies the degree of opacity when the Quad is in rollover state. Values between 10 (very transparent) and 100 (fully opaque) are accepted.
- *Rollover selection set*: Controls the content of Rollover tips when multiple entities are selected.

2. To edit a setting, select the setting then do one of the following:

   - Type a new value.
   - Click the check box.
   - Choose an option.

3. Close the *Settings* dialog.

The Quad Layout

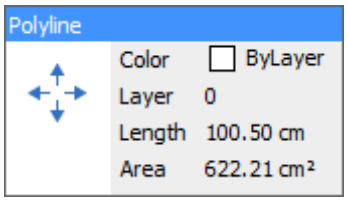
The layout of the Quad cursor menu depends on:

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

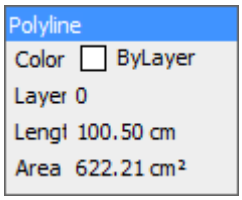
- When the cursor hovers over an entity, the entity highlights and the Quad displays the command most recently used with this entity type.
  When multiple entities are under the cursor, hit the TAB key to highlight all entities one by one.

  If QUADCOMMANDLAUNCH = 1, right click to launch the command.
  If QUADCOMMANDLAUNCH = 0, click the command button.

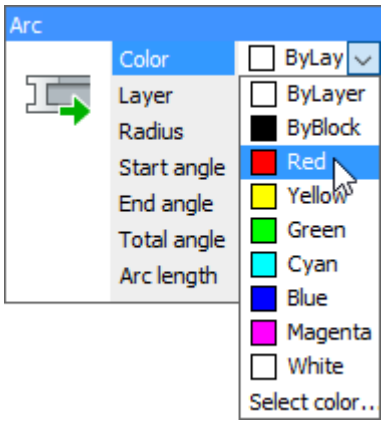
- If Rollover Tips = ON the properties of the entity under the cursor display in the Quad.



If the Quad display is off, rollover tips display if:
- SELECTIONPREVIEW = 1 or 3
- ROLLOVERTIPS = ON

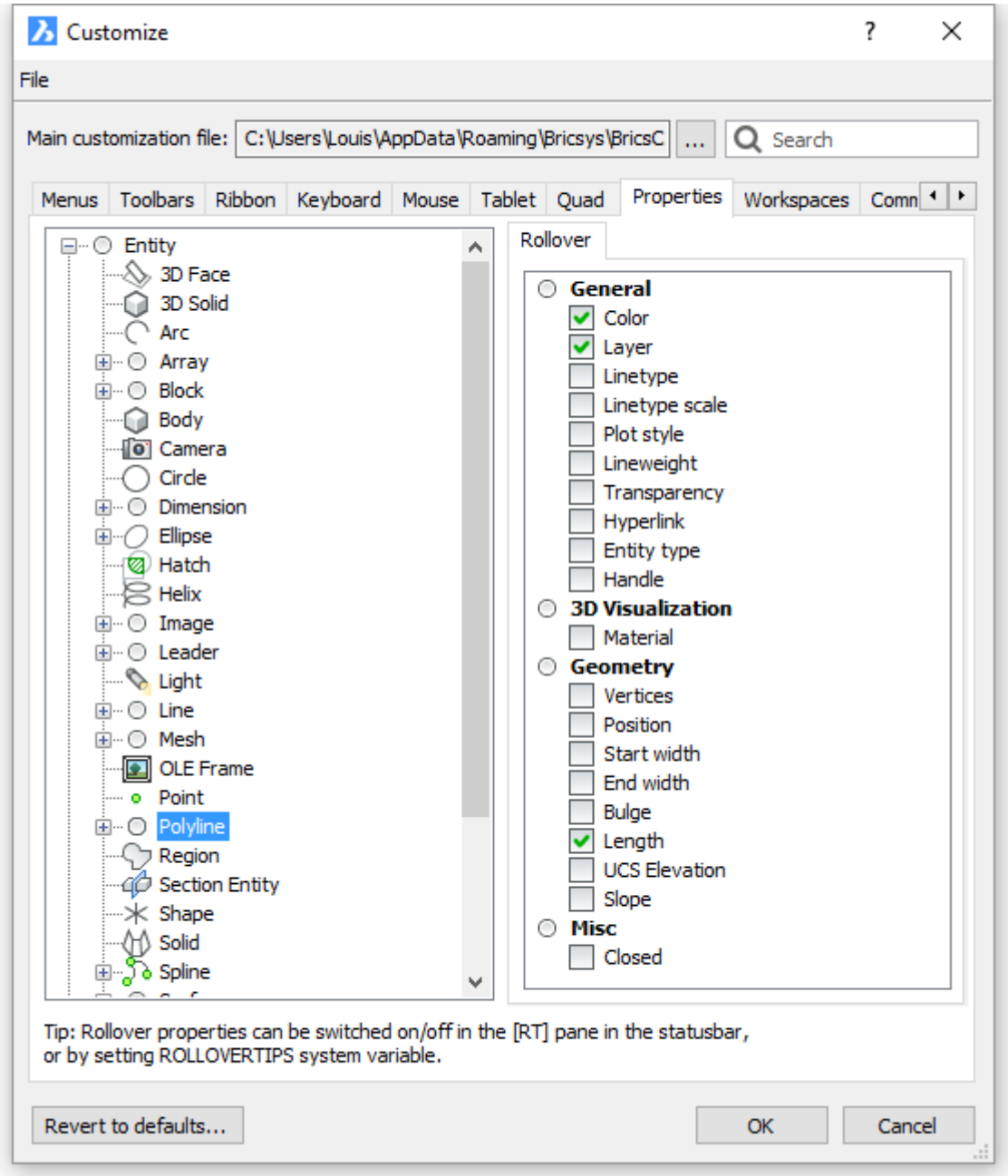


Click a property to edit:

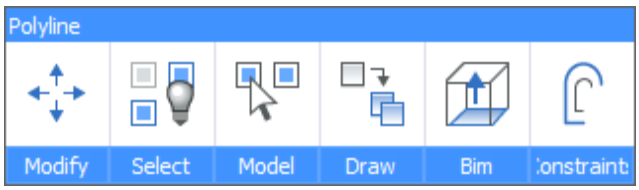


When the Quad is expanded, click the title bar of the Quad to display the Rollover tips.

Which properties display for each entity type is specified in the Properties tab of the Customize dialog box.
Select an entity type in the left pane, then check properties in the *Rollover* pane.



- Move the cursor to the Quad to display the most recently used commands with this entity type.
  A number of command category tabs display at the bottom.

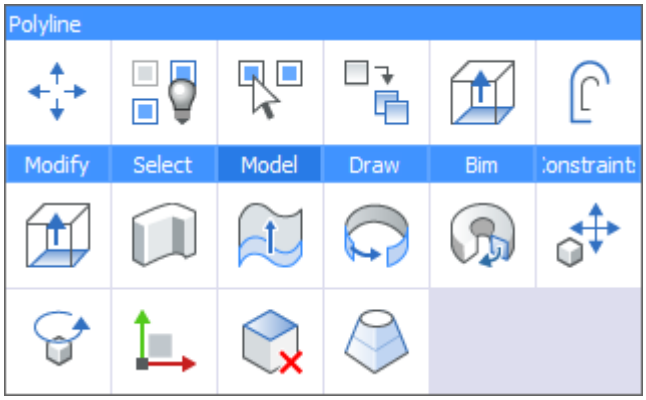


The number of buttons is controlled by the *QuadWidth* user preference.

Which command group tabs display depends on the current Workspace. See To edit the workspace command groups to switch closed command groups on.

Click an icon to launch the command.

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



Click an icon to launch the command.

Using the Quad

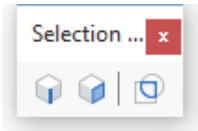
1. Move the cursor to the entity you want to manipulate.
   The entity under the cursor highlights and the Quad displays the most recently used command with this entity type.
2. (option) Select more entities.
   See Selecting multiple entities.
3. Do one of the following:
   - Depending on the value of the QUADCOMMANDLAUNCH system variable, either right click or click the Quad to repeat the most recently used command.
   - Move the cursor to the Quad and select a different command.
     See The Quad Layout.
4. Execute the command.

Selecting Multiple Entities

1. Move the cursor over the first entity, then click when the entity highlights.
   The entity is selected and remains highlighted.
2. Move the cursor to another entity, then click when the entity highlights.
   The entity is added to the selection set and remains highlighted.
3. (option) Repeat step 2 to select more entities.
4. (option) Move the cursor to a selected entity, then press and hold the Shift key and click.
   The entity is removed from the selection set.

NOTES

- When the Quad is active, SELECTIONPREVIEW is neglected, unless a command prompts for entity selection.
- When working with solids:
  - The SELECTIONMODES system variable controls which subentities are detected: edges, faces and/or boundaries.
    Holding down the Ctrl key inverts the current selection mode.
  - Click a button on the *Selection Modes* toolbar to toggle an option. A pressed button indicates the option is selected.



Enable detection of 3D solid edges

Enable detection of 3D solid faces

Enable boundary detection

- Repeatedly press the TAB key to highlight all (sub)entities under the cursor.

Selecting Edges of Faces or Solids

Command: DmSelectEdges

1. Select the faces and or solids.
   See Selecting multiple entities.
2. In the *Direct Modeling* command group of the Quad cursor menu, click the *Select Edges* tool ( ).
   All edges of the selected faces and solids are selected.
3. (option) To remove edges from the selection set: press and hold the Shift key, then click the edge(s).

Using the Quad to Create Entities

If the flag of the QUADDISPLAY system variable is set, entity creation commands are available in the Quad.

To set the *Display the Quad on right click* option: Right click the Quad field in the *Status Bar* and check the option in the context menu.

The available entity creation commands in the Quad:

- The most recently used commands. The number of available commands depends on the *QuadWidth* user preference.
- A number of command group tabs, depending on the current workspace.

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

Customizing the Quad

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

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

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

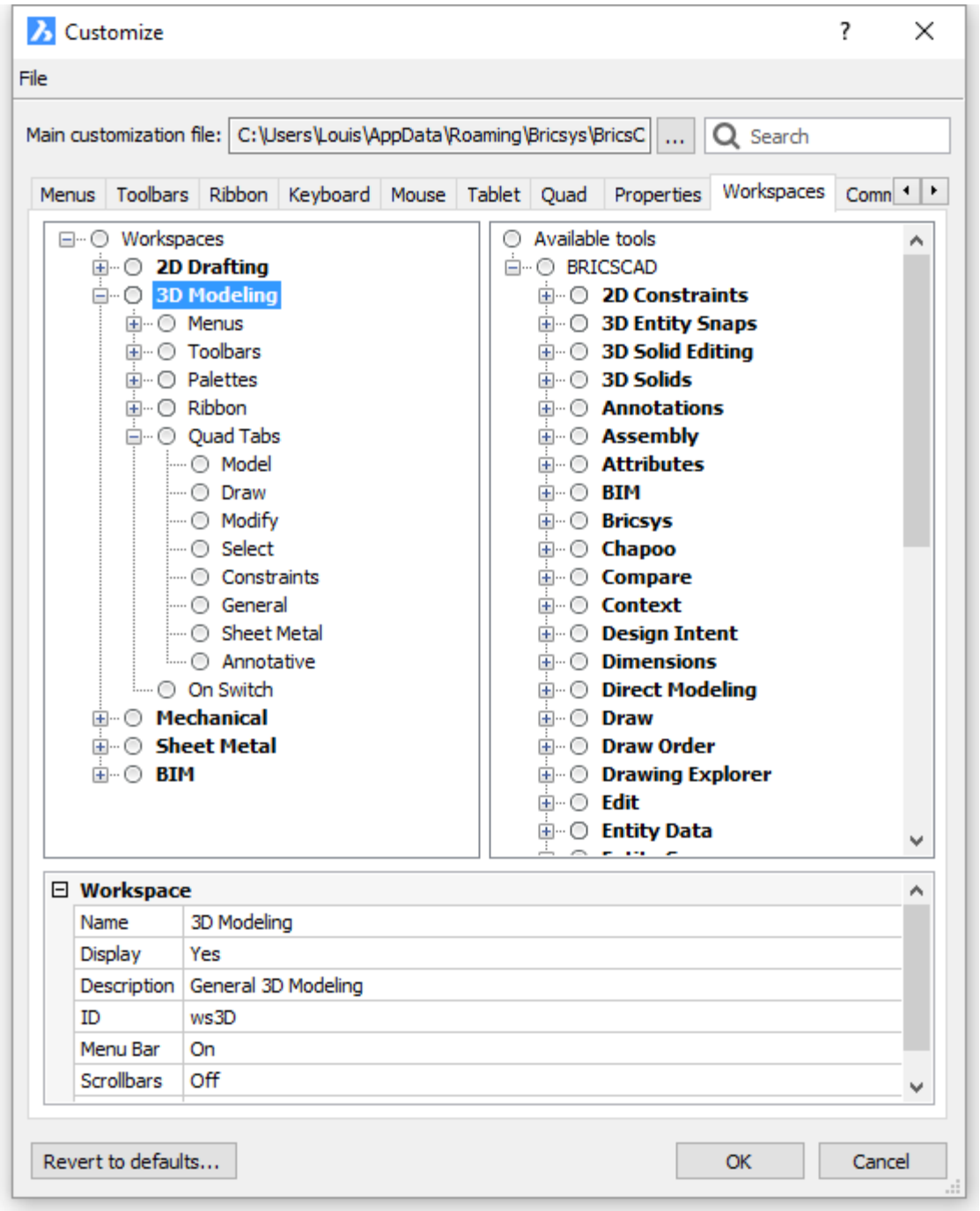
- The *Quad* section:
  In this section, the name and content of all Quad Tabs can be specified, similar to the way Ribbon Tabs are customized. Quad Tabs can be created, copied, renamed and deleted. Manage the content of a Quad Tab by dragging buttons from the right side of the dialog, or by using the right-click context menu to add, rename, move or delete Quad Buttons. The right side of the dialog contains the full list of all buttons which can be shown in the quad. These

include the tools defined in the cui file, plus all internally defined Quad Buttons, which are listed under the new 'Built-in' branch at the bottom. These built-in buttons are similar to the tools, only they can not be edited: their name, icon, command string etc.. are read-only. The condition which specifies whether a certain Quad Button will appear in the quad when the cursor hovers on an entity, is specified for each Quad Button individually in the 'Object Filter' property, similar to the way Object Context Menu Items are configured. It is not possible to modify the Object Filter of a built-in Quad Button.

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

To Edit the Workspace Command Groups

1. Move the cursor over a toolbar, then right click and choose *Customize* in the context menu.
2. Click the *Workspace* tab on the *Customize* dialog box.
3. Expand the workspace you want to edit.
4. Expand the *Quad Tabs* list.
   Tabs that are currently switched off display in grey.



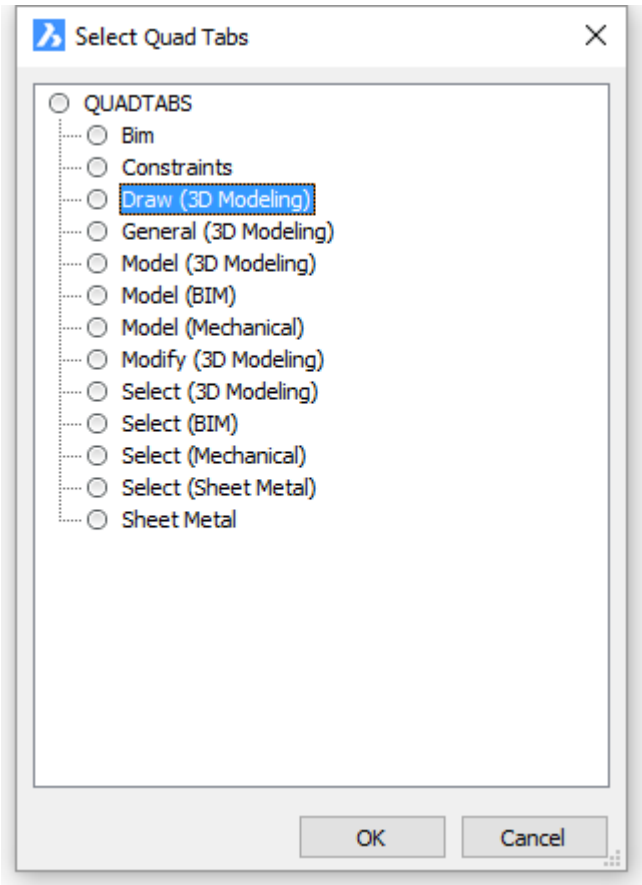
5. To set the availability of a tab, select the tab then click the *Display* field in the Properties grid and choose *Yes* or *No*.

6. To add a tab do one of the following:

7. Right click the Quad Tabs node and choose Appand Quad tab in the context menu to add a tab at the bottom of the tab list.

8. Select a tab, then right click and choose Insert in the context menu to add a tab above the selected tab.

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



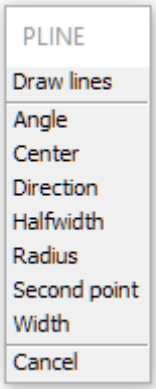
7. To remove a tab, select the tab, then right and choose *Delete* in the context menu.
8. To change the tab order: drag and drop the tabs in the list.

Prompt Menus

TUTORIALS

The Prompt Menu and Its Many Uses

BricsCAD commands often provide several options. These options appear in the status bar or command bar and also in a prompt menu, which initially displays in the top right corner of the screen. Choose the appropriate option by clicking in the prompt menu. E.g. click *Radius* in the *PLINE* prompt menu to define the radius of the current arc segment of a polyline.



To control the display of prompt menus

Whether a prompt menu displays and its location is controlled through the *PROMPTMENU* system variable and the *PromptMenuFlags* user preference.

Type *promptmenu* in the command bar, then type a value of 0 to 5 (see table below) and press Enter.

Value	Description
0	No prompt menus
1	Display prompt menu at user defined location (*)
2	Display prompt menu at top left corner
3	Display prompt menu at top right corner
4	Display prompt menu at bottom left corner
5	Display prompt menu at bottom right corner

(*) You can move the prompt menu by dragging it; future prompt menus will appear where you last placed it.

NOTE The PROMPTMENU system variable is saved in the registry and therefore applies to all drawings.

The PromptMenuFlags user preference has two options:

- 1 = Show hidden options. Hidden options display in italics.
- 2 = Hide the prompt menu when a command prompts to select entities.

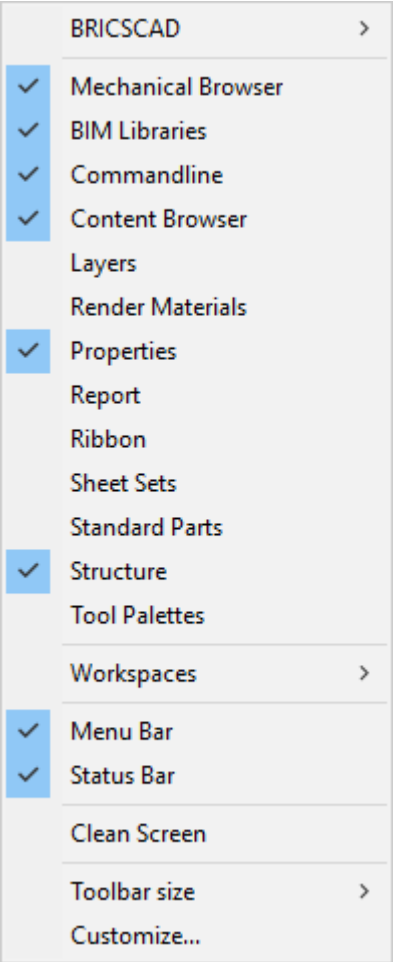
Dockable Panels

The following panels can be docked: Command Bar, BIM Compositions, Content Browser, Layers Panel, Mechanical Browser, Properties Bar, Render Materials, Ribbon, Sheet Sets, Structure, Tool Palettes

NOTE Use the -TOOLPANEL command to show, hide or toggle the display of the various tool panels.

To open or close a panel

1. Move the cursor over a toolbar or ribbon panel, then right click.
   A context menu displays.



   Panels that are currently open are checked.
2. Click a panel in the menu to toggle its display.

To dock a panel

Although panels can be docked at either side of the graphical screen, they are best placed at the left or right.

Do one of the following:

- Drag the panel by its title bar. A rectangle indicates the boundary of the panel. Release the left mouse button when the boundary rectangle is over an edge of the graphical screen.
- Double click the title bar to dock the panel at its previous location.

NOTE The LOCKUI system variable controls whether the current position docked and/or floating panels is locked or not.

Press and hold the Ctrl-key to override. Click the *LOCKUI* field in the Status Bar to toggle the locking of the user interface.

To dock a panel with respect to another docked panel

When another panel is already docked, you can dock another panel at the top, bottom, left or right of the docked panel or add the panel as a tabbed panel.

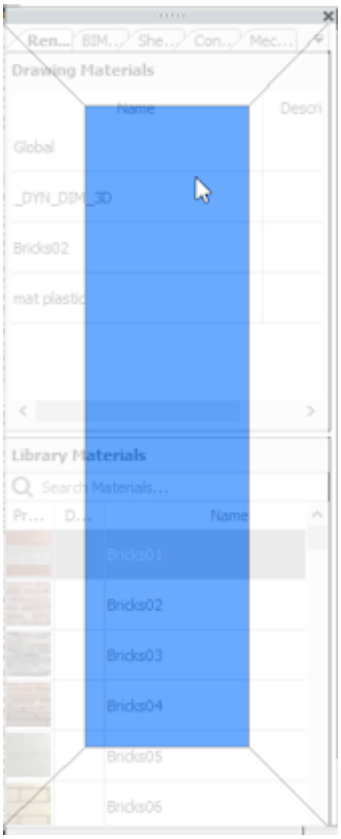
1. Drag the panel by its title bar over a docked panel.
   The docking location selector displays. The blue zone indicates the selected docking position with respect to the docked panel.

2. Move the cursor over the top, right, bottom, or left side of the docking location selector to choose the docking position with respect to the already docked panel and click when the appropriate position highlights.

To stack panels in a tabbed panel

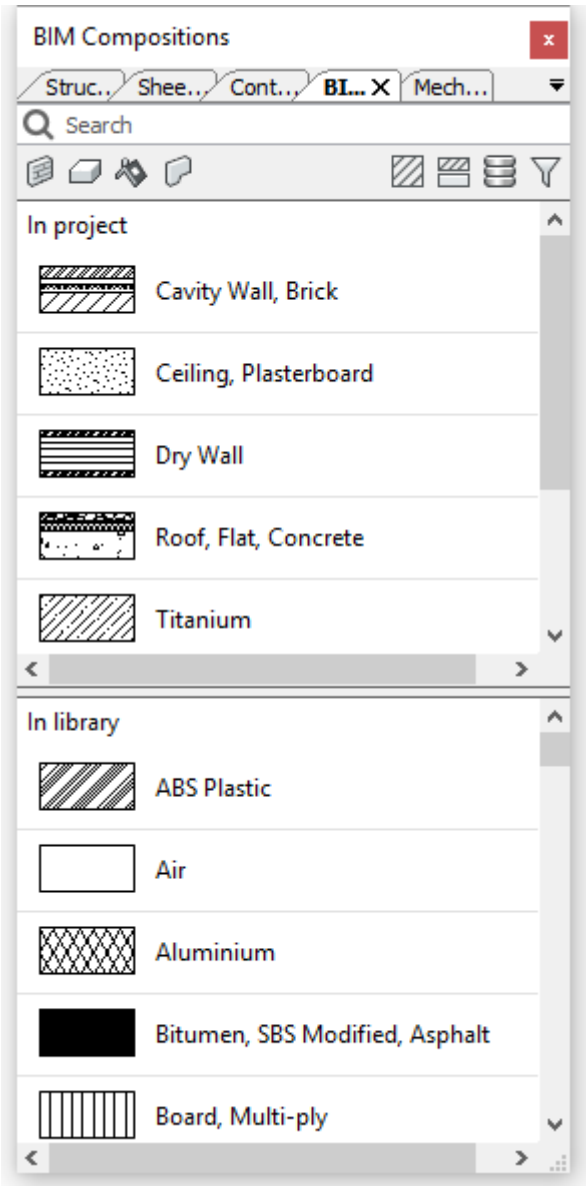
Panels can be grouped into a tabbed panel. A tabbed panel can be either docked or floating.

1. Drag a panel by its title bar.
   The docking location selector displays.



2. Move the cursor over the center of the docking location selector to add the panel in a new tab.

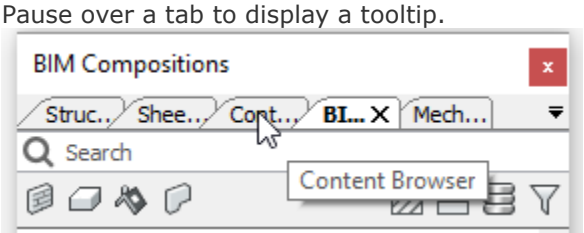
Using tabbed panels



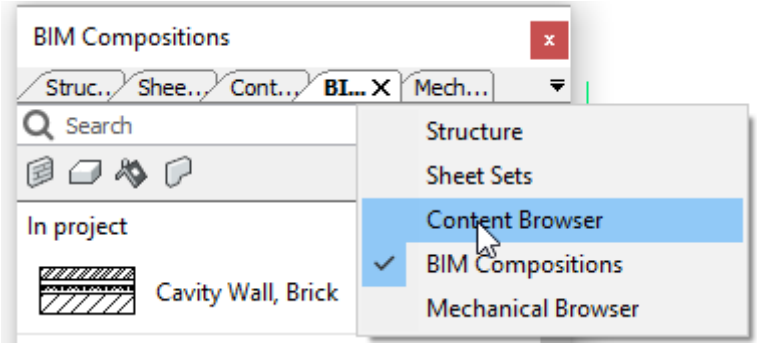
To select a tabbed panel

Do one of the following:

- Click the panel tab.
  Pause over a tab to display a tooltip.



- Click the down arrow button, then select the panel in the list.



To modify the order of the tabs

Drag a tab to its new position.

To tear off a tabbed panel

1. Select the tab.
2. Drag the tab away from the tabbed panels and drop it on the graphic screen.

To close a tabbed panel

1. Select the tab.
2. Click the *Close* button of the tab (X).

Customizing BricsCAD

Startup Options

When starting up BricsCAD it is possible to add arguments or switches. E.g. the command line "BricsCAD.exe drawing1.dwg drawing2.dwg drawing3.dwg" will start the BricsCAD application and load the specified drawings while the command line "bricscad.exe /P MyProfile" will load BricsCAD using user profile MyProfile.

"/P MyProfile" is called a command line switch. Both a slash or a hyphen are accepted to precede the switch, e.g. "-P MyProfile" will also work. The space is optional, e.g. "/PMyProfile" would also work, and the switch character is case insensitive, e.g. "-p MyProfile" is also fine.

Switch	Description
/P [userProfileName]	Starting without the /P option, BricsCAD re-uses the last used user profile (called current user profile), which is not necessarily the Default profile. If the user profile specified with the /P option doesn't exist at startup, a new empty user profile will be created with that name. If the /P argument is an .arg file the arg file is used to initialize the profile settings on the first run (See Starting BricsCAD with a specific profile).
/B [scriptFileName.scr]	Runs a script at startup.
/L	Suppresses the startup splash screen.
/LD [application.arx/brx/drx]	Loads an application.
/S [path]	Redefines the SRCHPATH system variable. e.g. BricsCAD.exe /S C:\BricsCAD\
/T [TemplateFileName]	Creates a new drawing based on the template drawing specified.
regserver unregserver	To register/unregister BricsCAD COM. Please note these arguments are to be used without a preceding "/".

NOTES

- Switch arguments containing spaces should be enclosed by double quotes, e.g. /P "My user profile"
- Switches can be combined, e.g. */P MyProfile /L /S C:\BricsCAD* will start BricsCAD using the *MyProfile* user profile without the splash screen and sets the SRCHPATH system variable to *C:\BricsCAD*.

Besides these switches, BricsCAD supports file names as command line arguments.
Supported file types are:
DWG - to load a drawing
DXF - to load a drawing
SCR - to load a script
LSP - to load a LISP program
DLL - to load an ADS/SDS or DRX/ARX program
SLD - to load a slide file
MNU - to load a menu file
CUI - to load a menu file

If GETSTARTED = ON when you launch BricsCAD, the Get Started dialog displays, which let you choose the workspace and create a new drawing or open an existing drawing.

What are BricsCAD User Files

In BricsCAD the preferences of each user are stored in specific folders, which are called the *Local root* folder and the *Roamable root* folder.

The *Local root* folder (Windows only) contains the *Templates* folder, where the drawing templates are saved.

The *Roamable root* folder contains several subfolders:

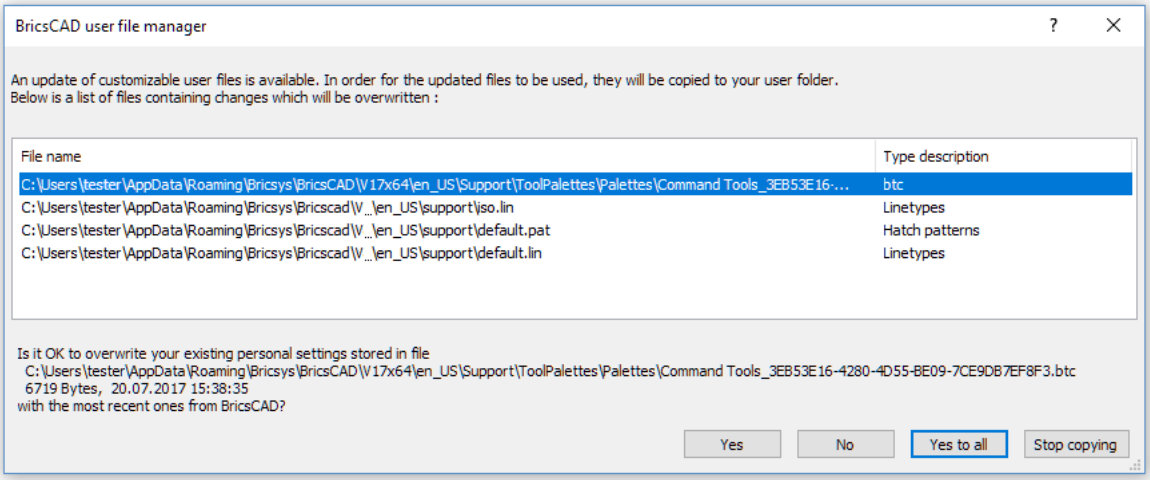
- *PlotConfig*: where the plotter configuration files (*.pc3) are saved
- *PlotStyles*: where the plot style table files (*.stb) and plot color table files (*.ctb) are saved
- *Support*: where the hatch pattern files (*.pat), line style files (*.lin), alias files (*.pgp), unit files (*.unt), custom user interface files (*.cui), etc. are saved
- *Templates*: (Mac and Linux only) where the drawing templates are saved.

Platform	Local / Roamable root folder
Windows	C:\Users\[UserName]\AppData**Local**\Bricsys\BricsCAD\V[Version]\en_US C:\Users\[UserName]\AppData**Roaming**\Bricsys\BricsCAD\V[Version]\en_US
Mac	/Users/[UserName]/Library/Preferences/Bricsys/BricsCAD/V[Version]/en_US
Linux	/home/[UserName]/Bricsys/BricsCAD/V[Version]/en_US

Using the BricsCAD User File Manager

When you install an update of the software, the first time you launch BricsCAD after the update, the content of the *UserDataCache* folder (C:\Program Files\Bricsys\BricsCAD\UserDataCache) is compared with the content of the *Local root* folder and the *Roamable root* folder of the current user. If you have customized the files in these folders and if one or more of the corresponding files in the *UserDataCache* folder are more recent, the *BricsCAD user file manager* is launched asking you to keep your files or to overwrite them with the new version of these files.

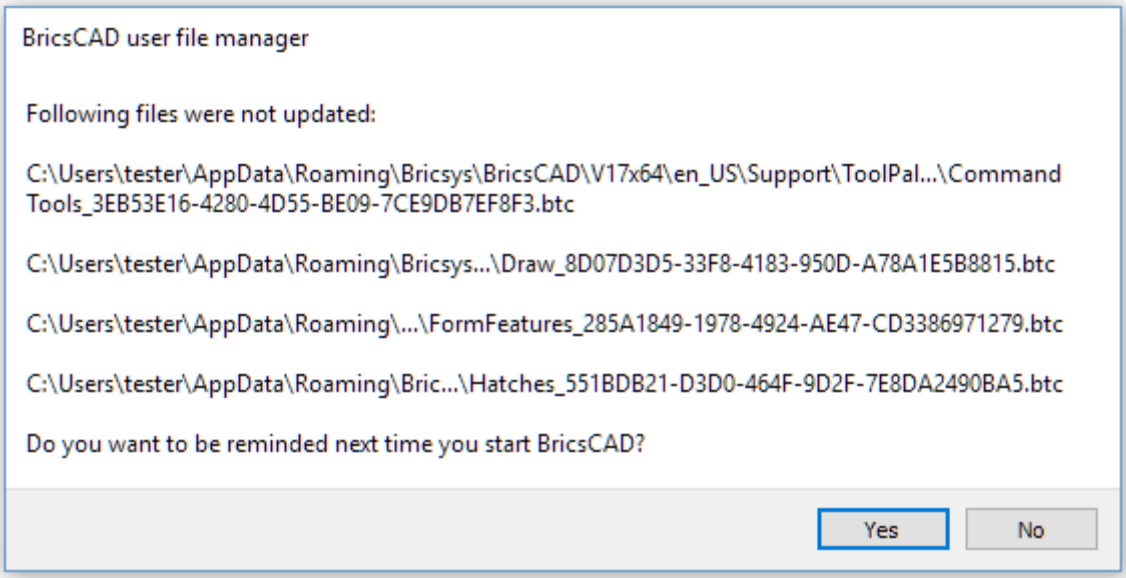
1. Do one of the following:
   - Select a file in the file list, then click the *Yes* button to overwrite or click the *No* button to keep your file.
     Repeat this procedure for each file in the list.
   - Click the *Yes to all* button to replace all your files with the more recent files from BricsCAD.
   - Click the *Stop Copying* button to keep all your files.



2. If you have decided not to update one or more files, an alert window displays.
   Do one of the following:

- Click the *Yes* button if you want to update one of these files later.
  In this case this procedure is restarted the next time you start BricsCAD.

- Click the *No* button if you don't want to be reminded.



What are the CUI Files

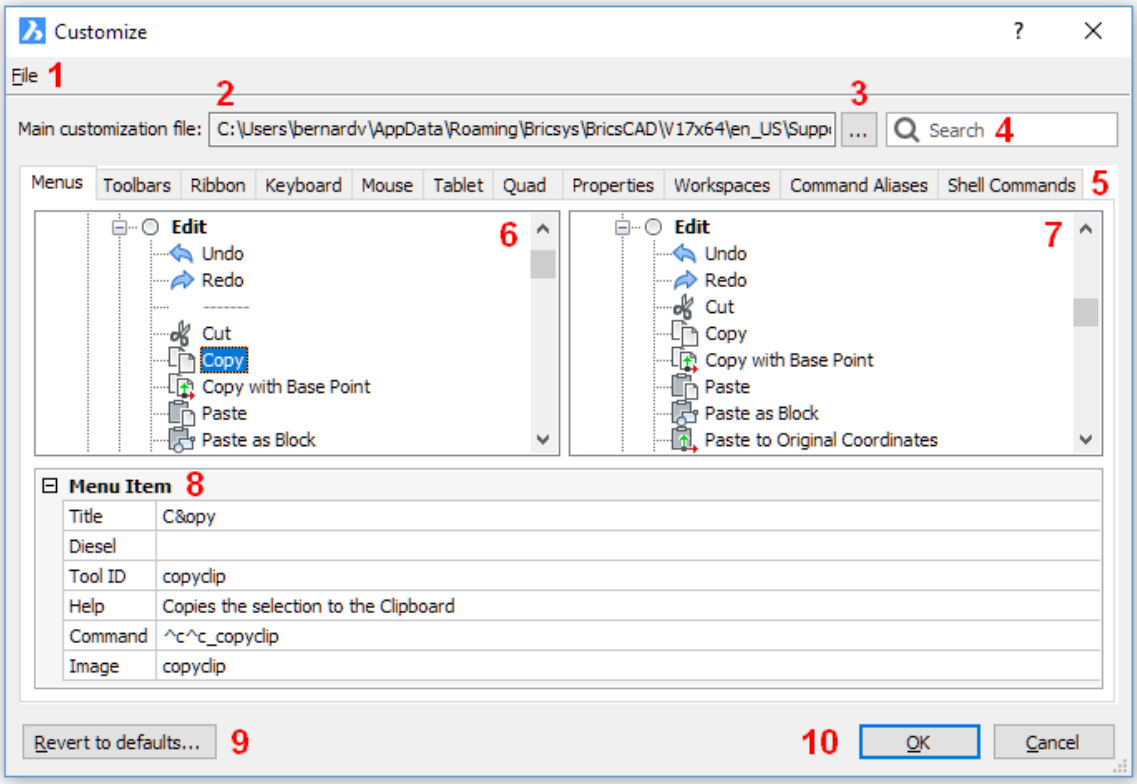
Files with the *CUI* extension are **C**ustomize **U**ser **I**nterface files for BricsCAD. These XML format files replace the previously used menu definition files (*.mnu *.mns). CUI files can be edited with the build-in interactive editor, launched by the CUSTOMIZE command. You can migrate CUI files between BricsCAD versions and combine them between different computers.

NOTE The Load partial CUI file ... procedure accepts IntelliCAD menu files (*.mnu *.mns) and converts them to CUI files.

Opening the Customize Dialog

To open the *Customize* dialog do one of the following:

- Choose *Customize* in the *Tools* menu
- In *Windows*: Hover over a toolbar or a blank portion of the toolbar area, then right click and choose *Customize* in the context menu.
- In *Linux*: Hover over a blank portion of the menu bar, status bar or toolbar area, then right click and choose *Customize* in the context menu.
- Type *customize* in the command window, then press Enter.
- Type *cui* in the command window, then press Enter.



The components of the *Customize* dialog are:

1. *File* menu
2. Main customization file name and location
3. Browse for main customization file button
4. Search field: type a search string then click the looking glass icon, click again for the next hit.
5. Section tabs
6. Content of the currently selected section
7. *Available tools* tree
8. Properties grid, displaying the properties of the selected item
9. Revert to defaults... Reloads default menus, toolbars, etc. overwriting the customized *default.cui* file.
10. OK Saves changes and closes the *Customize* dialog.
    Cancel Closes the *Customize* dialog, without saving changes.

Working with CUI files

Commands: MENULOAD, CUILOAD, MENUUNLOAD and CUIUNLOAD

When you install BricsCAD the default CUI files are copied to the *Support* folder of your Roamable root folder. (e.g. C:\Users\<user_name>\AppData\Roaming\Bricsys\Bricscad\Vxx.x\en_US\Support).

When you launch BricsCAD the first time the *default.cui* file is loaded automatically.

There are two procedures to load a CUI file:

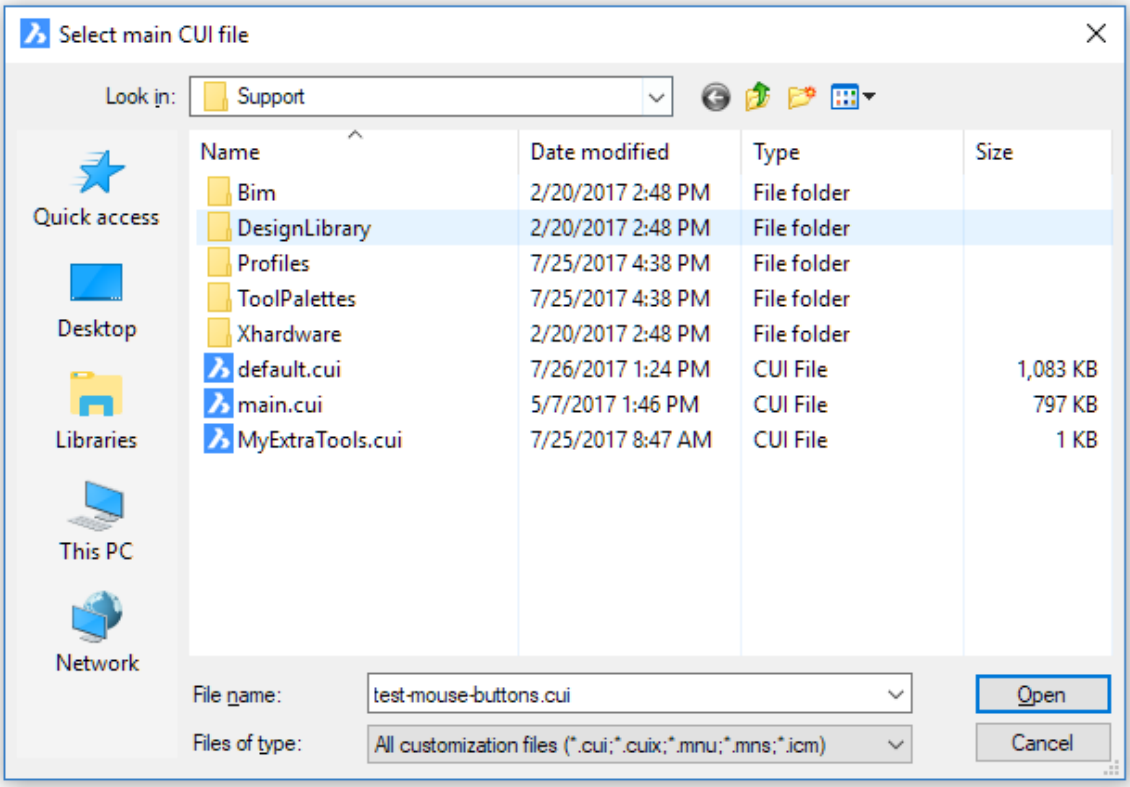
- Load main CUI file, which replaces all existing content with the content of the new CUI file.
- Load partial CUI file, which appends its content to the content of the currently loaded main CUI file.

NOTE You can load an IntelliCAD menu file if you select *MNU files (*.mnu;*.mns)* in the *Files of Type* list box of the *Choose a Customization file* dialog in the Load main CUI file, Load partial CUI file and Manage Customization Groups procedures. The menu file will be converted to a CUI file automatically.

To Load a Main CUI File

1. Do one of the following:
   - Click the *Load Menu* tool button ( ) on the *Tools* toolbar.
   - Choose *Load Menu* in the *Tools* menu.
   - Type *menu* in the command bar, then press Enter.
   - Open the Customize dialog, then click the *Browse* button.
   - Open the Customize dialog, then choose *Load main CUI file ...* in the *File* menu.

The *Select main CUI file* dialog displays.



2. Do one of the following:

- Double click a *CUI* file.
- Select a *CUI* file, then click the *Open* button.

The selected *CUI* file is loaded.

3. Close the *Customize* dialog.

To Load a Partial CUI File

1. Open the Customize dialog, then choose *Load partial CUI file ...* in the *File* menu.
   The *Choose a Customization file* dialog displays.
2. Do one of the following:

- Double click a CUI file.
- Select a CUI file, then click the *Open* button.

The partial CUI file is loaded.

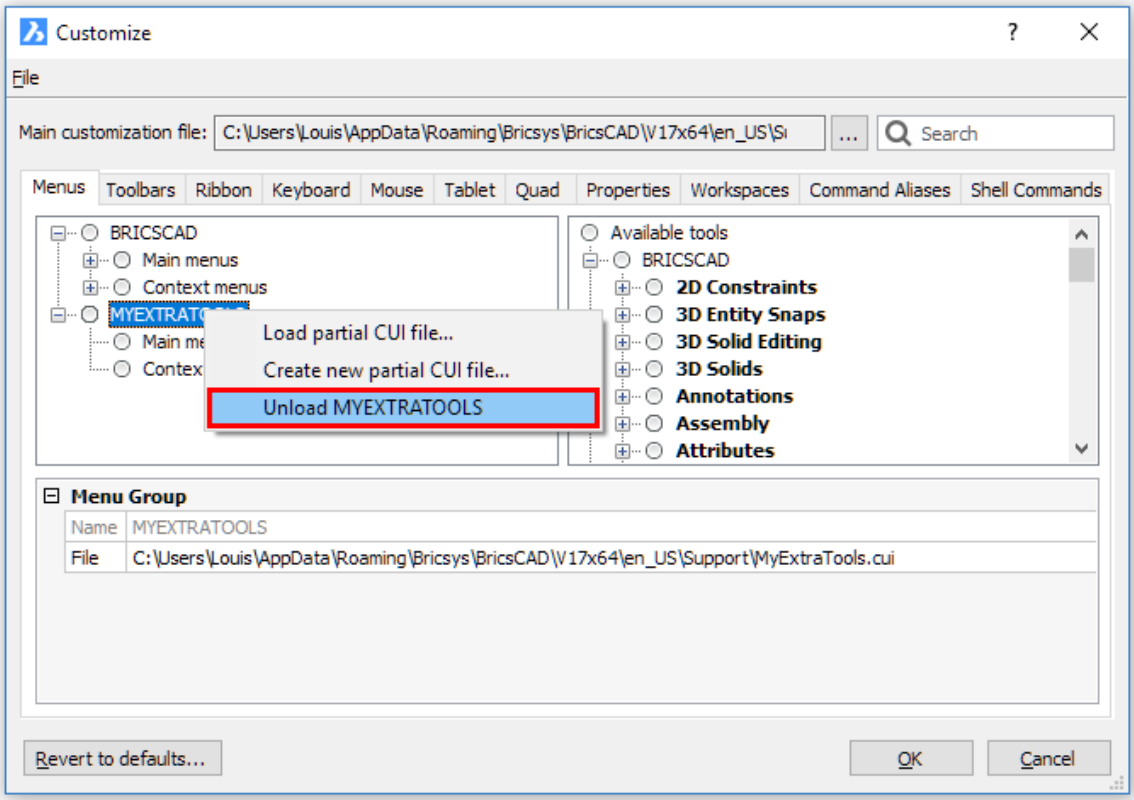
3. (option) Repeat steps 1 and 2 to load another partial CUI file.
4. Do one of the following:

- Click the *OK* button (OK) to confirm the loading of the partial CUI file(s) and close the *Customize* dialog.
- Click the *Cancel* button to (Cancel) close the *Customize* dialog and unload the partial CUI file(s).

NOTE If a CUIX file is selected in step 2 it will be converted to a CUI file on-the-fly. When a CUIX file contains bitmaps, they are copied into a .RESZ resource file, which are supported as an alternative to resource DLL (zip file containing tool bitmaps; works on all platforms). The following image formats are supported: BMP, JPG, PNG, ICO, GIF, TIFF, XBM, XPM.

To Unload a Partial CUI File

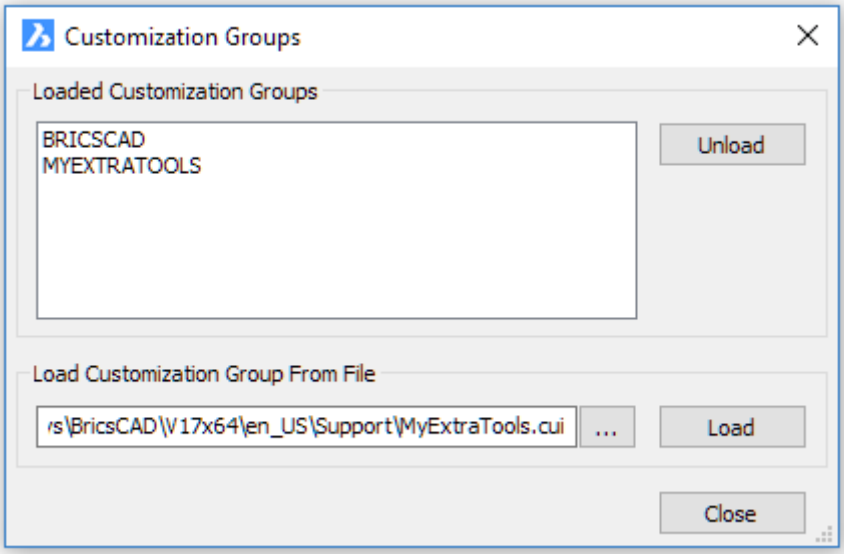
1. Open the Customize dialog.
2. In the *Menus*, *Toolbars* or *Keyboard* section select the partial CUI file you want to unload.



3. Right click and select *Unload <selected partial CUI>* in the context menu.

Managing customization groups

1. Type *Menuload* in the command bar, then press Enter.
2. The *Customization groups* dialog opens.



3. (option) To load a partial CUI file, click the browse button (...) to select a partial CUI file, then click the *Load* button.
4. (option) To unload a partial CUI file, select the partial CUI file in the *Loaded Customization Groups* list, then click the *Unload* button.
5. Click the *Close* button to stop.

Creating Custom Tools

Overview

1. Create a Partial CUI file.
2. (option) Add toolbox(es) in a Menu Group.
3. (option) Add tools in a toolbox
4. (option) Add main menu(s)
5. (option) Add sub menu(s)
6. (option) Add context menu(s)
7. (option) Add toolbar(s)
8. (option) Add flyout(s) to toolbar(s)
9. (option) Add control(s) to toolbar(s)
10. (option) Add keyboard shortcuts
11. (option) Add existing tools to menus, toolbars or shortcuts
12. (option) Create new tools in menus, toolbars or shortcuts
13. (option) Change the tool order in menus and toolbars

Create a Partial CUI File

1. In the Customize dialog do one of the following:
   - In the *File* menu, choose *Create new partial CUI file ...* .
   - Place the cursor on Menu group, then right click an choose *Create new partial CUI file ...* in the context menu.

The *Create a Customization File* dialog opens.

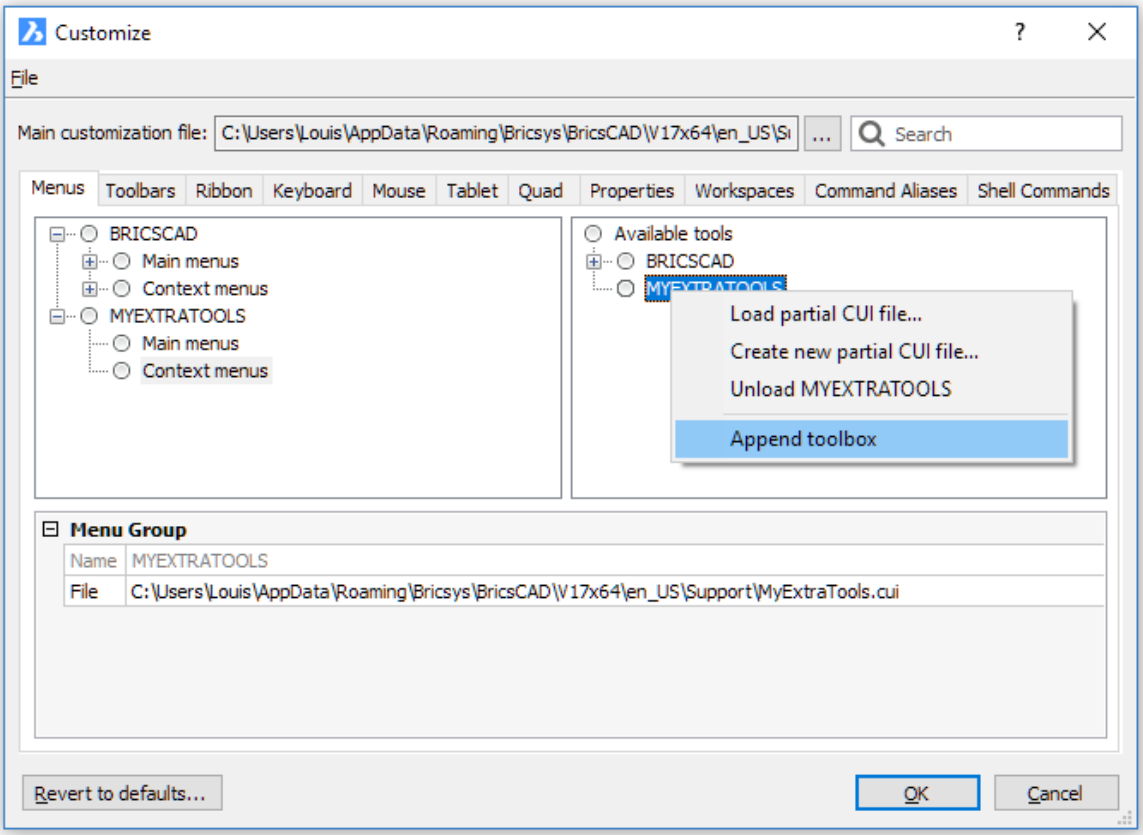
By default the new partial CUI file will be created in the *Support* folder of the Roamable root folder.

2. (option) Select a different folder to save the CUI file.
3. Type a name for the new CUI file in the *File name* field.
4. Click the *Save* button.

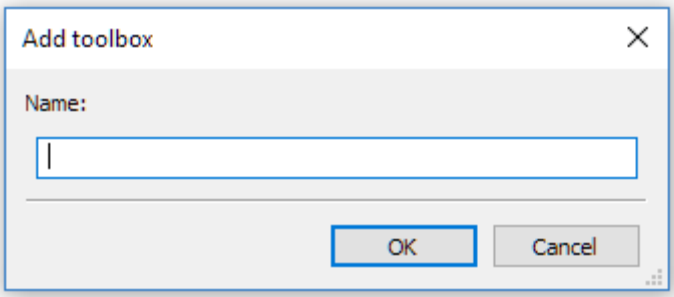
The newly created *Partial CUI file* is loaded and a *Menu Group* of the same name is created in the *Menus* tree, *Toolbars* tree, *Keyboards* tree and *Available Tools* tree.

Add a Toolbox to a Menu Group

1. In the Customize dialog select a menu group.
2. Right click and choose *Append toolbox* in the context menu.



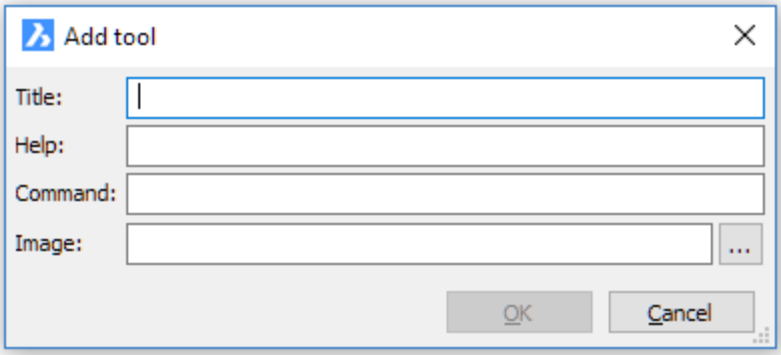
4. Type a name for the new *toolbox* in the *Title* field of the *Add toolbox* dialog.



5. Click the *OK* button.
   The toolbox is added in the *Available Tools* tree.
6. Click the *Apply* button to save the definition.

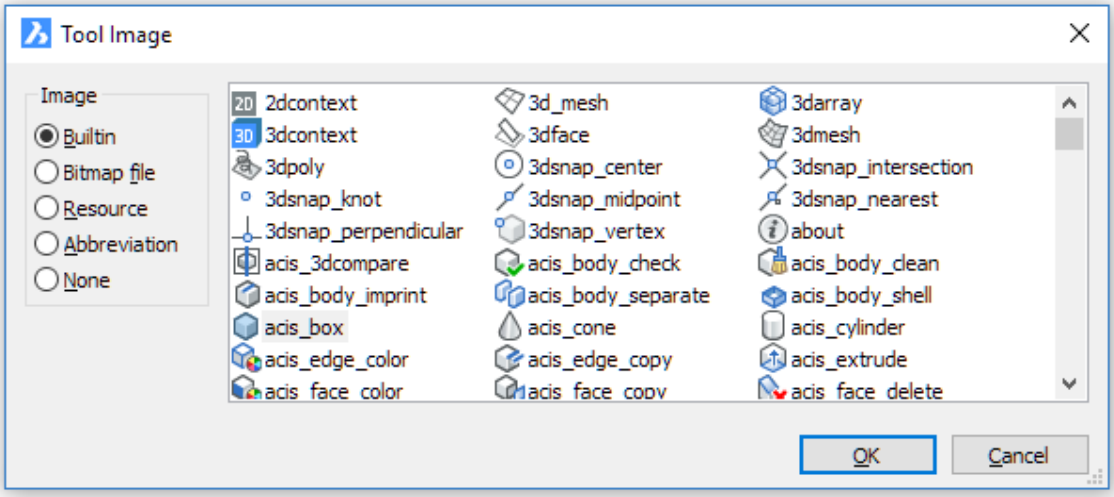
Add Tools to a Toolbox

1. In the Customize dialog select a toolbox in the *Available Tools* tree.
2. Right click and choose *Append tool* in the context menu.
   The *Add tool* dialog displays.



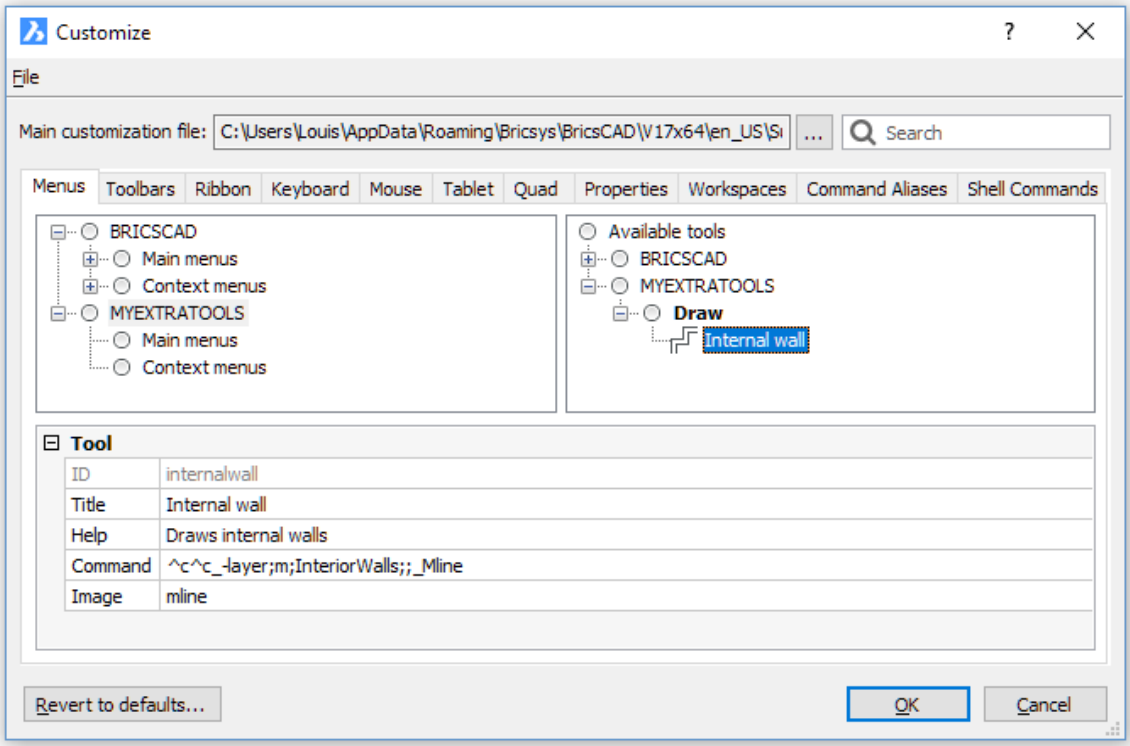
3. Type a name for the new tool in the *Title* field of the *Add tool* dialog.
4. (option) Type a help string in the *Help* field of the properties grid.
5. Type a command or command macro in the *Command* field of the properties grid.
6. Click the *Browse* button (...), then select an image for the new tool.
   Do one of the following:

   - Select *Builtin*, then choose one of the built-in icons.



   - Select *Bitmap file*, and choose a bmp, gif, jpg, jpeg or png image file.

7. Click the *OK* button.
   The tool is added in the toolbox.



In the above example the *Command* field contains a macro that makes the *InteriorWalls* layer current, then launches the *Mline* command.
If the layer does not exist yet, it is created first.

NOTE

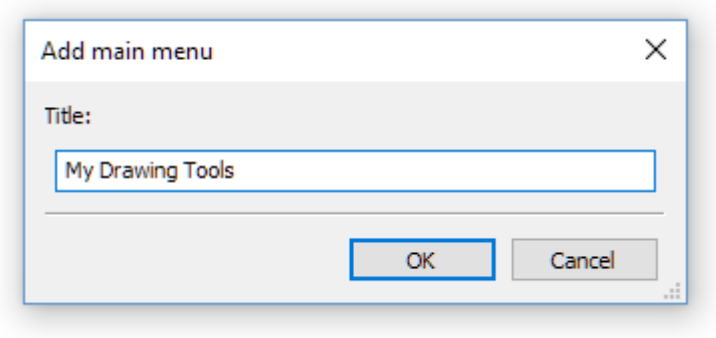
- *^c^c* is the equivalent of pressing the Esc key twice, which first ends a running command if necessary.
- an underscore (_) in front of a command name ensures that the macro still works in a non-English version of BRICSCAD.
- a point (.) in front of a command name ensures that the macro still works if this command is temporarily undefined (see the UNDEFINE and REDEFINE commands).
- a minus sign (-) in front of a command launches the command line version of a command, thus preventing the command dialog window to open.
- a semicolon (;) is the equivalent of pressing the Enter key
- a backslash (\) waits for user input, e.g. specifying a point.

Add a Main Menu

1. In the Customize dialog click the *Menus* tab.
2. Do one of the following:
   - Click the *Main Menus* subgroup of the menu group, then right click and choose *Append main menu* in the context menu.
   - Select an existing menu in the *Main Menus* subgroup, then right click and choose *Insert main menu* in the context menu.

New menus are added above the selected existing menu or as the first menu if you select the *Main Menus* subgroup.

3. Type a name in the *Title* field of the *Add main menu* dialog.



4. Click the *OK* button.
5. (option) Repeat steps 2 through 5 to add more menus.
6. Click the *Apply* button to save the definition(s).

NOTE The new main menu is not added to the Menu bar until at least one menu items is added to it.

Add a Submenu to a Menu

1. In the Customize dialog click the *Menus* tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. If collapsed, expand the *Main menus* group of the menu group.
4. Do one of the following:
   - Select the menu, then right click and choose *Append submenu* in the context menu.
   - Select an existing item in the menu, then right click and choose *Insert submenu* in the context menu.

New submenus are added above the selected existing item or as the first item if you select the menu.

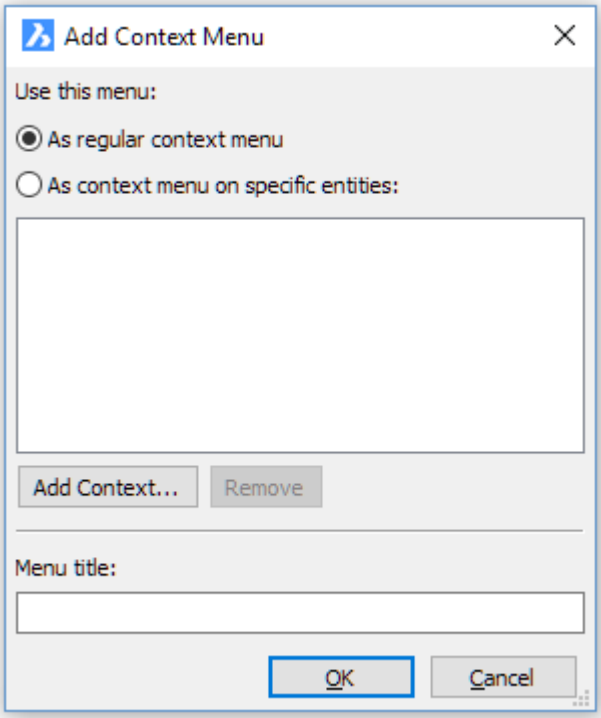
5. Type a name in the *Title* field of the *Add submenu* dialog.
6. Click the *OK* button.

Add a Context Menu

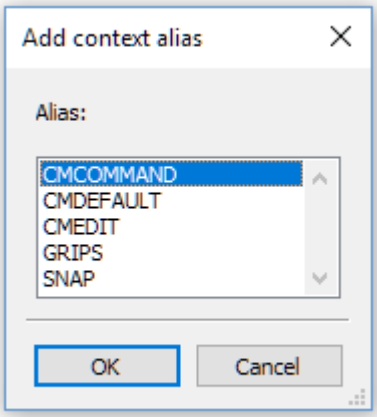
1. In the Customize dialog click the *Menus* tab.
2. Do one of the following:
   - Select the *Context Menus* subgroup of the menu group, then right click and choose *Append context menu* in the context menu.
   - Select an existing context menu, then right click and choose *Insert context* menu in the context menu.

New context menus are added above the selected existing context menu or as the first context menu if you select the *Context Menus* subgroup.

3. Select a menu in the *menu* list of the *Add context menu* dialog.



4. (option) Select *As regular context menu*, then click the *Add Context...* button.
The *Add context alias* dialog box displays:



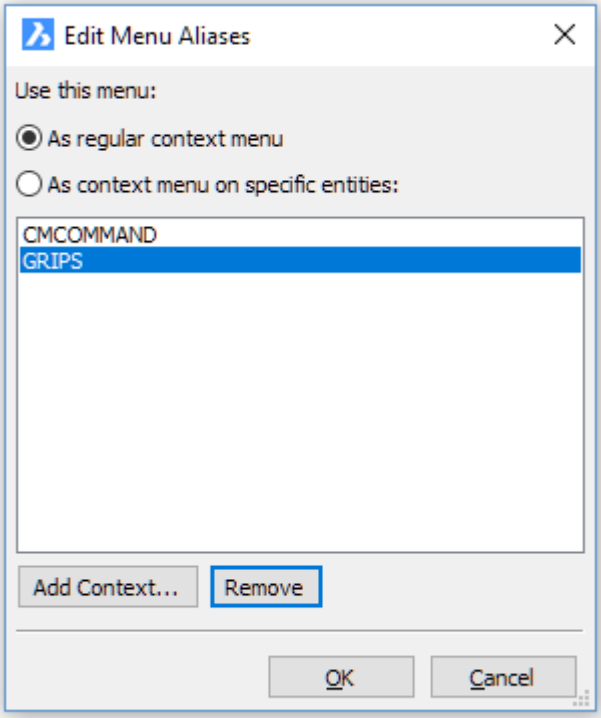
Choose:

- CMCOMMAND: to replace the default *Command* context menu.
- CMDEFAULT: to replace the default *Default* context menu.
- CMEDT: to replace the default *Edit* context menu.
- GRIPS: to replace the default *Grips* context menu.
- SNAP: to replace the default *Entity Snap* context menu.

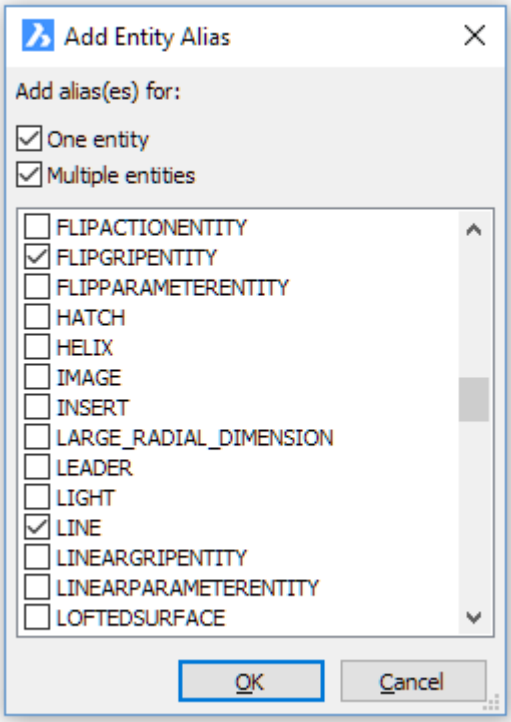
Click the *OK* button to confirm.

Optionally repeat the previous step to add another alias.

Select an alias, then click the *Remove* button to remove the alias.



5. (option) Select *As context menu on specific entities:*, then click the *Add Entity Type(s)...* button.
The *Add Entity Alias* dialog box displays:



Do one of the following:

- Check the *One entity* option: the context menu applies if one entity is selected only..
- Check *Multiple entities*: the context menu applies in multiple entities are selected only.
- Check both options: the context menu applies if one or more entities are selected.

Select one or more entity types in the list.

Click the *OK* button to confirm.

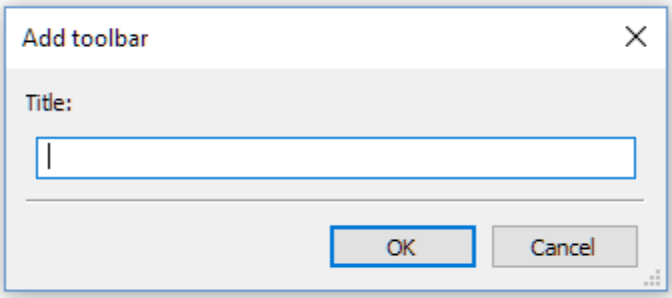
6. Type a name in the Menu title field.
7. Click the OK button to save the context menu.
8. (option) Repeat steps 2 through 5 to add more context menus.

Add a Toolbar

1. In the Customize dialog click the *Toolbars* tab.
2. Do one of the following:
   - Select a menu group, then right click and choose *Append toolbar* in the context menu.
   - Select an existing toolbar in a menu group, then right click and choose *Insert toolbar* in the context menu.

New toolbars are added above the selected existing toolbar or as the first toolbar if you select the menu group.

4. Type a name in the *Title* field of the *Add toolbar* dialog.



5. Click the *OK* button.
6. (option) Adjust the Toolbar properties.

Toolbar	
Title	My Drawing Tools
Alias	TB_MyDrawingTools
Position	Floating
Visible	Hide
Rows	1
Xval	
Yval	

- *Title*: The toolbar name must include alphanumeric characters with no punctuation other than a dash (&endash;) or an underscore (_). The *Title* along with the *Alias* enables the toolbar to be referenced programmatically.
- *Position*: Defines the display position of the toolbar when the CUI file is loaded. The options are: *Floating*, *Top*, *Left*, *Bottom* or *Right*. The latter four refer to the window edge where the toolbar will be docked.
- *Display*: Defines whether the toolbar displays or not when the CUI file is loaded.
- *Rows*: Specifies the number of rows when the toolbar position is floating. View bars with controls cannot have multiple rows.
- *Xval*: When the position is Floating, specifies the X coordinate in pixels, measured from the left edge of the screen to the left side of the toolbar.
- *Yval*: When the position is Floating, specifies the Y coordinate in pixels, measured from the top edge of the screen to the top of the toolbar.

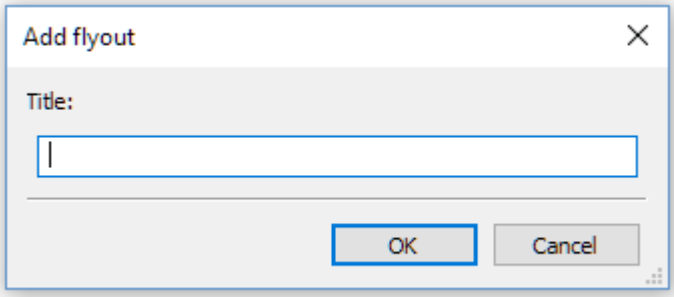
6. Click the *Apply* button to save the definition(s).

NOTES The new toolbar is loaded only if the *Display* property is set to *Show*. See Opening a Toolbar for more information on opening toolbars.

Add a Flyout to a Toolbar

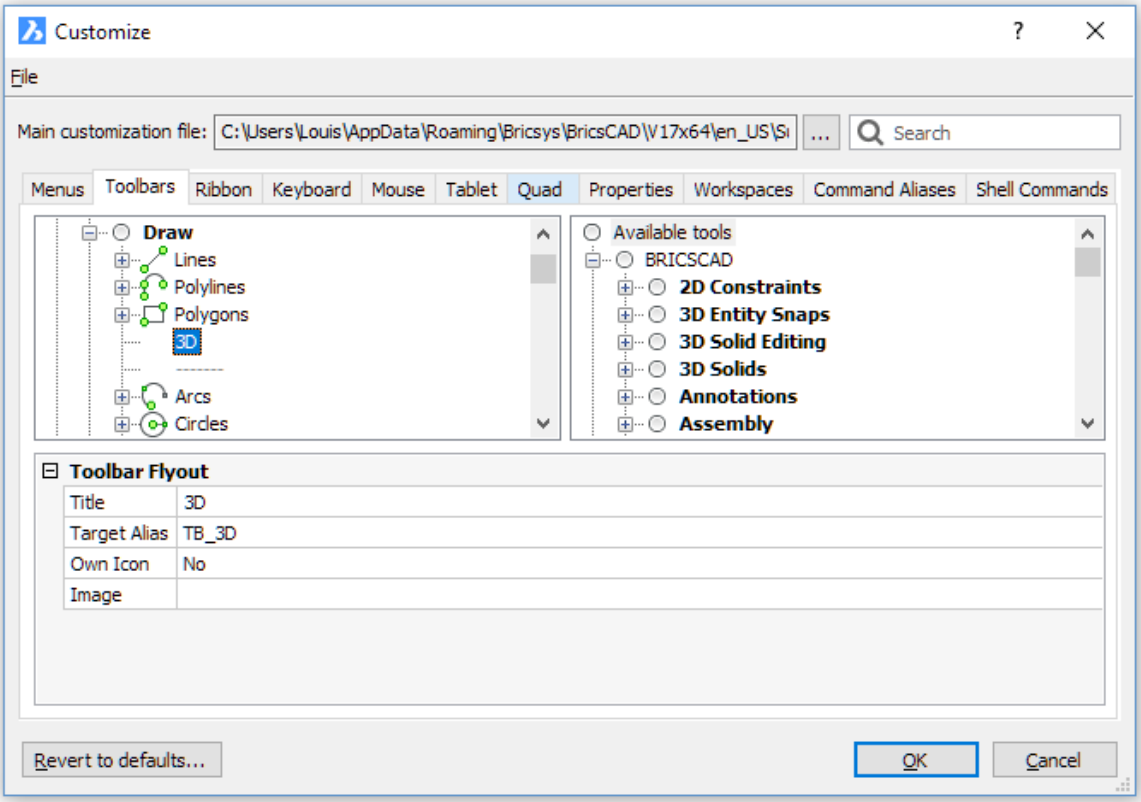
1. In the Customize dialog click the *Toolbars* tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. Do one of the following
   - Select a toolbar, then right click and choose *Append Flyout* in the context menu, to add the flyout at the end of the toolbar.
   - Expand the toolbar, right click a tool then choose *Insert flyout* in the context menu, to add the flyout above the selected tool.

The *Add flyout* dialog box displays:



4. Type a name in the *Title* field of the *Add flyout* dialog, then click the *OK* button.

A new empty toolbar is created and inserted as a flyout in the selected toolbar:



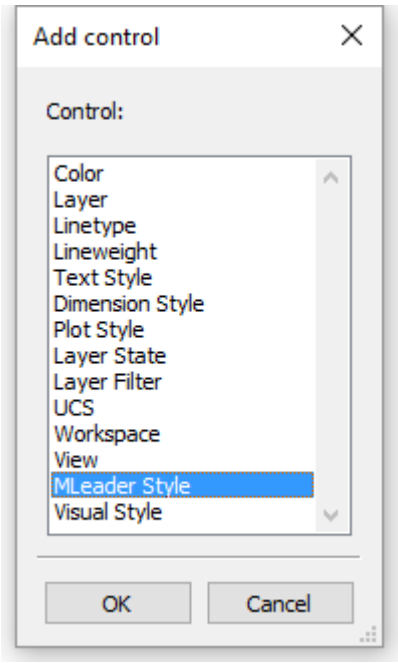
5. Do one of the following:

- Click the *Target Alias* field, then click the settings field and select an existing toolbar in the drop-down list.
- Add tools to the new toolbar.

Add a Control to a Toolbar

1. In the Customize dialog click the *Toolbars* tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. Do one of the following:
   - Select the toolbar, then right click and choose *Append control* in the context menu.
     The new control will be added as the first item in the toolbar.
   - Select an existing item in the toolbar, then right click and choose *Insert control* in the context menu.
     The new control will be added above the selected existing item.

4. Select a control in the *Add control* dialog.



5. Click the *OK* button.
6. To specify the width of the control, do one of the following:

   - Accept the *Default* width.
   - Type the width in pixels in the *Width* field.

Toolbar Control	
Control	Dimension Style
Width	Default

NOTE Type *Default* or *0* (zero) in the *Width* field to restore the default width of the control field.

Add a Keyboard Shortcut

1. In the Customize dialog click the *Keyboard* tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. Do one of the following:
   - Select the menu group, then right click and choose *Append shortcut* in the context menu.
     The new shortcut will be added at the end of the shortcut list.
   - Select an existing shortcut in the toolbar, then right click and choose *Insert shortcut* in the context menu.
     The new shortcut will be added above the selected existing shortcut.
6. Do one of the following:
   - Proceed with the Add an existing tool procedure.
   - Proceed with the Add a new tool procedure.

5. In the *Properties* grid of the new shortcut select the *Key* field, then press the key combination you want to assign to the shortcut.

Keyboard Shortcut	
Key	Ctrl+G
Tool ID	ID_GROUP_DIRECT
Help	Creates an unnamed group of the selected entities
Command	-group;;*;;
Image	custom_2

To add an existing tool to a menu, toolbar or shortcut

1. In the Customize dialog select the item you want to add an existing tool to.
2. In the *Available Tools* pane, expand the toolbox that contains the tool you want to add.
3. Drag the tool to the target item:
   - Drop the tool on top of the item name to append the tool.
   - Drop the tool on top of another tool to insert the new tool above the existing tool.

4. (option) Edit the tool in the settings grid, e.g. to add a diesel expression in the *Diesel* field.

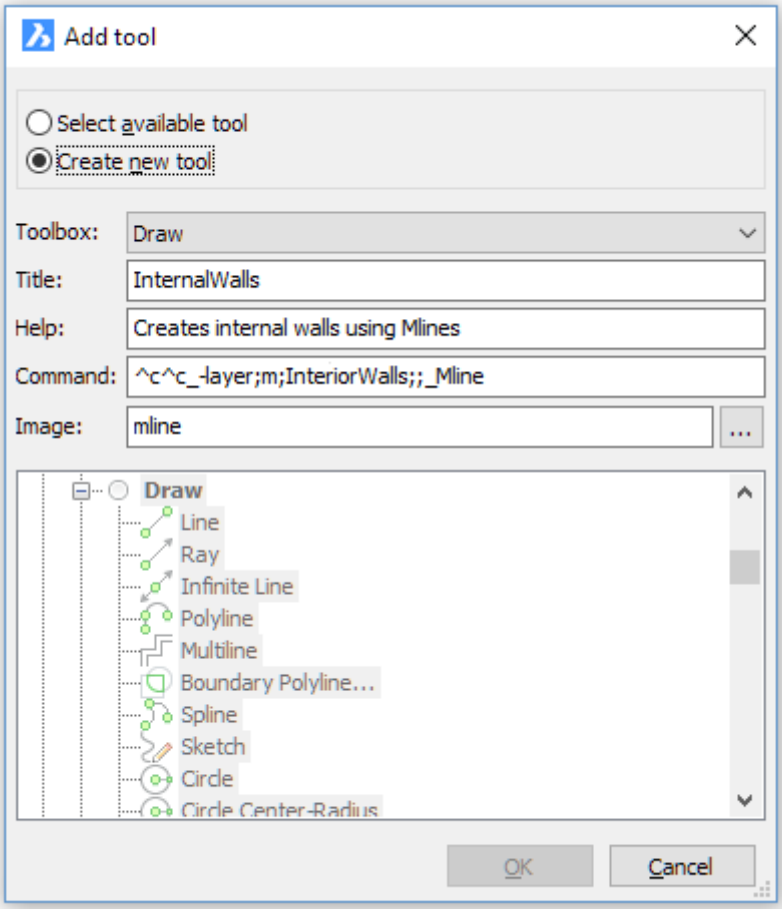
Toolbar Button	
Title	Snap to Nearest
Diesel	$(if,$(=,$(and,$(getvar,OSMODE),0x0200),0),,!.)
Tool ID	esnap_nearest
Help	Snaps to the nearest point on an entity
Command	_nearest
Image	esnap_nearest

To change the order of the tools in a menu or toolbar

1. In the Customize dialog select the menu or toolbar.
2. Drag the tool to its new position:
   - Drop the tool on top of the item name to move it to the last position.
   - Drop the tool on top of another tool to move the tool above this tool.

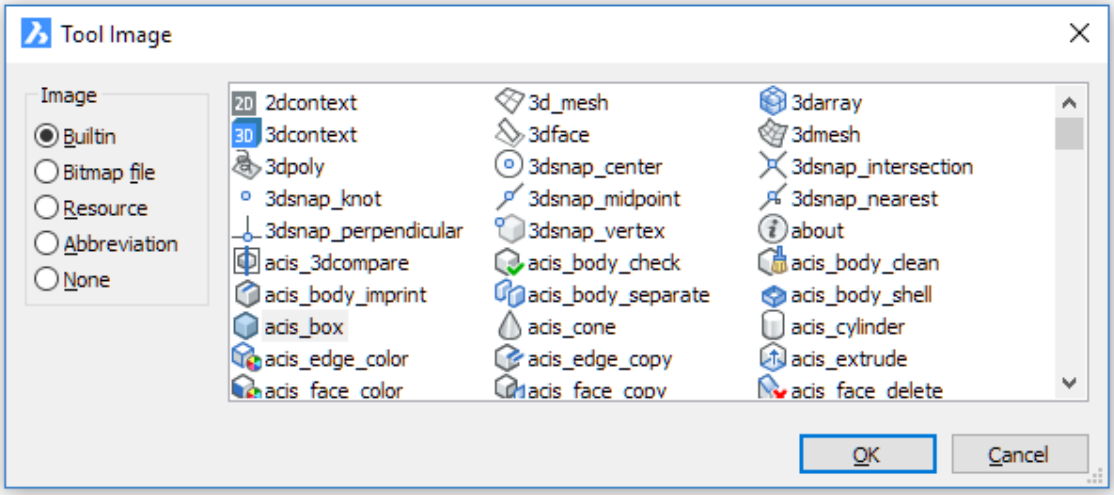
Create a New Tool

1. In the *Add* dialog choose *Create new tool.*
2. Select the *Toolbox* in which you want to create the new tool.



3. Type a name for the new tool in the *Title* field of the *Add tool* dialog.
4. (option) Type a help string in the *Help* field of the properties grid.
5. Type a command or command macro in the *Command* field of the properties grid.
   In the screenshot above the *Command* field contains a macro that makes the *InteriorWalls* layer current, then launches the *Mline* command.
   If the layer does not exist yet, it is created first.
6. Click the *Browse* button (...), then select an image for the new tool.
   Do one of the following:

- Select *Builtin*, then choose one of the built-in icons.



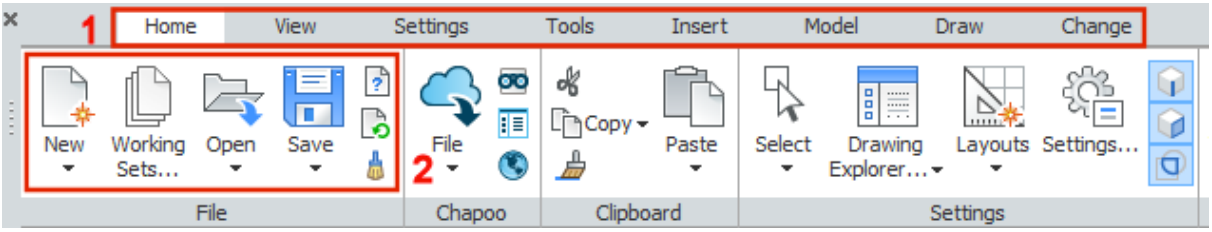
- Select *Bitmap file*, and choose a bmp, gif, jpg, jpeg or png image file.

7. Click the *OK* button to create the new tool.

Ribbon

Commands: RIBBON and RIBBONCLOSE

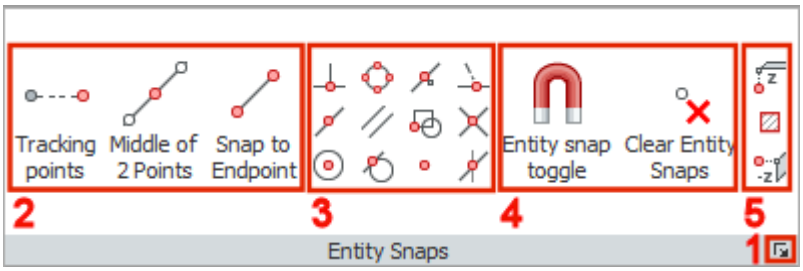
A ribbon organizes tools in a series of *Panels* (2) which are grouped in different *Tabs* (1).
The number of tabs and their order depends on the current workspace and is defined in the *Ribbon* node of the workspace (see Editing a Workspace).



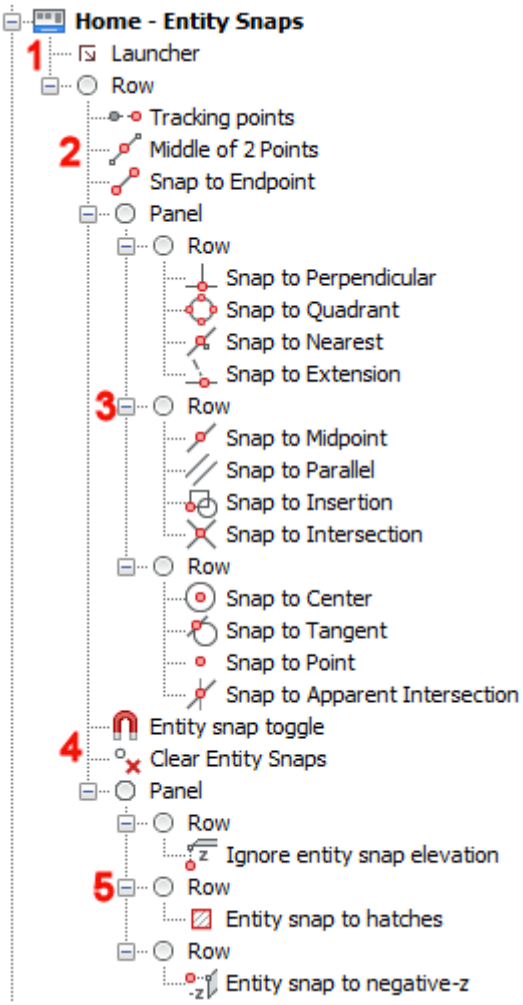
The RIBBONDOCKEDHEIGHT system variable stores the preferred height of the Ribbon bar. Values between 0 and 500 are accepted. Set the value to 0 (zero) for automatic height calculation.

Ribbon Panels

A ribbon panel is composed of one or more rows. In a row you can add one or more command buttons or row panels, which in turn can hold one or more command buttons.



Ribbon Panel



Structure of the Ribbon Panel in the Customize dialog

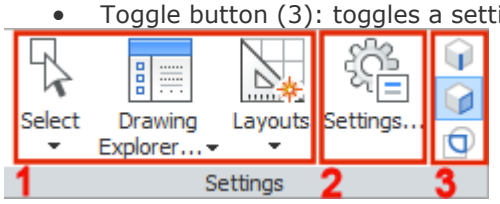
The *Entity Snaps* ribbon panel above is composed of:

- a launcher (1)
- a single row, holding three command buttons (2), a row panel (3) with three rows of four command buttons each, two command buttons (4) and a row panel (5) with three rows of one command each.

Ribbon Buttons

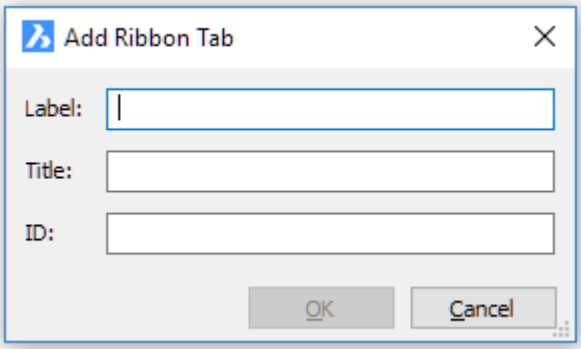
The following button types are available:

- Command button (2): launches a command
- Split button (1): groups a series of command buttons, much like a flyout toolbar.
- Toggle button (3): toggles a setting; shows the current status of setting



Creating a Ribbon Tab

1. Open the Customize dialog.
2. Click the *Ribbon* tab on the *Customize* dialog.
3. Right click the *Ribbon tabs* group and choose *Append ribbon tab* in the context menu.
4. Fill out the fields in the *Add Ribbon Tab* dialog box:



Label: Caption on the ribbon tab.
Title: Used in the *Select Ribbon Tab* dialog when selecting ribbon tabs for a workspace (see Editing a Workspace).
ID: A unique name to identify the ribbon tab.

5. Click the *OK* button.

Editing a ribbon tab

1. Open the Customize dialog.
2. Click the *Ribbon* tab on the *Customize* dialog.
3. Expand the *Ribbon tabs* group.

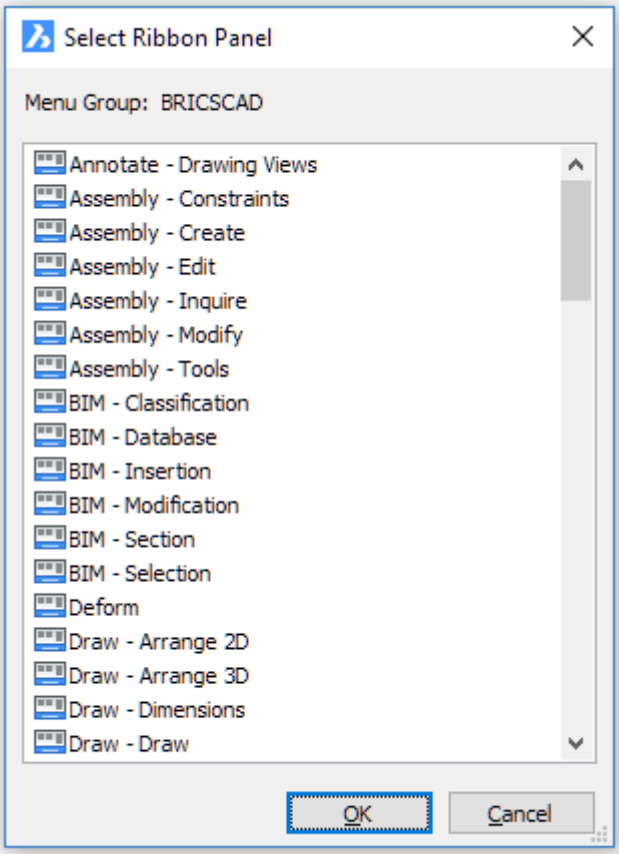
To add a panel to a ribbon tab

1. Do one of the following:

   Right click the ribbon tab and choose *Append panel* in the context menu, to add a panel at the end.

   Right click an existing panel and choose *Insert panel* in the context menu, to ad a panel above the existing panel.

2. Select the panel in the *Select Ribbon Panel* dialog, then click the *OK* button.



3. (option) Drag the panel to a different position.

To remove a panel from a ribbon tab

1. Select the panel.
2. Do one of the following:

   Press the *Delete* key.

   Right click and choose *Remove* in the context menu.

To add a row panel to a row

Do one of the following:

- Right click the row and choose *Append Row Panel* in the context menu.
  The row panel is added at the bottom of the row tree.
- Right click an existing row panel and choose *Insert Row Panel* in the context menu.
  The row panel is inserted above the existing row panel.

To add a command button to a row

- Drag and drop a tool from the *Available Tools* list onto the row.
  Drop the tool on the row to add a command button at the bottom of the row tree.
  Drop the tool on an existing command button to add a command button above an existing command button.
- Set the *Button Style* in the *Command Button* grid.

Command Button	
Label	Line
Button Style	Small without text
Key Tip	
Tool ID	line
Help	Draws a line
Command	^c^c_line
Image	line

The options are:

Icon	Style
	Small
Polyline	Small with text
	Large
Spline	Large with text

- (option) Drag the command button to a different position in the same row or onto a different row.

To add a split button to a row

- Select the row, then right click and choose *Append split button* in the context menu. The split button is added at the bottom of the row tree.
- Drag a series of related tools from the *Available Tools* list onto the split button.
- Set the *Button Style* in the *Split Button* grid.

Split Button	
Label	
Button Style	Small without text
Behavior	Split with recent
List Style	Icon with text
Grouping	No
Image	
Key Tip	

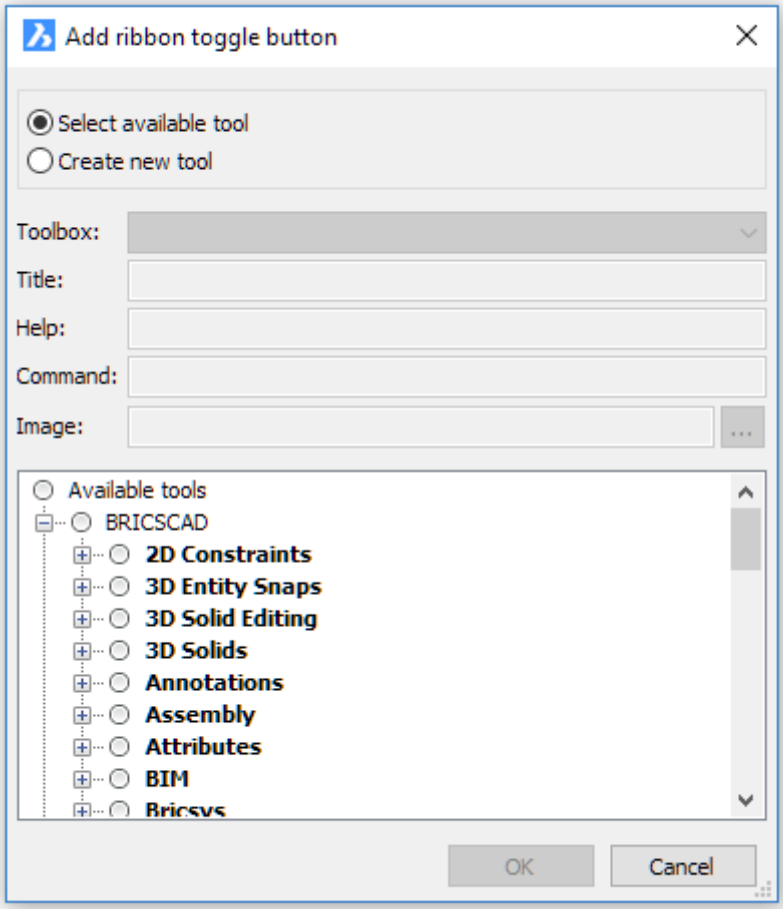
- The options are:

Icon	Style
	Small
Polyline	Small with text
	Large
Spline	Large with text

- Type a label text in the *Label* field.
- (option) Set the *Behavior* to *Split with recent (static text)*: always display the split button label, instead of the label of the most recently used tool.
- (option) Drag the split button to a different position in the same row or onto a different row.
- Drag and drop tools from the *Available Tools* list onto the split button.

To add a toggle button to a row

- Select the row, then right click and choose *Append Toggle Button* in the context menu.
- Either select an available tool or Create a New Tool, then click the *OK* button.



- The toggle button is added at the bottom of the row tree.
- (option) Drag the toggle button to a different position in the same row or onto a different row.

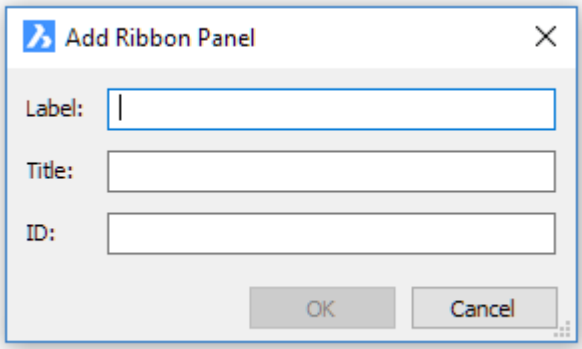
To add a control to a row

Drag a control from the controls list onto the row.

- Drop the control on the row to add it at the bottom of the row tree.
- Drop the control on a existing command button to add it above the command button.

Creating a Ribbon Panel

1. Open the Customize dialog.
2. Click the *Ribbon* tab on the *Customize* dialog.
3. Right click the *Ribbon Panels* group and choose *Append ribbon panel* in the context menu.
4. Fill out the fields in the Add Ribbon Panel dialog box:



Label: Caption of the panel in the ribbon.
Title: Used in the *Select Ribbon Panel* dialog when selecting the panels for a ribbon tab
ID: A unique name to identify the ribbon panel.

5. Click the *OK* button.

Editing a Ribbon Panel

1. Open the Customize dialog.
2. Click the *Ribbon* tab on the *Customize* dialog.
3. Expand the *Ribbon Panels* group.
4. Expand the ribbon panel.

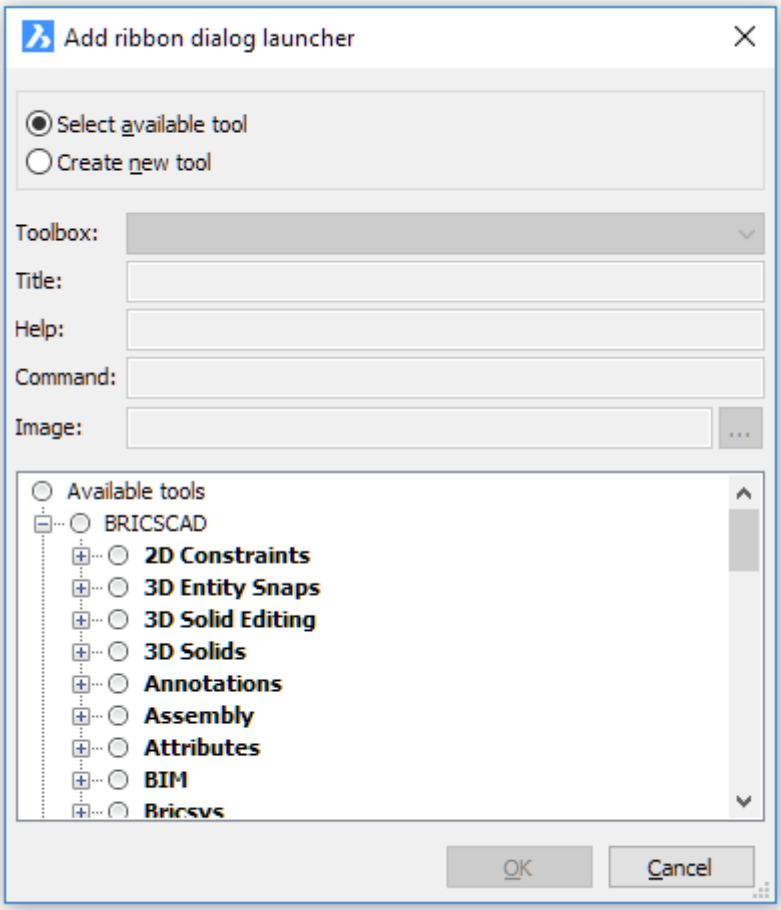
To add a row to a ribbon panel

Do one of the following:

- Right click the ribbon panel and choose *Append Row* in the context menu.
  The row is added at the bottom of the ribbon panel tree.
- Right click an existing row and choose *Insert Ribbon Row* in the context menu.
  The row is added above the existing row.
- (option) Drag the row to a different location in the ribbon panel tree.

To add a launcher to a ribbon panel

- Right click the ribbon panel and choose *Add launcher* in the context menu.
- Either select an available tool or Create a New Tool, then click the *OK* button.



Editing a Panel Row

1. Open the Customize dialog.
2. Click the *Ribbon* tab on the *Customize* dialog.
3. Expand the *Ribbon panels* group.
4. Expand the ribbon panel.
5. Expand the ribbon rows and ribbon row panels.

To add a row panel to a row

Do one of the following:

- Right click the row and choose *Append Row Panel* in the context menu.
  The row panel is added at the bottom of the row tree.
- Right click an existing row panel and choose *Insert Row Panel* in the context menu.
  The row panel is inserted above the existing row panel.

To add a command button to a row

- Drag and drop a tool from the *Available Tools* list onto the row.
  Drop the tool on the row to add a command button at the bottom of the row tree.
  Drop the tool on an existing command button to add a command button above an existing command button.
- Set the *Button Style* in the *Command Button* grid.

Command Button	
Label	Line
Button Style	Small without text
Key Tip	
Tool ID	line
Help	Draws a line
Command	^c^c_line
Image	line

The options are:

Icon	Style
	Small
Polyline	Small with text
	Large
Spline	Large with text

- (option) Drag the command button to a different position in the same row or onto a different row.

To add a split button to a row

- Select the row, then right click and choose *Append split button* in the context menu. The split button is added at the bottom of the row tree.
- Drag a series of related tools from the *Available Tools* list onto the split button.
- Set the *Button Style* in the *Split Button* grid.

Split Button	
Label	
Button Style	Small without text
Behavior	Split with recent
List Style	Icon with text
Grouping	No
Image	
Key Tip	

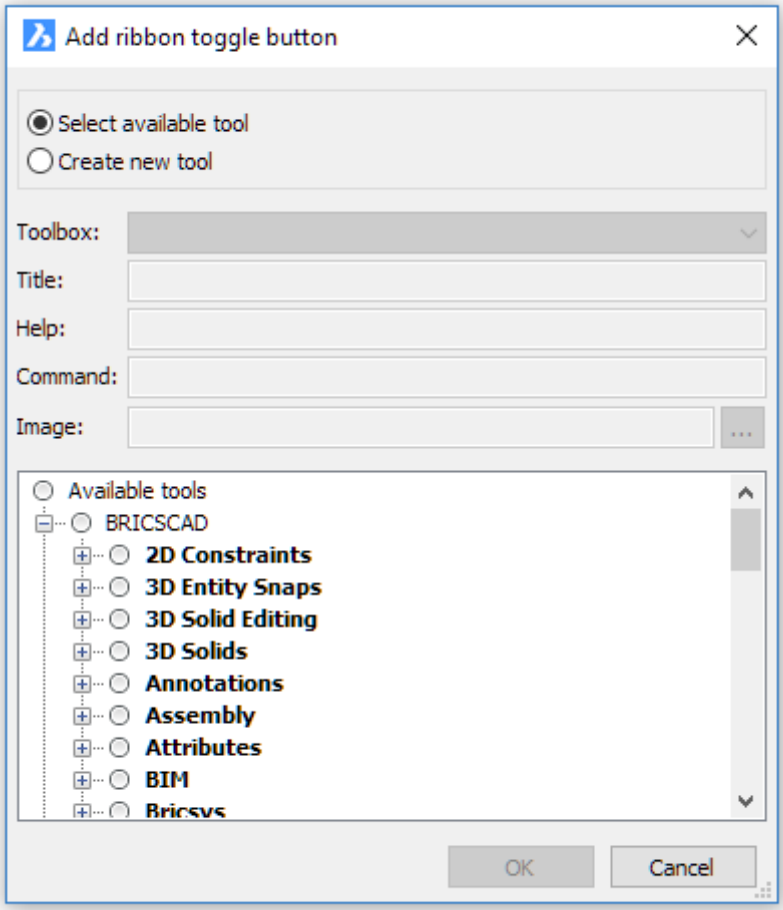
The options are:

Icon	Style
	Small
Polyline	Small with text
	Large
Spline	Large with text

- Type a label text in the *Label* field.
- (option) Set the *Behavior* to *Split with recent (static text)*: always display the split button label, instead of the label of the most recently used tool.
- (option) Drag the split button to a different position in the same row or onto a different row.
- Drag and drop tools from the *Available Tools* list onto the split button.

To add a toggle button to a row

- Select the row, then right click and choose *Append Toggle Button* in the context menu.
- Either select an available tool or Create a New Tool, then click the *OK* button.



- The toggle button is added at the bottom of the row tree.
- (option) Drag the toggle button to a different position in the same row or onto a different row.

To add a control to a row

Drag a control from the controls list onto the row.

- Drop the control on the row to add it at the bottom of the row tree.
- Drop the control on a existing command button to add it above the command button.

Tool Palettes

Command: TOOLPALETTES, TOOLPALETTESCLOSE, TPNAVIGATE

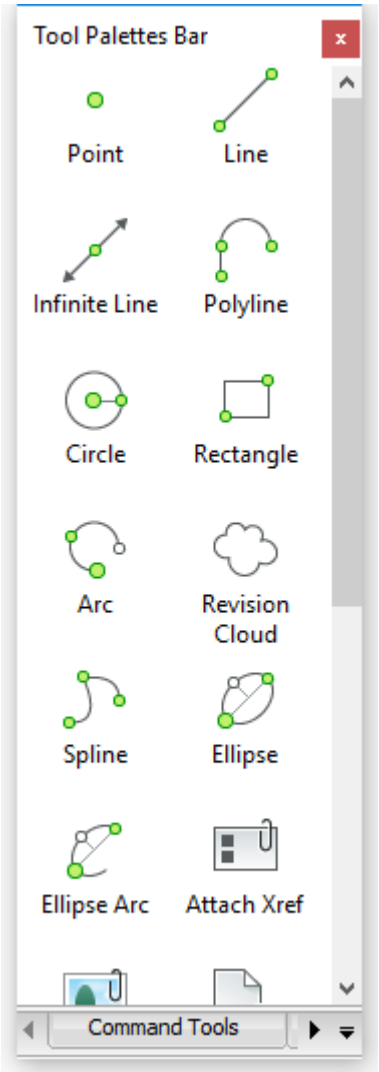
The *ToolPalettes* command opens the Tool Palettes bar.
The *ToolPalettesClose* command closes the Tool Palettes bar.
The *Tpnavigate* command displays a tool palette or a tool palette group.

On a tool palette you can add blocks, hatches and commands in a tabbed window.
Drag a tool from the palette to the graphic screen to launch the related tool.

To Open the Tool Palettes Bar

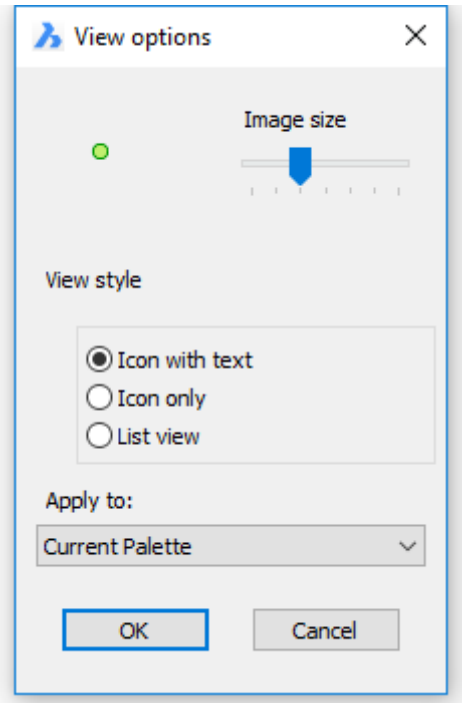
Do one of the following:

- Hover over a toolbar (Windows) or an empty toolbar area (Windows and Linux), then right click and choose *Tool Palettes Bar* in the context menu.
- Type *toolpalettes* in the command bar, then press Enter.



To Set the View Options

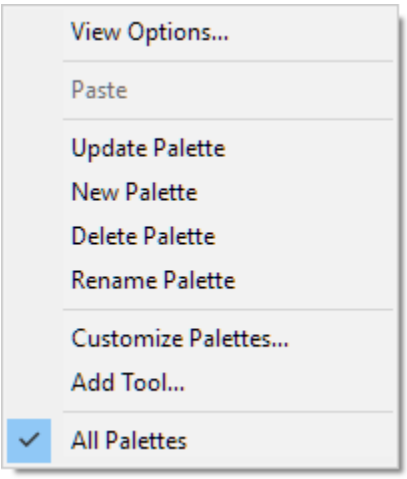
1. Right click in an empty space of the icon area on the *Toolpalettes Bar* and choose *View Options...* in the context menu.
   The *View Options* dialog displays.



2. Drag the *Image Size* slider to adjust the image size.
3. Select either *Icon with text*, *Icon only* or *List view.*
4. Choose either *Current Palette* or *All Palettes* from the *Apply to* list button.
5. Click the *OK* button to confirm.

Adding a New Tool Palette

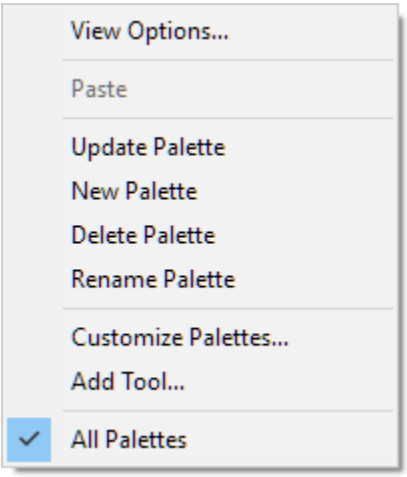
1. Move the cursor to the icon area of the *Tool Palettes Bar*, then right click.
   A context menu displays:



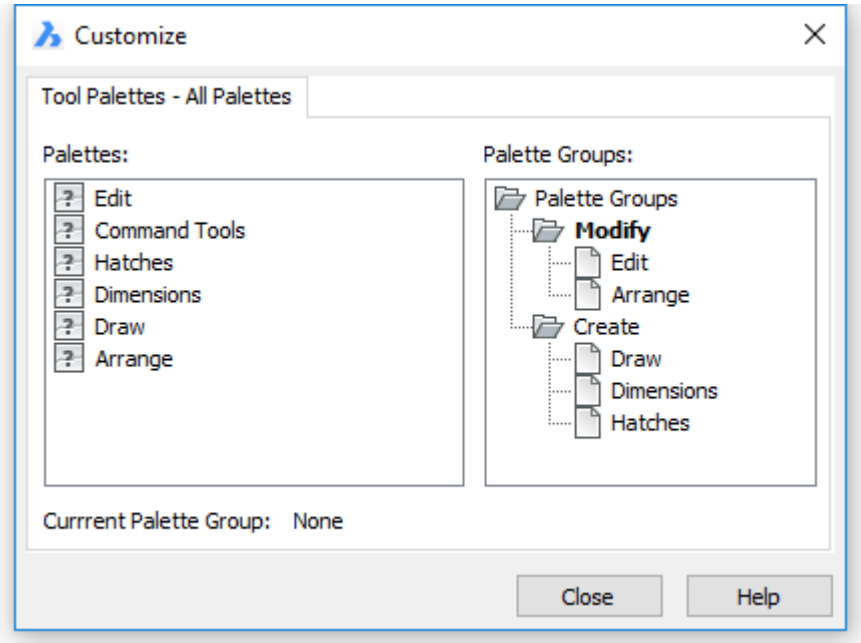
2. Choose *New Palette* in the context menu.
   A new tab is added.
3. Type a name for the new palette in the *Name* field.

To open the customize dialog box

1. Move the cursor to the icon area of the *Tool Palettes Bar*, then right click.
   A context menu displays:



2. Choose *Customize Palettes* in the context menu.
   The *Customize* dialog box displays:



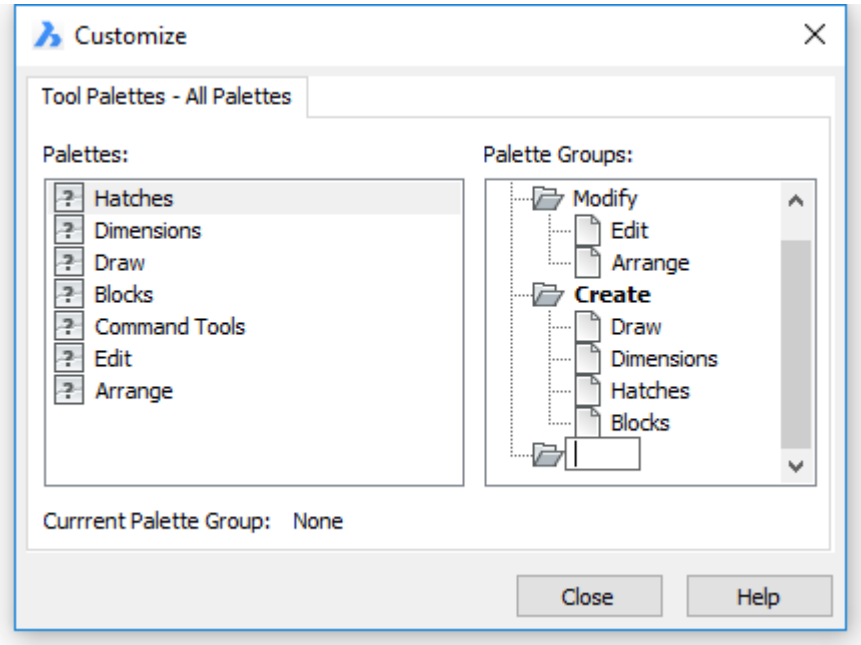
Importing a Tool Palette

1. Open the Customize dialog box.
2. In the *Palettes* pane on the *Customize* dialog box, right click and choose *Import* in the context menu.
3. On the *Import Palette* dialog, select a folder, then select a palette file (*.btc or *.xtp).
4. Click the *Open* button or double click to import the palette.
   The palette is added in the *Palettes* list.
5. Click the *Close* button to close the *Customize* dialog box.

Working with Palette Groups

To create a palette group

1. Open the Customize dialog box.
2. Place the cursor on the *Palette Groups* root folder name.
3. Right click and choose *New Palette* in the context menu.
   A new group folder is added at the bottom of the palette groups tree.



4. Type a name for the new group.

To create a palette subgroup

1. Open the Customize dialog box.
2. Place the cursor on an existing group name.
3. Right click and choose *New Palette* in the context menu.
   A new group folder is added as a subgroup of the selected group.
4. Type a name for the new group.

To add a palette to a group

1. Open the Customize dialog box.
2. Place the cursor on a palette.
3. Press and hold the left mouse button, then drag the palette onto a palette group name.
4. Release the left mouse button to add the palette to the group.

To remove a palette from a group

1. Open the Customize dialog box.
2. Select the palette in the group.
3. Right click and choose *Remove* in the context menu.

To set a palette group current

1. Open the Customize dialog box.
2. Select the palette group in the palette groups tree.
3. Right click and choose *Set Current* in the context menu.

To delete a palette group

1. Open the Customize dialog box.
2. Select the palette group in the palette groups tree.
3. Right click and choose *Delete* in the context menu.

To Rename a Tool Palette

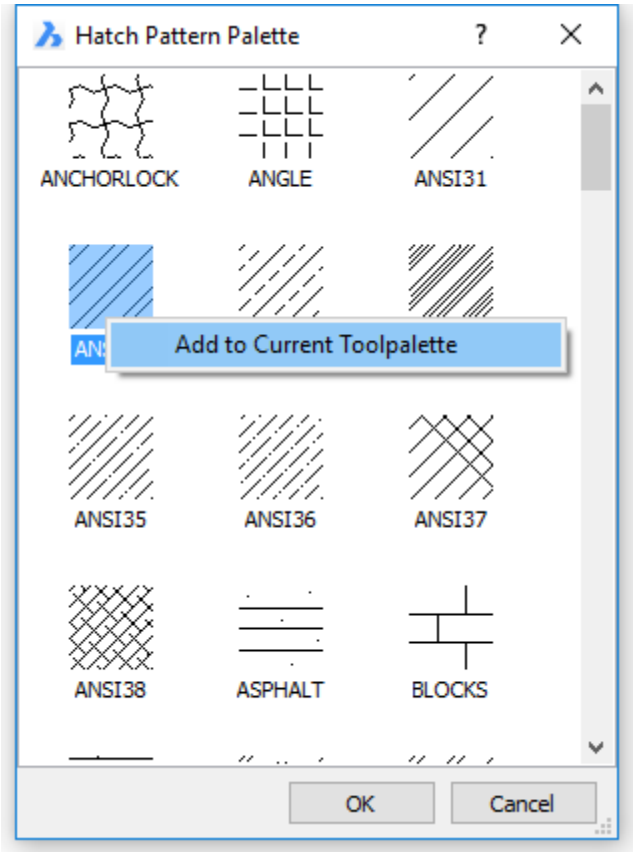
1. Right click tool palette tab and choose *Rename Palette* in the context menu.
2. Type a new name for the palette in the *Name* field.
3. Press Enter to confirm.

To Delete a Tool Palette

1. Right click tool palette tab and choose *Delete Palette* in the context menu.
2. Click the *OK* button on the *Confirm Palette Deletion* box.

Adding a Hatch Pattern

1. Select the palette you want to add a hatch pattern to.
2. Open the Hatch and Gradient dialog box.
3. Select either *Predefined* or *Custom* from the *Type* list button.
4. Double click the *Swatch* field on the *Hatch and Gradient* dialog box.
   The *Hatch Pattern Palette* displays.
5. Select the hatch pattern you want to add, then right click and Choose *Add to Current Tool Palette*.
   The hatch pattern is added on the current tool palette.



6. (option) Repeat steps 5 and 6 to add more hatch patterns.
7. (option) Drag-and-drop patterns to change their order on the palette.

Adding blocks

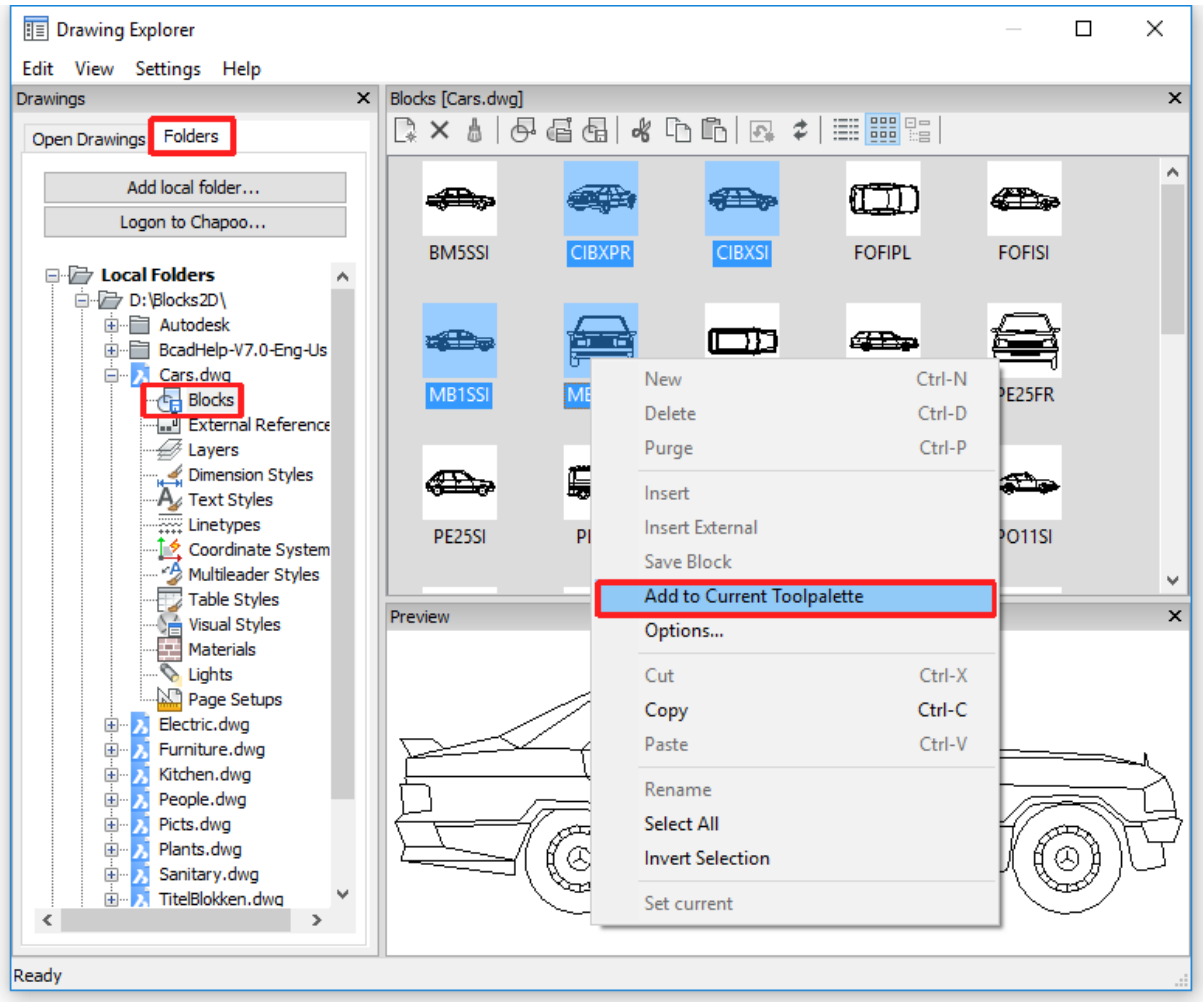
Blocks can be added from the current drawing or from any drawing.

Adding Blocks from the Current Drawing

1. Choose *Drawing Explorer - Blocks* in the *Tools* menu.
2. On the *Drawing Explorer - Blocks* dialog box select the block or blocks you want to add.
3. Right click and choose *Add to Current Tool Palette* in the context menu.

Adding Blocks from Any Drawing

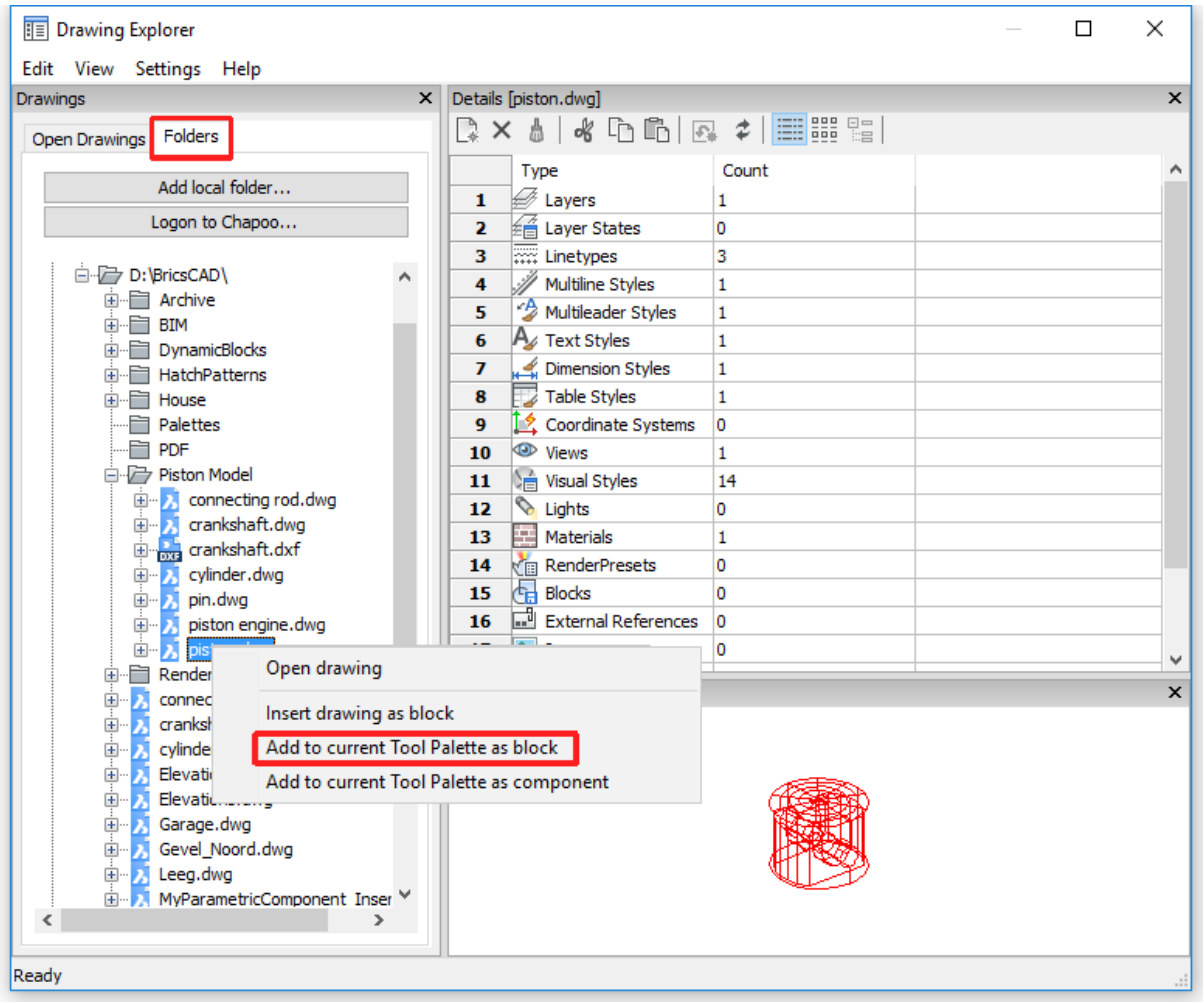
1. If not already available, add the containing folder to the Folders list on the *Drawing Explorer - Drawings* dialog box.
2. Select the drawing in the containing folder on the *Drawing Explorer - Drawings* dialog box.
3. Click the *Expand* icon (+) of the containing drawing.
   The *Blocks* icon of the drawing displays.
4. Click the *Blocks* icon of the containing drawing.
   The blocks in the drawing display in the *Details* pane.
5. Select the block or blocks you want to add.
   Press and hold the Ctrl key to select multiple blocks.
6. Right click and choose *Add to Current Tool Palette* in the context menu.



7. (option) Drag-and-drop blocks to change their order on the palette.

Adding a Drawing as a Block

1. If not already available, add the containing folder to the Folders list on the *Drawing Explorer - Drawings* dialog box.
2. Select the drawing in the containing folder on the *Drawing Explorer - Drawings* dialog box.
3. Right click and choose *Add to Current Toolpalette as block* in the context menu.

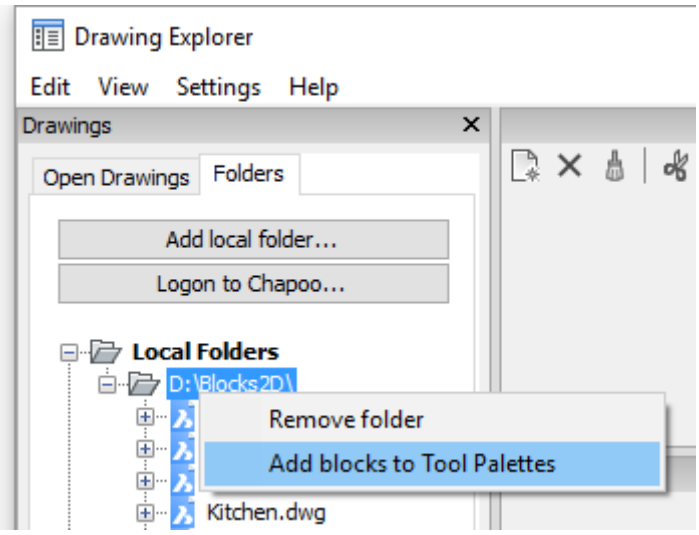


When placing a block from a toolpalette the -Insert command is launched.

Adding All Blocks from Multiple Drawings

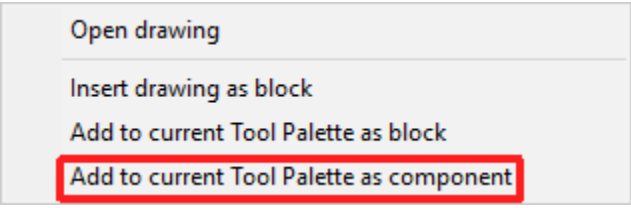
This procedure collects all blocks in each of the drawings of the selected folder and add them to a palette of the drawing name. New palettes are created if necessary.

1. If not already available, add the containing folder to the Folders list on the *Drawing Explorer - Drawings* dialog box.
2. Select the folder, then right click and choose *Add blocks to Tool Palettes* in the context menu.



To add a drawing as a component.

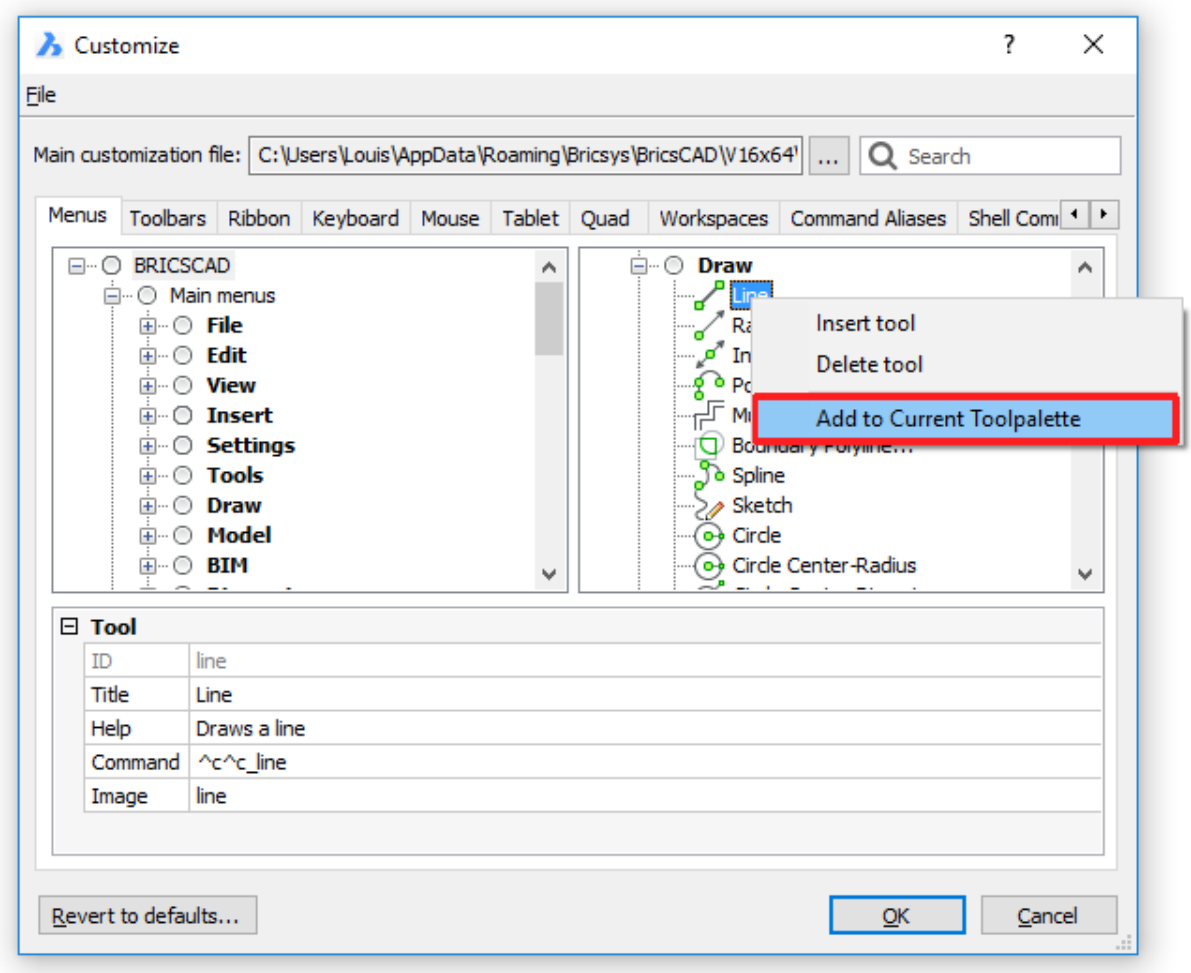
1. If not already available, add the containing folder to the Folders list on the *Drawing Explorer - Drawings* dialog box.
2. Select the drawing in the containing folder on the *Drawing Explorer - Drawings* dialog box.
3. Right click and choose *Add to Current Toolpalette as component* in the context menu.



When placing a component from the tool palette, the -BmInsert command is launched.

Adding a Component

1. Choose *Customize* in the *Tools* menu.
2. Under *Available Tools* on the *Customize* dialog, expand the containing toolbox, then select the command in the commands list.
3. Right click and choose *Add to Current Tool Palette* in the context menu.

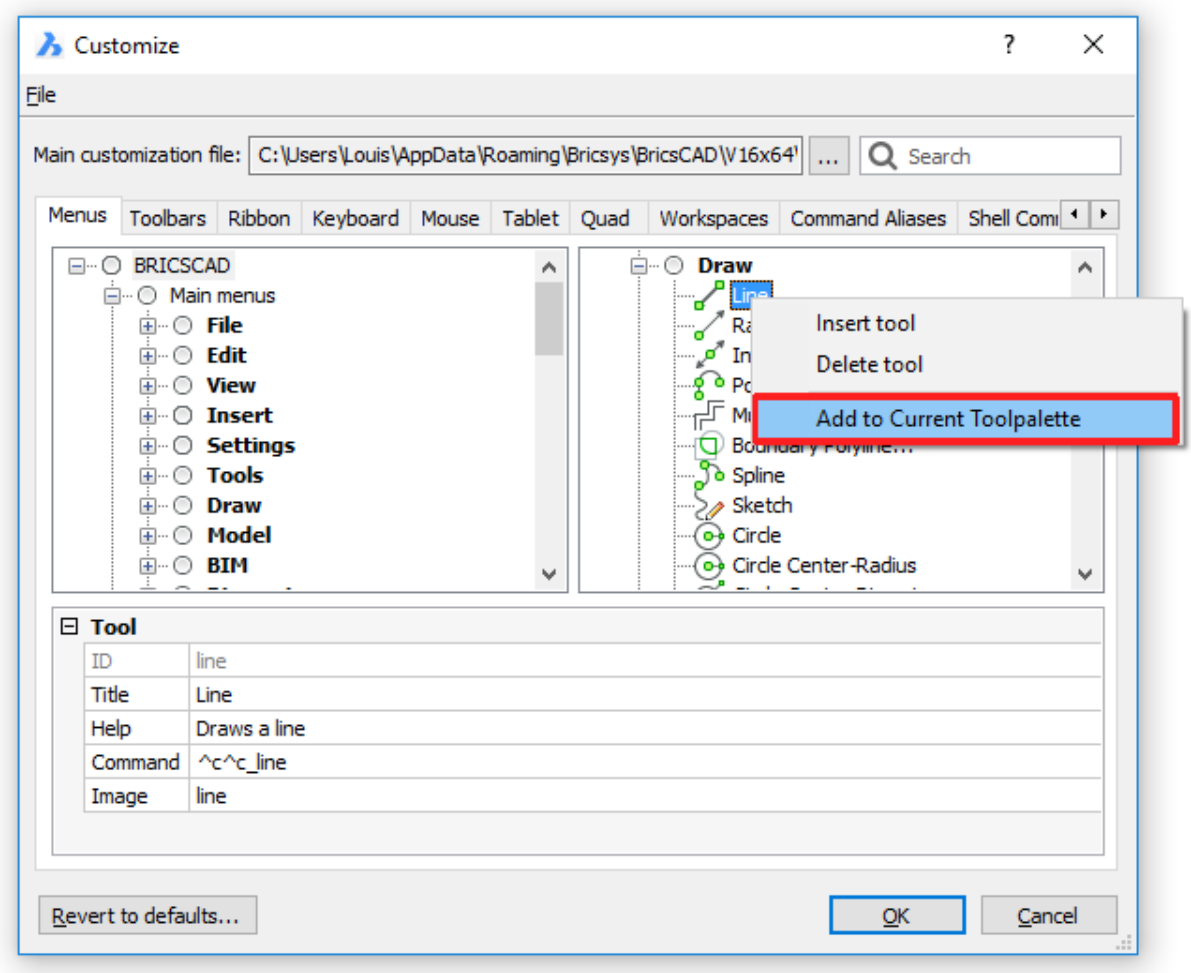


4. (option) Drag-and-drop commands to change their order on the palette.

NOTE See the topic Creating Custom Tools to create your own tools, which you can then add to a tool palette.

Adding a Command

1. Choose *Customize* in the *Tools* menu.
2. Under *Available Tools* on the *Customize* dialog, expand the containing toolbox, then select the command in the commands list.
3. Right click and choose *Add to Current Tool Palette* in the context menu.



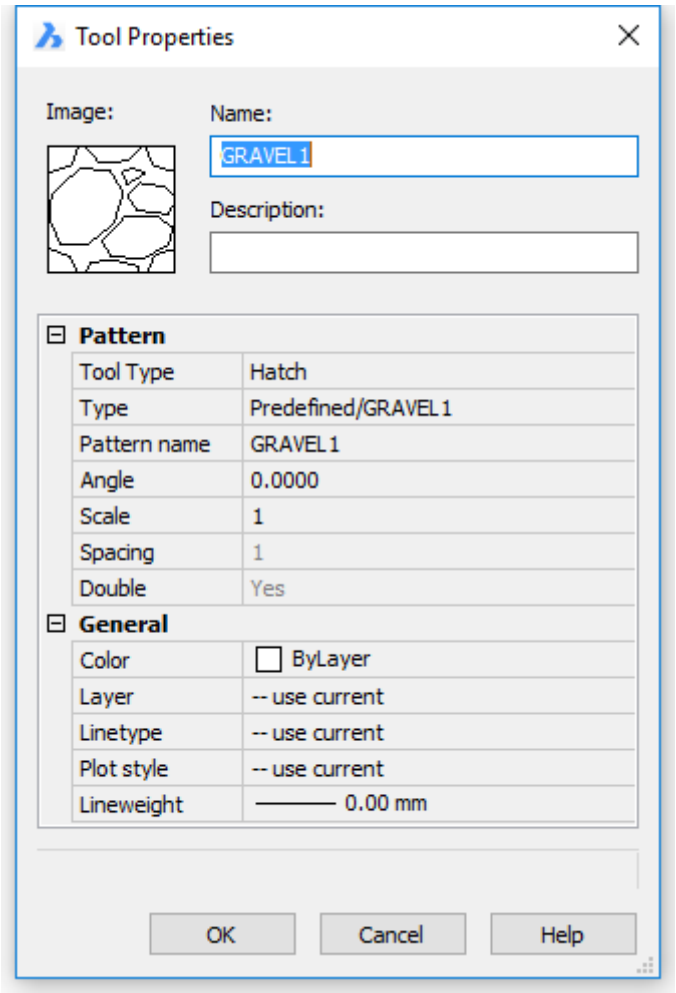
4. (option) Drag-and-drop commands to change their order on the palette.

NOTE See the topic Creating Custom Tools to create your own tools, which you can then add to a tool palette.

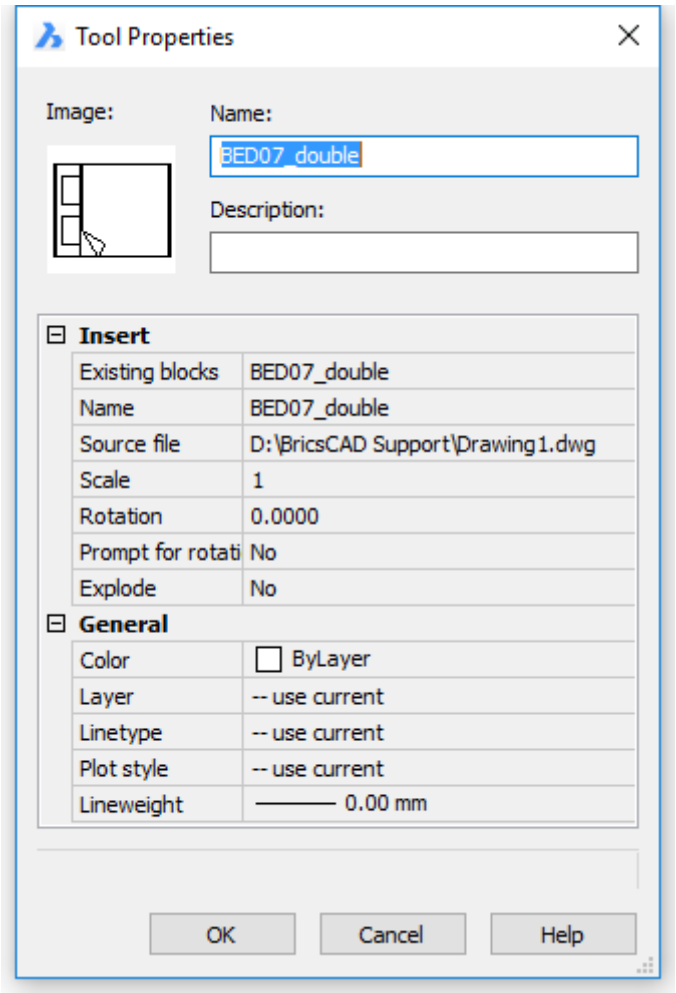
Editing a Tool

General procedure to edit a tool on a tool palette:

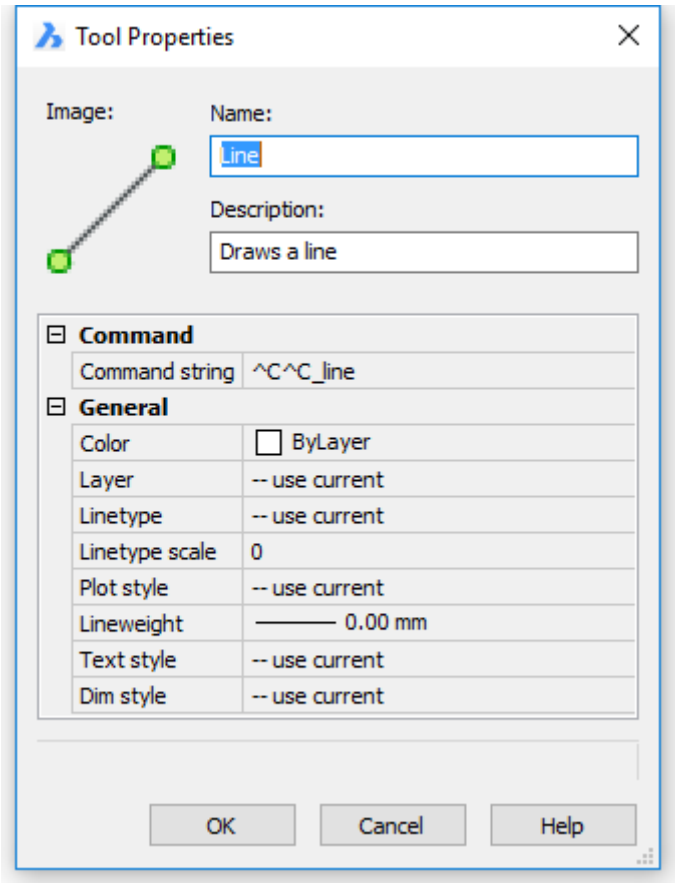
1. Right click the tool, then choose *Properties* in the context menu.
   The *Tool Properties* dialog box displays.



Hatch tool properties



Block tool properties



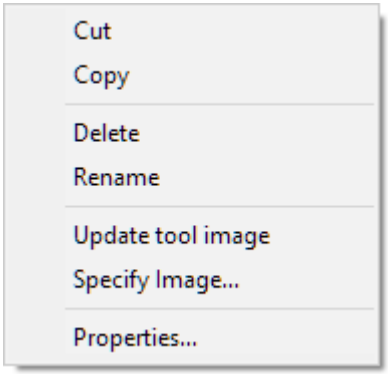
Command tool properties

2. Edit the tool properties.
3. Click the *OK* button to confirm.

Managing tools

To manage the tools on a tool palette do the following:

1. Right click the tool.
   A context menu displays:



2. Choose an option in the context menu.

To Specify a Tool Image

1. Right click the tool and choose *Specify Image* in the context menu.
   A file dialog box displays.
2. Select the image file, then click the *Open* button.

NOTE To restore the original tool image: Right click the tool and choose *Remove Image* in the context menu.

To Move a Tool

Drag-and-drop tools to re-arrange tools on a palette: drag the tool to the desired location and drop the tool on top of the existing tool.

To move a tool from one palette to another, do the following:

1. Right click the tool on the source palette and choose *Cut* in the context menu.
2. Select the target palette.
3. Right click and choose *Paste* in the context menu.
   The tool is added at the bottom of the palette.

To Copy a Tool

To copy a tool from one palette to another, do the following:

1. Right click the tool on the source palette and choose *Copy* in the context menu.
2. Select the target palette.
3. Right click and choose *Paste* in the context menu.
   The tool is added at the bottom of the palette.

Command Aliases

Command aliases are alternative names for BricsCAD commands. You can launch the LINE command by entering L at the command prompt. There is no need to type the complete name. After typing the command alias at the command prompt the command is executed. Some commands have more than one alias: e.g. REC, RECT and RECTANGLE are all aliases for RECTANG, whereas the last alias is even longer than the command name. In general, command aliases can increase your speed in entering the commands at keyboard.

Command aliases are saved in a *.pgp file (**P**ro**G**ram **P**arameters file). By default the *default.pgp* file in the *Support* folder of the Roamable Root folder is used.

If a command can be executed transparently (= while another command is being executed), the alias can also be used transparently. To execute a command or an alias transparently, precede the command name or alias with an apostrophe (').

NOTES

- Unlike commands, aliases are only local and cannot be used as global (preceded by a underscore (_).
- Command aliases cannot be used in scripts (*.scr).

To Edit the Alias File in a Text Editor

In order to modify existing aliases or to create new ones, the *default.pgp* file can be edited using an ASCII text editor (e.g. Notepad).

The first section of the *default.pgp* file defines the *Shell* commands. The second section of the *default.pgp* file defines command aliases.

The following syntax is used to define a command alias:

alias, **command*

alias = the character string to be entered at the command prompt. The command alias must be followed by a comma (,).

command = the BricsCAD command. The command name must be preceded by an asterisk (*).

Comment lines must be preceded by a semicolon (;). Comment lines can be used to add additional information, e.g. when the file was last edited and by whom.

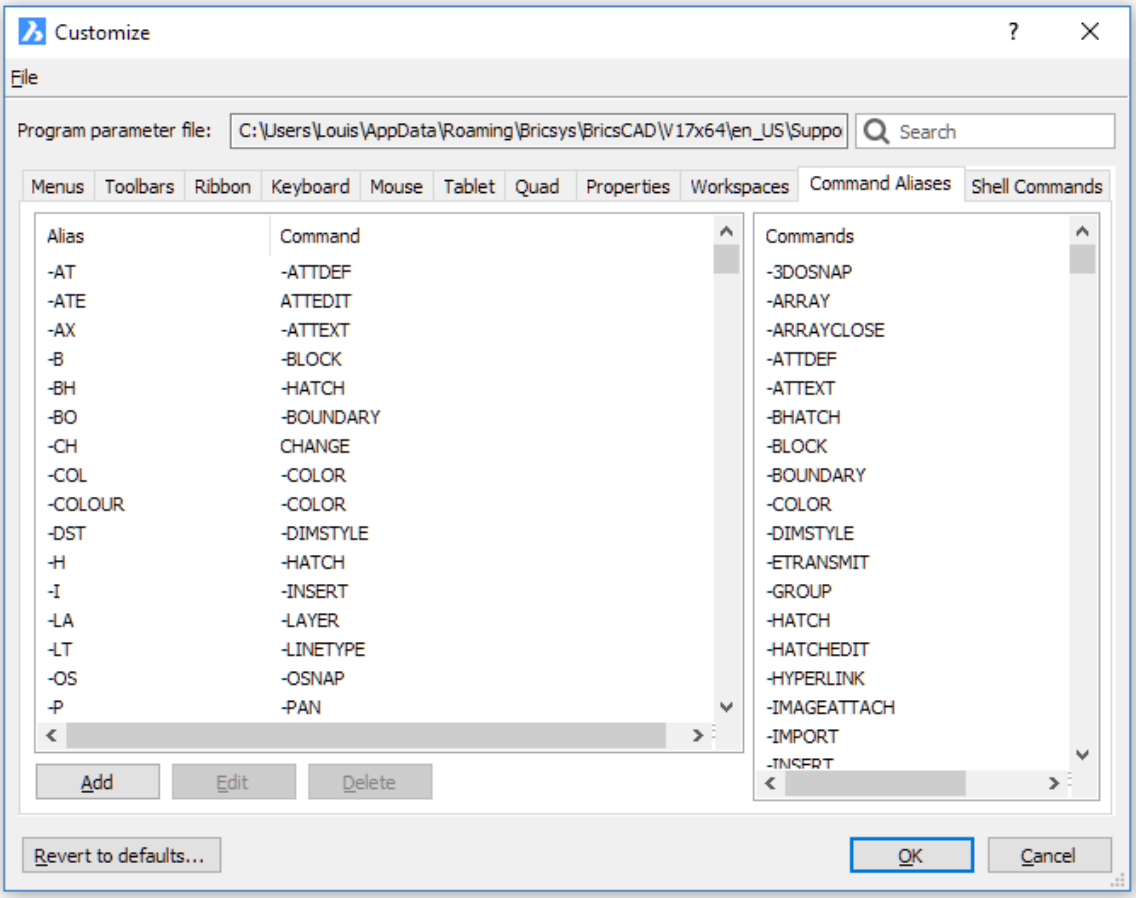
Some commands, such as HATCH or BOUNDARY, can be preceded by a hyphen (-) to access the command line version of the command instead of the dialog box version. Command aliases can be created for these commands also, e.g.:

vi, **-view*

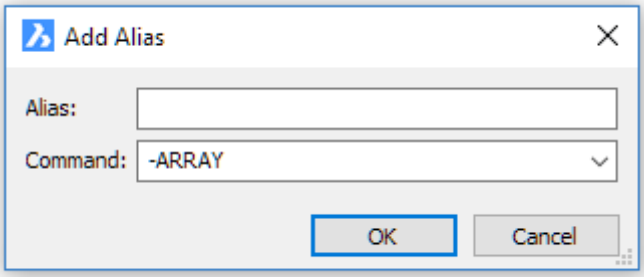
Execute the REINIT command to reload the alias file when it was externally edited.

Using the Customize Dialog to Edit the Alias File

1. Open the Customize dialog.
2. Click the *Aliases* tab on the *Customize* dialog.



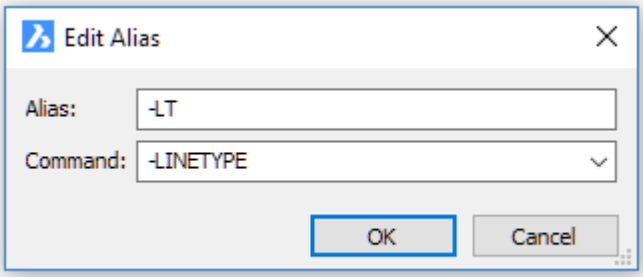
3. (option) To create a new alias:
   - Click the *Add* button.
     The *Add Alias* dialog box displays:



   - Click the down arrow at the right hand side of the *Command* field, then select a command.
     You can start typing the command name in the *Command* field to easily find the command in the list.
   - Type the command alias for the command in the *Alias* field.
   - Click the *OK* button.

4. (option) To edit an existing alias:

- Select the alias in the list.
- Click the *Edit* button.
  The *Edit Alias* dialog box displays:



- Type a new alias in the Alias field.
  If the alias already exists you are prompted to overwrite the existing alias.
- Click the *OK* button.

5. (option) To delete an alias:

- Select the alias in the list.
- Click the *Delete* button.
  A *Confirm* box displays.
- Click the *OK* button on the Confirm box to delete the alias.

6. Click the *OK* button on the *Customize* dialog to stop.

NOTE If you use the *Customize* dialog to edit the alias file, it is not necessary to execute the REINIT command.

User Profile Manager

Content of a User Profile

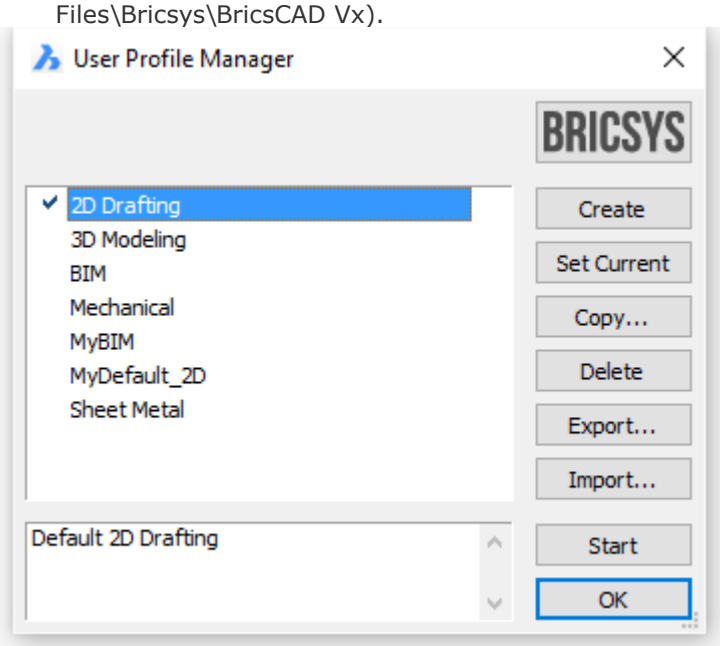
User profiles save many settings that control the drawing environment. Once you start BricsCAD using a user profile, it automatically tracks and stores changes that you make to your drawing environment.

- Main Cui file.
- Current Workspace: BricsCAD has five stanard workspaces: 2D Drafting, 3D Modeling, Mechanical, Sheet Metal and BIM.
- Configuration settings: all system variables which are saved in the Registry ( ) and user preferences.
- Default plot settings.
- Settings in dialog windows, e.g. search options in the *Settings* dialog.
- Project settings
- Recent paths
- Status bar settings
- Properties Bar settings
- Toolbar and menu configuration
- Worksets.

Launch the User Profile Manager

Do one of the following:

- When BricsCAD is running: choose *User Profile Manager* in the *Tools* menu.
- From the *Windows Start menu*, choose *All Programs > Bricsys > BricsCAD > User Profile Manager*.
- Double click *userprofilemanager.exe* in the BricsCAD program folder (e.g. C:\Program Files\Bricsys\BricsCAD Vx).



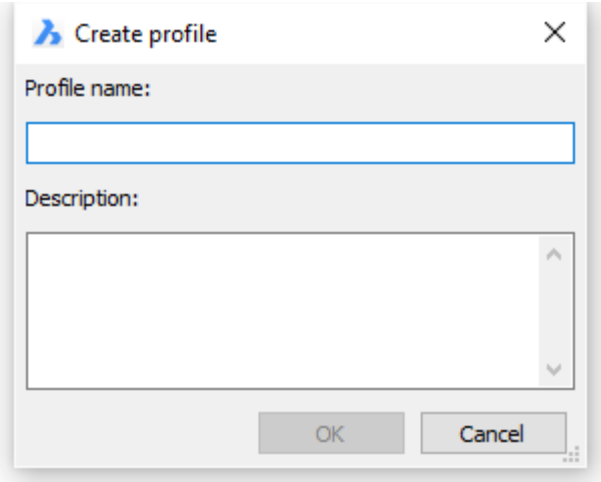
In the *User Profile Manager* dialog:

- The *Current Profile* is marked.
- *Create*: Creates a new profile, using default settings.
- *Set Current*: Sets the selected profile as the current profile, which will be used the next time you start BricsCAD.
- *Copy*: Click to make a copy of the selected profile.
- *Delete*: Click to delete the selected profile.
- *Export...* : Click to save the selected profile to a file on your computer, disk, or network. The file is saved with an *.arg* extension.
- *Import...* : Click to open a profile that has been saved to *.arg* file).
- *Start*: Starts BricsCAD with the selected user profile.
- OK: Closes the *User Profile Manager* dialog
- Double click a profile in the list of the available profiles to start BricsCAD using this profile.

Creating User Profiles

To create a new user profile:

1. Launch the User Profile Manager.
2. Click the *Create* button on the *User Profile Manager* dialog window.
   The *Create Profile* dialog opens.



3. Type a name for the new profile in the *Profile name* field.
4. (option) Type a description for the new profile in the *Description* field.
5. Click the *OK* button to create the profile.
   The new profile is added in the available profiles list.
   The new profile is created using default hard-coded values for all settings.

NOTE Use the Copy option to create a new user profile as a copy of an existing profile.

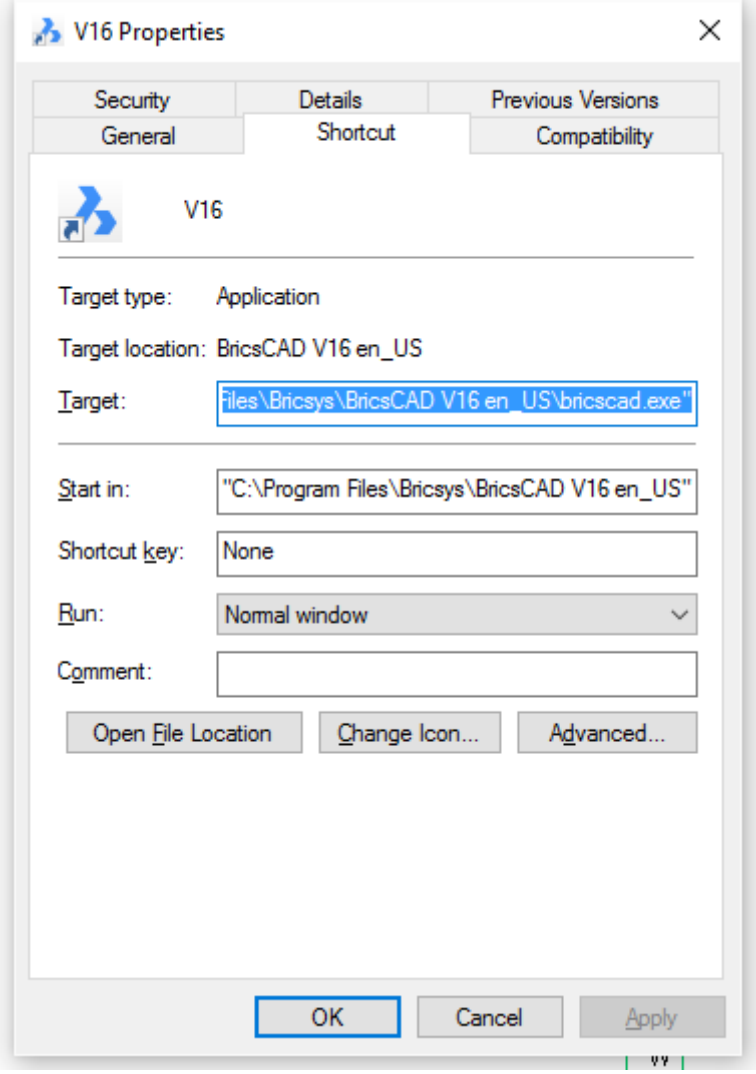
Loading a User Profile

When you start BricsCAD, the user profile that was current when you closed the previous BricsCAD session is used.

If the GETSTARTED system variable is ON, the *GET STARTED* page of the Welcome dialog box displays on startup. Here you can select the user profile you want to use.

To create a desktop shortcut to launch BricsCAD using a specific user profile, do one of the following:

- Add the */P option* to the *Target* settings of the BricsCAD shortcut.
  To open the shortcut settings dialog: right click the shortcut icon on the desktop and choose *Properties* in the context menu.
  A typical BricsCAD shortcut could be *"C:\Program Files\Bricsys\BricsCAD Vx\bricscad.exe" /P <UserProfileName>*



- Launch the User Profile Manager, then either double click a user profile or select a user profile and click the *Start* button.

NOTE

- The */P* option is case insensitive.
- If you load a user profile when BricsCAD is already running a second instance of BricsCAD is launched, at least on condition that the *SINGLETON* variable is OFF. This variable controls whether multiple BricsCAD sessions can run simultaneously. See the Settings dialog for more information about variables and settings.
- If the user profile in the argument of the */P* option does not exist, the user profile is created automatically using default hard-coded values for all settings.
- If the user profile name contains spaces, use double quotes: e.g. /P "background white".

Restarting BricsCAD Using a Different User Profile

1. Choose *User Profile Manager* in the *Tools* menu.
2. Select the user profile you want to start up with.
3. Click the *Set Current* button on the *User Profile Manager* dialog.
4. Close and restart BricsCAD.

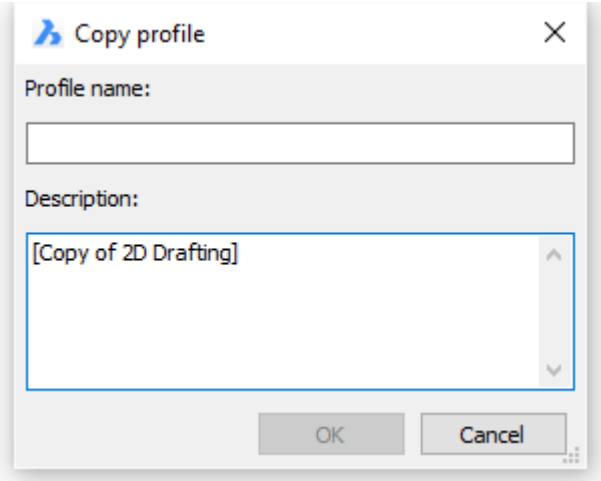
NOTE If the GETSTARTED system variable is ON, the *GET STARTED* page of the Welcome dialog box displays on startup. Here you can select the user profile you want to use.

Managing User Profiles

Once you start using user profiles, you may need to copy, rename or delete them. Copying a profile is a quick way to create a new profile based on an existing profile.

Copy a User Profile

1. Launch the User Profile Manager.
2. Select the user profile you want to copy.
3. Click the *Copy* button.
   The *Copy Profile* dialog displays.



4. Type a name for the new profile in the *Profile name* field.
5. (option) Type a description for the new profile in the *Description* field.
6. Click the *OK* button to create the profile.

Rename a User Profile

1. Launch the User Profile Manager.
2. Select the user profile you want to rename.
3. Click the name of the selected user profile.
   The *Profile name* field is active now.

4. Type a new name in the *Profile name* field.
5. Click outside the *Profile name* field.

Delete a User Profile

1. Launch the User Profile Manager.
2. Select the user profile you want to delete.
3. Click the *Delete* button.
4. Click the *Yes* button on the confirmation box to delete the profile.

NOTE The *current profile* cannot be deleted.

Export a User Profile

1. Launch the User Profile Manager.
2. Select the profile you want to export in the profile list.
3. Click the *Export* button.
   The *Save As* dialog displays.
4. Select a folder to save the user profile in.
5. (option) Edit the name of the profile in the *File name* field.
6. Click the *Save* button.
   The file is saved with an **.arg* extension.

Edit the User Profile Description

To edit the *Description* field:

1. Select the profile you want to edit the description of.
2. Edit the *Description* field.

Import a User Profile

1. Launch the User Profile Manager.
2. Click the *Import* button.
   The *Open* dialog displays.
3. Select a user profile file (*.arg).
4. Click the *Open* button.
   The *Import Profile* dialog displays.
5. (option) Type a different name in the *Profile name* field.
6. (option) Edit the profile description.
7. Click the OK button on the *Import Profile* dialog.
   The profile is added to the profile list.

Import a Profile from Another User

When you import a user profile of another user all user dependent search paths (e. g. the *local root* and *roamable root* folders - see BricsCAD User Files) still refer to the other user, as a result the folders cannot be found and error messages such as "Unable to open program parameters file: default.pgp" might pop up when you start BricsCAD.

To correct this problem do the following:

1. Open the exported user profile (*.arg) in a text editor such as Notepad.
2. Replace the user name dependent references with an environment variable.

On a Windows XP computer replace *C:\\Documents and Settings\\username* with *%USERPROFILE%.*

On a Windows Vista or Windows 7 computer replace *C:\\Users\\username* with *%USERPROFILE%.*

Use the *Find/Replace* procedure to quickly replace all references.

3. Save the user profile (*.arg)
4. Import the user profile.
5. Start BricsCAD using the imported profile.
   The environment variable *%USERPROFILE%* will be replaced with:
   C:\\Users\\currentusername.

NOTE Environment variables in profiles, using the %<variable_name>% format, are supported in V10.3.11. and higher.

Workspaces

Definition

The *Workspace* command sets the current workspace and lets you create, modify and save workspaces.

The *WsSave* command saves the current workspace under a different name, thus creating a new workspace.

Workspaces define settings and preferences regarding a specific working environment.

The following workspaces are available:

- 2D Drafting
- 3D Modeling
- Mechanical
- Sheet Metal
- BIM (Building Information Model)

The name of the workspace is stored in the WSCURRENT system variable of the current User Profile.

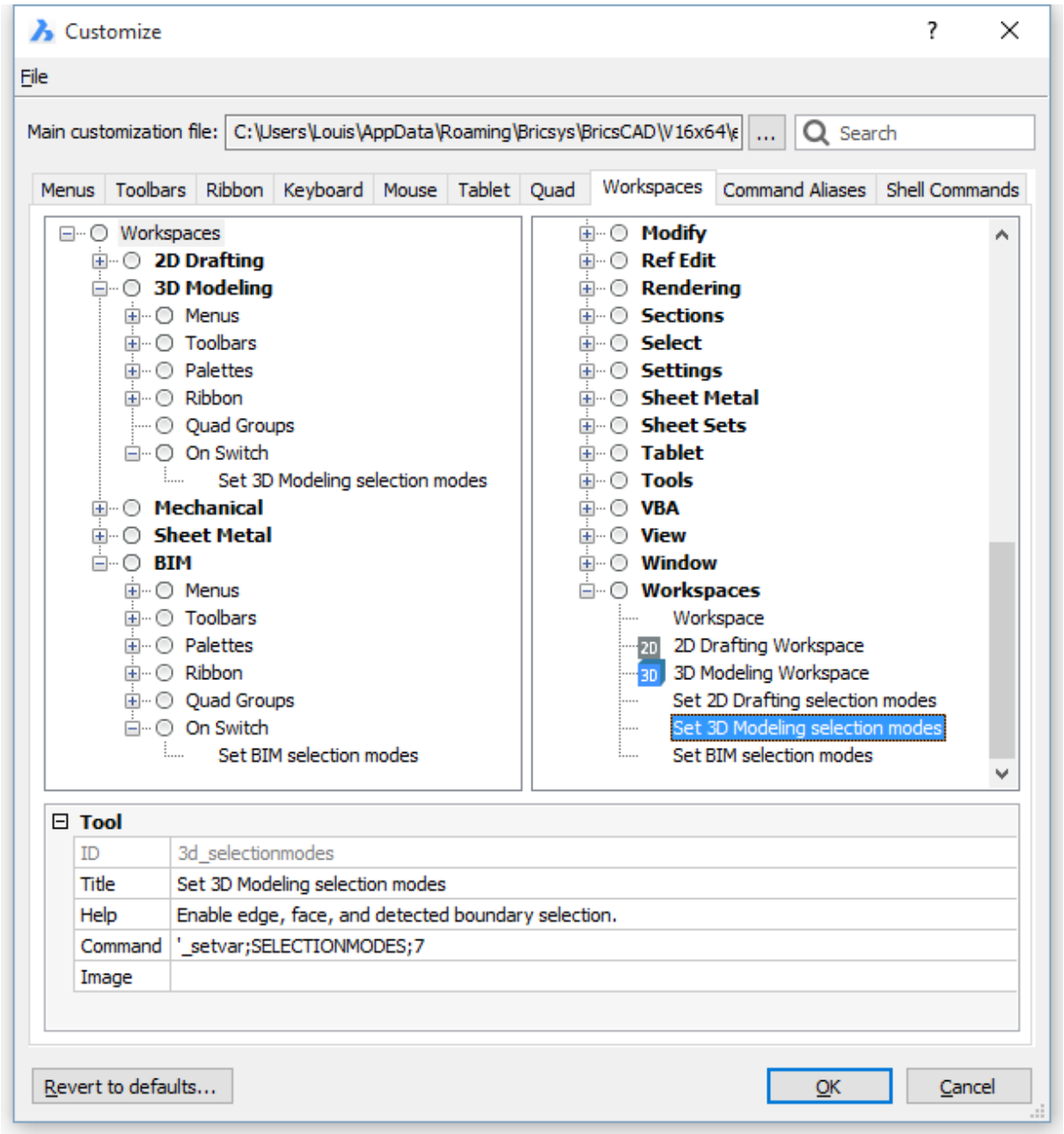
If the GETSTARTED system variable is ON, the *GET STARTED* page of the Welcome dialog box displays on startup. Here you can select the user profile you want to use.

Managing Workspaces

Workspaces are defined in the *Workspace* tab page on the *Customize* dialog.

Do one of the following:

- Open the Customize dialog, then select the *Workspace* tab.
- Right click the *Current Workspace* field in the Status Bar, then choose *Customize Workspaces...* in the context menu.



A workspace definition is composed of:

- **Menus**: defines which menus are available.
- **Toolbars**: defines which toolbars display by default.
- **Palettes**: controls the visibility of various palettes.

  Palettes
  - Command Line
  - Material Browser
  - Mechanical Browser
  - Properties
  - Ribbon
  - Tool Palettes

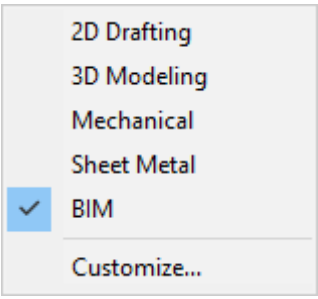
  Right click a palette then choose an option in the context menu:
  - Show:
  - Hide:
  - Don't change:

- **Ribbon**: defines the available ribbon tabs.
- **Quad Group Order**: defines the order and availability of the Quad command groups.
- **On Switch**: defines the system variables and user preferences to be set and the commands to be executed when switching to the workspace.

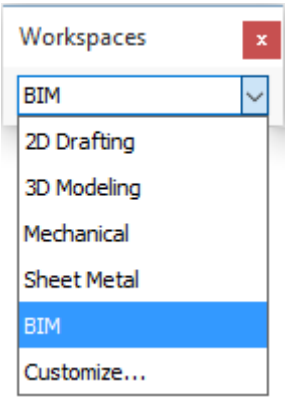
Setting the Current Workspace

Do one of the following:

- Right click the *Current Workspace* field in the Status Bar, then select a workspace in the context menu.



- Choose a workspace in the *Workspaces* toolbar.



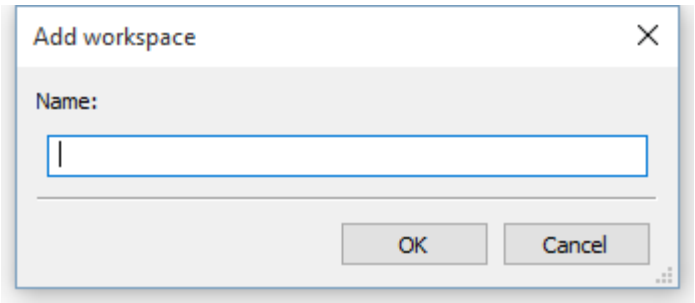
- Type *wscurrent* at the command prompt.
  You are prompted: New value for WSCURRENT, or . for none/<"2d DRAFTING">:
  Type the name of the new current workspace, then press enter.

NOTES

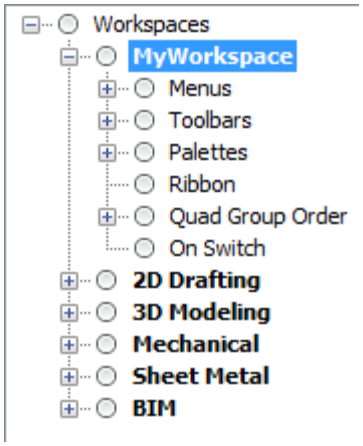
- If the WSAUTOSAVE system variable is On, upon switching to another workspace, the toolbar positions of the current workspace are saved in the CUI file. The default value for WSAUTOSAVE is ON.
- It is not recommended to alter the workspace if the name of the user profile refers to a workspace.

Creating a Workspace

1. Right click the *Current Workspace* field in the Status Bar, then choose *Customize Workspaces...* in the context menu.
2. Do one of the following:
   - Select *Workspaces*, then right click and choose *Append Workspace* in the context menu.
   - Select an existing workspace, then right click and choose *Insert Workspace* in the context menu.
3. Type a name for the new workspace, then click the *OK* button.



When appending a workspace, the new workspace is created below the last existing workspace. When inserting a workspace, the new workspace is created above the selected workspace.



A newly created workspace contains the following items:

- A standard menu set
- A standard toolbar set
- All palettes set to *Don't change*
- An empty ribbon
- A default Quad group order
- An empty *On Switch* sequence

Editing a Workspace

1. Right click the *Current Workspace* field in the Status Bar, then choose *Customize...* in the context menu.
2. Click the *Expand* button (+) of the workspace.
3. Edit the menu set:
   - Change the order: Drag a menu to another location.
   - Delete a menu: select the menu, then right click and choose *Remove* in the context menu.
   - Insert a menu:
     - Right click a menu in the list and choose *Insert menu* in the context menu. The *File* menu is inserted by default.
     - In the *Workspace Menu* grid, select the *ID* field and select the desired menu in the list.

4. Edit the toolbar set:

   - Change the order: Drag a toolbar to another location.
   - Delete a toolbar: select the toolbar, then right click and choose *Remove* in the context menu.
   - Insert a toolbar:
     - Right click a toolbar in the list, and choose *Insert toolbar* in the context menu.
       The *Standard* toolbar is inserted by default.
     - In the *Workspace Toolbar* grid, select the *ID* field and select the desired toolbar in the list.

5. Edit the visibility of the palettes:

   - Select the palette.
   - Click the *Display* property in the *Workspace Palette* grid, then set the desired 'on switch' behavior: *Show*, *Hide* or *Don't change*.

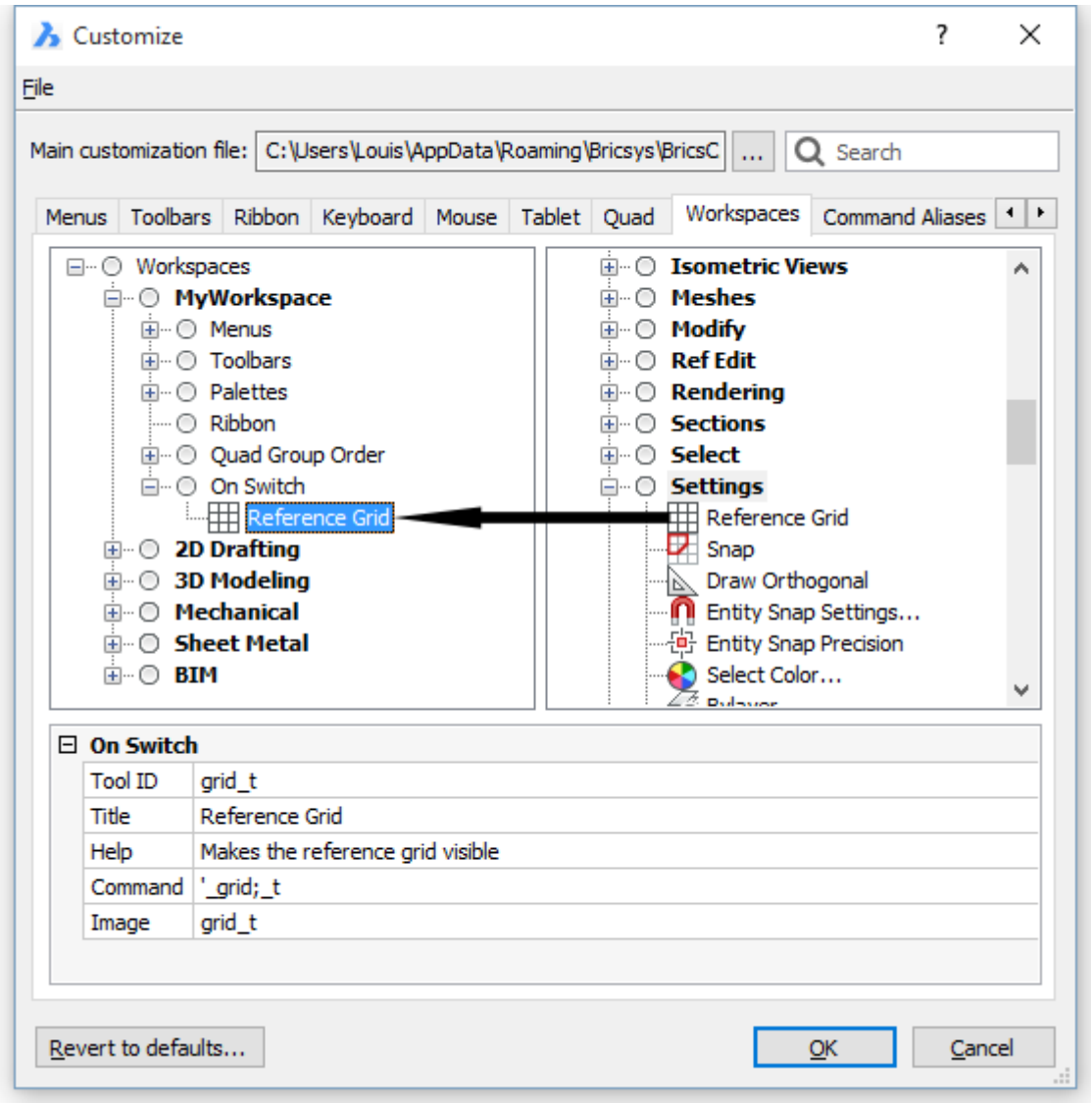
6. Edit the composition of the ribbon:

   - To add a tab: right click an existing tab, then choose *Insert tab* in the context menu and select the tab in the *Select Ribbon Tab* dialog.
   - To remove a tab: right click the tab and choose *Remove* in the context menu.
   - To change the position of a tab: press and hold the left mouse button, then drag the tab to the desired place.

7. Edit the Quad settings:

   - To change the order of the Quad command groups: press and hold the left mouse button, then drag a command group to its new position.
   - Right click a Quad command group, then choose *Switch off* in the context menu to remove the command group from the Quad cursor menu.
   - Right click a currently switched off Quad command group then choose *Switch on* in the context menu to add the command group to the Quad cursor menu.

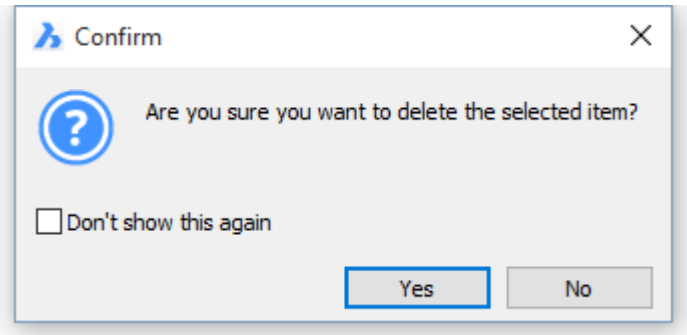
8. Edit the *On Switch* procedure:
   To add a command to the *On Switch* sequence: select the command in the available commands pane and drag it to the *On Switch* node.



NOTE See To create a new tool if you want to add a custom tool to the *On Switch* sequence.

Deleting a Workspace

1. Right click the *Current Workspace* field in the Status Bar, then choose *Customize Workspaces...* in the context menu.
2. Right click the workspace, then choose Delete in the context menu.
   A Confirm dialog box displays:

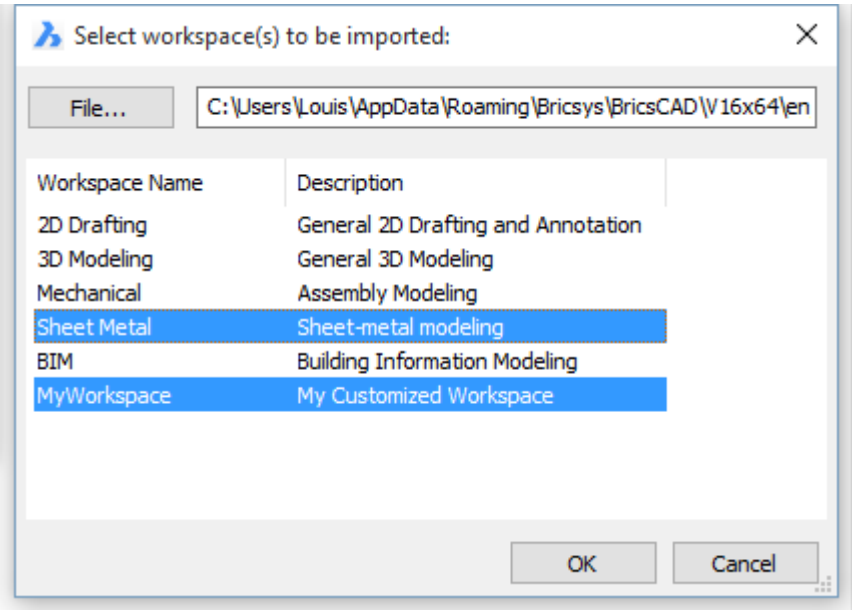


3. Press the *Yes* button to delete the selected workspace.

Importing a Workspace

The option in the *File* menu on the *Customize* dialog box can import workspaces from another CUI file into the current main CUI.

1. Open the Customize dialog.
2. In the *File* menu choose *Import Workspace(s)...*
   The *Choose a Customization File* dialog displays.
3. Browse to the folder where the CUI file, you want to import the workspace from, is saved.
   By default BricsCAD saves CUI files in the *Support* folder of the Roamable Root folder.
   e.g. *C:\Users\<user_name>\AppData\Roaming\Bricsys\Bricscad\Vxx\en_US\Support*
4. Select the CUI file.
   A dialog box, listing all workspaces in the selected CUI file, displays:



5. Select the workspace to import in the list, then click the *OK* button.
   Press and hold the Ctrl-button to select multiple workspaces.

Managing Drawings

Creating a Drawing

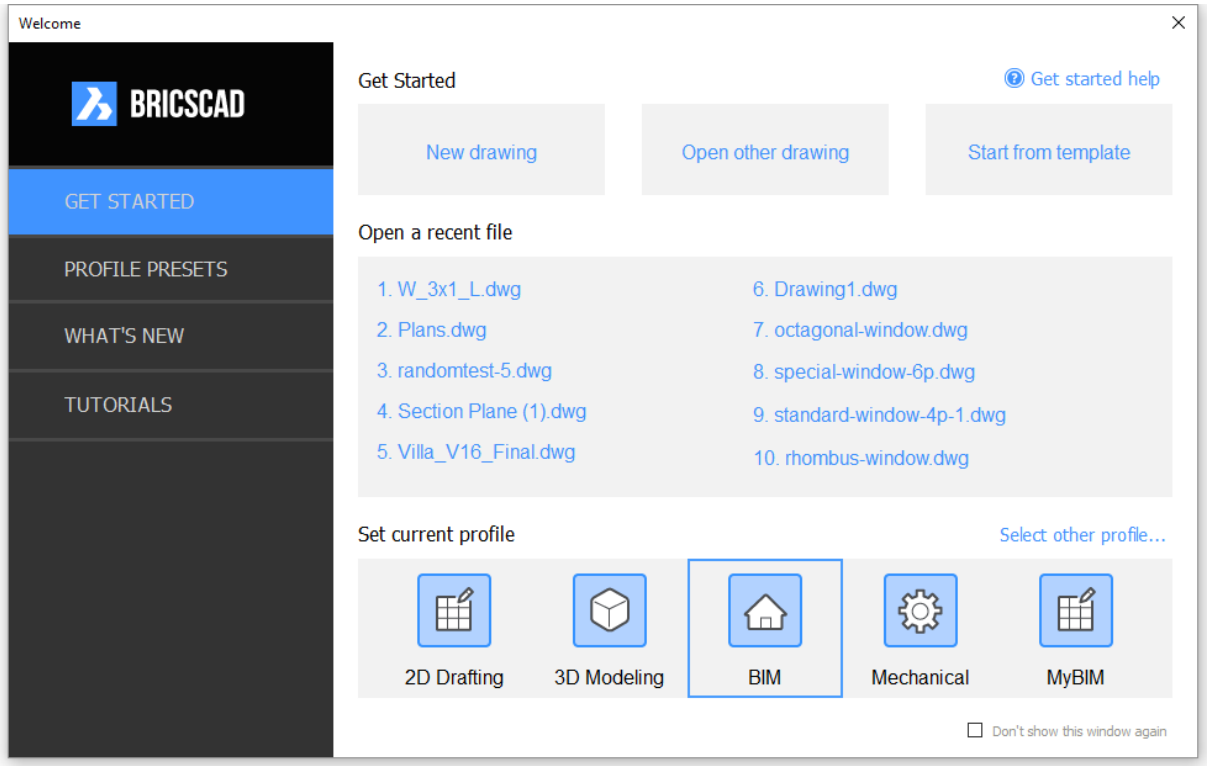
In BricsCAD you can create new drawings:

- Using a default template drawing.
- Starting from scratch.
- Using a template drawing of your choice.
- Using a wizard.
- Double clicking a **.dwt* file in a *Windows Explorer* window.

Using the Welcome Dialog Box

When you launch BricsCAD and the value of the GETSTARTED system variable is ON, the *GET STARTED* page of the Welcome dialog box displays.

Check the *Don't show this window again* option to suppress the display of the *Welcome* dialog.



On the *GET STARTED* page, do one of the following:

- Select a profile in the *Set current profile* list, then click *New Drawing* to start BricsCAD with a new drawing using the default template of the selected profile.
  Default template drawings in the *Profile Presets* are:

Profile	**Template**	
	Imperial	**Metric**
2D	Default-imperial.dwt	Default-mm.dwt
3D	Default3D-imperial.dwt	Default3D-metric.dwt
Mechanical	Mechanical-imperial.dwt	Mechanical-metric.dwt
Sheet Metal	Mechanical-imperial.dwt	Mechanical-metric.dwt
BIM	BIM-imperial.dwt	BIM-metric.dwt

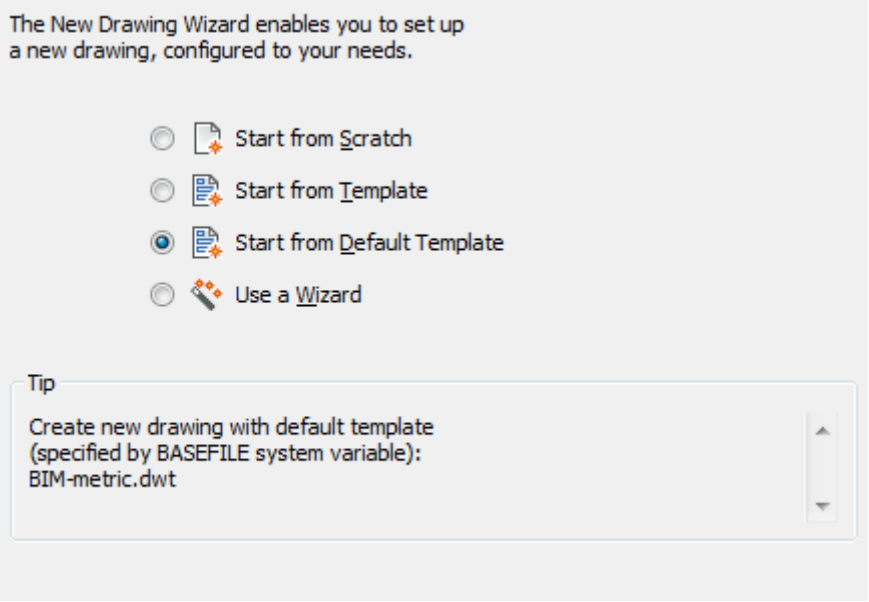
- Select a profile in the *Set current profile* list, then click Start from template to start BricsCAD with a new drawing using a template, different from the current default template of the selected profile.
  The selected template becomes the default template of the selected profile.

To create a drawing using a default template drawing

First method:

1. Do one of the following:
   - Choose *New wizard...* in the *File* menu.
   - Type *newwiz* in the command window, then press Enter.

2. On the *Create New Drawing* dialog choose *Start from Default template*, then click the *Next* button.
   The drawing is created as a copy of the default template.



Second method:

Do one of the following:

- Click the *QNew* tool button ( ) on the *Standard* toolbar.
- type *qnew* in the command bar, then press Enter.

The drawing is created as a copy of the default template defined by the *BASEFILE* system variable. If the default template drawing is not specified, the *Create New Drawing* wizard is launched if the *STARTUP* system variable is *ON*, otherwise the *Select Template* dialog displays.

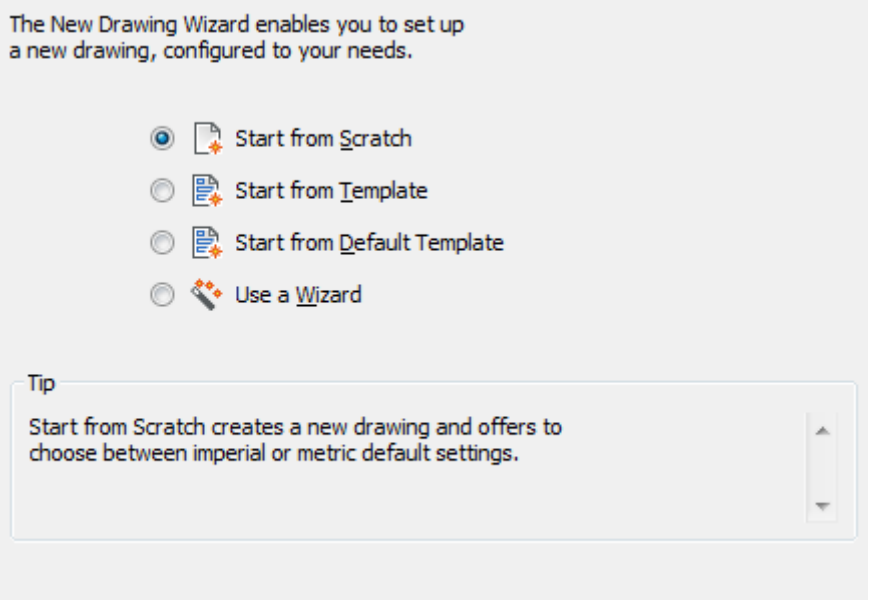
Setting the Default Template Drawing

1. Open the Settings dialog.
2. Under *Program Options / Files / Templates* select *Template (BASEFILE)*
3. Click the *Browse* button.
4. On the *Choose a File* dialog, select a drawing file (*.dwg) or drawing template (*.dwt), then click the *Open* button.

Creating a Drawing from Scratch

1. Do one of the following:
   - Choose *New wizard...* in the *File* menu.
   - Type *newwiz* in the command window, then press Enter.

2. On the *Create New Drawing* dialog choose *Start from Scratch*, then click the *Next* button.



3. Choose between either *Imperial* or *Metric* default settings.

   Click the *Finish* button to create the drawing.

NOTE Using the above procedure it depends on the status of the *BASEFILE* and *PSTYLEPOLICY* variables whether *CTB* (Color table) or *STB* (Named plot style table) will be used to set up plotter configurations in the newly created drawing.

BASEFILE	PSTYLEPOLICY	CTB or STB?
not specified	OFF	STB
not specified	ON	CTB
specified	ON or OFF	as defined in the template drawing

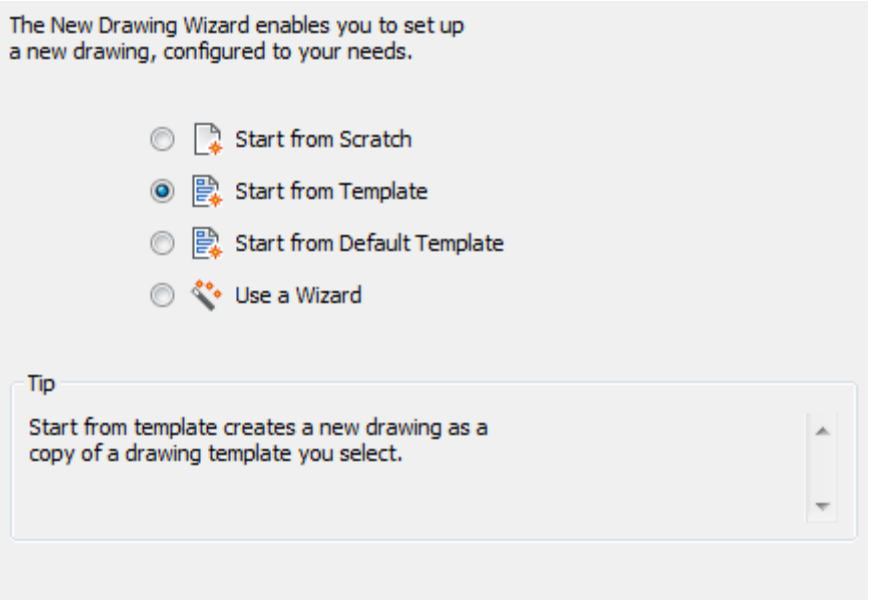
Setting the Plot Style Policy

1. Open the Settings dialog.
2. Under *Program Options / Plot and publish* select *Plot style policy (PSTYLEPOLICY)*.
3. Do one of the following:
   - Choose *Associate an entity's plot style with its color* for *Color dependent* plot styles.
   - Choose *No association between color and plot style* for *Named* plot styles.

Creating a Drawing Using a Template Drawing

1. Do one of the following:
   - Choose *New wizard...* in the *File* menu.
   - Type *newwiz* in the command window, then press Enter.

2. On the *Create New Drawing* dialog choose *Start from template*, then click the *Next* button.



3. Do one of the following:

   - Choose a template drawing in the *Select a template list*.
     By default this is the content of the *Template* folder in the Local root folder.
     Edit the *Template Path* variable to set your template folder differently.
   - Click the *Browse...* button to select a template drawing.
     You can choose between **.dwg* and **.dwt* file types.

The drawing is created as a copy of the selected template.

Creating a Drawing Using the New Command

1. Do one of the following:
   - Choose *New...* in the *File menu*.
   - Type *new* in the command bar, then press Enter.

The *Select template* dialog opens showing the content of the *Template* folder as defined in the *Template Path* variable.

2. Select the template drawing, then click the *Open* button.
The drawing is created as a copy of the selected template.

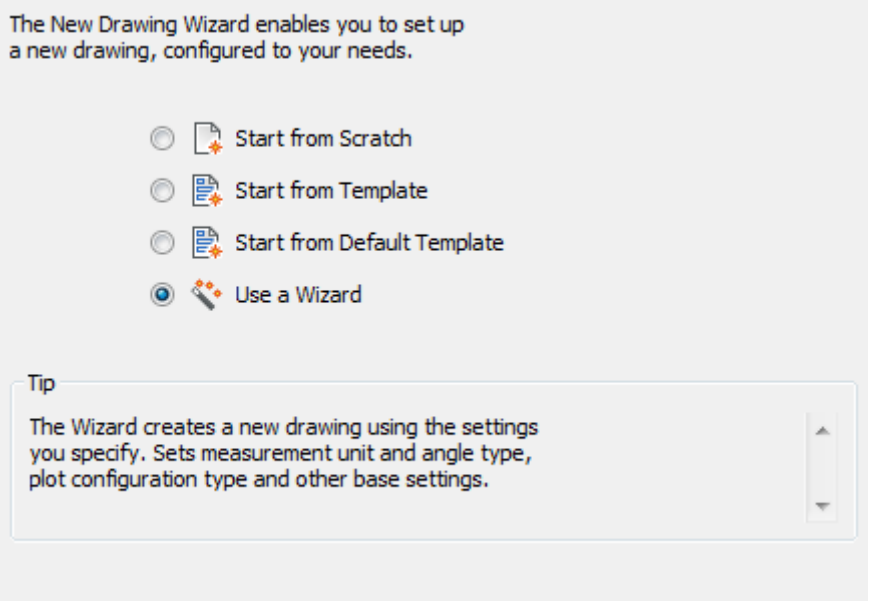
Setting the Template Path Variable

1. Open the Settings dialog.
2. Under *Program Options / Files / Templates* select *Template Path (templatePath)*
3. Click the *Browse* button.
4. On the *Browse for folder* dialog, select a folder , then click the *OK* button.

Creating a Drawing Using a Wizard

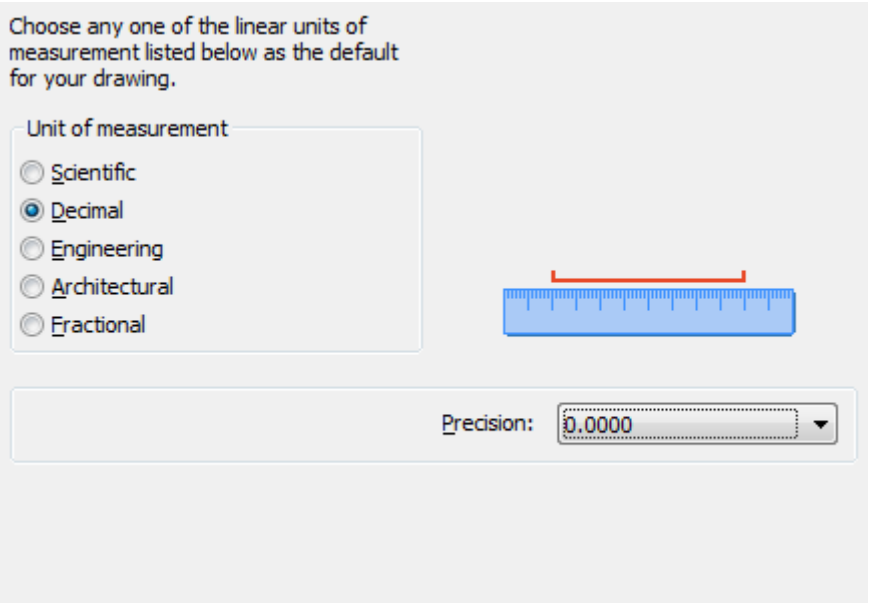
1. Do one of the following:
   - Choose *New wizard...* in the *File* menu.
   - Type *newwiz* in the command window, then press Enter.

2. On the *Create New Drawing* dialog choose *Use a Wizard*, then click the *Next* button.

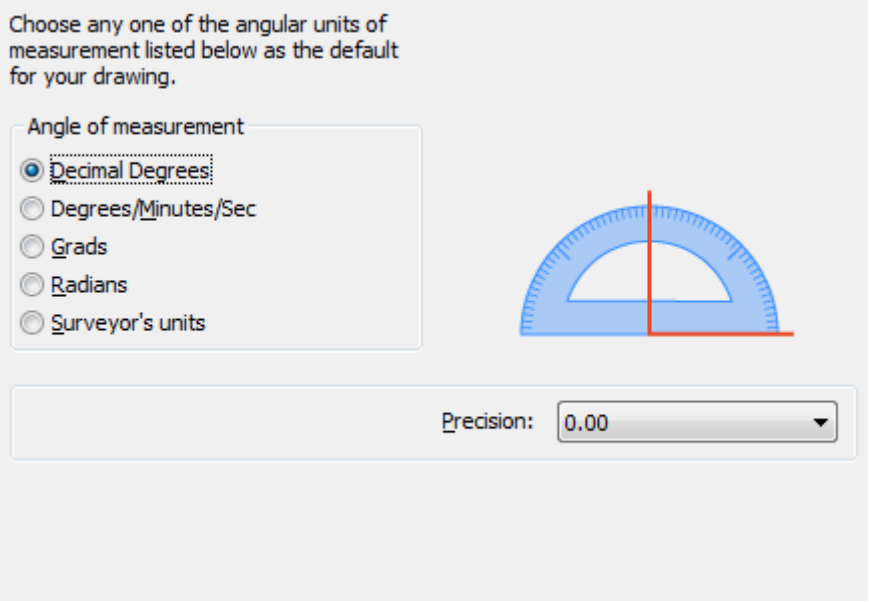


3. Choose between either *Imperial* or *Metric* default settings, then click the *Next* button.

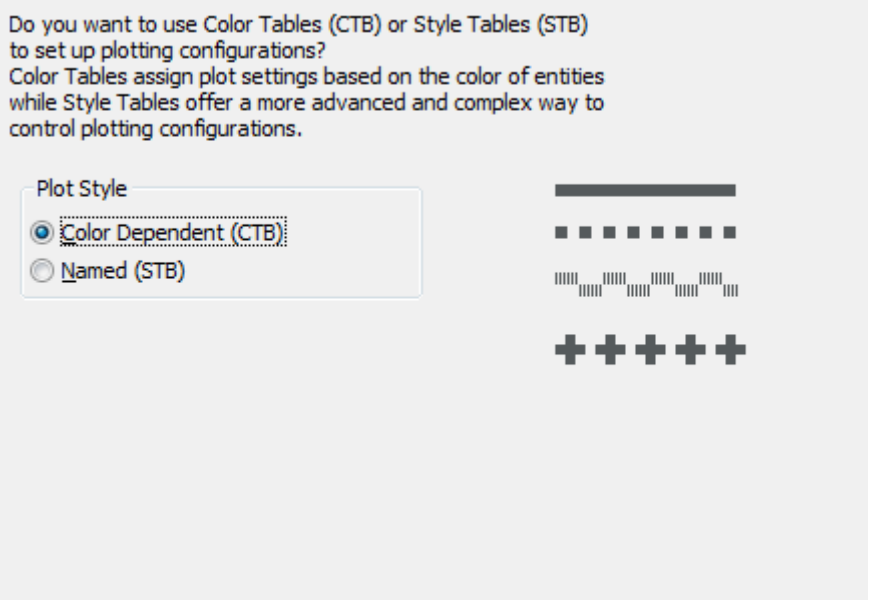
   Select the *Unit of measurement* and the display *Precision*, then click the *Next* button.



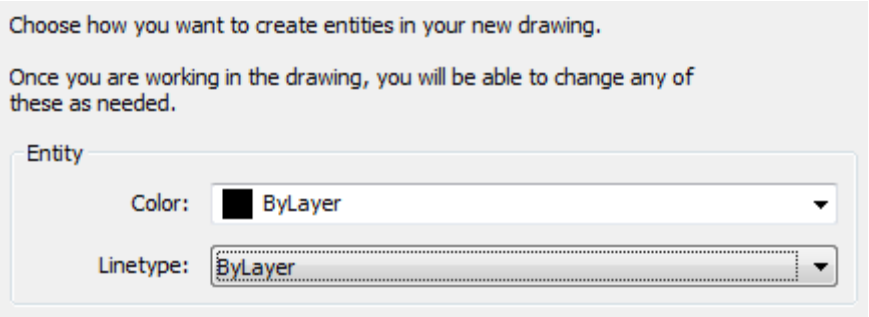
4. Select the *Angle of measurement* and the display *Precision*, then click the *Next* button.



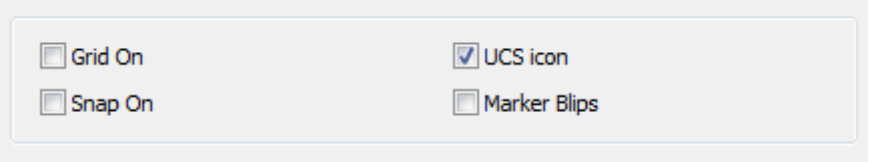
5. Choose between *Color Dependent* plot styles (CTB) or *Named* plot styles (STB), then click the *Next* button.



6. Select the default *Color* and *Linetype*.



7. Define the *Grid*, *Snap*, *UCS icon* and *Marker Blips* settings.



8. Click the *Finish* button to create the drawing.
9. (option) Choose *Save As...* in the *File* menu to save the drawing as a template in the *Templates* subfolder of the Local root folder.

Drawing Explorer

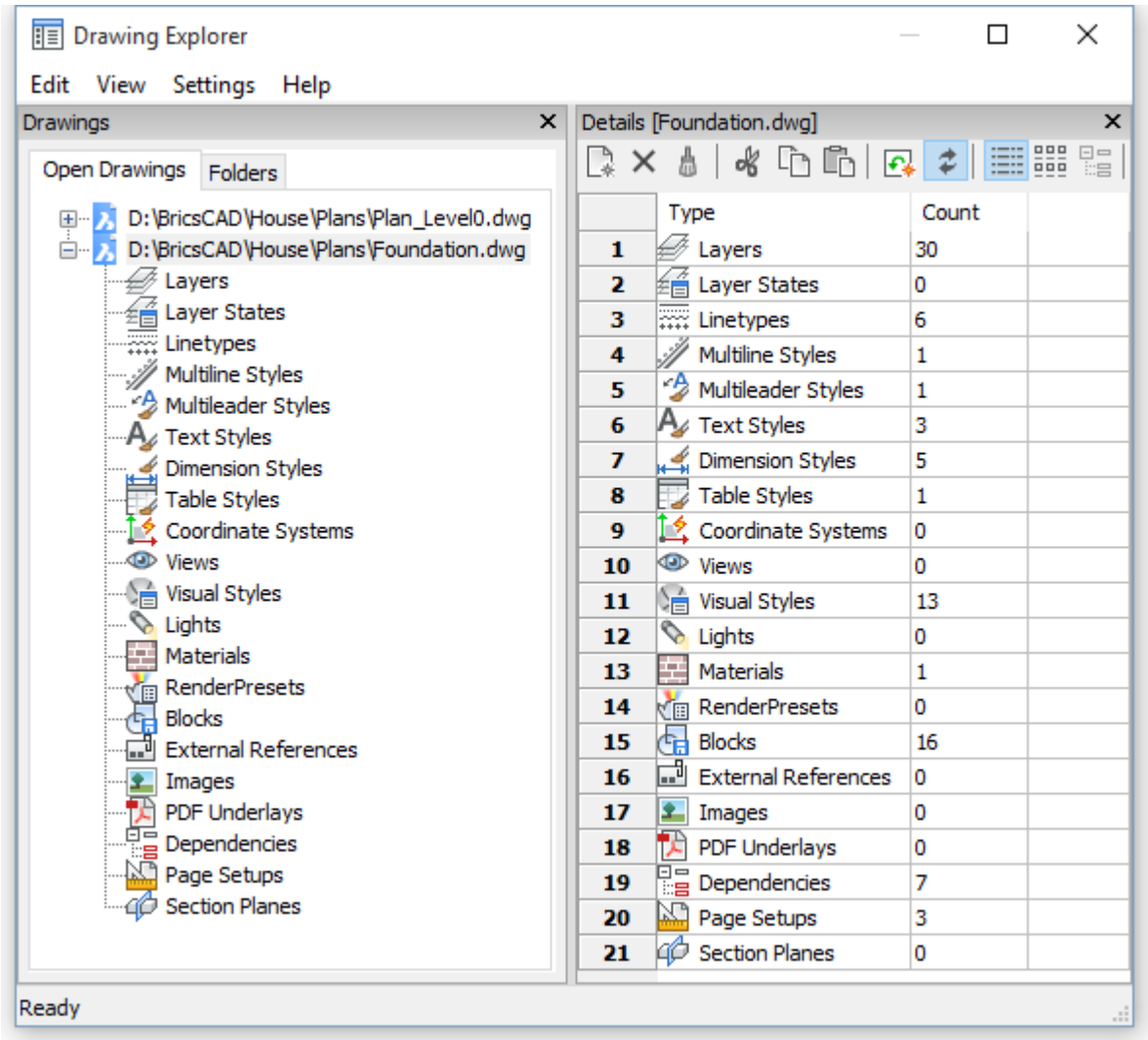
Commands: EXPLORER, LAYER, LAYERSTATE, LINETYPE, MLSTYLE, MLEADERSTYLE, STYLE, DIMSTYLE, TABLESTYLE, EXPUCS, VIEW, VISUALSTYLES, LIGHTLIST, MATERIALS, RENDERPRESETS, EXPBLOCKS, XREF, IMAGE, PDF, eTRANSMIT, PAGESETUP, SECTIONPLANESETTINGS

The *Drawing Explorer* window consists a number of sub-windows or panes:

- *Drawings*:
  - *Open Drawings*: a list of all drawings that are currently open.
  - *Folders*: your favorite drawing folders
- *Details*: the details of the selected drawing or the details of the selected category in a drawing, e.g. layers, blocks, images, ...
- *Preview*: a preview of the selected drawing, layer, block, view, dimension style, table style, XRef, image or material.
  Use the arrow keys to browse through the items.
- *Editor*: to define or edit the properties of *Multiline Styles*, *Multileader Styles*, *Table Styles*, *Views*, *Visual Styles*, *Sunlight settings*, *Materials*, *Render Presets* and *Section Planes*.

You can select which panes you want to have open or closed in the View menu.

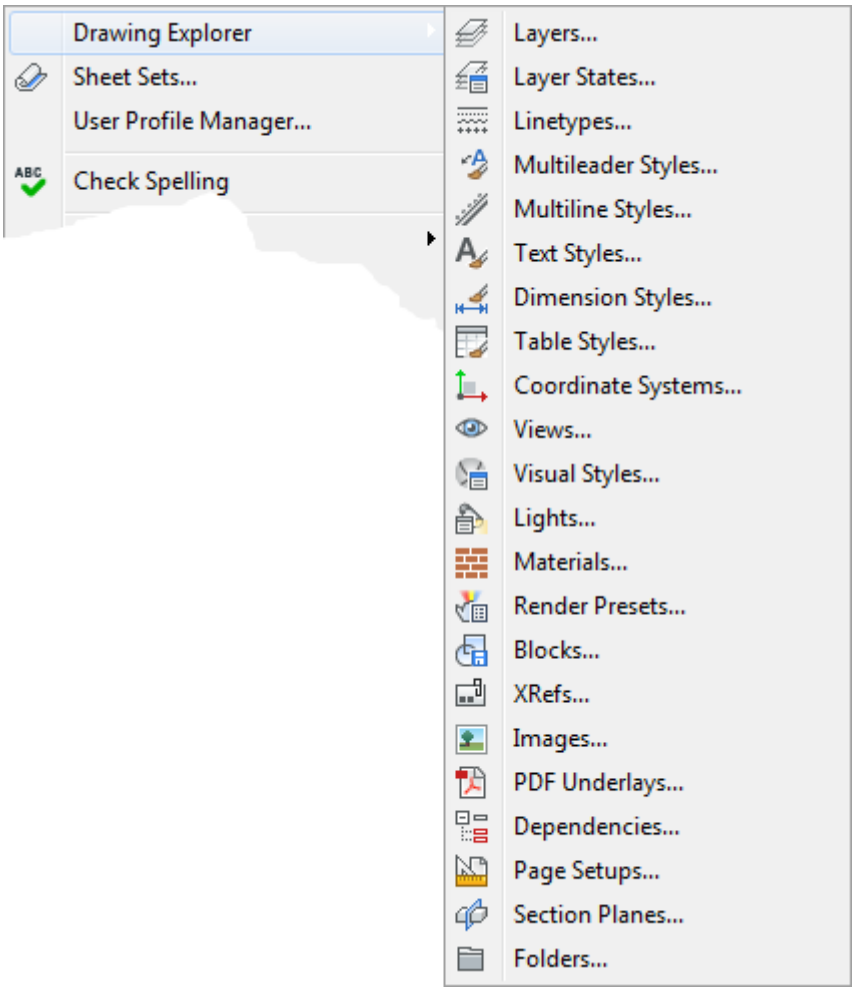
BricsCAD remembers the display settings for each category.



To Open the Drawing Explorer

To open the *Drawing Explorer* window do one of the following:

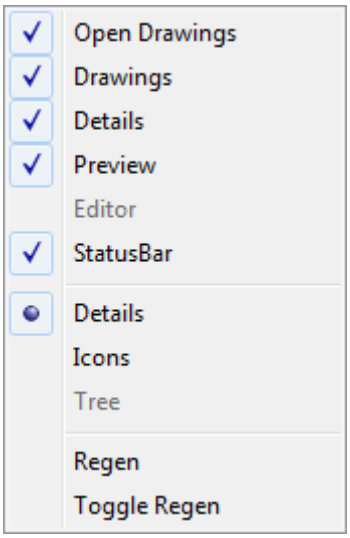
- Click the *Drawing Explorer* tool button ( ) on the *Standard* toolbar.
  The *Drawing Explorer* window opens showing the details of the most recently used category.
- Click the *Layer*, *Linetype*, *Text Style* or *Dimension Style* field in the *Status Bar*.
  The *Drawing Explorer* window opens showing the details of the clicked definitions category in the current drawing.
  (See Customizing the Status Bar to enable/disable fields in the *Status Bar*.)
- Choose a *Drawing Explorer* in the *Tools* menu, then select a category in the flyout.
  The *Drawing Explorer* window opens showing the details of the selected category.



- Type *explorer* in the command bar, then press Enter.
  The *Drawing Explorer* window opens showing the details of the most recently used category.

To Control the Drawing Explorer Layout

- In the *View* menu, check/uncheck the sub-window you want to open or close.

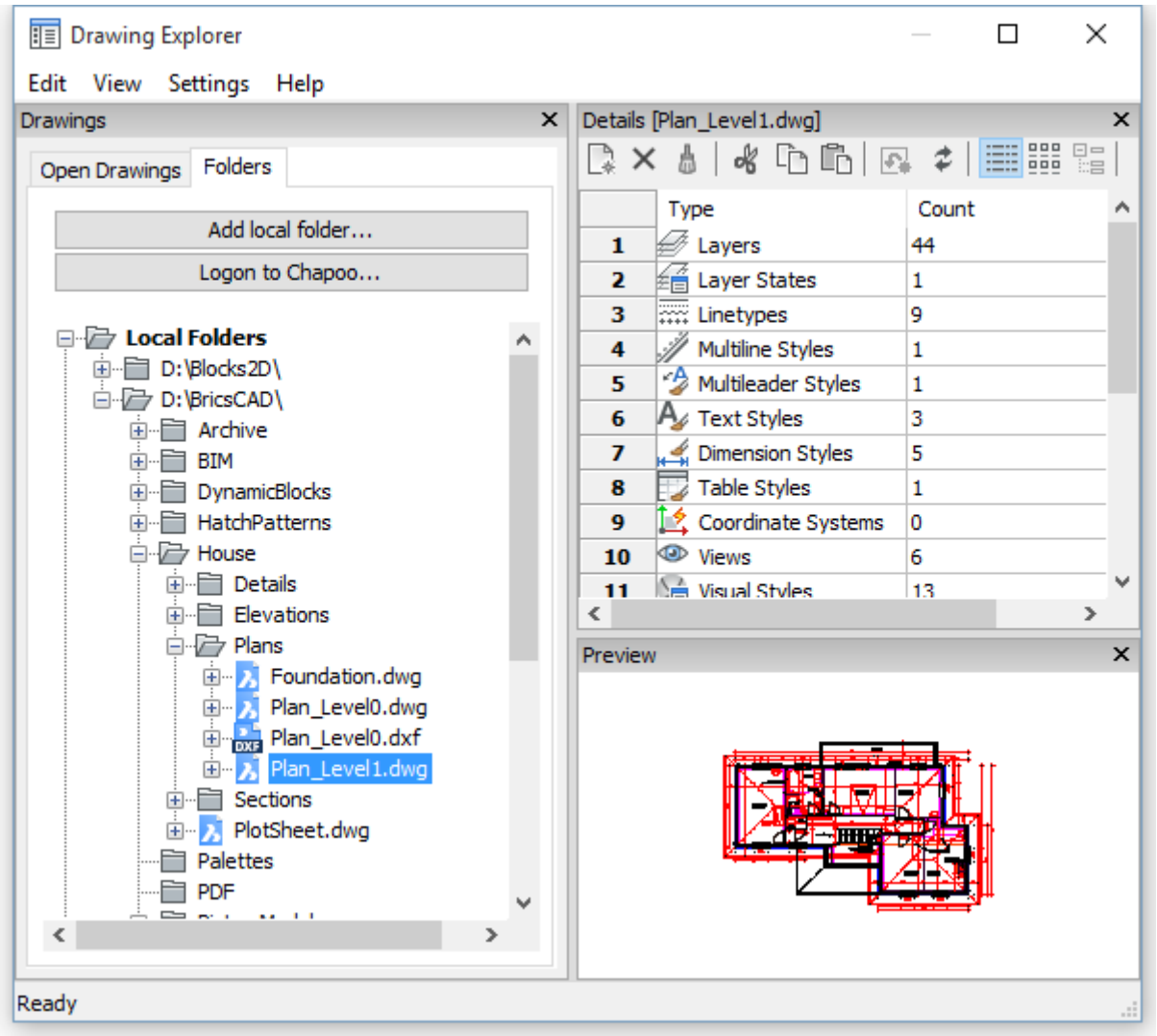


- If you click a settings category in the *Drawings* sub-window when the *Details* sub-window is closed, the *Details* window is re-opened, showing the content of the selected settings category.
- To close a sub-window, click the close button (✕) in its title bar.
- To restore the default layout choose *Restore Default Explorer Layout* in the *Settings* menu.

Opening a Drawing

To open a drawing using the Drawing Explorer do the following

1. Launch the *Drawing Explorer*.
2. In the *Drawings > Folders* sub-window browse to the folder of the drawing.



3. Double click the drawing.
   The drawing opens, while the *Drawing Explorer* window stays open.
4. (optional) Repeat steps 2 and 3 to open more drawings.
5. Close the *Drawing Explorer*.

NOTE When you select a drawing, a preview displays in the *Preview* sub-window.

Adding a Drawing Folder

1. In the *Drawings > Folders* sub-window of the *Drawing Explorer* main window click the *Add local folder...* button.

   The *Browse For Folder* window opens.
2. In the *Browse For Folder* window do one of the following:

   - To add an existing folder: select the folder you want to add.
   - To create a new folder: click the *Make New folder* button.

3. Click the *OK* button.
   The folder is added.

NOTE

- Drawings in the subfolders of the selected folder can be opened too.
- The parent folder(s) of the selected folder is (are) greyed out in the drawing folder tree. Drawings in such folders cannot be opened.

To Insert a Block from an Unopened Drawing

The Block Manager feature of the *Drawing Explorer* allows to insert blocks in the current drawing from unopened drawings.

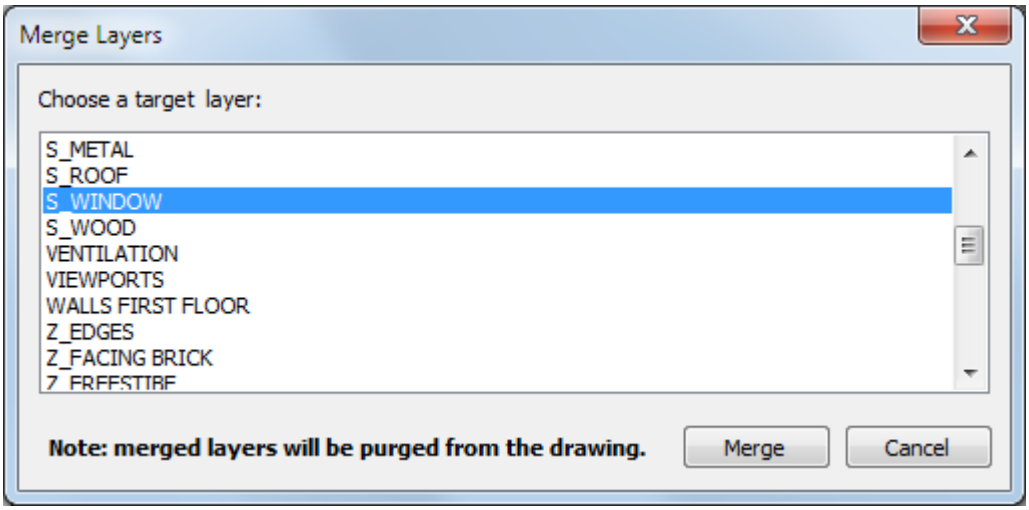
Copy Definitions Between Drawings

1. Open both the source and the target drawing.
2. Launch the Drawing Explorer.
3. In the source drawing select the category you want to copy items from, e.g. *Blocks*.
4. (option) In the *Details* toolbar, click the *Icon View* button ( )
5. Select the item(s) you want to copy, e.g. a series of blocks.
   Press and hold the Ctrl key to select multiple items.
6. Press and hold the left mouse button to drag the selection set to the *Open Drawings* section of the *Drawing Explorer*.
7. Release the left mouse button when on the target drawing name.
   The selected items are copied to the target drawing.
   You will be prompted to overwrite items of the same name.

Merging Layers

1. Open the Drawing Explorer > Layers dialog window.
2. Select the layer you want to merge into another layer.
3. Choose *Merge to...* in the *Edit* menu.
   The *Merge Layers* dialog box displays.

4. Select the layer you want to merge to.



5. Click the *Merge* button.
   The content of the first layer is merged to the second layer.
   The first layer is purged from the drawing.

Drawing Properties

The *Dwgprops* command opens the *Drawing Properties* dialog box, showing the general information and user defined properties stored with a drawing.

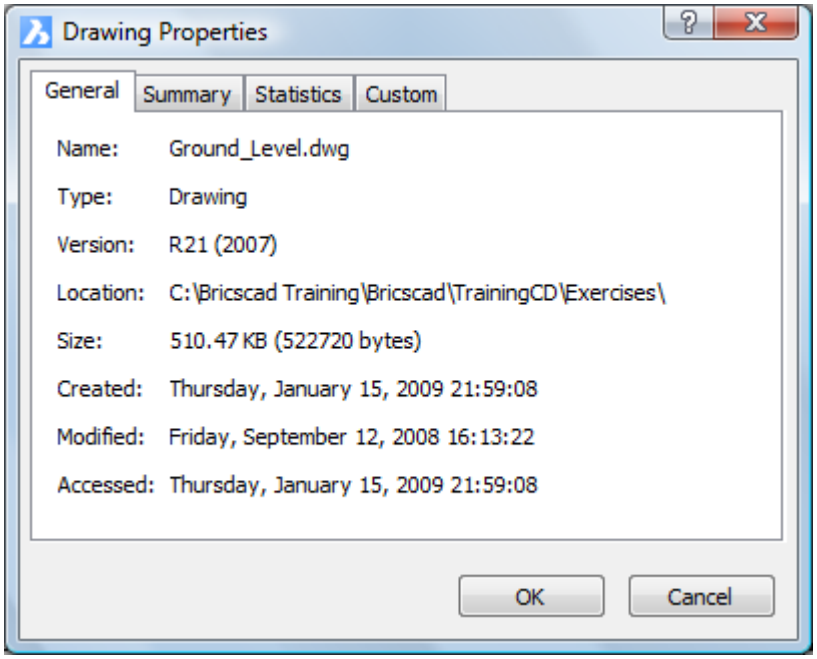
Drawing properties include: general information, summary, statistics and user defined custom properties.

Drawing properties can be referenced in a field in multiline text entities or tables.

To Open the Drawing Properties Dialog

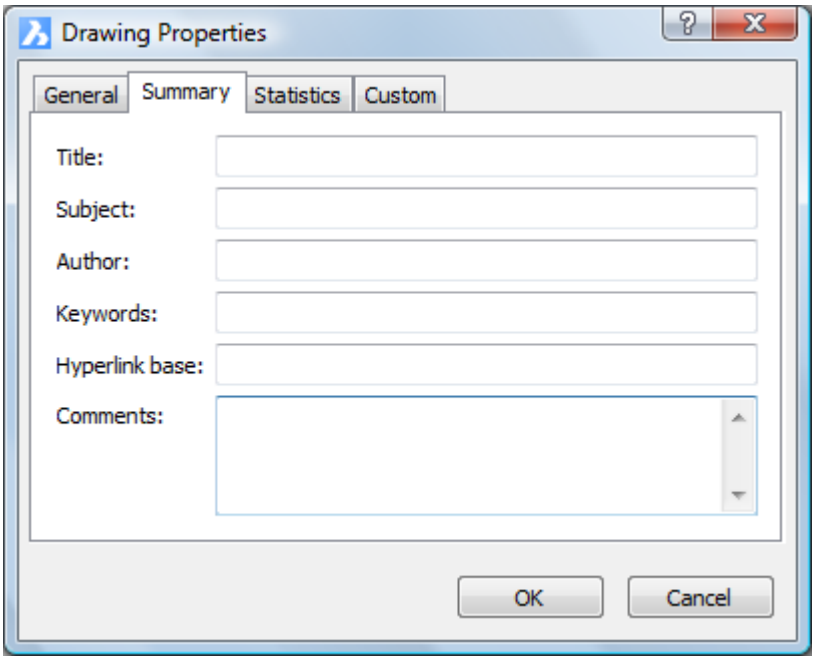
do one of the following:

- Choose *Drawing Properties...* in the *File* menu.
- Type *dwgprops* in the command bar and press Enter.



Defining Default Properties

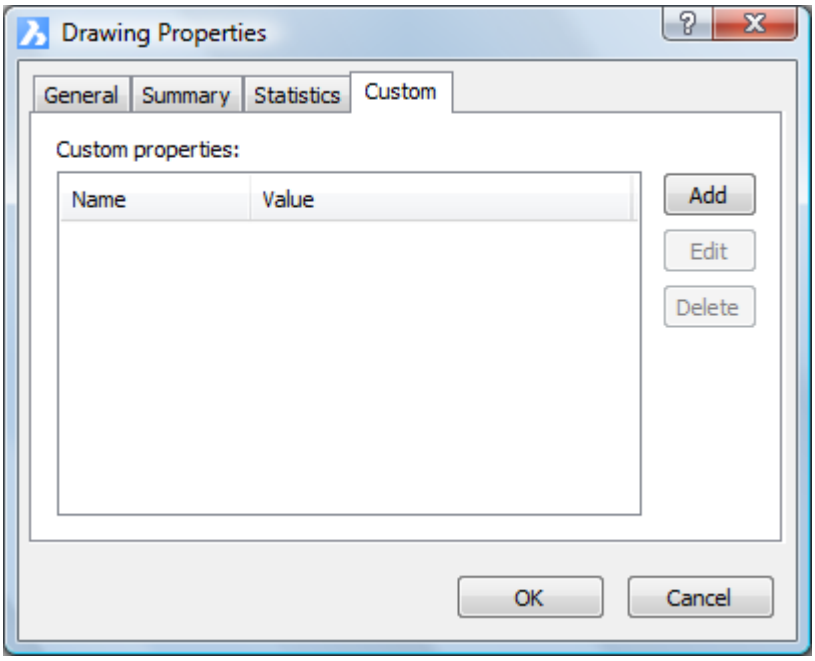
1. Open the Drawing Properties dialog.
2. Open the *Summary* tab page.



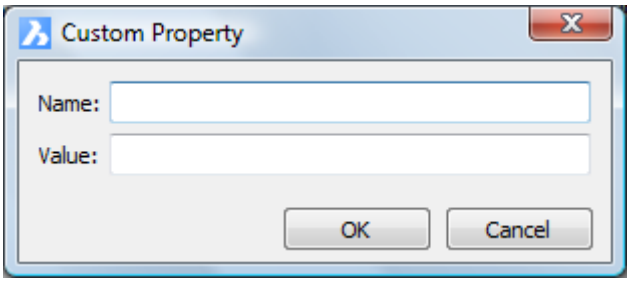
3. Fill out the property fields
4. Click the *OK* button.

Adding Custom Properties

1. Open the Drawing Properties dialog.
2. Open the *Custom* tab page.



3. Click the *Add* button.
   The *Custom Property* dialog displays.



4. Type a name in the *Name* field.
   The custom property name is listed under *Document* in the Field dialog.
5. Type a value in the *Value* field.
   The value displays in multiline texts where the custom property is inserted as a field.

Drawing Security Options

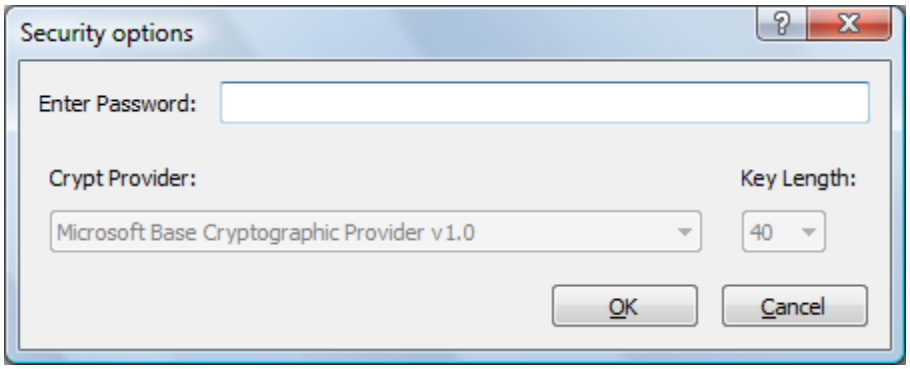
Command

The *Securityoptions* command allows to protect your drawings with a password. Password protected drawings cannot be opened without entering the password first.

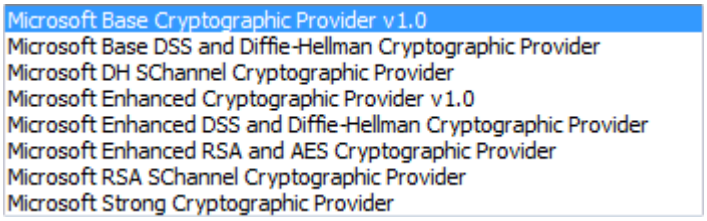
To Set a Password

1. Do one of the following:
   - Choose *Security Options...* in the *File | Drawing Utilities* menu.
   - Type *securityoptions* in the command bar, then press Enter.

The *Security options* dialog opens:



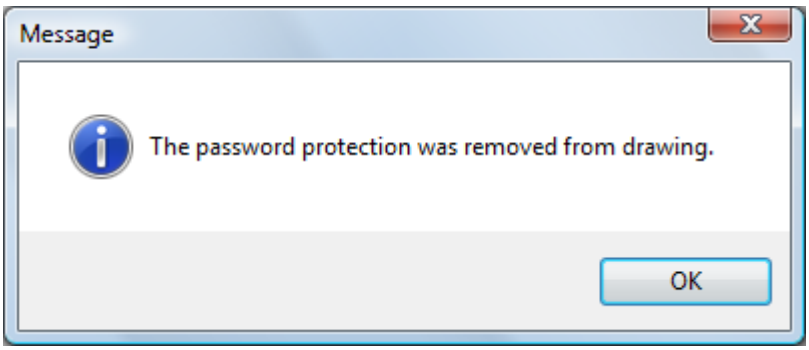
2. Type the password in the *Enter Password* field.
3. Choose a crypt provider from the *Crypt Provider* list.



4. Set the *Key Length* to *40*, *48* or *56*.
5. Click the *OK* button or press Enter.
   The *Security options* dialog closes.
   The *Password* dialog opens.
6. Type the password in the *Confirm Password* field.
7. Click the *OK* button or press Enter.
   The password is defined.

To Remove a Password

1. Type *securityoptions* in the command bar, then press Enter.
   The *Security options* dialog opens.
2. Empty the *Enter Password* field.
3. Click the *OK* button or press Enter.
   A message box displays to confirm that the password is removed.



4. Click the *OK* button or press Enter to close the message box.

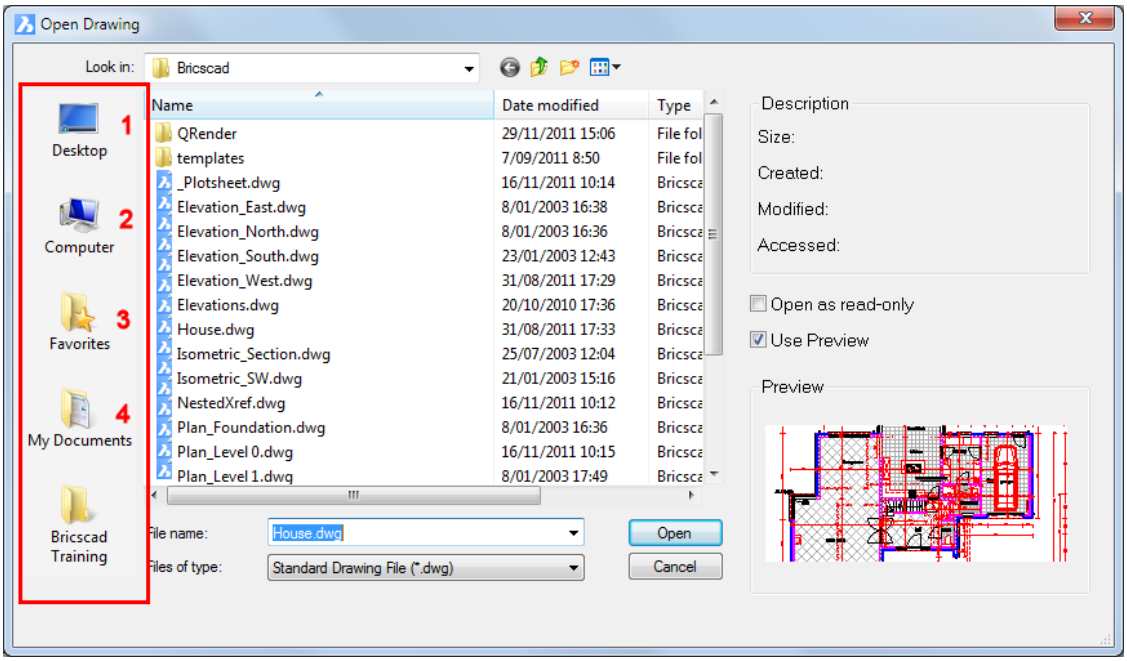
Opening and Saving Drawings

Commands: OPEN, SAVE, SAVEAS, SAVEALL, SAVEFILEFOLDER

The way BricsCAD opens and saves drawings is controlled through a series of system variables and user preferences.

In the table below the SYSTEM VARIABLES are fully capitalized.

Name	Title	Description
ShowFullPathInTitle	Display full path in title	Controls wether the full path of a drawing displays in the title bar, or the file name only.
DrawingPath	Drawings path	Specifies an additional folder in the file dialog for the Open and SaveAs commands. On the Windows platform: the fifth folder in the *Open File* dialog's *Places* bar.
PlacesBarFolder1 - 4	First, Second, Third and Fourth folder	Sets the folders 1 through 4 in the *Places* bar of the non-standard *Open File* dialog (Windows platform only). Values between 0 and 5 are accepted: 0=Desktop, 1=My Computer, 2= My Documents, 3= Favorites, 4=Network, 5=My Recent Documents. Tip: Place shortcuts to your favorite drawing folders on your desktop or in your Favorites folder.



THUMBSIZE	Thumbnail preview image size	Range is 0 - 8; Default = 1 0 (64 x 64) 1 (128 x 128) 2 (256 x 256) 3 (512 x 512) 4 (1024 x 1024) 5 (1440 x 1440) 6 (1600 x 1600) 7 (1920 x 1920) 8 (2560 x 2560)
INDEXCTL	Index Control	Determines whether layer and/or spatial indexes are created and saved. The layer index lists all entities in the drawing and what layer they are on. The spatial index stores the location of all the entities in 3D space. The indexes are used to load Xrefs if demand loading is enabled. If a layer index exists, entities on layers which are Off or Frozen are not loaded. If a spatial index exists, entities in clipped Xrefs which are outside the clipping boundary are not loaded. The demand loading of Xrefs is controlled through the XLOADCTL system variable). Creating indexes slows down the saving of a drawing, but increases the speed of Xrefs. Therefore create indexes only for drawings that will be used as an external reference. 0 = No indexes, 1 = Layer index only, 2 = Spatial index only, 3 = Layer index and spatial index.
XLOADCTL	Xref load control	Controls demand loading of entities in external references. If demand loading is turned on you will experience a performance increase: • When clipping external references which are saved with a spatial index (INDEXCTL = 2 or 3), entities in the external reference within the clipped area are loaded only. If the clipping area is modified, more or less entities are loaded from the reference drawing. • When freezing layers in external references which are

		saved with a layer index (INDEXCTL = 1 or 3), entities in the external reference on thawed layers are loaded only. If xref-dependent layers are thawed more entities are loaded. 0: Demand loading is OFF - The entire external reference is loaded. 1: Demand loading is ON - Referenced drawings are kept open and locked. Other users can open such drawings only as read-only. 2: Demand loading is ON - Temporary copies of referenced drawings are kept open and locked. Other users can open and modify the referenced drawings.
ISAVEBAK	Incremental save backup	Controls the creation of a backup file (BAK). Improves the speed of incremental saves if switched off, especially for large drawings. Please notice that in some cases (i.e. a power failure during a save), it might be possible drawing data are lost.
ISAVEPERCENT	Save percent	This system variable controls how often BricsCAD performs a full save. A full save reorganizes all the data in the drawing database and creates a compact file. A partial save stores all changes a the end of the drawing data base. A full save takes longer to be executed than a partial save. If ISAVEPERCENT = 0, each save is a full save. The higher the value of ISAVEPERCENT, the longer the time between two full saves.
NFILELIST	Recent File List	Specifies the number of files shown in the *Recent Files* flyout of the *File* menu. Values between 0 and 20 are accepted. Default =10.
PROXYGRAPHICS	Proxy graphics	Determines whether images of proxy entities(1) are saved in the drawing. If switched off, a bounding box displays instead.
PROXYNOTICE	Proxy notice	Displays a notice when you open a drawing containing proxy entities(1) created by an application that is not present.
PROXYSHOW	Proxy show	Controls how proxy entities(1) display in a drawing. 0 = Not displayed, 1 = Graphic images, 2 = Bounding box.
PROXYWEBSEARCH	Proxy web search	Determines whether BricsCAD checks for object enablers: 0 = Never, 1 = If a live Internet connection is present.
SaveFormat	Save format	Sets the default dwg version to save a drawing.
SAVEFILEPATH	Save file path	Specifies the path and folder where automatic saves and temporary files are stored. The SaveFileFolder command opens the SAVEFILEPATH location in the system file browser.
SAVETIME	Save time interval	Sets the interval in minutes for automatic saves. If set to zero, automatic saves are turned off. Automatic saves are created with a *.SV$* extension in the folder defined by the SAVEFILEPATH system variable.
WorspaceSecurity	Enable macros	Defines whether macros are enabled when loading a VBA-project.

AutosaveChecksOnlyFirstbitDBMOD	Ignore all but first bit of DBMOD(2) for autosave	If this user preference is switched on, no autosave file will be created for files that are viewed only.
CloseChecksOnlyFirstBitDBMOD	Ignore all but first bit of DBMOD(2) for close	If this user preference is switched on, you can view a file, then close it without getting a request to save the file.

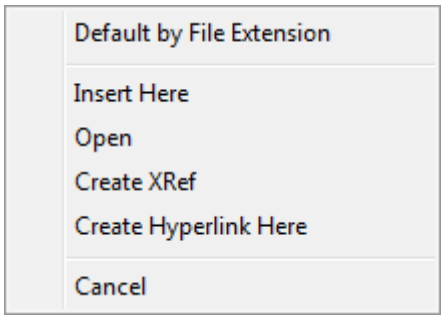
(1) Proxy entities are custom entities which are created by an add-on application. If the add-on is currently not loaded or not installed, such entities cannot be displayed properly.

(2) DBMOD: Stores the status of the drawing modification as a bit-code as the sum of the following values: 1 = Object database modified, 4 = Database variable modified, 8 = Window modified, 16 = View modified, 32 = Field modified.

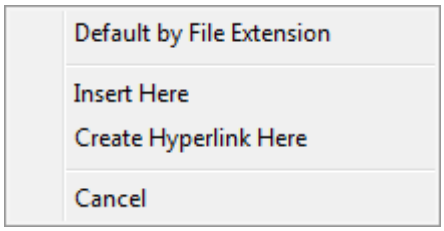
Using the Drag-and-Drop option menu

1. Select a file in a *Windows Explorer* dialog, then hold down the right mouse button and drag the file onto the drawing area of the BricsCAD application window.
   A context menu displays

When a drawing file is selected:



When another file type is selected:



2. Do one of the following:

- Choose *Default by File Extension* to
  - open a drawing file (*.DWG, *.DXF or *.DWT)
  - attach a PDF
  - attach a raster image file
  - insert a file as an OLE object (e.g. an Excel spreadsheet)
- Choose *Insert Here* to:
  - insert a drawing file (*.DWG, *.DXF or *.DWT) as a block in the current drawing.
  - attach a PDF
  - attach a raster image file
  - insert the file as an OLE object
- Choose *Open* to open a drawing file (*.DWG, *.DXF or *.DWT).
- Choose *Create Xref* to attach a drawing file (*.DWG, *.DXF or *.DWT) as an external reference to the current drawing.
- Choose *Create Hyperlink Here* to create an hyperlink to the selected file.

NOTE When multiple files are selected, the files are processed one after the other.

Getting Drawing Information

You can retrieve two types of information from a drawing:

- General information about the drawing.
- Information about the entities in the drawing

General drawing information

Commands: DWGPROPS, SETVAR, STATUS, TIME

The *Dwgprops* command opens the Drawing Properties dialog box, showing the general information and user defined properties stored with a drawing.

The *Setvar* command displays or changes the values of system variables (short for "set variables").

The *Status* command reports the status of the drawing: drawing name, the status of the most important system variables and the total number of entities in the drawing.

The *Time* command reports on time in the drawing: current time, creation time, last update, total editing time and user elapsed time.

Information about Entities

Commands: STATUS, AREA, MASSPROP, ID, LIST

The *Status* command reports the total number of entities in the drawing.

Using Quick Select you can see the number of entities for each category.

The *Area* command finds the area and perimeter (or length) of 2D entities.

The *Massprop* command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties").

The *ID* command reports the x,y,z coordinates of picked points.

The *List* command lists the properties of selected entities.

Comparing Drawings

The *DwgCompare* command compares a selected drawing with the current drawing during a comparison session.

The *EndCompare* command concludes a comparison session.

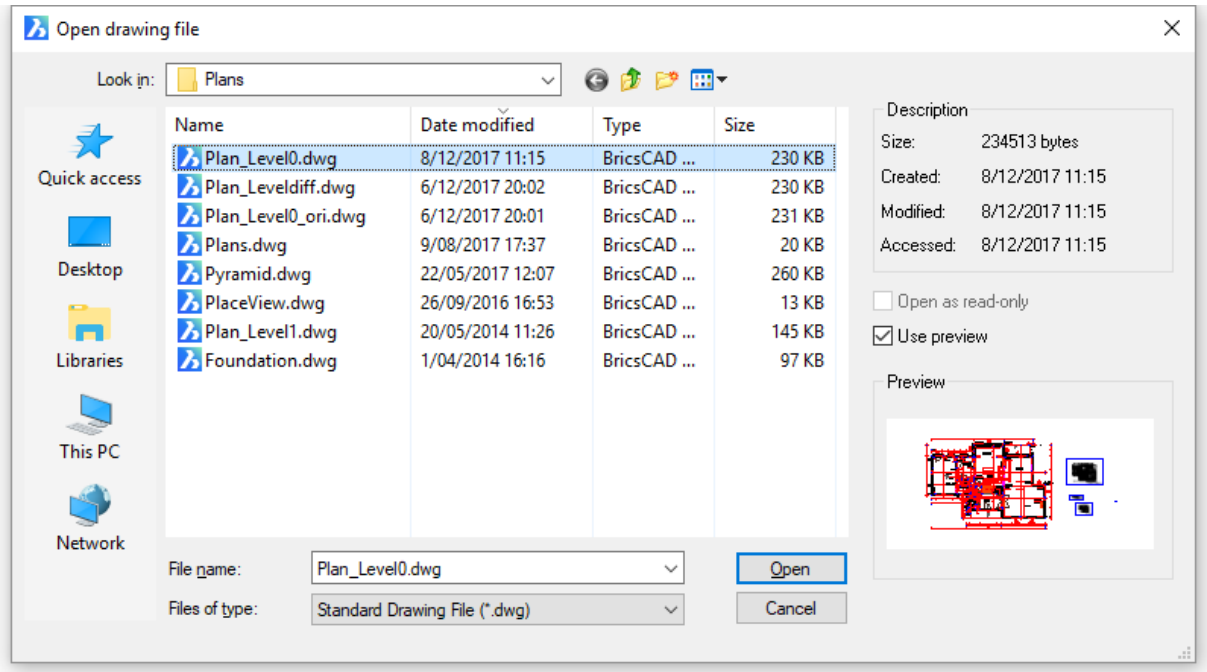
The *KeepMe* command allows to merge the compared drawings visually.

Comparison System Variables

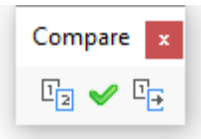
Name	Title	Description
CMPCLRMISS	Compare missing entities display color	Sets the color for displaying entities which are missing in the other drawing. The default color is Red .
CMPCLRMOD1	Compare modified entites display color 1	Sets the color for displaying modified entities in the current drawing. The default color is index color 253.
CMPCLRMOD2	Compare modified entities display color 2	Sets the color for displaying modified entities in the other drawing. The default color is Yellow.
CMPCLRNEW	Compare new entities display color	Sets the color for displaying entities which are new in the other drawing. The default color is Green.
CMPDIFFLIMIT	Maximal number of output entities	Sets the limit for entities comparison. The comparison will be aborted after reaching the difference limit. Values between 1 and 10 000 000 are accepted.

Executing a Comparison Session

1. Launch the DWGCOMPARE command.
   You are prompted: CMPDIFFLIMIT = 10000000. Select the file to compare with or [Limit]:
2. (option) Choose the Limit option to set a new value for entities comparison.
3. Press Enter to display *File* dialog box.

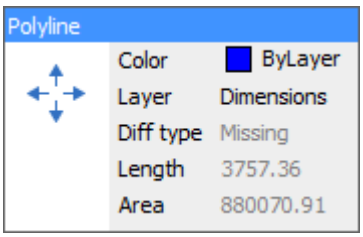


4. Select the drawing to be compared with the current drawing, then click the *Open* button or double click the drawing name.
   The differences between the two drawings display using the Compare Display Colors.
   The Compare toolbar displays:

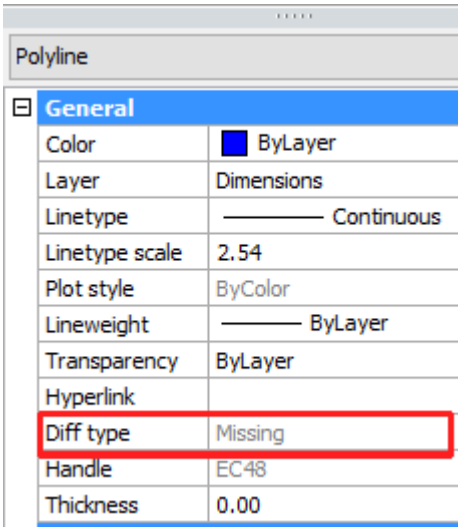


5. To check the *Diff Type* of an entity, do one of the following:

- If Rollover Tips are active, hover over the entity:



- Select the entity and check the *Diff type* field in the Properties panel.



6. Analyze the comparison in the Structure panel.
7. Merge the compared drawings.
8. Click the *Endcompare* tool button ( ) on the *Compare* toolbar to conclude the comparison session.

Analyzing the Comparison in the Structure Panel

1. Open the Structure panel.
2. Load the *Default.cst* configuration. See Load a Structure configuration file.
3. Expand the *Comparison* node.
   Four subnodes display: *Missing*, *Modified (Original)*, *New* and *Modified (Compared)*.
   The number of entities in each subnode is indicated between brackets.

4. Click a subnode to select the corresponding entities in the drawing (see also the note below).
5. Expand a subnode, then click an entity to highlight the corresponding entity in the drawing. Press and hold the Ctrl-key to select multiple entities.
6. Right click, then choose *Zoom* in the context menu to zoom onto the selected entity or entities.

NOTE When selecting entities in the Structure panel, entities are either selected or highlighted in the drawing, or tree selection is ignored. See setting the options in the Structure panel.

Merging the Compared Drawings

1. Missing entities only exist in the current drawing; do one of the following:

- Select the entities you want to keep, then click the *Keep Me* tool button ( ) on the *Compare* toolbar.
- Select the entities you don't want to keep, then click the *Delete* tool button ( ).

2. Modified entities exist in both drawings; do the following:

- In the *Modified (Original)* node: select the entities you want to keep, then click the *Keep Me* tool button ( ) on the *Compare* toolbar.
  The corresponding entities in the *Modified (Compared)* node are removed.
- In the *Modified (Compared)* node: select the entities you want to keep, then click the *Keep Me* tool button ( ) on the *Compare* toolbar.
  The corresponding entities in the *Modified (Original)* node are removed.

3. New entities exist in the compared drawing only. Do the following:

- Select the entities you want to keep, then click the *Keep Me* tool button ( ) on the *Compare* toolbar.
- Select the entities you don't want to keep, then click the *Delete* tool button ( ).

The merge is complete when the *Comparison* node in the *Structure* panel disappears.
A message displays in the Command bar: No more differences to show. Session is closed

Repairing a Drawing

The *Audit* command analyzes the integrity of the current drawing and optionally tries to fix errors.
The *Recover* command repairs a damaged drawing, then opens the drawing.

To Analyze the Integrity of the Current Drawing

Do one of the following:

- Choose *Audit* in the *File | Drawing Utilities* menu.
- Type *audit* in the command bar, then press Enter.

You are prompted: Fix any errors detected? Yes/<No>

Do one of the following:

- Choose *Yes* in the context menu or type *Y*, then press Enter to analyze the integrity of the current drawing and fix the errors detected.
- Choose *No* in the context menu or type *N*, then press Enter to analyze the integrity of the current drawing.

The drawing is analyzed.

(option) Press F2 to open the *Prompt History* window in order to read the audit report:

objects audited
Total errors found during audit #, fixed #.

NOTE

- Set the *AUDITCTL* settings 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 .
- Use the *Recover* command to repair the errors that *Audit* cannot fix.

To Repair a Drawing

1. Do one of the following:
   - Choose *Recover* in the *File | Drawing Utilities* menu.
   - Type *recover* in the command bar, then press Enter.

The *Open Drawing* dialog window displays.

2. Select the drawing, then press the *Open* button on the *Open Drawing* dialog window.
   The drawing is repaired and - if the repair succeeded - opened.
3. (option) Press F2 to open the *Prompt History* window in order to read the recover report.

Exporting Drawings

eTransmit

Command: ETRANSMIT

The *Etransmit* command creates a package of a drawing file and all its dependencies, such as: external references, images, font files, plot configuration files, plot style tables and font map files.

The root drawing and all its dependencies can be copied to a folder are compressed in a ZIP file.

Optionally you can send the ZIP file attached to an email.

TUTORIALS

Package Your Drawings with eTransmit

Export to PDF

Drawing files can be saved as a PDF file, optionally including layers.

Export to PDF Settings

Choose *PDF Export Options...* in the *File* menu to display the *PDF Export* settings in the *Settings* dialog.

PDF export	
Pdf embedded fonts	☑ Enable embedded TTF fonts
Pdf TTF text as geometry	☐ Convert TTF text to geometry
Pdf SHX text as geometry	☐ Convert SHX text to geometry
Pdf text is searchable	☐ Enable pdf text to be searchable
Pdf simple geometry optimization	☑ Enable simple geometry optimization
Pdf zoom to extents mode	☑ Zoom to extents
Pdf Merge Control	[0] Lines Overwrite
Pdf layer support	[1] Use all layers with visible entities
Pdf layouts to export	[0] Active
Pdf papersize override	☐ Enable papersize override
Pdf overridden paper width	210
Pdf overridden paper height	297
Pdf use plotstyles	☑ Enables usage of plotstyles
Export solid hatches	[2] use PDF paths
Image anti-aliasing	☑ Enables anti-aliasing for images
Image compression	[1] JPEG
Vector Resolution DPI	600
Image DPI	300
Render DPI	300
Hatch to bitmap DPI	300

- *Pdf embedded fonts*: Controls whether True Type Fonts are embedded in the PDF export.
- *Pdf TTF text as geometry*: If selected, TTF text is exploded to geometry.
- *Pdf SHX text as geometry*: If selected, SHX text is exploded to geometry.
- *Pdf text is searchable*: If selected, it is possible to search text strings in the PDF document.
- *Pdf simple geometry optimization*: Enable simple geometry optimization: e.g. separate segments to one polyline, use of Bezier curve control points.
- *Pdf zoom to extents mode*: If exporting model space, zoom to extents first, if the current model space view is zoomed in.
- *Pdf merge control*: Controls the appearance of crossing lines.
  The options are:

Lines Overwrite: The topmost line obscures the line(s) under it.
Lines Merge: The colors of crossing lines are merged.

- *Pdf layer support*: Controls whether layers are included in the PDF export.
  The options are: do not export layers, export layers which are on only, export all layers.
- *Pdf layouts to export*: Controls which layouts are exported.
  The options are:
  Active: only the active layout or model space is exported as a single-page pdf.
  All: All layouts, including model space are exported as a multi-page pdf.
- *Pdf papersize override*: If on, the papersize as defined in the BricsCAD Print settings is overridden. The papersize width and height defined by *PdfPaperWidth* and *PdfPaperHeight* is used instead.
- *Pdf overridden paper width*: Width (in mm) of the PDF paper, if *PdfPaperSizeOverride* is on.
- *Pdf overridden paper height*: Height (in mm) of the PDF paper, if *PdfPaperSizeOverride* is on.
- *Export solid hatches*: Controls the export of solid hatches. The options are: *as bitmap*, *use vectorizer*, *use PDF paths*.
- *Pdf use plotstyles*: If on, the plotstyle of the layout controls the color and lineweight in the PDF export.
- *Image anti-aliasing*: Enables anti-aliasing for images that require upscaling when exporting.
- *Image compression*: Defines the compression for exported images: *None* or *JPEG*.
- *Vector Resolution DPI*: Controls the resolution of vector graphics when exporting to PDF. Default is 600 DPI.
- *Image DP*I: When a raster image is exported, this value controls the minimal resolution for the image.
- *Render DPI*: Resolution of a rendering when exporting.
- *Hatch to bitmap DPI*: Defines the resolution of hatch patterns and gradient fills in DPI (= dots per inch).

Export a Drawing To PDF

TUTORIALS

Output as PDF

1. (option) Adjust the Export to PDF settings.
2. Choose *Export...* in the *File* menu.
   The *Export Drawing As* dialog opens.
3. In the *Save as type*: list select *Adobe PDF (*.pdf)*.
4. Select a folder to save the PDF file.
5. (option) Type a name in the *File name* field to replace the default <DRAWING_NAME>.pdf name.
6. Click the *Save* button to create the PDF.

Projects

Projects are especially useful to manage xrefs and images when exchanging drawings. If external references (xrefs) and images are not found in the saved path, the project search paths are used to find the external references and images.

Projects are saved in the registry and contain one or more search paths.

Opening the Projects Settings

1. Click the *Settings* tool button ( ) on the *Standard* toolbar.
   or
   Choose *Settings...* in the *Settings* menu.

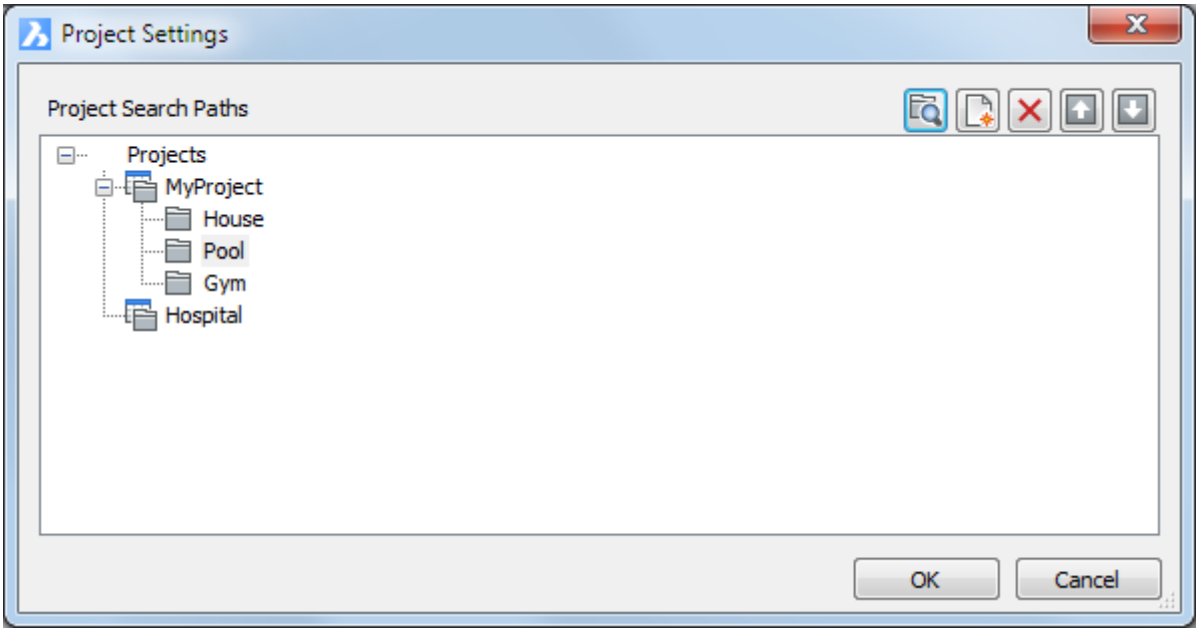
   The *Settings* dialog opens.
2. Go to *Program Options* > *Files* > *Projects* > *Project search paths*.

Creating Projects

To create a project:

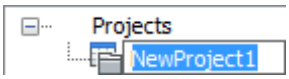
1. Open the Project settings.
2. Click the *Browse* button at the right hand side of the *Project search paths* settings field.

The *Project Settings* dialog opens.



3. To create the first project:

- Click the *New* button ( ) on the *Project Settings* dialog.



- Type a name to replace the *NewProject1* default name.

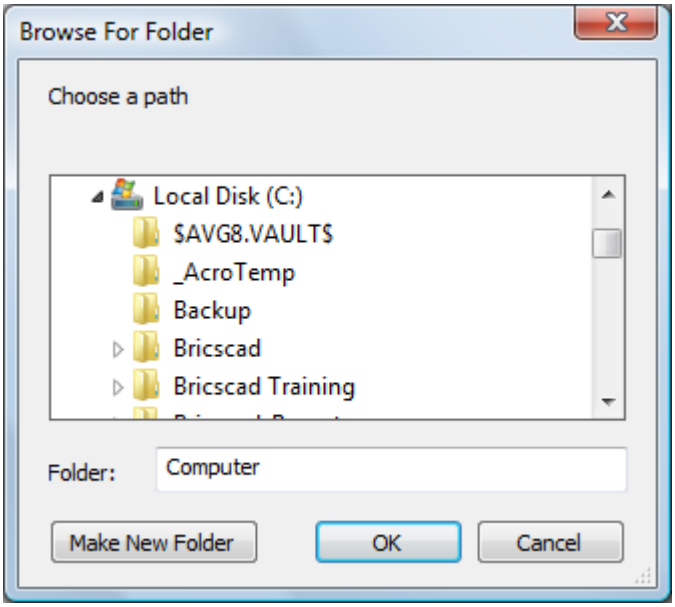
4. To create additional projects.
5. Click *Projects* (on top of the projects tree) then click the *New* button ( ) .

6. Type a name to replace the *NewProject1* default name.

Adding Search Paths to a Project

To add a search path to a project:

1. Open the Project settings.
2. Select the project in the *Projects* tree.
3. Click the *Browse* button ( ).
   The *Browse for folder* dialog opens.



4. Do one of the following:

- Select a folder.
- Select a parent folder, then click the *Make New Folder* button to create a new folder.

5. Click the *OK* button to add the selected folder to the project search paths.

Changing the Order of the Search Paths

1. Open the Project settings.
2. Click the *expand* button (+) of the project you want to edit.
3. Select a search path, then click the up ( ) or down ( ) buttons to change the order of the search paths.

Assigning a Project to the Current Drawing

1. Open the Project settings.
2. Type the name of an existing project in the *Project name* settings field (*PROJECTNAME* system variable).

Drawing Accurately

TUTORIALS

Snap and Grid

In BricsCAD the following drawing aids are available:

- Entity Snaps
- Dynamic dimensions
- Polar Tracking
- Entity Snap Tracking
- Coordinate Input
- Snap and Grid
- Drawing Limits
- Ortho Mode
- User Coordinate Systems
- Direct Distance Entry

Orthogonal Mode

Orthogonal Mode restricts the movement of the cursor to be parallel to the X-axis or the Y-axis of the current coordinate system. For example, with the default 0-degree orientation (angle 0 at the "three o'clock" or "east" position), when the *Orthogonal Mode* setting is enabled, lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. As you draw lines, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is nearest to the cursor: type the length of the line in the command bar, then press Enter.

NOTES

- When you enable the Isometric snap style, cursor movement is restricted to orthogonal equivalents within the current isometric plane.
- You can rotate the *Orthogonal Mode* axes using the Snap Angle setting.
- Press and hold the Shift key to reverse the *Orthogonal Mode* setting.
- *Orthogonal Mode* is overruled by entity snaps.
- Turning on *Ortho Mode* automatically disables Polar Tracking and vice versa.

To toggle Orthogonal Mode

Do one of the following:

- Click the *ORTHO* field in the Status Bar.
- Press the *F8* function key.
- Press and hold the Shift key.
- Type *ortho* in the command bar, then choose the appropriate setting.
- Set the *Orthogonal Mode* in the *Settings* dialog.

Direct Distance Entry

Direct Distance Entry (DDE) means that you can type the length of lines, polyline segments, circle radii and diameters, movement vectors in commands such as Move, Copy and Stretch, etc. in the command bar.

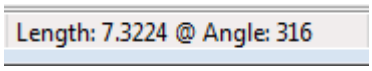
When combined with either Orthogonal Mode or Polar Tracking, *Direct Distance Entry* is a particularly efficient method to draw lines of a specified length and direction, and to move or copy objects over a specified distance.

To help you keep track of the position of the cursor it is recommended to set the readout of the coordinate field in the Status Bar to *Relative*.

To define the Coordinates setting

Do one of the following:

- Right click the then choose *Relative* in the context menu.
- Click the *Coordinates* field in the status bar to cycle the *Coordinate* setting. The sequence is *Off*, *Relative*, *Absolute.*
- Press the *F6* function key or use the *Ctrl + I* keyboard shortcut to cycle the *Coordinate* setting. The sequence is *Off*, *Relative*, *Absolute.*
- Type *coords* in the command bar, then press Enter.
  Type 2 and press Enter.
- Open the Settings dialog and expand the *Display/Viewing* settings sub-category under *Drafting* in the *Drawing* settings class.
  Set the *Coordinates* setting to *Coordinates in polar form for point, distance and angle selection*.



Unit Settings

Commands: UNITS and -UNITS

The *Units* command sets units of linear and angular measurements through the Settings dialog box.

The *-Units* command sets units of linear and angular measurements in the command bar.

The INSUNITS (Insertion Units) system variable defines a drawing units value for automatic scaling when inserting blocks or attaching Xrefs.

Any length, area or volume property can be displayed using the unit(s) preferred by the user.

The unit defined by INSUNITS is used as the base unit for any requested unit conversions. If INSUNITS = 0 (undefined) the PROPUNITS mechanism is disabled because the base drawing unit is undefined.

NOTES

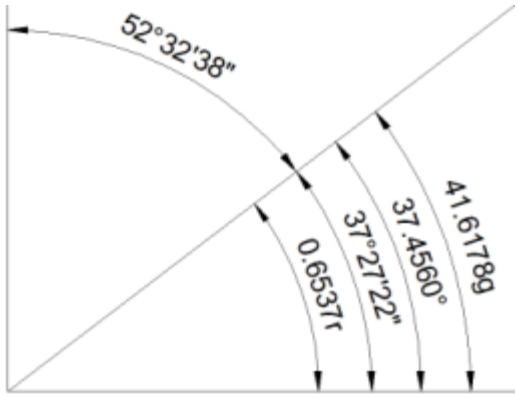
- When LUNITS = 3 (Engineering) or 4 (Architectural), property values will be formatted in imperial notation.
- Automatic unit conversion during length input can be used when bitcode 1 of PROPUNITS (Format length properties) is set. E.g. when INSUNITS is set to "mm", entering "2m" in a length property field will result in entering [2000.0000 mm]. Entering "1ft" will give [304.8000 mm].

Name	Title	Description
INSUNITS	Insertion Units	Defines the value of 1 Drawing Unit (DU), which is used for automatic scaling when inserting or attaching blocks, images or Xrefs. When both INSUNITS and PROPUNITS are ON, Length, Area, Volume and/or Inertia properties will be formatted with a dedicated unit.
MEASUREMENT	Measurement	Sets drawing units as Imperial or Metric. The MEASUREMENT system variable controls whether ANSI or ISO hatch pattern and linetype files are used.
PROPUNITS	Property Units	Defines which property values are automatically formatted with a dedicated unit when INSUNITS is not zero. The value is stored as a bitcode, using the sum of the values of all selected options.
LUPREC	Linear Unit Precision	Sets the number of decimal places displayed for linear units.
AREAPREC	Area Precision	Sets the number of decimal places displayed for areas when the Area bit of PROPUNITS is ON. If set negative, the value of LUPREC is used.
VOLUMEPREC	Volume Precision	Sets the number of decimal places displayed for volumes when the Volume bit of PROPUNITS is ON. If set negative, the value of LUPREC is used.
LENGTHUNITS	Length Units	Defines a list of units used for displaying lengths when the Length bit of PROPUNITS is ON. If empty, all lengths display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.
AREAUNITS	Area Units	Defines a list of units used for displaying areas when the Area bit of PROPUNITS is ON. If empty, all lengths display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.
VOLMEUNITS	Volume Units	Defines a list of units used for displaying volumes when the Volume bit of PROPUNITS is ON. If empty, all lengths display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.
UNITMODE	Unit Mode	Specifies how Imperial units display.

Entering angles

The *AUNITS* system variable defines the angular unit type.
You can enter angles in any of 5 formats:

- **Decimal degrees**: 37.456 (AUNITS = 0)
- **Degrees / Minutes / Seconds**: 37°27'22" or 37d27'22" (AUNITS = 1)
- **Gradians**: 41.6178g (AUNITS = 2)
- **Radians**: 0.6537r (AUNITS = 3)
- **Surveyor's Units**: N52°32'38"E or N52d32'38"E (AUNITS = 4)
  Surveyor's units are measured from the North (N) or the South (S) to the East (E) or to the West (W).



Coordinate Reference System

Command: GEOGRAPHICLOCATION

A Coordinate System is a set of mathematical rules for specifying how coordinates are to be assigned to points. The coordinate system is unrelated to the Earth. A Coordinate Reference System (CRS) is a coordinate system related to the Earth through a "Datum". A Geodetic Datum is a model of the Earth that specifies a reference surface (Ellipsoid or Spheroid).

A projected coordinate reference system is the result of the application of a map projection to a geographic coordinate reference system. A map projection is a type of coordinate conversion. It uses an identified method with specific formulas and a set of parameters specific to that coordinate conversion method.

Coordinates may be changed from one CRS to another through the coordinate operation. Two types of coordinate operation may be distinguished:

- **coordinate conversion**, where no change of datum is involved and the parameters are chosen and thus error free.
- **coordinate transformation**, where the target CRS is based on a different datum to the source CRS. Transformation parameters are empirically determined and thus subject to measurement errors. (Geodetic, Seven paramets, Molodenksy & etc.)

Geographic datasets are defined in *geodatabase.xml* that is placed in *<User folder>\AppData\Roaming\Bricsys\BricsCAD\<RELEASE>\en_US\Support* for Windows. The content of this file is composed of the 5 sections:

- CoordinateReferenceSystems
- Datums
- Ellipsoids
- Transformations
- Locations
- ProjectionCode
- Methods

1. CoordinateReferenceSystems

This section is the main xml node where all Coordinate Reference Systems are defined. It has many sub-nodes called **CRS**, each xml node must define a unique projected coordinate reference system and the necessary projected parameters.

The supported CRS projection types are described in the ***ProjectionCode*** and ***Methods*** sections.

CRS node attributes:
Follow Table 1 of OGP Geomatics Guidance Note No 7, part 2

Attribute name	**Description**	**Units**
epsg	Unique EPSG database id.	Integer
name	Human-readable name of CRS, Example: "WGS 84 -- WGS84 - World Geodetic System 1984, used in GPS"	String
codeSpace	Owner of CRS. It is option.	String
x, **y**	Orientation of projected direction. Possible values: "Westing", "Easting", "Southing" and "Northing". Directions MUST be defined.	String
xAxis, yAxis	Orientation of geographic direction. Possible values: "Latitude" and "Longitude".	String
Greenwich	Greenwich Meridian relation.	Degree
proj	Projection method of CRS. Possible values: "LL", "LatLon", "LonLat" is a Geodetic Latitude-Longitude. "MercA", "Mercator_1SP" is a Mercator (variant A). "MercB", "Mercator_2SP" is a Mercator (variant B). "MercC" is Mercator (variant C). "MercSp" is a Mercator (Spherical). "MercPv" is a Popular Visualisation Pseudo Mercator. "TMerc" is a Transverse Mercator. "TMercSO" is a Transverse Mercator South Orientated. "LCC_1SP" is a Lambert Conic Conformal 1SP. "LCC_2SP" is a Lambert Conic Conformal 2SP. "LCC_2SP_BE" is a Lambert Conic Conformal 2SP Belgium.	String
AngleSG	Angle from Rectified to Skew Grid	Degree
Azimuth	Azimuth of initial line	String
FE	Easting at false origin, False easting	Float-Point value in CRS units
EC	Easting at projection center	Float-Point value in CRS units
FN	Northing at false origin, False northing	Float-Point value in CRS units
FC	Northing at projection center	Float-Point value in CRS units
LonI	The longitude of the western limit of the first zone of a Transverse	Degree

	Mercator zoned grid system	
Lat	Latitude of natural origin, Latitude of standard parallel. It depends on Projection method.	Degree
Lat1	Latitude of 1st standard parallel	Degree
Lat2	Latitude of 2nd standard parallel	Degree
LatF	Latitude of false origin	Degree
LatC	Latitude of projection center	Degree
Lon	Longitude of natural origin, Longitude of origin	Degree
LonF	Longitude of false origin	Degree
LonC	Longitude of projection center	Degree
SF	Scale factor at natural origin	Float-Point value, unity
SFIL	Scale factor on initial line	Float-Point value, unity
SFPSP	Scale factor on pseudo standard parallel	Float-Point value, unity
W	Zone width in longitude	Degree
zone	Zoned Grid System	Degree
units	Units of conversion result from geographic coordinates to projected. Example: "Meter", unit "Degree" means no conversion, Geographic Coordinate System	String

Example: common attributes for all map projections based on EPSG:31468.

```
    <CRS epsg="31468" codeSpace="OGP" name="DHDN / 3-degree Gauss-Kruger zone 4"
y="Easting" x="Northing" Greenwich="0" proj="TMerc" Lon="12" Lat="0" SF="1"
FE="4500000" FN="0" zone="4" units="Meter">
```

For each node ***CRS*** there should be at least one sub-node ***Datum*** that refers via **epsg** or **alias** attributes to sub-node ***Datum*** in the ***Datums*** section. The sub-node ***Datum*** MUST have the **id** attribute of a unique CRS name. At least one of the epsg code and the alias of Datum must be valid.

Attribute name	**Description**	**Units**
epsg	Unique EPSG database id. Example: "4326"	Integer
alias	Unique Datum name. Example: "WGS 84"	String
id	Unique CRS name: combination of Projected method and Datum. Example: "WORLD-MERCATOR". It related to CS-MAP, AutoCAD names.	String
pjcode	Indicate code for the coordinate reference system projection method types. Corresponded to AutoCAD projection codes.	Integer

	For instanse, 3 - Transverse Mercator, 44 - UTM, 45 - Transverse Mercator of Snyder formulation and etc.	

Example: there are unique CRS definitions according to Datums inside **CRS** node:

```
<Datum epsg="6314" alias="DHDN/3" id="DHDN/3.Gauss3d-4"  pjcode="3" />
<Datum epsg="6314" alias="DHDN/2" id="DHDN/2.Gauss3d-4"  pjcode="3" />
<Datum epsg="6314" alias="DHDN"   id="DHDN.Gauss3d-4"  pjcode="3" />
```

2. Datums

This section is a main node where all Datums are stored. Datum is a combination of the Earth model (Ellipsoid or Spheroid) and the transformation method to model WGS84. A ***Datum*** node specifies common parameters and contains at leat one inner sub-nodes ***Datum*** exactly one sub-node ***Ellipsoid***. Each inner sub-node ***Datum*** should define a unique name within the **id** attribute and may specify transformation parameters to WGS84 model via sub-node ***Transformation*** that refers via **epsg** or **alias** attributes to a sub-node ***Transformation*** in the ***Transformations*** section.

Attribute name	**Description**	**Units**
alias	Unique id. Example: "DHDN/3". It related to CS-MAP, AutoCAD names.	String
epsg	Unique EPSG database id. Example: "6314"	Integer
name	Human-readable name of Datum, Example: "Deutsches Hauptdreiecksnetz"	String
codeSpace	Owner of CRS. It is option.	String

Example:

```
<Datums>
    <Datum epsg="6314" codeSpace="OGP" name="Deutsches Hauptdreiecksnetz ">
        <Datum id="DHDN/3">
            <Transformation epsg="1777" alias="DHDN/3_to_WGS84" />
        </Datum>
        <Datum id="DHDN/2" name="Deprecated - Replaced by DHDN/3">
            <Transformation epsg="1777" alias="DHDN/2_to_WGS84" />
        </Datum>
        <Datum id="DHDN"   name="Deprecated - Replaced by DHDN/2">
           <Transformation epsg="1673" alias="DHDN_to_WGS84" />
        </Datum>
        <Ellipsoid epsg="7004" alias="BESSEL" />
    </Datum>
    â€¦
</Datums>
```

3. Ellipsoids

The Ellipsoids section is the main section where models of the Earth are defined. Each ***Ellipsoid*** node should define an identification by a unique name (**alias**) and a semi-major axis (**a**) and one of the following attributes at least: flattering (**f**), semi-minor axis (**b**) or eccentricity (**e**).

Attribute name	Description	Units
alias	Unique id. Example: "WGS84". It related to CS-MAP, AutoCAD names.	String
epsg	Unique EPSG database id. Example: "7030"	Integer
name	Human-readable name of Datum, Example: "WGS 84"	String
codeSpace	Owner of CRS. It is option.	String
a	Length of the semi-major axis of the ellipsoid, the radius of the equator	Float-Point value, strongly in Meter
b	Length of the semi-minor axis of the ellipsoid, the distance along the ellipsoid axis between equator and pole	Float-Point value, strongly in Meter
f	Flattening	Float-Point value, unity
e	Eccentricity	Float-Point value, unity

Example:

```
<Ellipsoid epsg="7008" alias="CLRK66" name="Clarke 1866, Benoit Ratio"
a="6378206.4000000004" b="6356583.7999999998"  f="294.9786982139" e="0.0822718542"
/>
```

4. Transformations

This section is a main node where all transformation methods are defined. Supported Transformation methods are described in the ***ProjectionCode*** and ***Methods*** sections.

Attribute name	Description	Units
alias	Unique id. Example: "DHDN_to_WGS84". It related to CS-MAP, AutoCAD names.	String
epsg	Unique EPSG database id. Example: "1673"	Integer
codeSpace	Owner of Transformation. It is option.	String
source	Source Datum	String
target	Target Datum	String
use	Transformation method. Supported transformations using geocentric methods: Geocentric translations, Four/Six/Seven Parameter Transformation, Similarity Transformation, Position Vector transformation and Coordinate Frame rotation.	String
method	Methods of building of rotation matrix, if applicable, "PVT" is Position Vector Transformation, "CFR" is Coordinate Frame Rotation	String

tX	X-axis translation	Meter
tY	Y-axis translation	Meter
tZ	Z-axis translation	Meter
rX	X-axis rotation	Degree
rY	Y-axis rotation	Degree
rZ	Z-axis rotation	Degree
dS	Scale difference	Unity
xp	Coordinate 1 of evaluation point	Meter
yp	Coordinate 2 of evaluation point	Meter
zp	Coordinate 3 of evaluation point	Meter
dtX	Rate of change of X-axis translation	Unity
dtY	Rate of change of Y-axis translation	Unity
dtZ	Rate of change of Z-axis translation	Unity
drX	Rate of change of X-axis rotation	Unity
drY	Rate of change of Y-axis rotation	Unity
drZ	Rate of change of Z-axis rotation	Unity
ddS	Rate of change of scale difference	Unity
t0	The reference epoch for time-dependent parameters.	Float point value
fallback	Alias of transformation from target datum to WGS84 datum.	String

Example:

```
<Transformation epsg="1679" alias="Pulkovo42/2_to_WGS84" src="Pulkovo42/2"
trd="WGS84" use="Param7" method="CFR" tx="-40.595" ty="-18.55" tz="-69.339" ds="-
4.299" rx="-2.508" ry="-1.832" rz="2.611" accuracy="9" />
```

5. ProjectionCode and Methods

These sections describe the ***ProjectionCode*** and ***Methods*** that are mapped to AutoCAD definitions of projections and transformations types. They are used to store the definitions in an XML file of coordinate reference system definition inside the **AcDbGeoData** object that represents a geographic location.

Parametric Constraints

TUTORIALS

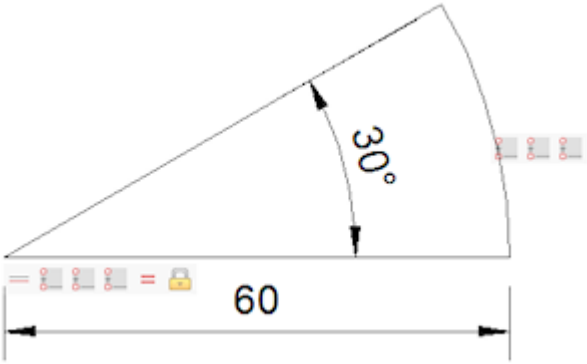
Transform a smartphone into a tablet - Using 2D Dimensional constraints

Parametric constraints are used to maintain relationships and control dimensions of 2D geometry. There are two types of constraints:

- *Geometrical* constraints control relationships between entities.
- *Dimensional* constraints are used to control the dimensions of an entity, such as length, angle, radius or diameter.

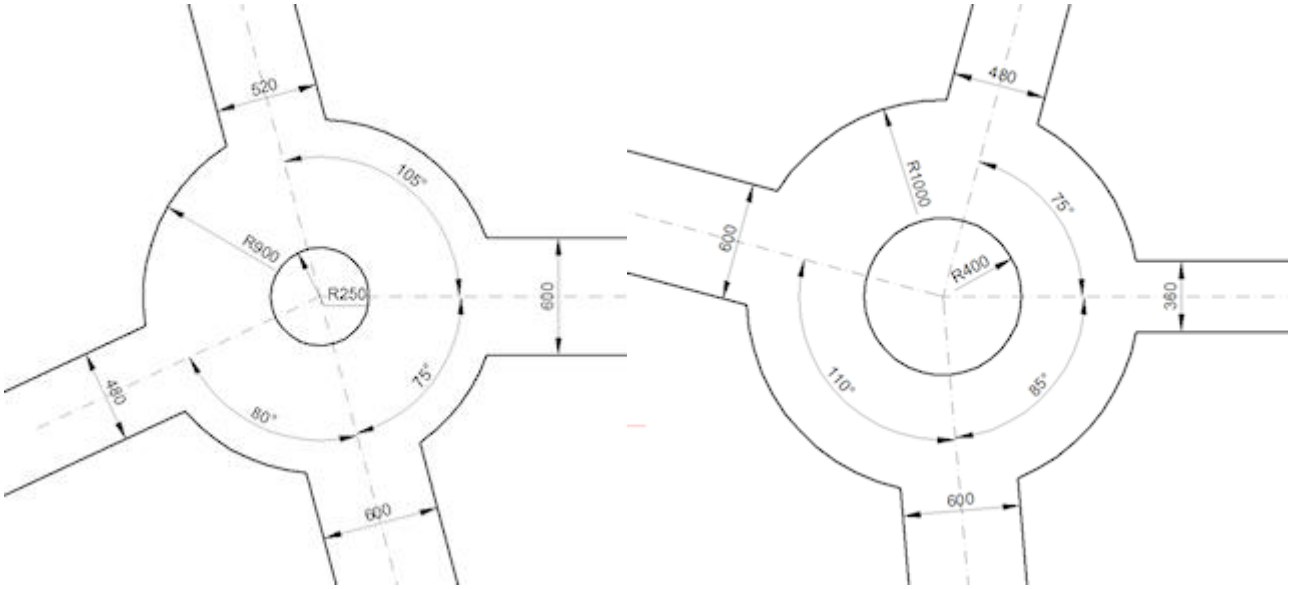
In the example below:

- The width and opening angle are controlled by dimensional constraints.
- Coincident geometrical constraints ( ) are applied to the endpoints of the arc and the polyline and to the center point of the arc and the polyline vertex.
- The polyline segments have an equal ( ) geometrical constraint.
- One segment of the polyline has a horizontal ( ) geometrical constraint.
- The polyline vertex has a fix ( ) geometrical constraint.



Parametric constraints allow to:

- Maintain geometric design intent.
- Make multiple versions of a design by applying different values to dimensional constrains.



Two versions of the same fully constrained drawing representing a roundabout

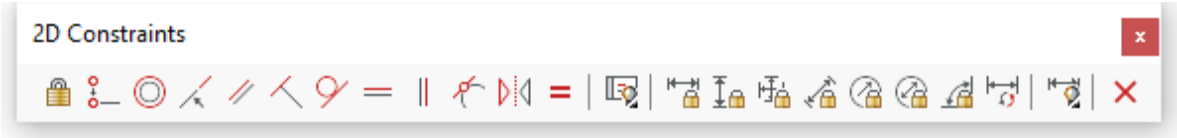
NOTE It is recommended to first apply geometric constraints, then dimensional constraints.

When using constraints a drawing can be in one of three states:

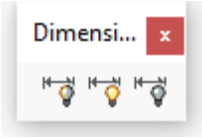
- Unconstrained: no entities are constrained. The drawing can be modified using modification commands and procedures.
- Underconstrained: some entities are constrained. It might be impossible to apply some changes using modification commands and procedures.
- Fully constrained: all possible and relevant geometric and dimensional constraints are applied to the design geometry. The drawing can be modified only by changing the values of the dimensional constraints.

Commands and toolbars

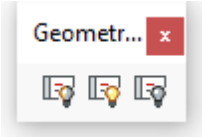
2D parametric constraints tools sit on the *2D Constraints* toolbar:



Tools to control the display of constraintbars and geometrical constraints sit on two flyout toolbars: *Dimensional Constraint*:



and *Geometric Constraint:*



Command	Icon	Description
Geometric Constraints		
DcDisplay		Allows to set dimensional constraints to be displayed or hidden.
		Shows all dimensional constraints.
		Hides all dimensional constraints.
GeomConstraint		Applies geometric relationships between entities and/or valid constraint points on entities.
GcFix		Constrains points and entities at a fixed position.
GcCoincident		Applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcConcentric		Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcCollinear		Forces lines or polyline segments to be collinear.
GcParallel		Forces two lines or linear polyline segments to be parallel to each other.
GcPerpendicular		Constrains two lines or polyline segments to lie perpendicularly to each other.
GcTangent		Constrains two entities to maintain a point of tangency to each other or their extensions. The following entities are accepted: lines, polyline segments, circles, arcs, ellipses or elliptical arcs.
GcHorizontal		Constrains lines, linear polyline segments or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcVertical		Constrains lines, linear polyline segments or pairs of points to lie parallel to the Y-axis of the current coordinate system.
GcSmooth		Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.

GcSymmetric		Constrains two entities or points to lie symmetrically with respect to a selected line.
GcEqual		Constrains arcs and circles to the same radius, or lines and polyline segments to the same length.
Dimensional Constraints		
ConstraintBar		Controls the display of the constraint bar of entities. Constraint bars are always hidden when opening a drawing.
		Shows all constraintbars.
		Hides all constraintbars.
Dimconstraint		Applies a dimensional constraint to an entity or between constraint points on entities. Converts associative dimensions to dynamic dimensions.
DcHorizontal		Constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcVertical		Constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcLinear		Constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcAligned		Constrains the distance between two points or the length of a line or polyline segment.
DcRadius		Constrains the radius of a circle, an arc or an arc polyline segment.
DcDiameter		Constrains the diameter of a circle, an arc or an arc polyline segment.
DcAngular		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.
DcConvert		Converts an associative dimension to a dimensional constraint.
Delconstraint		Removes all dimensional and geometrical constraints from an entity.

NOTE The PARAMETERCOPYMODE system variable controls copying of constraints and related parameters when entities are copied.

0: Do not copy 2D constraints.

1: Replace all expressions with constants.

2: Use existing parameters; if a parameter is missing, replace it with a constant.

3: Use existing parameters; create a new parameter if a parameter is missing.

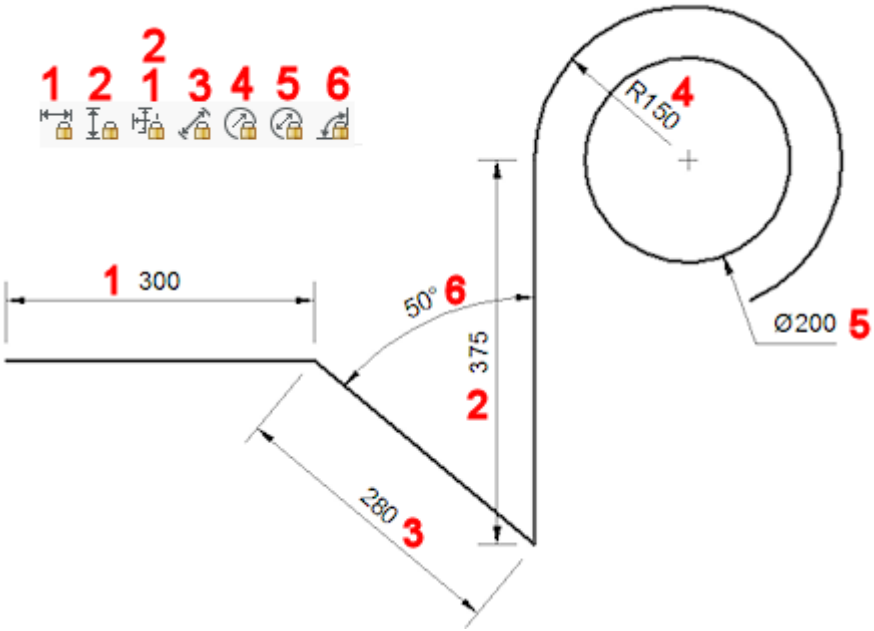
4: Use existing parameters; create a new parameter if a parameter is missing, or has a different value.

Using Dimensional Constraints

Commands: DIMCONSTRAINT, DCALIGNED, DCANGULAR, DCCONVERT, DCDIAMETER, DCDISPLAY, DCHORIZONTAL, DCLINEAR, DCRADIUS, DCVERTICAL, DELCONSTRAINT

The dimensions of the geometry can be controlled by dimensional constraints:

- Horizontal distance between points (1).
- Vertical distance between points (2).
- Distance between points or the length of lines and polyline segments (3).
- Radius of arcs, circles and arc polyline segments (4).
- Diameter of arcs, circles and arc polyline segments (5).
- Angle between entities or between points and entities (6).



NOTE Dimensional constraints are created using the current dimension style.

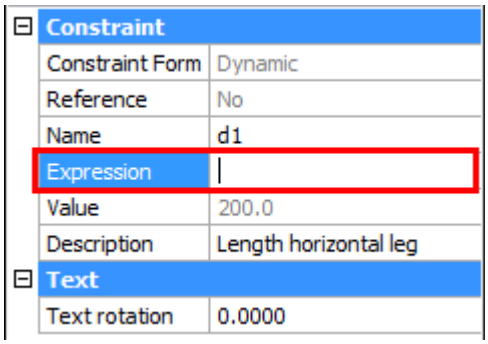
To Edit Dimensional Constraints

Dimensional constraints can be edited:

- using the DDEDIT command
- in the Properties bar
- in the Mechanical Browser

To edit dimensional constraints in the Properties bar.

1. Select the dimensional constraint.
   The properties of the entity display in the *Properties* bar.
2. Type a new value or expression in the *Expression* field, then press Enter.
   The readout of the *Value* field changes accordingly.

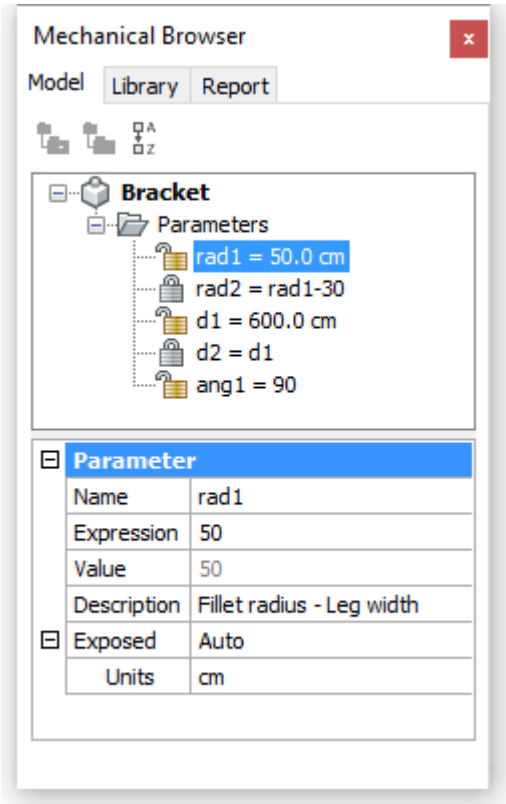


3. (option) Edit the *Name* field.
4. (option) Edit the *Description* field.
5. (option) Edit the *Text rotation* field.
6. Press the Esc key to unselect.

NOTE Select multiple dimensional constraints simultaneously to set their value equally.

To edit dimensional constraints in the Mechanical Browser:

1. If not already open, open the Mechanical Browser.



2. Select a parameter.
   The properties the selected parameter display in bottom part of the *Mechanical Browser*.
3. (option) Select the *Name* field, then type a new name.
4. (option) Select the *Expression* field, then type a new value or expression.
5. (option) Select the *Description* field, then type a new description.
6. (option) Set the *Exposed* field, which specifies whether the parameter is exposed when the drawing is inserted in another drawing using BmInsert.

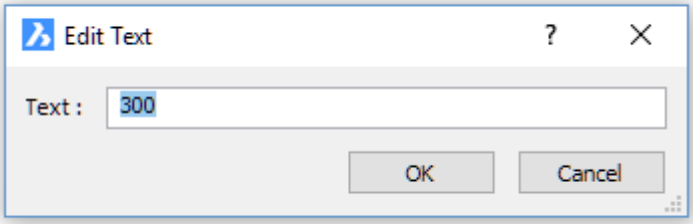
The options are:

- **Auto**: the parameter is exposed only if it does not depend on other parameters.
- **ON** = the parameter is always exposed.
- **OFF** = the parameter is never exposed.

7. (option) Set the *Units* property: Undefined, 1-dimensional (linear), 2-dimensional (planar) or 3-dimensiona (volume)l.
   Select the field, then click the down-arrow button and choose an option.

To edit a single dimensional constraint:

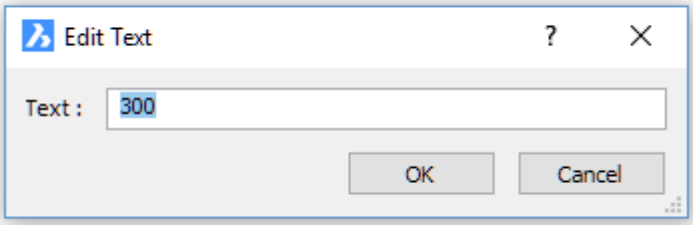
Double click the dimensional constraint.
You are prompted: _ddedit
The *Edit Text* dialog box displays.



Type a new value in the *Text* field, then press Enter or click the *OK* button.
The new value is applied and the geometry is rebuild accordingly.

To edit multiple dimensional constraints:

1. Type *ddedit* or *ed* in the command bar, then press Enter.
   You are prompted: Select entity to modify:
2. Click the a dimensional constraint.
   The *Edit Text* dialog box displays.



2. Type a new value in the *Text* field, then press Enter or click the *OK* button.
3. (option) Repeat steps 2 and 3 to edit more dimensional constraints.
4. Press Enter or right click to stop editing dimensional constraints.
   All edited dimensional constraints are updated simultaneously.
   The geometry is rebuild accordingly.

To Set the Visibility of Dimensional Constraints

Tools to control the display of dimensional constraints sit on the *Dimensional Constraint* toolbar.

Dimensional constraints are hidden when a drawing is closed and reopened.

To set the display of dimensional constraints of a selection of entities

1. Do one of the following:

   Click the *Show/Hide Dimensional Constraint* tool button ( ) on the *2D Constraints* or *Dimensional Constraint* toolbar.

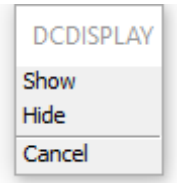
   Choose *Show/Hide Dimensional Constraint* in the *Parametric* menu.

   Type *dcdisplay* in the command bar.

You are prompted: Select Entities:

2. Select the entities, then right click or press Enter.
   BricsCAD reports the number of selected entities.
   You are prompted: Enter an option [Show/Hide] <Show>:

   A prompt menu displays:



3. Do one of the following:

   Press Enter to display the dimensional constraints of the selected entities.

   Choose *Hide* or *Show* in the context menu.

   Type *H* or *S* in the command bar, then press Enter.

To set the visibility of the dimensional constraints of all entities

Do one of the following:

- Click the *Show All* tool button ( ) on the *Dimensional Constraint* toolbar to display the dimensional constraints of all entities.
- Click the *Hide All* tool button ( ) on the *Dimensional Constraint* toolbar to hide the dimensional constraints of all entities.

NOTE The DYNCONSTRAINTMODE system variable controls the visibility of hidden dimensional constraints when selecting entities:

- ON: Display dimensional constraints when entities are selected.
- OFF: Dimensional constraints remain hidden when entities are selected.

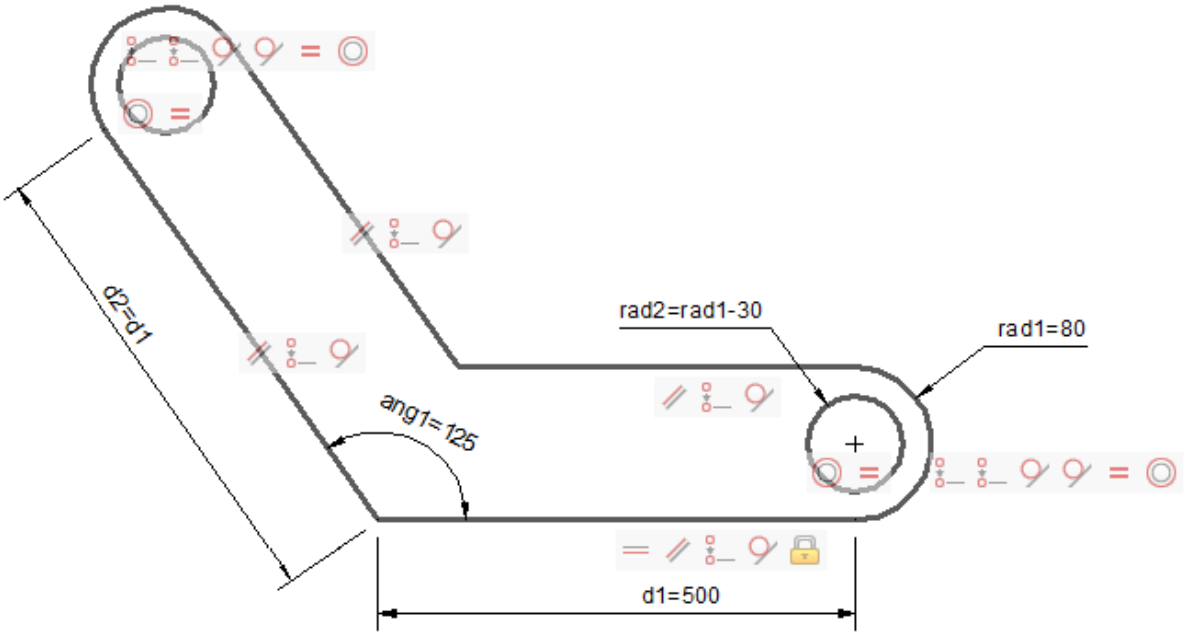
The DYNCONSTRAINTMODE system variable controls the visibility of hidden dimensional constraints when the constrained entity is selected. When hovering over an entity with a dimensional constraint, the blue constraint glyph ( ) displays if the SELECTIONPREVIEW system variable is set to 1 or 3

Using expressions to define dimensional constraints

Each dimensional constraint is named automatically upon its creation.

The naming is as follows:
distance: d1, d2, d3, ...
radius: rad1, rad2, rad3, ...
diameter: dia1, dia2, dia3, ..
angular: ang1, ang2, ang3, ..

The name of a constraint can be used as a parameter in a mathematical expression to define another dimensional constraint.
The dimensions of the bracket below are controlled through the *rad1* radius, *d1* distance and *ang1* angular dimensional constraints.
The radius of the hole (*rad2*) and the length of the second leg (*d2*) are defined by expressions.
Geometrical constraints force the width of the second leg and the hole in it to be equal to the first leg.



Creating a New Parameter

1. Select either the part name or *Parameters* in the *Mechanical Browser*, then right click.
   A context menu displays.
2. Select *Add new parameter* in the context menu.
   A new parameter is created.
3. Edit the properties of the new parameter.

Parameter	
Name	v6
Expression	1
Value	1
Description	
Exposed	Auto
Units	

NOTES

- The constants *Pi* and *e* can be used in expressions.
- The CLEANUNUSEDVARIABLES command purges parameters that are not used in constraint expressions or linked to dimensions.

Using Operators in Expressions

The following operators can be used in expressions:

Operator	Description
+	Addition
-	Subtraction or Negative
*	Multiplication
/	Division
^	Exponentiation
%	Modulo or Remainder operator The expression "5 % 2" would evaluate to 1, because 5 divided by 2 leaves a quotient of 2 and a remainder of 1 .

Expressions are evaluated according to the standard mathematical rules of precedence:

1. Expressions within brackets; innermost sets first.
2. Standard operations order:
    1. exponent
    2. multiplication and division
    3. addition and subtraction
3. Operators of equal precedence from left to right.

Using Functions in Expressions

The following functions can be used in expressions:

Function	Syntax
Cosine	cos(expression)
Sine	sin(expression)
Tangent	tan(expression)
Arc cosine	acos(expression)
Arc sine	asin(expression)
Arc tangent	atan(expression)
Hyperbolic cosine	cosh(expression)
Hyperbolic sine	sinh(expression)
Hyperbolic tangent	tanh(expression)
Arc hyperbolic cosine	acosh(expression)
Arc hyperbolic sine	asinh(expression)
Arc hyperbolic tangent	atanh(expression)
Square root	sqrt(expression)
Signum function (-1,0,1)	sign(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Round down	floor(expression)
Round up	ceil(expression)
Absolute value	abs(expression)
Largest element in array	max(expression1;expression2) [1]
Smallest element in array	min(expression1;expression2) [1]
Degrees to radians	d2r(expression)
Radians to degrees	r2d(expression)

Logarithm, base e	ln(expression)
Logarithm, base	10 log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Power function	pow(expression1;expression2) [1]
Random decimal, 0-1	Random

(1) Use the list separator character as defined on your system: , (comma) or ; (semicolon).

Using Geometric Constraints

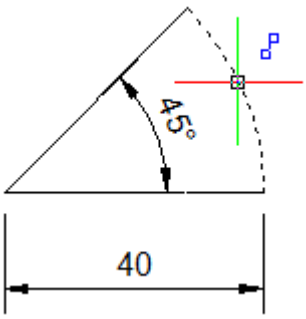
Commands: GEOMCONSTRAINT, GCCOINCIDENT, GCCOLLINEAR, GCCONCENTRIC, GCCENTER, GCEQUAL, GCFIX, GCHORIZONTAL, GCPARALLEL, GCPERPENDICULAR, GCSMOOTH, GCSYMMETRIC, GCTANGENT, GCVERTICAL

A geometric constraint is a non-numerical relationship between the parts of a geometric figure. Geometric constraints:

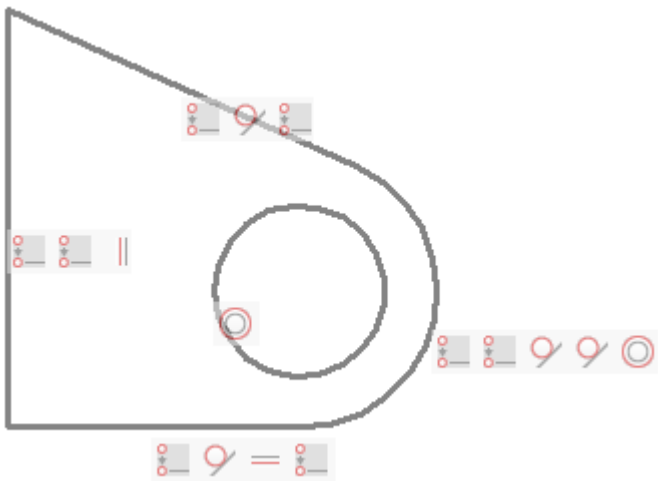
- associate geometric entities together two by two (coincident, concentric, collinear, parallel, perpendicular, tangent, smooth, symmetric, equal)
- specify a fixed angle (horizontal, vertical)
- specify a fixed location (fix)

When a geometric constrained is applied to an entity:

- the position of the entity is adjusted according to the applied constraint.
- an icon displays next to the entity, to indicate the applied constraint. If multiple geometric constraints are applied, the icons are joined to a *constraint bar*.
- If the SELECTIONPREVIEW system variable is set to 1 or 3, a blue constraint glyph ( ) displays next to the cursor when you move the cursor over entities that have constraints applied to them.



In the image below the endpoints of the three lines and the arc are joined by coincident constraints. The midpoint of the circle and the arc are made concentric. Two lines are forced to be tangent to the arc. One line has a vertical constraint (= parallel to the Y-axis of the current coordinate system), one line has a horizontal constraint (= parallel to the X-axis of the current coordinate system)



To Set the Display of Constraints Bars

Tools to control the display of constraintbars sit on the *Geometric Constraint* toolbar.

Constraint bars are hidden when a drawing is closed and reopened.

To set the display of constraint bars of a selection of entities

1. Do one of the following:

   Click the *Show/Hide Geometric Constraint* tool button ( ) on the *2D Constraints* or *Geometric Constraint* toolbar.

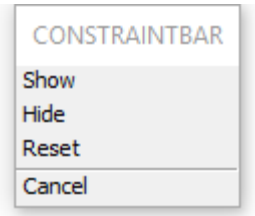
   Choose *Show/Hide Geometric Constraint* in the *Parametric* menu.

   Type *constraintbar* in the command bar.

You are prompted: Select Entities.

2. Select the entities, then right click or press Enter.
   BricsCAD reports the number of selected entities.
   You are prompted: Select option for constraints [Show/Hide/Reset] <Show>:

   A prompt menu displays:



3. Do one of the following:

   - Press Enter to display the constraint bar of the selected entities.
   - Choose *Hide* or *Show* in the context menu.
   - Type *H* or *S* in the command bar, then press Enter.

To set the display of constraint bars of all entities

Do one of the following:

- Click the *Show All* tool button ( ) on the *Geometric Constraint* toolbar to display all constraint bars.
- Click the *Hide All* tool button ( ) on the toolbar *Geometric Constraint* to hide all constraint bars.

NOTE The value of the CONSTRAINTBARDISPLAY system variable controls the visibility of geometric constraints:

- 1: Display constraint bars when geometrical constraints are added.
- 2: Display hidden dimensional constraints when the constrained entities are selected.

To Control the Position of a Constraints Bars

By default constraint glyph bars are created close to the midpoint of the entity and are kept at that relative position when the entity position changes. You can drag the constraint bar to a different location. This new relative position is then maintained until the *Reset* option of the CONSTRAINTBAR command restores the default position of the constraint bar.

To relocate a constraint bar:

1. Place the cursor on the constraint bar.
2. Press and hold the left mouse button to move the constraint bar.
3. Release the left mouse button at the desired location.

To restore the default position of the constraint bars:

1. Do one of the following:

   Click the *Show/Hide Geometric Constraint* tool button ( ) on the *2D Constraints* or *Geometric Constraint* toolbar.

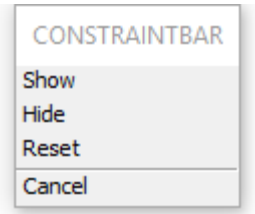
   Choose *Show/Hide Geometric Constraint* in the *Parametric* menu.

   Type *constraintbar* in the command bar.

You are prompted: Select Entities.

2. Select the entities, then right click or press Enter.
   BricsCAD reports the number of selected entities.
   You are prompted: Select option for constraints [Show/Hide/Reset] <Show>:

   A prompt menu displays:



3. Do one of the following:

   - Choose *Reset* in the context menu.
   - Type *R* in the command bar, then press Enter.

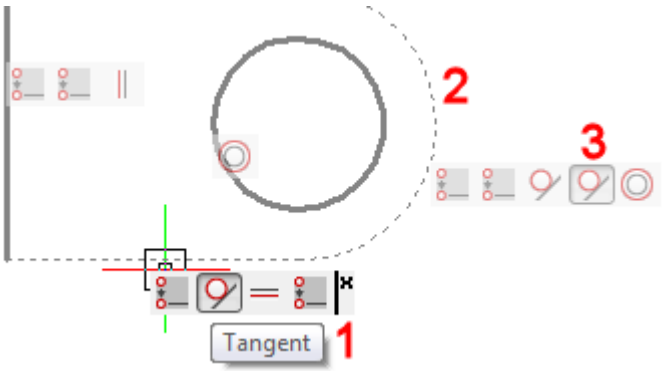
Working with the Constraints Bar

Using the constraint bar you can:

- Control a constraint.

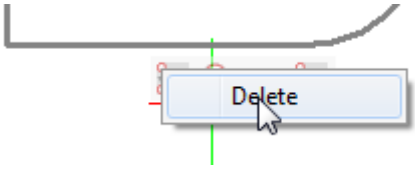
  Move the cursor over a constraint icon.

  A tooltip displays, indicating the constraint type (1)
  The associated entity highlights (2)
  The corresponding icon on the constraint bar of the associated entity highlights (3)



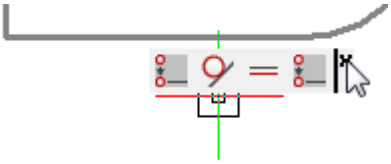
- Delete a constraint.

  Move the cursor over the constraint icon, then right click.
  Click *Delete*.
  The constraint is deleted and the icon is removed from this constraint bar and from the constraint bar of the corresponding entity.



- Hide the constraint bar of an entity.

  Move the cursor of the constraint bar, then click the *Close* button.



Deleting Constraints

To delete a single constraint: right click the constraint icon, then click the *Delete* button.

To delete all constraints from a selection set:

1. Click the *Delete 2D Constraints* icon (✕) on the *2D Constraints* toolbar.
   You are prompted: Select entities to delete all constraints:
2. Select the entities you want to delete the constraints from.
3. Right click to stop selecting entities and delete all constraints from the selection set.

Using Fix Constraints

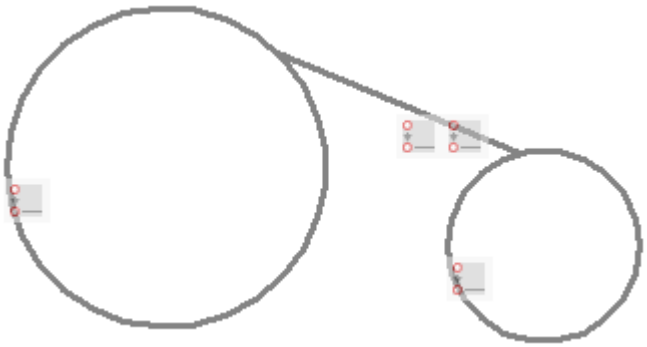
A fix geometric constraint ( ) forces a point or an entity to a fixed location.

- If a fix geometric constraint is applied to a line or a polyline segment, the angle of the line or polyline segment is fixed. The endpoints can still be moved.
- If you fix an arc or a circle, the center point is fixed too.

NOTE It is recommended to apply a fix constraint to important geometric features in your design. This prevents the geometry from moving unexpectedly when you edit the design.

Using Coincident Constraints

A coincident geometric constraint can be applied to two points or to a point and an entity. The endpoints of lines, polylines, splines, rays and arcs (circular and elliptical) and the center point of circles, ellipses and arcs (circular and elliptical) are accepted. These points can be made coincident (= to lie on the entity or on the extension of the entity) with a line, a polyline segment, a circle or an arc (circular and elliptical).



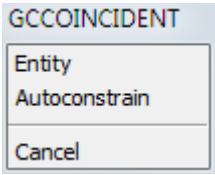
Coincident constraints between the endpoint of a line and two circles

If the endpoints or center points of entities already coincide, the *Autoconstraint* option of the *GcCoincident* command automatically applies coincident constraints to such points.

To apply a coincident constraint between a point and an entity:

1. Click the *Coincident* tool button ( ) on the *2D Constraints* toolbar.
   You are prompted: Select first point or [Entity/Autoconstrain] <Entity>:

   A prompt menu displays:

2. Do one of the following:
   - Press Enter.
   - Choose *Entity* in the prompt menu.

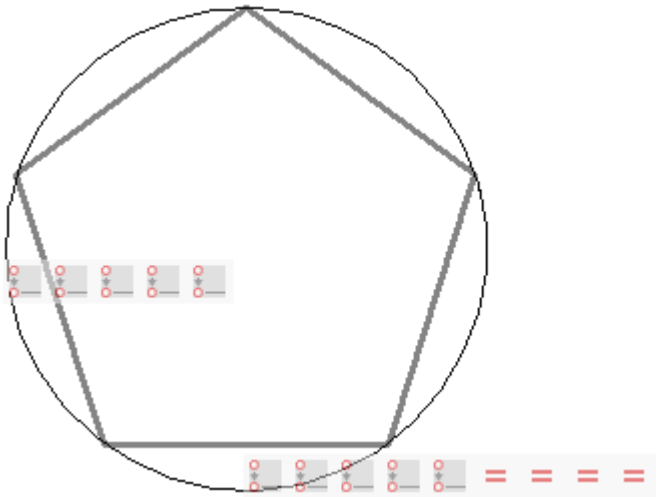
You are prompted: Select an entity:

3. Select the entity.
   The entity highlights.
   You are prompted: Select point or [Multiple]:

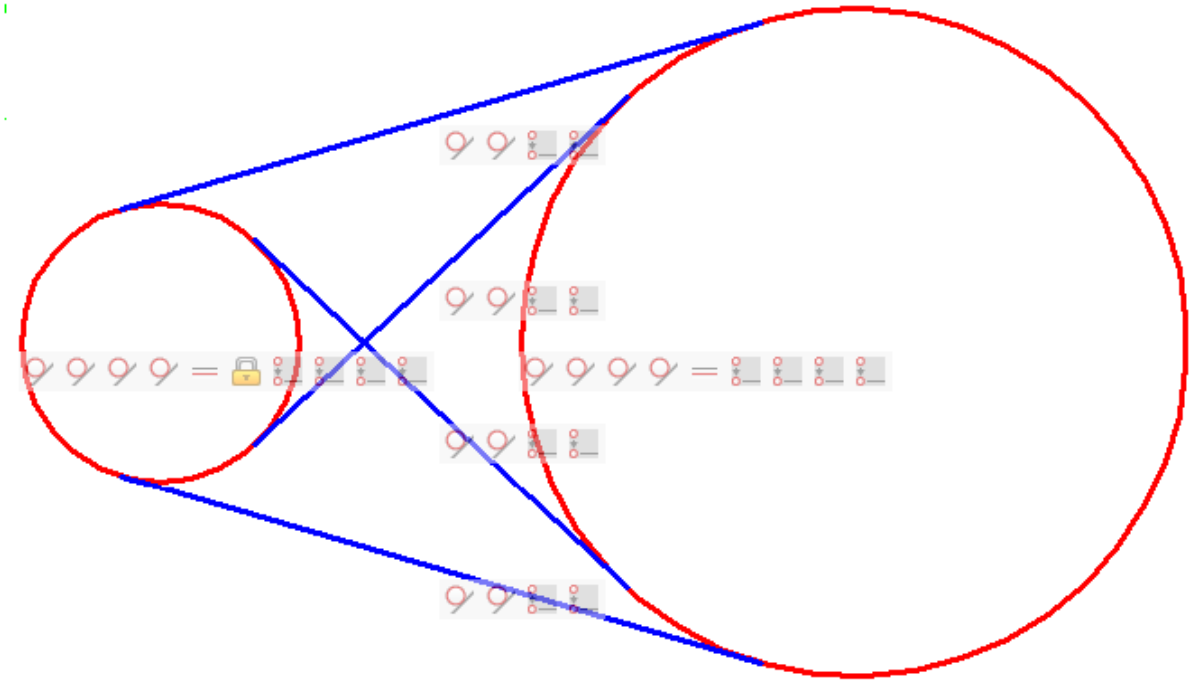
4. Snap to the endpoint of an entity or the center point of a circle or an arc.
   The selected point is forced to lie on the selected entity (or its extension).

NOTE If you choose *Multiple* in step 3, you can select multiple points.

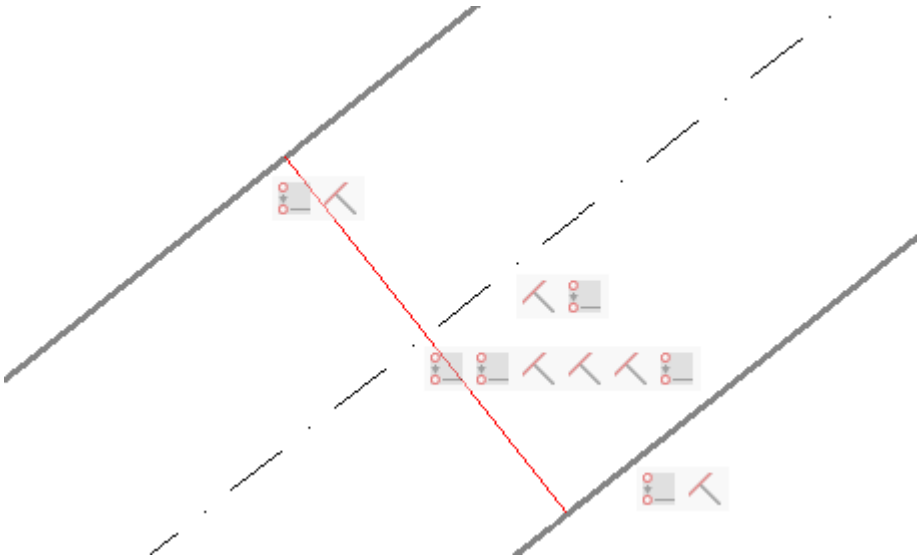
Usage of a coincident constraint between a point and an entity



By applying coincident constraints between the pentagon vertices and the circle, and equal constraints between one side and the four other sides, the circle radius defines the size of the pentagon.



Coincident constraints between the endpoints of the tangent lines and the circles prevent the tangent lines to extend beyond the tangent points.



Coincident constraints are used to:

- force the endpoints of the red line to lie on the bold lines;
- force the midpoint of the red line to lie on the dash-dot line.

The bold lines and the dash-dot line have a perpendicular constraint with the red line.

As a result the dash dot line will always lie in the middle of the two bold lines.

Using Horizontal and Vertical Constraints

Horizontal and vertical geometric constraints force two points, a line or a polyline segment to be parallel to the X-axis (horizontal) or the Y-axis (vertical) of the current coordinate system. Therefore these constraints can be used to keep lines at a fixed angle, rather than only horizontal or vertical.

To Constrain a Line to a Fixed Angle

1. Align the X-axis of the coordinate system to the desired angle.
   - Type *UCS* in the command bar, then press Enter.
   - Choose *Z* in the prompt menu or type *Z*, then press Enter.
   - Type the angle, then press Enter.

2. Click the *Horizontal* tool button (=) on the *2D Constraints* toolbar.
   You are prompted: Select an entity or [2Points] <2Points>:
3. Click the line or polyline segment.
   The selected entity is constrained to be parallel to the X-axis of the current UCS.
4. (option) Restore the WCS: Type *UCS* in the command bar, then press Enter twice.

NOTE Use the *Vertical* tool in the above procedure to constrain entities to be parallel to the Y-axis of the current UCS.

Dynamic Dimensions

Dynamic dimensions provide an interface at the cursor position to specify the length and angle of the linear entity being created or grip-edited. As such dynamic dimensions help you to focus on the drawing area.

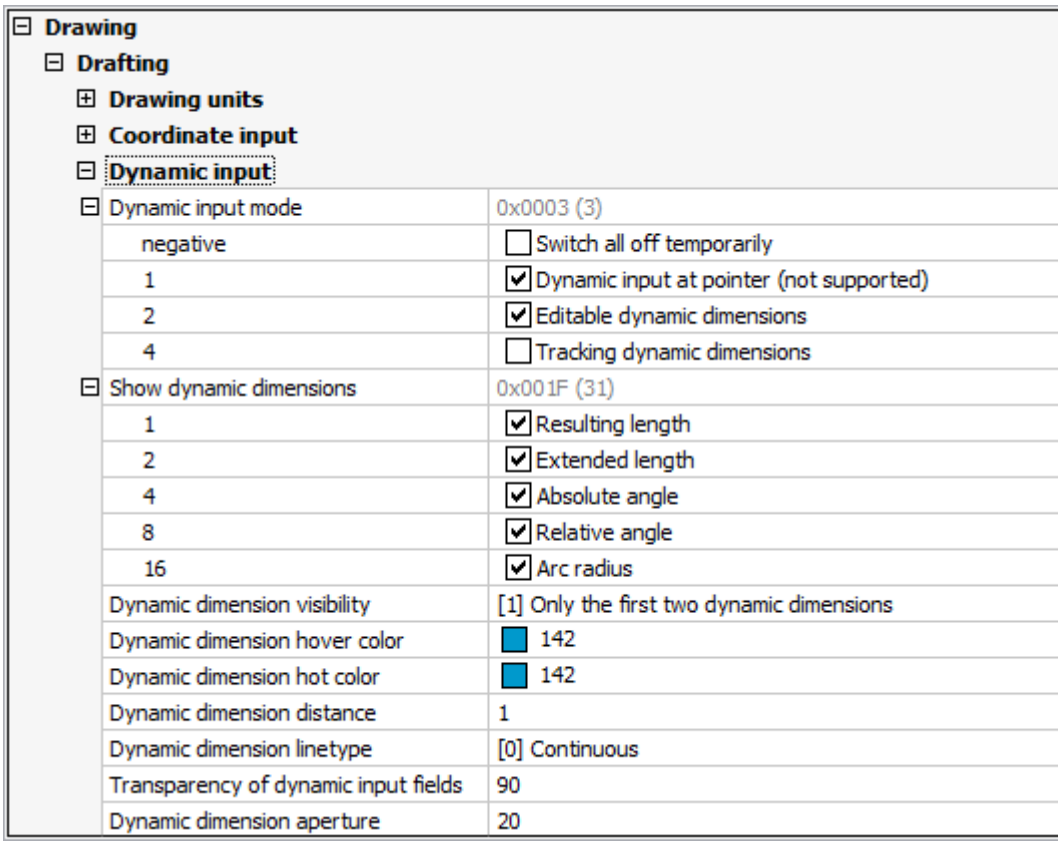
The behavior and appearance of dynamic dimensions is controlled through a series of system variables and user preferences.

TUTORIALS

Defining Dynamic Dimensions

1. Right click the *DYNMODE* field in the Status Bar, then choose *Settings* in the context menu.

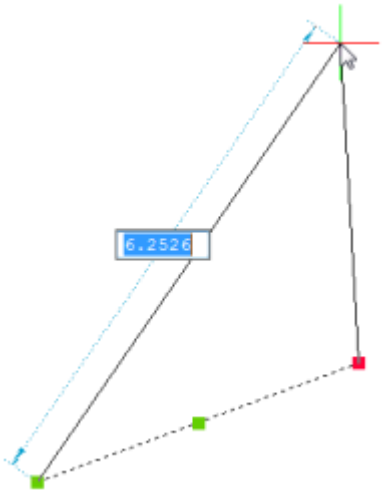
The *Dynamic input* settings in the Settings dialog display:



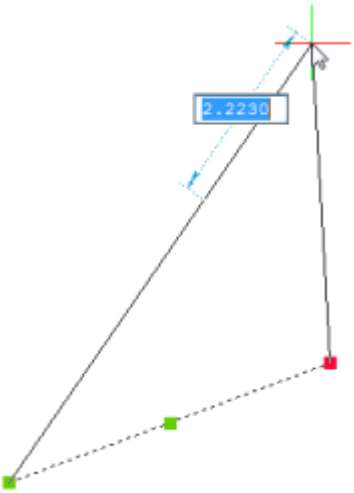
Drawing	
Drafting	
Drawing units	
Coordinate input	
Dynamic input	
Dynamic input mode	0x0003 (3)
negative	☐ Switch all off temporarily
1	☑ Dynamic input at pointer (not supported)
2	☑ Editable dynamic dimensions
4	☐ Tracking dynamic dimensions
Show dynamic dimensions	0x001F (31)
1	☑ Resulting length
2	☑ Extended length
4	☑ Absolute angle
8	☑ Relative angle
16	☑ Arc radius
Dynamic dimension visibility	[1] Only the first two dynamic dimensions
Dynamic dimension hover color	142
Dynamic dimension hot color	142
Dynamic dimension distance	1
Dynamic dimension linetype	[0] Continuous
Transparency of dynamic input fields	90
Dynamic dimension aperture	20

2. Define the *DYNMODE* system variable, to switch the display of dynamic dimensions and/or dynamic input on/off.
   Please notice that dynamic input at the pointer is not implemented yet.
3. Define the *DYNDIGRIP* system variable, to select the dynamic dimension types to be displayed.
   Distance fields have a blue background; angle fields have a green background.

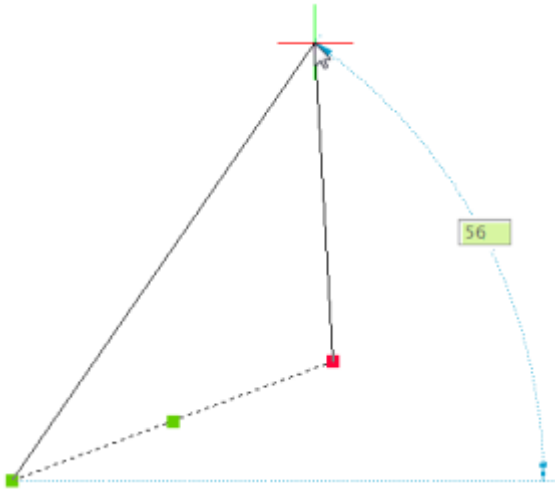
- Resulting length: total length of the line or polyline segment



- Extended length: incremental length of the line

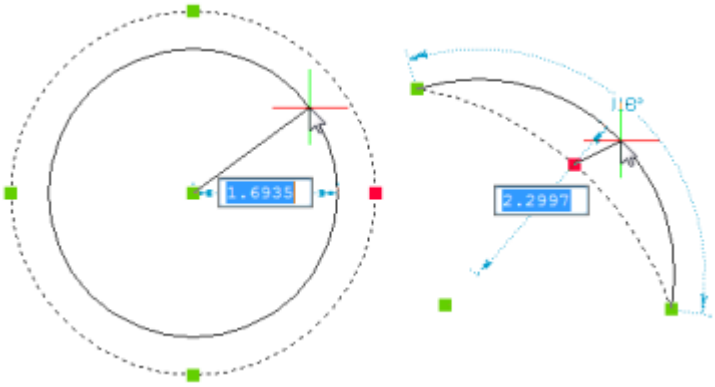


- Absolute angle: angle relative to the X-axis of the current UCS



- Relative angle: angle relative to the original angle of the linear entity

- Arc radius: radius of arcs and circles

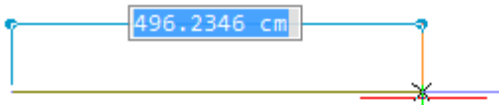


5. Define the *DYNDIVIS* system variable to choose the number of dynamic dimensions that display simultaneously.

   - *Only the first dynamic dimension* (*DYNDIVIS*=0): hit the TAB key to display and edit the next dynamic dimension, as defined by the *DYNDIGRIP* system variable.
   - *Only the first two dynamic dimension* (*DYNDIVIS*=1): hit the TAB key to edit the next dynamic dimension, as defined by the *DYNDIGRIP* system variable and display the next plus one.
   - *All dynamic dimensions as defined by DYNDIGRIP* (*DYNDIVIS*=2): hit the TAB key to edit the next dynamic dimension

6. Set the dynamic dimensions user preferences to define the appearance (color, linetype and position) of dynamic dimensions.

NOTE If bitcode 16 (= format dynamic dimensions) of the PROPUNITS system variable is set, dynamic dimensions are formatted using a dedicated unit.



Automatic unit conversion during length input can be used when bitcode 1 of PROPUNITS (Format length properties) is set. E.g. when INSUNITS is set to "mm", entering "2m" in a

length property field will result in entering [2000.0000 mm]. Entering "1ft" will give [304.8000 mm]. If no unit is specified in the entry field, the unit as specified by INSUNITS is used.

Create Entities

1. Launch the command to create a 2D entity (e.g. Line and *DYNDIVIS* = 1 or 2).
2. Specify the first point, then move the cursor to display the dynamic dimensions.
   The *Length* dynamic dimension highlights.
3. Do one of the following:
   - Type the desired length of the line, then hit the TAB key to jump to the *Angle* dynamic dimension.
     The *Length* dynamic dimension is locked and displays in red and the *Angle* dynamic dimension is selected.
   - Type the desired length of the line, then click to create the line at the angle defined by the cursor position.
   - Hit he TAB key to jump to the *Angle* dynamic dimension.

4. Do one of the following:

   - Type the desired angle, then hit the Enter key to create the line.
   - Type the desired angle, then hit the TAB key to return to the *Length* dynamic dimension.
   - Hit the Enter key to create the line at the angle defined by the cursor position.
   - Hit the TAB key to return to the *Length* dynamic dimension.

NOTE

- If *DYNDIVIS* = 0, only one dynamic dimension displays. Hit the TAB key to display the next field.
- If you type a value in the *Length* field, and add a comma (,) the content of the *Length* field is copied to the command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates with respect to the previous point.

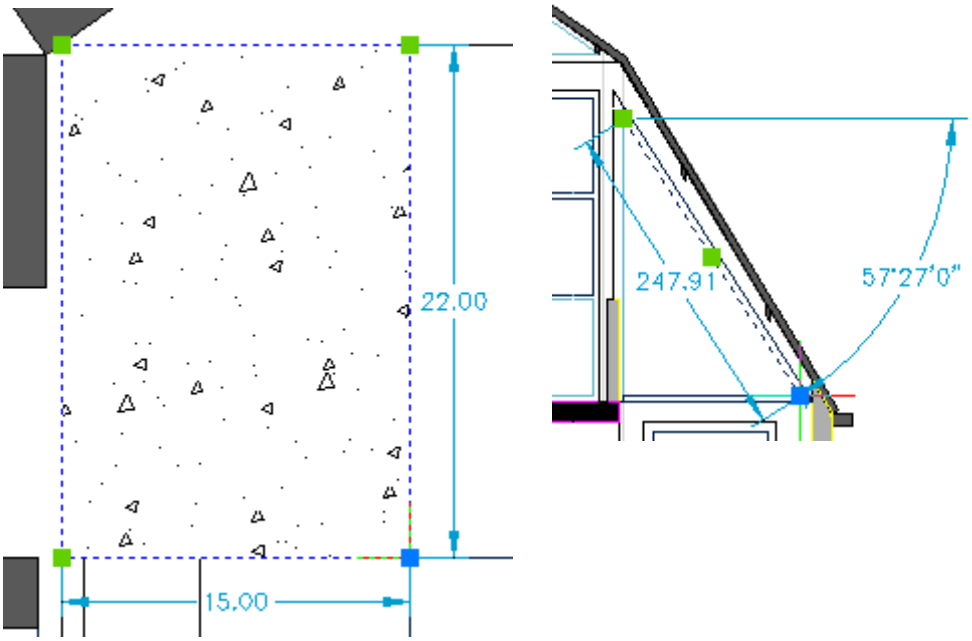
Grip-edit Entities

1. Select the entity or entities.
   All grips on the selected entities display.
2. Click the grip you want to edit, then move the cursor.
   Depending on the value of the DYNDIVIS system variable one, two or all dynamic dimensions display simultaneously .
   The first dynamic dimension highlights.
3. Do one of the following:
   - Type the desired value for the currently highlighted dynamic dimension, then hit the TAB key to jump to the next dynamic dimension.
     The dynamic dimension is locked and displays in red.
   - Type the desired value for the currently highlighted dynamic dimension, then press Enter to stop.
   - Hit the TAB key to jump to the next dynamic dimension.

4. Repeat step 3 or press Enter to stop.

Measure Entities

1. Select the entity or entities.
   All grips on the selected entities display.
2. Hover the cursor over a grip.
   The grip highlights and all dimensions related to the selected grip display.



Dynamic dimensions used to measure a rectangle (left) and a line (right).

(option) Click to start editing the selected grip.

NOTE Dynamic dimensions when hovering over a non-selected grip display in 2DWireframe visual style only.

Using Dynamic Dimensions with Tracking Points

If the *Tracking dynamic dimensions* option of the DYNMODE system variable is set, dynamic dimensions display when entity snap tracking points or temporary tracking points are acquired.

Coordinate Input

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. Every drawing has a fixed coordinate system called the *World Coordinate System* (WCS).

You can also define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS) and can be located anywhere in the WCS and oriented in any direction.

To specify points and distances using the keyboard you can use the following formats:

- Cartesian coordinates: x,y,z
- Cylindrical coordinates: R<alpha,z
- Spherical coordinates: R<alpha<beta

Relative Coordinates

If you place the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called: *Relative Coordinates*.

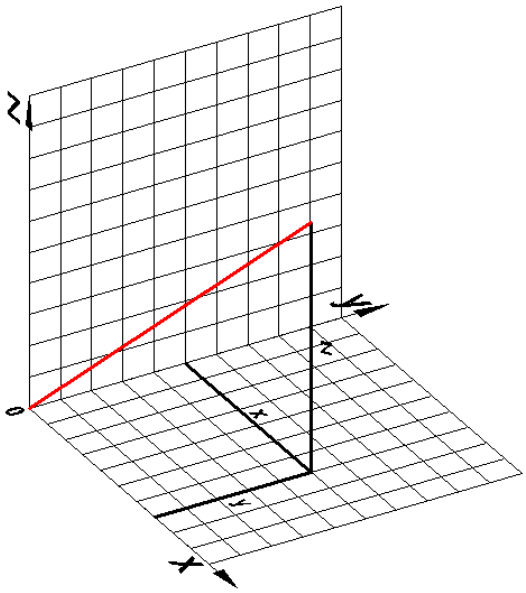
When Dynamic Dimensions are active and you type a value in the *Length* field and add a comma (,) the content of the *Length* field is copied to the command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates with respect to the previous point.

Absolute Coordinates

When a user defined coordinate system is active you can enter absolute coordinates (World coordinates) if you place an asterisk (*) in front. E.g. *0,0 refers to the origin of the WCS (World Coordinate System).

Cartesian Coordinates

In the Cartesian coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined by its distances to the yz-, xz- and xy- planes. These distances are called the xyz-coordinates of a point.



If you want to enter the absolute Cartesian coordinates of a point, type the x-, y- and z-coordinates separated by commas: 45.5,57.3,60

If you omit the z-coordinate, the point is placed in the xy-plane (Z = 0).

If you place the @-character in front of the entry (@x,y), the coordinates are calculated with respect to the previous point. This technique is called *Relative Cartesian coordinates*.

Using relative Cartesian coordinates to draw a rectangle

1. Launch the RECTANGLE command.
2. Specify the first corner of the rectangle.
3. In the command bar type: @<width>,<height>
   - <width> = the width of the rectangle in drawing units, measured along the x-axis
   - <height> = the height of the rectangle in drawing units, measured along the y-axis

Cylindrical Coordinates

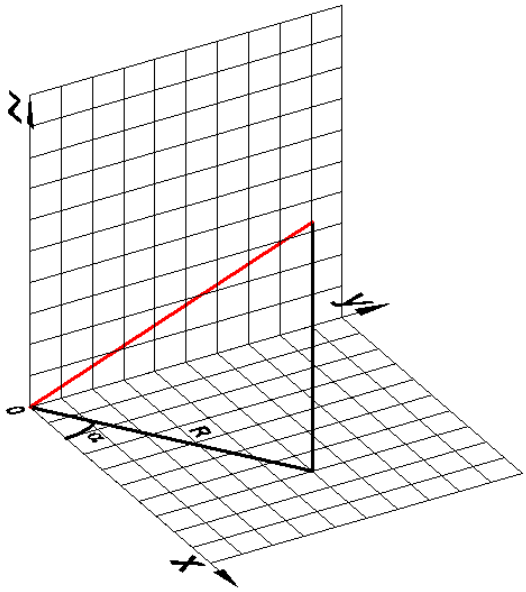
In a cylindrical coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes.

A point is defined using the following format: R<alpha, z.

- R = distance to the origin in the xy-plane
- <alpha = the angle between R and the x-axis (positive angles are measured counter clockwise)
- z = the height above the xy-plane.

If the z-coordinate is omitted, cylindrical coordinates are referred to as polar coordinates.

If you place the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called *Relative Cylindrical coordinates*.

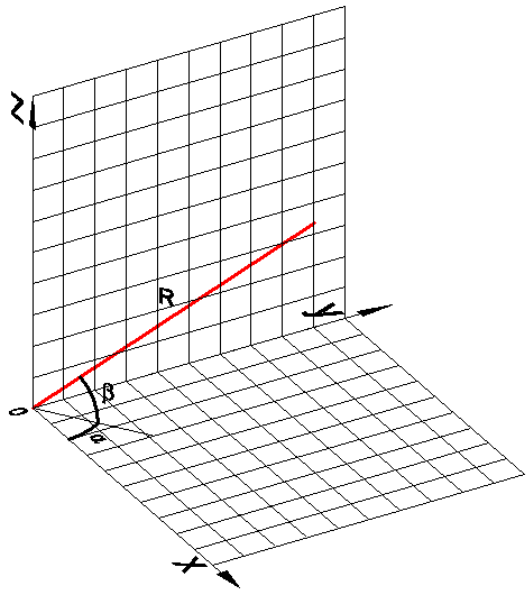


Spherical Coordinates

In a cylindrical coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes.

A point is defined using the following format: R<alpha<beta

- R = distance from the origin
- <alpha = angle in the xy-plane (positive angles are measured counter clockwise)
- <beta = angle measured from the xy-plane (positive angles are measured counter clockwise, above the xy-plane)



Snap and Grid

TUTORIALS

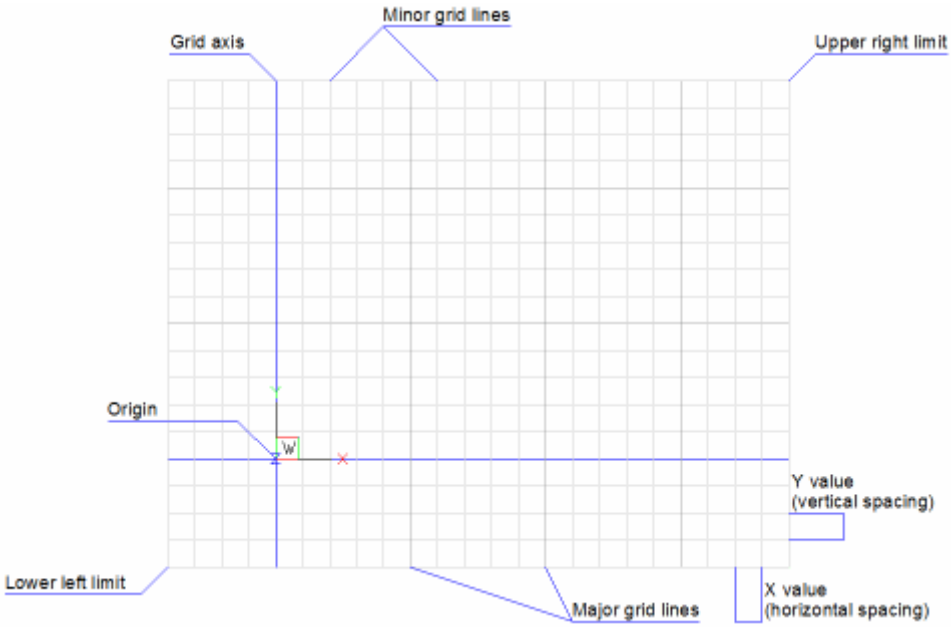
Snap and Grid

Grid and snap help you to draw fast and accurately.

A *grid* is a set of evenly spaced lines or dots that serve as a visual distance reference.

The GRIDSTYLE system variable controls whether grid lines or dots display.

The GRIDDISPLAY system variable controls the display of the grid. The grid can also indicate how far the drawing limits extend.



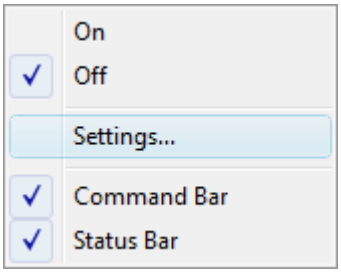
The *snap* feature creates a set of evenly spaced, invisible magnetic points, which make the crosshairs move in even increments. Both grid and snap are like the intersection points of the lines on a piece of grid paper. Grid points are for visual reference only and they do not print. *Snap* constrains the points that you can pick with the mouse.

- Click the *Grid* tool ( ) on the *Settings* toolbar or the *Grid* field in the Status Bar, or press *Ctrl+G* to toggle grid display on/off.
- Click the *Snap* tool ( ) on the *Settings* toolbar or the *Snap* field in the Status Bar, or press *Ctrl+B* to toggle snap on/off.
- Both *snap* and *grid* can be set differently in each viewport.
- The *Snap* option of the *Grid* command equals the grid spacing to the current snap spacing.
- Grid lines or dots do not print.

Snap and Grid Settings

Do one of the following:

- In the Status Bar, right click on the *Snap* field, then choose *Settings* in the context menu.



- Open the Settings dialog, then click the *Drawings* button.
  Under *Drafting*, expand *Coordinate Input* and *Snap/Grid*.

To Synchronize Snap and Grid Spacing

1. Type *Grid* in the command bar, then press Enter.
   You are prompted: Grid is on: OFF/Aspect/<Grid spacing (synced with snap)>:
2. Type *s*, then press Enter.
3. Type *Snap* in the command bar, then press Enter.
   You are prompted: Snap is on, x and y = 0.2000: OFF/Rotate/Style/Aspect/<Snap spacing>:
4. Type a new value for the snap spacing, then press Enter.
   The value is applied to both snap and grid.

NOTE The grid spacing remains in sync with the snap spacing until you set the grid spacing differently using the Grid command or in the Settings dialog.

To set the Drawing Limits

Click the *Drawing Limits* tool button ( ) on the *Settings* toolbar to toggle the LIMCHECK system variable on/off.

If LIMCHECK = ON, it is not possible to draw outside the drawing limits.

The appearance (pressed or unpressed) of the *Drawing Limits* tool button reflects the current status of the *Drawing Limits.*

The *Display beyond LIMITS area* option of the GRIDDISPLAY system variable defines wether or not the grid display indicates the drawing limits.

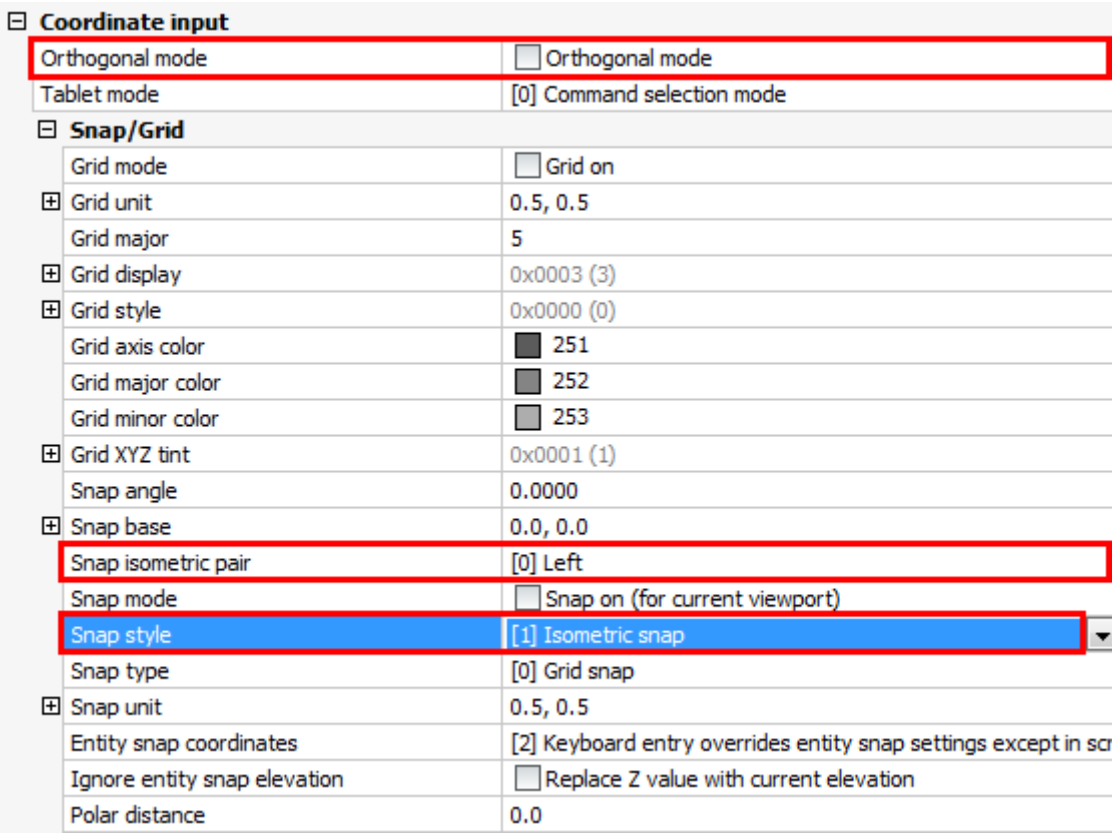
Using Isometric Snap

You can use the Isometric snap and grid option to create two-dimensional isometric drawings. With the isometric option, you can draw a simulated three-dimensional view on a two-dimensional plane, much the same as you might draw on a piece of paper. Do not confuse isometric drawings with three-dimensional drawings.

The isometric option always uses one of three preset planes, which are denoted as *Left*, *Right* and *Top*. You cannot alter the arrangement of these planes. If the Snap Angle is 0, the three isometric axes are 30 degrees, 90 degrees, and 150 degrees.

When you set the *Snap Style* setting to *Isometric Snap* and then set the *Snap Isometric Pair* setting to *Left*, *Top* or *Right*, the snap intervals, grid, and crosshairs align with the selected plane. The grid is always shown as isometric and uses y-coordinates to calculate the grid spacing. If the Orthogonal Mode is active, the movement of the crosshairs is constrained to the current isometric plane. Press the *F5* function key to toggle the *Snap Isometric Pair* setting.

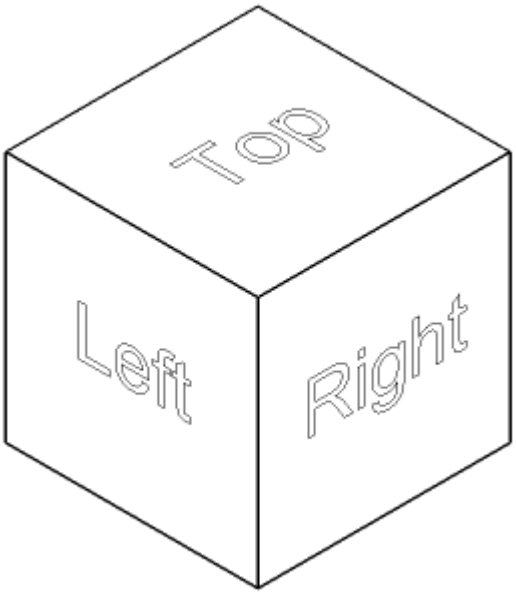
1. Display the *Snap/Grid* settings in the Settings dialog:



Coordinate input	
Orthogonal mode	Orthogonal mode
Tablet mode	[0] Command selection mode
Snap/Grid	
Grid mode	Grid on
Grid unit	0.5, 0.5
Grid major	5
Grid display	0x0003 (3)
Grid style	0x0000 (0)
Grid axis color	251
Grid major color	252
Grid minor color	253
Grid XYZ tint	0x0001 (1)
Snap angle	0.0000
Snap base	0.0, 0.0
Snap isometric pair	[0] Left
Snap mode	Snap on (for current viewport)
Snap style	[1] Isometric snap
Snap type	[0] Grid snap
Snap unit	0.5, 0.5
Entity snap coordinates	[2] Keyboard entry overrides entity snap settings except in scr
Ignore entity snap elevation	Replace Z value with current elevation
Polar distance	0.0

2. Turn on the *Orthogonal Mode*.
3. Use the SNAP command to set the *Snap Style* setting to *Isometric Snap*.

4. Press the *F5* function key to set the appropriate drawing plane: *Top*, *Left* or *Right*.



Setting the Snap Angle

You can rotate a dotted grid using the Snap Angle setting (if *Snap Angle* is different from zero, grid lines do not display).

1. Display the *Snap/Grid* settings in the Settings dialog:
2. Select the *Snap Angle* setting.
3. Type a new value in the *Snap Angle* setting field.
4. Close the *Settings* dialog.

NOTE You can also set the *Snap Angle* by typing *snapang* in the command bar.

When in a command, type *'snapang* (with an apostrophe in front), to set the *Snap Angle* variable transparently (= without interrupting the running command).

Entity Snaps

Commands: 2DINTERSECTION, 3DINTERSECTION, NEAREST, CENTER, ENDPOINT, EXTENSION, FROM, GCENTER, INSERTION, INTERSECTION, MIDPOINT, MTP, NONE, PARALLEL, PERPENDICULAR, POINT, QUADRANT, TANGENT

Entity snaps enable you to quickly select exact geometric points on existing entities without having to know the exact coordinates of those points. With entity snaps, you can select the end point of a line or arc, the center point of a circle, the intersection of any two entities, or any other geometrically significant position. You can also use entity snaps to draw entities that are tangent or perpendicular to an existing entity. You can use entity snaps any time you need to specify a point.

You can work with entity snaps in one of two ways

- Enable a running entity snap that remains in effect until you turn it off by choosing an entity snap when no other command is active.
- Enable a one-time entity snap for a single selection by choosing an entity snap when another command is active. You can also use a one-time entity snap to override a running entity snap.

When using entity snaps, the program recognizes only visible entities or visible portions of entities. You cannot snap to entities on layers that have been turned off or to the blank portions of dashed lines.

If the *APBOX* system variable is on, the entity *Snap Aperture Box* is added to the crosshairs when one or more entity snaps are active.

When you move the cross hairs, the program snaps to the snap point closest to the center of the *Snap Aperture Box*. The *Snap Marker* indicates the current snap point.

The DRAGSNAP system variable controls the snap behavior while 'dragging', providing an increased WYSIWIG experience. DRAGSNAP controls whether rubberband dynamics are displayed at the current cursor location or at the current entity snap location. DRAGSNAP applies to all modify commands that display dynamics, such as COPY, PASTECLIP, PASTEBLOCK, MOVE, ROTATE, MIRROR, SCALE and STRETCH.

- DRAGSNAP = 0 (Default),:dragged entities display at the cursor location.
- DRAGSNAP = 1: dragged entities display at the current entity snap location.

The OSOPTIONS system variable suppresses entity snaps on certain entity types. The value is stored as a bitcode using the sum of the values of the selected options.

- 1: Entity snapping is disabled for hatch entities.
- 2: Entity snapping is disabled for geometry with negative Z values when using a dynamic UCS.
- 4: Entity snapping is disabled for endpoints of dimension extension lines.

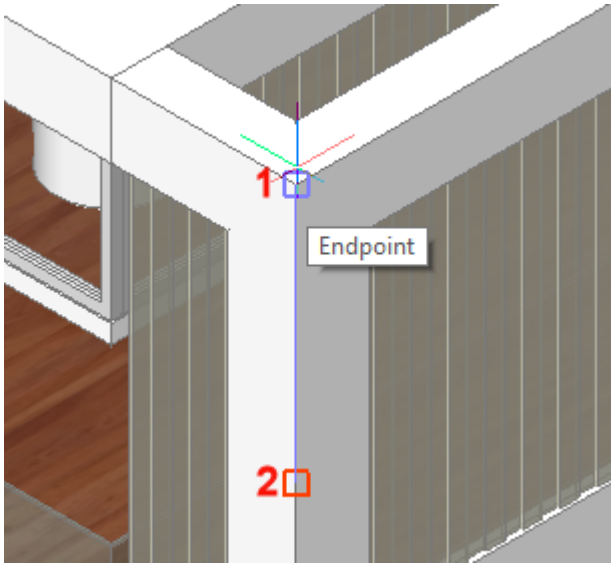
NOTE Press the *TAB* key, without moving the mouse, to cycle through all possible entity snaps.

If the *DYNMODE* variable has a positive value (Dynamic Dimensions switched ON) entity snap cycling is not possible. Click the *DYNMODE* field in the Status Bar to toggle the display of dynamic dimensions.

To Define the Entity Snap Settings

1. Open the Settings dialog.
2. In the *Settings* dialog, expand the *Program Options* settings class.
3. Under *Program Options* expand the *Display* settings group, then scroll down to the *Entity Snap* settings.

Snap marker size	6
Snap marker thickness	2
Snap marker color	20
3d snap marker color	Blue
Auto tracking vector color	171
Snap marker in all views	Display snap marker in all views
Locked layer fade control	50
Xref database fade control	70
Reference editing fade control	50
Snap tooltips	Enable snap tooltips

User Preference	Title	Description
SnapMarkerSize	Snap marker size	Sets the size of the snap marker. (default size is 6)
SnapMarkerThickness	Snap marker thickness	Sets the thickness of the snap marker. (default thickness is 2)
SnapMarkerColor	Snap marker color	Sets the color of the snap marker. (default color = 20)
3dSnapMarkerColor	3d snap marker color	Sets the color of the 3d snap marker (default color = 5 (Blue)). The 3d snap marker (1) displays the identified 3d point, the snap marker (2) displays at the current Elevation height. The 3d snap marker displays only if OSNAPZ = 1 (Ignore entity snap elevation). 

DisplaySnapMarkerInAllViews	Snap marker in all views	If multiple viewports are open, enables the display of the snap marker in all viewports.
DisplayTooltips	Snap tooltips	Enables the display of the Entity Snap tooltips.

4. Under *Drawing* expand the *Drafting* settings group then go to *Coordinate Input* > *Entity Snaps*.

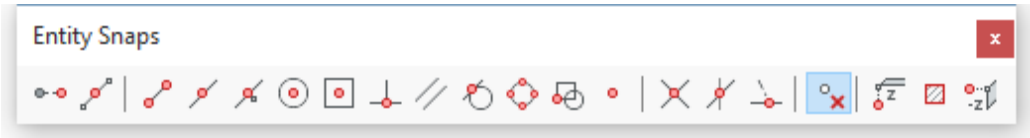
Entity snaps	
Entity snap mode	0x0003 (3)
Entity 3d snap mode	0x0000 (0)
Entity snap coordinates	[2] Keyboard entry overrides entity snap settings except in scripts
Ignore entity snap elevation	☐ Replace Z value with current elevation
Entity snap options	0x0007 (7)
1	☑ Entity snap ignores hatches
2	☑ Entity snap ignores negative Z values in Dynamic UCS mode
4	☑ Entity snap ignores end points of dimension extension lines
Entity snap aperture box	☑ Display Aperture box
Entity snap aperture	10
Dgn entity snap	☑ Enable DGN entity snap
Dwf entity snap	☑ Enable DWF entity snap
Pdf entity snap	☑ Enable PDF entity snap

System Variable	Title	Description
OSMODE	Entity snap mode	Sets the entity snap modes and toggles entity snap on/off.
3DOSMODE	3D entity snap mode	Sets the entity snap modes for 3D entity features such as the midpoint of an edge or the center of a face.
OSNAPCOORD	Entity snap coordinates	Determines whether entity snaps override keyboard coordinate entry.
OSNAPZ	Ignore entity snap elevation	Replaces the Z-coordinate of the snapping point with the current value of the ELEVATION system variable.
OSOPTIONS	Entity snap options	Suppresses entity snaps on hatches and/or dimension extension lines or negative z-values in dynamic UCS mode.
APBOX	Entity snap aperture box	Toggles the display of the aperture box (2).
APERTURE	Entity snap aperture	Sets the size of the Autosnap aperture box. (default size is 10).
DGNOSNAP	DGN entity snap	Enables entity snapping to entities in DGN underlay files.
DWFOSNAP	DWF entity snap	Enables entity snapping to entities in DWF underlay files.
PDFOSNAP	PDF entity snap	Enables snapping to the geometry in PDF underlay files.

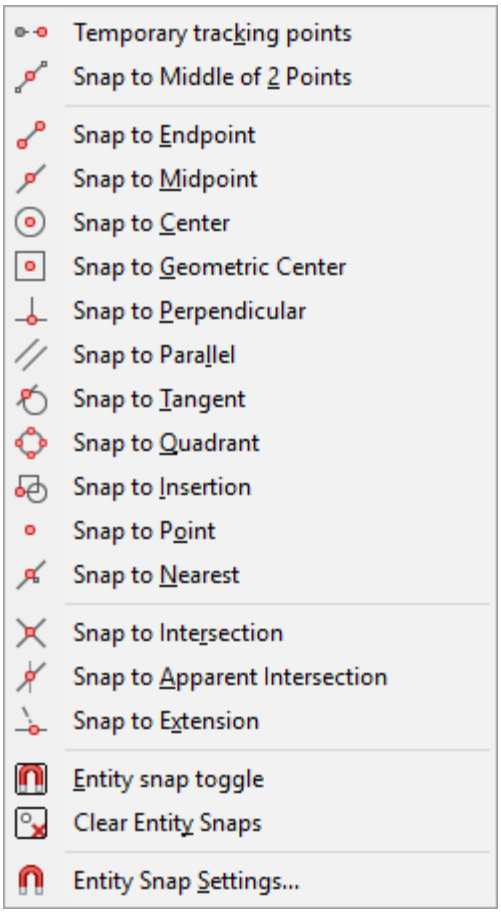
To Set the Entity Snaps

Do one of the following:

- Click the buttons on the *Entity Snaps* toolbar.
  The buttons of the currently active entity snap modes are pressed.



- Press and hold the Shift key, then right click and select the Entity Snap mode in the context menu.
  The icons of the currently active Entity Snap modes are outlined.



When no command is active, the above procedures toggle the Entity Snap modes on/off.

When a command is active, the above procedures set a 'one shot' override of the current Entity Snap modes.

Alternatively you can type a single letter shortcut as indicated in the table below.

Entity Snap Modes

Name	Icon	Marker	Context menu shortcut*	Transparent shortcut**	Description
TK				tk	Use temporary tracking points. Prompts you to identify temporary points; press Enter to accept the last temporary point.
MTP			2	mtp	Snaps to a point in the middle between two points. Prompts you to identify two points.
ENDPOINT			E	end	Snaps to the nearest endpoint of an entity or polyline segment.
MIDPOINT			M	mid	Snaps to the midpoint of an entity or polyline segment.
NEAREST			N	nea	Snaps to the nearest point on an entity.
CENTER			C	cen	Snaps to the center point of an arc, circle, polygon, ellipse or elliptical arc. Snaps to the center of gravity of a closed polyline.
GEOMETRIC CENTER			G	gce	Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids.
PERPENDICULAR			P	per	Snaps to the perpendicular point of another entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline or edge of a plane to form a perpendicular alignment with that entity or with an extension of that entity.
PARALLEL			L	par	Displays a tracking line parallel to the selected entity.
TANGENT			T	tan	Snaps to the point on an arc, ellipse, spline or circle that, when connected to the previous point, forms a line tangent to that entity.
QUADRANT			Q	qua	Snaps to the closest quadrant of an arc, circle, ellipse, or elliptical arc.
INSERTION			I	ins	Snaps to the insertion point of an attribute, block or text entity.

POINT			O	poi nod	Snaps to a point entity.
INTERSECTION			R	int	Snaps to the intersection of any combination of entities.
APPARENT INTERSECTION			A	app	Snaps to the apparent intersection in the current view of two entities that do not intersect in three-dimensional space.
EXTENSION		+	X	ext	Snaps to the extension of an entity or to the intersection of the extension of two entities.
NONE			Y	non	Turns off all entity snap modes.

* Type this letter to quickly select the entity snap mode in the right click context menu.

** Type the transparent shortcut in the command bar when a command prompts for a point. Other running entity snaps are temporarily disabled.

System Variable	Icon	Description
OSNAPZ		Ignore entity snap elevation: replaces the Z coordinate of the snapped point with the current value of the ELEVATION system variable.
OSOPTIONS		Entity snap to hatches: allows to snap to hatches.
OSOPTIONS		Entity snap to negative-z: allows entity snaps with a negative z-value when a Dynamic UCS is active,

NOTE You can toggle the *Entity Snaps* on/off:

- by clicking the *ESNAP* field in the Status Bar.
- by hitting the *F3* function key.

Working with Multiple Entity Snap Modes

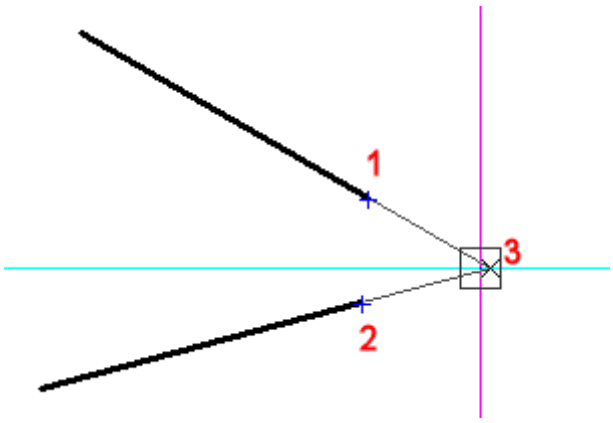
1. Move the cursor to the entity you want to snap.
   One of the active Entity Snap modes markers display.
2. Do not move the mouse; press the TAB key.
   The entity the snap point is on highlights.
3. Do one of the following:
   - Click to accept the snap point.
   - Press the TAB key.
     The next possible Entity Snap mode marker displays.
     The entity the snap point is on highlights.
4. (option) Repeat step 3 until the right snap point is found.

NOTES

- Do not move the mouse while pressing the TAB key.
- Entity snap cycling is not possible when Dynamic Dimensions are active. Click the *DYNMODE* field in the Status Bar to toggle Dynamic Dimensions on/off.

To Snap to the Extension of Two Entities

1. If not already on, turn on the *Extension* Entity Snap mode ( ).
2. Launch a drawing tool, e.g. Draw Line.
3. Move the cursor over the endpoint of the first entity (1).
   A small cross ( ) indicates the entity is marked for extension.
4. Move the cursor over the endpoint of the second entity (2).
   A small cross ( ) indicates the entity is marked for extension.
5. Move the cursor near the intersection of the extensions of the two entities.
   An X indicates the intersection (3).



6. Click to accept the snap point.

NOTE You can snap to the extension of lines, polylines, arcs and elliptical arcs

Using Temporary Tracking Points

When using temporary tracking points, Polar Tracking must be on and it is recommended to have dynamic dimensions active as well.

1. Whenever a command prompts for a point, do one of the following:
   - Click the mouse wheel.
   - Click the *Temporay tracking points* tool button ( ) on the *Entity Snaps* toolbar.
   - Hold down the Shift key, then right click and choose *Temporay tracking points* in the context menu.
   - Type *TK* in the command bar, then press Enter.

The command is suspended temporarily.

You are prompted: Specify first temporary tracking point:

2. Enter a point, e.g. use an entity snap to specify a point.
   You are prompted: Specify next temporary tracking point (press ENTER to end)
3. Do one of the following:

   - Enter a point.
   - Press Enter to accept the last temporary tracking.

The command resumes.

Using 'Middle of 2 Points' Snap

1. When prompted to specify a point, do one of the following:
   - Click the *Snap to the middle of 2 points* button ( ) on the *Entity Snaps* toolbar.
   - Type *mtp* or *m2p* in the command bar, then press Enter.
   - Press and hold the Shift key, then right click and choose *Snap to middle of 2 points* in the context menu.

You are prompted: First point of mid:

Specify the first point
You are prompted: Second point of mid:

Specify the second point.
The point in the middle of the two points is snapped.

Using Parallel Entity Snap

1. Make sure Entity Snap Tracking is active.
2. Launch an entity create command to create a linear entity (e.g. Line).
3. Specify the first point of the entity.
   The line displays dynamically.
4. Do one of the following:

   - Press and hold the Shift key, then right click and choose *Snap to parallel* in the context menu.
   - Click the *Snap to parallel* tool button ( ) in the *Entity Snaps* toolbar.

5. Hover the cursor over the entity you want to draw parallel to.

   The *Parallel Entity Snap* marker ( ) displays on the selected entity.
   Wait until a small cross (+) indicates the entity is recognized.
6. Move the cursor in order to place the dynamics of the line more or less parallel to the selected entity.
   A tracking line displays from the start point of the line.
7. Do one of the following to specify the second point while the tracking line displays:

   - Click a point.
   - Enter the desired length and press Enter.
   - Use a second entity snap tracking point, then move the cursor to the intersection of the parallel tracking line and the second entity snap tracking line.

Polar Tracking

Polar tracking assists you to draw at exact angles:

- at fixed intervals, starting from 3 O'clock (or East)
- at specific additional angles

A polar tracking path line displays from the origin point when you move the cursor close to one of the polar tracking angles. The tracking path lines display as long as the *Snap Aperture* box overlaps the polar tracking line.

In 3D views an additional tracking line displays parallel to the Z-axis of the current UCS.

Snap track lock: to lock a snap track path hit the Shift key. To unlock a track path, hit Shift again: it works as a toggle. Locked tracking paths display in the color defined by the SnapMarkerColor user preference..

NOTE The display of the tracking path line is controlled through the *TRACKPATH* system variable.

The color of the tracking path line is controlled through the *AutoTrackingVecColor* user preference.

Tracking paths parallel to the X-, Y- or Z-axis of the current coordinate system (WCS or UCS) display in the color of the axis they are parallel with (defined by *COLORX*, *COLORY* or *COLORZ* variables)..

To Define the Polar Tracking Settings

1. Do one of the following:
   2. Right click on the *POLAR* field in the Status Bar, then right click and choose *Settings* in the context menu.
   3. Open the *Settings* dialog, then expand the *Coordinate input* sub-category under *Drafting* in the *Drawing* settings class.
      Expand the *Snap Tracking* settings group.

Snap tracking	
AutoSnap	0x003F (63)
1	☑ AutoSnap marker
2	☑ AutoSnap tooltips
4	☑ AutoSnap magnet
8	☑ Polar tracking
16	☑ Entity snap tracking
32	☑ Tooltips for polar tracking and entity snap tracking
Polar angle	45.0
Polar add angles	30;60
Polar mode	0x000F (15)
1	☑ Relative
2	☑ Use polar tracking settings in entity snap tracking
4	☑ Use additional polar tracking angles
8	☑ Press SHIFT to acquire entity snap tracking points
Track path	[0] Display full-screen entity snap tracking path

   2. Adjust the settings:
   3. 1: Autosnap marker (not used, required for AutoCad® compatibility only)
   4. 2: Autosnap tooltips (not used, required for AutoCad® compatibility only)
   5. 4: Autosnap magnet (not used, required for AutoCad® compatibility only)
   6. 8: If checked, *Polar Tracking* is on.
   7. 16: If checked, *Object Snap Tracking* is on.
   8. 32: If checked, tooltips display for polar tracking and entity snap tracking.
   9. Polar angle: Sets the polar angle increment.

10. Polar add angles: Sets the additional polar angles. Separate angles by semicolons.
11. 1: If checked, polar angles are measured relatively from selected entities.
12. 2: If checked, the polar angles (interval and additional angles) are also used in entity snap tracking.
13. 4: If checked, the additional angles for polar tracking are used.
14. 8: If checked, press and hold the Shift key to acquire entity tracking points.

- Track path: Controls the display of the polar and entity snap tracking paths.

Display full-screen object snap tracking path
Display object snap tracking path only between the alignment point and the From point to the cursor location
Do not display polar tracking path
Do not display polar or object snap tracking paths

Using Polar Tracking to Draw a Line

1. (option) Check the *Polar Tracking* settings.
2. (option) If not already on, click the *POLAR* field in the Status Bar.
3. Launch the *Draw Line* tool.
4. Specify the start point of the line.
   Polar tracking lines display at the specified intervals and additional angles.

If the cursor is on or next to a tracking line:

- The current distance and angle from the start point display next to the cursor when
- A cross marker (X) indicates the current point.

5. Do one of the following to define the endpoint of the line.
   When the appropriate tracking line displays:
6. Click when the cursor is at the desired position.
7. Type the length of the line in the command bar and press Enter.

NOTE Turning on *Polar Tracking* automatically disables Orthogonal Mode.

Using Snap Track Lock

Hit the Shift key while a snap track marker line is displayed to lock a tracking path, any (snap)point that is picked will be projected perpendicularly onto the tracking path. Hit the Shift key again to unlock.

A locked tracking path changes to the color defined by the SnapMarkerColor user preference.

Using Temporary Tracking Points

When *Polar Tracking* is ON, temporary tracking points can be used to pick points at locations where no existing geometry is available to snap directly to, while you know the desired offset distance(s) from snap points on existing geometry in the neighborhood.

TT option: whenever prompted to enter a point, type *TT* on the command line, then press Enter to define a single temporary tracking point. After the tracking point has been picked, normal command execution continues. Repeat typing *TT* to enter additional temporary tracking points.

TK option: whenever prompted to enter a point, type *TK* on the command line, then press Enter to define one or more temporary tracking points. When eventually the desired location has been reached, enter the point by pressing the Right Mouse button or the Enter key.

Using the TT option

1. Launch a draw command, e.g. Line.
   You are prompted: ENTER to use last point/Follow/<Start of line>:
2. Type *TT* in the command bar, then right click or press Enter.
   You are prompted: Specify temporary tracking point:
3. Use an Entity Snap to pick a point on existing geometry.
   A small red cross indicates the temporary tracking point.

4. Move the cursor in the desired direction.
   Depending on the value of the POLARANG and POLARADDANG system variables tracking lines display from the first temporary tracking point.
5. When a tracking line in the appropriate direction displays, type the distance from the temporary tracking point.
6. Specify the endpoint of the line.

Using the TK option

1. Launch a draw command, e.g. Line.
   You are prompted: ENTER to use last point/Follow/<Start of line>:
2. Type *TK* in the command bar, then right click or press Enter.
   You are prompted: Specify first temporary tracking point:
3. Use an Entity Snap to pick a point on existing geometry.
   A small red cross indicates the temporary tracking point.
   You are prompted: Specify next temporary tracking point (press ENTER to end):
4. Move the cursor in the desired direction.
   Depending on the value of the POLARANG and POLARADDANG system variables tracking lines display from the first temporary tracking point.
5. When a tracking line in the appropriate direction displays, type the distance from the temporary tracking point.
   A small red cross indicates the second temporary tracking point.
   You are prompted: Specify next temporary tracking point (press ENTER to end):
6. Do one of the following:

   Right click or press Enter to accept the second tracking point as the start point of the line.

   Repeat steps 4 and 5 until the desired point is specified, then right click or press Enter to accept the point.
7. Specify the endpoint of the line.

Snap Tracking

Entity Snap Tracking assists you to draw entities with respect to other entities.

When *Entity Snap Tracking* is turned on, BricsCAD displays temporary alignments based on entity snap points. *Entity Snap Tracking* works in combination with Entity Snap tools. At least one *Entity Snap* mode must be on if you want to use *Entity Snap Tracking.*

Click the *STRACK* field in the Status bar to enable/disable *Entity Snap Tracking*.

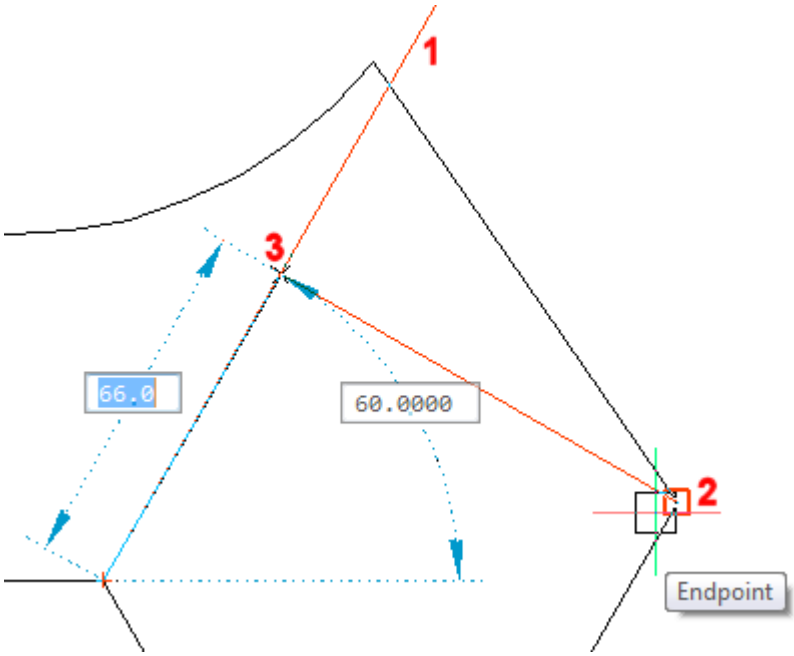
Entity Snap Tracking alignments are parallel to the X- and Y-axis of the current UCS by default.

If the Polar mode > 0x0002 setting is checked, the polar angles used in Polar Tracking (interval and additional angles) are also used in *Entity Snap Tracking*.

Alignment points are acquired when you hover over a point, using entity snaps. Acquired points display a small plus sign (+). To remove an acquired point, hover over the point again. The small plus sign (+) is then removed. If the Polar mode > 0x0008 is checked, you must press and hold the Shift key to acquire entity tracking points. Alignments relative to an acquired point are displayed as you move the cursor over their tracking paths. You can then define points either on the alignments or at the intersection of two alignments.

Esnap track markers: to distinguish between regular esnap markers and markers drawn together with related snap tracking lines, esnap track markers display in the color of the tracking lines.

Hitting the Shift key when a tracking line displays, locks the tracking line. Locked tracking lines display in red (1). Entity snaps (2) or specified points are projected perpendicularly onto the tracking line (3). Hit the Shift key again to unlock a tracking line.

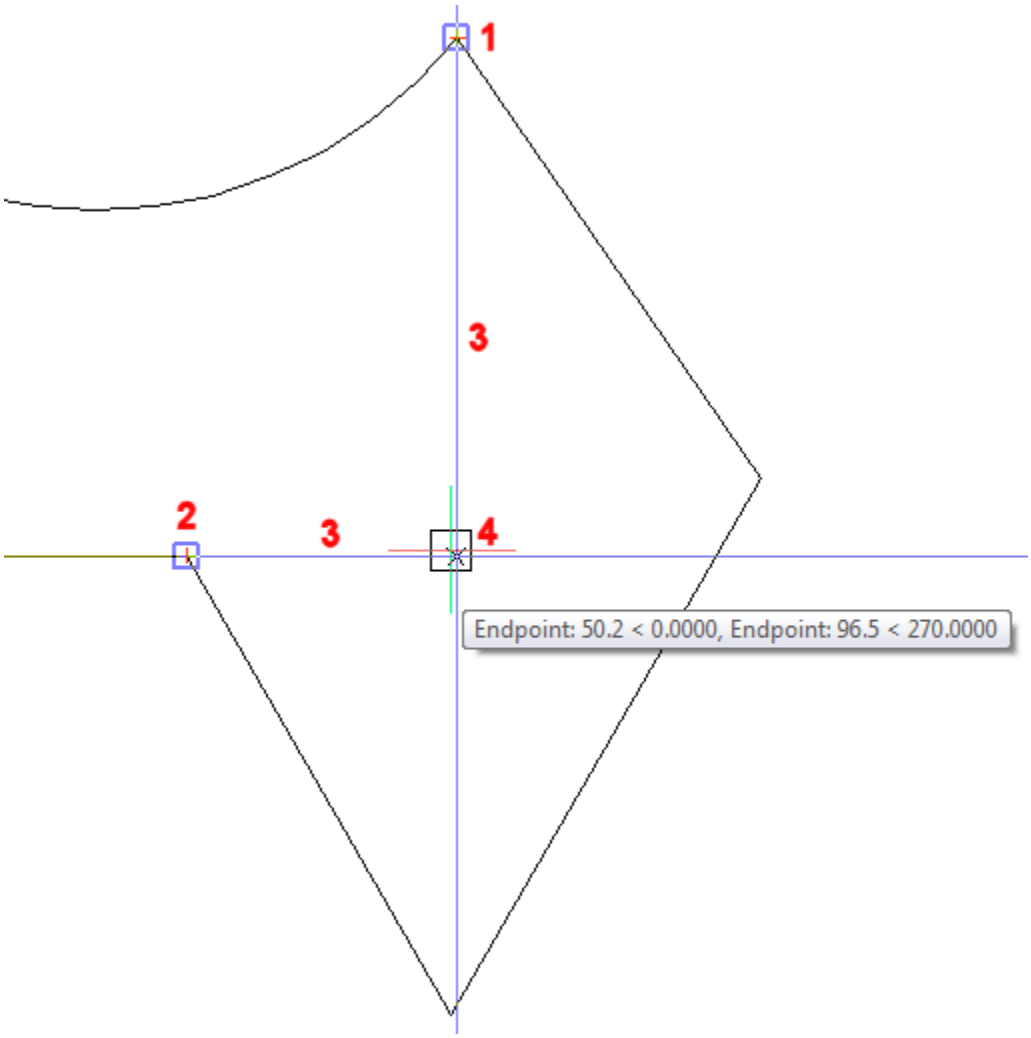


More options with locked tracking lines are:

- If the Intersection entity snap is active, it is possible to snap to the intersections of a locked tracking line and entities in the drawing.
- Acquire additional tracking lines and snap to the locations where these intersect with the locked tracking line.

Using Entity Snap Tracking

1. Launch a drawing tool, e.g. *Draw Line*.
2. Hover the cursor over the first tracking point (1).
   A small plus sign (+) displays to indicate the tracking point is active.
3. Hover the cursor over the second tracking point (2)
   A small plus sign (+) displays to indicate the tracking point is active.
4. Move the cursor close to position 4.
   - Tracking lines display from both tracking points (3).
   - A cross marker (X) indicates the intersection of the tracking lines.
   - Snap markers display at the tracking points.
   - An *Entity Snap Tracking* tooltip at the intersection of the tracking lines shows the angles and distances from the tracking points.



5. Click to accept the *Entity Snap Tracking* point.

User Coordinate Systems

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. Every drawing has a fixed coordinate system called the *World Coordinate System* (WCS).

You can also define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS) and can be located anywhere in the WCS and oriented in any direction.

To specify points and distances using the keyboard you can use the following formats:

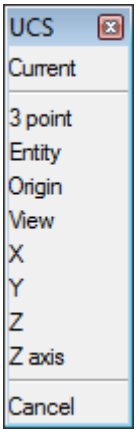
- Cartesian coordinates: x,y,z
- Cylindrical coordinates: R<alpha,z
- Spherical coordinates: R<alpha<beta

To Move the Origin of the Current Coordinate System

1. Type *ucs* at the command prompt.
   You are prompted: Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:
2. Identify the new origin, then press Enter to confirm.

To Define a User Coordinate System

1. Choose *Drawing Explorer > Coordinate Systems* in the *Tools* menu.
   The *Drawing Explorer - Coordinate Systems* dialog opens.
2. Click the *New* tool button ( ) on the *Drawing Explorer* dialog.
   The *BricsCAD Explorer* dialog closes.
   You are prompted: Current/Entity/Origin/View/X/Y/Z/ZAxis<3point>:
   A prompt menu displays:



2. Press Enter to define the UCS using points.
   You are prompted: New origin <current origin>:
3. Specify the origin point of the UCS.
   You are prompted: Point of positive X axis <current point>:
4. Specify a point to define the positive X-axis.
   The command window reads: Point in X-Y plane with positive Y value <current point>:
5. Specify a point to define the positive Y-axis.
   The UCS is defined.
   The *BricsCAD Explorer* dialog reopens.
6. Click in the *UCS Name* field of the newly defined UCS to replace the <NewUCS> default name.
7. Close the *BricsCAD Explorer* dialog.

Command Options

See the EXPUCS and UCS commands.

To Restore the WCS

1. Type *UCS* in the command bar, then press Enter.
   You are prompted:
   ?/3point/Delete/Entity/Origin/Previous/Restore/Save/View/X/Y/Z/Zaxis/<World>:
   A prompt menu displays.
2. Do one of the following:
   - Press Enter to accept the default option.
   - Choose *World* in the prompt menu.
   - Type *W* in the command bar and press Enter.

To Restore a UCS

1. Choose *Drawing Explorer > Coordinate Systems* in the *Tools* menu.
   The *Drawing Explorer - Coordinate Systems* dialog opens.

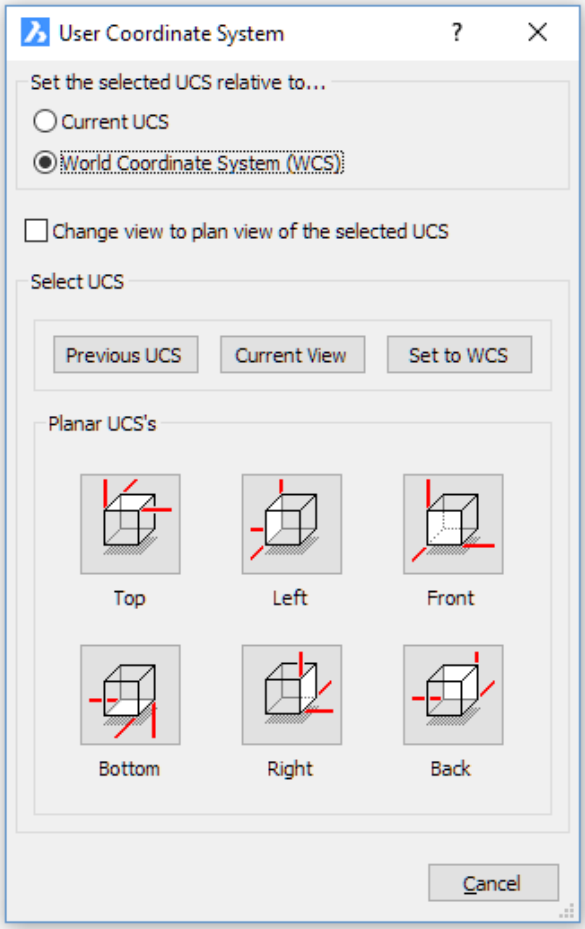
2. Click the blank tile in front of the UCS name to make it current.
3. Close the *BricsCAD Explorer* dialog.

NOTE If no UCS is marked, the WCS is the current coordinate system.

To Set a Relative UCS

1. Do one of the following:
   - Click the *UCS...* tool button ( ) on the *Settings* toolbar.
   - Choose *UCS...* in the *Settings* menu.

The *User Coordinate System* dialog opens.



Choose one of the following:

- Set the selected UCS relative to the *Current UCS*.
- Set the selected UCS relative to the *World coordinate system (WCS)*.

3. (option) Enable the *Change view to plan view of the selected UCS* option.
4. Click one of the *Planar UCS's* buttons.
   The User Coordinate System dialog closes.

NOTES

- If the UCSORTHO system variable is ON an orthographic UCS is restored automatically when the related orthographic view is restored by the -VIEW command or selected on the Look From control.
  An orthographic UCS has its origin at the WCS origin.
- The UCSVP (UCS viewports) system variable controls whether the UCS in a viewport is fixed (UCSVP = ON) or changes to match the UCS in the current viewport (UCSVP = Off). As a result, in a multiple viewport drawing, the UCS of all viewports that have UCSVP = Off matches the orthographic UCS of the current viewport if UCSORTHO = On.

Command Options

See the SETUCS command.

Dynamic UCS

With the Dynamic UCS feature enabled, the coordinate system is aligned automatically with a face of a 3D solid under the cursor when drawing entities.

This feature is controlled by the UCSDETECT system variable.

The RedHilite_DUCSLocked_Face_Color user preference controls the highlight color for the face to which the Dynamic UCS has been locked by pressing the Shift key.

The RedHilite_DUCSLocked_Face_Alpha user preference controls the highlight transparency for the face to which the Dynamic UCS has been locked by pressing the Shift key.

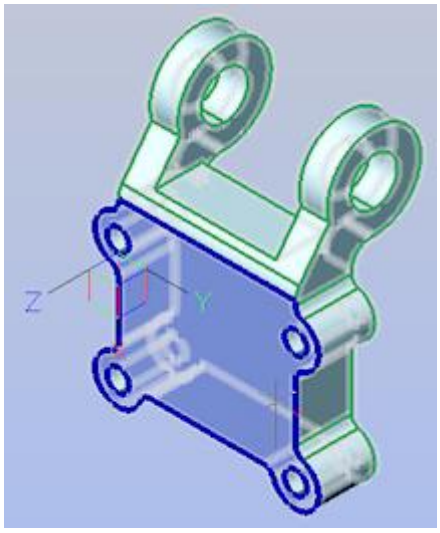
To toggle the dynamic UCS

Do one of the following:

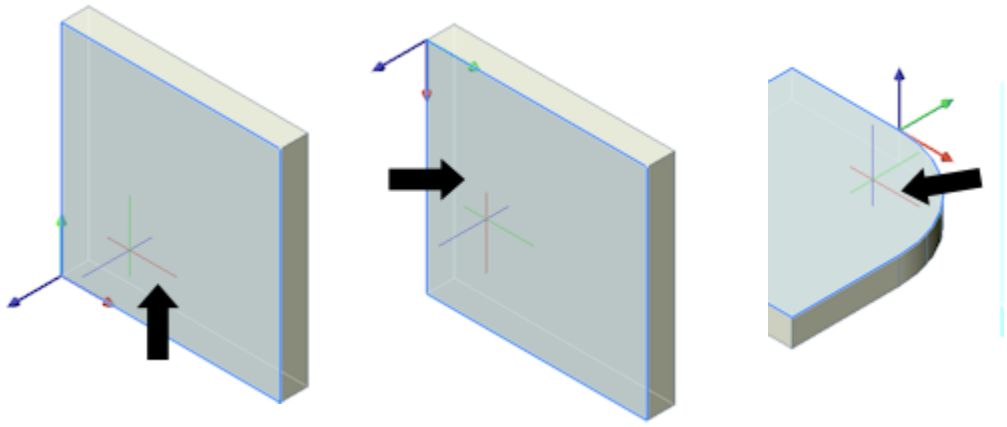
- Press the *F6* function key.
- Click the *DUCS* field in the status bar.

Using the Dynamic UCS

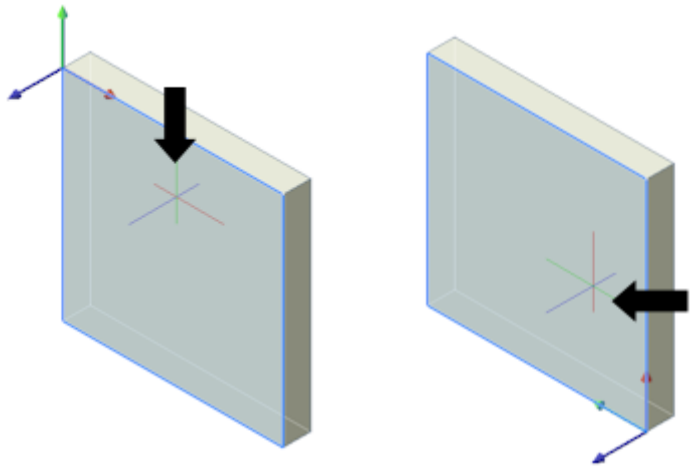
1. With the Dynamic UCS feature active, launch a Draw command.
2. Move the cursor over the face of a 3D solid.



- The face highlights.
- If ON, the grid is aligned with the selected face.
- If the UCSICON system variable = 3 (On at Origin), the UCS icon displays in the selected face.
- The dynamic UCS is oriented according to the edge by which the cursor 'entered' the hovered face, thus providing users a way to manipulate the orientation of the DUCS. The origin of the DUCS is set to the start point of this edge and the X-axis is chosen codirectional to the tangent at the start point of the edge.



Bottom Left Curved



Top Right

3. (option) Hit the Shift key to lock the DUCS, which allows you to start drawing outside the highlighted face.
   Hit the Shift key again to unlock the DUCS.
4. Pick a point to start drawing.
5. When the drawing command concludes, the WCS or previously defined UCS is restored.

Measuring

The *Dist* command reports the distance and angle between two points.

The *Area* command finds the area and perimeter (or length) of 2D entities.

The *Massprop* command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties").

The *ID* command reports the x,y,z coordinates of picked points.

The *List* command lists the properties of selected entities.

Measuring Distances

To measure a distance between two points:

1. Do one of the following:
   - Click the *Distance* tool button ( ) on the *Inquiry* toolbar.
   - Type *dist* in the command bar and press Enter.

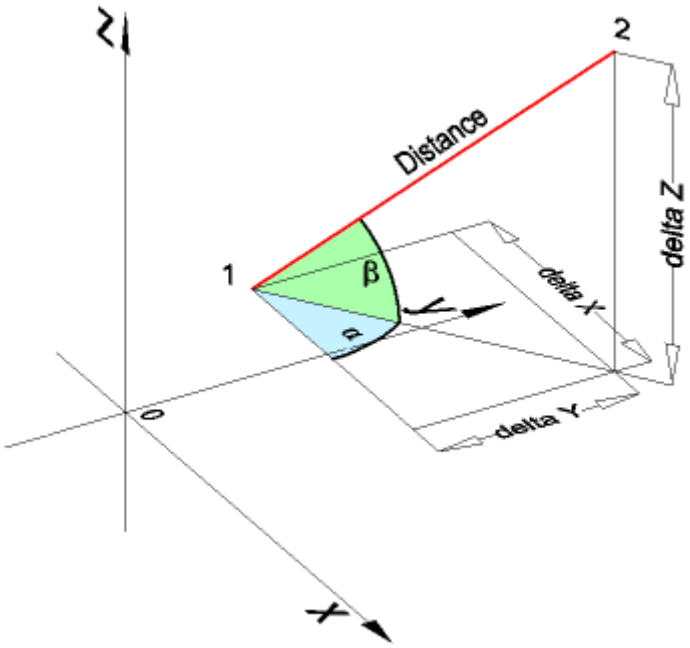
You are prompted: Starting point for distance:

2. Identify the first point.
   You are prompted: End point:
3. Identify the second point.
   If DRAGSNAP = 1, the distance displays in the dynamic dimension field.

The following information displays in the command bar (see image below):

Distance, Angle in XY Plane (alpha), Angle from XY Plane (beta),
Delta X, Delta Y, Delta Z.

The values are calculated with respect to the current coordinate system.



Measuring Lengths

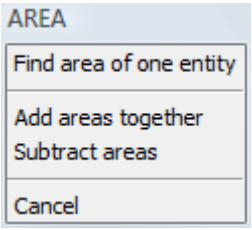
To measure the length of single linear entity, click the entity then read the *Length* field in the Properties Bar.

To add the lengths of a series of linear entities:

1. Do one of the following:
   - Click the *Area* tool button ( ) on the *Inquiry* toolbar.
   - Type area in the command bar and press Enter.

You are prompted: Entity/Add/Subtract/<First point>:

A prompt menu displays:

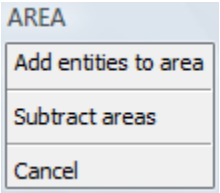


2. Do one of the following:

   - Choose *Add areas together* in the prompt menu.
   - Type *A* in the command bar and press Enter.

You are prompted: Adding: Entity/Subtract/<First point>:

A prompt menu displays:



4. Do one of the following:

   - Choose *Add entities to area* in the prompt menu.
   - Type *E* in the command bar and press Enter.

You are prompted: Adding area<Select entities>:

5. Click the first entity.
   The area and length of the first entity display in the command bar.
   You are prompted: Adding area<Select entities>:
6. Click the second entity.
   The area and length of the second entity and the total area and total length display in the command bar.
   Optionally press the *F2* function key to display the *Prompt History* window; press *F2* again to close the *Prompt History* window.
   You are prompted: Adding area<Select entities>:
7. Repeat step 5 to add more entities
   or
   press the *Esc* key to stop.

Measuring Areas

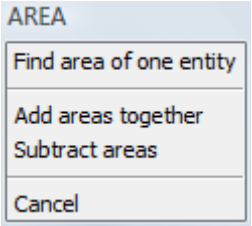
To measure the area and length of single closed entity, click the entity then read the *Area* and *Length* fields in the Properties Bar.

To measure areas and perimeters by specifying points in a drawing:

1. Do one of the following:
   - Click the *Area* tool button ( ) on the *Inquiry* toolbar.
   - Type area in the command bar and press Enter.

You are prompted: Entity/Add/Subtract/<First point>:

A prompt menu displays:

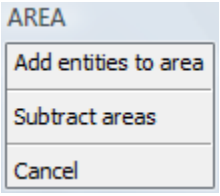


3. Do one of the following:

   - Choose *Add areas together* in the prompt menu.
   - Type *A* in the command bar and press Enter.

You are prompted: Adding: Entity/Subtract/<First point>:

A prompt menu displays:

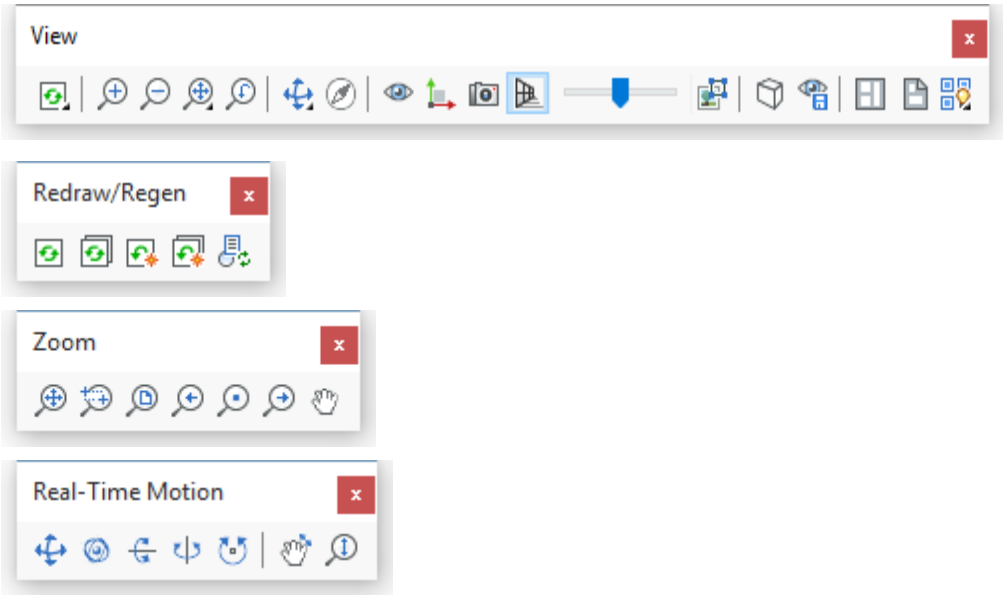


4. Click the first corner point of the area.
   A dashed line displays dynamically from the first corner point.
   You are prompted: Adding: <Next point>:
5. Click the second corner point of the area.
   A dashed polygon displays dynamically, indicating the currently defined area.
   You are prompted: Adding: <Next point>:
6. Click the third corner point of the area.
   You are prompted: Adding: <Next point>:
7. Repeat step 5 to add more corner points
   or
   Right click to stop adding corner points.
   The *Total length* and *Total area* display in the command bar.
   Optionally press the *F2* function key to display the *Prompt History* window; press *F2* again to close the *Prompt History* window.
8. Do one of the following:

   - Repeat steps 3 through 6 to add another area.
   - Press the *Esc* key to stop.

Viewing your Drawing

BricsCAD provides various tools to control the display and view orientation of your drawing. These tools can be found either in the *View* menu or the *View* toolbar or both.



View toolbar and its flyouts

The Look From control

Icon	Tool name	Command	Description
	Redraw	REDRAW	Refreshes the screen display in the current viewport.
	Redraw All	REDRAWALL	Refreshes the screen display in all open viewports.
	Regenerate	REGEN	Recalculates the screen display in the current viewport.
	Regenerate All	REGENALL	Recalculates the screen display in all open viewports.
	Update Fields	UPDATEFIELD	Updates the selected fields.
	Real Time Pan	RTPAN	Pans the view dynamically.
	Pan	PAN	Pans the view.
	Real Time Zoom	RTZOOM	Zooms in/out dynamically.
	Zoom In	ZOOM + I(*)	Zooms in on the center of the window by a factor of 2.
	Zoom Out	ZOOM + O(*)	Zooms out from the center of the window by a factor of 1/2.

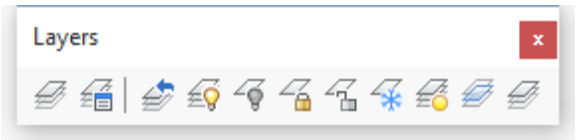
	Zoom Extents	ZOOM + E(*)	Displays all the entities in the drawing (referred to as the *drawing extents*).
	Zoom Window	ZOOM + W(*)	You are prompted to pick two corners of a box on the existing view in order to enlarge that area to fill the display.
	Zoom Previous	ZOOM + P(*)	Restores the displayed view prior to the current one.
	Zoom All	ZOOM + A(*)	Displays the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).
	Zoom Left	ZOOM + L(*)	Pick the lower left corner of the next view, then specify the magnification factor or the view height.
	Zoom Center	ZOOM + C(*)	Pick the center of the next view, then specify the magnification factor or the view height.
	Zoom Right	ZOOM + R(*)	Pick the upper right corner of the next view, then specify the magnification factor or the view height.
	Look Around	RTLOOK	Look around in a 3D scene.
	Look Up and Down	RTUPDOWN	Move up / down and left / right in a 3D scene.
	Walk	RTWALK	Walk left / right and forward / backward in a 3D scene.
	Real-Time Constrained Sphere	RTROT	Rotates the view dynamically in 3D. • Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the world z-axis. • Moving the mouse vertically (parallel to the screen y-axis) rotates the screen x-axis.
	Real-Time Sphere Center	RTROTCTR	Rotates the view dynamically in 3D about a user-defined center point. Behaves like RTROT.
	Real-Time Free Sphere	RTROTF	Rotates the view dynamically in 3D. • Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the screen y-axis. • Moving the mouse vertically (parallel to the screen y-axis) rotates the view about the screen x-axis.
	Real-Time X	RTROTX	Rotates the view dynamically about the screen x-axis.
	Real-Time Y	RTROTY	Rotates the view dynamically about the screen y-axis.
	Real-Time Z	RTROTZ	Rotates the view dynamically about the screen z-axis.
	Set View Point	DDVPOINT	Opens the *Preset Viewpoints* dialog.
	Plan View	PLAN	Restores the plan view with respect to the WCS, the current UCS or a saved UCS.
	Define View	DVIEW	Defines parallel and visual perspective views.

	Save/Restore view	VIEW	Saves and restores named views.
	Viewports	VPORTS	Manages viewports in model space.
	Paper Space Views	MVIEW	Creates viewports in paper space.
	Generate Drafting Views	BMGENDRAFT	Generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
	Generate Section	BMGENSECTION	Creates a cross section view based on a standard drafting view generated by the *BmGenDraft* command in a paper space layout.
	2D Context	2DCONTEXT	Switches the drawing to a 2D drafting context.
	3D Context	3DCONTEXT	Switches the drawing to a 3D modeling context.

(*) <command> <option>: type *zoom* followed by Enter, then type the command option letter, followed by Enter.

Layer Utilities

The layer utility commands help you to easily manipulate the *On/Off*, *Thaw/Freeze* and *Unlocked/Locked* layer properties by selecting entities. You can then save the result in a Layer State.

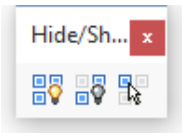


Command	Icon	Description
LAYON		Turns on all layers that have been turned off
LAYOFF		Turns off layers associated with selected entities.
LAYFRZ		Freezes layers associated with selected entities.
LAYTHW		Thaws all frozen layers.
LAYLCK		Locks layers associated with selected entities.
LAYULK		Unlocks layers associated with selected entities.
LAYISO		Isolates layers associated with selected entities. If no entity on the current layer is selected, the layer of the entity that is selected first temporarily becomes the current layer until the LAYUNISO command is executed.
LAYUNISO		Restores layers isolated by the LAYISO command.

See also Hide and Show Entities.

Hide and Show Entities

The tools on the *Hide/Show Entities* toolbar allow to temporarily hide or isolate a selection of entities.



Command	Icon	Description
UNISOLATEOBJECTS		Brings back all previously hidden entities.
HIDEOBJECTS		Hides the selected entities.
ISOLATEOBJECTS		Displays the selected entities only.

The OBJECTISOLATIONMODE system variable controls whether hiding and isolating entities is saved or not.

See also Layer Utilities.

Redrawing and Regenerating a Drawing

Commands: REGEN, REGENALL, REGENAUTO, REDRAW and REDRAWALL

The *Regen* command regenerates the current viewport (short for "regenerate").

The *Regenall* command regenerates all viewports (short for "regenerate all").

The *Regenauto* command controls when BricsCAD regenerates the drawing automatically.

The *Redraw* command redraws the current viewport to clean it up.

The *Redrawall* command redraws all viewports to clean them up.

As a matter of fact the screen display of a drawing is a simplified version of the drawing database. From time to time it is necessary to synchronise the screen display and the drawing database.

The *REGENMODE* system variable controls whether BricsCAD regenerates the drawing automatically. You can change the status of *REGENMODE* through the *RegenAuto* command. If *REGENMODE* is ON BricsCAD regenerates the display automatically, but in a few cases a forced regeneration of the drawing might be necessary. This is done by the *Regen* command.

Don't confuse the *Regen* command with the *Redraw* command, which simply repaints the screen, without attempting to synchronize the screen display with the drawing database.

Redraw All

Do one of the following:

- Click the *Redraw All* tool button ( ) on the *View* toolbar.
- Type *ra* or *redrawall* in the command bar, then press Enter.

All viewports are refreshed.

Regenerate All

Do one of the following:

- Click the *Regenall* tool button ( ) on the *View* toolbar.
- Type *rea* or *regenall* in the command bar, then press Enter.

All viewports are regenerated.

Regenerate

Do one of the following:

- Click the *Regen* tool button ( ) on the *View* toolbar.
- Choose *Regen* in the *View* menu.
- Type *re* or *regen* in the command bar, then press Enter.

The current viewport is regenerated.

Redraw

Do one of the following:

- Click the *Redraw* tool button ( ) on the *View* toolbar.
- Choose *Redraw* in the *View* menu.
- Type *r* or *redraw* in the command bar, then press Enter.

The current viewport is refreshed.

Panning

The *Pan* command moves the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space. The only change is the portion of the drawing being displayed. The cursor changes to a hand ( ) when a *Pan* tool is active.

The *Rtpan* command pans the drawing in real-time.

See also: View manipulation using the mouse.

Real Time Pan Command

1. To launch the Real-time Pan command do one of the following:
   - Click the *Real-Time Pan* tool button ( ) on the *Real-Time Motion* toolbar
   - Choose *Real-Time Motion* > *Real-Time Pan* in the *View* menu.
   - Type *rtpan* in the command bar, then press Enter.

You are prompted: >>ENTER, Right click or Esc to complete...

2. Press and hold the left mouse button to pan the view.
3. To abort the *Real-Time Pan* command, do one of the following:

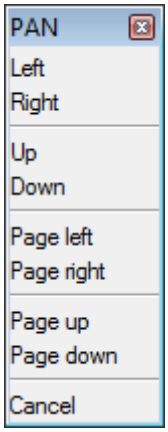
   - Right click.
   - On the keyboard, press Enter, space bar or Esc.

Pan Command

1. To launch the Pan command do one of the following:
   - Click the *Pan* tool button ( ) on the *View* toolbar (*Zoom* flyout).
   - Choose *Zoom* > *Pan* in the *View* menu.
   - Type *p* or *pan* in the command bar, then press Enter.

You are prompted: Left/Right/Up/Down/PGLeft/PGRight/PGUp/PGDown/<Pan base point>:

A prompt menu opens:



2. Specify the *Pan base point*.
   You are prompted: Pan displacement point:
3. Specify the *Pan displacement point*.
   The display shifts over the specified distance and in the specified direction.

Zooming

The *Zoom* command visually changes the size of the drawing within the current viewport.

The *Rtzoom* command zooms the drawing in real-time.

You can change the magnification of your drawing at any time by zooming. The cursor changes to a magnifying glass ( ) when a *Zoom* tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail. Changing the magnification of the drawing affects only the way the drawing is displayed; it has no effect on the dimensions of the entities in your drawing.

See also: View manipulation using the mouse.

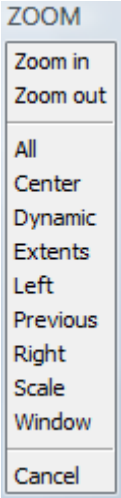
Zoom Command

Do one of the following:

- Choose *zoom* in the *View* menu.
- Type *zoom* in the command bar.
- Type *Z* in the command bar.

You are prompted: In/Out/All/Center/Dynamic/Extents/Left/Previous/Right/Window/<Scale (nX/nXP)>:

A prompt menu opens:



Do one of the following:

- Type the zoom factor, followed by *X*. E.g. type *2x* to magnify the display 2 times; typing *0.5x* changes the display to half its original size.
- In paper space viewports, type the zoom factor followed by *XP* to define the zoom factor relative to the viewport, thus defining the scale of the viewport content.
- Pick two corners of a box on the existing view in order to enlarge that area to fill the display.

Real Time Zooming

1. Do one of the following:
   - Click the *Real-Time Zoom* tool button ( ) in the View toolbar.
   - Type *rtzoom* in the command bar, then press Enter.

You are prompted: >>ENTER, Right click or Esc to complete...

2. Press and hold the left mouse button.
   Move the mouse forward to zoom in.
   Move the mouse backward to zoom out.
3. To abort the *Real-Time Zoom* command, do one of the following:

   - Right click.
   - On the keyboard, press Enter, space bar or Esc.

Dynamic zooming

1. Do one of the following:
   - Choose *zoom* in the *View* menu.
   - Type *zoom* in the command bar.
   - Type *Z* in the command bar.

A prompt menu displays.

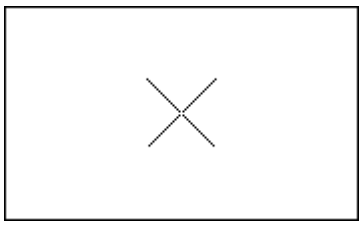
You are prompted: In/Out/All/Center/Dynamic/Extents/Left/Previous/Right/Window/<Scale (nX/nXP)>:

2. Do one of the following:

   - Choose *Dynamic* in the prompt menu.
   - Type *D*, then press Enter.

A view extents or view limits (if larger) is executed first.

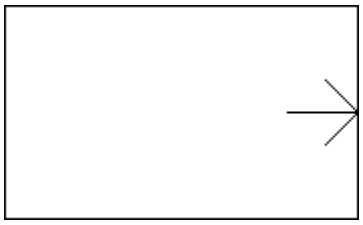
A view box displays. The original size of the view box matches the size of the previous view.



A dashed rectangle indicates the area of the previous view.

3. (option) To resize the view box:

   - Click to switch to resize mode. A right pointing arrow indicates you are in resize mode.



   - Move the mouse to adjust the size.
   - Click to leave resize mode.

4. Move the view box to the area you want to enlarge, then right click.
   The drawing is zoomed in on the selected area.
5. Repeat steps 1 through 4 to zoom in on another part of the drawing.

View Manipulation Using the Mouse and Keyboard

In BricsCAD you can navigate through a 3D model using the mouse and keyboard shortcuts.

In perspective mode you can walk inside the 3D model. In isometric mode you can only zoom in very close, but not go inside.

Mouse / Key	Action	Result
Wheel	scroll	Zoom in / out. The incremental change in zoom with each mouse-wheel action is controlled through the *Zoom Factor* variable (ZOOMFACTOR). Default = 60. Values between 3 and 100 are accepted. When zooming in the incremental step decreases gradually allowing to focus on a particularly detail easily. In a 3D model the entity under the cursor has the zoom focus. The ZOOMWHEEL system variable controls the zoom direction: • 0 or OFF: scrolling forwards zooms in, backwards zooms out. • 1 or ON: scrolling forwards zooms out, backwards zooms in.
Middle Button or wheel (**)	press and hold while moving the mouse	Real-time pan (RTPAN command)
Middle Button or wheel (**)	double click	Zoom extents
Ctrl + Shift + Left Button	press and hold while moving the mouse	Real-time zoom in / out (RTZOOM command)
Ctrl + Shift + Right Button	press and hold while moving the mouse	Real-time pan (RTPAN command)
Ctrl + Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time sphere (RTROTF command)
Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time constrained sphere (RTROT command)
Ctrl + Right Button	press and hold while moving the mouse	Real-time rotate about the screen Z-axis (RTROTZ command)

Walk Through Navigation

Alt + Left Button
press and hold while moving the mouse
Walk left / right and forward / backward (RTWALK command)
Alt + Middle Button
press and hold while moving the mouse
Move up / down and left / right (RTUPDOWN command)
Ctrl + Middle Button
press and hold while moving the mouse
Look around (RTLOOK command)
Ctrl + Arrow Keys
press the arrow keys while holding the Ctrl key
Walk left / right and forward / backward (RTWALK command)
Ctrl + Shift + Arrow Keys
press the arrow keys while holding the Alt + Ctrl keys
Look up / down and left / right (RTUPDOWN command)
Ctrl + Home

Sets the view direction horizontal
Alt + Home

Moves target point to center of scene
Alt + plus/minus

Increase / decrease movement speed
Ctrl + plus/minus

Increase / decrease rotation speed

(**) On condition the *Middle Button Pan* (MBUTTONPAN) variable is set *ON*.

Zoom Factor Variable

1. Type *zoomfactor* in the command bar, then press Enter.
   You are prompted: New current value for ZOOMFACTOR (3 to 100) <current value>:
2. Type a value between 3 and 100 in the command bar, then press Enter.

or

1. In the Settings dialog, open the *Drawing* category, then expand the *Display/Viewing* sub-category.
   In the *Viewing* group, select the *ZOOMFACTOR* variable.
2. Type a value between 3 and 100 in the *zoomfactor* field.

NOTE Increase the zoom factor to speed up scroll wheel zooming in large drawings.

Middle Button Pan Variable

1. Type in *mbuttonpan* the command bar, then press Enter.
   You are prompted: New current value for MBUTTONPAN (Off or On) <current value>:
2. Type *On* or *Off* in the command bar.

or

1. In the Settings dialog, open the *Program Options* category, then expand the *User Preferences* sub-category and select the *MBUTTONPAN* variable.
2. Choose either *Support panning* or *Support action defined in menu file*.

View Rotation

Commands: LOOKFROM, RTROT, RTROTCTR, RTROTF, RTROTX, RTROTY, RTROTZ, DDVPOINT, -VIEW, PLAN and VPOINT

The LookFrom command controls the display of the Look From control.

The *Rtrot* (Real-Time Constrained Sphere) command rotates 3D drawings in real-time.

The *Rtrotctr* (Real-time Sphere Center) command freely rotates 3D drawings in real-time about a user-defined center point.

The *Rtrotf* (Real-time Free Sphere) command freely rotates 3D drawings in real-time.

The *Rtrotx* , *Rtroty* and *Rtrotz* commands rotates 3D drawings in real-time about the x, y or z screen axis.

The *Ddvpoint* command sets 3D viewpoints or plan view, through a dialog box (short for "dynamic dialog view point").

The *Plan* command displays the plan viewpoint of drawings.

The *Vpoint* command changes the 3D viewpoint.

In order to view your 3D drawings from any angle, you can rotate a view. The *Real-Time Motion* tools of BricsCAD allow you to rotate a view in real-time. You can rotate the view about the X, Y or Z screen axis or in any direction (real-time sphere). If the *Continuous Motion* variable is set, the view rotation continues until you conclude the *Real-Time Motion* command.

See also: View manipulation using the mouse.

NOTE 'Real-time' commands should not be used when drawing in 2D. Use the Plan View tool to restore top vi

View Transition Options

- **VTENABLE**: determines whether animated view transitions are enabled for zoom/pan and/or for view rotation operations.
- **VTDURATION**: determines the duration of view transition animations in milliseconds.
- **VTFPS**: defines the minimum frame rate required to allow animation. That is 7 by default, which means that the redraw time should take less than 143 (=1000/7) milliseconds. If the computer is not capable to redraw the view fast enough, no animation will be available.

Restoring Orthographic and Isometric Views

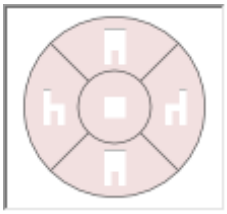
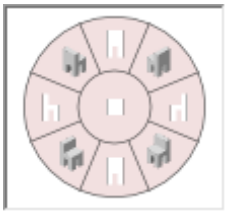
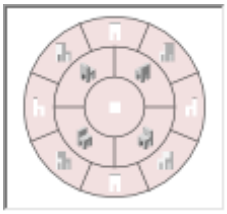
The *Look From* control allows to select a number of preset views such as orthographic views and isometric views. By default the *Look From* control displays in the top right corner of the graphic screen.

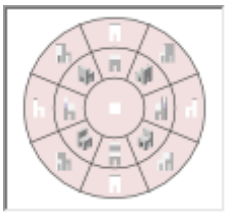
If the UCSORTHO system variable is *ON* an orthographic UCS is restored automatically when the related orthographic view is chosen on the *Look From* control.

System variables and user preferences

The behavior of the *Look From* control depends on a number of system variables and user preferences

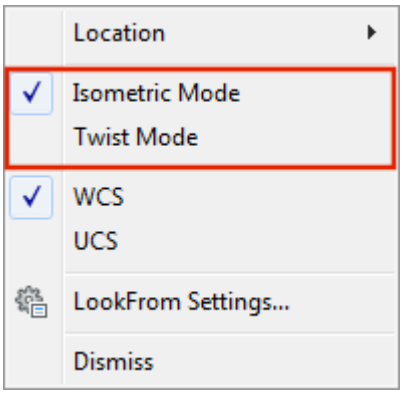
LookFrom control	
LookFrom display	☑ Display the LookFrom control
LookFrom location	[0] Top right corner
LookFrom opacity	80
LookFrom orientation	[0] WCS
LookFrom feedback	[1] Tooltips
LookFrom direction mode	[3] 8 top/down corners(26 directions)
LookFrom zoom extents	☑ Zoom extents

Name	Title	Description / Options
NAVVIEWCUBEDISPLAY	Look From Display	Controls whether the *Look From* control displays.
NAVVIEWCUBELOCATION	Look From Location	Controls the location of the *Look From* control in the drawing area. The options are: 0: Top right corner. 1: Top left corner 2: Bottom left corner 3: Bottom right corner
NAVVIEWCUBEOPACITY	Look From Opacity	Sets the opacity of the *Look From* control when not active. Values between 1 (invisible) and 100 (no dimming) are accepted.
NAVVIEWCUBEORIENT	Look From Orientation	Specifies the view orientation reference: either the WCS (World Coordinate System or the current UCS (User Coordinate System).
LookFromFeedback	Look From Feedback	Controls the display and location of tooltips. 0: No tooltips 1: Tooltips next to the *Look From* control. 2: Tooltips in the status bar.
LookFromDirectionMode	Look From Direction Mode	Specifies the number of available view directions. Holding down the Ctrl-key switches between top-down and bottom-up view directions 0: Orthogonal views only (6 directions).  1: 6 Orthogonal views and 8 Isometric views (14 directions).  2: 4 Rotated orthogonal, 6 Orthogonal and 8 Isometric views (18 directions)  3: 8 Rotated isometric, 4 Rotated orthogonal, 6 Orthogonal and 8 Isometric views (26 directions).

		
LookFromZoomExtents	Look From Zoom Extents	Determines whether a 'zoom extents' is performed whenever a new view direction is selected.

Setting the Look From mode

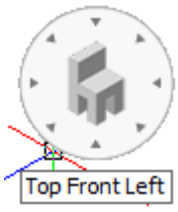
1. Move the cursor to the *Look From* control.
2. Right click, then select either *Isometric Mode* or *Twist Mode* in the context menu.



Using the Look From control in Isometric mode

Isometric mode is best used in 3D models.

1. Move the cursor to the *Look From* control.
   The *Look From* control becomes active.
2. Position the cursor on the *Look From* control to choose a view orientation.
   A preview image and a tooltip indicate the currently selected view orientation.



3. Click to confirm.
   The view is updated.

NOTE VTENABLE setting determines whether animated view transitions are enabled for zoom/pan and/or for view rotation operations.

VTDURATION setting determines the duration of view transition animations in milliseconds.

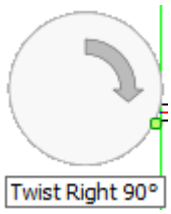
VTFPS setting defines the minimum frame rate required to allow animation. That is 7 by default, which means that the redraw time should take less than 143 (=1000/7) milliseconds. If the computer is not capable to redraw the view fast enough, no animation will be available.

Using the Look From control in Twist mode

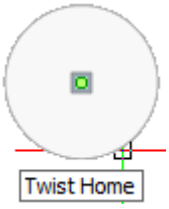
Twist mode is best used in 2D drawings.

In *Twist* mode, the *Look From* control allows to rotate the view around the screen Z-axis

1. Move the cursor to the *Look From* control.
   The *Look From* control becomes active.
2. Do one of the following:
   - Move the cursor on the edge of the the *Look From* control. Left half for counterclockwise, right half for clockwise rotation. Click when the arrow is at the desired rotation angle.
     The following angles are available: 15°, 30°,45°, 60°, 90°, 120°, 135° and 180°, both clockwise and counterclockwise.
     If *LookFromFeedback* = 1, the angle displays in a tooltip.
     Click again to rotate over the same angle again.



   - Move the cursor to the center of the *Look From* control, then click to restore the view (= unrotated state).



Real Time Sphere

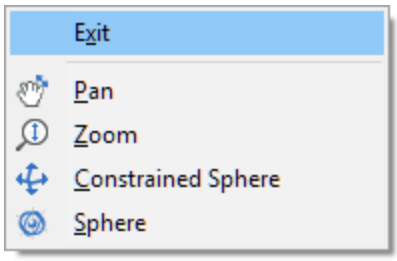
1. Do one of the following:
   - Click the *Real-Time Constrained Sphere* tool button ( )
     or the *Real-Time Sphere* tool button ( ) on the *Real-Time Motion* toolbar.
   - Choose *Real-Time Motion* > *Real-Time Constrained Sphere*
     or *Real-Time Motion* > *Real-Time Sphere* in the *View* menu.
   - Type *rtrot* , *rtrotf* or *rtrotctr* in the command bar, then press Enter.

You are prompted: >> Press ENTER or Esc to complete, or right click to display context menu ...

3. Press and hold the left mouse button.
   Move the mouse to rotate the view.

   - Using the *Real-Time Constrained Sphere* command horizontal mouse movement (parallel to the screen X-axis) rotates the 3D model about the world Z-axis.
   - Using the *Real-time Sphere Center* command you are prompted to specify a center point for the rotation first.

4. (option) Right click to display a context menu:



5. To abort the *Real-Time Sphere* commands, do one of the following:

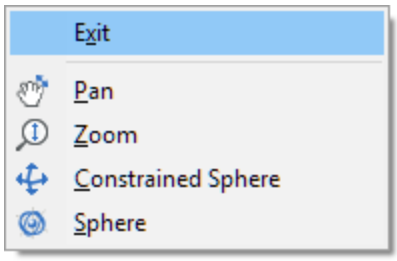
   - Right click, then choose *Exit* in the context menu..
   - Press Enter, space bar or Esc.

Real Time X

1. Do one of the following:
   - Click the *Real-Time X* tool button ( ) on the *Real-Time Motion* toolbar.
   - Choose *Real-Time Motion* > *Real-Time X* in the *View* menu.
   - Type *rtrotx* in the command bar, then press Enter.

You are prompted: >> Press ENTER or Esc to complete, or right click to display context menu ...

2. Press and hold the left mouse button.
   Move the mouse to rotate the view.
3. (option) Right click to display a context menu:



4. To abort the *Real-Time X* command, do one of the following:

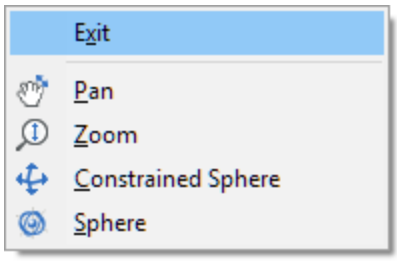
   - Right click.
   - Press Enter, space bar or Esc.

Real Time Y

1. Do one of the following:
   - Click the *Real-Time Y* tool button ( ) on the *Real-Time Motion* toolbar.
   - Choose *Real-Time Motion* > *Real-Time Y* in the *View* menu.
   - Type *rtroty* in the command bar, then press Enter.

You are prompted: >> Press ENTER or Esc to complete, or right click to display context menu ...

2. Press and hold the left mouse button.
   Move the mouse to rotate the view.
3. (option) Right click to display a context menu:



4. To abort the *Real-Time Y* command, do one of the following:

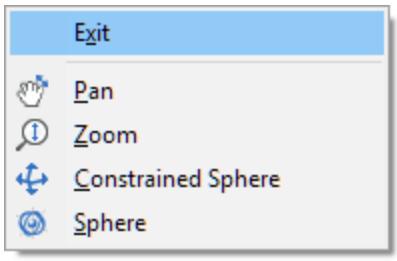
   - Right click.
   - Press Enter, space bar or Esc.

Real Time Z

1. Do one of the following:
   - Click the *Real-Time Z* tool button ( ) on the *Real-Time Motion* toolbar.
   - Choose *Real-Time Motion* > *Real-Time Z* in the *View* menu.
   - Type *rtrotz* in the command bar, then press Enter.

You are prompted: >> Press ENTER or Esc to complete, or right click to display context menu ...

2. Press and hold the left mouse button.
   Move the mouse to rotate the view.
3. (option) Right click to display a context menu:



4. To abort the *Real-Time Z* command, do one of the following:

   - Right click.
   - Press Enter, space bar or Esc.

Preset Viewpoints

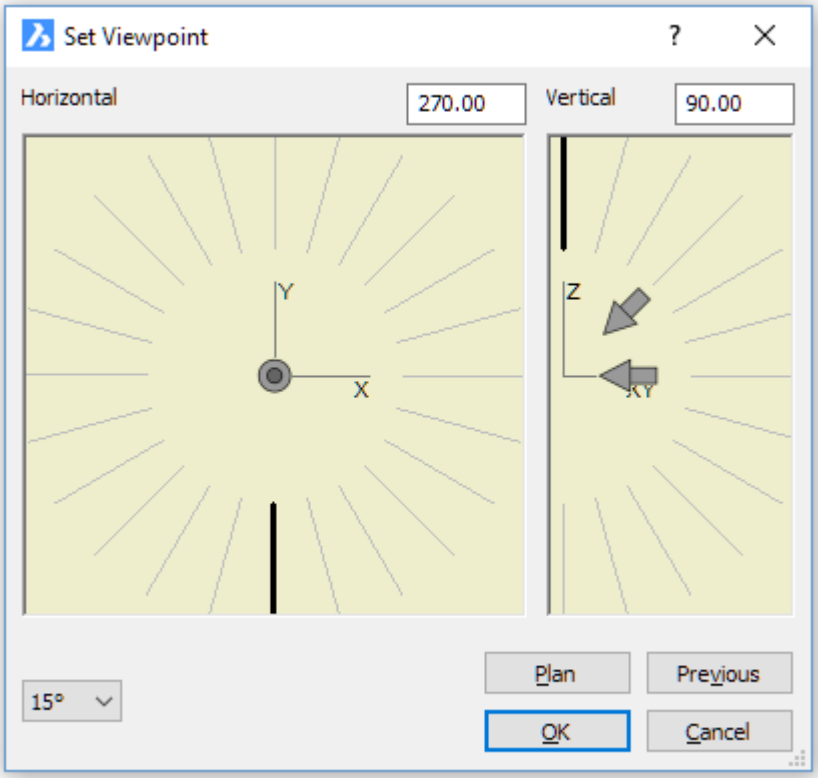
1. Do one of the following:

   - Click the *Set Viewpoint ...* tool button ( ) on the *View* toolbar.
   - Choose *Set Viewpoint ...* in the *View* menu.
   - Type *ddvpoint* in the command bar, then press Enter.

The *Preset Viewpoint* dialog opens.

2. Click the *Angle Precision* button (45°) to choose a different display mode for the *Set Viewpoint* dialog.

Then choose either *45°*, *15°* or *5°*.



45° display mode

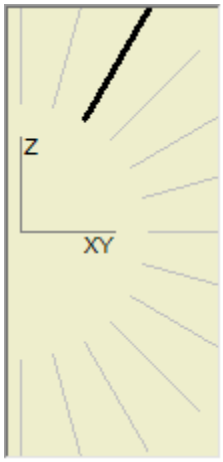
In *15°* or *5° mode* click in the inner area of the *Horizontal* and *Vertical* fields to choose a *45°* display mode viewpoint.
Click in the outer area of the *Horizontal* and *Vertical* fields to choose another angle.

3. To set the *Vertical* view angle, do one of the following:

   - Skip this step to accept the default vertical view directions:
   - Horizontal for East, North, West and South.
   - Downwards for the other directions.

If both the down- and left-arrow show in the *Vertical* field, the default vertical view directions are active.

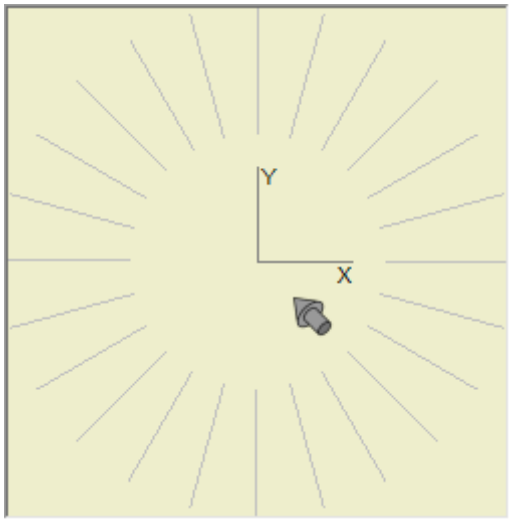
- Select a *45° mode* angle in the inner area, then click to confirm your choice.
  A single arrow indicates the selected vertical view direction.
- If in *15°* or *5° mode* in, select a view direction in the outer area, then click to confirm your choice.
  The selected angle displays in bold.



The view is updated as you click.

4. To set the *Horizontal* view angle, do one of the following:

- Select a *45° mode* angle in the inner area, then click to confirm your choice.
  An arrow indicates the selected horizontal view direction.
  The perspective of the arrow reflects the vertical view direction.



- If in *15°* or *5° mode* in, select a view direction in the outer area, then click to confirm your choice.
  The selected angle displays in bold.

The view is updated as you click.

NOTE Use the *Isometric Views* toolbar for standard view rotations: top, front, back, left, right and isometric views.

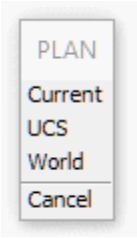
Plan View

Do one of the following:

- Click the *Plan View* tool button ( ) on the *View* toolbar.
- Choose *Plan View* in the *View* menu.
- Type *plan* in the command bar, then press Enter.

You are prompted: Plan view of: UCS/World/<current UCS>:

A prompt menu opens:



Press Enter to restore the plan view with respect to the current coordinate system.

NOTES

- If the WCS is the current coordinate system, the *Current* and *World* options have the same result.
- If the *UCSFOLLOW* variable is set to *ON*, the plan view is generated whenever the UCS changes.

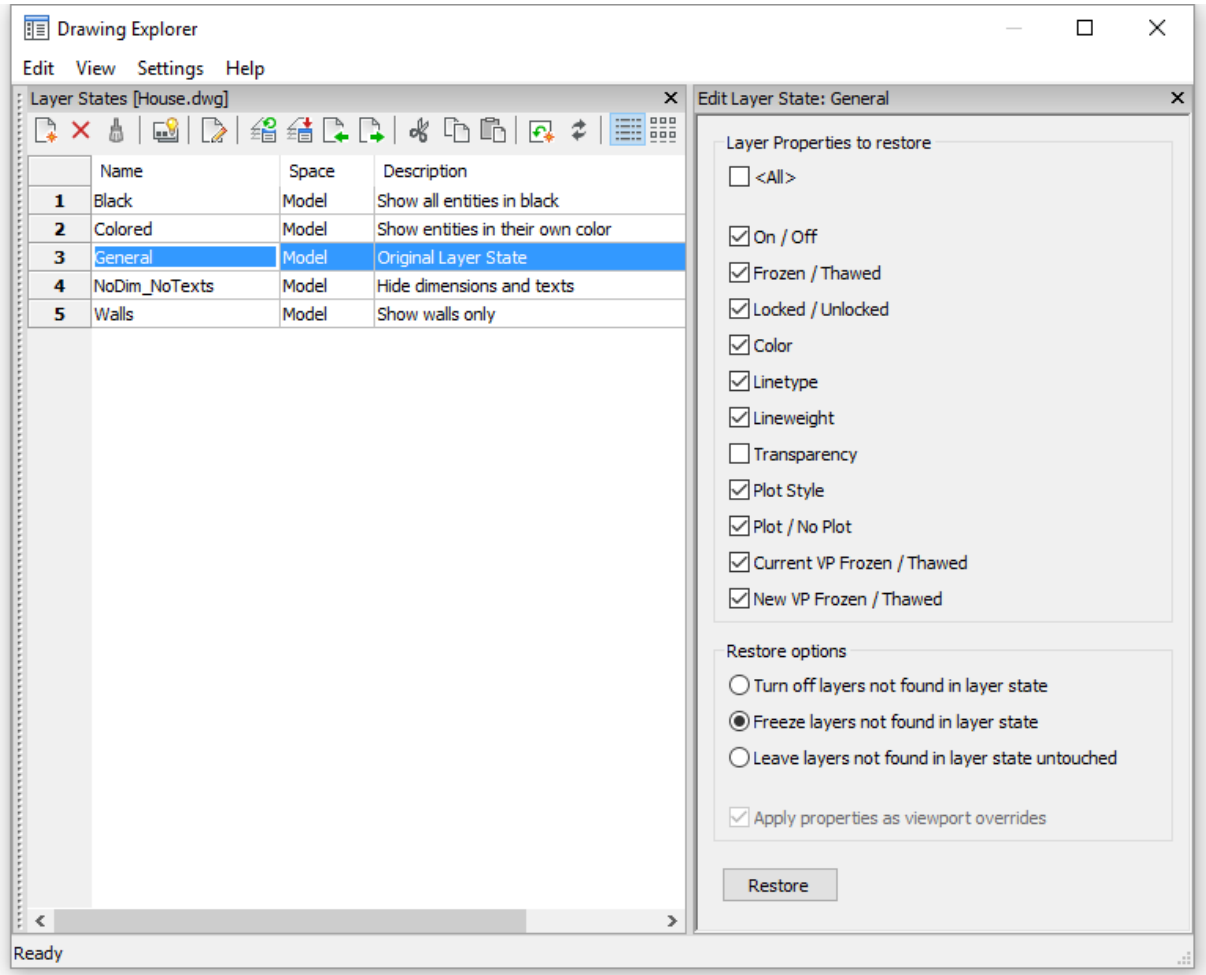
Layer States

TUTORIALS

Use Layer States instead of Clicking Bulbs

Layer states let you save and restore configurations of layer properties and states. You might want a layer to display in blue sometimes and green at other times, or you need some layers to be off or frozen or locked when editing a specific part of a drawing. After spending a lot of time adjusting all layer settings you can use layer states to save your settings. You can save multiple layer states in a each drawing. Layer states can be copied between drawings and you can save a layer state to an external file, which can then be imported in another drawing.

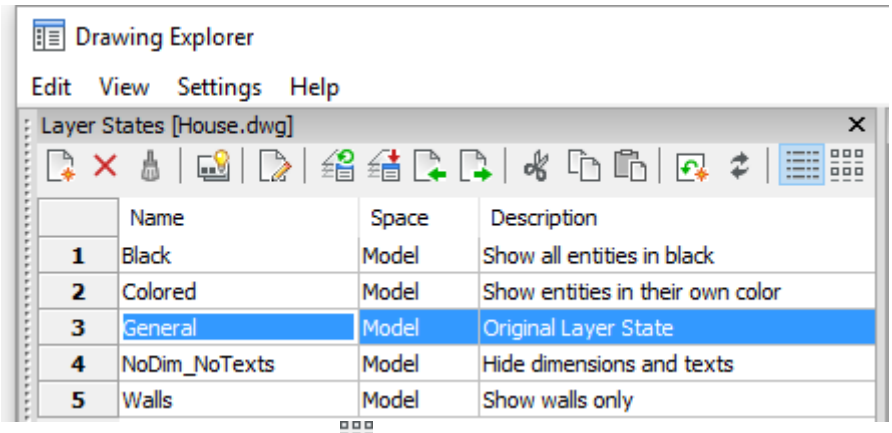
To easily create layer states, the Layer Utility commands might be useful.



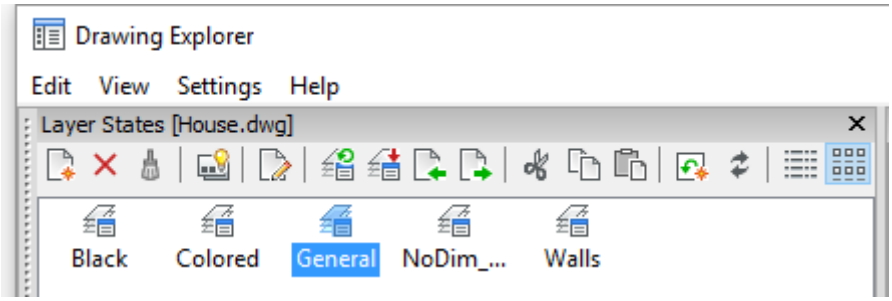
Drawing Explorer: Layer States (Detail view)

Opening the Layer States Explorer

1. In the *Tools > Drawing Explorer* menu choose *Layer States...* .
2. (option) Do one of the following:
   - Click the *Detail View* button ( ).

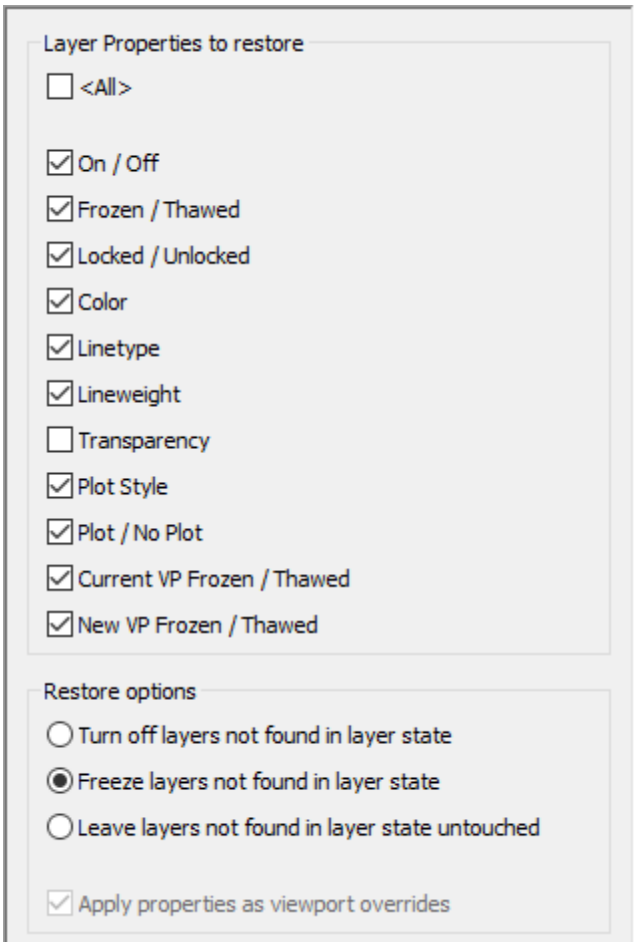


   - Click the *Icon View* button ( ).



Save the Current Layer State

1. Open the Layer States explorer.
2. Click the *New* button. ( ).
   A new layer state is created.
3. Type a name for the layer state, replacing the *NewLayerState1* default name.
4. (option) When in *Detail View* type a description in the *Description* field.
5. Select the *Layer Properties to restore*.
   Click the check box to select a layer property, click again to remove the selection.



NOTE By default in a new layer state all layer properties are selected to be restored.

Edit a Layer State

1. Open the Layer States explorer.
2. (option) Click the *Overwrite* button ( ) to overwrite the layer state with the current layer settings.
3. If necessary, click the *Tree View* button ( ).
4. Click the expand button of the layer state.
5. (option) Rename the layer state.
6. (option) Edit the layer state description.
7. (option) Expand the *Layer Properties to restore* list.
   Click the check box to select a layer property, click again to remove the selection.

Restore a Layer State

1. Open the Layer States explorer.
2. Select the layer state you want to restore.
3. Select a *Restore Option*.
   The restore option applies to layers which are added after the selected layer state was created.
   The options are:
   - Turn Off.
   - Freeze.
   - Do nothing.
4. Optionally check the *Apply properties as viewport overrides* option, which applies the layer state as viewport overrides (VP layer properties) to the current layout viewport.
   This option is available only on a layout within a layout viewport.
5. Do one of the following:
   - Click the *Restore* button ( ).
   - Right click, then choose *Restore* in the context menu.

4. Click the *Regen* button ( )

NOTES

- If *REGENAUTO* is on ( ), it is not necessary to press the *Regen* button in step 4.
- Layer states can be restored using the Layer State control on the Layers panel.
  You can add the Layer State control to a toolbar.

Copy Layer States to Another Drawing

1. Open both, the source and the target drawing.
2. Make the source drawing the current drawing.
3. Open the Layer States explorer.
4. If the layer states display in *Tree View*, choose either *Detail View* ( ) or *Icon View* ( ).
5. Do one of the following
   - Select the layer state you want to copy.
   - Press and hold the Ctrl key to select multiple layer states.

6. With the cursor on a selected layer state, press and hold the left mouse key, then drag the selection to the target drawing in the *Open Drawings* section of the Drawing Explorer dialog window.
7. When on the target drawing, release the left mouse button.
   The selected layer states are copied to the target drawing.

Export a Layer State

1. Open the Layer States explorer.
2. Select the layer state.
3. Click the *Export* button ( ).
   The *Export Layer States* dialog opens.
4. In the *File Name* field, type a name for the layer state.
5. Choose a folder to save the layer state in.
6. Click the *Save* button.

Import a Layer State

1. Open the Layer States explorer.
2. Click the *Import* button ( ).
   The *Import Layer States* dialog opens.
3. Browse to the folder where you have saved the layer state file.
4. Select the layer state.
5. Click the *Open* button.
   The layer state is imported.

NOTE If a layer state of the same name already exists in the drawing a warning displays. It is not possible to overwrite a layer state definition.

Named Views

Exploring Views

You can save the view in the current window as a named view. Saved views can be restored at any time.

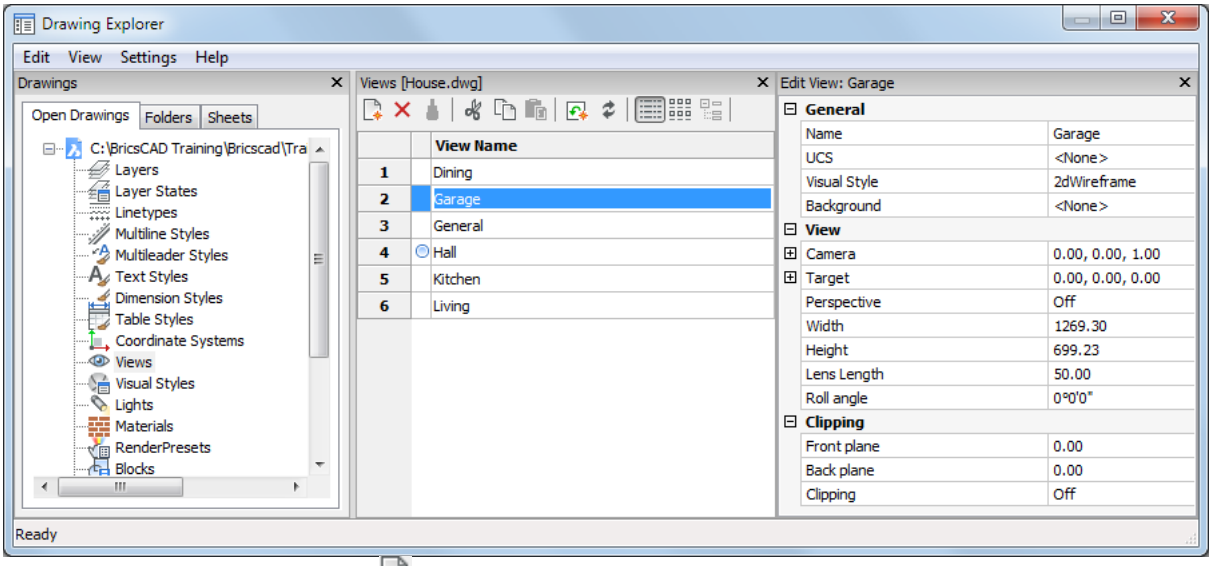
A Background can be assigned to a named view, which will be used when the view is rendered.

Exploring views

1. Do one of the following:
   - Choose *Drawing Explorer > Views...* in the *Tools* menu.
   - Type either *view* or *V* in the command bar, then press Enter.

The *Drawing Explorer - Views* window opens.

2. (option) Click the blank tile in front of the *View Name* to restore a saved view.
   The current view is marked ( ).



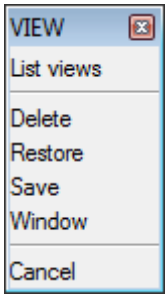
3. (option) Click the *New* button ( ) in the *Drawing Explorer* toolbar to save the current view.
4. (option) Click the *Delete* button ( ) in the *Drawing Explorer* toolbar to delete the selected view.
5. (option) Click the *View Name*, then right click and choose Rename in the context menu. Type a new name for the saved view.

6. (option) Check the *Perspective* property to make the view a visual perspective view.
   If not checked the view is a parallel perspective view.
   See also Define a View.
7. (option) Click the *Background* column of a view to choose a background.
   The background definition of the view is applied during rendering.

Saving a View

1. Do one of the following:
   - Click the *Save/Restore View* tool button ( ) on the *View* toolbar.
   - Choose *Save/Restore View* in the *View* menu.
   - Type *-view* in the command bar, then press Enter.

You are prompted: View: ? to list saved views/Delete/Restore/Save/Window:



3. Do one of the following:

   - Choose *Save* in the *View* prompt menu.
   - Type *S* in the command bar, then press Enter

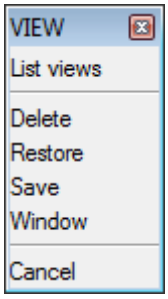
4. Type a name for the view in the command bar, then press Enter.
   The current view is saved.

NOTE If you type a name that is already used, the current view is saved, replacing the previously saved view. There is no warning when overwriting a saved view.

Restoring a View

1. Do one of the following:
   - Click the *Save/Restore View* tool button ( ) on the *View* toolbar.
   - Choose *Save/Restore View* in the *View* menu.
   - Type *-view* in the command bar, then press Enter.

You are prompted: View: ? to list saved views/Delete/Restore/Save/Window:



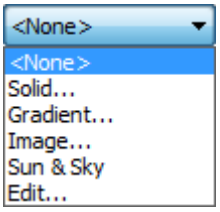
3. Do one of the following:

- Choose *Restore* in the *View* prompt menu.
- Type *R* in the command bar, then press Enter

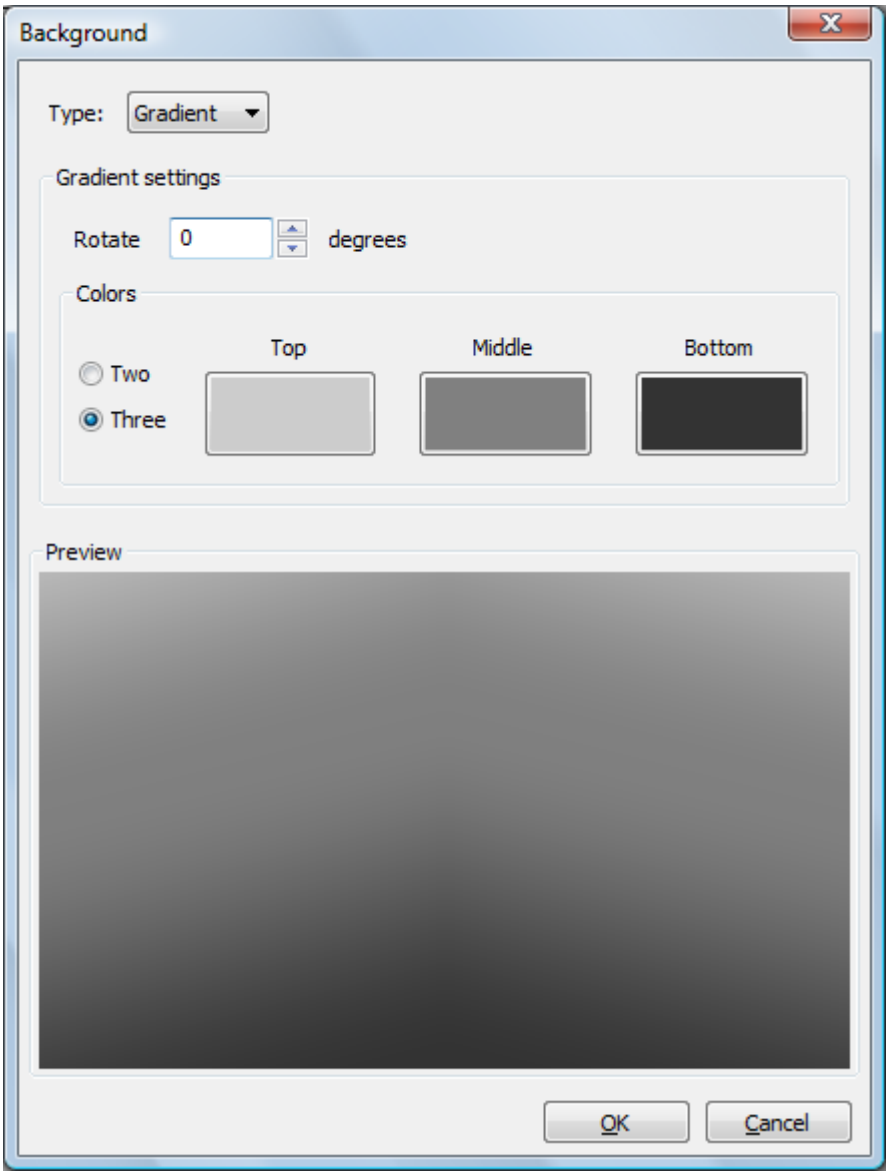
4. Type the name of the saved view you want to restore in the command bar, then press Enter.
   The saved view is restored.

Defining the View Background

1. In the Drawing Explorer - Views dialog window, select *Background* in the *Edit* grid of the view.
2. Click the settings field to display the *Background* options.



3. Choose an option in the list.
   The *Background* dialog box displays:

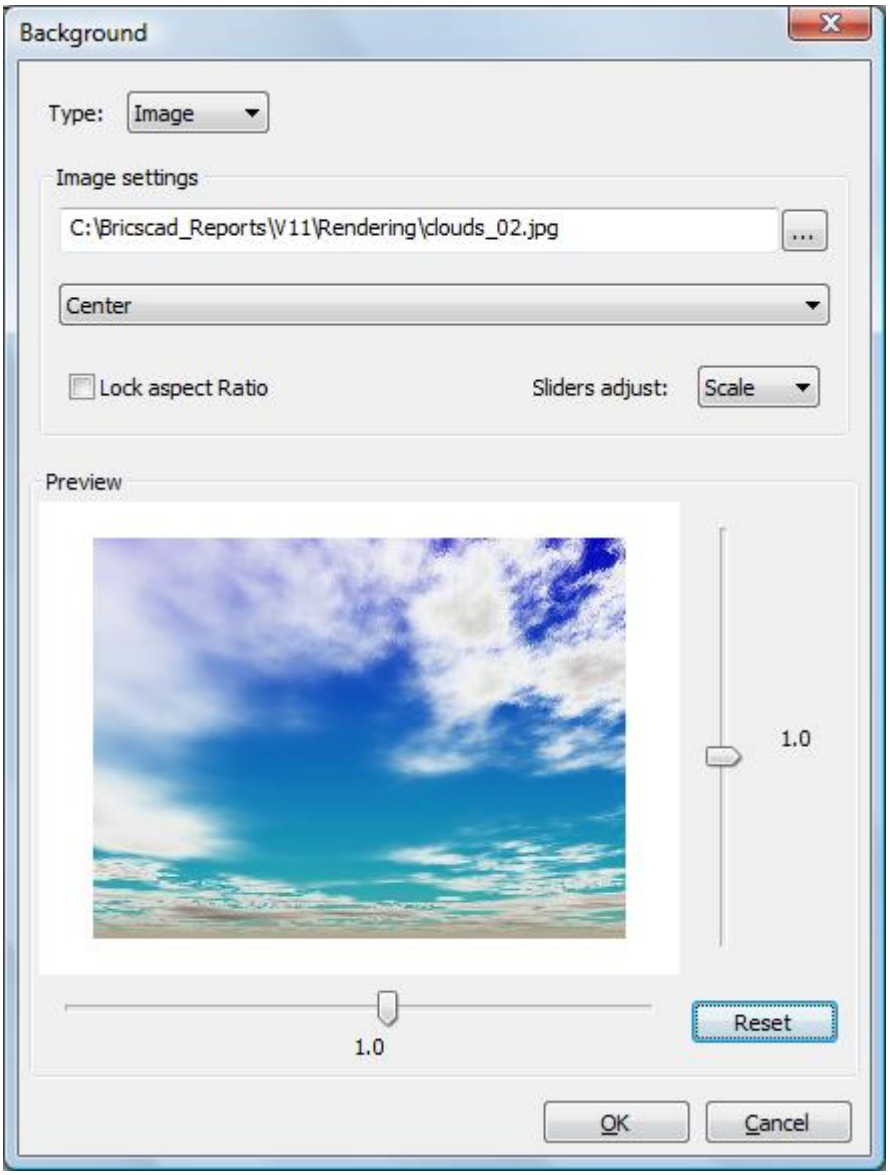


4. (option) Select *Solid* in the *Type* list.
   Click the *Color* button to define the color.

5. (option) Select *Gradient* in the *Type* list.

   - Set the rotation angle
   - Choose *Two* or *Three* colors.
   - Click the *Top*, *Middle* and *Bottom* color buttons to define the color.

6. (option) Select *Image* in the *Type* list.

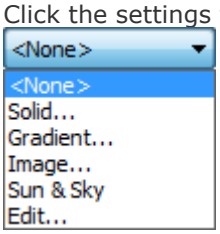


- Click the browse button next to the Image settings field to select a raster image file.
- Choose an alignment. The options are: *Center*, *Stretch* and *Tile*.

7. (option) Right click, then choose *Set Current* in the context menu if you want to use the newly defined background in a rendering right away.

To Edit the View Background

1. In the Drawing Explorer - Views dialog window, select *Background* in the *Edit* grid of the view..
2. Click the settings field to display the *Background* options.



3. Choose *Edit...* in the list to edit the current settings.
   The *Background* dialog box displays.
4. Edit the settings needed, then click the *OK* button.

Perspective Views

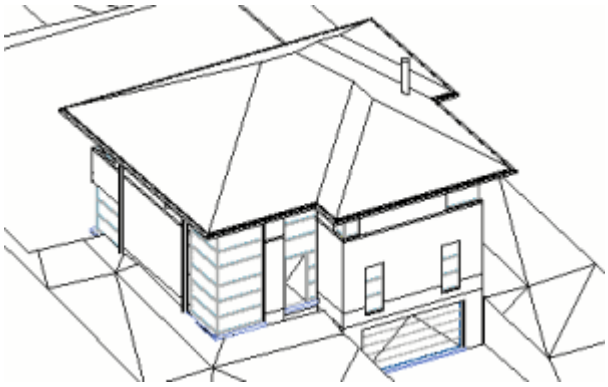
Commands

The *Define View* command dynamically defines parallel perspective or visual perspective views of your 3D models.

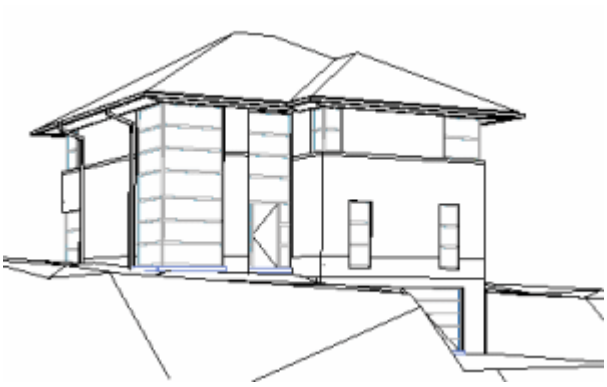
The *Camera* command creates perspective views.

The projection method of a view is controlled by the *Perspective* property of the view:

- If ON, the view is a visual perspective view.
- If OFF, the view is a parallel perspective view.



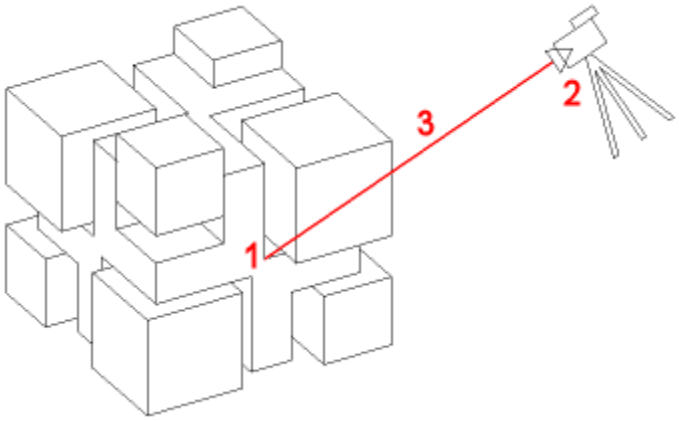
Parallel Perspective View



Visual Perspective View

The following parameters are used in the *Define View* procedure:

- *Target Point*: Defines the center of the view (1)
- *Camera Point*: Defines the view point and view direction (2).
- *Distance to Target*: Distance between the camera position an the target point (3).



NOTE View manipulation tools, such as zoom, pan, real-time motion, are not available in visual perspective views. However, you can use the mouse to manipulate the view.

Defining a Camera View

The *Camera* command creates a named view.

1. Do one of the following:

   - Click the *Camera* tool button ( ) on the *View* toolbar.
   - Choose *Camera* in the *View* menu.
   - Type *Camera* in the command bar.

You are prompted: Specify camera location:

2. Do one of the following:

   - Click a point in the drawing.
   - Type the coordinates of the camera location in the command bar and press Enter.

You are prompted: Specify target location:

3. Do one of the following:

   - Click a point in the drawing.
   - Type the coordinates of the target location in the command bar and press Enter.

You are prompted: Enter an option: ?/Name/LOcation/Height/Target/LEns/Clipping/View/<eXit>:

4. Do one of the following:

   - Press Enter to create the camera view using a default name (e.g. Camera1).
   - Enter an option:
     - **Name**:
       Prompts you: Enter name for new camera <Camera1>:
       Type a name at the command prompt, then press Enter.
     - **Location**:
       Prompts you: Specify camera location <current location>:
       Click a point or enter new coordinates of the camera location, then press Enter.
     - **Height**:
       Prompts you: Specify camera height <current height>:
       Enter a new height (Z-coordinate) for the camera location.
     - **Target**:
       Prompts you: Specify target location <current location>:
       Click a point or enter new coordinates of the target location, then press Enter.
     - **Lens**:
       Prompts you: Specify lens length in mm <50.00>:
       Enter an new lens length, then press Enter.
     - **Clipping**:
       Prompts you: Enable front clipping plane? <Yes>/No:
       Choose Yes to define the front clipping plane.
       Prompts you: Specify front clipping plane offset from target plane <0.00>:
       Type value in the command bare or specify the distance by clicking two points.

       Prompts you: Enable back clipping plane? Yes/<No>:
       Choose Yes to define the front clipping plane.
       Prompts you: Specify front clipping plane offset from target plane <0.00>:
       Type value in the command bare or specify the distance by clicking two points.

- **View**:
  Prompts you: Switch to camera view? Yes/<No>:
  Choose Yes to display the camera view.
  This option concludes the *Camera* command.

NOTE Choose *Drawing Explorer > Views...* in the *Tools* menu to edit a camera view.

Toggle the Perspective Property of a View

Do one of the following

- Click the *Perspective* tool button ( ) on the *View* toolbar.
  The state of the tool button, pressed or not, indicates the current value of the PERSPECTIVE system variable.
- Choose *Perspective* in the *View* menu.
- Type *perspective* in the command bar.
  You are prompted: New current value for PERSPECTIVE (Off or On) <On>:
  A prompt menu displays.
  Do one of the following:
  - Type *off* or *on* in the command bar, then press Enter.
  - Choose *Off* or *On* in the prompt menu.

NOTE In named views you can set the *Perspective* property in the Drawing Explorer - Views dialog.

Model Space and Paper Space

When you start a drawing session, your initial working area is called *Model Space*. Model Space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the *Model* tab.

In general model space consists of a single view that fills the screen. If needed, you can create additional views, called viewports, which can show different views of your drawing or 3D model. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. Click in a viewport to make it the current viewport. You can print the current viewport only. The Viewports command lets you manage your viewports in model space.

BricsCAD provides an additional work area, called *Paper Space*. Paper space represents a paper layout of your drawing. In this work area, you can create and arrange different views of your drawing similar to the way you arrange drawings on a sheet of paper. In paper space you can also add keynotes, annotations, borders, title blocks, and other print-related entities, which you don't want to see in model space.

Each drawing has at least one *Layout* in which you can have one or more *Layout Viewports*. Such layout viewports are to be considered as a view window on your drawing in model space. Layout viewports can be placed anywere in a paperspace layout. Each viewport has its own scale and layer visibility. All viewports in the same layout can be printed simultaneously.

The Mview command lets you manage your viewports in paper space.

Although not necessary to print your drawing, paper space offers a lot of advantages:

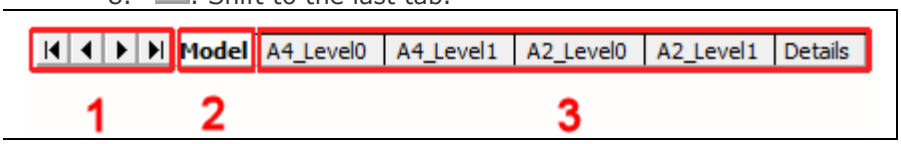
- Create multiple layouts to print the same drawing with different print settings, such as pen widths, printer configuration files, lineweight settings, drawing scale, and more.
- Add print-related entities that are not essential to the model itself, such as keynotes, annotations, title blocks, etc.
- For a single layout, create multiple layout viewports to print multiple views of your drawing at different scales.

Switching between Model Space and Paper Space

- To switch between Model Space and Paper Space Layouts use the Model and Layout tabs at the bottom of the drawing window.
- When working in a layout, use the MSPACE and PSPACE commands to toggle between paper space and model space.

Using the Model and Layout Tabs

1. To open model space, click the *Model* tab (2) at the bottom of the drawing window.
2. To open a paper space layout, click the corresponding *Layout* tab (3) at the bottom of the drawing window.
3. Use Ctrl-PgUp or Ctrl-PgDn to switch to the previous or next layout tab.
4. Click the navigation buttons to shift the tab display (1):
   5. : Display the model tab.
   6. : Shift one tab to the left.
   7. : Shift one tab to the right.
   8. : Shift to the last tab.



NOTE A drawing contains at least one layout, which is named *Layout1* by default.

Toggle between Model Space and Paper Space

Commands: MSPACE and PSPACE

1. (option) When in *model space* (Model Space, with floating viewports) do one of the following to switch to *paper space*:
   - Type either *pspace* or *ps*, then press Enter
   - Double click outside a viewport.

2. (option) When in *paper space* do one of the following to switch to *model space* (Model Space, with floating viewports):

   - Type either *mspace* or *ms*, then press Enter.
   - Double click inside a viewport.

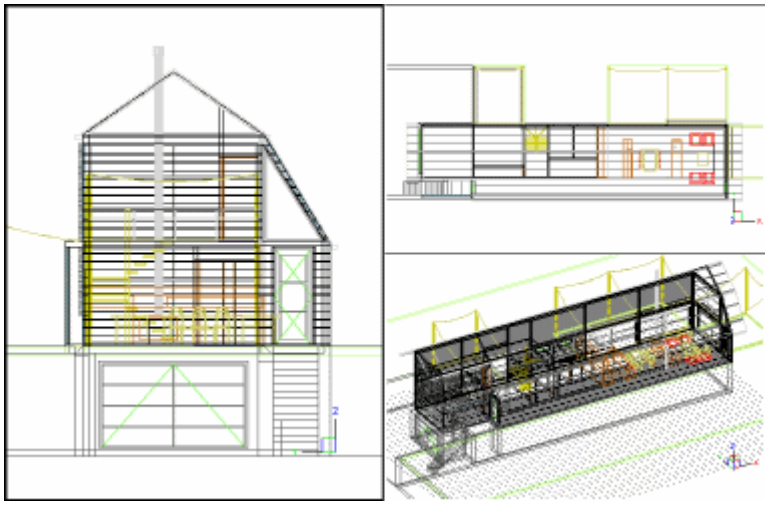
NOTE *MSPACE* and *PSPACE* commands are available when working in a layout only.

Model Space Viewports

The display in model space (Model Space, with tiled viewports) can be divided into multiple viewports, each of which can contain a different view of the current drawing. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. When no command is active click in a viewport to make it the current viewport. You can print the current viewport only.

The following settings can be defined differently for each viewport:

- grid display and snap
- coordinate system: WCS or UCS



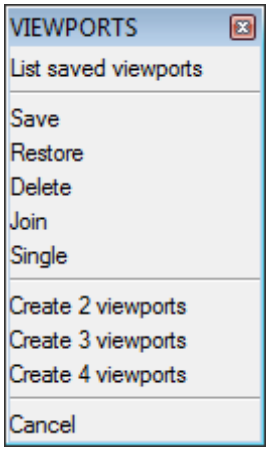
Model Space divided in 3 viewports

Creating Viewports in Model Space

1. Do one of the following:
   - Click the *Viewports* tool button ( ) on the *Views* toolbar.
   - Choose *Viewports* in the *Views* menu.
   - Type *vports* in the command bar.

You are prompted: Viewports: ? to list/Save/Restore/Delete/SIngle/Join/2/3/4/<3>:

A prompt menu opens.

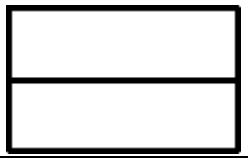
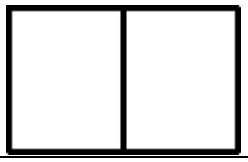


2. (option) To create 2 viewports, do one of the following:

   - Choose *Create 2 viewports* in the prompt menu.

- Type *2* in the command bar, then press Enter.

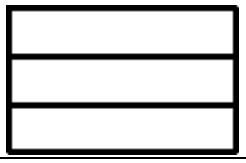
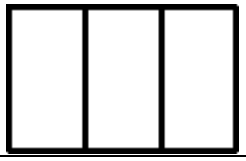
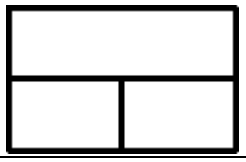
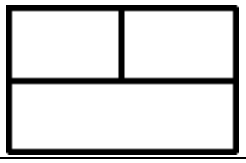
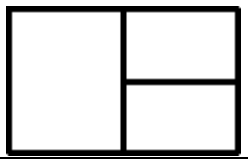
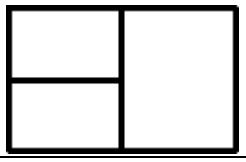
You are prompted. Two viewports: Horizontal/<Vertical>:

Option	**Result**
Horizontal	
Vertical	

3. (option) To create 3 viewports, do one of the following:

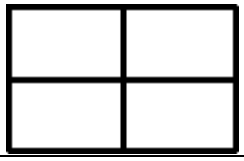
- Choose *Create 3 viewports* in the prompt menu.
- Type *3* in the command bar, then press Enter.

You are prompted. Three viewports: Horizontal/Vertical/Above/Below/Left/<Right>:

Option	**Result**
Horizontal	
Vertical	
Above	
Below	
Left	
Right	

4. (option) To create 4 viewports, do one of the following:

   - Choose *Create 4 viewports* in the prompt menu.
   - Type *4* in the command bar, then press Enter.

Option	Result
4	

Drawing in Multiple Viewports

1. Click in a viewport to make it current.
   The border of the current viewport highlights.
2. Draw the entities in the current viewport.
3. (option) Repeat step 1 to make a different viewport current.

NOTES

- The crosshairs show in the current viewport only.
- You can start a command in the current viewport, and finish the operation in a different viewport.

Joining Viewports

1. Launch the *Viewports* command.
2. Select the *Join* command option.
   You are prompted: Select inside dominant viewport <Current>:
3. Click in the dominant viewport or press Enter to choose the current viewport.
   The border of the selected viewport highlights.
   You are prompted. Select inside viewport to join:
4. Click the viewport to join again to confirm.
   The two selected viewports are joined.

NOTE You can join viewports which share an edge of equal length only.

Paper Space Viewports

The *Mview* command creates one or more viewports in layout tabs (short for "make viewports").

The *MvSetup* command creates multiple viewports; in model space the command creates a rectangle, representing a paper sheet with respect to a specified scale.

The *AlignSpace* command adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space.

The *Vpclip* command clips viewports in layouts (short for "view port clipping").

In a layout you can create multiple viewports each of which displays a unique view of the entities created in model space. Each layout viewport functions as a window into your model space drawing. You can control the view, scale, and content of each layout viewport separately.

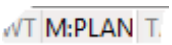
A layout viewport is created as a separate entity that you can copy, delete, move, scale, and stretch as you would any other drawing entity. You can snap to the viewport borders using entity snap. When you are working in model space with floating viewports (see Toggle between model space and paper space), click any layout viewport to make it the current viewport, and then add or modify model space entities in that viewport. Any changes you make in one layout viewport are immediately visible in the other viewports (if the other layout viewports are displaying that portion of the drawing). Zooming or panning in the current viewport affects only that viewport.

Each viewport has its own layer visibility settings. You can also turn off the display of the content of a viewport.

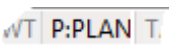
To preserve the scaling of a viewport you can lock the display. It is no longer possible to zoom or pan in a locked viewport.

Apart from viewports, you can add print-related entities in a paper space layout that are not essential to the model itself, such as keynotes, annotations, title blocks, etc. Such entities are part of a specific paper space layout and do not appear in other layouts or in model space.

When you are working in a layout, either *Model Space* or *Paper Space* is your current workspace. The *Workspace* field in the Status Bar indicates which workspace is current: *M:<Layout Name>* indicates you are working in *Model Space*, while *P:<Layout Name>* means *Paper Space* is the current workspace.

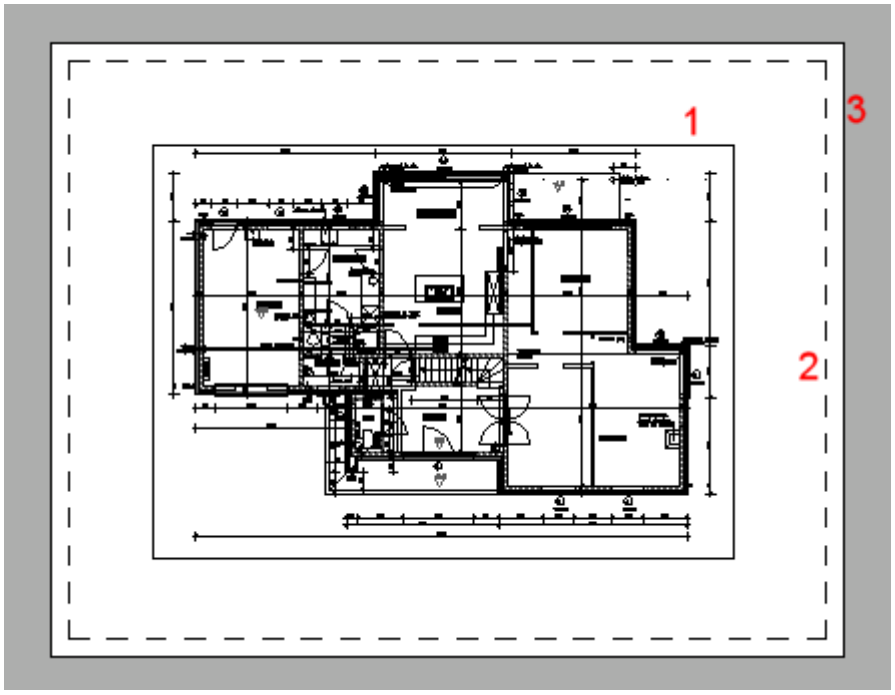


Model Space, with floating viewports of layout PLAN



Paper Space of layout PLAN

Displaying Paper Sheet and Printable Area



1. Viewport
2. Printable area.
   The display of the printable area is controlled through the DISPPAPERMARGINS system variable.
3. Paper sheet.
   The display of the paper sheet is controlled through the DISPPAPERBKG system variable.

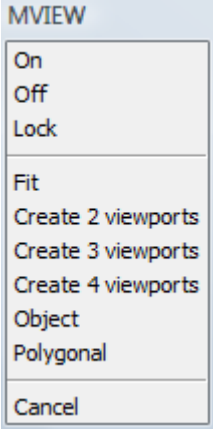
Creating Viewports in a Layout

Command: MVIEW

1. Click the appropriate layout tab at the bottom of the drawing window.
2. Do one of the following:
   - Click the *Paper Space Views* tool button ( ) on the *Views* toolbar.
   - Choose *Paper Space Views* in the *Views* menu.
   - Type *mview* at the command prompt, then press Enter.

You are prompted: Viewports. ON/OFF/Fit/2/3/4/<First corner>:

A prompt menu opens:



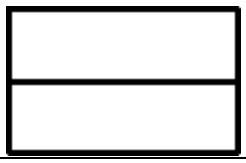
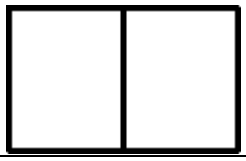
(option) To add 1 viewport. do one of the following:

- Click to specify the first corner of the viewport, then specify the opposite corner.
  A single viewport which fits in the specified rectangle is created.
- Choose *Fit to view* in the prompt menu or type *F*, then press Enter.
  A single viewport which fits in the current drawing display window is created.

(option) To create 2 viewports, do one of the following:

- Choose *Create 2 viewports* in the prompt menu.
- Type *2* in the command bar, then press Enter.

You are prompted. Two viewports: Horizontal/<Vertical>:

Option	Result
Horizontal	
Vertical	

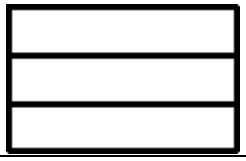
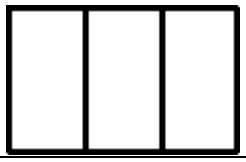
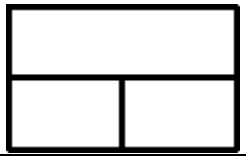
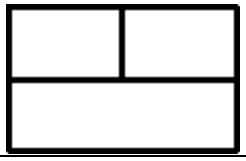
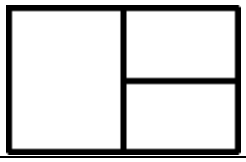
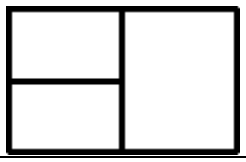
You are prompted: Fit to screen/<First corner of bounding rectangle>:

- Click to specify the first corner of the bounding rectangle, then specify the opposite corner.
  Two viewports which fit in the specified bounding rectangle are created.
- Choose *Fit to view* in the prompt menu or type *F*, then press Enter.
  Two viewports which fit in the current drawing display window are created.

5. (option) To create 3 viewports. do one of the following:

- Choose *Create 3 viewports* in the prompt menu.
- Type *3* in the command bar, then press Enter.

You are prompted. Three viewports: Horizontal/Vertical/Above/Below/Left/<Right>:

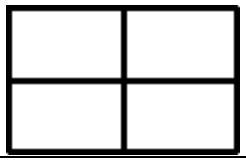
Option	**Result**
Horizontal	
Vertical	
Above	
Below	
Left	
Right	

You are prompted: Fit to screen/<First corner of bounding rectangle>:

- Click to specify the first corner of the bounding rectangle, then specify the opposite corner.
  Three viewports which fit in the specified bounding rectangle are created.
- Choose *Fit to view* in the prompt menu or type *F*, then press Enter.
  Three viewports which fit in the current drawing display window are created.

6. (option) To create 4 viewports. do one of the following:

   - Choose *Create 4 viewports* in the prompt menu.
   - Type *4* in the command bar, then press Enter.

Option	Result
Create 4 viewports	

You are prompted: Fit to screen/<First corner of bounding rectangle>:

   - Click to specify the first corner of the bounding rectangle, then specify the opposite corner.
     Four viewports which fit in the specified bounding rectangle are created.
   - Choose *Fit to view* in the prompt menu or type *F*, then press Enter.
     Four viewports which fit in the current drawing display window are created.

7. (option) To create a non-rectangular (clipped) viewport using an existing polyline or circle, do one of the following:

   - Choose *Object* in the prompt menu.
   - Type *O* in the command bar, then press Enter.

You are prompted: Select Object to clip viewport.

Select a closed polyline or a circle in the layout.

8. (option) To create a non-rectangular (clipped) viewport, do one of the following:

   - Choose *Polygonal* in the prompt menu.
   - Type *P* in the command bar, then press Enter.

You are prompted: Specify start point:

Specify the vertices of the non-rectangular viewport. Right click or press Enter to create the viewport.

NOTES

- Make sure no other viewports display when choosing the *Fit to view* option because they will be hidden behind the newly created viewport.
- Each newly created viewport shows all entities on the layers which are currently visible in model space.
- Since a new viewport is created on the current layer it is recommended to use a dedicated layer for viewport entities. Make this layer current when you want to add a new viewport. If you don't want the viewport outlines to be printed, set the *Do Not Plot* property of the viewport layer.
- Only circles and closed polylines can be used to create a clipped viewport. If you want to create an elliptical viewport, use the Polyline Ellipse variable to create a polyline representation of an ellipse.
- To select a clipped viewport do the following: Click the viewport border. Both the viewport and the clipping polygon are selected then, as indicated in the Properties Bar: the list button caption reads: *All (2)*. Press the list button and select *Viewport* to see the properties of the viewport.

Creating an Array of Layout Viewports

Command: MVSETUP

1. Type *mvsetup* in the command bar.
   You are prompted: Enter an option [Align/Create/Scale viewports/Undo]:
2. Do one of the following:
   - Type *C* in the command bar, then press Enter.
   - Choose *Create* in the context menu.

You are prompted: Enter an option [Delete objects/Create viewports/Undo] <Create>:

3. Press Enter to accept the default option.
   You are prompted: Enter choice for layout options <0-3> [none(0)/single(1)/std. engineering(2)/array of viewports(3)] <0>:
4. Do one of the following:

   - Type *3* in the command bar, then press Enter.
   - Choose *Array of Viewports* in the prompt menu.

You are prompted: Specify first corner of bounding area for viewport(s):

5. Pick a point in the layout.
   You are prompted: Other corner of rectangle:
6. Pick a second point in the layout.
   You are prompted: Enter number of viewports in X direction <1>:
7. Specify the number columns in the array.
   You are prompted: Enter number of viewports in Y direction <1>:
8. Specify the number of rows in the array.
   You are prompted: Specify distance between viewports in X direction <0.000>:
9. Do one of the following:

   - Press Enter, to create contiguous viewport columns.
   - Specify the horizontal distance between the viewports.

You are prompted: Specify distance between viewports in Y direction <horizontal distance>:

Do one of the following:

- Press Enter to set the vertical spacing equal to the horizontal spacing.
- Type *0* (zero) to create contiguous viewport rows.
- Specify the vertical spacing between the viewports.

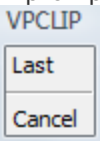
The viewports are created.

Press Enter to conclude the command.

To Clip a Viewport

Command: VPCLIP

1. Type *vpclip* in the command bar, then press Enter.
   You are prompted: Select viewport to clip:
2. Select the viewport.
   You are prompted: Polygonal <Select clipping object>:
   A prompt menu displays.

   Do one of the following:

   - Select the clipping object.
   - Choose *Polygonal* in the prompt menu or type *P*, then press Enter.
     You are prompted to draw a polyline.

The viewport is clipped.

NOTE

- Only closed polylines and circles are accepted as clipping boundaries.
- If you want elliptical viewports, use the polyline representation to create the ellipses.

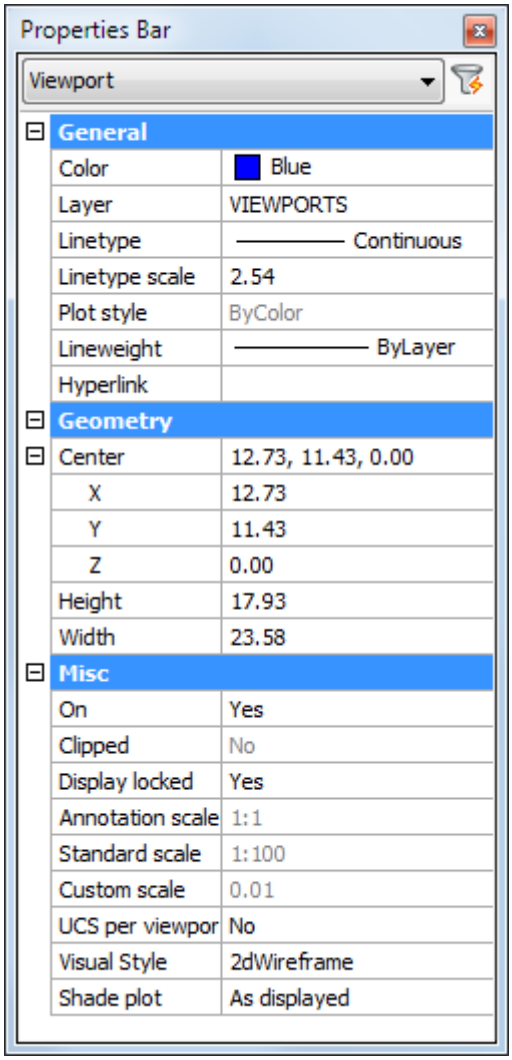
To Remove the Clipping Boundary of a Viewport

Command: VPCLIP

1. Type *vpclip* in the command bar, then press Enter.
   You are prompted: Select viewport to clip:
2. Select a clipped viewport.
   You are prompted: Polygonal/Delete<Select clipping entity>:
3. Do one of the following:
   - Type *D*, then press Enter.
   - Choose *Delete* in the prompt menu.

Defining Viewport Properties

1. Select the viewport.
   The viewport properties display in the Properties Bar.



2. Click the settings field of a property to modify.

Properties

Name	Description
Center	XYZ coordinates of the center point of the viewport. To specify the center of the viewport graphically, drag the center handle of the viewport.
Height (*)	Height of the viewport in *drawing units*. To specify the height of the viewport graphically, drag the top or bottom handle of the viewport.
Width (*)	Width of the viewport in *drawing units*. To specify the width of the viewport graphically, drag the left or right handle of the viewport.
On	Controls the display of the content of the viewport.
Clipped	Allows to create non-rectangular viewports.

Display locked	Locks the scaling of the viewport content to preserve the scaling factor.
Scale	The SCALELISTEDIT command allows to edit the available scales in the current drawing. **Annotation scale** Sets the annotation scale for the viewport. If the ANNOALLVISIBLE system variable is OFF, annotative entities which do not supoort the annotation scale do not display. **Standard scale** The *Standard scale* field automatically matches the *Annotation scale* field. Although you can set the *Standard scale* field different from the *Annotation scale* field manually, it is not recommended if annotative entities exist in the drawing. The *Standard scale* field reads 'Custom' if the value in the *Custom scale* field is not available in the drawings scale list. **Custom scale** Displays the viewport scale factor. Allows to set the viewport scale to a scale which is not available in the drawings scale list.
Standard scale (*)	Lets you choose a standard scaling factor.
Custom scale (*)	Lets you define the scaling factor in a decimal format. Displays the current scaling factor.
UCS per viewport	If Yes, lets you define a UCS for this viewport.
Shade Plot	Defines how the viewport will be plotted. The options are: As displayed Legacy wireframe Legacy hidden Rendered 2dWireframe 3dWireframe 3D Hidden Realistic Conceptual High Quality Modeling

(*) You must choose the width and height of a paperspace viewport with respect to the viewport scale.

Navigating in a Paper Space Layout

View manipulation commands, such as zoom, pan and view manipulation using the mouse, act slightly different in a paper space layout compared to model space (with tiled viewports).

If you are working in paper space (no viewport selected), view manipulation commands act on the complete paper space layout.

If you are working in a viewport (model space with floating viewports), view manipulation commands act on the active viewport only. Except if the display of the current viewport is locked, then the view manipulation commands act on the complete paper space layout.

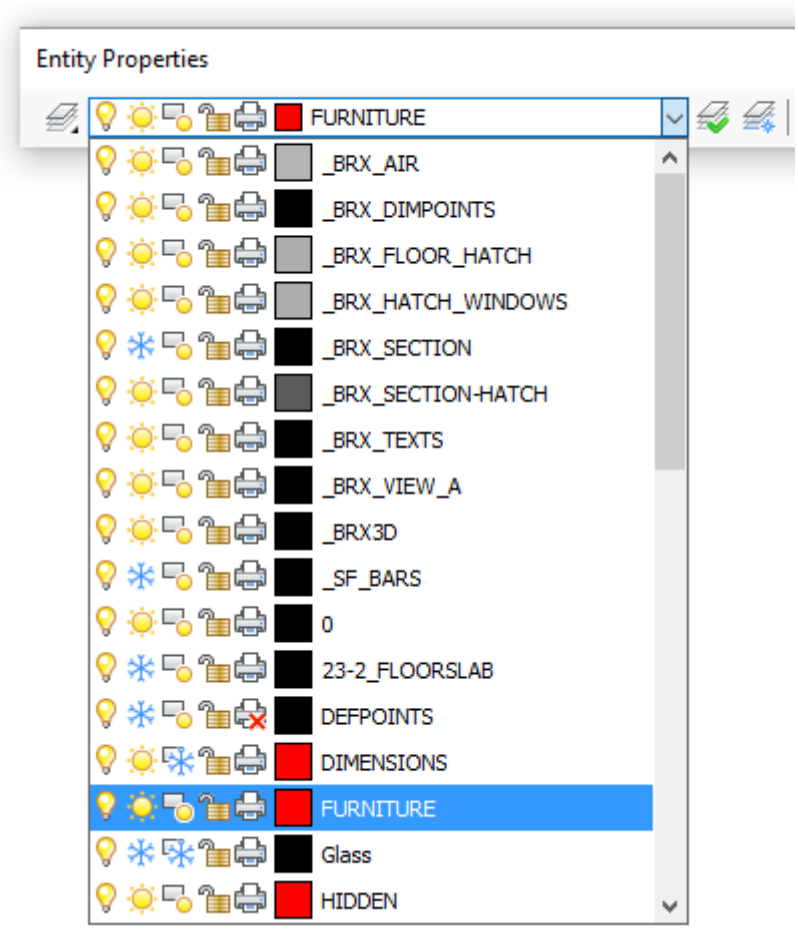
NOTE It is not possible to rotate the display of a paper space layout.
View Rotate commands are available in viewports of which the display is not locked only.

Setting the Layer Visibility in a Viewport

1. Do one of the following:

   - If *Paper Space* is the current workspace: double click inside the viewport to make it the current viewport.
   - If *Model Space with floating viewports* is the current workspace: click inside the viewport to make it the current viewport.

2. Select the layer in the layer list of the *Entity Properties* toolbar.



3. Do one of the following:
4. Click the *Thawed* icon ( ) of the layer(s) you want to freeze in the current viewport.

5. Click the *Frozen* icon ( ) of the layer(s) you want to thaw in the current viewport.
6. (option) Repeat the previous steps to set the layer visibility in another viewport.

Using the Drawing Explorer / Layers dialog

1. Do one of the following:
   - If *Paper Space* is the current workspace: double click inside the viewport to make it the current viewport.
   - If *Model Space with floating viewports* is the current workspace: click inside the viewport to make it the current viewport.

2. Do one of the following.

   - Click the *Layers...* tool button on the *Settings* toolbar.
   - Choose *Layers...* in the *Settings* menu.
   - Type *layer* in the command bar, then press Enter.

The *Drawing Explorer - Layers* window opens.

3. In the *VP Freeze* column:

   - Click the *Thawed* icon ( ) of the layer(s) you want to freeze.
   - Click the *Frozen* icon ( ) of the layer(s) you want to thaw.

4. Close the *Drawing Explorer - Layers* window.
5. (option) Repeat the previous steps to set the layer visibility in another viewport.

NOTE Use the *Viewports / Vpfreeze* option of the LAYOFF and LAYFRZ commands to freeze layers in a viewport by clicking entities.

Setting the Scale of a Viewport

1. Make sure the Plot Scale property of the layout is set correctly.
2. Switch to *paperspace*.
   (See Toggle between model space and paper space)
3. Click the viewport border.
   The viewport properties display in the BricsCAD Properties Bar.
4. Do one of the following:
   - Select the *Annotation scale* field, then click the *Browse* button ( ) and select a scale in the scale list.
     The *Standard scale* and *Custom scale* fields are updated automatically.
   - Select the *Standard scale* field, then click the *Browse* button ( ) and select a scale in the scale list.
     The *Custom scale* field is updated automatically.
   - Select the *Custom scale* field and type the scale factor.
     If the scale factor is not available in the drawings scale list, the *Standard scale* field reads 'Custom'.

5. (option) If necessary, adjust the size of the viewport to the new scale.
6. (option) Set the *Display Locked* property to *Yes*.
   The scale fields are greyed out and cannot be edited anymore.

NOTE In a *locked viewport* it is impossible to:

- zoom, pan or rotate the view
- modify the scale

Setting the Scale of a Selection of Viewports

Command: MVSETUP

1. Make sure the Plot Scale property of the layout is set correctly.
2. Type *mvsetup* in the command bar.
   You are prompted: Enter an option [Align/Create/Scale viewports/Undo]:
3. Do one of the following:
   - Type *S* in the command bar, then press Enter.
   - Choose *Scale viewports* in the prompt menu.

You are prompted: Select entities

4. Select the viewports.
5. Press Enter or right click to stop selecting viewports.
   You are prompted: Set the scaling mode of viewports: Interactively/<Uniform>:
6. Do one of the following:

   - Press Enter to set the scale of the selected viewports uniformly.
     You are prompted: Enter the number of paper space units <1.0>:
     - Press Enter to accept the number of paper space units.
       You are prompted: Enter the number of model space units <1.0>:
     - Type the desired scaling factor: e.g. 50 to apply a 1/50 scale.
     - Press Enter to apply the scale to all selected viewports.
     - Press Enter to conclude the command.
   - Choose *Interactively* in the prompt menu or type *I*, then press Enter.
     The first viewport in the selection set highlights.
     You are prompted: Enter the number of paper space units <1.0>:
     - Press Enter to accept the number of paper space units.
       You are prompted: Enter the number of model space units <1.0>:
     - Type the desired scaling factor: e.g. 50 to apply a 1/50 scale.
     - Press Enter to apply the scale to the viewport.
       The next viewport highlights.
       You are prompted: Enter the number of paper space units <1.0>:
     - Repeat the previous steps for each of the selected viewports.

7. Press Enter to conclude the command.

Rotating the Display in a Viewport

The display of a layout viewport can be rotated only if the display of the viewport is not locked.

If the VPROTATEASSOC system variable is ON, the ROTATE command rotates the content of a viewport over a specified angle.

The *Align > Rotate* option of the MVSETUP command rotates the content of a viewport relative to the X-axis of the world coordinate system (WCS).

Rotating the viewport display

1. Select the viewport, then right click and choose *2D Rotate* in the context menu.
   You are prompted: Rotation point.
2. Use the *Center* ( ) entity snap to select the center point of the viewport as the rotation point.
   You are prompted: Copy/Base angle/<Rotation angle> <0>:
3. Type the rotation angle, then press Enter.
4. The content of the viewport is rotated over the specified angle.

Defining the rotation angle of the viewport display

1. Type *mvsetup* in the command bar.
   You are prompted: Enter an option [Align/Create/Scale viewports/Undo]:
2. Do on of the following:
   - Type *A* in the command bar, then press Enter.
   - Choose *Align* in the prompt menu.

You are prompted: Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:

3. Do one of the following:

   - Type *R* in the command bar, then press Enter.
   - Choose *Rotate view* in the prompt menu.

You are prompted: Specify basepoint in the viewport to be rotate:

4. Specify the rotation point in the viewport.
   You are prompted: Specify rotation angle:
5. Type the rotation angle in the command bar, then press Enter.
   The viewport display is rotated over the specified angle with respect to the X-axis of the WCS.

NOTE Type 0 (zero) in the final step of the above procedure to undo the rotation of the viewport display.

To Align Viewports

Command: MVSETUP

1. Type *mvsetup* in the command bar.
   You are prompted: Enter an option [Align/Create/Scale viewports/Undo]:
2. Do on of the following:
   - Type *A* in the command bar, then press Enter.
   - Choose *Align* in the prompt menu.

You are prompted: Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:

3. Do one of the following:

   - To align two viewports horizontally:
     - Choose *Horizontal alignment* in the context menu or type *H* in the command bar, then press Enter.
       You are prompted: Specify basepoint:
     - Pick a point in the reference viewport.
       You are prompted: Specify point in viewport to be panned:
     - Pick a point in the viewport to be aligned.
       The display in the viewport is moved vertically to align the two points horizontally.
   - To align two viewports vertically:
     - Choose *Vertical alignment* in the context menu or type *V* in the command bar, then press Enter.
       You are prompted: Specify basepoint:
     - Pick a point in the reference viewport.
       You are prompted: Specify point in viewport to be panned:

- Pick a point in the viewport to be aligned.
  The display in the viewport is moved horizontally to align the two points vertically.

To Align Model Space and Paper Space Points

Command: ALIGNSPACE

Aligning 1 point

1. Type *alignspace* in the command bar, then press Enter.
   You are prompted: First alignment point in Model space:
2. Click inside a layout viewport, then pick a point.
   You are prompted: Second alignment point in Model space or <Return> for none:
3. Right click or press Enter.
   You are prompted: Alignment point in Paper space:
4. Pick a point in the layout.
   You are prompted: Activate the viewport which will be used for alignment.
5. Click inside the viewport you want to align.
6. Right click or press Enter to confirm.
   The display in the selected viewport is moved to make the two points coincident.

Aligning 2 points

1. Type *alignspace* in the command bar, then press Enter.
   You are prompted: First alignment point in Model space:
2. Click inside a layout viewport, then pick a point.
   You are prompted: Second alignment point in Model space or <Return> for none:
3. Pick a second point in the layout viewport.
   You are prompted: First alignment point in Paper space:
4. Pick a point in the layout.
   You are prompted: Second alignment point in Paper space:
5. Pick a second point in the layout.
   You are prompted: Activate the viewport which will be used for alignment.
6. Click inside the viewport you want to align.
7. Right click or press Enter to confirm.
   The display in the selected viewport is moved, rotated and/or scaled to make the points two by two coincident.

Layouts

In BricsCAD, you can create up to 255 layouts in a single drawing. Each layout represents a sheet of paper. For each layout you can specify the print area, print scale, lineweight scale, pen mappings, and add viewports, dimensions, a title block, and other geometry specific to the layout. The entities you add to a layout in paper space do not appear in model space.

Each layout requires at least one layout viewport. If the CREATEVIEWPORTS system variable is ON a viewport is added automatically in a new layout. If this variable is OFF, newly created viewports are empty. To add new layouts you can either define them yourself or your can import layouts from another drawing: template file (.dwt), drawing file (.dwg) or drawing interchange file (.dxf).

Viewports can display all or part of the drawing's model space entities.

The CACHELAYOUT system variable controls whether the content of layouts is cached. The extra memory required for caching is made up for by improvements to the memory requirements for displaying entities, the net result being that the memory needed to display drawings has been reduced. As a result of this switching between layouts is up to 10 times faster now.

The Ctrl-PgUp or Ctrl-PgDn keyboard shortcuts allow to open the next or previous layout.

Creating a new Layout

1. Do one of the following:
   - Click the *New Layout* tool button ( ) on the *Layout* or the *Insert* toolbar.
   - Choose *New Layout* in the *Insert* menu.
   - Type *layout* in the command bar, press Enter, then type *N* and press Enter.

You are prompted: Enter a layout name. <Layout1>:

2. Do one of the following:

   - Type a unique name for the layout, then press Enter.
     The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_), or any combination.
   - Right click or press Enter to accept the default name (e.g. Layout1).

The new layout tab is added.

3. Create at least one viewport (see Creating viewports in a layout).

If the CREATEVIEWPORTS system variable is ON a viewport is added automatically in a new layout.

Copying Layouts

1. Right click the layout tab you want to copy.
   A context menu displays.
2. Choose *Copy* in the context menu.
   You are prompted: Enter new layout name:
3. Type a new unique name in the command bar, then press Enter.
   A copy of the selected layout is added.

Importing Layouts

1. Right click the *Model* tab or one of the layout tabs.
   A context menu displays.
2. Choose *From Template...* in the context menu.
   The *Select Template From File* window opens.
3. Browse to the drawing that you want to import layouts from.
4. Click the *Open* button on the *Select Template From File* window.
   The *Insert Layout(s)* window opens.
5. Select the layout(s) you want to import.
   Press and hold the Ctrl key to select multiple layouts.
6. Click the OK button on the *Insert Layout(s)* window.
   The selected layouts are imported.

Renaming Layouts

1. Double click the layout tab you want to rename.
   The *Rename Layout* dialog opens.
2. Type a new unique name in the *Name* field of the *Rename Layout* dialog.
3. Click the *OK* button on the *Rename Layout* dialog.

Arranging Layout Tabs

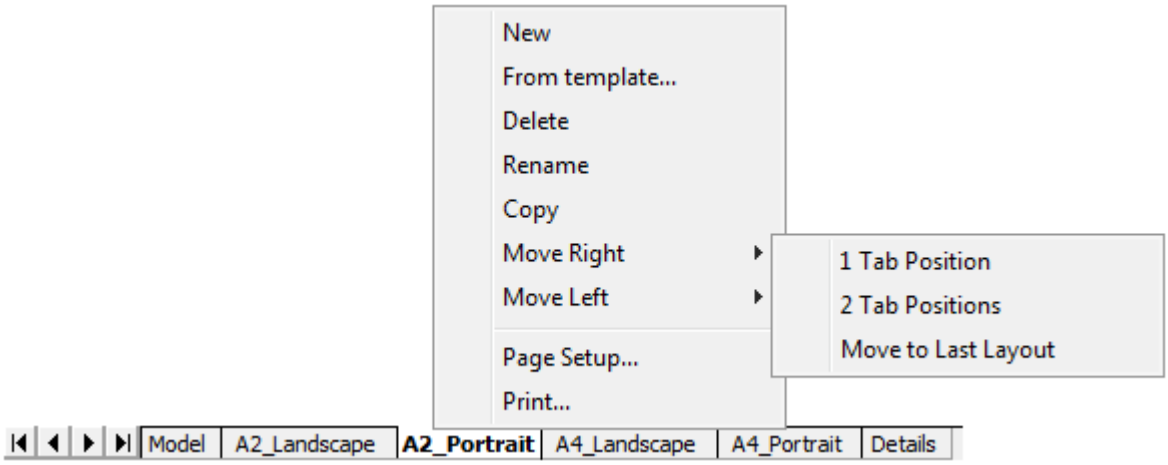
Drag the layout tab to the desired position

or

1. Right click the layout tab you want to move.
   A context menu displays.
2. Do one of the following:
   - Choose *Move Right*.
   - Choose *Move Left*.

3. To move the selected layout tab do one of the following:

   - Select the number of tab positions.
   - Choose *Move to Last Layout* or *Move to First Layout*.



Deleting a Layout

1. Right click the layout tab you want to move.
   A context menu displays.
2. Choose *Delete* in the context menu.
   The selected layout is deleted.

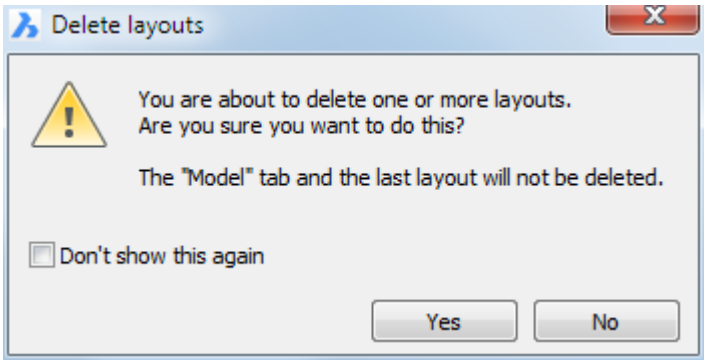
Deleting Multiple Layouts

1. Do one of the following:
   2. Choose *Drawing Explorer > Page Setups...* in the *Tools* menu.
   3. Choose *Page Setup...* in the *File* menu.

The *Drawing Explorer - Page Setups* dialog displays.

2. Select the layouts you want to delete.
   To select a series of layouts: select the first layout, then press and hold the Shift key and select the last layout.
   Press and hold the Ctrl-key to add/remove layouts.

3. Click the *Delete* tool button ( ) in the toolbar on the *Drawing Explorer - Page Setups* toolbar.
   An alert box displays:



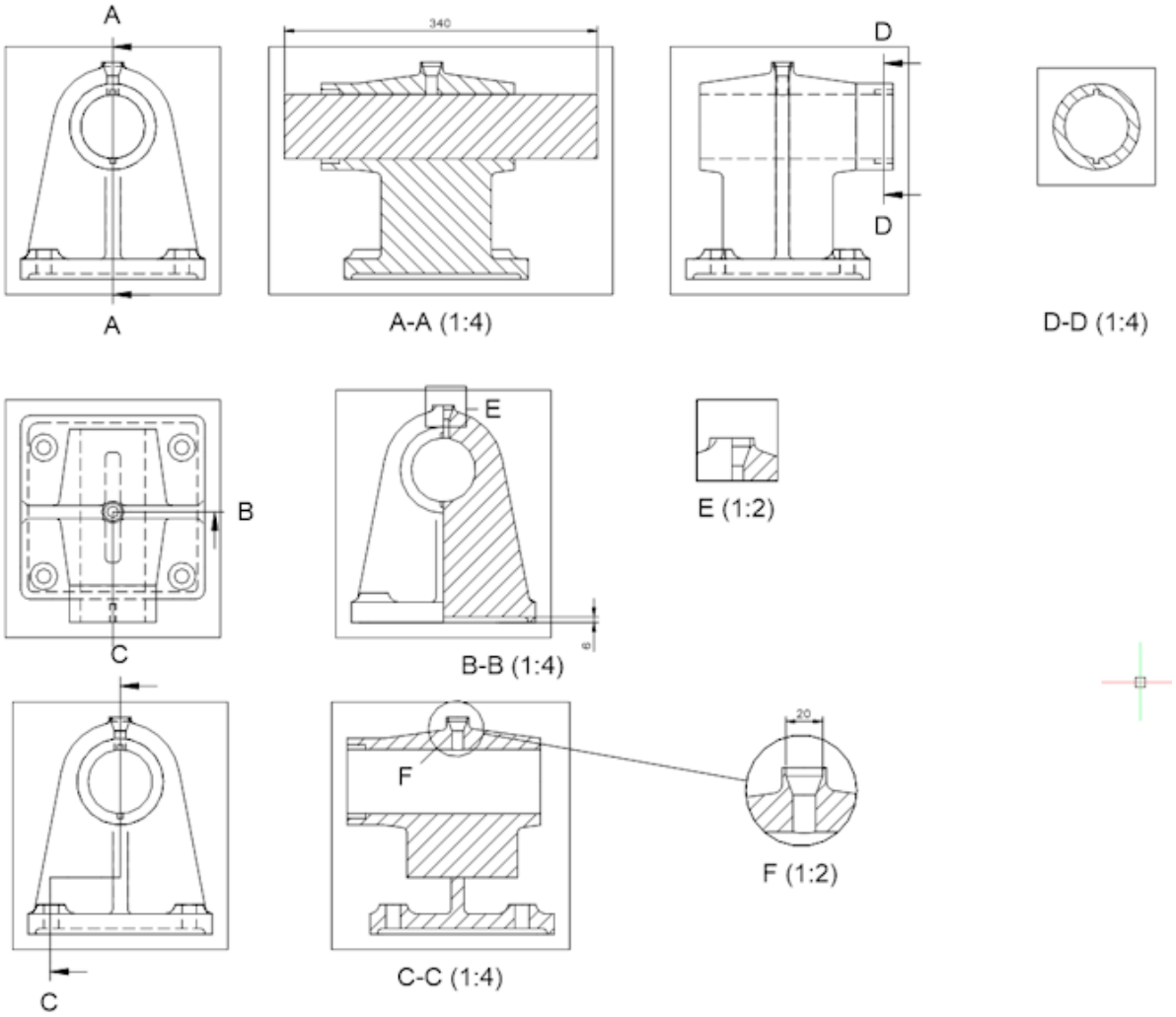
4. Press the *Yes* button on the *Delete Layouts* alert box.

Generated Drawing Views

Commands: VIEWBASE, VIEWSECTION, VIEWSECTIONSTYLE, VIEWPROJ, VIEWDETAIL, VIEWDETAILSTYLE, VIEWUPDATE, VIEWEXPORT and VIEWEDIT

The Generated Drawing Views functionality allows to automatically generate associative orthographic and standard isometric views of a 3D solid model. All drawing views are placed in a paper space layout, they are not visible in model space.

Analytical hidden line removal (HLR) procedures are used to create the drawing views using standard 2D entities, mostly lines and arcs. Much like the result of the FlatShot command. Drawing views created in V14.1 will be automatically converted into HLR geometry by the ViewUpdate command.



The *ViewBase* command generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.

The *ViewSection* command creates a cross section view based on a standard drawing view generated by the *ViewBase* command in a paper space layout.

The *ViewSectionStyle* command allows to modify the appearance of the section views created by the *ViewSection* command.

The *ViewProj* command generates additional projected views from an existing generated drawing view.

The *ViewDetail* command creates a detail view of a portion of a standard generated drawing, at a larger scale.

The *ViewdetailStyle* command allows to modify the appearance of the detail views created by the *ViewDetail* command.

The *ViewUpdate* command Updates a selection of drawing views obtained by *ViewBase* and *ViewSection* when VIEWUPDATEAUTO = 0.

The *ViewExport* command exports the content of drawing views obtained by *ViewBase* and *ViewSection* to the Model Space of the drawing. This command can be used in paper space only.

The *ViewEdit* command allows changing the scale and the hidden line visibility of drawing views.

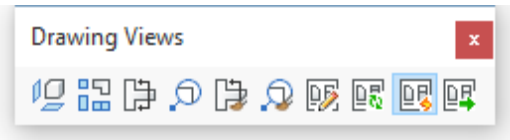
NOTE Drawing views generated in V14.2 or higher will show correctly in BricsCAD V13. The most recent V13.2 version is recommended.

Drawing views generated in V14.2 or higher do not display in V12 and older BricsCAD versions.

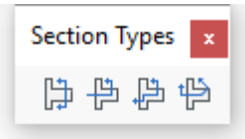
Commands and Toolbars

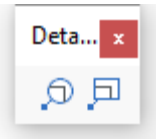
Tools to generate drawing views of your 3D solid model are available on:

The *Drawing Views* toolbar, which is a flyout of the *View* toolbar:

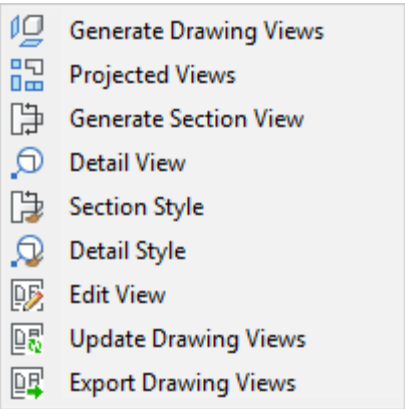


The S*ection Types* and *Detail Section Types* toolbars:

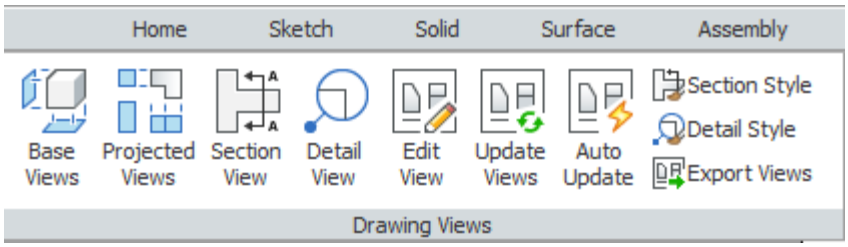




The *View | Drawing Views* menu:



The *Drawing Views* panel of the *Annotate* Ribbon tab.



Standard Drawing Views

Standard views include multiview orthographic and isometric projections.

In a technical drawing, a multiview orthographic projection is an illustration technique in which up to six images of an object are generated, with each projection plane parallel to one of the coordinate axes of the object.

Isometric projection is a method represent three-dimensional objects in two dimensions in technical and engineering drawings. It is an axonometric projection in which the three coordinate axes appear equally foreshortened and the angles between any two of them are 120 degrees.

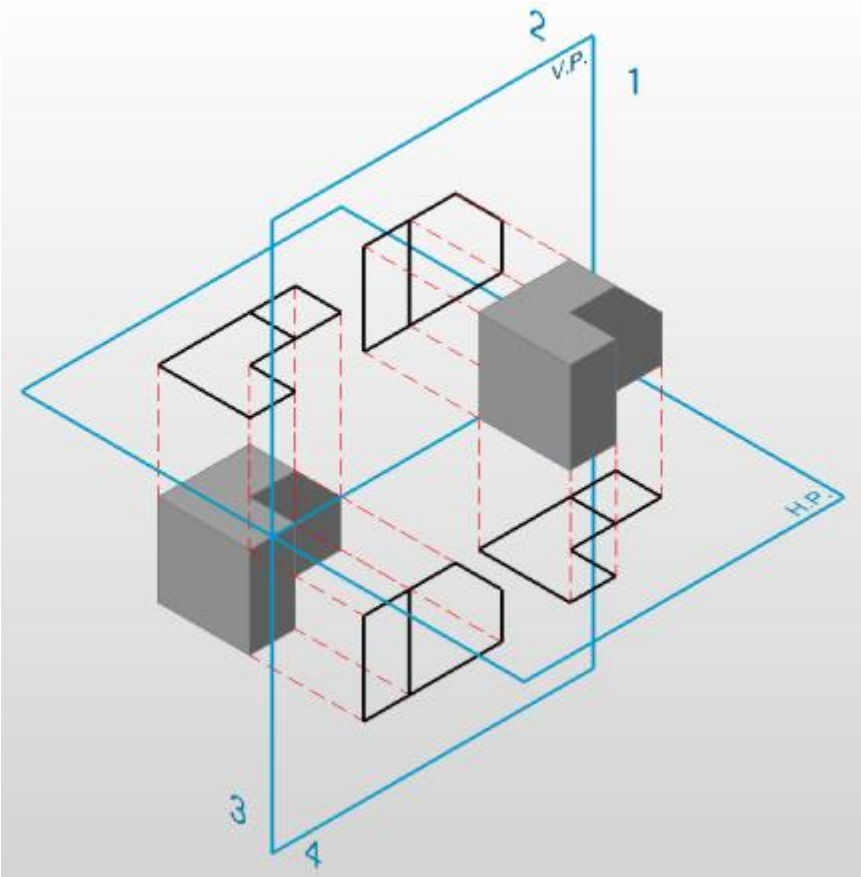
To generate the standard views of your 3D solid model, run the ViewBase command from model space.

You can choose between first angle (European) projection or third angle (American) projection.

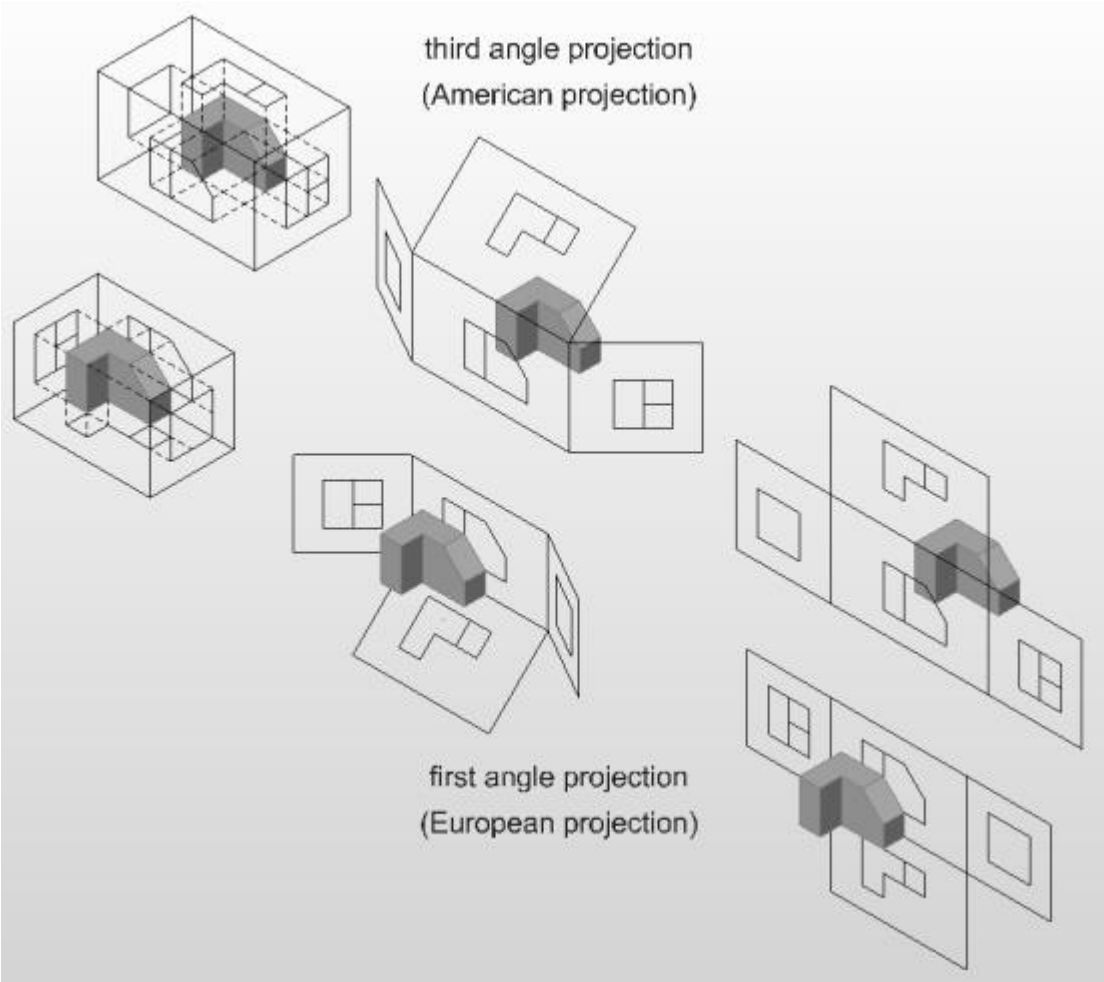
Orthographic projection (or orthogonal projection) is a means of representing a three-dimensional object in two dimensions. It is a parallel projection, where all the projection lines are orthogonal to the projection plane, resulting in every plane of the scene appearing in affine transformation on the viewing surface. It is further divided into multiview orthographic projections and axonometric projections.

Two projection planes, one horizontal (H.P.) and one vertical (V.P.) divide the space in four quarters.

- In the first angle projection system the model is placed in the first quarter space (first angle).
- In the third angle projection system the model is placed in the third quarter (third angle).



With multiview orthographic projections, up to six pictures of a 3D model are produced, with each projection plane parallel to one of the coordinate axes of the model. The views are positioned relative to each other according to either of two schemes: first-angle or third-angle projection. In each, the appearances of views may be thought of as being projected onto planes that form a 6-sided box around the model.



The projection type is set by the *Projection Type* option of the ViewBase command or by editing the PROJECTIONTYPE system variable:

- 0 = First angle projection
- 1 = Third angle projection

The value of the PROJECTIONTYPE system variable is saved in the drawing.

Generated drawing view lines are created on default layers:

- BM_Isometric_Hidden
- BM_Isometric_Visible
- BM_Ortho_Hidden
- BM_Ortho_Visible

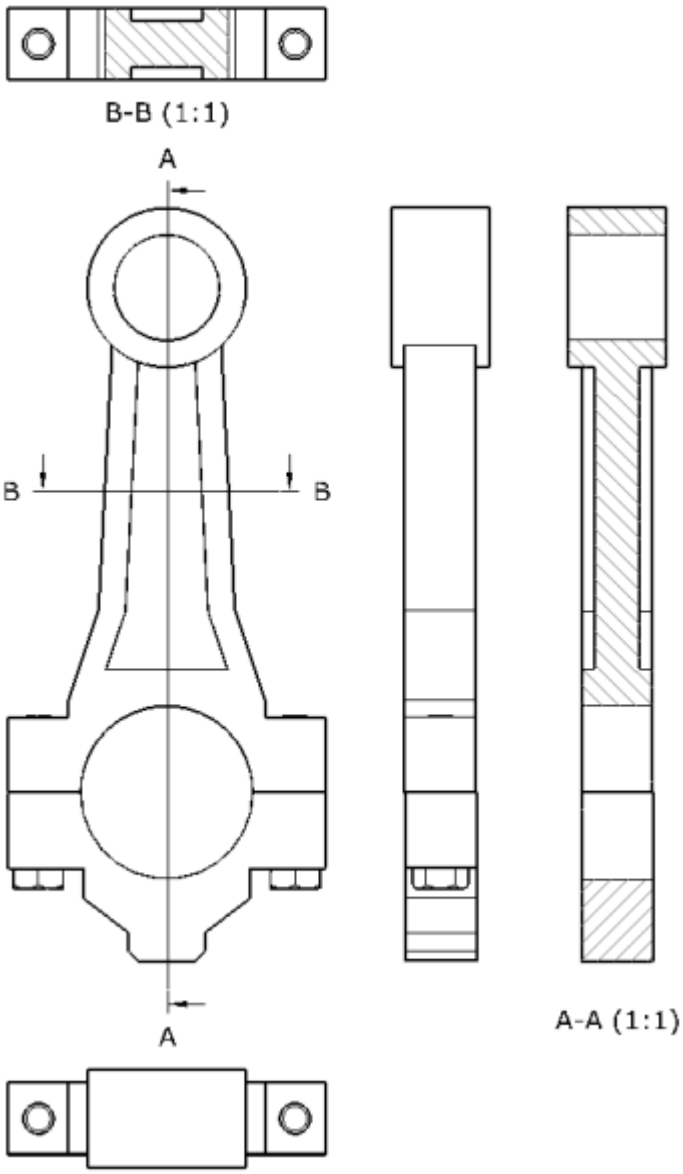
The display of the *BM_Isometric_Hidden* layer is turned off by default.

Additional projected views can be added using the ViewProj command. A projected view inherits the scale, display settings and alignment from the parent view.

Generating Cross Section Views

A cross section is the intersection of a 3D solid model with a section plane. In technical drawings, the internal parts of the 3D model are hatched in cross sections.

To generate section views of a 3D solid model, run the ViewSection command from the paper space layout where the standard drawing views exist.



Updating Drawing Views

By default BricsCAD checks whether the source 3D solid model was modified and automatically recalculates the drawing when:

- opening a paper space layout containing out-of-date views if the VIEWUPDATEAUTO system variable is ON.
- running the ViewUpdate command.
- running the BmUpdate command (BricsCAD Platinum only).

NOTES

- Hidden line removal calculation might take some time for complex 3D models. In such cases it is recommended to set VIEWUPDATEAUTO = OFF. The viewport border of out-of-date drawing views turns red to indicate an update is needed. When executing the ViewUpdate command, BricsCAD prompts you to either select the drawing views to be updated or update all drawing views.
- If GENERATEASSOCVIEWS = ON, associative dimensions will be updated according to the modified 3D geometry.

Exporting Views to Model Space

The associativity mechanism of drawing views in a paper space layout allow limited control by the user. Such views can be moved or scaled, but you cannot edit the geometry. In order to get the full control to the drawing views geometry the ViewExport command allows to move or copy drawing views to model space or to a separate drawing. Exported drawing views loose their associativity with the 3D model and become standard blocks, which can be exploded, edited, erased, etc&ldots;

Customizing drawing Views

The first time the ViewBase command is launched, four new layers are created:

- *BM_Ortho_Hidden*: hidden lines in drawing views
- *BM_Ortho_Visible*: visible lines in drawing views
- *BM_Tangent_Hidden*: hidden tangent lines in drawing views
- *BM_Tangent_Visible*: visible tangent lines in drawing views

Use the ViewEdit command to change the visibility of these layers for all view or for selection of views only.

- Edit the properties of these layers, such as linetype, lineweight or color, to modify the display of the drawing views.
- Use ViewSectionStyle command to adjust different properties of section views such as:style, color, and height of identifier text
- start and end symbols, their colors, size and direction of arrows scale, patter, color, and angle of hatches
- color, type, and weight of a line representing a cutting plane

Use the ViewDetailStyle command to adjust different properties of details views such as:

- style, color, height of identified text, and the type of identifier
- symbol, and its color and size
- color, type, and weight of detail view boundary

Creating Entities

In a drawing you can create a variety of different entity types. Drawing entities can be very simple such as lines, circles, arcs, points and rays or complex such as polylines, splines and planes. Apart from 2D entities you can also create 3D surface entities or 3D solids in BricsCAD.

To further complete your drawing BricsCAD provides a series of tools to add text, hatching and dimensions.

To create an entity you can choose between selecting the command in the Draw menu, using the tools on the Draw, Draw 3D and Solids toolbars or typing the command in the command bar.

When you use a tool or a drawing command, the program prompts you to enter coordinate points, such as endpoints or insertion points. You can enter the points or distances either using a mouse or by typing coordinate values in the command bar. As you draw, BricsCAD also displays a context-sensitive prompt menu with appropriate additional options for the type of entity you are drawing.

New entities are created on the current layer, using the current color, linetype and lineweight.

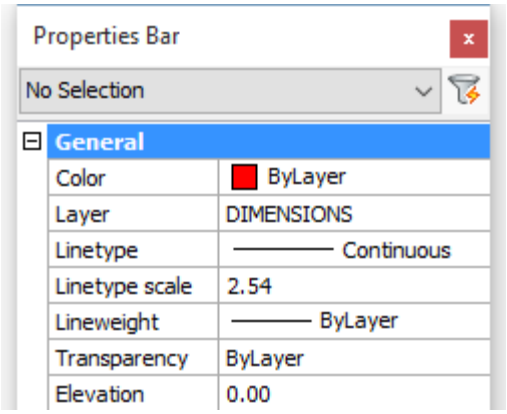
Entity Creation Settings Overview

The following settings control the display and/or creation of entities.

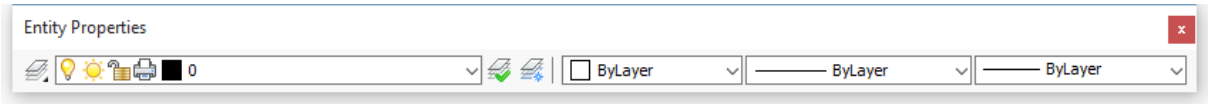
Name	Command or System Variable	Description
Creation		
Current Entity Color	COLOR	Sets the color of new entities.
Current Layer	CLAYER	Sets the layer for new entities.
Current Entity Linetype	CELTYPE	Sets the linetype for new entities.
Current Entity Linetype Scale	CELTSCALE	Sets the linetype scaling factor for new entities.
Current Entity Lineweight	CELWEIGHT	Sets the lineweight for new entities.
Transparency	CETRANSPARENCY	Sets the transparency level for new entities.
Elevation	ELEVATION	Specifies the current elevation for new entities relative to the current UCS.

Display		
Lineweight Display	LWDISPLAY	Controls whether lineweights display on the screen.
Default Lineweight	LWDEFAULT	Defines the default lineweight.
Lineweight Units	LWUNITS	Defines whether lineweights are expressed in millimeters or inches.
Linetype Scale	LTSCALE	Sets the linetype scaling factor for all entities in the drawing (global linetype scale).
Transparency Display	TRANSPARENCYDISPLAY	Specifies whether entity transparencies display in 2DWireFrame visual style. Rendered visual styles always display entity transparency.
Whip Arcs	WHIPARC	Determines whether circles, arcs, ellipses, elliptical arcs and arc segments of polylines display as true curves or as a series of vectors.
Fill mode	FILL	Specifies whether multilines, traces, solids, hatches (including solid-fill) and wide polylines are filled in.

The current properties display in the *Properties Bar* and in the *Entity Properties* toolbar.



Properties Bar



Entity Properties toolbar

Entity Creation Settings

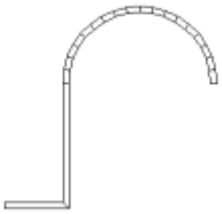
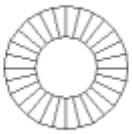
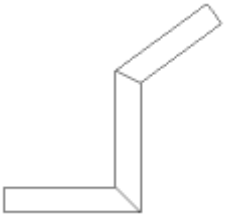
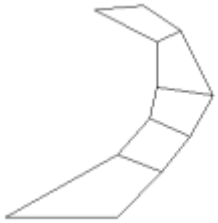
Fill Mode

Command: FILL

The *Fill* command toggles the display of filled and hatched areas. Specifies whether multilines, traces, solids, hatches (including solid-fill) and wide polylines are filled in.

If *FILLMODE* is OFF, all filled entities display and print as outlines.

You can reduce the time it takes to display or print a drawing by turning FILLMODE off.

	Polyline	**Donut**	**Trace**	**Plane (Solid)**
Fill On				
Fill Off				

Setting Fill Mode

1. To toggle *Fill Mode* on / off do one of the following:
   - Choose *Fill* in the *Settings* Menu.
   - Type *fill* or *fillmode* in the command bar, then type *T* + Enter.

2. Regenerate the viewport.

NOTE If multiple viewports are open, click the Regen All tool button ( ) on the *View* toolbar (*Redraw/Regen* flyout) to regenerate all open viewports simultaneously.

Entity Color

The *Color* command specifies the current color, through a dialog box.

The *-Color* command sets the current color, through the command bar.

An entity's color determines how it is displayed and how it prints. Entities are created in the current color.

You can choose between 255 *index colors* or define a *true color.*

Index color

Index color is the specification of the color of a pixel on a display screen using a an 8-bit color value, allowing up to 256 possible colors.

Each of the Index Colors has a unique number from 1 to 255. Seven of the index colors can also be referred to by name: red (1), yellow (2), green (3), cyan (4), blue (5), magenta (6) and white/black (7). Index color 7 displays white on a black screen background and black on a white screen background. Index color 7 always prints in black.

The two additional color properties are BYLAYER and BYBLOCK. These color properties cause an entity to adopt the color either of the layer or of the block in which it is a member. BYLAYER is color number 256, and BYBLOCK is color number 0. In all commands where you would use a color, you can indicate BYLAYER and BYBLOCK as well as by numbers 256 and 0, respectively.

Color BYLAYER:

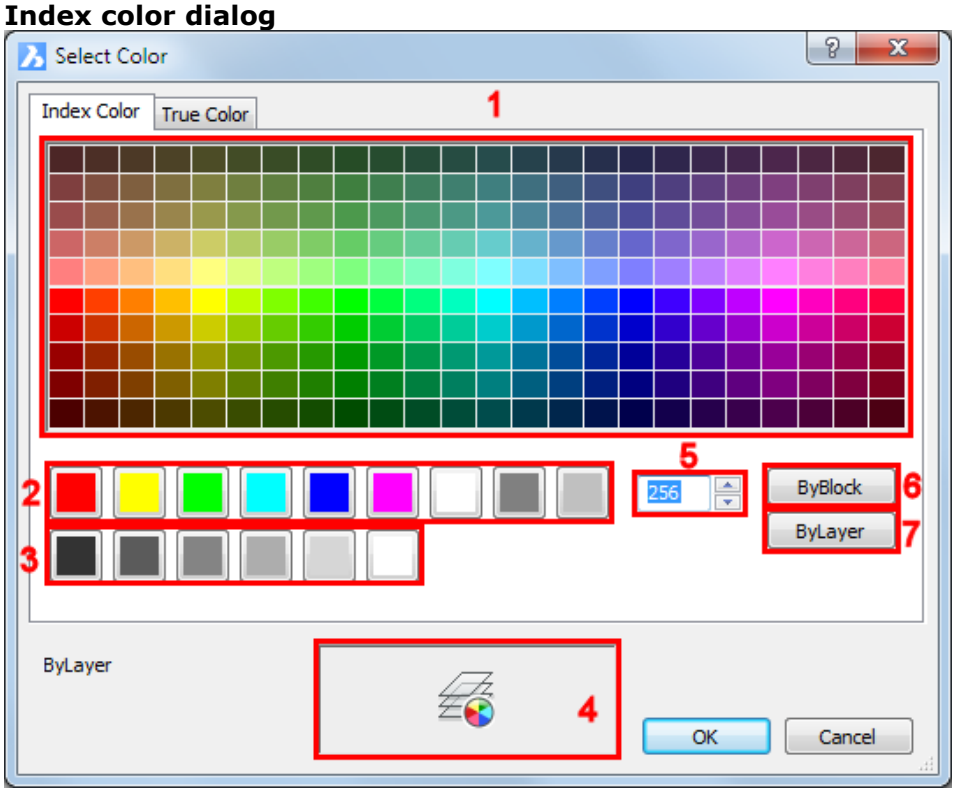
Entities which have a color *BYLAYER* adopt the color of their layer. This allows you to change the color of all such entities by adjusting the color of the layer.

Color BYBLOCK:

Entities which have a color *BYBLOCK* are drawn in index color 7 (black or white, depending on the screen background color). When included in a block definition, such entities adopt the color of the block.

NOTE Whether entities using an index color will print in this color, depends on the *Color Table* (CTB) or *Style Table* (STB) that is used for printing. Only if the *Color* setting in the CTB or STB definition file is set to *Use Object Color*, the printed color matches the entity color.

Index color dialog



1. Colors 10 - 249
2. Colors 1 to 9
3. Colors 250 - 255
4. Current Color
5. Color Number field
6. 'Color By Block' button
7. 'Color By Layer' button

True Colors

True color is the specification of the color of a pixel on a display screen using a 24-bit value, which allows up to 16 777 216 possible colors.

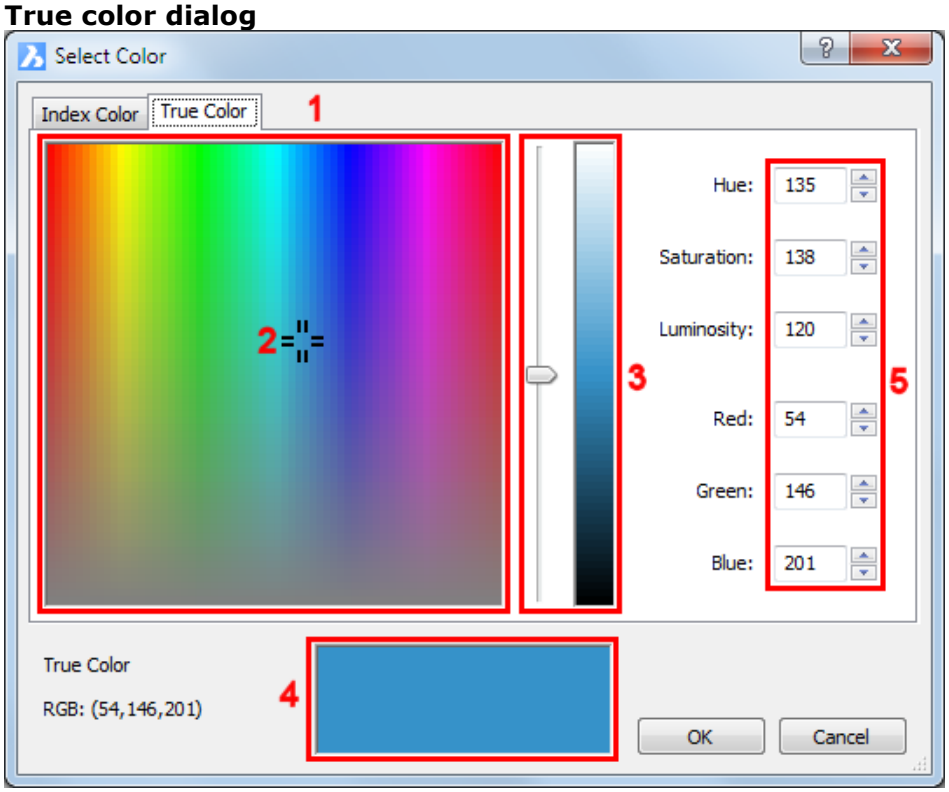
The number of bits used to define a pixel's color shade is its *bit-depth*. True color is sometimes known as *24-bit color*. Some new color display systems offer a 32-bit color mode. The extra byte, called the *alpha channel*, is used for control and special effects information.

True colors use a *RGB* color definition (Red, Green, Blue). Each of these parameters has a range from 0 to 255. The RGB definition for black is (0,0,0), the RGB definition for white is (255,255,255).

NOTE

- In drawings that use *Style Tables* (STB) to set up plotting configurations, entities in a true color will only be printed in this color if the *Color* setting in the STB definition file is set to *Use Object Color.*
- In drawings that use *Color Tables* (CTB) to set up plotting configurations, entities in a true color always print in this color.

True color dialog



1. Color Selection pane
2. Color Picker
3. Luminosity slider
4. Current Color
5. Color Parameter fields

To Set the Current Entity Color

1. To open the *Select Color* dialog, do one of the following:
   - In the *Color* field of the *Entity Properties* toolbar, choose *Select Color*.
   - Click the *Select Color* tool button ( ) on the *Settings* toolbar.
   - In the BricsCAD Properties Bar click *Color*, then choose *Select Color* in the drop down list.
   - Double click the *Color Field* in the Status Bar.
   - Type *color* in the command bar, then press Enter.

2. (option) To select an index color, click the *Index Color* tab on the *Select Color* dialog, then do one of the following:

   - Click one of the colored tiles.
   - Type the color number in the *Color Number* field.

3. (option) To define a true color, click the *True Color* tab on the *Select Color* dialog, then do one of the following:

   - Click the in *Color Selection* pane.
   - Set the color parameters in the *Color Parameter* fields.

4. Click the *OK* button to confirm.

Lineweight

Lineweights determine how thick or thin entities appear on the screen and / or when printed.

The following lineweights are available: BYLAYER, BYBLOCK, DEFAULT and many additional lineweights in millimeters or inches, depending on the *Lineweight Units* (LWUNITS) system variable..

You cannot assign lineweights to planes, points, TrueType fonts, and raster images.

New entities are drawn using the current lineweight.

Lineweight BYLAYER:

When you create an entity, it is created using the current lineweight. By default, the current lineweight for a new entity is BYLAYER. This means that the entity lineweight is determined by the current layer. When you assign BYLAYER, changing a layer's lineweight changes the lineweight of all such entities on that layer.

Lineweight BYBLOCK:

Entities created using lineweight BYBLOCK, are drawn using the DEFAULT lineweight until include them into a block. The entities then inherit the block's lineweight setting when you insert the block into a drawing.

DEFAULT lineweight:

The default lineweight is saved in the *Default Lineweight* (LWDEFAULT) system variable. Entities created using the default lineweight will be adjusted if the *Default Lineweight* setting is redefined.

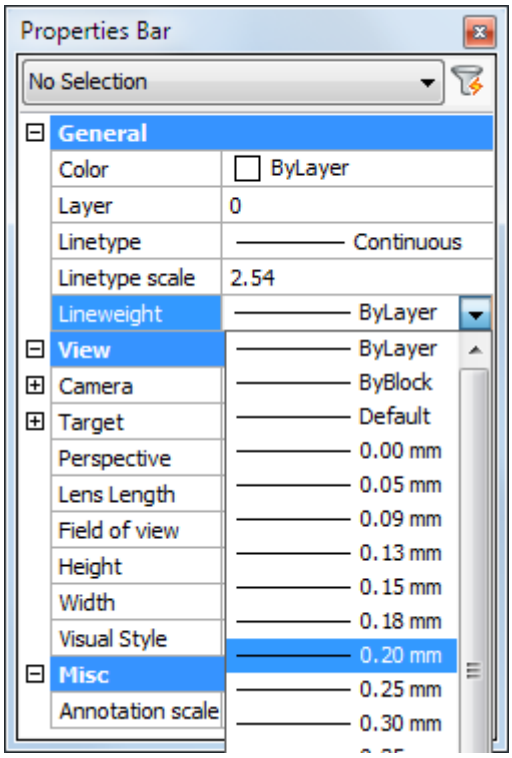
NOTE For entities using an index color, lineweights will be used for printing only if the *Color Table* (CTB) or *Plot Style* (STB) is defined as 'Use object lineweight'. Otherwise the line weight is defined by the objects color or plot style.

Entities which are created in a true color always use the object line weight when printed.

Setting Current Lineweight

1. (option) On the BricsCAD Properties Bar, select *Lineweight*.
   Make sure, no entity is selected.

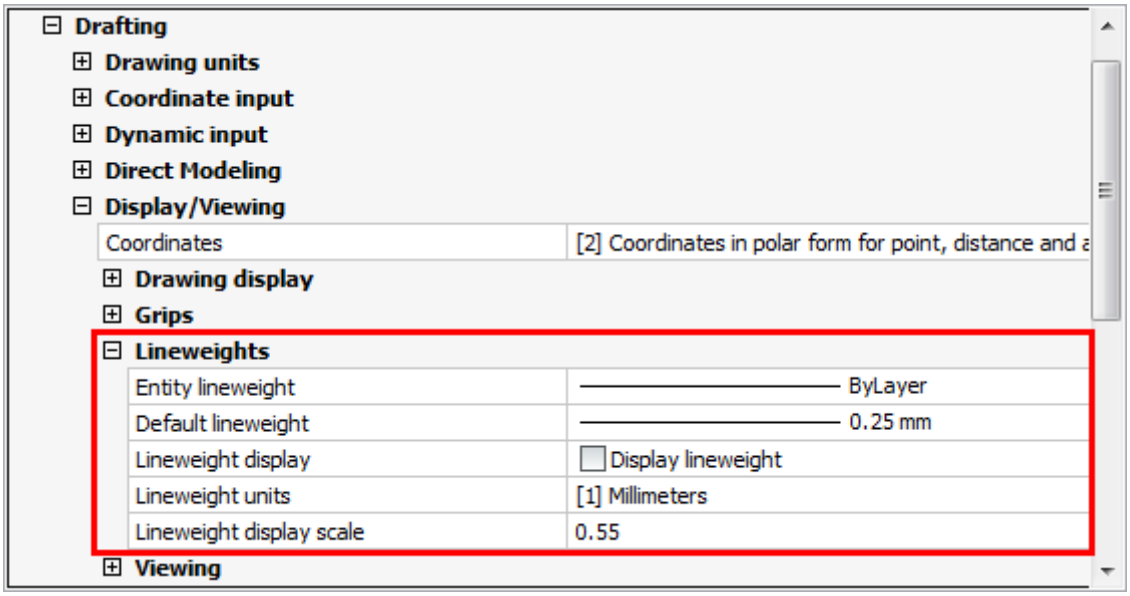
Choose a lineweight in the drop down list.



2. (option) On the *Entity Properties* toolbar, click the down arrow next to the *Lineweight* field, then choose a lineweight in the drop down list.

Defining the Lineweight Settings

1. Choose *Settings* in the *Settings* menu.
   The Settings dialog opens.
2. Select the *Drawing settings* category.
3. Expand the *Display/Viewing* settings sub-category.
4. Expand the *Lineweights* settings group.



5. (option) Select the *Default Lineweight* (LWDEFAULT) system variable, then select a lineweight in the drop down list.
6. (option) Select the *Lineweight Display* (LWDISPLAY) system variable, then click the checkbox to toggle the display of lineweights.
7. (option) Select the *Lineweight Units* (LWUNITS) system variable, then choose *Millimeters* or *Inches*.
8. Close the *Settings* dialog.

NOTE Click the lineweight field (LWT) in the Status Bar to toggle the display of lineweights on/off.

Linetype

You use different linetypes to differentiate the purpose of one line from another. A linetype consists of a repeating pattern of dots, dashes, or blank spaces. Linetypes determine the appearance of entities both on the screen and when printed. By default, every drawing has at least three linetypes: CONTINUOUS, BYLAYER and BYBLOCK. Your drawing may also contain an unlimited number of additional linetypes. Linetype definitions are saved in the drawing. New linetypes are either imported from other drawings or loaded from a linetype file (*.lin). Which linetype file is used is controlled by the MEASUREMENT system variable:

- MEASUREMENT = ON uses *Iso.lin*: Linetype definitions are expressed in mm.
- MEASUREMENT = OFF: *Default.lin*: Linetype definitions are expressed in inch.

New entities are drawn using the current linetype.

Linetype BYLAYER:

Entities which have a linetype BYLAYER adopt the linetype of their layer. This allows you to change the linetype of all such entities by adjusting the linetype of the layer.

Linetype BYBLOCK:

Entities created using linetype BYBLOCK, are drawn as continuous lines until you include them into a block. The entities then inherit the block's linetype setting when you insert the block into a drawing.

Linetype scale (LTSCALE)

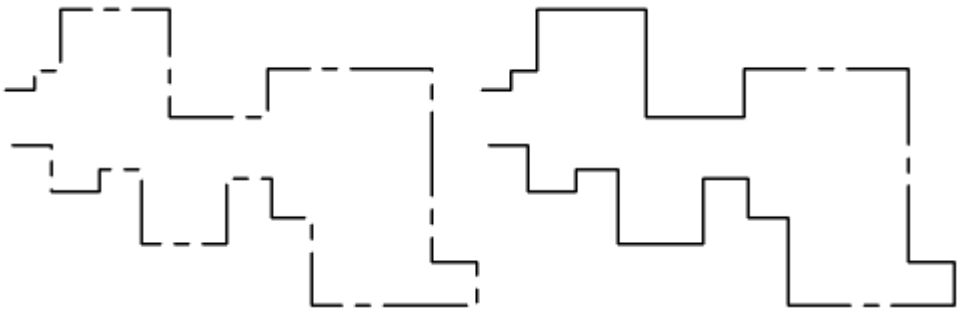
Non-continuous linetypes consist of lines, gaps and dots put together in a variety of patterns. In complex linetypes also text or shapes can be included.

In the linetype definition the length of the lines and gaps are either defined in inch (default.lin) or mm (iso.lin). If the drawing units are different from inch or mm, linetypes need to be scaled. The linetype scaling is controlled by the *Linetype Scale* (LTSCALE) system variable, which must be set with respect to the drawing units (DU):

Drawing Unit	**LTSCALE**
Imperial (MEASUREMENT = OFF)	
Inch	1
Foot	1/12
Metric (MEASUREMENT = ON)	
mm	1
cm	0.1
m	0.001

BricsCAD will always put a dash at the start and end of a line, polyline, arc or spline. The linetype pattern is also nicely centered, so that both ends of the entity look the same.

Polyline Generation (PLINEGEN)
Linetypes are normally generated from vertex to vertex. Polylines of which the vertices are very close together might be rendered as a continuous line, if the linetype pattern does not fit between two subsequent vertices. The *Polyline Generation* (PLINEGEN) system variable addresses this problem: when set to 1 or ON, the linetype is drawn from one end of the polyline to the other end, instead of from vertex to vertex.



The same polyline drawn with PLINEGEN on (left) and off (right).

Paper space linetype scale (PSLTSCALE)
PSLTSCALE controls linetype scaling in paperspace layouts. If ON, linetype scaling is independent from the viewport scale.

In general, PSLTSCALE is best set ON. Remember that this variable can be set per layout.

Model space linetype scale (MSLTSCALE)
If MSLTSCALE is ON, the current annotation scale (CANNOSCALE) is applied to linetypes in model space.

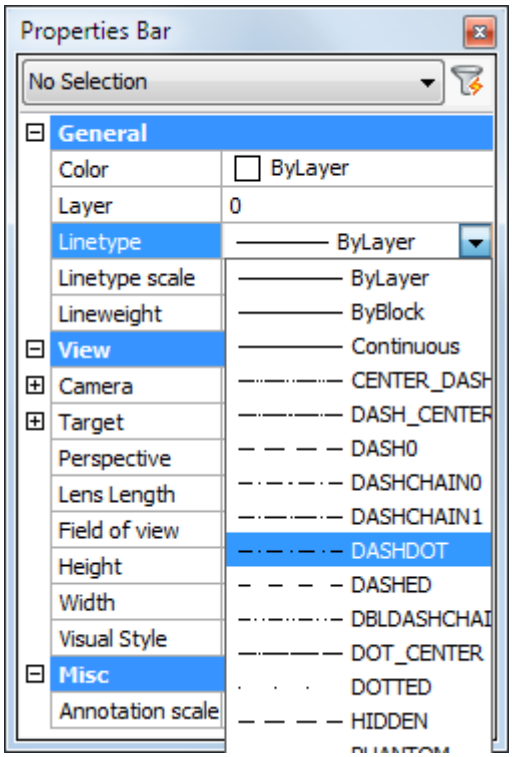
MSLTSCALE is best set OFF. It might be usefull to switch it ON when printing from model space.

Current Linetype

Do one of the following:

- On the BricsCAD Properties Bar, select *Linetype*.
  Make sure, no entity is selected.

Choose a linetype in the drop down list.



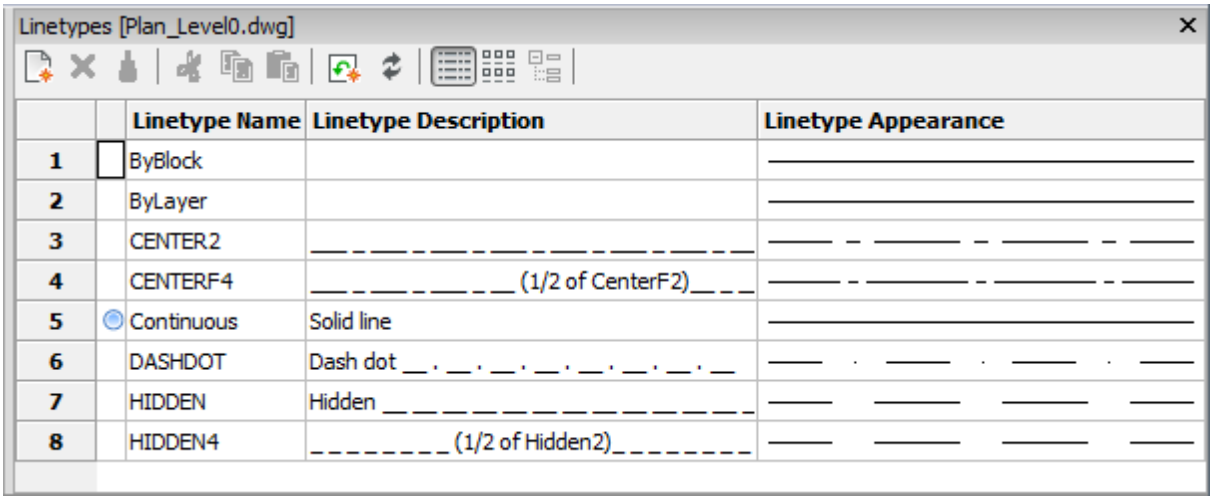
- On the *Entity Properties* toolbar, click the down arrow next to the *Linetype* field, then choose a linetype in the drop down list.

- Right click on the *Linetype* field in the Status Bar, then choose a linetype in the context menu.

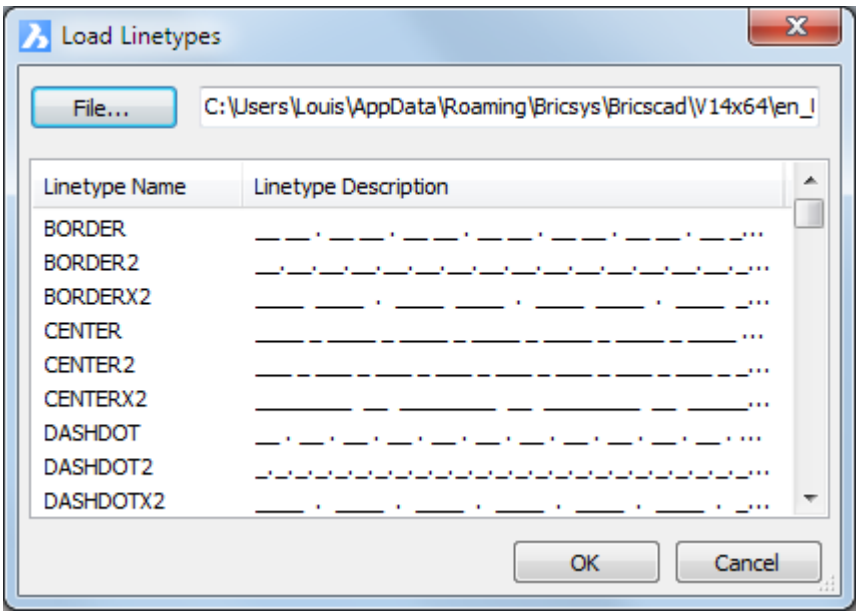
Add a New Linetype

1. In the *Tools > Drawing Explorer* choose *Linetype...*

The *Drawing Explorer - Linetypes* dialog displays.



2. Click the New tool button ( ) in the Details toolbar.
   The Linetypes dialog opens.

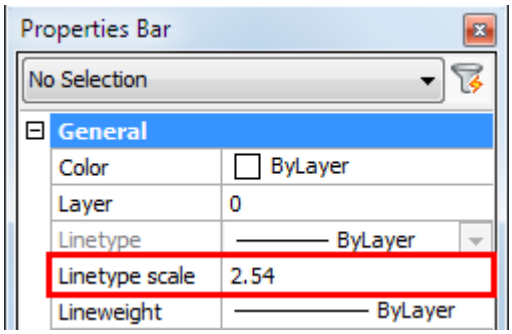


3. (option) Click the *File* button on the *Linetypes* dialog to load a different linetype file (*.lin).
4. Select a linetype, then click the *OK* button.
   The linetype is added.
5. (option) Click the linetype name to rename the linetype.
6. (option) click the linetype description to edit the description of the linetype.

Entity Linetype Scale

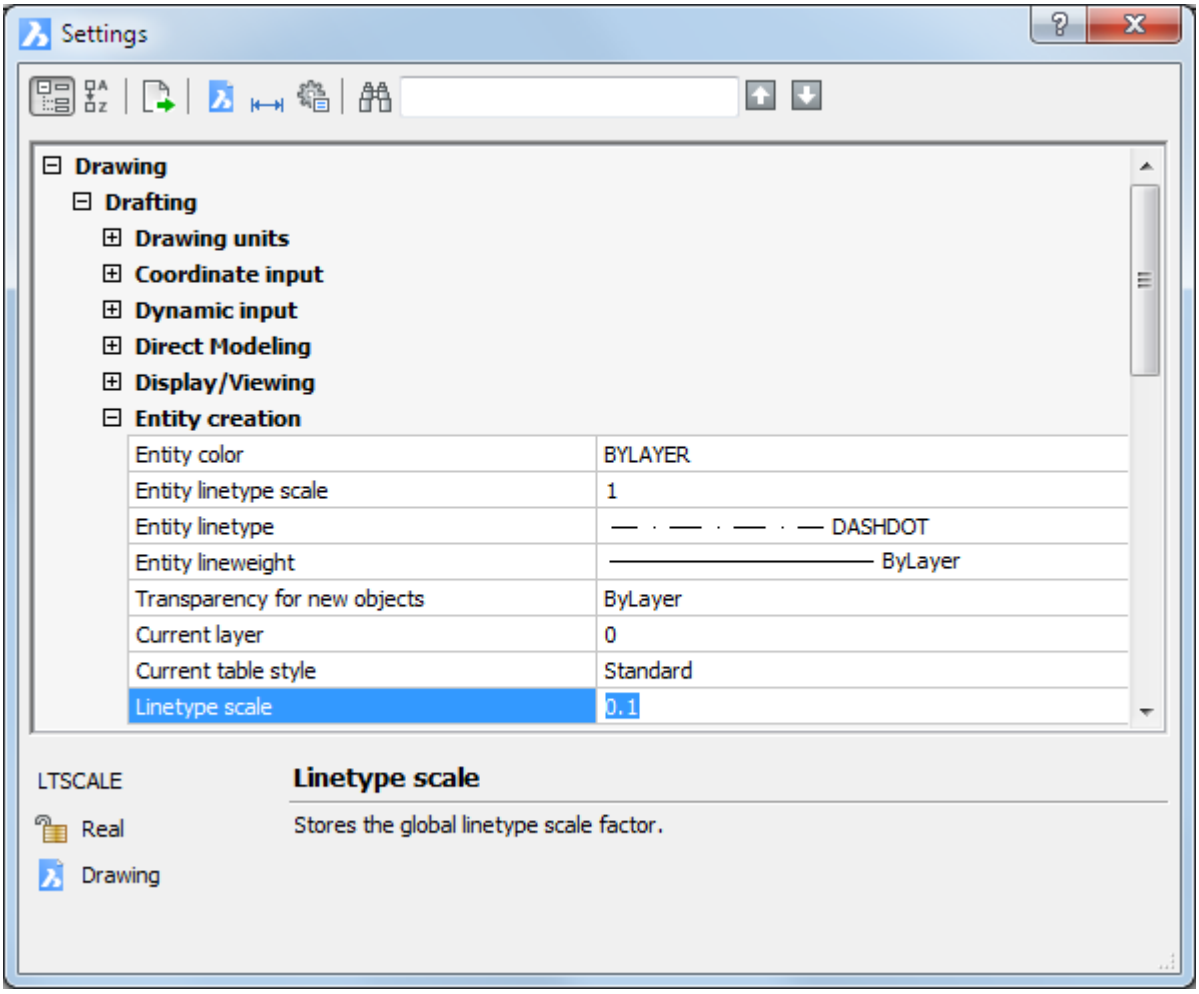
The CELTSCALE system variable sets the value of the Linetype Scale property for new entities. This variable is best set to 1, except for special linetypes, such as BATTING or GAS_LINE, which most likely need to be scaled.

1. On the BricsCAD Properties Bar, select *Linetype scale*.
   Make sure, no entity is selected.
2. Type a new value in the Linetype scale field.



Global Linetype Scale

1. Choose *Settings* in the *Settings* menu.
   The Settings dialog opens.
2. Select the *Drawing settings* category.
3. Expand the *Entity Creation* settings sub-category.
4. Select the *Linetype Scale* setting (LTSCALE).



5. Type a new value in the *Global Linetype Scale* field, then press Enter.
   The drawing is regenerated.
6. Close the *Settings* dialog.

Current Layer

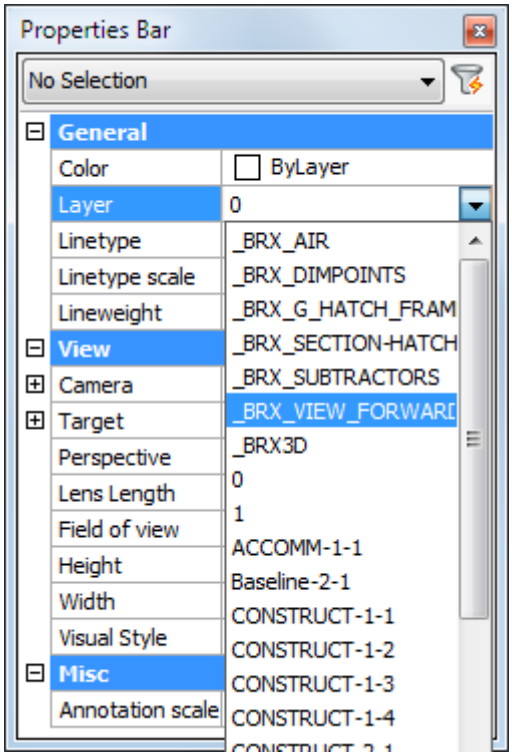
When you create new entities, they are drawn on the current layer. To draw new entities on a different layer, you must first make that layer the current layer.

Setting the current layer

Do one of the following:

- On the BricsCAD Properties Bar, select *Layer*.
  Make sure, no entity is selected.

Choose a layer in the drop down list.



- On the *Entity Properties* toolbar, click the down arrow next to the *Layer* field, then choose a layer in the drop down list.
- Right click on the *Layer* field in the Status Bar, then choose a layer in the context menu.

NOTES

- It is possible to turn the display of the current layer off ( ). As a result, everything you draw will be invisible until the display of the current layer is turned on ( ) again. A warning displays if you try to turn off the current layer.
- The LAYISO command allows to isolate layers by either lock or turn off other layers. If no entity on the current layer is selected, the layer of the entity that is selected first temporarily becomes the current layer until the LAYUNISO command is executed.
- It is not possible the freeze ( ) the current layer.

Transparency

The Transparency property allows to control the transparency of drawing entities. You can set the transparency for a selection of entities or for all entities on the same layer. The default transparency for new entities is controlled by the CETRANSPARENCY system variable. The default transparency for Hatches and Gradient Fills is controlled by the HPTRANSPARENCY system variable.

Transparency can be used to enhance the display of your drawings, e.g. to de-emphasize less important details or entities that are used for reference only.

Transparencies will only be printed if the Plot Transparencies option is checked on the Print or Page Setup dialog boxes.

The options to set the Transparency are:

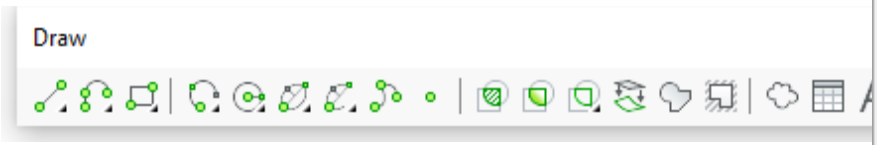
- **ByLayer** - apply the value of the transparency property of the layer.
- **ByBlock** - the transparency value is controlled by block.
- **Value** - values between 0 (fully opaque) and 90 are accepted.
  The transparency level is limited to 90 % to avoid confusion with layers that are frozen or turned off.

Drawing 2D Entities

Drawing 2D Overview

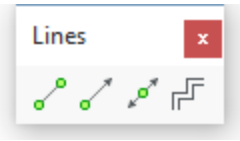
All tools to create 2D entities can be found either on the *Draw* toolbar or in the *Draw* menu.

Toolbars

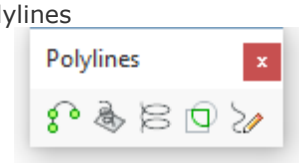


Draw flyouts:

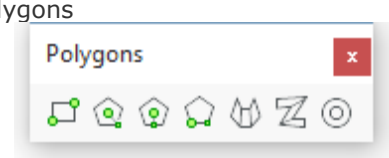
Lines



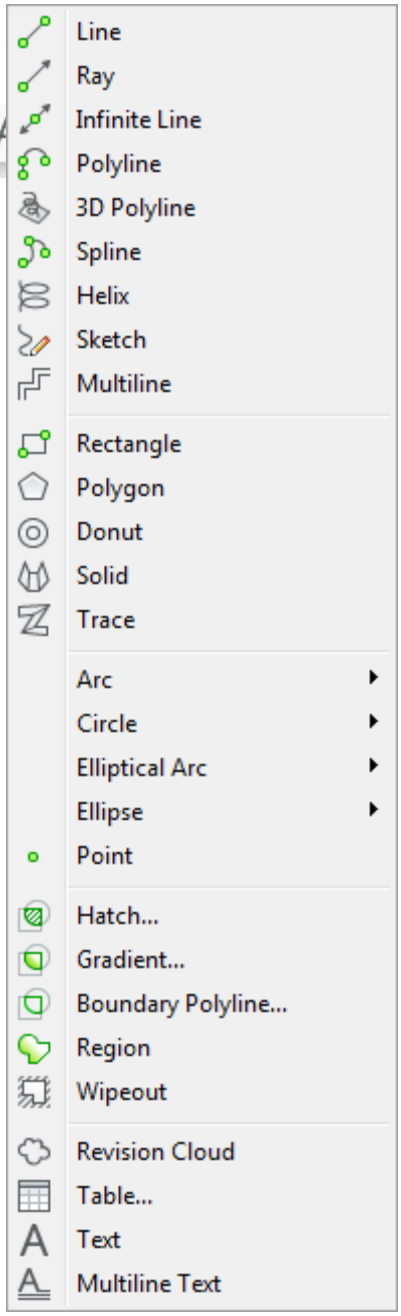
Polylines



Polygons



Menu



Icon	Tool Name	Command	Description
	Line	LINE	Creates single lines or a series of connected lines.
	Ray	RAY	Creates a line that starts at a point and extends to infinity.
	Infinite Line	XLINE	Creates a line through a given point, oriented at a specified angle and extending to infinity in both directions.
	Polyline	PLINE	Creates a single open or closed entity, composed of lines and/or arcs.
	3D Polyline	3DPOLY	Creates 3D polylines.
	Boundary Polyline	BOUNDARY	Calculates a closed polyline from a boundary set.
	Spline	SPLINE	Creates open or closed curved lines.
	Helix	HELIX	Creates a 2D spiral or a 3D helix.
	Sketch	SKETCH	Creates lines as if you were sketching, using a pencil.
	Rectangle	RECTANG	Creates Rectangles.
	Polygon	POLYGON	Creates equal sided polygons
	Solid	SOLID	Creates planes (solids).
	Trace	TRACE	Creates traces.
	Arc	ARC	Creates circular arcs.
	Circle	CIRCLE	Creates circles.
	Donut	DONUT	Creates donuts.
	Elliptical Arc	ELLIPSE+A	Creates elliptical arcs.
	Ellipse	ELLIPSE	Creates ellipses.
	Point	POINT	Creates points.
	Hatch	HATCH	Fills closed areas with repeating patterns.
	Gradient Fill	GRADIENT	Fills closed areas with solid fills in nine different patterns and one or two colors.
	Region	REGION	Creates regions from 2D entities.
	Wipeout	WIPEPOUT	Creates wipeouts, blank areas in the drawing.
	Revision Cloud	REVCLOUD	Creates revision clouds.
	Table	TABLE	Creates a table entity in the drawing through a dialog box.
	Text	TEXT	Places single lines of text in the drawing.
	Multiline Text	MTEXT	Places multi-line text in the drawing through a dialog box.

Lines

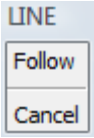
Command: LINE

A line consists of two points: a start point and an endpoint. Using the *Line* command you can draw a series of connected lines, but each line is considered a separate line entity.

1. Do one of the following
   - Click the *Line* tool button ( ) on the *Draw* toolbar.
   - Choose *Line* in the *Draw* menu.
   - Type *line* in the command bar, then press Enter.
   - Type *L* in the command bar, then press Enter.

You are prompted: ENTER to use last point/Follow/<Start of line>:

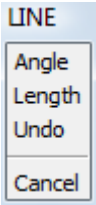
A prompt menu displays:



2. Specify the start point.
   The line displays dynamically.
3. Specify the end point.
   The line is drawn and the following line displays dynamically starting in the endpoint of the first line.

   - Using Ortho Mode or Polar Tracking you can type the length of the line in the command bar.
   - Using Snap Tracking you can position the end point with respect to the start point and one or two snap points on entities in the drawing.

The prompt menu changes:



4. (option) Repeat step 3 to draw a series of connected lines.

When the second line is drawn, the options in the prompt menu are:

After the third line, the options in the prompt menu are:

4. Right click to stop drawing lines.
5. (option) Right click to restart.

Rays

Command: RAY

A *Ray* is a line that starts at a point and extends to infinity. Because rays extend to infinity, they are not calculated as part of the drawing extents.

The default method for drawing a ray is to select the start point of the ray, and then specify its direction.

Rays and Infinite Lines or sometimes referred to as construction lines.

1. Do one of the following:

   - Click the *Ray* tool button ( ) on the *Lines* flyout of the *Draw* toolbar.
   - Choose *Ray* in the *Draw* menu.
   - Type *ray* in the command bar, then press Enter.

You are prompted: Infinite ray: Bisect/Horizontal/Vertical/Angle/Parallel/<Start of Ray>:

A prompt menu opens:

2. Specify the start point of the ray.
   The ray displays dynamically.
3. Specify the direction of the ray.
   The next ray displays dynamically.
4. (option) Keep specifying directions to draw a series of rays, starting from the same point.
5. Right click to stop drawing rays.

Infinite Lines

Command: XLINE

The *Xline* command creates infinite lines.

An *Infinite Line* is a line through a given point, oriented at a specified angle and extending to infinity in both directions. Because infinite lines extend to infinity, they are not calculated as part of the drawing extents. The default method for drawing an infinite line is to select a point and then specify its direction.

Rays and Infinite Lines or sometimes referred to as construction lines.

1. Do one of the following:

   - Click the *Infinite Line* tool button ( ) on the *Lines* flyout of the *Draw* toolbar.
   - Choose *Infinite Line* in the *Draw* menu.
   - Type *xline* in the command bar, then press Enter.

You are prompted: Xline: Bisect/Horizontal/Vertical/Angle/Parallel/<Point along line>:

A prompt box displays:

2. Specify the start point of the infinite line.
   The infinite line is created.
   A new infinite line through the same start point displays dynamically.
3. (option) Specify the direction of the infinite line.
   The infinite line is created.
   A new infinite line through the same start point displays dynamically.
4. (option) Repeat step 3 to draw a series of infinite lines, through the same start point.
5. Right click to stop drawing infinite lines.
6. (option) Right click to restart.

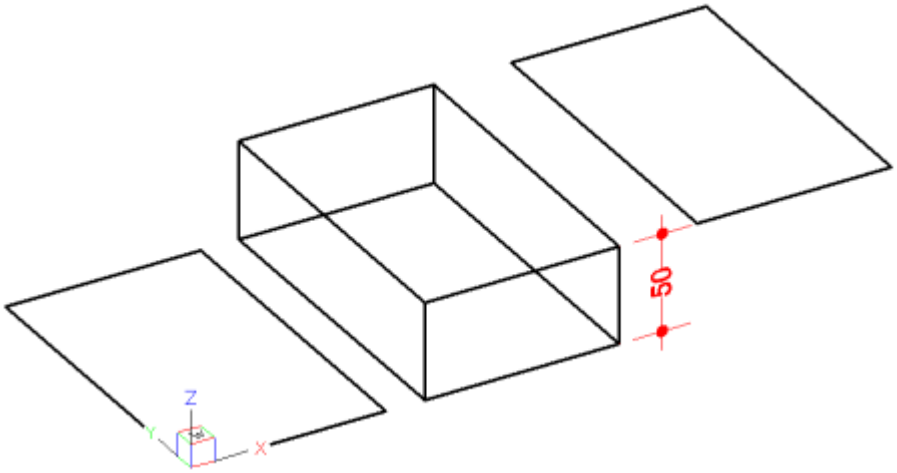
Polylines

A *polyline* is an open or closed sequence of connected line and/or arc segments, which are treated as a single entity. Each segment of a polyline can have a width that is either constant or tapers over the length of the segment.

The *Pline* command interactively creates open and closed polylines entering points.

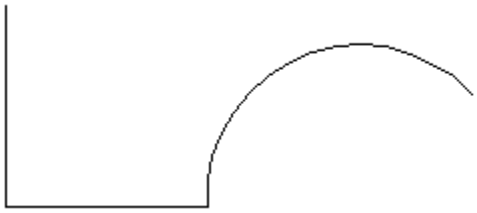
The *Boundary* and *- Boundary* commands create closed polylines from an enclosed area, defined by other entities.

The *THICKNESS* system variable specifies the height of a polyline. If *THICKNESS* = 0 (zero) there is no visual difference between polylines and a series of lines and/or arcs.

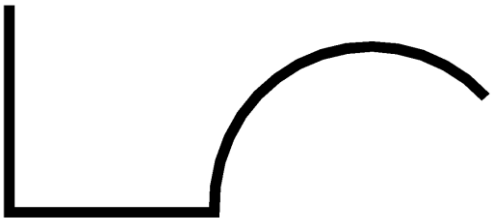


THICKNESS = 0 (left), THICKNESS = 50 (middle), ELEVATION = 50 (right)

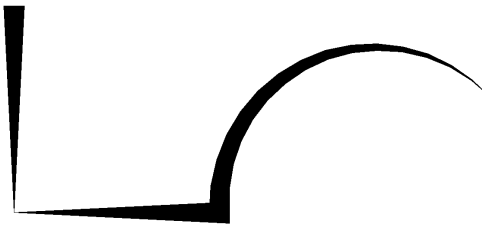
When a polyline is edited, you can modify the entire polyline or change individual segments and you can add or remove vertices.



zero width polyline



constant width polyline



tapered width polyline

NOTE When Fill Mode is turned OFF, all filled entities, such as wide polylines and planes, display and print as outlines.

Depending on the value of the GRIPS system variable have grips on their endpoints and vertices and/or at the midpoint of each segment.

The value of GRIPS can be 0, 1 or 2

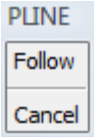
Value	Grips display
0	No grips
1	Endpoint grips only
2	Endpoint and midpoints grips

Drawing Polylines

1. Do one of the following
   - Click the *Polyline* tool button ( ) on the *Draw* toolbar.
   - Choose *Polyline* in the *Draw* menu.
   - Type *pline* in the command bar, then press Enter.
   - Type *PL* in the command bar, then press Enter.

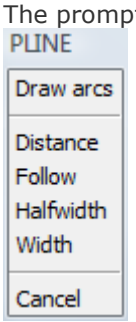
You are prompted: ENTER to use last point/Follow/<Start of polyline>:

A prompt menu displays:

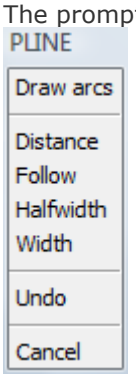


2. Specify the start point of the polyline.

The prompt menu changes:



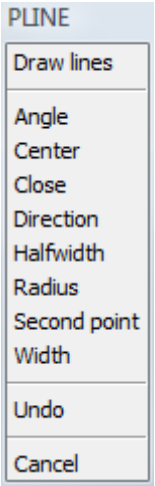
3. Specify the second point of the polyline.
   The prompt menu changes:



4. (option) Repeat step 3 to add more straight segments.
5. (option) Do one of the following to start drawing arc segments:

   - Type *A*, then press Enter.
   - Choose *Draw arcs* in the prompt menu.

The prompt menu changes:



An arc segment displays dynamically. The arc is tangent to the previous line segment.

6. Specify the endpoint of the arc.
7. (option) Repeat step 6 to add more arc segments.
8. (option) Do one of the following to start drawing line segments:

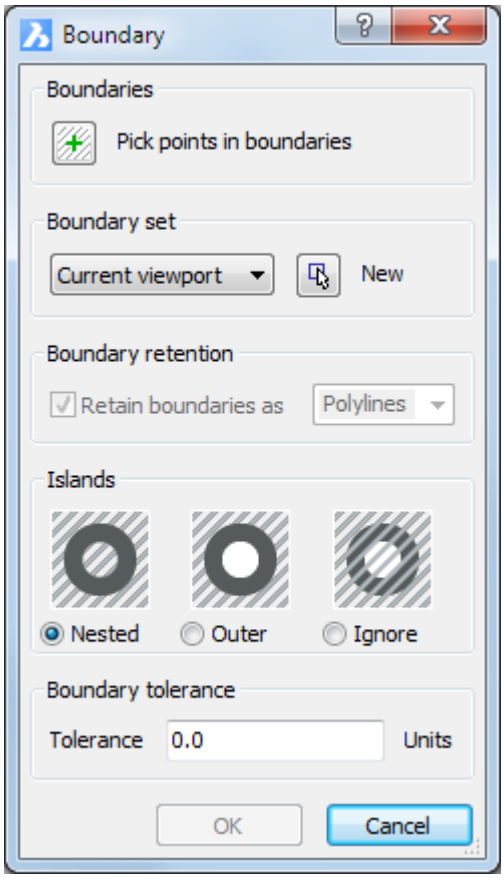
   - Type *L*, then press Enter.
   - Choose *Draw lines* in the prompt menu.

9. Specify the endpoint of the line segment.
10. Right click or choose *Done* in the prompt menu to stop.
11. (option) Right click to restart.

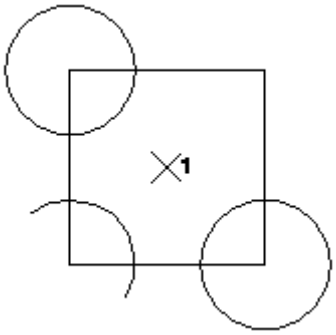
Boundary Polylines

1. Do one of the following
   - Click the *Boundary Polyline* tool button ( ) on the *Draw* toolbar.
   - Choose *Boundary Polyline* in the *Draw* menu.
   - Type *boundary* in the command bar, then press Enter.

The *Boundary* dialog box opens.

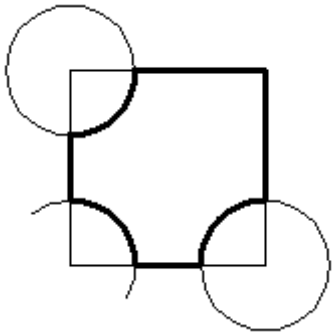


2. (option) Click the *Select* button ( )
   The *Boundary* dialog closes.
   Select the boundary entities in the drawing, then Press Enter.
   The *Boundary* dialog opens.
3. Click the *Pick points in boundaries* button on the *Boundary* dialog box.
   The dialog box closes.
4. Click in the area where you want to create the boundary polyline: point 1 in the image below.



5. The boundary polyline displays in dashed lines.
6. (option) Keep picking points to create more boundary polylines.
7. Right click to accept the polyline(s).
   The *Boundary* dialog box opens again.

8. Click the *OK* button on the *Boundary* dialog box.
   The polyline(s) is (are) created.



NOTES

- If you select the boundary entities before launching the *Boundary Polyline* tool, you can make the selection set active by pressing the *Select* button ( ) in step 2.
- If you type *-boundary* (don't forget the "minus" sign) in the command bar, you are prompted to click the area where you want to create the polyline(s) immediately (no dialog opens) then right click to create the polyline(s).

Splines

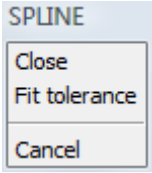
Command: SPLINE

A spline is an open or closed smooth curve defined by a set of points. You can use splines to draw curved shapes, which cannot be drawn as a polyline.

1. Do one of the following
   - Click the *Spline* tool button ( ) on the *Draw* toolbar.
   - Choose *Spline* in the *Draw* menu.
   - Type *spline* in the command bar, then press Enter.

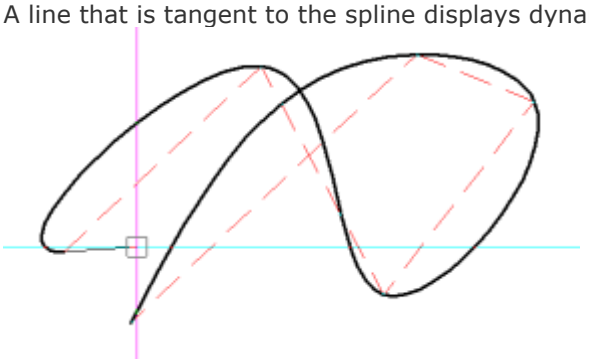
You are prompted: First point of spline:

2. Specify the first point of the spline.
3. Define the second point of the spline.
   You are prompted: Close/Fit Tolerance/<next point>:
   A prompt menu displays.

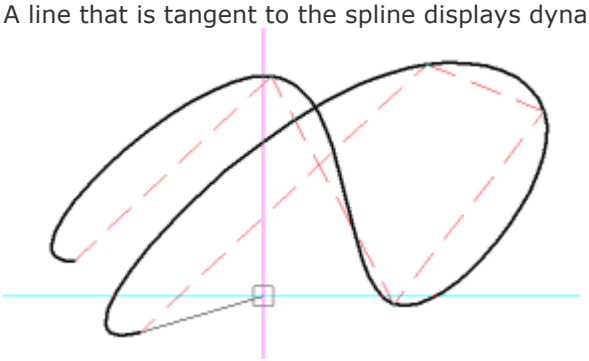


4. (option) Repeat step 3 to define more points.
5. Right click to stop adding more points.
   You are prompted: Select starting tangent point:

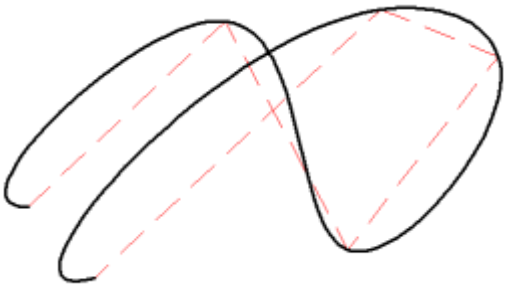
A line that is tangent to the spline displays dynamically from the start point of the spline.



6. Click to define the curve of the spline.
   You are prompted: Select ending tangent point:
   A line that is tangent to the spline displays dynamically from the start end of the spline.



7. Click to conclude the spline command.



8. (option) Right click to restart.

NOTES

- The red dashed lines in the above images represent the polyline through the control points of the spline.
- The *SPLFRAME* (Spline Frame) settings variable defines whether the control polygon displays for splines and spline-fit polylines.

Sketch

Command: SKETCH

A freehand sketch consists of a series of straight line segments, created either as individual line entities or as a polyline. Before you begin creating a freehand sketch, you must set the length, or increment, of each segment. The smaller the segments, the more accurate your sketch, but small segments increase the file size.

After you specify the length of the sketch segments, the crosshairs change to a pencil tool ( ).
A freehand sketch line is not added to the drawing until you "write" the sketch into your drawing.

Setting the Sketch Poly (SKPOLY) system variable

Whether a chain of individual lines or a polyline is created by the *Sketch* tool is controlled through the *Sketch Poly* (SKPOLY) system variable. When the setting is *ON* the *Sketch* tool creates polylines

Do one of the following:

- In the command bar type *skpoly*, then press Enter.
- In the Settings dialog, go to *Drawing > Drafting > Entity Creation > Freehand Sketches*.

Creating a freehand sketch

1. Do one of the following
   - Click the *Sketch* tool button ( ) on the *Draw* toolbar.
   - Choose *Sketch* in the *Draw* menu.
   - Type *sketch* in the command bar, then press Enter.

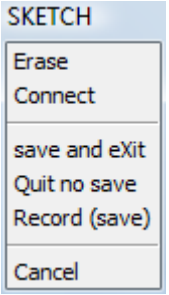
You are prompted: Length of segments <current length>:

2. Do one of the following to define the length of the segments::

   - Press Enter to accept the current length.
   - Type a new value in the command bar.
   - Click two points to define the length graphically.

The length of the segments is saved to the *Sketch Increment* (SKETCHINC) system variable.

3. Click to start sketching (pen down). The crosshairs are replaced by the *Pencil* tool ( )
You are prompted: Press ENTER or click to toggle Pen/Quit without save/Record (save)/Erase/Connect/Straight/eXit with save::
A prompt menu displays:



4. Move the cursor to sketch.
   The sketched line displays in light green.
5. (option) Click to stop sketching (pen up), then click again (pen down) to resume.
6. (option) Choose *Erase* on in the prompt menu or type *E* + Enter in the to start erasing sketch lines.

   The cursor changes to the *Erase* tool ( ).

   - Move the cursor to the end of a sketch line to start erasing.
   - Click to stop erasing and resume sketching.

7. (option) Choose *Record (save)* in the prompt menu or type *R* + Enter to save the sketch lines, and keep sketching.
8. (option) Choose *save and eXit* in the prompt menu or type *X* + Enter to save the sketch lines and exit the sketch command.
9. (option) Choose *Quit no save* in the prompt menu or type *Q* + Enter to exit the sketch command without saving.

Circles

The default method for drawing a circle is to specify a center point and radius.

Other methods to draw circles can be found in the *Circle* submenu of the *Draw* menu and in the *Circles* toolbar.

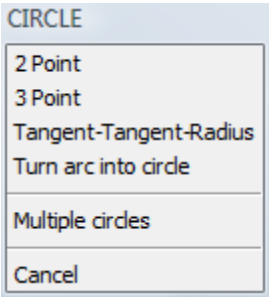
The *Ai_CircTan* command draws circles tangent to three entities.

General Procedure to Draw a Circle

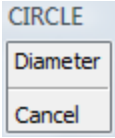
1. Do one of the following.
   - Click the *Circle* tool button ( ) on the *Draw* toolbar.
   - Type *circle* in the command bar, then press Enter.
   - Type *C* in the command bar, then press Enter.

You are prompted: 2Point/3Point/RadTanTan/Arc/Multiple/<Center of circle>:

A prompt menu displays:



2. Specify the center point of the circle.
   You are prompted: Diameter/<Radius> <current radius>:
   The prompt menu changes:



3. Do one of the following:

   - Press Enter to accept the current radius.
   - Type a new radius and press Enter.
   - Click to define the radius graphically.

Drawing a Circle Tangent to Three Entities

1. Type *ai_circtan* in the command bar.
   You are prompted: Enter Tangent spec:
2. Click a linear entity.
   Lines, arcs, circles, polyline segments, xlines and rays are accepted.
   You are prompted: Enter second Tangent spec:
3. Click the second linear entity.
   You are prompted: Enter third Tangent spec:
4. Click the third linear entity.
   A circle tangent to the three selected entities is created.

Editing a Circle

1. Select the circle.
   The properties of the circle display in the Properties bar.

Geometry	
Center	85.0057, 163.4489, 0
X	85.0057
Y	163.4489
Z	0
Radius	59.6358
Diameter	119.2716
Circumference	374.7029
Area	11172.8589
Normal	0, 0, 1
X	0
Y	0
Z	1

2. Type a new value in the *Center*, *Radius*, *Diameter*, *Circumference* or *Area* field, then press Enter.
3. Press the Esc key to stop editing the circle.

Arcs

The default method for drawing circular arcs is to specify three points: the start point, a second point and the endpoint.

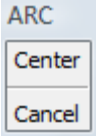
Other methods to draw arcs can be found in the *Arc* submenu of the *Draw* menu and in the *Arcs* toolbar.

General Procedure to Draw an Arc

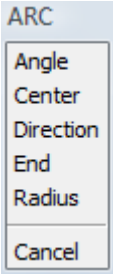
1. Do one of the following.
   - Click the *Arc* tool button ( ) on the *Draw* toolbar.
   - Type *arc* in the command bar, then press Enter.
   - Type *A* in the command bar, then press Enter.

You are prompted: Enter to use last point/Center/Follow/<Start of arc>:

A prompt menu displays:



3. Specify the start point of the arc.
   You are prompted: Angle/Center/Direction/End/Radius/<Second point>:
   The prompt menu changes:



4. Specify the second point of the arc.
   You are prompted: End point:
5. Specify the end point of the arc.

Editing an Arc

1. Select the arc.
   The properties of the arc display in the Properties bar.

Geometry	
Start point	38.5381, 191.1854, 0
X	38.5381
Y	191.1854
Z	0
Center	3.7399, 153.1902, 0
X	3.7399
Y	153.1902
Z	0
End point	-47.553, 158.0457, 0
X	-47.553
Y	158.0457
Z	0
Radius	51.5223
Start angle	48
End angle	175
Total angle	127
Arc length	114.2725
Area	1884.8676
Normal	0, 0, 1
X	0
Y	0
Z	1

2. Type a new value in the *Center*, *Radius*, *Start angle* or *End angle*, then press Enter.
3. Press the Esc key to stop editing the arc.

Ellipses

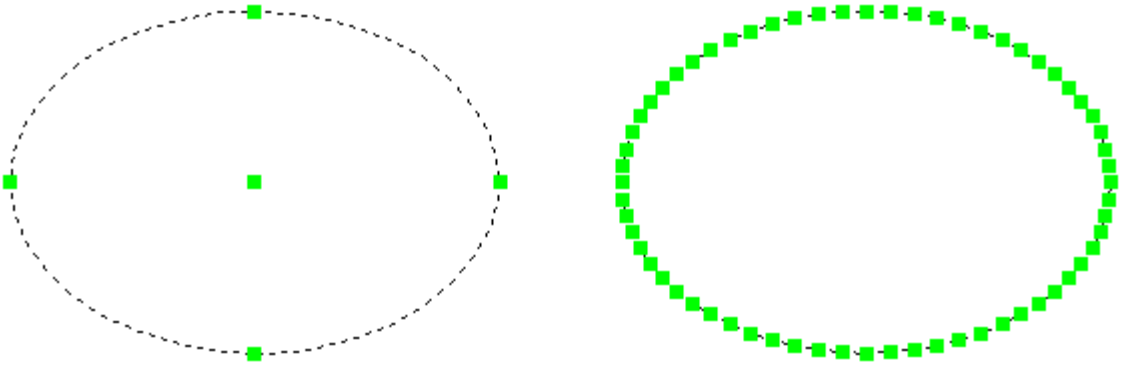
The default method for drawing an ellipse is to specify the endpoints of one axis of the ellipse and then specify a distance representing half the length of the second axis. The endpoints of the first axis determine the orientation of the ellipse. The longer axis of the ellipse is called the major axis, and the shorter one is the minor axis. The order in which you define the axes does not matter. The program determines the major and minor axes based on their relative lengths. Half the major axis is called the major radius, half the minor axis is the minor radius. The ratio (minor radius divided by the major radius) defines the eccentricity of the ellipse. A circle is an ellipse with an eccentricity of 1.

Other methods to draw ellipses can be found in the *Ellipse* submenu of the *Draw* menu and in the *Ellipses* toolbar.

The *Polyline Ellipse* (PELLIPSE) settings variable: controls whether real ellipses or a polyline representation is created.

If the variable is ON, the *Ellipse* tool creates a closed polyline, if the variable is OFF real ellipses are created.

The difference between a real ellipse and a polyline representation of an ellipse is visible when you select the ellipse. A real ellipse has five handles: center point and the endpoints of the axes. A polyline representation of an ellipse is a closed polyline composed of arc segments.



real ellipse (left) and the polyline representation of an ellipse (right)

NOTE Since ellipses cannot be used as a clip boundary for external references or to create clipped paperspace viewports, you must use a polyline representation to do these jobs.

Setting the Polyline Ellipse Variable

Do one of the following:

- In the command bar type *pellipse*, then press Enter.
- In the Settings dialog, go to *Drawing > Drafting > Entity Creation > Other entities*.

Other entities	
Circle radius	59.6358
Polyline ellipse	[0] Create real ellipses
Polygon sides	4
Shape name	

General Procedure to Draw an Ellipse

1. Do one of the following.
   - Click the *Ellipse* tool button ( ) on the *Draw* toolbar.
   - Type *ellipse* in the command bar, then press Enter.
   - Type *EL* in the command bar, then press Enter.

You are prompted: Arc/Center/<First end of ellipse axis>:

A prompt menu displays:

ELLIPSE
Arc
Center
Cancel

2. Specify the the first end of the ellipse axis.
   You are prompted: Second end of axis:
3. Specify the second end of the ellipse axis.
   You are prompted: Rotations/<Other axis>:

A prompt menu displays:

ELLIPSE
Rotation
Cancel

4. Specify the length of half the other axis of the ellipse.

NOTE If the *Polyline Ellipse* variable is off, the *Arc* option is not available.

Editing an Ellipse

1. Select the ellipse.
   The properties of the ellipse display in the Properties bar.

Geometry	
Start point	-55.838, 242.3358, 0
X	-55.838
Y	242.3358
Z	0
Center	-17.4752, 242.3358, 0
X	-17.4752
Y	242.3358
Z	0
End point	-55.838, 242.3358, 0
X	-55.838
Y	242.3358
Z	0
Major radius	38.3628
Minor radius	22.6197
Radius ratio	0.5896
Start angle	0
End angle	0
Major axis vector	-38.3628, 0, 0
X	-38.3628
Y	0
Z	0
Minor axis vector	0, -22.6197, 0
X	0
Y	-22.6197
Z	0
Area	2726.1383

2. Type a new value in the *Center*, *Major radius*, *Minor radius*, *Radius ratio*, *Start angle* or *End angle* field, then press Enter.
3. Press the Esc key to stop editing the ellipse.

Elliptical Arcs

An elliptical arc is a portion of an ellipse. The default method for drawing an elliptical arc is to specify the endpoints of one axis of the ellipse, and then specify a distance representing half the length of the second axis. Then you specify the start and end angles for the arc, measured from the center of the ellipse in relation to its major axis.

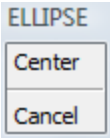
Other methods to draw elliptical arcs can be found in the *Ellipse* submenu of the *Draw* menu and in the *Elliptical Arcs* toolbar.

General Procedure to Draw an Elliptical Arc

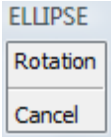
1. Do one of the following.
   - Click the *Elliptical Arc* tool button ( ) on the *Draw* toolbar.
   - Launch the Ellipse command using the *Arc* option.

You are prompted: Center/<First end of ellipse axis>:

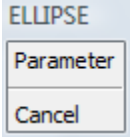
A prompt menu displays:

2. 

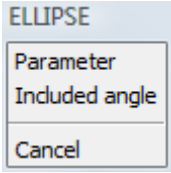
3. Specify the first end or the ellipse axis.
   You are prompted: Second end of axis:
4. Specify the second end of the ellipse axis.
   You are prompted: Rotation/<Other axis>
   A prompt menu displays:



5. Specify the length of half the other axis of the ellipse.
   You are prompted: Parameter/<Start angle of arc>:
   The prompt menu changes:



6. Specify the start angle of the elliptical arc.
   The elliptical arc is drawn dynamically when you move the crosshairs.
   You are prompted: Parameter/Included/<End angle>:
   The prompt menu changes:



7. Specify the end angle of the elliptical arc.

Editing an Elliptical Arc

1. Select the elliptical arc.
   The properties of the elliptical arc display in the Properties bar.

Geometry	
Start point	-10.743, 100.3847, 0
X	-10.743
Y	100.3847
Z	0
Center	-25.58, 70.3336, 0
X	-25.58
Y	70.3336
Z	0
End point	17.1995, 57.1941, 0
X	17.1995
Y	57.1941
Z	0
Major radius	54.7481
Minor radius	29.1773
Radius ratio	0.5329
Start angle	234
End angle	154
Major axis vector	-54.0321, -8.8252, 0
X	-54.0321
Y	-8.8252
Z	0
Minor axis vector	4.7033, -28.7958, 0
X	4.7033
Y	-28.7958
Z	0
Area	4196.6226

2. Type a new value in the *Center*, *Major radius*, *Minor radius*, *Radius ratio*, *Start angle* or *End angle* field, then press Enter.
3. Press the Esc key to stop editing the elliptical arc.

Rectangles

Command: RECTANG

A rectangle is created as a closed, four sided polyline.

A rectangle is drawn by specifying two opposite corners. The *Dimensions* option allows to specify the width and height of the rectangle.

Unless the *Rotated* option is chosen, the sides of a rectangle are always parallel to the x-axis and y-axis of the current coordinate system.

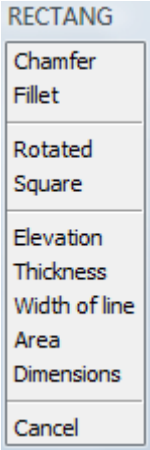
NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and solids, display and print as outlines.

General Procedure to draw a rectangle

1. Do one of the following
   - Click the *Rectangle* tool button ( ) on the *Draw* toolbar.
   - Choose *Rectangle* in the *Draw* menu.
   - Type *rectang* in the command bar, then press Enter.
   - Type *rec* in the command bar, then press Enter.

You are prompted:
Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/Area/Dimensions<select first corner of rectangle>:

A prompt menu opens:



2. Specify the first corner of the rectangle.
   The rectangle displays dynamically when you move the cursor.
3. Specify the opposite corner of the rectangle.
   The rectangle is created.
4. (option) Right click to draw more rectangles.

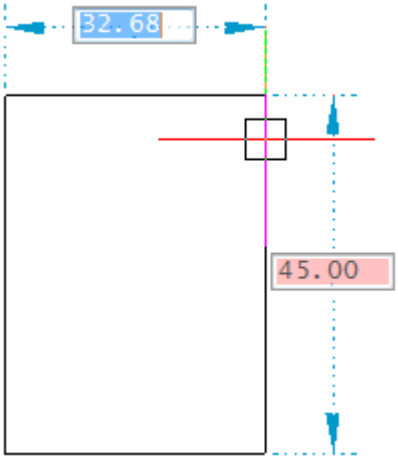
NOTE Because a rectangle is created as a closed polyline, rectangles have a direction. The direction of a closed polyline is positive if it is drawn counter-clockwise and negative if it is drawn in a clockwise direction. Rectangles have a counter-clockwise direction if both the X- and Y-coordinate of the second point are higher or lower than the coordinates of the first point (= movement of the cursor is positive or negative along both X- and Y-axis). Rectangles have a clock-wise direction if the X- or Y-coordinate is higher or lower than the X- or Y-coordinate of the first point (= movement of the cursor is positive along the X-axis and negative along the Y-axis or vice versa).

The *Reverse Direction* option of the *Edit Polyline* tool changes the direction of a polyline.

Using Dynamic Dimensions

Using Dynamic Dimensions the width and length of the rectangle can be controlled dynamically.

1. Launch the *Rectang* command.
   You are prompted:
   Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/Area/Dimensions<select first corner of rectangle>:
2. Specify the first corner of the rectangle.
   The rectangle displays dynamically when you move the cursor.
   The width and height display in the dynamic dimension fields.
3. Type a value in the *Height* field.
4. Press the TAB key.
   The *Height* field is locked and turns red.
5. Type a value the *Width* field.



6. Do one of the following

   - Press Enter to create the rectangle.
   - Press the TAB key until the desired field is active, then type a different value.

7. Press Enter to create the rectangle.

Polygons

Command: POLYGON

Polygons are closed polylines comprised of a minimum of three and a maximum of 1,024 equal-length sides.

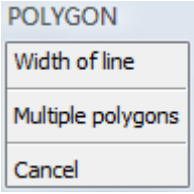
In BricsCAD you can draw polygons using the following methods:

- *Centre - Vertex* ( ): first define the center point, then the vertex (= radius of the circumscribed circle method).
- *Center - Side* ( ): first define the center point, then the midpoint of a side (= radius of the inscribed circle method).
- *Edge* ( ): define the length of the side of the polygon.

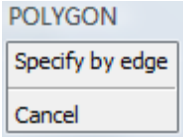
1. Do one of the following
   - Click the *Polygon, Center - Vertex* tool button ( ) on the *Polygons* flyout of *Draw* toolbar.
   - Choose *Polygon* in the *Draw* menu.
   - Type *polygon* in the command bar, then press Enter.

You are prompted: Polygon: Multiple/Width of line/<Number of sides> <4>:

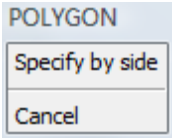
A prompt menu opens:



2. Specify the number of sides, then press Enter.
   You are prompted: Specify by: Edge/<Center of polygon>:
   The prompt menu changes:



3. Specify the center of the polygon.
   You are prompted: Specify by: Side/<Select vertex point>:
   the prompt menu changes:



4. Specify a vertex point of the polygon.
   The polygon is created.

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.

Traces

Command: TRACE

Traces are 4-sided filled entities, created with two parallel sides. When you create a series of traces, they are L-connected to each other.

1. Do one of the following
   - Click the *Trace* tool button (Z) on the *Polygons* flyout of the *Draw* toolbar.
   - Choose *Trace* in the *Draw* menu.
   - Type *trace* in the command bar, then press Enter.

You are prompted: Width of trace <current width>:

2. Do one of the following:

   - Right click to accept the current width.
   - Type a new current width in the command bar.
   - Define a new current width graphically by clicking two points.

You are prompted: Start of trace:

3. Specify the start point of the trace.

You are prompted: Next point:

4. Specify the end point of the first trace.
   A line indicates the direction of the first trace. The direction of the second trace displays dynamically.
5. Do one of the following:

- Right click to create the trace.
- Specify the endpoint of the next trace.

6. (option) Repeat step 5 to keep adding traces.

NOTE When Fill Mode is turned off, all filled entities, such as traces, display and print as outlines.

Donuts

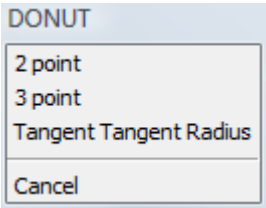
Command: DONUT

Donuts are solid, filled circles or rings created as closed, 2-segment, wide polylines. The default method to draw donuts is to specify its inside and outside diameters, and then specify its center. If the inside diameter is zero, a filled circle is created.

1. Do one of the following:
   - Click the *Donut* tool button ( ) *Draw* toolbar.
   - Choose *Donut* in the *Draw* menu.
   - Type *donut* in the command bar, then press Enter.

You are prompted: 2Point/3Point/RadTanTan/<Inside diameter of donut> <current inside diameter>:

A prompt menu opens:



2. Do one of the following:

   - Right click to accept the current inside diameter.
   - Type a new inside diameter in the command bar.
   - Define a new inside diameter graphically by clicking two points.

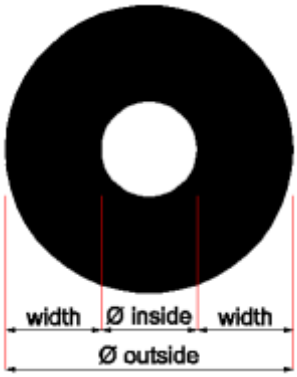
You are prompted: Outside diameter of donut <current outside diameter>:

3. Do one of the following:

   - Right click to accept the current outside diameter.
   - Type a new outside diameter in the command bar.
   - Define a new outside diameter graphically by clicking two points.

You are prompted: Center of donut:

4. Specify the center of the donut.
   The donut is created.
5. (option) Specify the center point of another donut.
6. Right click to conclude the *donut* command.



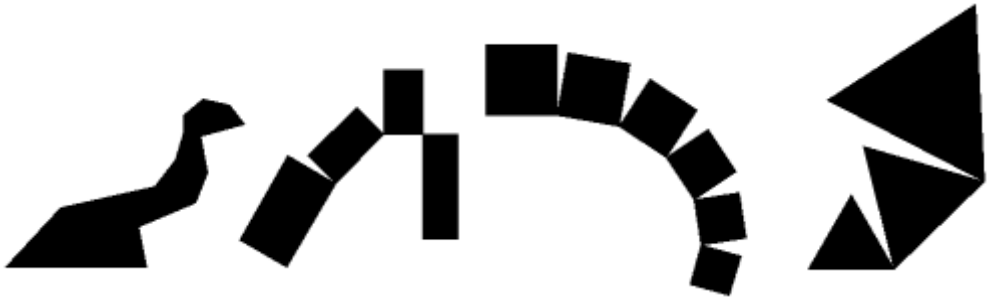
NOTE

- When Fill Mode is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.
- Set the inside diameter to 0 (zero) to create filled circles.

Solids

Command: SOLID

2D Solids are rectangular, square, triangular or quadrilateral areas filled with a solid color.



Left to right: quadrilateral, rectangular, square and triangular solids

1. Do one of the following:
   - Click the *Solid* tool button ( ) on the *Polygons* flyout of *Draw* toolbar.
   - Choose *Solid* in the *Draw* menu.
   - Type *solid* in the command bar, then press Enter.

You are prompted: Rectangle/Square/Triangle/<First point of solid>:

A prompt menu displays:

2. Specify the first point of the solid.
3. Specify the second point of the solid.
4. Specify the third and fourth point of the solid.
   The solid is created.
5. (option) Repeat step 4 to add more solids.
   A series of connected solids is created.
6. Right click to conclude.

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and solids, display and print as outlines.

Revision Clouds

Command: REVCLOUD

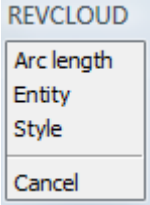
A *Revision Cloud* is an open or closed polyline composed of arcs. Revisions clouds are used to for reviewing or redlining purposes to indicate the parts of the drawing that need to be adjusted or annotated.

Before you start drawing revision clouds, first define the *Arc length* and *Style*.

1. Do one of the following:
   - Click the *Revision Cloud* tool button ( ) on the *Draw* toolbar.
   - Choose *Revision Cloud* in the *Draw* menu.
   - Type *revision cloud* in the command bar, then press Enter.

You are prompted: Specify start point or [Arc length/Object/Style] <Object>:

A prompt menu opens:



2. (option) To define the *Arc length*, do one of the following:

   - Choose *Arc length* in the prompt menu.
   - Type *A* in the command bar, then press Enter.

You are prompted: Specify minimum length of arc <current minimum length>:

Do one of the following:

- Press Enter or right click to accept the current minimum length.
- Type a new value in the command bar, then press Enter.
- Define a new minimum length graphically by specifying two points.

You are prompted: Specify maximum length of arc <current maximum length>:

Do one of the following:

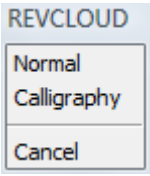
- Press Enter or right click to maximum length equal to the minimum length.
- Type a new value in the command bar, then press Enter.
- Define a new maximum length graphically by specifying two points.

3. (option) To define the revision cloud *Style*, do one of the following:

- Choose *Style* in the prompt menu.
- Type *S* in the command bar, then press Enter.

You are prompted: Select arc style [Normal/Calligraphy] <current style>:

A prompt menu opens:



Do one of the following:

- Press Enter or right click to accept the current style
- Select a new style in the prompt menu.
- Type *N* or *C* in the command bar, then press Enter.

4. (option) To convert an existing polyline, arc or circle to a revision cloud, do one of the following:

- Press Enter, then select the entity you want to convert.
- Choose Object in the prompt box, then select the entity you want to convert.

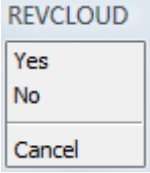
5. (option) To draw the revision cloud, click where you want to start and move the cursor. Each time the cursor movement exceeds the minimum length, an arc is added.

To stop do one of the following:

- Move the cursor over the start point to close the revision cloud.
- Right click.

You are prompted: Reverse direction [Yes/No] <No>:

A prompt menu opens:



6. To create the revision cloud, do one of the following:

- To accept the revision cloud, right click.
- To reverse the direction, type *Y* or choose *Yes* in the prompt menu.
- To leave the command without creating the revision cloud, choose *Cancel* in the prompt menu or press the Esc key.

Revision Clouds: 1. Normal 2. Calligraphy 3. Reversed calligraphy (based on a rectangle)

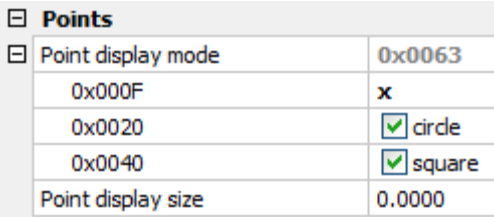
Points

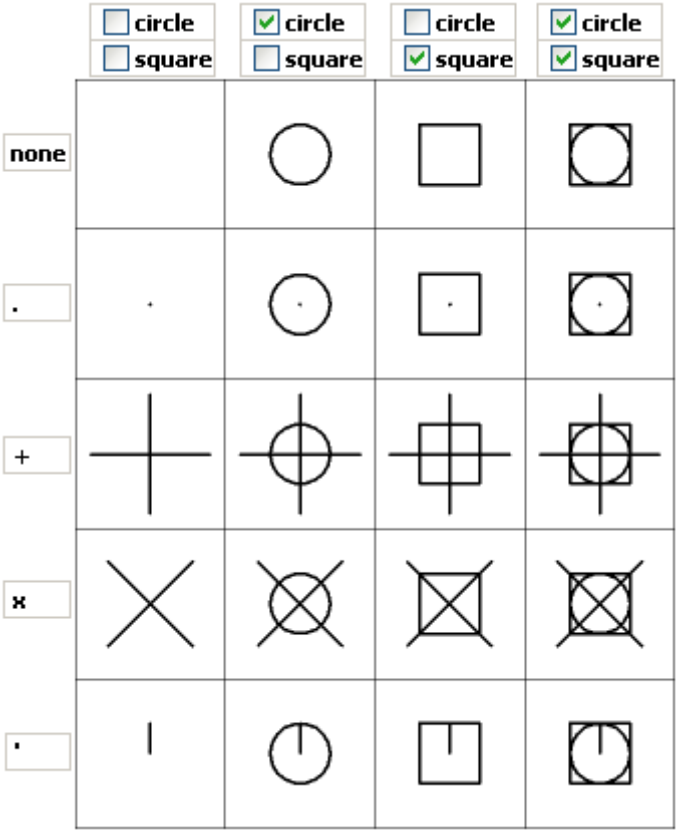
Since points are dimensionless, a single pixel should be the (nearly) correct representation on the screen. But this is hardly visible, especially in a complex drawing. Therefore you can choose between a number of possible display styles.

The point display style is chosen by means of the *Point Display Mode* (PDMODE) system variable. The size of the point representation style is controlled through the *Point Display Size* (PDSIZE) system variable.

Defining the Point Display Settings

1. Open the Settings dialog.
2. In the *Drawing* settings category, expand the *Entity Creation* sub-category.
3. Expand the *Points* settings group, then expand the *Point display mode* setting.





Point display modes

4. Set the *Point Display Mode* setting:

   - Choose the *point location marker* style: *point*, *none*, *X* or |
   - Choose to add a *circle* and/or a *square*.

5. Set the *Point Display Size* setting.

- Relative: Type 0 (zero) to scale the point display style at 5% of the screen.
- Absolute: Type the size of the point display style in drawing units.

6. Close the *Settings* dialog.

NOTE If you need to draw a 'dot', use a Donut with a zero inside diameter instead of a point entity.

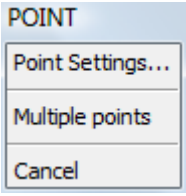
Drawing Points

Do one of the following

- Click the *Point* tool button ( ) on the *Draw* toolbar.
- Choose *Point* in the *Draw* menu.
- Type *point* in the command bar, then press Enter.

You are prompted: Settings/Multiple/<Location of point>:

A prompt menu displays:



Specify the location of the point.
The point is created.

NOTE If the *Point Display Size* setting is set to 0 (zero), use the Regen tool to resize the display of the points to 5% of the screen size after zooming in or out.

Wipeouts

A *wipeout* is a polygonal area that hides the underlying objects using the current background color. Most commonly wipeouts are used to mask part of a drawing for clarity. For example, you may want to add text to a complicated part of a drawing. A wipeout could be used then to mask an area behind the text to increase the readability of the text.

The *Frames* option of the *wipeout* command determines whether the edges of all wipeout entities in the current drawing are displayed or hidden.

To Create a Wipeout Using Points

1. Do one of the following:
   - Click the *Wipeout* tool button ( ) on the *Draw* toolbar.
   - Choose *Wipeout* in the *Draw* menu.
   - Type *wipeout* in the command bar, then press Enter.

You are prompted: Frames/Polylines/<Start point>:

2. Specify the first vertex of the wipeout.
   You are prompted: Next point.
3. Specify the second vertex of the wipeout.
   You are prompted: Undo/<Next Point>:
4. Do one of the following:

   - Specify the third vertex of the wipeout.
   - Type *U*, then press Enter or choose *Undo* in the context menu to delete the previous vertex.

You are prompted: Undo/Close/<Next Point>:

5. Do one of the following:

   - Specify the next vertex of the wipeout.
   - Type *U*, then press Enter or choose *Undo* in the context menu to delete the previous vertex.
   - Type *C*, then press Enter or choose *Close* in the context menu to create the wipeout.
   - Right click to create the wipeout.

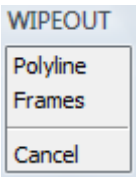
6. Repeat step 5 to create more vertices of the wipeout.

To Create a Wipeout from a Polyline

1. Do one of the following:
   - Click the *Wipeout* tool button ( ) on the *Draw* toolbar.
   - Choose *Wipeout* in the *Draw* menu.
   - Type *wipeout* in the command bar, then press Enter.

You are prompted: Frames/Polylines/<Start point>:

The *Wipeout* context menu displays:



2. Do one of the following:

   - Choose *Polyline* in the context menu.
   - Type *P*, then press Enter.

You are prompted: Select a closed polyline:

3. Click a closed polyline.
   Only polylines composed of line segments can be selected.
   You are prompted: Delete selected polyline? Yes/<No>:
4. Do one of the following:

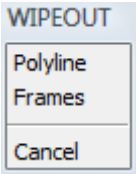
   - Type *Y*, then press Enter or choose *Yes* in the context menu to create the wipeout and delete the selected polyline.
   - Press Enter or choose *No* in the context menu, to create the wipeout and delete the selected polyline.

To Control the Display of the Edges of Wipeouts

1. Do one of the following:
   - Click the *Wipeout* tool button ( ) on the *Draw* toolbar.
   - Choose *Wipeout* in the *Draw* menu.
   - Type *wipeout* in the command bar, then press Enter.

You are prompted: Frames/Polylines/<Start point>:

The *Wipeout* context menu displays:



2. Do one of the following:

   - Choose *Frames* in the context menu.
   - Type *F*, then press Enter.

You are prompted: Enter display mode: OFF/<ON>:

3. Do one of the following:

   - Choose *OFF* in the context menu or type *off*, then press Enter, to hide the edges of all wipeouts in the drawing.
   - Choose *ON* in the context menu or type *on*, then press Enter, to display the edges of all wipeouts in the drawing.

Direct Modeling

BricsCAD provides advanced easy-to-use functions for working with ACIS solid and surface geometry, including:

- The creation of solid and surface geometry using Extrude (dmExtrude) and Revolve (dmRevolve) commands applied to 2D contours, 3D curves and edges of existing solids.
- The creation of fillets (dmFillet) and chamfers (dmChamfer) on existing solid and surface geometry.
- Modifying (dmPushPull) or deletion (Erase) of fillets and chamfers.
- Direct interactive manipulation of solid and surface faces using PushPull (dmPushPull), Move (dmMove) and Rotate (dmRotate) operations.
- The creation or removing of a volume when 2D contour attached to a face of a solid is being pushed or pulled.
- The removing of holes in surfaces by deleting its edges (Erase).
- Deletion (Erase) and copying (dmCopyFaces) of protrusion or depression features, such as holes, pockets, extrusions, ribs, and their combinations within the same 3D solid or from one 3D solid to another.

All these operations are performed in interactive mode: the intermediate result of the operation is shown dynamically, according to user input defined by mouse movement.

Depending on the value of the DELOBJ (Delete Entity) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities (see table below)

Value	Result
0	Retain defining entities
1	Delete profile entities; retain paths and guide curves
2	Delete all defining entities
-1	Prompt to delete profile entities; retain paths and guide curves
-2	Prompt to delete all defining entities

The key feature of direct editing capabilities provided by BricsCAD consists in preserving of design intent. Design intent can be expressed by 3D dimensional and geometrical constraints which can be explicitly specified by user (see 3D Constraints. 3D dimensional constraints (Distance, Radius and Angle) imposed for faces and edges give user full parametric control over the solid model, different configurations and scales of the model can be obtained simply by changing the parameter of a dimensional constraint. Geometrical constraints (Coincident, Tangent, Parallel and others) define how these changes influence the geometry of the model. However, there is no need to introduce all the geometrical constraints manually since most of them can be recognized automatically by BricsCAD (see Design Intent) to provide the expected result.

Another feature of Direct Editing operations is that they can be easily applied with the Quad cursor menu. This tool provides a set of operations for the currently selected entities. At the same time the Quad provides the other basic solid editing operations such as copy, boolean union and subtract.

Direct modeling operations apply to all ACIS solid geometry, whether created in BricsCAD or imported, and can be combined with regular solid modeling operations, for example SOLIDEDIT.

3D Constraints

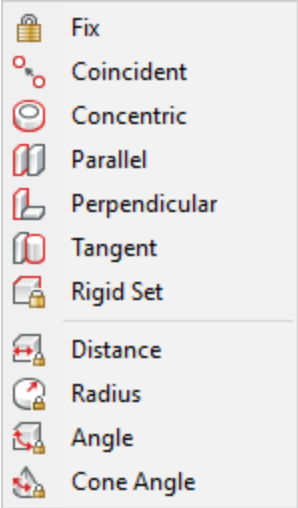
Commands: BMBROWSER, DMUPDATE, DMFIX3D, DMCOINCIDENT3D, DMCONCENTRIC3D, DMPARALLEL3D, DMPERPENDICULAR3D, DMTANGENT3D, DMDISTANCE3D, DMRADIUS3D, DMANGLE3D

Variational Direct Modeling

BricsCAD offers powerful tools for parametric direct solid and surface modeling using geometric constraints solving (aka variational direct modeling). Solid and surface models can be modified by defining constraints between their elements (edges and faces), as soon as constraint is added BricsCAD automatically modifies the model accordingly, and these constraints are kept satisfied upon following modifications of the model. The key advantage of variational direct modeling is that all constraints are taken into account simultaneously and the model behavior does not depend on the constraints creation order. It allows you to parameterize any feature of the 3D model and to do not care about the model creation history.

Dimensional constraints allow controlling the dimensions of the model, when such constraint is introduced or the value of its parameter is changed BricsCAD automatically updates the solid and surface geometry to satisfy the constraint. For example, the dimensions of a box can be controlled by the parameters of three distance constraints applied to its opposite faces.

Constraints can be created using the *3D Constraints* toolbar, the *Parametric/3D Constraints* menu or the *Constraints* command group in the Quad.



BricsCAD provides the following 3D constraints:

- Geometrical constraints: *Fix* ( ), *Coincident* ( ), *Concentric* ( ), *Parallel* ( ), *Perpendicular* ( ), *Tangent* ( ) and *Rigid Set* ( ).
- Dimensional 3D constraints: *Distance* ( ) , *Radius* ( ), *Angle* ( ), Cone Angle ( ).

Mechanical Browser

The *Mechanical Browser* ( ) allows to navigate through all the constraints in the model and to edit dimensional constraints.

See the BmBrowser command to learn more about the *Mechanical Browser*.

3D Constraints types

The following entities are supported for constraints creation:

- Faces of ACIS solids and surfaces of planar, cylindrical, spherical, toroidal and conical geometry.
- Linear and circular edges of ACIS solids and surfaces.
- Lines, rayes, xlines, circles and arcs.

3D Constraints and Direct Modeling

3D Constraints are taken into account when direct modeling operations are applied. If there is a 3D constraint which fixes the placement of faces or edges this constraint will be preserved during Push/Pull, Move and Rotate operations. The dynamically presented result of the operation respects existing constraints as well.

If some faces or edges disappear in a direct modeling or in a Boolean solid editing operation 3D constraints applied to such entities are removed. However, if there is an exact correspondence

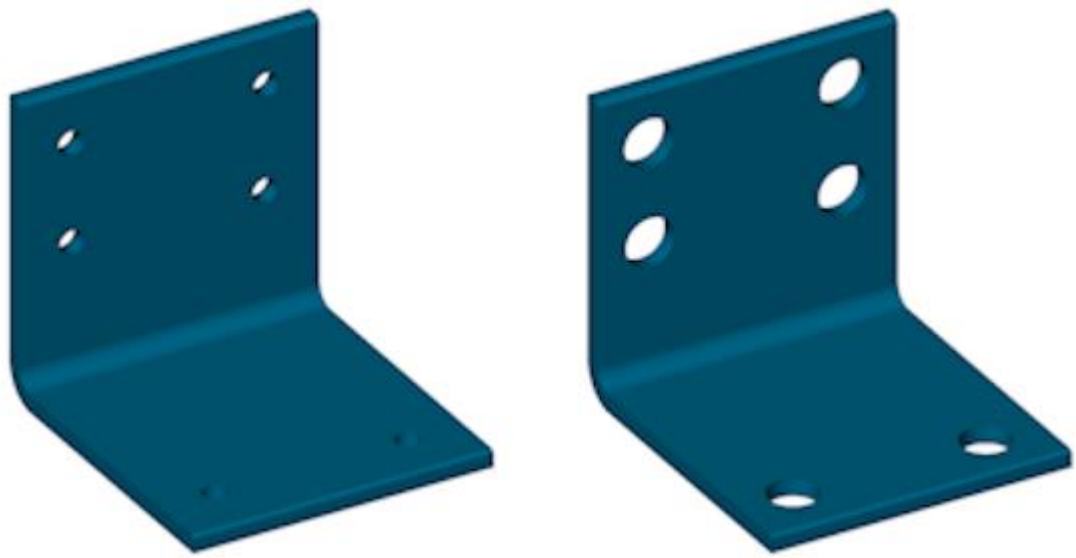
between the initial faces and the faces obtained in a result of a Boolean operation 3D constraints are automatically applied to new entities.

If 3D constraints prevent modification of the model with a particular direct modeling operation (e.g. a Push/Pull operation applied to a face with a *Fix* constraint) the status line of the 3D Constraints Bar receives a *No Solution* status.

Depending on the value of the DMRECOGNIZE system variable, BricsCAD automatically recognizes geometrical relations between surfaces of a solid and preserves them during direct modeling operations (see Design Intent Recognition).

Design Intent

Direct modeling operations allow to change geometry of 3D solid models easily: when a single face of a solid is modified with a particular operation other faces are modified accordingly in order to keep design intent of the model and its topological and geometrical validity. For example, consider a model with a number of holes of the same radius. When the radius of a cylindrical face is modified with a push/pull operation or by changing the value of a radial constraint other faces that share the same diameter are recognized by BricsCAD and modified automatically to keep the radii synchronized.



Apart from the recognition of equal radii, BricsCAD automatically recognizes other geometrical relations between surfaces of a solid and preserves them during direct modeling operations.

To define the design intent recognition, click the buttons on the *Design Intent* toolbar. A pressed button indicates the feature is selected. An unpressed button indicates the feature is not active.

- tangent surfaces (planes, cylinders and cones) ( )
- coincident planes ( )
- parallel planes ( )
- perpendicular planes ( )
- cylinders perpendicular to planes ( )
- coaxial surfaces (cylinders and cones) ( )
- surfaces of equal radius (cylinders and spheres) ( )
- vertices between 4 or more faces ( )
- edges between coincident faces ( )
- switch off/on all ( )

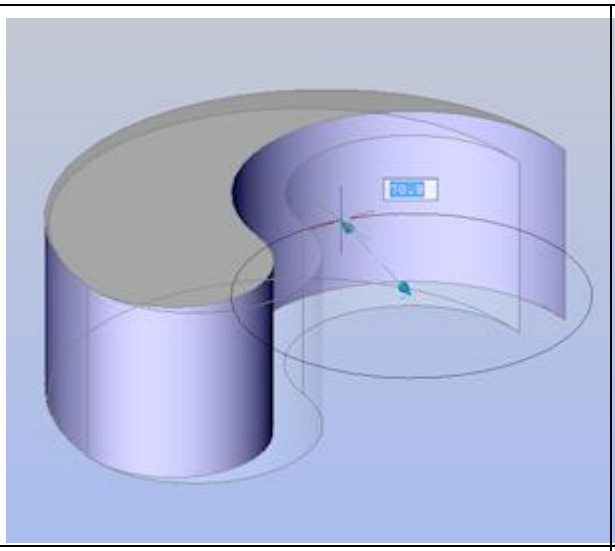
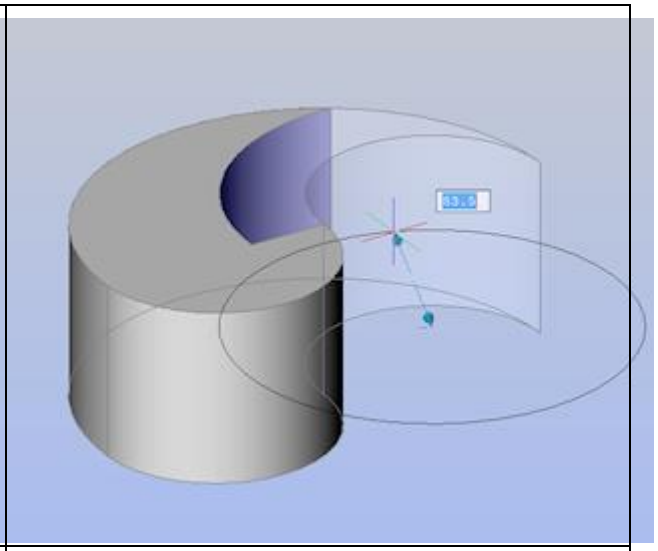
The design intent options can also be set by editing the DMRECOGNIZE system variable in the Settings dialog.

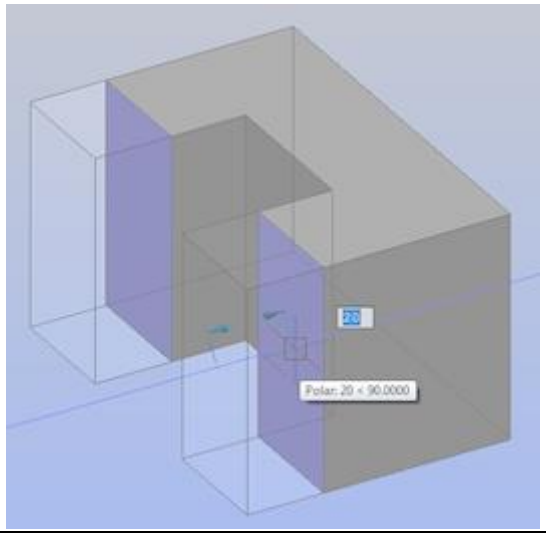
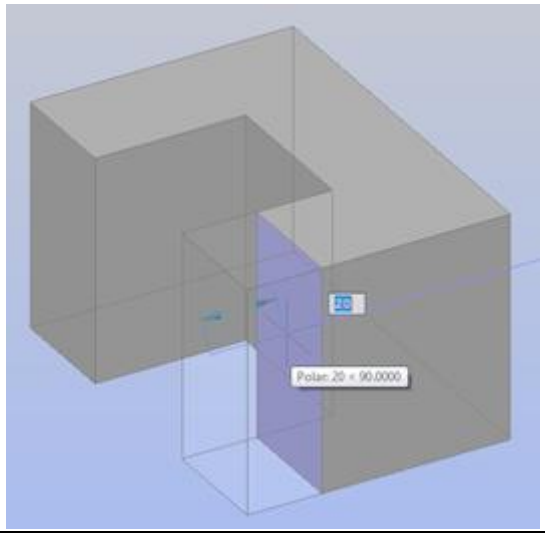
Click the *Options* tool button ( ) on the *3D Constraints* toolbar to set the *Automatic 3D geometry constraints recognition*. The state of tool buttons on the *Design Intent* toolbar (pressed or unpressed) is adjusted accordingly.

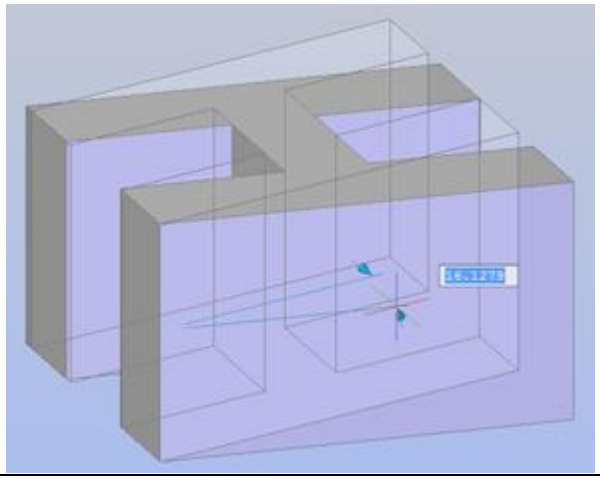
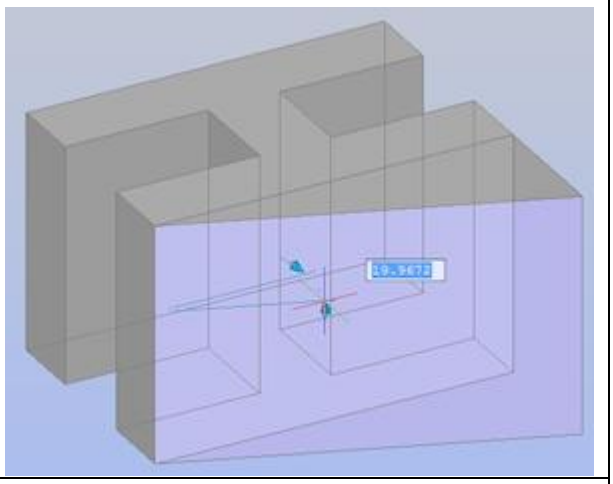
Automatic 3D geometry constraints recognition	0x0061 (97)
negative	☐ Switch off automatic 3D geometry constraints recognition
1	☑ Tangent surfaces
2	☐ Coincident planes
4	☐ Parallel planes
8	☐ Perpendicular planes
16	☐ Cylinders perpendicular to planes
32	☑ Coaxial surfaces
64	☑ Cylinders and spheres of equal radius
128	☐ Vertices between 4 or more faces
256	☐ Edges between coincident faces

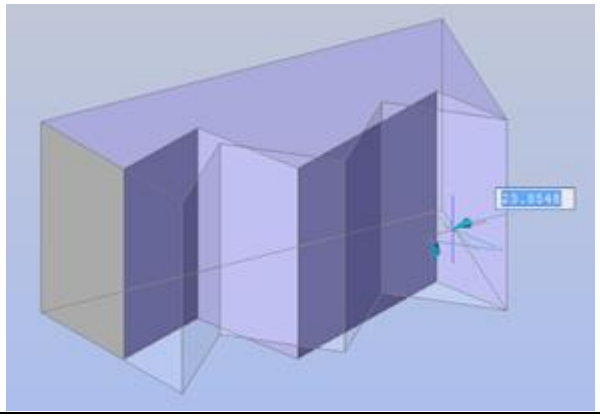
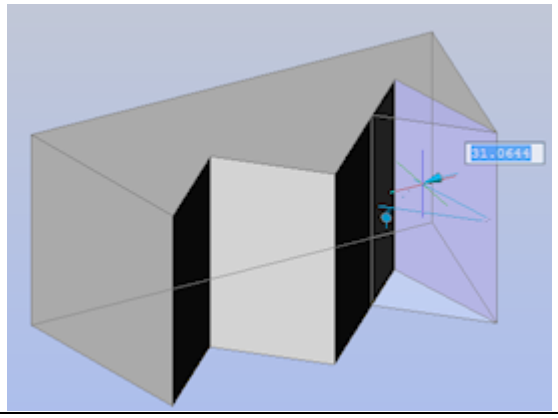
NOTE 3D Constraints which are explicitly defined by the user are taken into account when solid model is being modified with direct modeling commands. These constraints have a higher priority than automatically recognized ones and allow the user to better control the behavior of solid model modifications.

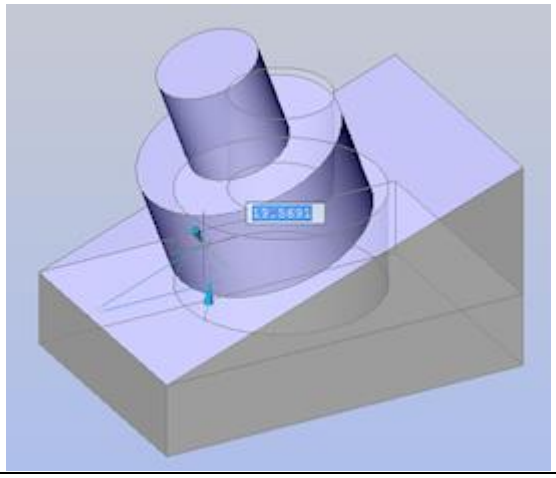
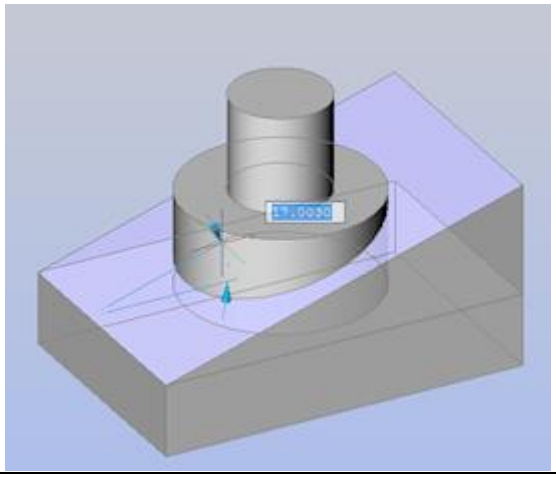
The following examples show the difference in the behavior whether automatic 3D geometry constraints are recognized or not.

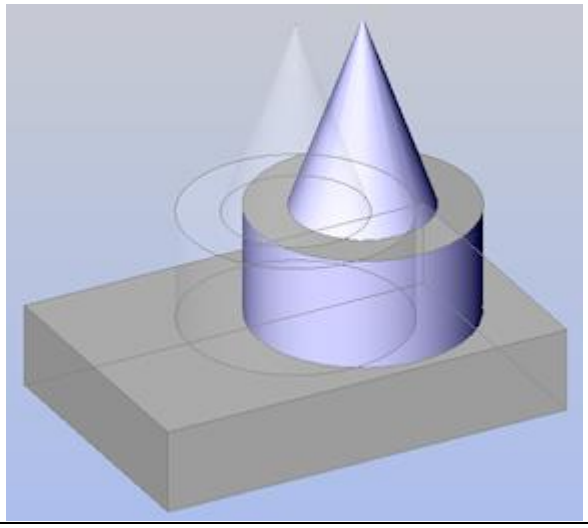
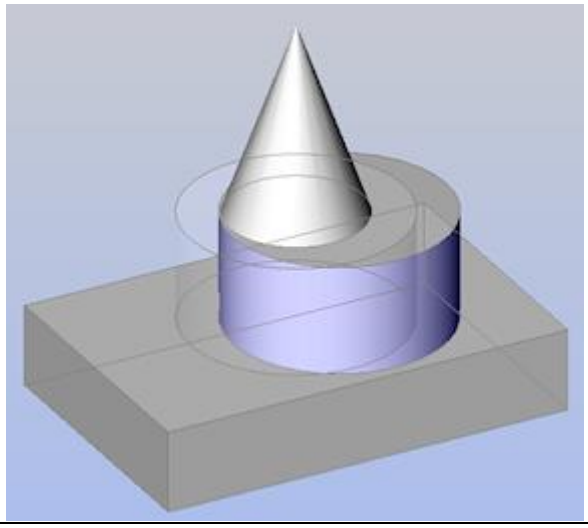
	
tangent surfaces recognized ( )	*tangent surfaces not recognized ( )*

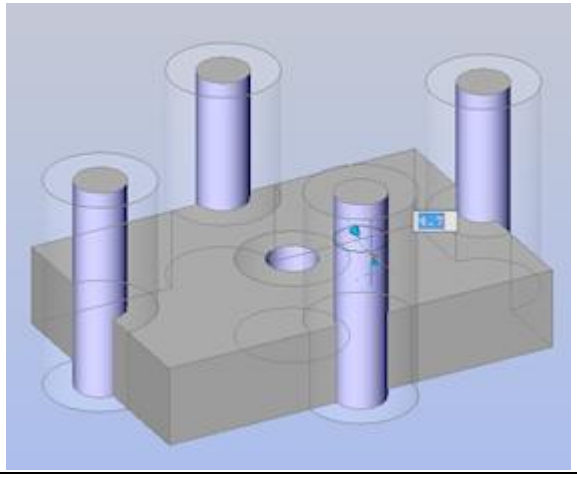
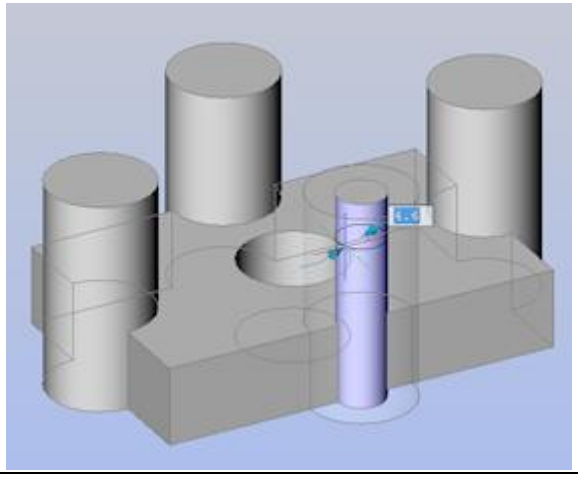
	
coincident planes recognized ( )	*coincident planes not recognized ( )*

	
parallel planes recognized ( )	*parallel planes not recognized ( )*

	
perpendicular planes recognized ( )	*perpendicular planes not recognized ( )*

	
perpendicular cylinders recognized ( )	*perpendicular cylinders not recognized ( )*

	
coaxial surfaces recognized ( )	*coaxial surfaces not recognized ( )*

	
equal radius recognized ( )	*equal radius not recognized ( )*

Direct Modeling Operations

This section describes direct modeling operations offered by BricsCAD. These operations are available from the Quad tool.

Alternatively you can use *Direct Modeling* Menu or *Direct Modeling* and *Deform* toolbars:

The *Align UCS* tool ( ) launches the UCS command with the *Face* option selected.

If the UCSDETECT system variable is *ON*, the UCS is aligned automatically to the currently highlighted face. The UCS-icon changes accordingly. Press the TAB key to select an obscured face. Click the *DUCS* field in the Status Bar to toggle the dynamic UCS feature.

The following direct modeling operations are available in BricsCAD:

Icon	Command	Applies to	Description
	dmPushPull	Any face of a 3D solid or surface	Adds volume to, or removes volume from the solid by cursor movement or direct distance input.
	dmMove	Cylindrical, spherical, conical and toroidal faces of a 3D solid or surface. Linear and circular edges of a 3D solid or surface. Whole 3D solids, surfaces and other entities.	Moves the selected geometry using a vector in the XY- plane of the current UCS. When a face or an edge of a solid or surface is moved, adjacent faces and edges are adjusted to preserve the correct solid/surface topology.
	dmRotate	Planar, cylindrical, spherical, conical and toroidal faces of a 3D solid or surface Whole 3D solids, surfaces and other entities.	Rotates the selected geometry around an axis. When a face of a solid or surface is rotated, adjacent faces and edges are adjusted to preserve the correct solid/surface topology.
	dmSimplify	3D solids, surfaces and their faces	Simplifies geometry and topology of selected entities or their subentities.
	dmStitch	Surfaces	Stitches disjoint surfaces into a single 3D solid or a single surface.
	dmAudit	3D solids and surfaces	Checks and fixes geometrical and topological errors in ACIS solids and surfaces.
	UCS	Planar faces of 3D solids and surfaces	Aligns the UCS to the selected face.
	dmExtrude	Closed 2D entities and detected closed	Creates solids by extruding the selected 2D entities or boundary.

		boundaries*	
		Planar faces of 3D solids and surfaces.	Adds volume to, or removes volume from the solid. Optionally the added volume is created as a separate solid
		Closed 2D entities and detected closed boundaries on a planar face of a solid	Creates solids or surfaces by extruding the selected 2D entities or boundary. Optionally the created volume is unified with or subtracted from the solid.
	dmRevolve	Closed 2D entities and detected closed boundaries*	Creates a solid or a surface by revolving a 2D entity about an axis.
		Planar faces of solids and surfaces	Adds volume to, or removes volume from the solid. Optionally the added volume is created as a separate solid.
		Closed 2D entities and detected closed boundaries on a planar face of a solid	Creates solids or surfaces by revolving the selected 2D entities or boundary. Optionally the created volume is unified with or subtracted from the solid.
	xFaces	Faces of 3D solids and surfaces	Extracts a set of 3D solid or surface faces as a new 3D surface object.
	dmCopyFaces	A set of 3D solid faces	Copies a feature (protrusion or depression) represented by the selected faces from one 3D solid to another, or to a different location on the same 3D solid.
	dmThicken	Faces of 3D solids and surfaces	Creates new 3D solids by thickening faces by a given value.
		Surfaces	Creates a new 3D solid by thickening the whole surface.
		Wire entities (lines, polylines, circles, ellipses, arcs, helices, splines)	Creates a new 3D solid by sweeping a circle along the selected wire.
	dmFillet	Sharp edges of a 3D solid or surface.	Creates a fillet between adjacent faces sharing a sharp edge(s).
	dmChamfer	Sharp edges of a 3D solid or surface	Creates a chamfer between adjacent faces sharing a sharp edge(s).
	Erase	Faces of a solid or a surface. Whole 3D solids and surfaces.	Deletes the selected entities. When faces of a solid entity are deleted, the gap is filled by extending the adjacent faces. When faces of a surface entity are deleted, the gap is filled only if the faces form a fillet.
		Free edges of a surface (a surface edge is called free if it has only one adjacent face)	If the selected free edges belong to one or more holes in the surface, then these holes are deleted and the gap if filled automatically.
		Edges between coplanar faces.	Deletes the edges between coplanar faces, thus unifying the faces.

	dmExtrude	3D curves and edges of 3D solids and surfaces	Creates surfaces by extruding the selected curves and edges in a particular direction, which is defined heuristically, but can also be set by the user.
	dmRevolve	3D curves and edges of 3D solids and surfaces	Creates surfaces by revolving the selected curves and edges around the selected axis.
	dmDeformPoint	Faces of a 3D solid or a surface	Deforms the selected set of faces by moving a point lying on one of them
	dmDeformMove	Edges of a 3D solid or a surface	Deforms faces by moving and rotating their edges.
	dmDeformCurve	Edges of a 3D solid or a surface	Deforms faces by changing the geometry of their edges.
	dmTwist	3D solids, surfaces or regions	Command to modify an object by twisting a portion around a given axis by a certain angle.
	dmSelect	Edges and faces of 3D solids and surfaces	Selects edges and faces of 3D solids or surfaces based on their geometric and topologic properties; i.e. selects faces of a smooth side of a given solid or selects a set of faces forming a protrusion or depression feature.

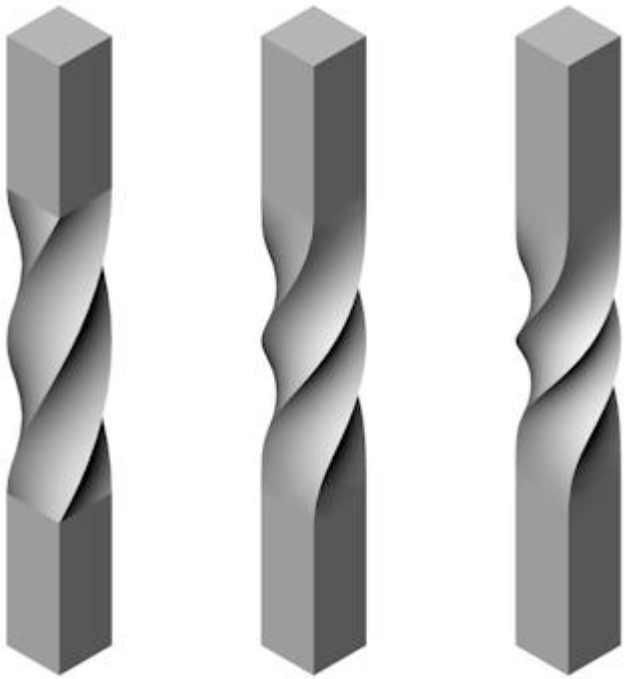
* If the *Select detected boundaries* option of the SELECTIONMODES system variable is ON, boundaries which are enclosed by 2D linear entities in the XY-plane of the current UCS, are detected automatically.

Selection Preview

When the Quad cursor menu is active, the SELECTIONPREVIEW system variable is neglected, unless a command prompts to select an entity (see Selection Settings).

Selecting Geometry

- Hit the *TAB* key to select obscured geometry.
- The SELECTIONMODES system variable controls which subentities highlight: edges, faces and/or detected boundaries.
- If a direct modeling command is entered at the command prompt or selected in a menu or toolbar, you need to press the Enter key (or right click) to conclude the selection procedure.
- If launched from the Quad cursor menu, the command is executed when you click to confirm the selection of the currently highlighted geometry.
- Window selection: Pressing the Ctrl key during window selection allows to select faces or edges of 3D solids and surfaces. An icon next to the cursor indicates what will be selected.
- The dmSelect command allows to select edges and faces based on their geometrical and topological properties:
    - Edges of particular length
    - Edges forming a concave/convex network
    - Edge loops representing the border of a set of connected faces
    - Faces of particular area
    - Smoothly connected faces (G1 - tangent faces, or G2 - curvature continuity)



G0: No continuity (left), G1: Tangent Faces (middle) and G2: Curvature continuity (right)

    - Fillets of particular radius
    - Cylindrical protrusions of particular radius
    - Protrusion or depression features, such as holes, pockets, extrusions or ribs

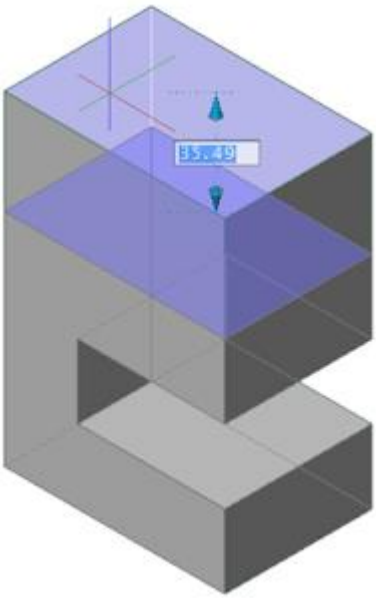
The value of a particular length, area or radius can be taken from a selected edge/face.

To Control Dimensions

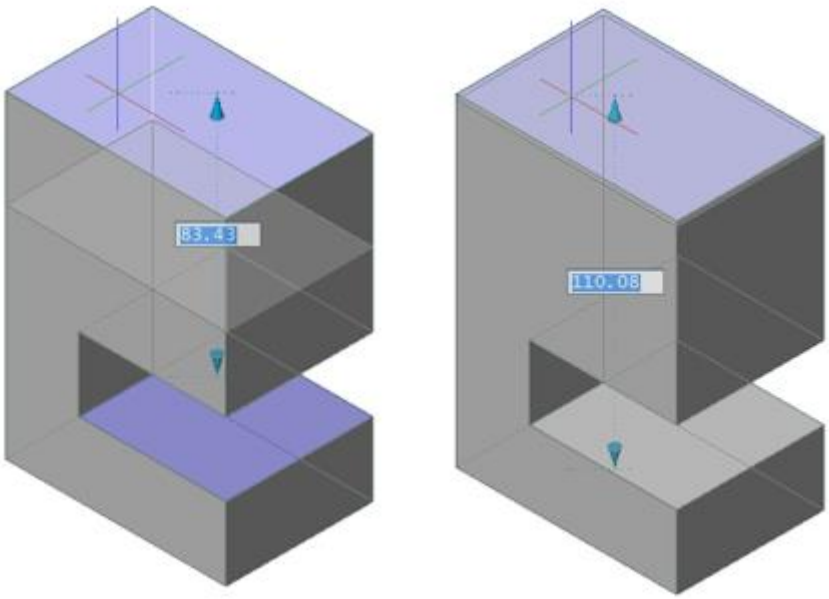
When using the *dmPushPull* command a dynamic dimension displays.

Make sure the Quad and Dynamic Dimensions or ON.

1. Hover the cursor over the face you want to move.
   The face highlights.
2. Choose *PushPull* ( ) in *Direct Modeling* command group of the Quad cursor menu.
   The selected face is attached to the cursor and a dynamic dimension field displays the relative distance.



3. (Option) repeatedly hit the TAB key to see the distance with respect to an internal face, parallel to the selected face (left) or the bottom face (right).
   Press and hold the Shift key while hitting the TAB key to reverse the display order of reference faces.



4. Type a value in the dynamic dimension field, then press Enter.

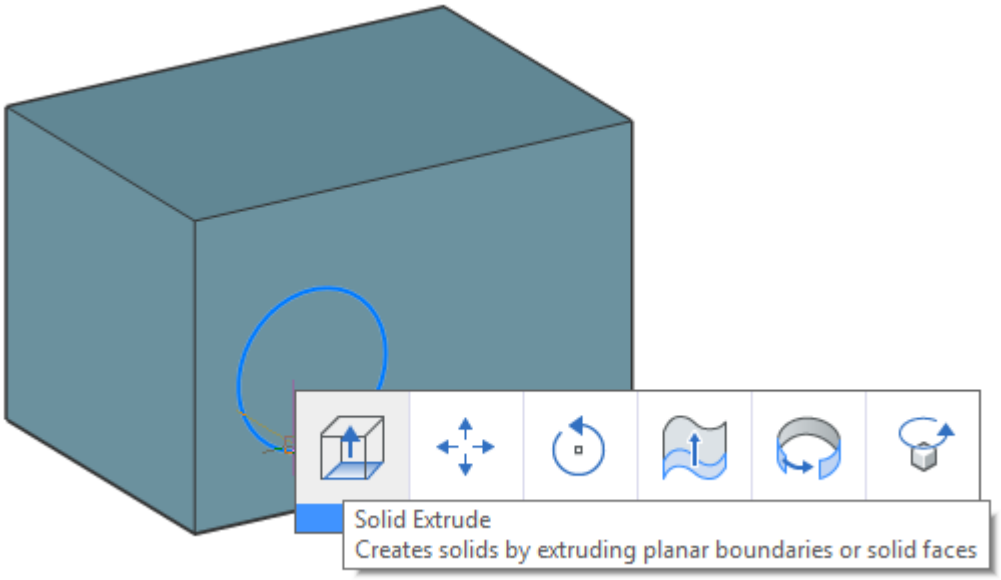
Making Holes

Typically holes are made using the SUBTRACT command.

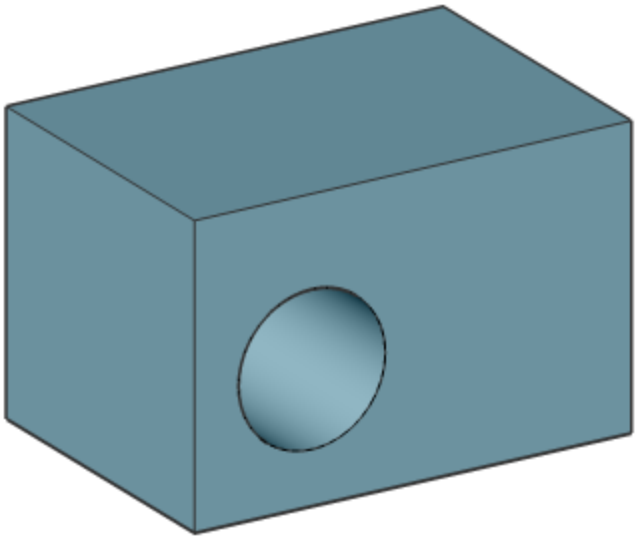
When pushpulling a closed 2D entity that lies on a face of a solid, a hole is created when you push the 2D entity through the solid.

To make a (circular) hole through a solid

1. Make sure Dynamic UCS is active.
2. Launch the Circle command.
3. Move the cursor to the face of the solid you wan to create the circular hole in.
   The UCS automatically aligns to the face the cursor is on.
4. Draw a circle on the face of the solid.
5. Hover over the circle, then choose *Extrude* ( ) in the *Model* command group of the Quad cursor menu.



4. Push the circle inside the solid.



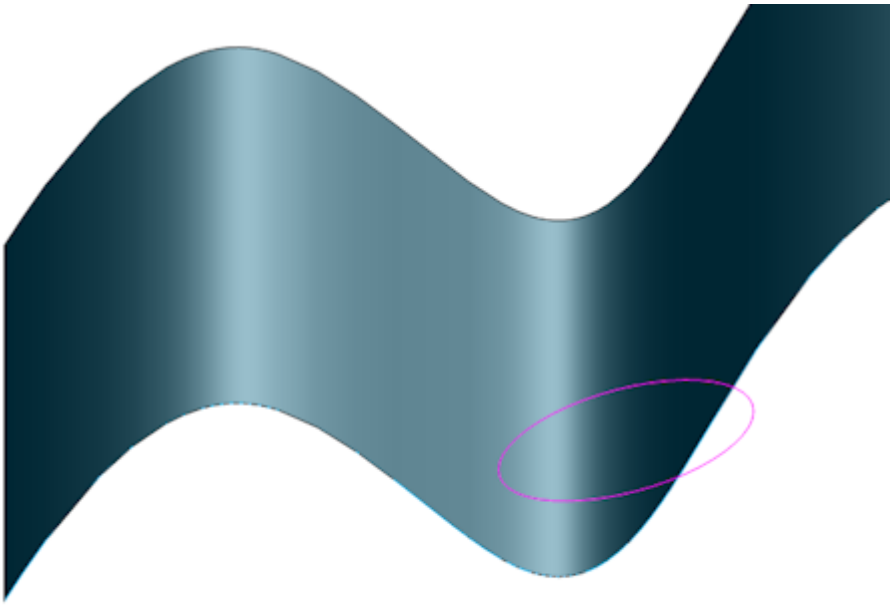
5. Do one of the following:

   - Type a distance in the dynamic field to define the depth of the hole.
   - Push the circle through the solid to create a through-hole.

6. (option) Choose *Push/Pull* ( ) in the *Model* command group of the Quad cursor menu to modify the radius of the hole.
7. (option) Use the dmDistance3d command to apply distance constraints between the side faces of the solids and the center of the cylindrical surface of the hole to control the position of the hole.
8. (option) Use the dmRadius3d command to control the radius of the hole.

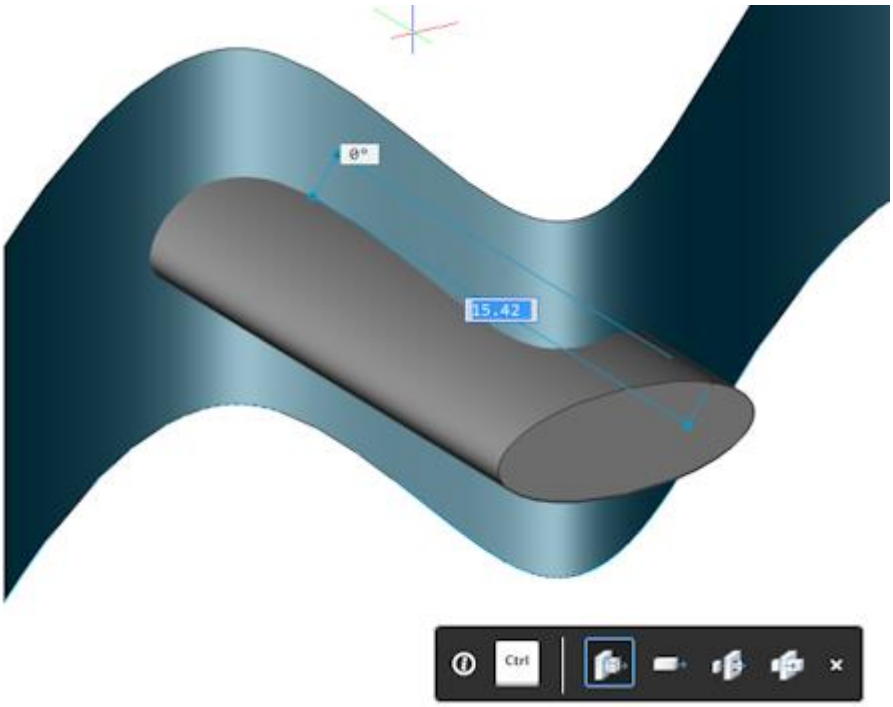
To make a hole in a surface

1. Create a 2D closed entity, e.g. an ellipse.

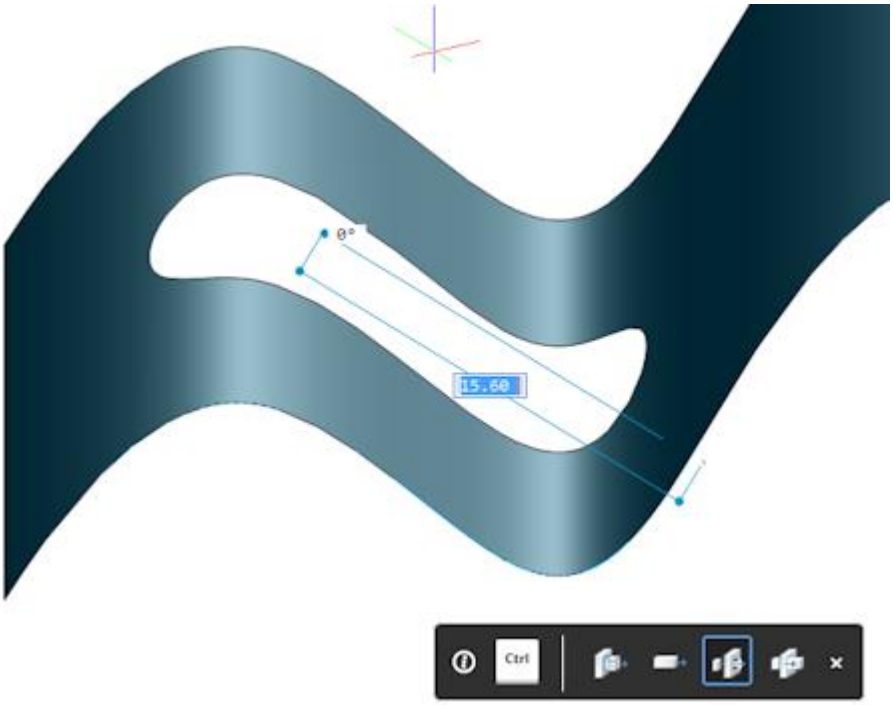


2. Hover over the 2D entity.

3. Choose *Extrude* ( ) in the *Model* command group of the Quad cursor menu.
4. Extrude the 2D entity in the direction of the surface.



5. Press the Ctrl-key twice to select the *Subtract* option of the DmExtrude command.



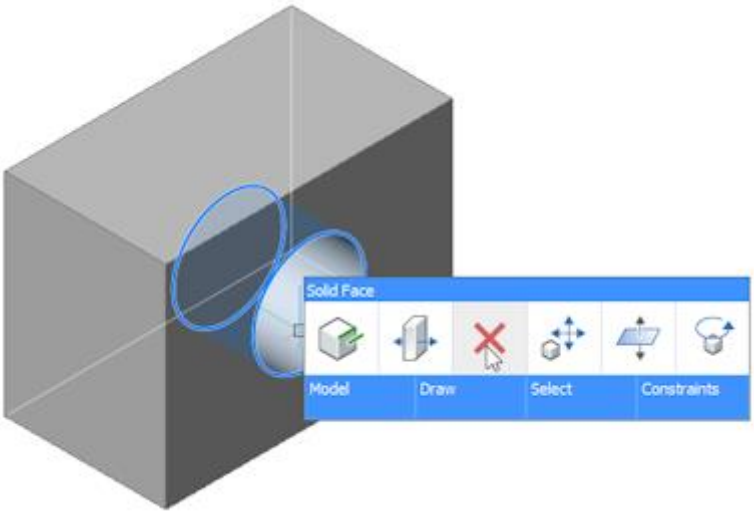
6. Click to create the opening.

Deleting Features

To delete a hole, you must delete all internal faces and/or surfaces of the hole.

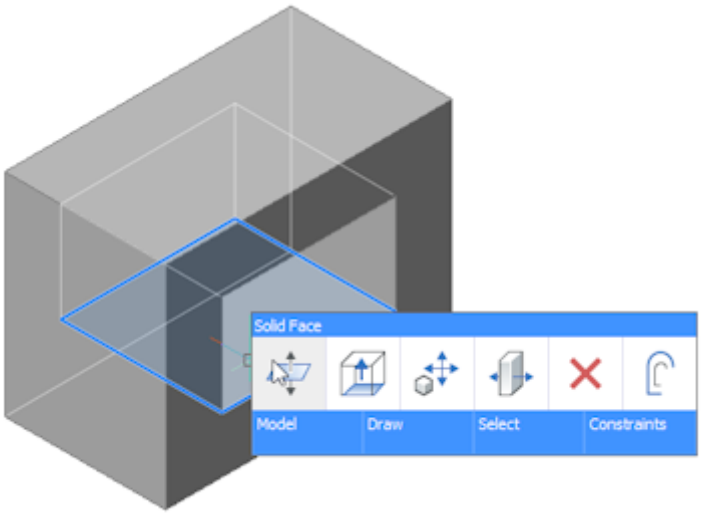
To delete a circular hole

1. Hover over the cylindrical surface of the hole.
2. Choose *Delete* ( ) in the *Model* command group of the Quad cursor menu.



To delete a rectangular hole

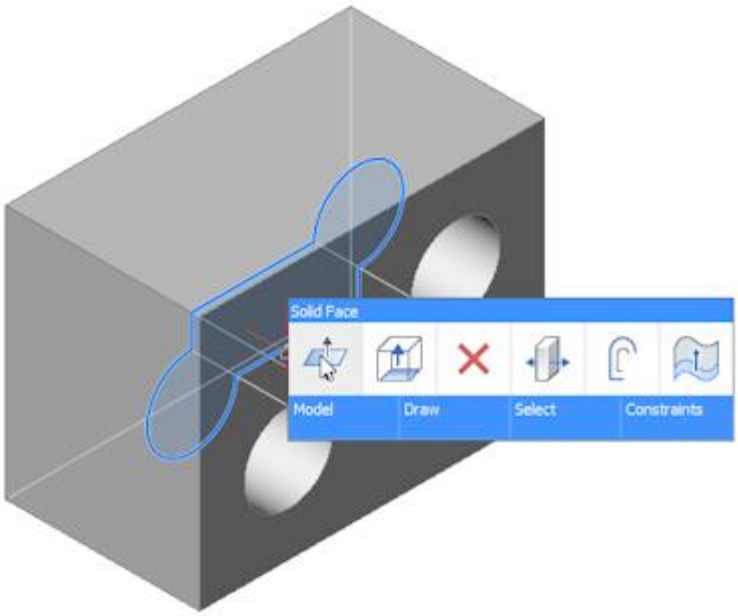
1. Hover over one of the visible internal faces of the hole.
   The face highlights.
2. Choose *PushPull* ( ) in the *Model* command group of the Quad cursor menu.



3. Drag the selected face to close the hole.

To delete a niche or recess

1. Hover over the back (bottom) face of the recess.
   If the face is obscured in the current view, hit the TAB key untill the face highlights.
2. Choose *PushPull* ( ) in the *Model* command group of the Quad cursor menu.



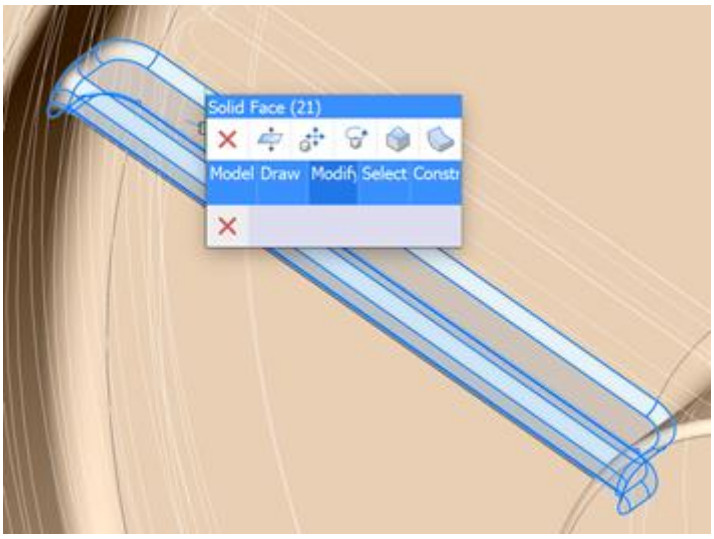
3. Drag the face outside the solid.

To delete a hole in a surface

1. Hover over the edge of the opening.
   The edge highlights.
2. Choose Erase ( ) in the *Model* command group of the Quad cursor menu..

To delete a depression (hole or pocket) or a protrusion feature (extrusion or rib)

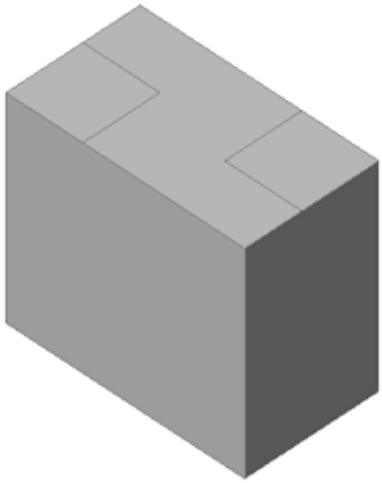
1. Select one or more faces of the feature you want to delete.
2. Click the *Protrusion* ( ) or *Depression* ( ) tool button in the *Select* panel of the *Solid* ribbon tab or in the *Select* command group of the Quad (not supported yet!).
3. Click the *Erase* ( ) tool button.



Removing Edges Between Coplanar Faces

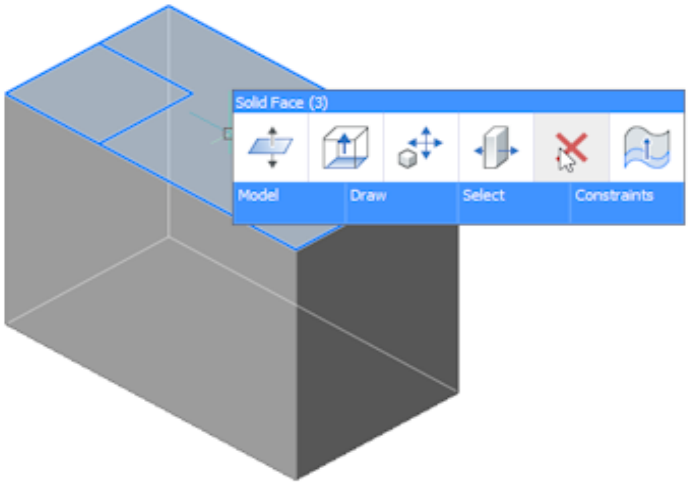
When two or more faces are made coplanar, e.g. using the PushPull command, it might be necessary to remove the shared edges.

Two depressed faces ...

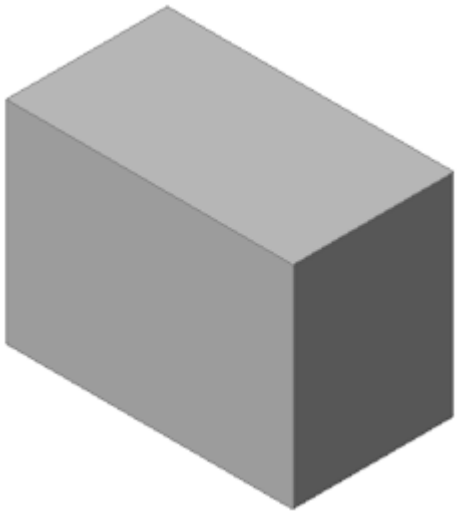


... moved to the top face, creating three coplanar faces.

1. Select the three faces.
2. Choose *Delete* (✕) in the *Model* command group in the Quad.



The shared edges are deleted and the three selected faces are unified.



Copying Features

the dmCopyFaces command allows to copy one or more features of a 3D solid, such as depressions (holes, grooves, pockets) or protrusions (extrusions, ribs) to a new location on the same or a different 3D solid.

To copy a feature do the following:

1. Select all faces of the entire feature(s) (see Selecting Geometry).
2. Make sure dynamic UCS is enabled.
   It is recommend to always keep DUCS enabled when using the Direct Modeling toolset.
3. Click the *Copy Faces* ( ) tool button:

- on the ribbon: *Solid | Edit | Copy Faces*
- on the *Direct Modeling* toolbar.
- in the menu bar: *Model | Direct Modeling | Copy Faces*

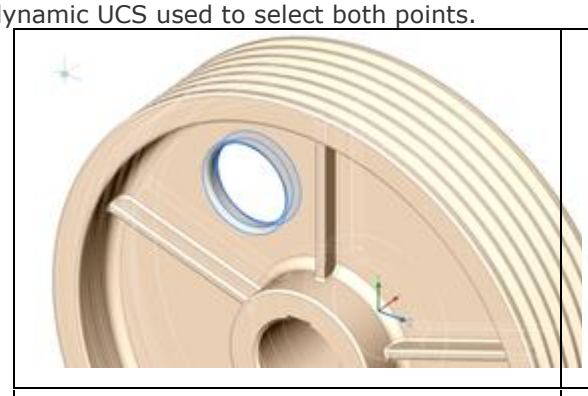
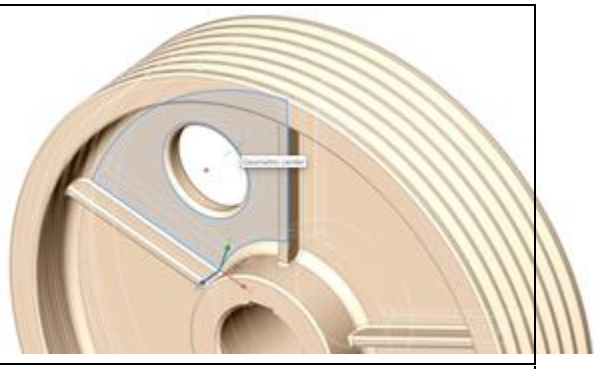
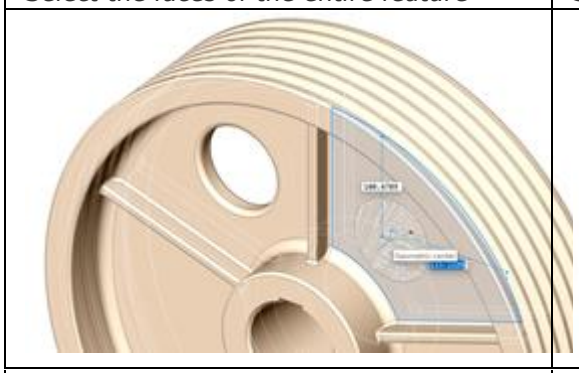
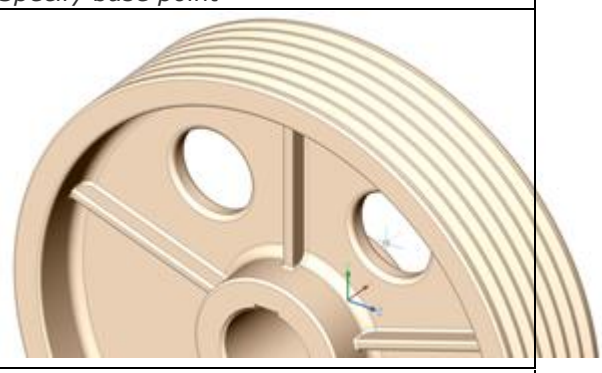
You are prompted: Enter base point:

4. Specify the base point.
   You are prompted: Select insertion point or [Rotate/mUltiple/MOde]:
5. Optionally:

- Use the *Rotate* option to rotate the faces around Z axis of the UCS.

- Use the *Multiple* option to insert the multiple copies of the selected feature.

6. Specify the insertion point.
   The feature is copied from the base point to the insertion point taking into account the dynamic UCS used to select both points.

	
Select the faces of the entire feature	*Specify base point*
	
Specify insertion point	*The feature is copied*

Deformable Modeling

Using the tools on the *Deform* toolbar, you can create complex aesthetic shapes from simpler ones by deforming a set of adjacent faces of 3d solid or a surface.

The faces are always deformed under the following conditions:

- Their external boundary (= edges, which are adjacent to only one of the selected faces) remain unchanged
- Their internal boundaries (= edges, which are adjacent to two of the selected faces) is deformed together with the faces
- The initial continuity between deformed faces (G1 &endash; tangent faces, or G2 &endash; curvature continuity) is kept during deformation

For a twist deformation it is possible to specify the continuity between the deformed and the fixed portion of the entities.

Deformation can be defined in several ways:

- By twisting the whole entity or its portion
- By moving a point lying on a face in any direction in 3D space
- By moving or rotating one or more edges
- By replacing an edge with a given 3D curve

Working with Sections

Commands: SECTION, SECTIONPLANE, LIVESECTION, SECTIONPLANESETTINGS, SECTIONPLANETOBLOCK.

The *Section* command creates section planes of 3D solids; the results are region entities.

The *Sectionplane* command creates a section entity that creates sections of 3D solids, surfaces and meshes.

The *Livesection* command toggles the *Live Section* property of a section plane.

The *Sectionplanesettings* command defines the properties of a section plane entity in the *Drawing Explorer - Section Planes* dialog.

The *Sectionplanetoblock* command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

NOTE The BIMSECTION command creates *BIM section* entities, which are SectionPlane entities with extended properties, such as type (plan, section, elevation)and scale.

SectionPlane command

Do one of the following:

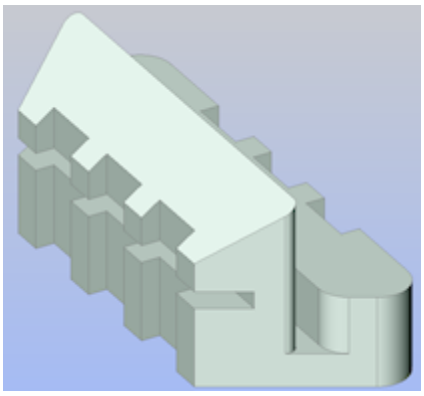
- Click the *Section Plane* tool button ( ) on the *Sections* toolbar.
- Choose *Sections > Section Plan*e in the *Model* menu.
- Type *sectionplane* in the command bar.

You are prompted: Specify start point or [select Face/Draw/Orthographic]:

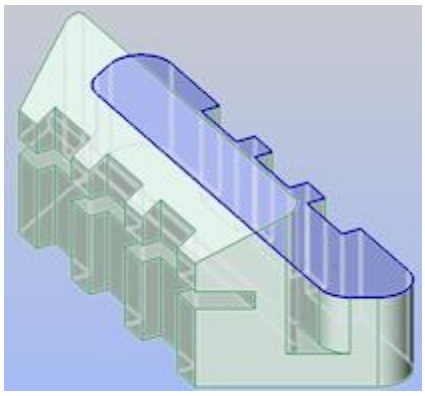
Align Section Plane to a Face

1. Launch the Sectionplane command.
2. Choose *Select Face* in the prompt menu or type *F* in the command bar.
   You are prompted: Select entity:

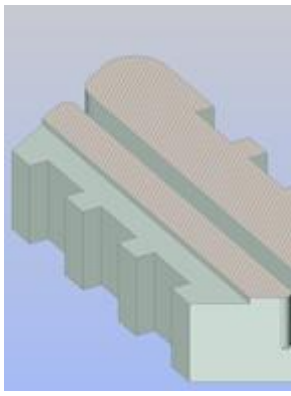
3. Select the face of a solid you want to align the section plane to.



Solid



Select face



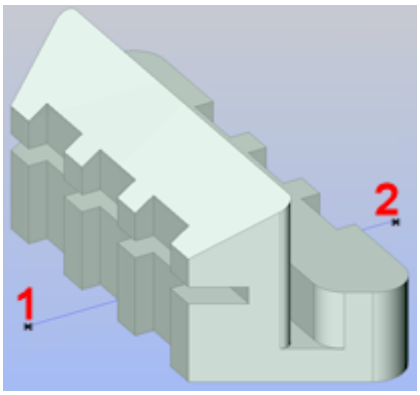
Section plane aligned to

The section plane is aligned to the selected face. The *Clip Display* property of the section entity is switched on automatically.

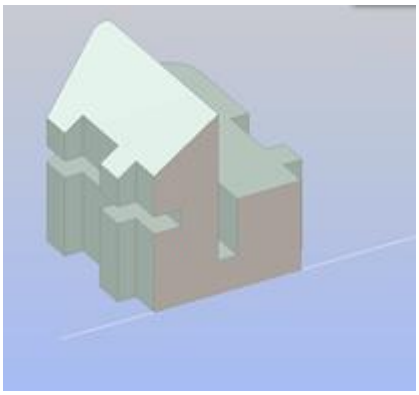
4. (option) Edit the properties of the section plane.

To Create a Vertical Section Plane

1. Make sure the WCS is the current coordinate system.
2. Launch the Sectionplane command.
3. Specify two points.
   The section plane is created through these points and perpendicular to the XY-plane of the current coordinate system.
   The part of the solid at the right hand side of the section plane is cut away.
   The *Clip Display* property of the section entity is not switched on automatically.



Specify two points

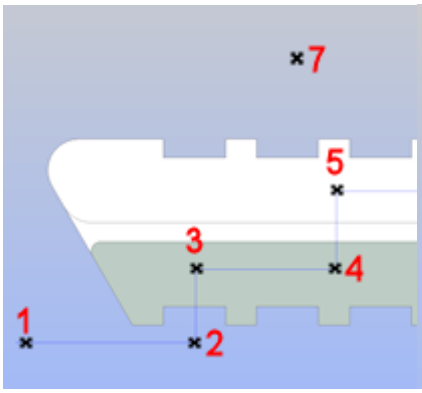


Vertical section plane

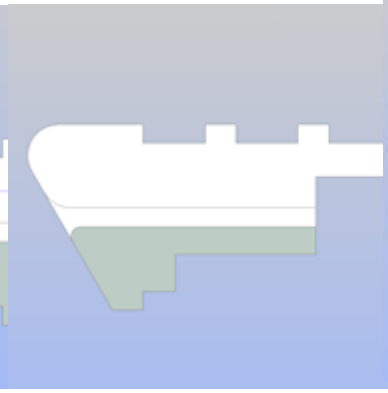
4. (option) Edit the properties of the section plane.

To Create a Vertical Jogged Section Plane

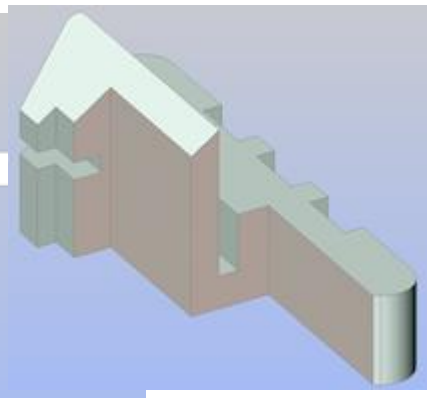
1. Make sure the WCS is the current coordinate system.
2. Launch the Sectionplane command.
3. Choose *Draw* in the prompt menu or type *D* in the command bar.
   You are prompted: Specify start point:
4. Specify the first point (1).
   You are prompted: Specify next point:
5. Specify the second point (2):
   You are prompted: Specify next point or ENTER to finish:
6. Specify the other points (3 - 6) , the right click or press Enter to stop.
   You are prompted: Specify the direction of the section entity:
7. Click a point (7) to specify the direction of the section entity.
   The part of the solid opposite to the direction of the section entity is cut away.
   The *Clip Display* property of the section entity is not switched on automatically.



Specify points



The part of the solid opposite to the direction of the section entity is cut



Vertical jogged section plane

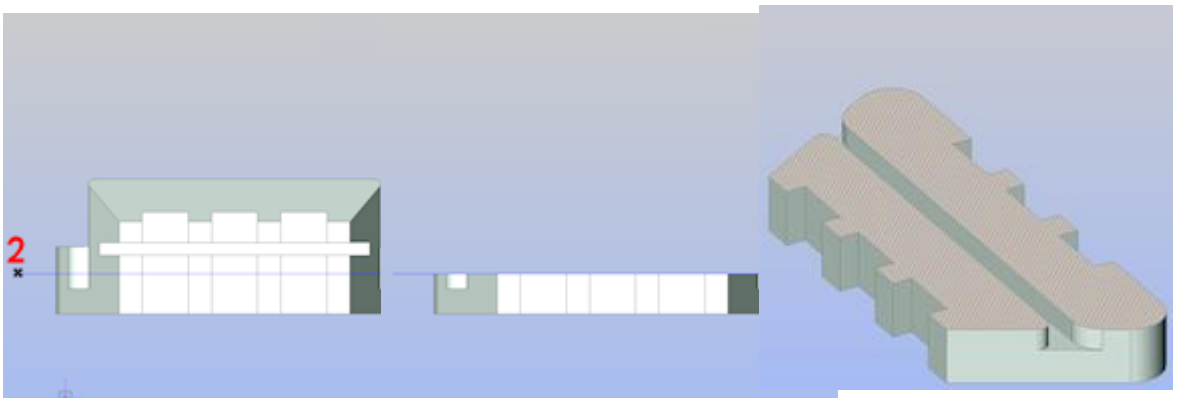
away.

8. (option) Edit the properties of the section plane.

To Create a Horizontal Section Plane

1. Choose *Front* in the *Look From* widget.

2. Align the UCS to the front view:
   Type *ucs* at the command prompt, choose the *View* option in the prompt menu or type *V* in the command bar.
   If UCSORTH0 = ON, the UCS is aligned with the view automatically.
3. Launch the Sectionplane command.
4. Specify two points.
   The section plane is created through these points and perpendicular to the XY-plane of the current coordinate system.
   The part of the solid at the right hand side of the section plane is cut away.
   The *Clip Display* property of the section entity is not switched on automatically.



Specify two points | *The part of the solid at the right hand side of the section plane is cut away.* | *Horizontal section plane*

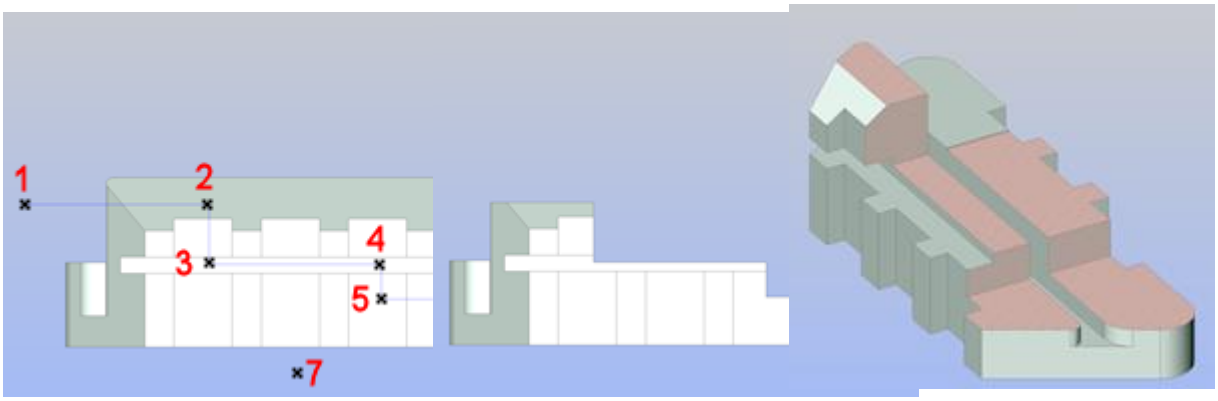
5. (option) Edit the properties of the section plane.

To Create a Horizontal Jogged Section Plane

1. Choose *Front* in the *Look From* toolbar.

2. Align the UCS to the front view:
   Type *ucs* at the command prompt, choose the *View* option in the prompt menu or type *V* in the command bar.
   If UCSORTH0 = ON, the UCS is aligned with the view automatically.

3. Launch the Sectionplane command.
4. Choose *Draw* in the prompt menu or type *D* in the command bar.
   You are prompted: Specify start point:
5. Specify the first point (1).
   You are prompted: Specify next point:
6. Specify the second point (2):
   You are prompted: Specify next point or ENTER to finish:
7. Specify the other points (3 - 6) , the right click or press Enter to stop.
   You are prompted: Specify the direction of the section entity:
8. Click a point (7) to specify the direction of the section entity.
   The part of the solid opposite to the direction of the section entity is cut away.
   The *Clip Display* property of the section entity is not switched on automatically.
   The section plane gets the *Boundary* state.



Specify points

The part of the solid opposite to the direction of the section entity is cut away.

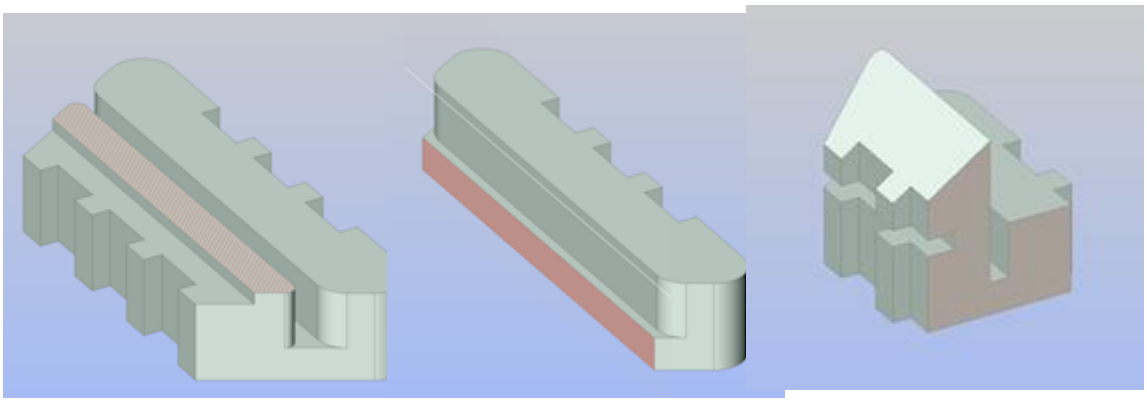
Horizontal jogged section plane

9. (option) Edit the properties of the section plane.

To Create a Orthographic Section Plane

1. Make sure the WCS is the current coordinate system.
2. Launch the Sectionplane command.
3. Choose *Orthographic* in the prompt menu or type *O* in the command bar.
   You are prompted: Align section to [Front/bAck/Top/Bottom/Left/Right] <Top>:
4. Pick the orthographic section orientation of your choice in the prompt menu or type the corresponding option in the command bar.
   Orthographic section planes are created through the center of the solid, parallel to the XY-plane (Top and Bottom), YZ-plane (Left and Right) or XZ-plane (Front and Back) of the current coordinate system.

The *Clip Display* property of the section entity is not switched on automatically.



Top *Front* *Right*

5. (option) Edit the properties of the section plane.

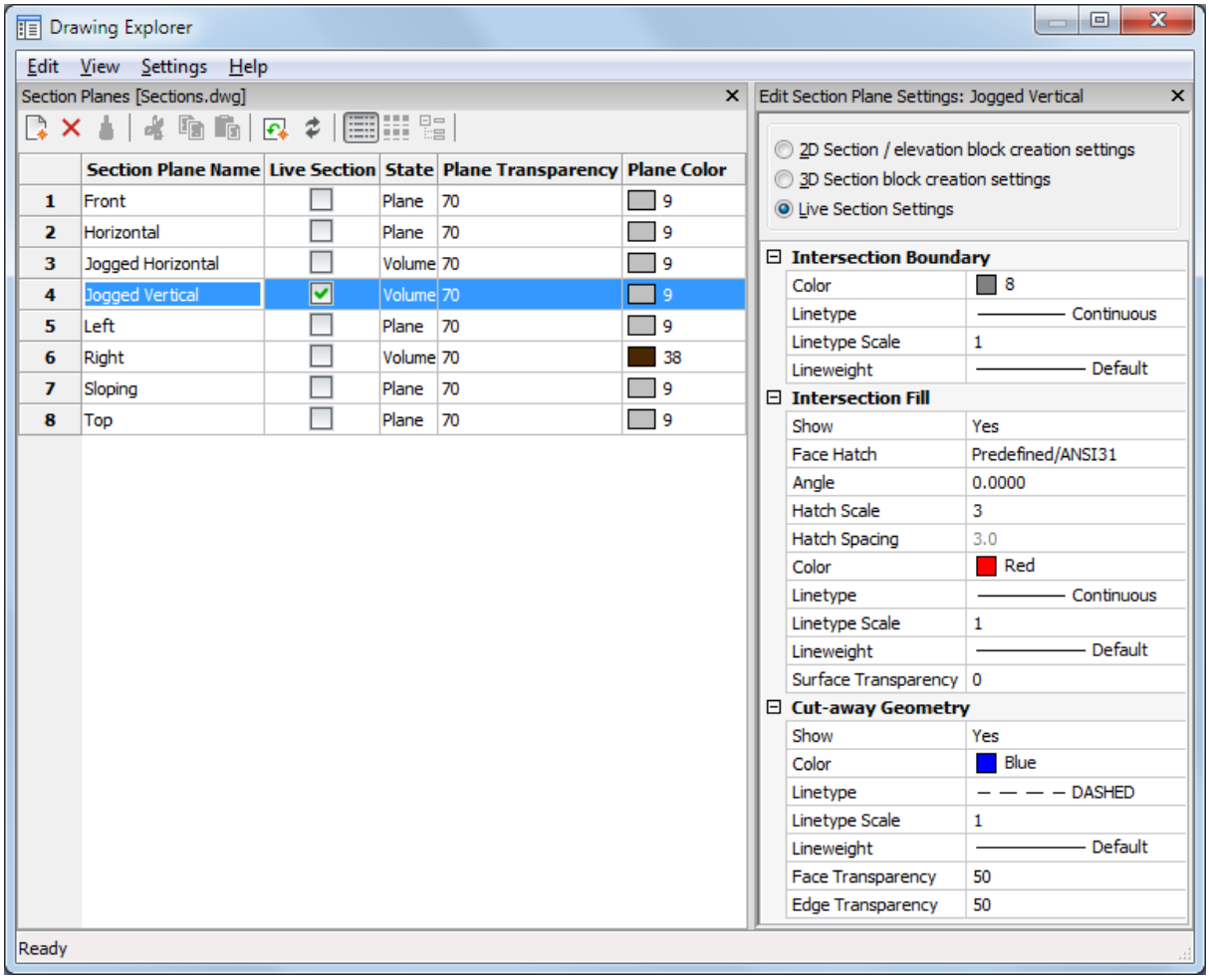
To open the Drawing Explorer - Section Planes dialog

Do one of the following:

- Choose *Drawing Explorer > Section Planes* in the *Tools* menu.
- Choose *Sections > Section Plane Settings...* in the *Model* menu.
- Type *sectionplanesettings* in the command bar, then press Enter.

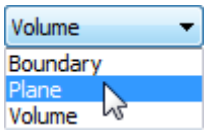
To edit the properties of a section plane in the Drawing Explorer

1. Open the Drawing Explorer - Section Planes dialog.



2. (option) Click the *Section Plane Name* field, then click again to type a new name.
3. (option) Click the *Live Section* check box of a section plane to toggle the *Live Section* property of the section plane.

   If necessary, click the *Regen* tool button ( ) in the *Details* toolbar to update the display.
4. (option) Click the *State* field of a section plane to choose a different state for the section plane.



5. (option) click the *Plane Transparency* field of a section plane to change the transparency of the section plane entity in the drawing.
   0 = Opaque, 100 = Transparent (invisible).
6. (option) Click the *Plane Color* field, then choose a new color in the Select Color dialog.

To edit the properties of a section plane in the properties bar:

Select the section entity in the drawing.

Section Entity	
Name	Plan
State	Plane
Live section	No
Clip display	No
Use lids	No
Plane transparency	70
Plane color	9

- **Name**: allows to rename the section entity.
- **State**: click, then select *Plane*, *Boundary* or *Volume*.
- **Live Section**: click, then select *Yes* or *No*.
- **Clip display**: click, then select *Yes* or *No*.
- **Use lids**: click, then select *Yes* or *No*.
- **Plane transparency**: allows to define the transparency of the section plane.
- **Plane color**: allows to define the color of the section plane.

NOTE **Difference between *Live Section* and *Clip Display*:**

In both statuses all drawing content is sectioned and clipped-away parts are no longer displayed.

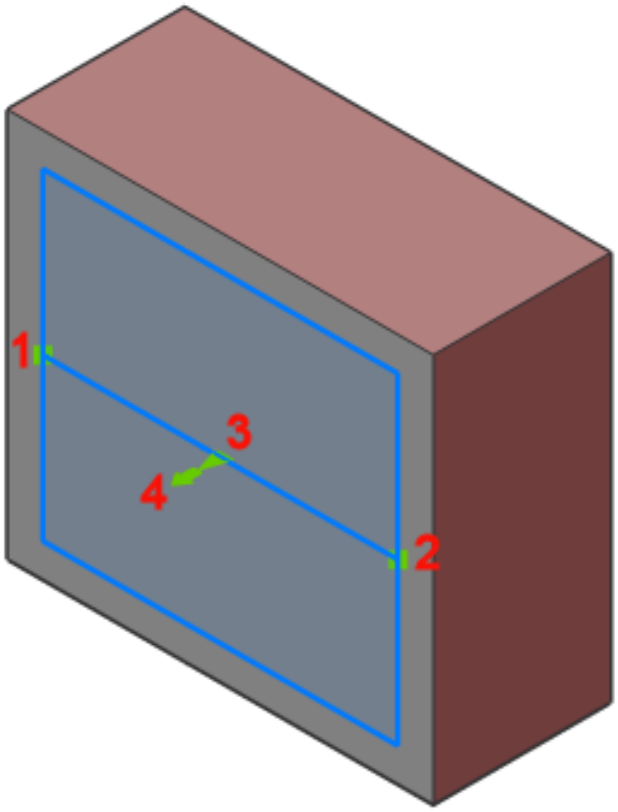
Live status: entities in a drawing are temporarily modified, e.g. clipped solids are replaced by temporary solids with a different shape.

Clip Display status: the display of entities is clipped. As in the AutoCAD 3DCLIP command, entities are not replaced by temporary versions. Different from the *Live* status, the *Clip Display* status allows to continue using all modeling and editing operations on all drawing content. While the 3DCLIP command allows to clip the display by a *Front* and a *Back* plane, it is not very useful because these planes must be reset whenever the view direction changes. Using a Section entity to control which part of a model is displayed, makes it easy to store and repeat display states. Besides the *Plane* state, which typically clips a model in two halves, also the *Boundary* and *Volume* state of section entity can be used.

As opposed to the *Live* status, the Clip Display status can be set for multiple section entities simultaneously. Clipped solids can optionally be closed by setting the *Use Lids* option. Althoug it has been made possible to snap to the geometry of these lids, they are not actually added to the drawing. If enabled, the hatch pattern defined in the *2dSection Intersection Fill* settings of the section entity displays on top of the lids.

To Modify the Size and Position of a Section Plane

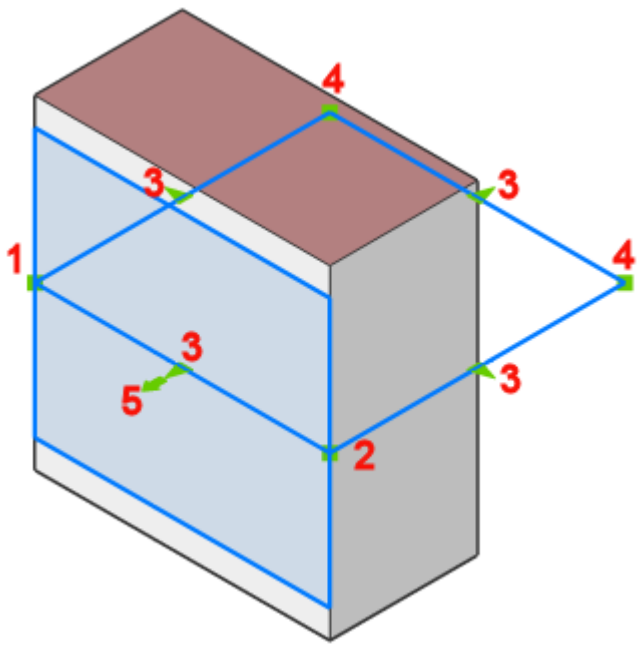
1. Select the section plane entity in the drawing.
   Depending on the current state of the section plane entity a number of grips display.
2. (option) Switch on the *Live Section* property of the section plane to see the result of the modifications dynamically.
3. (option) To grip edit a *Plane* state section plane entity:



Four grips display.

- 1 (■) = origin point: moves the position of the section plane in the XY-plane.
- 2 (■) = through point: redefines the through point of the section plane.
  The section plane is rotated around the origin point (1).
- 3 (▲) = midpoint: moves the section plane parallel in a direction perpendicular to the section plane.
- 4 (⬆): click to flip the view direction of the section plane.

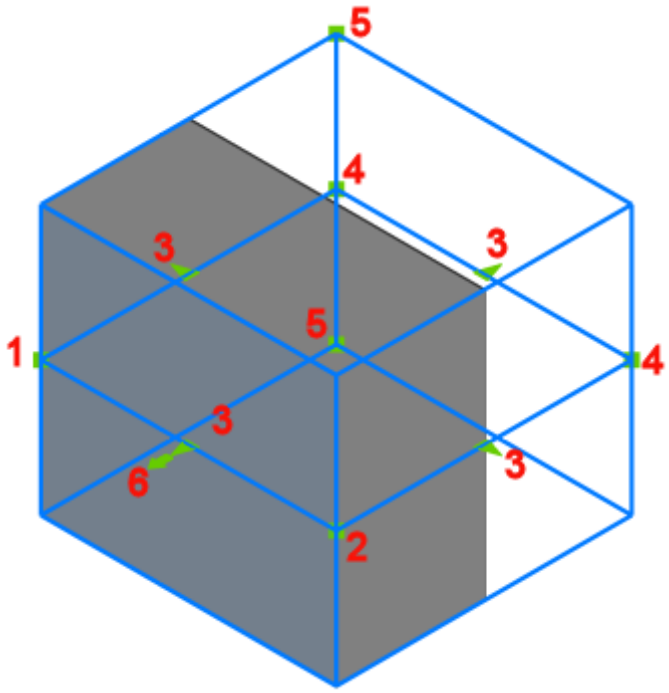
3. (option) To grip edit a *Boundary* state section plane entity:



Ten grips display.

- 1 (■) = origin point: moves the position of the section plane in the XY-plane.
- 2 (■) = through point: redefines the through point of the section plane.
  The section plane boundary is stretched and rotated around the origin point (1).
- 3 (▲): stretches the section plane boundary parallel.
- 4 (■): modifies the shape of the section plane boundary.
- 5 (⬆): click to flip the view direction of the section plane.

4. (option) To grip edit a *Volume* state section plane entity:

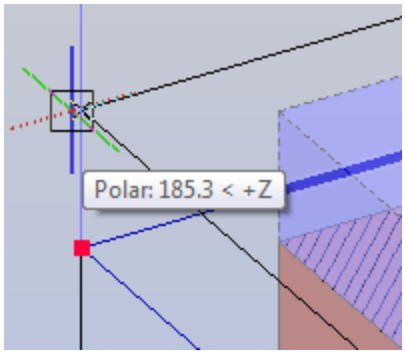


Twelve grips display.

- 1 (■) = origin point: moves the position of the section plane in the XY-plane.

- 2 (■) = through point: redefines the through point of the section plane.
  The section plane volume is stretched and rotated around the origin point (1).
- 3 (▲) :stretches the section plane volume parallel.
- 4 (■): modifies the shape of the section plane volume.

Polar Tracking must be on to move grips 5.



- 5 (■): modifies the height of the section plane volume.
- 6 (↑): click to flip the view direction of the section plane.

Using Live Section

When the *Live Section* property of a section plane is on, the sectioning of a 3D solid, surface or region displays in the drawing. If the section entity is moved or modified, the sectioning is updated dynamically. The *Live Section Settings* of the section entity control the appearance of the live section.

To set the *Live Section* property of a section plane:

Method 1: Using the LIVESECTION command.

1. Do one of the following:
   - Click the *Live Section* tool button ( ) on the *Sections* toolbar.
   - Choose *Live Section* in the *Model > Sections* menu.

You are prompted: Select a section entity:

2. Click a section entity in the drawing.
   The *Live Section* property of the selected section plane is toggled.

Method 2: Using the Properties bar.

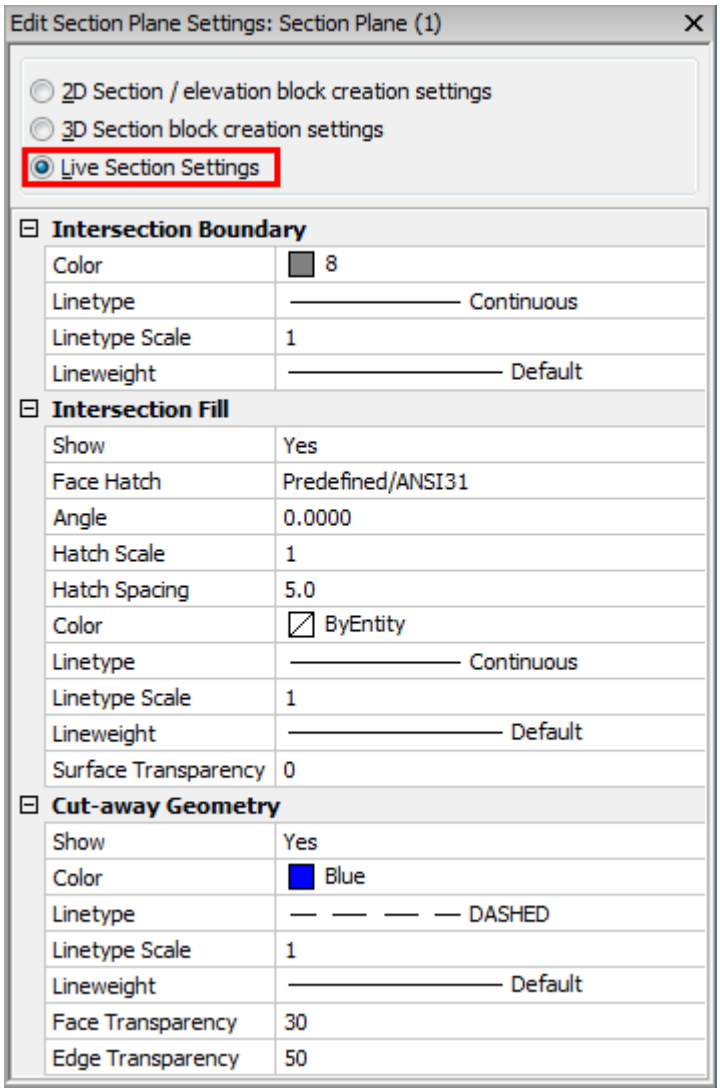
1. Select the section plane entity in the drawing.
2. In the Properties Bar, under *Selection Entity*, click the *Live Section* field and select *Yes*.

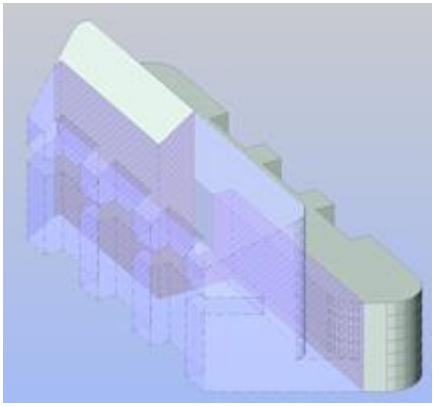
Method 3: Using the *Drawing Explorer - Section Planes* dialog.

1. Select the section plane in the Drawing Explorer - Section Planes dialog
2. Click the check box in the *Live Section* field.

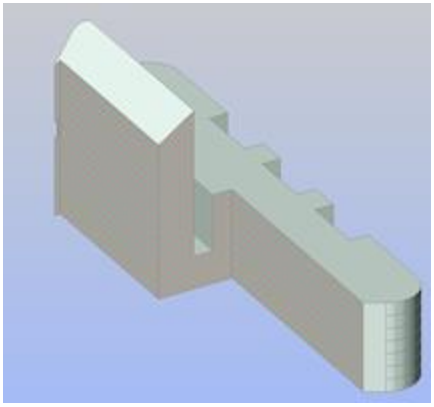
To define the live section settings

1. Open the Drawing Explorer - Section Planes dialog.
2. Select the section plane.
3. Select *Live Section Settings* in the *Edit* pane on the *Drawing Explorer - Section Planes* dialog.
   The live section settings display.





Cut-away Geometry ON



Cut-away Geometry OFF

To Save a Section

The SECTIONPLANETOBLOCK command allows you to:

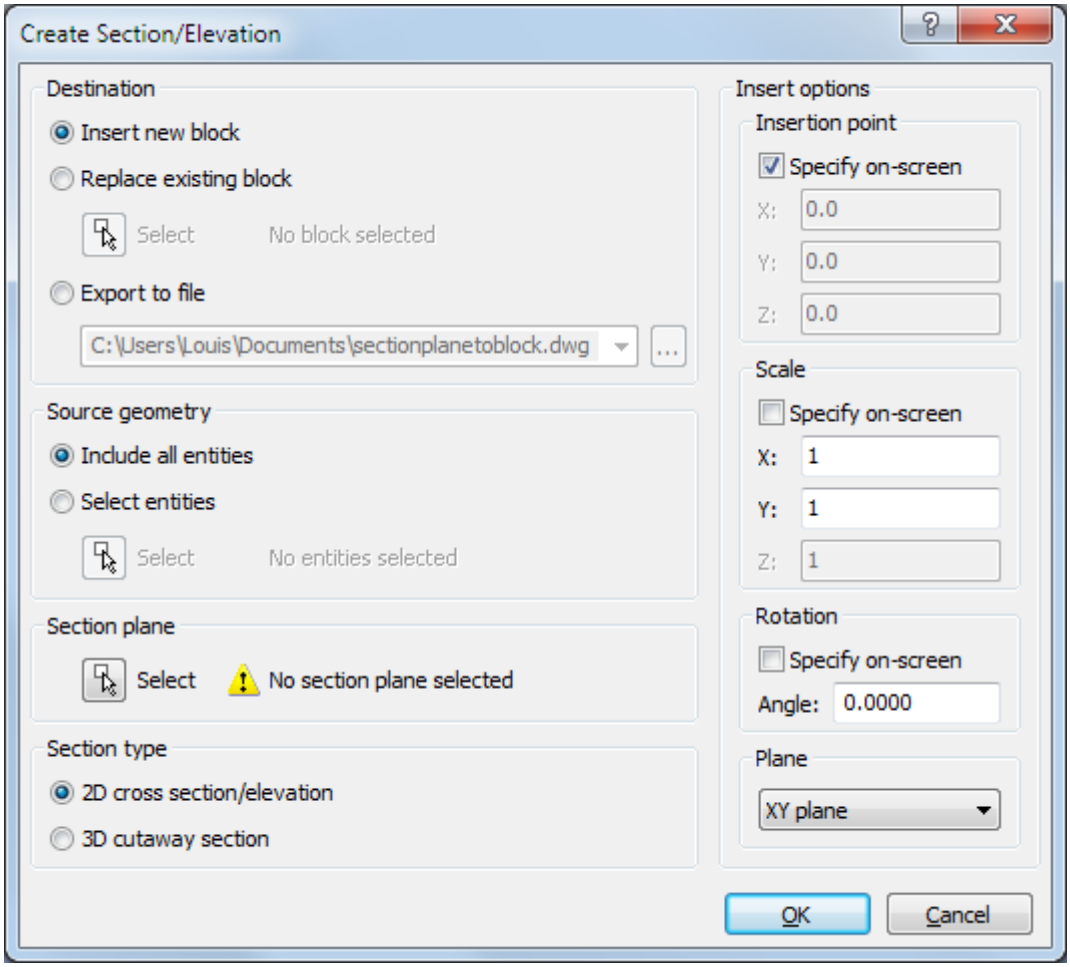
- Insert a section in the drawing as a 2D section / elevation block or a 3D cutaway block.
- Replace a section block (e.g. when the section entity has been modified).
- Save a section to a new drawing.

To launch the SectionPlaneToBlock command

Do one of the following:

- Click the *Section Plane to Block* tool button ( ) on the *Sections* toolbar.
- Choose *Section Plane to Block* in the *Model > Sections* menu.
- Type *sectionplanetoblock* in the command bar, then press Enter

The *Create Section/Elevation* dialog box displays:



To Insert a Section as a 2D Block

1. (option) Adjust the 2D Section / Elevation settings of the section plane entity.
2. Launch the Section Plane to Block command.
3. Set the *Destination* to *Insert new block*.
4. Under *Source Geometry*, do one of the following:
   - Choose *Include all entities* to include all 3D solids in the drawing.
   - Choose *Select entities*, then click the *Select* button ( ) to select 3D solids.

4. Under *Section plane*, click the *Select* button ( ), then select a section plane entity in the drawing.
5. Set the *Section type* to *2D cross section/elevation*.
6. Adjust the *Insert* options (see Insert a block).
7. Click the *OK* button.
8. The *Create Section/Elevation* dialog closes.
   The block is attached to the cursor.
   Follow the instructions in the command bar to place the block in the drawing.

To Insert a Section as a 3D Block

1. (option) Adjust the 3D cutaway section settings of the section plane entity.
2. Launch the Section Plane to Block command.
3. Set the *Destination* to *Insert new block*.
4. Under *Source Geometry*, do one of the following:
   - Choose *Include all entities* to include all 3D solids in the drawing.
   - Choose *Select entities*, then click the *Select* button ( ) to select 3D solids.

4. Under *Section plane*, click the *Select* button ( ), then select a section plane entity in the drawing.
5. Set the *Section type* to *3D cutaway section*.
6. Adjust the *Insert* options (see Insert a block).
7. Click the *OK* button.
8. The *Create Section/Elevation* dialog closes.
   The block is attached to the cursor.
   Follow the instructions in the command bar to place the block in the drawing.

To Replace an Existing block

1. Launch the Section Plane to Block command.
2. Set the *Destination* to *Replace existing block*.
3. Click the *Select* button ( ), then select the block to be replaced in the drawing.
4. Under *Source Geometry*, do one of the following:
   - Choose *Include all entities* to include all 3D solids in the drawing.
   - Choose *Select entities*, then click the *Select* button ( ) to select 3D solids.

4. Under *Section plane*, click the *Select* button ( ), then select a section plane entity in the drawing.
5. Under *Section type*, do one of the following:

   - Choose *2D cross section/elevation*, to replace the block by a 2D section / elevation block.
   - Choose *3D cutaway section,* to replace the block by a 3D cutaway section block.

6. Click the *OK* button.
   The selected block is replaced.

To Export a Section to a File

1. Launch the Section Plane to Block command.
2. Set the *Destination* to *Export to file*.
3. Click the *Browse* button (...), then do the following in the *Select export file* dialog box:
   - Select a folder.
   - Type a name in the *File name* field.
   - Click the *Save* button.
4. Under *Source Geometry*, do one of the following:
   - Choose *Include all entities* to include all 3D solids in the drawing.
   - Choose *Select entities*, then click the *Select* button ( ) to select 3D solids.

   5. Under *Section plane*, click the *Select* button ( ), then select a section plane entity in the drawing.
6. Under *Section type*, do one of the following:
   - Choose *2D cross section/elevation*, to replace the block by a 2D section / elevation block.
   - Choose *3D cutaway section*. to replace the block by a 3D cutaway section block.
7. Click the *OK* button.
   Section is saved to a drawing.

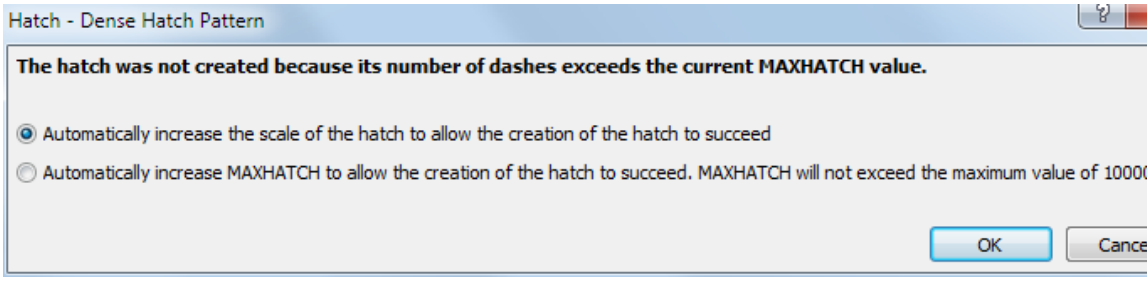
Hatches and Gradient Fills

Commands: HATCH, -HATCH, GRADIENT, HATCHEDIT, -HATCHEDIT

When you add hatching to a drawing, BricsCAD fills entities or enclosed areas with a predefined pattern or lines. First you specify the hatch pattern and other options, and then you choose which entities or enclosed areas that you want to hatch.

NOTES

- Depending on the pattern, hatchings are memory intensive and might take a considerable amount of time to draw and display. To improve performance, add hatching as one of the last steps when you create a drawing, or insert hatches on a separate layer that you can freeze as you continue to work on your drawing. Or use the *FILLMODE* system variable to switch off the display of hatches, gradient fills and filled entities.
- The maximum number of dashes is controlled by the *MAXHATCH* settings variable. When the number of dashes is higher than the current MAXHATCH value, the hatch cannot be created.
  A warning box displays offering possible choices for subsequent actions:



  By default the maximum number of dashes is 100 000.

The predefined and user pattern files sit in the *Support* folder of the current user.

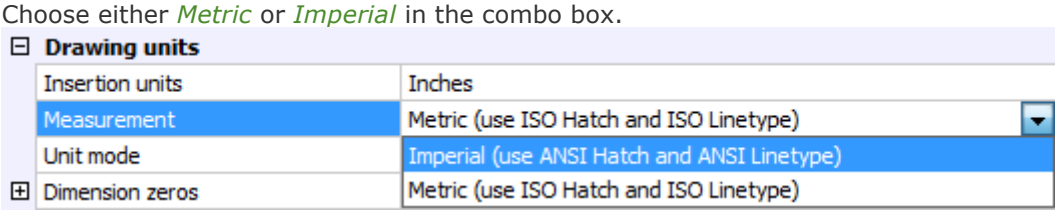
The *MEASUREMENT* setting controls which predefined hatch pattern file will be used.

- *Imperial*: uses Default.pat
- *Metric*: uses Iso.pat

Defining the MEASUREMENT setting

Do one of the following:

- In the command bar type *measurement*, then press Enter.
  Type *ON*, then press Enter to set *measurement* to *Metric*.
  Type *OFF*, then press Enter to set *measurement* to *Imperial*.
- In the Settings dialog go to *Drawing > Drafting > Drawing units.*
  Choose either *Metric* or *Imperial* in the combo box.



Defining the MAXHATCH setting

Do one of the following:

- In the command bar type *maxhatch*, then press Enter.
  Type a new value and press enter.
- In the Settings dialog, go to *Drawing > Drafting > Entity Creation > Hatches*.
  Type a new value in the Maximum hatch dashes settings field.

Hatches	
Hatch pattern angle	0
Hatch pattern boundary	Polyline
Hatch pattern doubling	Hatch pattern doubling
Hatch pattern name	ANSI31
Hatch pattern origin	0.0000, 0.0000
Hatch pattern scale	1.0000
Hatch pattern spacing	1.0000
Hatch pattern style	Standard
Maximum hatch dashes	100000

The *Hatch* command fills an enclosed area in your drawing with a pattern.

The *Gradient* command creates solid fills in nine different patterns and one or two colors.

The area can either be a single entity such as a circle or a closed polyline or an area enclosed by a selection of entities.

To create the hatch you can click inside the closed perimeter of a boundary or you can select entities.

General procedure to create hatches and gradient fills

1. Do one of the following:
   - Click the *Hatch* tool button ( ) or the *Gradient* tool button ( ) on the *Draw* toolbar.
   - Choose *Hatch...* or *Gradient...* in the *Draw* menu.
   - Type *hatch* or *gradient* in the command bar, then press enter.

The *Hatch and Gradient* dialog displays.

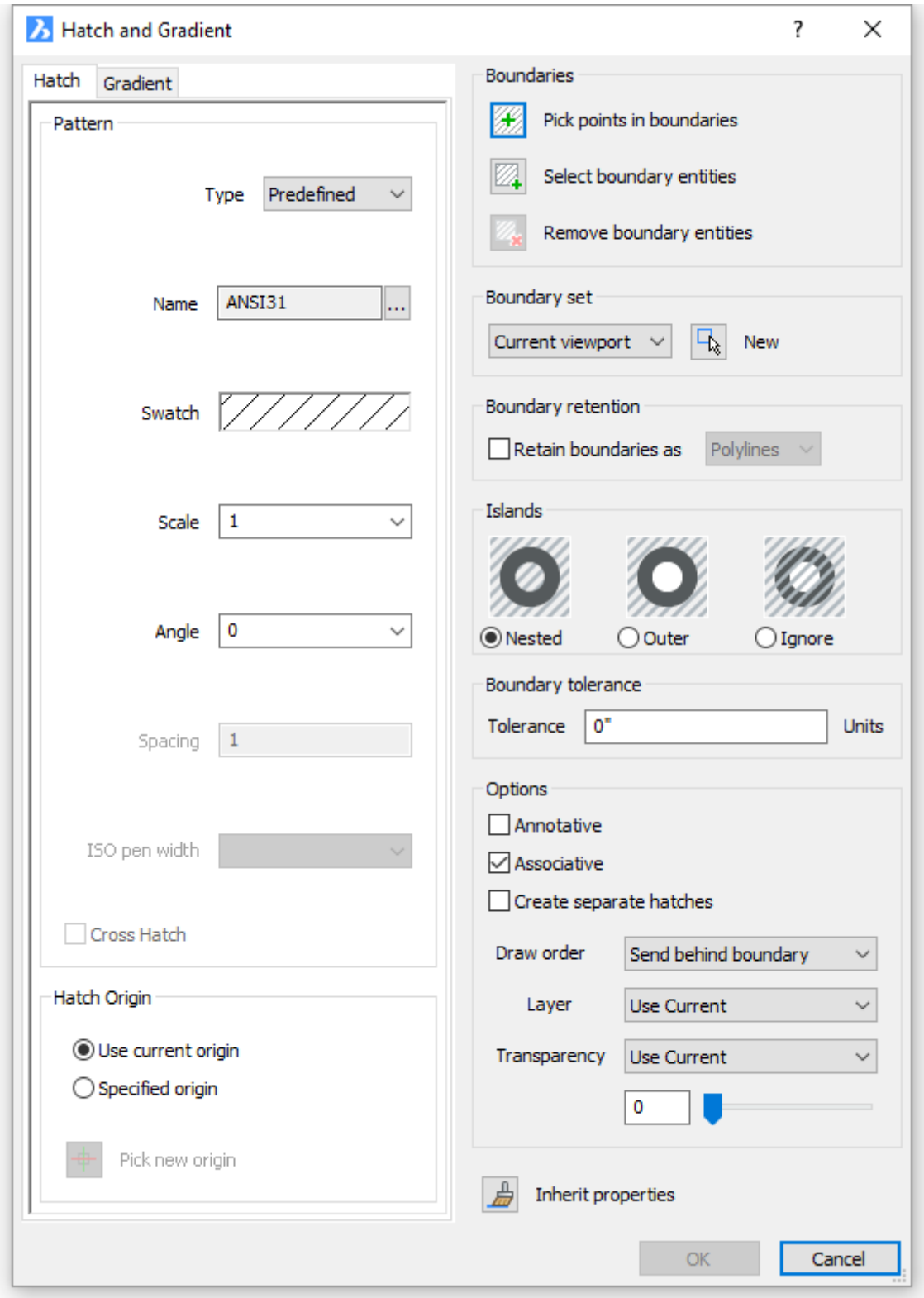
2. Do one of the following:

   - Click the *Hatch* tab to create a hatch, then define the hatch properties.
   - Click the *Gradient* tab to create a gradient fill, then define the gradient fill properties.

3. Define the boundary for the hatch or gradient fill.
4. Set the options for the hatch or gradient fill.
5. Click the *OK* button to create the hatch or gradient fill.

To Define Hatch Properties

1. On the Hatch and Gradient dialog click the *Hatch* tab.

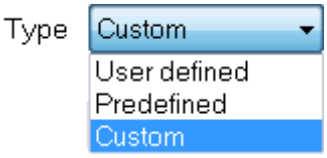


2. (option) Click the *Inherit properties* button ( ).

   - The *Hatch and Gradient* dialog closes temporarily.
     Prompts you: Select hatch entity to copy properties from:

- Select a hatch entity.
  The *Hatch and Gradient* dialog reopens, displaying the properties of the selected hatch entity.

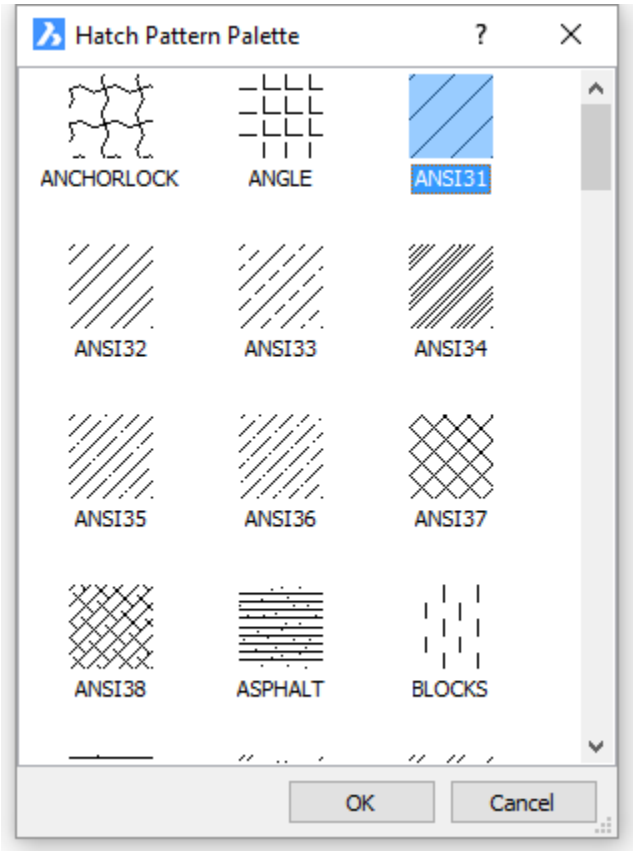
3. (option) Select the *Pattern Type*.



- **User Defined** - uses the pattern defined in this dialog box with the Angle, Spacing, and Cross Hatch options.
- **Predefined** - uses patterns defined in *iso.pat* (metric units - MEASUREMENT = ON) and *default.pat* (imperial units - MEASUREMENT = OFF).
- **Custom** - uses patterns defined in *.pat files (one pattern definition per file).

BricsCAD searches for *.pat files in the folders that are defined in the SRCHPATH (Support File Search Path) system variable.

4. (option) If the *Pattern Type* is *Predefined* or *Custom* select a *Pattern Name* in the *Hatch pattern* palette,
   then double click the pattern or click the *OK* Button.



5. (option) If the *Pattern Type* is *Predefined* or *Custom*, do one of the following:

- Type a scale in the *Scale* settings field.
- Choose the scale from the drop down list.

5. (option) Do one of the following:

   - Type an angle in the *Angle* settings field.
   - Choose the angle from the drop down list.

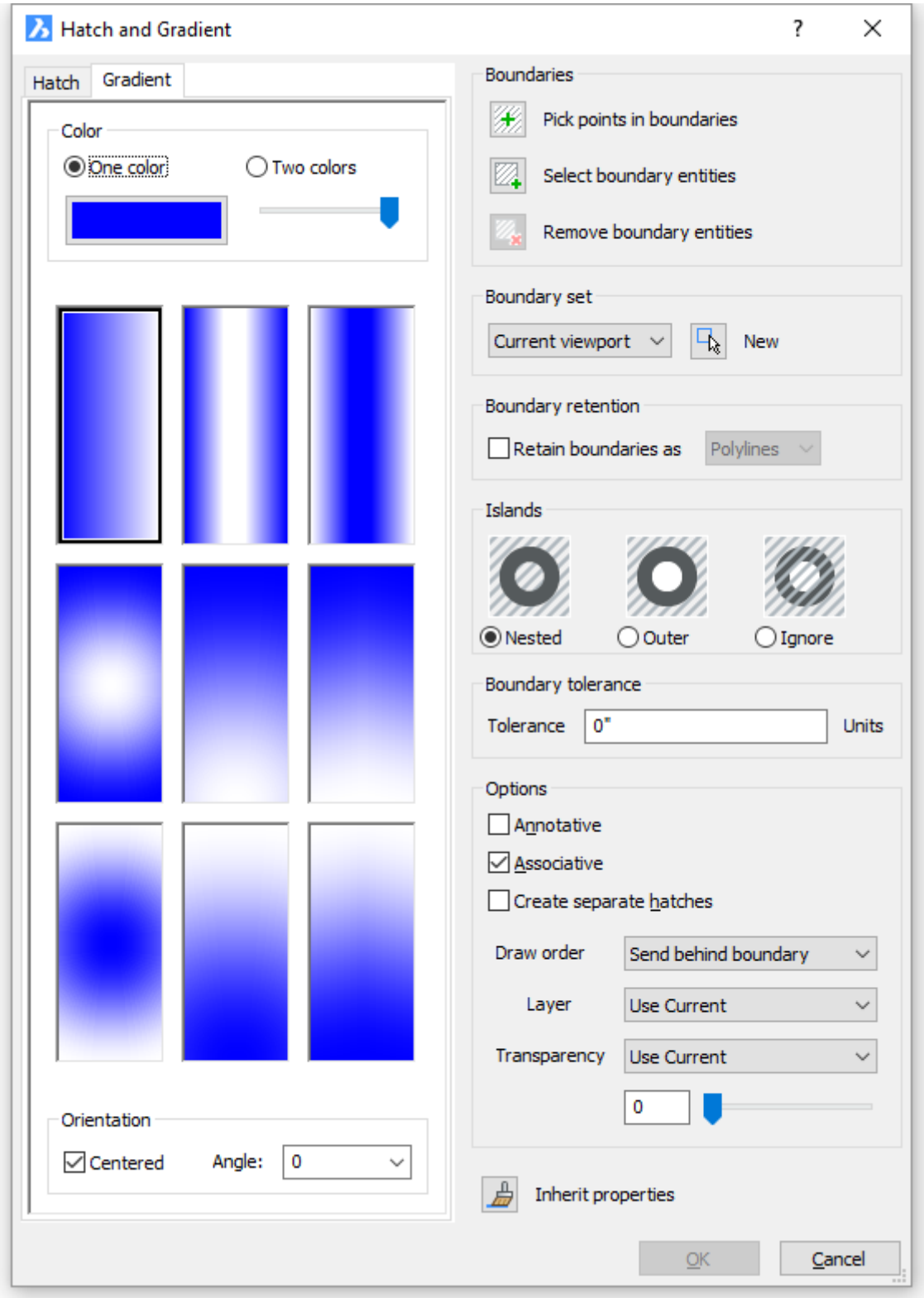
6. (option) If the *Pattern Type* is *User defined*, specify the *Pattern Spacing*.
7. (option) If the *Pattern Type* is *User defined*, select the *Cross Hatch* option.
8. Define the Hatch Origin. The options are:

   - *Use current origin*: The origin as defined by the HPORIGIN system variable is reused.
   - *Specified origin*: Click the *Pick new origin* button ( ), then pick a point in the drawing. The new origin is stored in the HPORIGIN system variable.

NOTE The BACKGROUNDCOLOR property of a hatch can be modified in the *Pattern / Background Color* field in the Properties panel.

To Define Gradient Fill Properties

1. On the Hatch and Gradient dialog click the *Gradient* tab.



2. (option) Click the *One color* radio button to create a 'color to white' or 'color to black' gradient fill.

- Click the colored tile to open the *Select color* dialog, to pick a different base color.
- The color slider sets the transition from color to white or black.

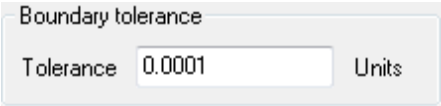
  Press and hold the left mouse button to move the color slider ( ) .
  Move the slider to the far right to create a 'color to white' gradient fill.
  Move the slider to the far left to create a 'color to black' gradient fill.

2. (option) Click the Two color radio button to create 'two color' gradient fill.
   Click the colored tiles to open the *Select color* dialog to pick the base colors.
3. Select one of the 9 gradient patterns.
   The options are: linear, cylindrical, inverted cylindrical, spherical, hemispherical, curved, inverted spherical, inverted hemispherical and inverted curved.
4. (option) *Centered*: If centered, the gradient fill is created symmetrical; if not centered, the gradient fill is built up from the left of the boundary.
5. (option) *Angle*: Specifies the angle of the gradient fill, relative to the current UCS.

To define the boundary for a hatch or gradient fill

1. (option) Select the *Boundary retention* option.
2. (option) Select an *Island* option.

3. (option) Adjust the *Boundary tolerance*.



4. (option) Click the *Select* button ( ) to select a *Boundary set*.
   The *Hatch and Gradient* dialog temporarily closes to let you select entities.
   (see also the note below)
5. (option) Click the *Specified Origin* radio button, then click the *Pick a new origin* button ( ).
   The *Hatch and Gradient* dialog temporarily closes to let you specify a new origin.
6. Do one or more of the following:

- Click the *Pick Points* button ( ).
  The *Hatch and Gradient* dialog closes.
  You are prompted: Select a point to define a boundary or hatch area:
  - Specify a point in each of the areas you want to hatch.
    You are prompted: Select a point to define a boundary or hatch area:
  - Right click or press Enter to stop adding areas.
    The *Hatch and Gradient* dialog displays again.
- Click the *Select Boundary Entities* button ( ).
  The *Hatch and Gradient* dialog closes.
  You are prompted: Select entities:
  - Select the entities you want to hatch.
  - Right click or press Enter to stop selecting entities.
    The *Hatch and Gradient* dialog displays again.
- Click the *Remove Boundary Entities* button ( ).
  The *Hatch and Gradient* dialog closes.
  You are prompted: Select entities:

- Select the current boundary entities to be removed from the selection.
- Right click or press Enter to stop selecting entities.
  The *Hatch and Gradient* dialog displays again.

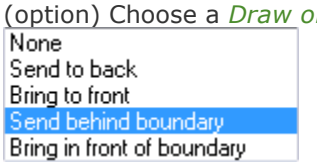
7. Define the Hatch Origin. The options are:

- *Use current origin*: The origin as defined by the HPORIGIN system variable is reused.
- *Specified origin*: Click the *Pick new origin* button (), then pick a point in the drawing. The new origin is stored in the HPORIGIN system variable.

NOTE If a selection set was active when you launch the *Hatch* tool, this selection will be used as the *Boundary Set* if you click the *New* button in step 11 in the above procedure. In this case the *Hatch and Gradient* dialog closes and immediately reopens.

Set the Hatch and Gradient options

1. (option) Select the *Annotative* option to create an annotative hatch.
2. (option) Select the *Associative* option to associate the hatch with its boundary.
3. (option) Select the *Create Separate Hatches* option.
   In case multiple boundaries are detected or multiple entities are selected in step 9, separate hatches are created instead of a single hatch.
4. (option) Choose a *Draw order* option.



   This option is reset to *Send behind boundary* when the drawing is closed and reopened.

Editing Hatches and Gradient Fills

If you select a hatch or gradient fill, its properties display in the Properties bar.

To edit a hatch or gradient fill in the properties bar

1. Select the hatch or gradient fill in the drawing.

The properties of the selected entity display in the *Properties* bar:

General	
Color	☐ ByLayer
Layer	0
Linetype	——— ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	——— ByLayer
Transparency	ByLayer
Hyperlink	
Handle	83
3D Visualization	
Material	ByLayer
Pattern	
Type	Predefined/ANSI31
Pattern name	ANSI31
Annotative	No
Angle	0
Scale	1
Origin point	0, 0
X	0
Y	0
Spacing	1
Double	No
Associative	Yes
Island detection style	Nested
Background color	■ ByBlock
Misc	
Elevation	0 mm
Area	54.05 cm²
Geometry	
UCS Elevation	0 mm, 0 mm
Minimum	0 mm
Maximum	0 mm
Slope	0

Hatch properties

General	
Color	ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Handle	8A
3D Visualization	
Material	ByLayer
Pattern	
Gradient name	Linear
Annotative	No
Origin point	0, 0
X	0
Y	0
Associative	Yes
Island detection style	Nested
Color 1	RGB:0,0,255
Color 2	RGB:255,255,255
Gradient angle	0
Centered	Yes
Misc	
Elevation	0 mm
Area	54.05 cm²
Geometry	
UCS Elevation	0 mm, 0 mm
Minimum	0 mm
Maximum	0 mm
Slope	0

Gradient fill properties.

2. Select the property you want to modify in the *Properties* bar.
   The selected property field is activated.
3. Modify the selected property
4. Do one of the following:

- Repeat steps 2 and 3 to modify another property.
- Press the Esc key to stop editing the hatch.

NOTES

- If you select multiple hatches you can you edit them simultaneously. Properties which are different display as **varies** in the BricsCAD Properties bar. If you edit such property, all edited hatches will then share this property.
- Linetype, Linetype scale and Lineweight apply to *User Defined* pattern types only .

Editing Hatches and Gradients in the Properties Bar

Commands: HATCHEDIT and -HATCHEDIT

If you select a hatch or gradient fill, its properties display in the Properties bar.

1. Select the hatch or gradient fill in the drawing.

The properties of the selected entity display in the *Properties* bar:

General	
Color	☐ ByLayer
Layer	0
Linetype	——— ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	——— ByLayer
Transparency	ByLayer
Hyperlink	
Handle	83
3D Visualization	
Material	ByLayer
Pattern	
Type	Predefined/ANSI31
Pattern name	ANSI31
Annotative	No
Angle	0
Scale	1
Origin point	0, 0
X	0
Y	0
Spacing	1
Double	No
Associative	Yes
Island detection style	Nested
Background color	■ ByBlock
Misc	
Elevation	0 mm
Area	54.05 cm²
Geometry	
UCS Elevation	0 mm, 0 mm
Minimum	0 mm
Maximum	0 mm
Slope	0

Hatch properties

General	
Color	ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Handle	8A
3D Visualization	
Material	ByLayer
Pattern	
Gradient name	Linear
Annotative	No
Origin point	0, 0
X	0
Y	0
Associative	Yes
Island detection style	Nested
Color 1	RGB:0,0,255
Color 2	RGB:255,255,255
Gradient angle	0
Centered	Yes
Misc	
Elevation	0 mm
Area	54.05 cm²
Geometry	
UCS Elevation	0 mm, 0 mm
Minimum	0 mm
Maximum	0 mm
Slope	0

Gradient fill properties.

2. Select the property you want to modify in the *Properties* bar.
   The selected property field is activated.
3. Modify the selected property
4. Do one of the following:

- Repeat steps 2 and 3 to modify another property.
- Press the Esc key to stop editing the hatch.

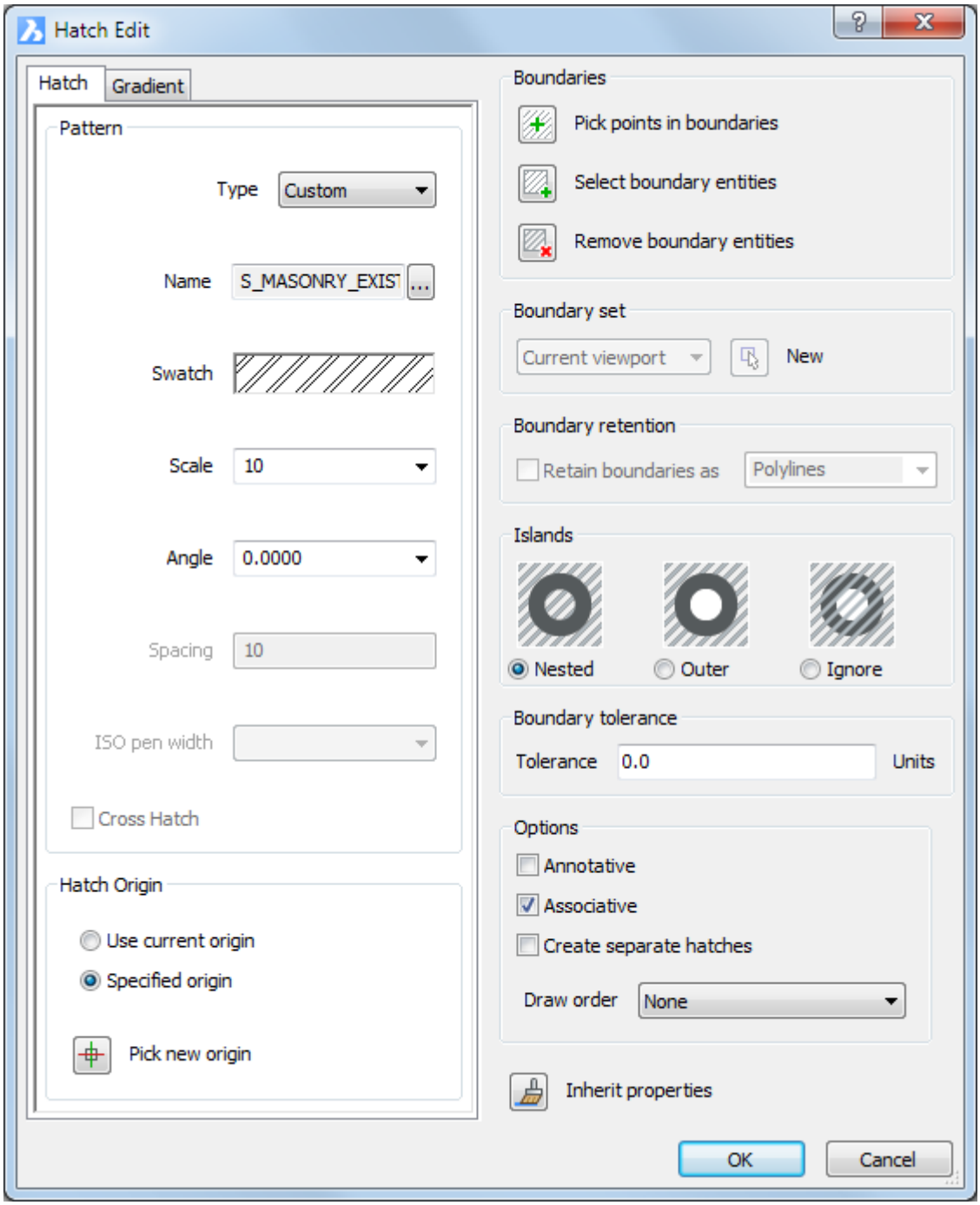
NOTES

- If you select multiple hatches you can you edit them simultaneously. Properties which are different display as **varies** in the BricsCAD Properties bar. If you edit such property, all edited hatches will then share this property.
- Linetype, Linetype scale and Lineweight apply to *User Defined* pattern types only .

To edit a hatch in a dialog box

1. Do one of the following:
   - Double click a hatch or gradient fill entity.
   - Type *hatchedit* at the command prompt, then select a hatch or gradient fill entity.

The Hatch Edit dialog displays:

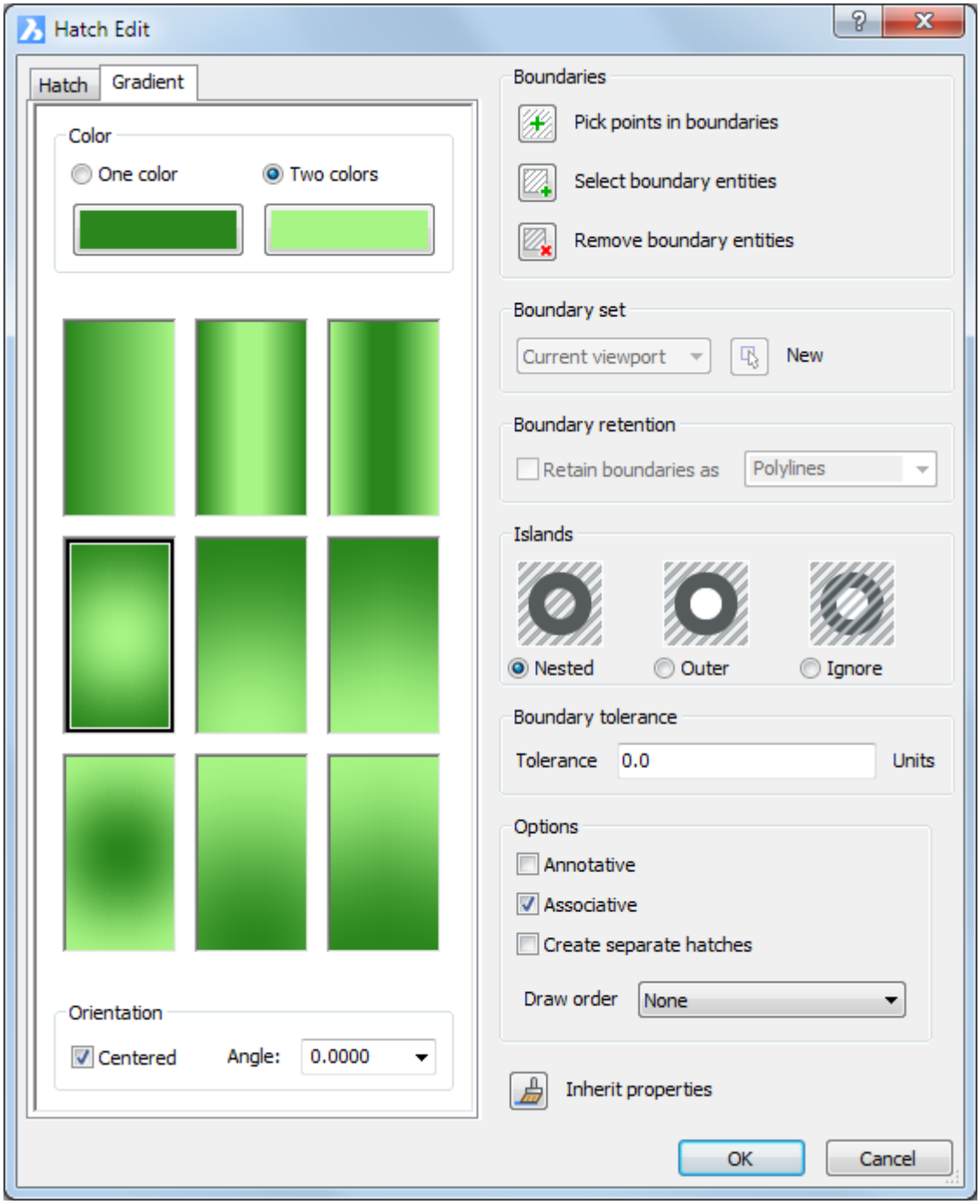


2. (option) Select the *Gradient* tab to turn the Hatch into a Gradient fill.
3. Edit the Pattern settings.
4. Edit the Boundary settings.
5. Click the *OK* button to apply the changes.

To edit a gradient fill in a dialog box

1. Do one of the following:
   - Double click a hatch or gradient fill entity.
   - Type *hatchedit* at the command prompt, then select a hatch or gradient fill entity.

The Hatch Edit dialog displays:



2. (option) Select the *Hatch* tab to turn the Gradient fill into a Hatch .
3. Edit the Color settings.
4. Edit the Boundary settings.
5. Click the *OK* button to apply the changes.

To edit a hatch or gradient fill using the Quad

1. If necessary, hit the F12 function key to activate the Quad cursor menu.
2. Pause the cursor of the hatch or gradient fill.
   The Quad displays.
3. Move the cursor to the Quad and expand the *2D Editing* command group.

4. Do one of the following:
   - Click the *Add Loop* button ( ).
     You are prompted: Specify internal point or [Select entities]:
     Click inside a closed area,
     or
     Type *S*, then press Enter to select entities.

Right click or press Enter to conclude.

   - Click the *Remove Loop* button ( ).
     The boundaries of the selected hatch highlight.
     You are prompted: Select entities:
     Click the boundaries you want to remove.

Right click or press Enter to conclude.

   - Click the *Associate* button ( ).
     You are prompted: Specify internal point or [Select entities]:
     You are prompted: Specify internal point or [Select entities]:
     Click inside a closed area,
     or
     Type *S*, then press Enter to select entities.

Right click or press Enter to conclude.

   - Click the *Generate Boundary* button ( ).
     The boundary of the hatch is generated.
   - Click the *Send all hatches to back* button ( ).
     You are prompted: # hatch entity(ies) sent to back.

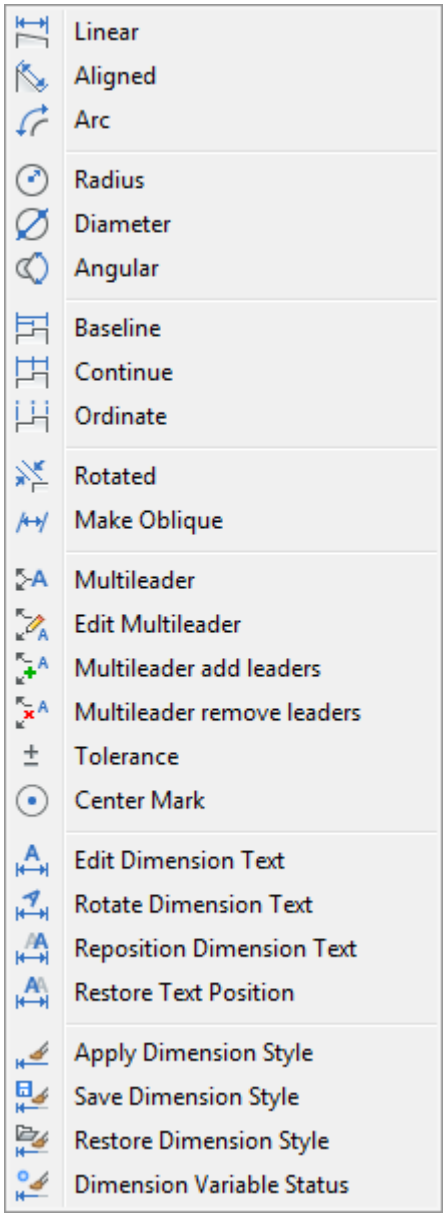
NOTE To repeat a Quad command, right click when the Quad displays.

Dimensions

All tools to create dimension entities can be found either on the *Dimensions* toolbar or in the *Dimensions* menu.

Toolbar

Menu

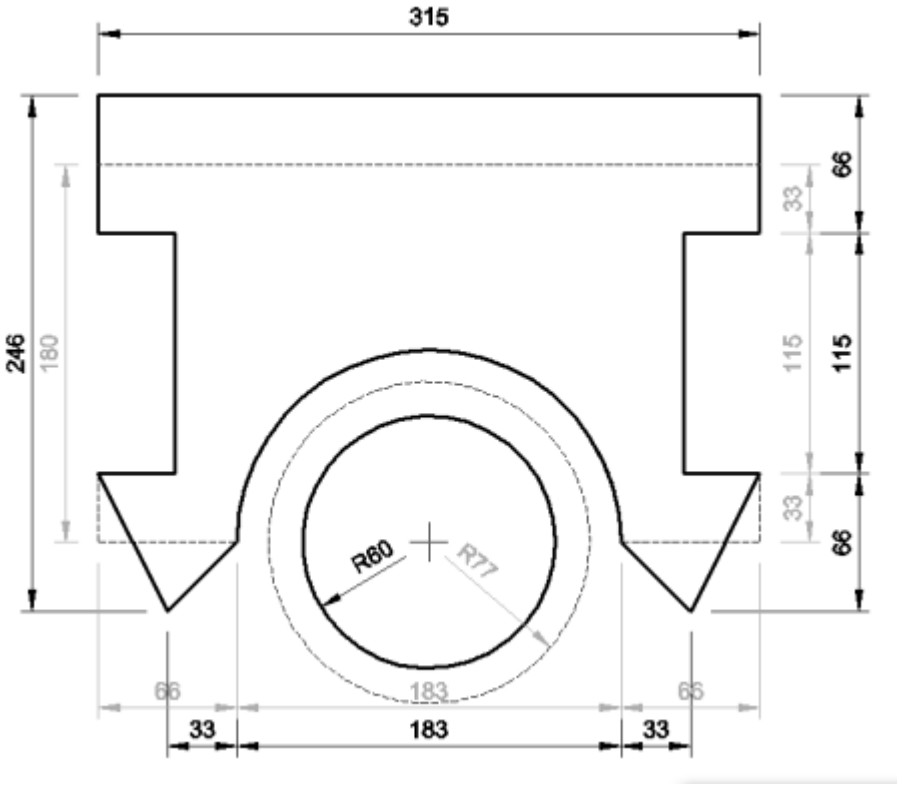


Icon	Tool name	Command	Description
	Linear	DIMLINEAR	Creates horizontal or vertical linear dimensions.
	Aligned	DIMALIGNED	Creates linear dimensions which are aligned with the dimension points.
	Arc	DIMARC	Creates arc length dimensions
	Radius	DIMRADIUS	Creates radial dimensions in circles and circular arcs.
	Diameter	DIMDIAMETER	Creates diametrical dimensions in circles and circular arcs.
	Angular	DIMANGULAR	Creates angular dimensions
	Baseline	DIMBASELINE	Creates stacked linear dimensions.
	Continue	DIMCONTINUE	Creates a chain of linear dimensions, starting from an existing linear dimension entity.
	Ordinate	DIMORDINATE	Creates ordinate dimensions.
	Rotated	DIMLINEAR	Creates rotated linear dimensions.
	Make Oblique	DIMEDIT + O	Makes the extension lines of the selected dimension(s) oblique.
	Multileader	MLEADER	Creates Mleaders (multileaders) using a multileader style.
	Edit Multileader	MLEADEREDIT	Adds leader lines to and removes leader lines from a multileader entity.
	Multileader add leaders	AIMLEADEREDITADD	Adds leader lines to a multileader.
	Multileader remove leaders	AIMLEADEREDITREMOVE	Removes leader lines from a multileader.
	Tolerance	TOLERANCE	Defines the layout of geometric tolerances.
	Center Lines	DIMCENTER	Draws center lines in circles and circular arcs.
	Edit Dimension Text	DIMEDIT+ E	Defines the dimension text of the selected dimension(s).
	Rotate Dimension Text	DIMEDIT+ R	Rotates the dimension text of the selected dimension(s).
	Reposition Dimension Text	DIMTEDIT	Lets you reposition the dimension text.
	Restore Text Position	DIMEDIT	Undoes the rotation or repositioning of the dimension text of the selected dimension(s).
	Apply Style	-DIMSTYLE + A	Applies the current dimension style to the selected dimension(s).
	Save Style	-DIMSTYLE + S	Saves the current dimension settings

			in a new dimension style.
	Restore Style	-DIMSTYLE	Sets the current dimension style.
	Dimension Variable Status	-DIMSTYLE + ST	Displays the status of the dimension variables in the command bar and the Prompt History window.
	Leader	DIMLEADER	Creates a leader with or without text.
	Quick Leader	QLEADER	Draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.

Associative Dimensions

Dimensions created in BricsCAD are associative: if the dimensioned entity is moved or modified, the dimensions will update automatically. The associative property of dimensions also applies to dimensions which are placed in paperspace and are associated with model space entities. If necessary, you can use the DIMREGEN command to update all associative dimensions.



The following Entity Snaps can be used to create associative dimensions:

endpoint, midpoint, center, perpendicular, quadrant, insertion, point, intersection and apparent intersection.

Dimension Styles Explorer

Command: DIMSTYLE

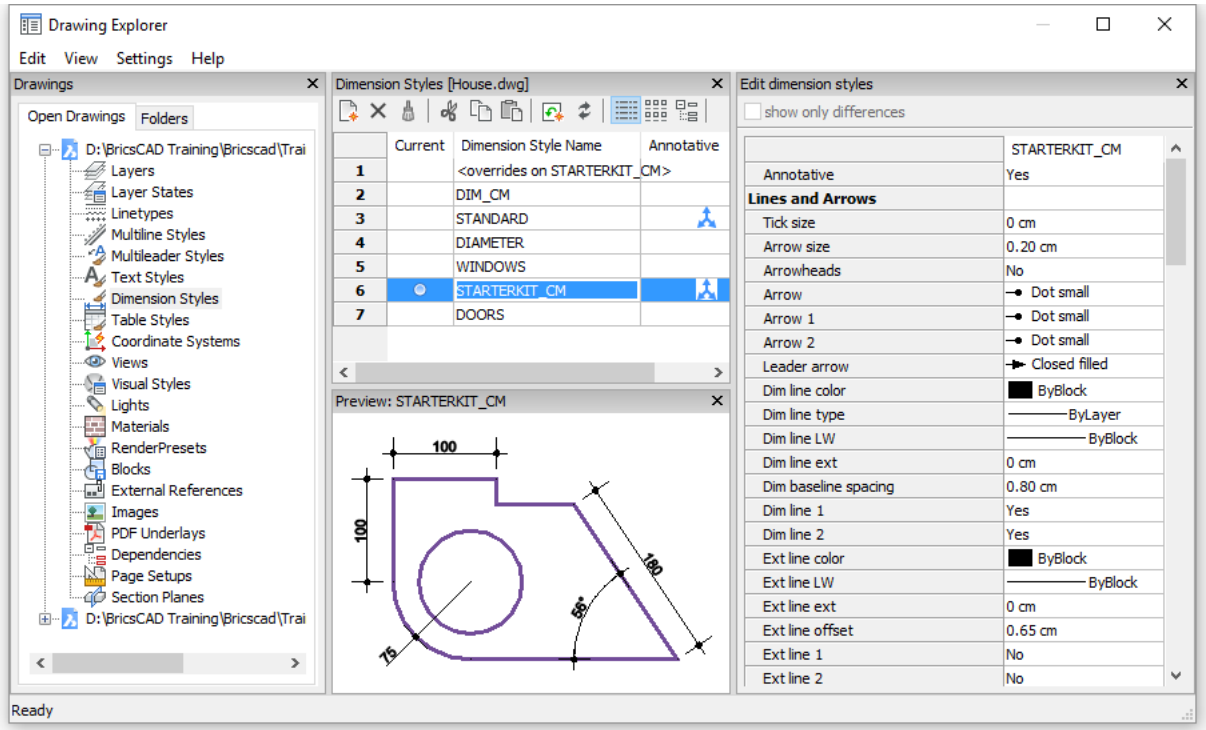
A dimension style is a collection of settings and definitions which determine the type of arrowheads, text style, color of text, lines and arrowheads, unit type, precision, etc. Dimension styles are saved in the drawing. You can create, edit and delete dimension styles. Dimension styles can be copied between drawings. Dimension styles which have the *Annotative* property create annotative dimensions.

NOTES

- You cannot delete the current dimension style.
- You cannot delete a dimension style that is used in the drawing.
- Each drawing contains at least one dimension style.

Exploring dimension styles

1. To open the *Dimension Styles Explorer*, do one of the following:
   - Choose *Dimension Styles ...* in the *Tools > Drawing Explorer* menu.
   - Double click the *Dimension Style* field in the Status bar.
   - Right click the *Dimension Style* field in the Status bar, then choose *Properties* in the context menu.
   - Type *dimstyle* in the command bar, then press Enter.



The current dimension style is marked in the *Current* column.

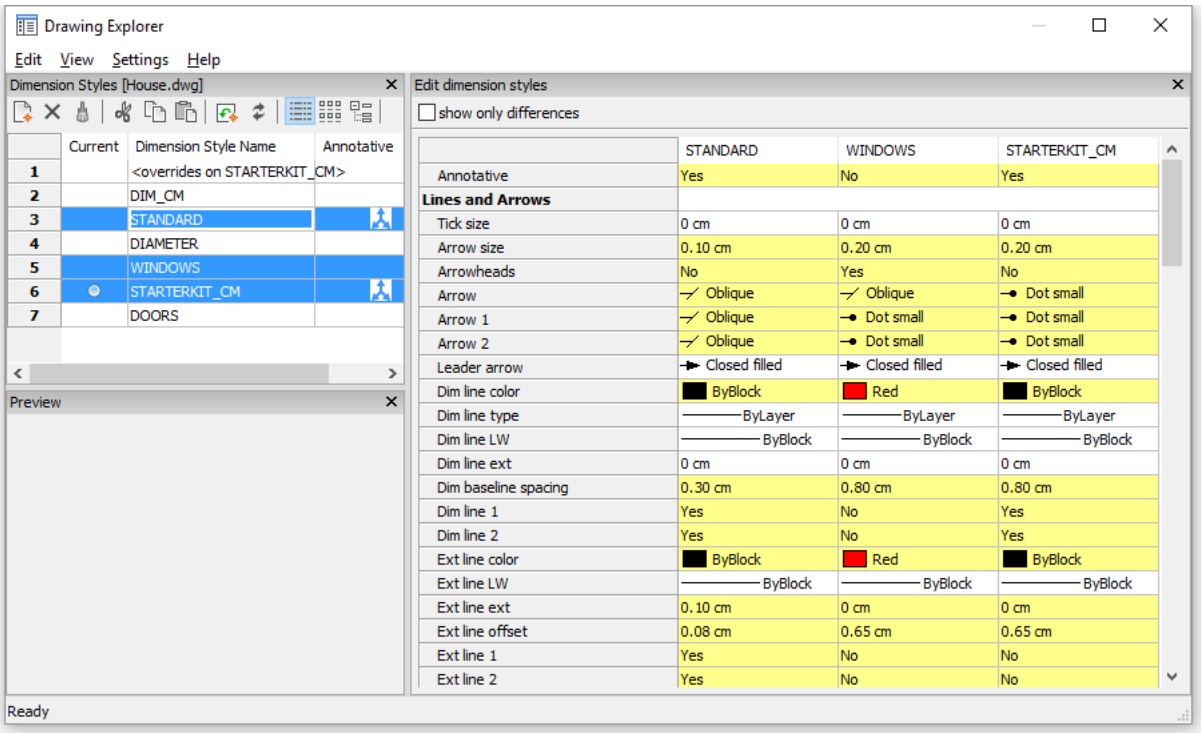
2. (option) Select a dimension style to see a preview.
3. (option) Select a dimension style then right click:

   - Choose *New* to create a new dimension style.
   - Choose *Delete* to delete the selected dimension style.
   - Choose *Cut* selected dimension style in the current drawing, then paste it in another open drawing.
   - Choose *Copy*, then paste it in another open drawing.
   - Choose *Paste*, to copy a previously cut or copied dimension style in the current drawing.
   - Choose *Rename*, to rename of the selected dimension style.
   - Choose *Set current* in the context menu to make it the current dimension style.
   - Choose *Save overrides to current style*, to save all current overrides to the current style.
   - Choose *Save to new style,* to create a new dimension style as a copy of the selected dimension style.

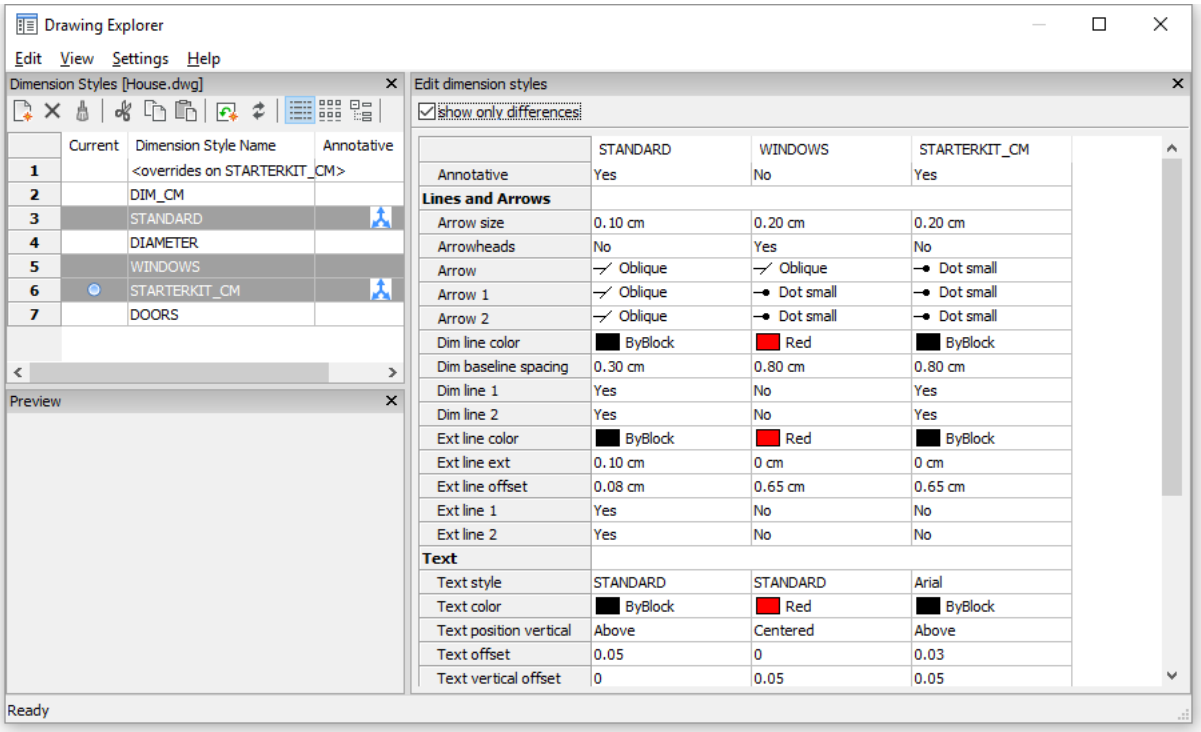
4. (option) Modify a dimension style definition.
5. (option) Select *<overrides>* to define override settings for the current dimension style.

Comparing Dimstyles

1. Open the Drawing Explorer > Dimension Styles dialog.
2. Select 2 or more dimension styles.
   Press and hold the Ctrl-key to add more dimension style to the selection set.
   The properties of the selected styles display in the Editor pane. Properties that are different in one of the selected styles display on a yellow background.

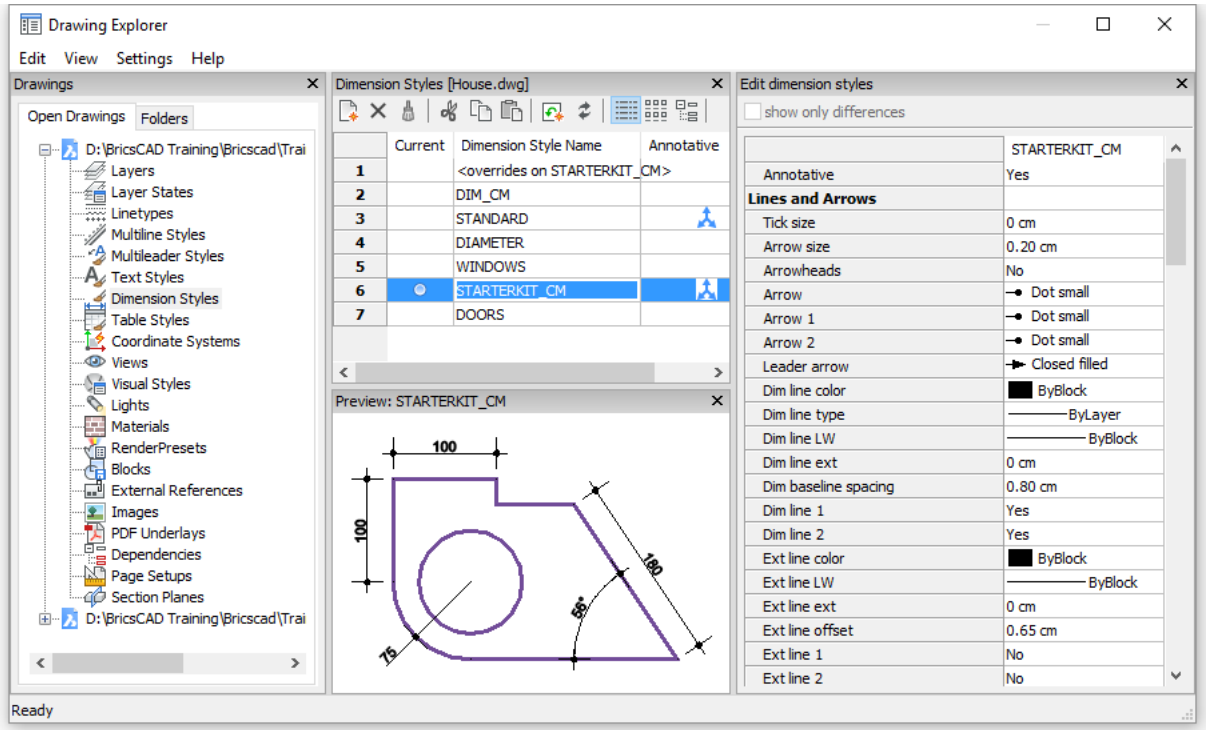


3. Check the *Show only differences* option to display the properties that are different for at least one of the selected dimension styles only.



Exploring Dimension Styles

1. To open the *Dimension Styles Explorer*, do one of the following:
   - Choose *Dimension Styles ...* in the *Tools > Drawing Explorer* menu.
   - Double click the *Dimension Style* field in the Status bar.
   - Right click the *Dimension Style* field in the Status bar, then choose *Properties* in the context menu.
   - Type *dimstyle* in the command bar, then press Enter.



The current dimension style is marked in the *Current* column.

2. (option) Select a dimension style to see a preview.
3. (option) Select a dimension style then right click:

   - Choose *New* to create a new dimension style.
   - Choose *Delete* to delete the selected dimension style.
   - Choose *Cut* selected dimension style in the current drawing, then paste it in another open drawing.
   - Choose *Copy*, then paste it in another open drawing.
   - Choose *Paste*, to copy a previously cut or copied dimension style in the current drawing.
   - Choose *Rename*, to rename of the selected dimension style.
   - Choose *Set current* in the context menu to make it the current dimension style.
   - Choose *Save overrides to current style*, to save all current overrides to the current style.
   - Choose *Save to new style,* to create a new dimension style as a copy of the selected dimension style.

4. (option) Modify a dimension style definition.
5. (option) Select *<overrides>* to define override settings for the current dimension style.

Dimension Settings

A dimension style consists of six settings groups

- Lines and Arrows
- Text
- Fit
- Primary Units
- Alternate Units
- Tolerances

To list the current status of the dimension variables

1. Do one of the following:
   - Click the *Dimension Variable Status* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Dimension Variable Status* in the *Dimensions* menu.

A list of all dimension variables displays in the command bar.

2. Press the *F2* function key to open the *BricsCAD Prompt History* window.
3. Press Enter to continue the listing.
4. Press the *F2* function key to close the *BricsCAD Prompt History* window.

Dimension Styles

In the Dimension Styles explorer you can:

- Create a new dimension style
- Edit a dimension style
- Define dimension style overrides
- Rename a dimension style
- Set a dimension style current
- Copy dimension styles between drawings
- Move dimension styles between drawings
- Delete a dimension style

To Create a New Dimension Style

1. Open the Dimension Styles explorer dialog.
2. Do one of the following:
   - Click the *New* button ( ).
   - Place the cursor on the dimension style list, then right click and choose *New* in the context menu.

A *New Dimension Style* is added.

3. Select the *Name* field and type a name for the new dimension style.
4. Define the settings for the new dimension style.

NOTE

- The *New* tool creates a new dimension style as a copy of the current dimension style.
- Use the *Save to new style* option of the context menu to create a new dimension style as a copy of the selected dimension style.
- Use the *Save overrides to current style* option of the context menu to create a new dimension style from the current override settings. This option is available if you select *<overrides>* only.
- Alternatively you can use the Save Style tool to create a new dimension style.

To create a dimension child style

For each dimension style you can create a child style for each dimension type: Linear, Angular, Radius, Diameter, Ordinate and Leader.

If the current dimension style has a child style for a dimension type, the child style will be used automatically when such dimension type is created.

1. Select the parent dimension style, then right click and choose a child style in the context menu.

   The name of the child style is: <parent_style>:<child_style>

2. Select the child style and define the properties that need to be different from the parent style.

To Edit a Dimension Style

1. Open the Dimension Styles explorer dialog in *Tree view* mode ( ).
2. Expand the dimension style.
3. Adjust the dimension settings.
4. Close the *Dimension Styles explorer* dialog.

To Define Dimension Style Overrides

1. Open the Dimension Styles explorer dialog in *Tree view* mode ( ).
2. Select *<overrides>*.
3. Define the dimension override settings.
4. Close the *Dimension Styles explorer* dialog.

NOTES

- It is not recommended to create dimensions with overrides.
- Use the dimension override settings to test changes, without modifying the current dimension style definition, then use the *Save overrides to current style* or *Save to new style* option.
- Setting another style current will discard the existing dimension style overrides.

To Rename a Dimension Style

1. Open the Dimension Styles explorer dialog.
2. Select the dimension style you want to rename.
3. Right click, then choose *Rename* in the context menu.
4. Type a new name in the *Name* field, then press enter.

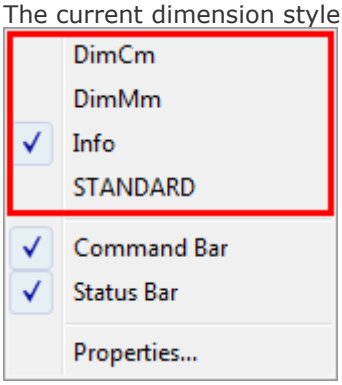
To Set a Dimension Style Current

1. Open the Dimension Styles explorer dialog.
   The current dimension style is marked in *Tree view* and *Detail view* mode.
2. Select the dimension style.
3. Right click and choose Set current in the context menu.
   In *Tree view* and *Detail view* mode a mark indicates the selected dimension style is made current.

NOTE If dimension style overrides are defined when making another dimension style current, an alert box opens saying that setting another style current will discard the existing dimension style overrides.

Using the Status Bar to Set a Dimension Style Current

1. Place the cursor on the *Dimension style* field in the Status bar.
2. Right click.
   A list of available dimension styles displays in the context menu.
   The current dimension style is checked.



3. Select the new current dimension style.
   The selected dimension style name displays in the *Dimension style* field of the *Status bar*.

NOTE Alternatively you can use the Restore Style tool to make a dimension style current.

To Copy Dimension Styles Between Drawings

1. Open the Dimension Styles explorer dialog.
2. In the source drawing, select the dimension style you want to copy.
3. Do one of the following:
   - Click the *Copy* button ( ).
   - Right click, then choose *Copy* in the context menu.

4. In the *Open Drawings* section, expand the target drawing.
5. Select *Dimension Styles* in the target drawing.
   The list of the dimension styles in the target drawing displays in the *Details* section.

6. Do one of the following:

   - Click the *Paste* button ( )
   - Place the cursor on one of the existing dimension style names, then right click and choose *Paste* in the context menu.

NOTE To select multiple dimension styles, choose *Detail view* ( ) or *Icon view* ( ) in the *Drawing Explorer* dialog.

To Move a Dimension Style Between Drawings

1. Open the Dimension Styles explorer dialog.
2. In the source drawing, select the dimension style you want to move.
3. Do one of the following:
   - Click the *Cut* button ( ).
   - Right click, then choose *Cut* in the context menu.

4. In the *Open Drawings* section, expand the target drawing.
5. Select *Dimension Styles* in the target drawing.
   The list of the dimension styles in the target drawing displays in the *Details* section.
6. Do one of the following:

   - Click the *Paste* button ( )
   - Place the cursor on one of the existing dimension style names, then right click and choose *Paste* in the context menu.

NOTE
- You cannot cut the current dimension style.
- You cannot cut a dimension style that is used in the drawing.
- To select multiple dimension styles, choose *Detail view* ( ) or *Icon view* ( ) in the *Drawing Explorer* dialog.

To Delete a Dimension Style

1. Open the Dimension Styles explorer dialog.
2. Select the dimension style you want to delete.
3. Do one of the following:
   - Click the *Delete* button ( ).
   - Right click, then choose *Delete* in the context menu.

The selected dimension style is deleted.

NOTE
- You cannot delete the current dimension style.
- You cannot delete a dimension style that is used in the drawing.
- To select multiple dimension styles, choose *Detail view* ( ) or *Icon view* ( ) in the *Drawing Explorer* dialog.

Dimension Style Tools

Command: -DIMSTYLE

Using the dimension style tools you can:

- Apply the current dimension style to a selection of dimension entities.
- Create a new dimension style.
- Restore a dimension style.
- List the current status of the dimension settings.

To apply the current dimension style

1. Do one of the following:
   - Click the *Apply Style* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Apply Style* in the *Dimensions* menu.
   - Type *-dimstyle* in the command window, press Enter then type *A* or *apply* and press Enter.

You are prompted: Select dimensions to apply the current style.

2. Select the dimensions, then press Enter.

NOTE Since the dimension style settings only apply to dimension entities, you can select all entities in step 2 to apply the current dimension style to all dimensions in the drawing.

Creating Linear Dimensions

Commands: DIMLINEAR, DIMALIGNED, DIMBASELINE, DIMCONTINUE, and DIMARC

Linear dimensions annotate linear distances or lengths and can be oriented either horizontally, vertically or aligned parallel to an existing entity or to the selected origin points.

Starting from a linear dimension, you can add a *baseline* dimension or a *continued* dimension.

- A baseline dimension inserts an additional dimension from a common first extension line origin of a previous linear dimension.
- A continued dimension continues a linear dimension from the second extension line of a previous linear dimension.

To Create a Horizontal or Vertical Linear Dimension

1. Do one of the following:
   - Click the *Linear* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Linear* in the *Dimensions* menu.
   - Type *dimlinear* in the command bar, then press Enter.

You are prompted: ENTER to select entity/<Origin of first extension line>:

2. Do one of the following:

   - Press Enter, then select the line or linear segment of a polyline you want to dimension.
   - Specify the origin of the first extension line, then specify the origin of the second extension line.

The dimension line displays dynamically.
Move the cross hairs vertically to create a horizontal dimension, or horizontally to create a vertical dimension.

3. Position the dimension line.

To Create an Aligned Linear Dimension

1. Do one of the following:
   - Click the *Aligned* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Aligned* in the *Dimensions* menu.
   - Type *dimaligned* in the command bar, then press Enter.

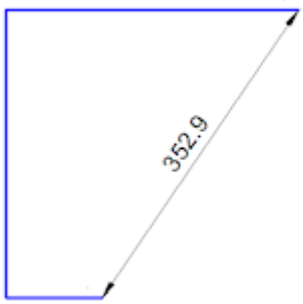
You are prompted: ENTER to select entity/<Origin of first extension line>:

2. Do one of the following:

   - Press Enter, then select the line or linear segment of a polyline you want to dimension.
   - Specify the origin of the first extension line, then specify the origin of the second extension line.

3. Do one of the following:

   - Click a point to position the dimension line.
   - Press *0* (zero) to place the dimension line at the dimension points, thus omitting the extension lines.



To Create a Rotated Linear Dimension

1. Do one of the following:
   - Click the *Rotated* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Rotated* in the *Dimensions* menu.

You are prompted: ENTER to select entity/<Origin of first extension line>:

2. Do one of the following:

   - Press Enter, then select the line or linear segment of a polyline you want to dimension.
   - Specify the origin of the first extension line, then specify the origin of the second extension line.

You are prompted: Angle of dimension line <0>:

3. Do one of the following:

   - Type the rotation angle in the command bar, then press Enter.
   - Enter the rotation angle by specifying two points.

The dimension line displays dynamically.

4. Position the dimension line.
You can place the dimension line either at the specified rotation angle or perpendicular to this direction.

To Create Baseline Dimensions

1. Do one of the following:
   - Click the *Baseline* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Baseline* in the *Dimensions* menu.
   - Type *dimbaseline* in the command bar, then press Enter.

You are prompted: Baseline: ENTER to select starting dimension/<Origin of next extension line>:

A baseline dimension displays dynamically starting from the most recently added linear or angular dimension (if any).

2. (option) Press enter, then select the starting dimension (see note below).
3. Specify the origin of the next extension line.

You are prompted: Baseline: ENTER to select starting dimension/<Origin of next extension line>:

4. Do one of the following:

   - Repeat steps 2 and 3.
   - Press the Esc key to stop creating baseline dimensions.

NOTES

- When selecting the starting dimension make sure to click the first extension line or the first half of the dimension line in order to use the first origin point of the starting dimension as the origin point for the baseline dimension.
- If you select an angular dimension, stacked angular dimensions are created.
- The distance between the stacked dimension lines is controlled by the *Dim baseline spacing* setting in the dimension style.

To Create Continued Dimensions

1. Do one of the following:
   - Click the *Continue* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Continue* in the *Dimensions* menu.
   - Type *dimcontinue* in the command bar, then press Enter.

You are prompted: Continue: ENTER to select starting dimension/<Origin of next extension line>:

A continued dimension displays dynamically starting from the most recently added linear or angular dimension (if any).

2. (option) Press enter, then select the starting dimension (see note below).
3. Specify the origin of the next extension line.

You are prompted: Continue: ENTER to select starting dimension/<Origin of next extension line>:

4. Do one of the following:

   - Repeat steps 2 and 3.
   - Press the Esc key to stop creating continued dimensions.

NOTE

- When selecting the starting dimension make sure to click the second extension line or the second half of the dimension line in order to use the second origin point of the starting dimension as the origin point for the continued dimension. Otherwise the new dimension line will partly overlap the starting dimension.
- If you select an angular dimension, continued angular dimensions are created.

To Create an Arc Length Dimension

1. Do one of the following:
   - Click the *Arc* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Arc* in the *Dimensions* menu.
   - Type *dimarc* in the command bar, then press Enter.

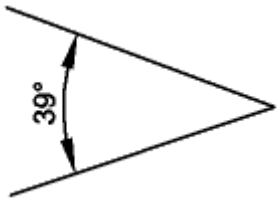
You are prompted: Select arc or polyline arc segment:

2. Select the arc or polyline arc segment.
   The arc length dimension displays dynamically.
3. Position the dimension line.

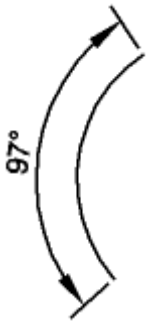
Creating Angular Dimensions

Angular dimensions annotate the angle measured between two lines, two linear segments of a polyline or the angle of a circular arc. You can also dimension an angle by selecting an angle vertex and two endpoints.

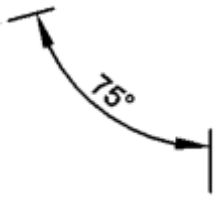
Starting from an existing angular dimension, you can add a baseline dimension or a continued dimension. An angular baseline dimension inserts an additional dimension from a common first extension line origin. An angular continued dimension continues an angular dimension from the second extension line of a previous angular dimension.



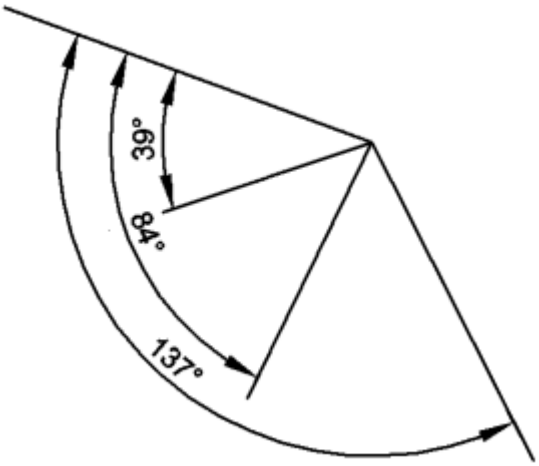
angular dimension measured between two lines



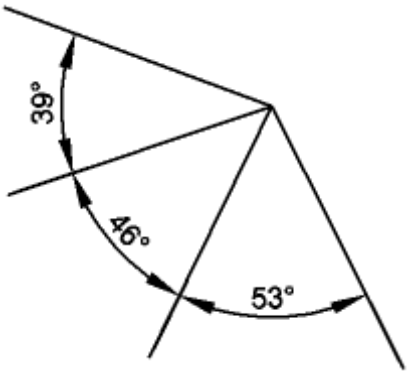
angular dimension of an arc



angular dimension defined by a vertex and two endpoints



angular baseline dimensions



angular continued dimensions

To Dimension an Angle Between Two Lines

1. Do one of the following
   - Click the *Angular* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Angular* in the *Dimensions* menu.
   - Type *dimangular* in the command bar, then press Enter.

You are prompted: Press Enter to specify angle/<select line, arc, or circle>:

2. Select the first line or line segment.
   You are prompted: Other line for angular dimension:
3. Select the second line or line segment.
   The angular dimension displays dynamically.
   You are prompted: Angle/Text/<Location of dimension arc>:
4. Do one of the following:

   - Position the dimension arc.
   - Type *A* and press Enter to define the angle of the dimension text, then position the dimension arc.
   - Type *T* and press Enter to modify the dimension text, then position the dimension arc.

To Dimension an Angle Encompassed by an Arc

1. Do one of the following
   - Click the *Angular* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Angular* in the *Dimensions* menu.
   - Type *dimangular* in the command bar, then press Enter.

You are prompted: Press Enter to specify angle/<select line, arc, or circle>:

2. Select the arc.
   The angular dimension displays dynamically.
   You are prompted: Angle/Text/<Location of dimension arc>:
3. Do one of the following:

   - Position the dimension arc.
   - Type *A* and press Enter to define the angle of the dimension text, then position the dimension arc.
   - Type *T* and press Enter to modify the dimension text, then position the dimension arc.

Creating an angular dimension defined by a vertex and two endpoints

1. Do one of the following
   - Click the *Angular* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Angular* in the *Dimensions* menu.
   - Type *dimangular* in the command bar, then press Enter.

You are prompted: Press Enter to specify angle/<select line, arc, or circle>:

2. Right click or press Enter.
   You are prompted: Vertex of angle.
3. Specify the vertex of the angle.
   You are prompted: first side of angle.
4. Specify the endpoint of the first side of the angle.
   You are prompted: Other side of angle.
5. Specify the endpoint of the second side of the angle.
   The angular dimension displays dynamically.
   You are prompted: Angle/Text/<Location of dimension arc>:
6. Do one of the following:

   - Position the dimension arc.
   - Type *A* and press Enter to define the angle of the dimension text, then position the dimension arc.
   - Type *T* and press Enter to modify the dimension text, then position the dimension arc.

Creating Diametrical and Radial Dimensions

Diametrical and radial dimensions annotate the radii and diameters of circles and circular arcs. The DIMCENTER command creates center marks in the center point of circles and circular arcs.

To Create a Diametrical Dimension

1. Do one of the following
   - Click the *Diameter* tool button ( ) on *Circles* toolbar or the *Circles* flyout of the *Dimensions* toolbar.
   - Choose *Diameter* in the *Dimensions* menu.
   - Type *dimdiameter* in the command bar, then press Enter.

You are prompted: Select arc or circle

2. Select the arc or circle you want to annotate.
   The diametrical dimension displays dynamically.
3. Position the dimension line.

To Create a Radial Dimension

1. Do one of the following
   - Click the *Radius* tool button ( ) on *Circles* toolbar or the *Circles* flyout of the *Dimensions* toolbar.
   - Choose *Radius* in the *Dimensions* menu.
   - Type *dimradius* in the command bar, then press Enter.

You are prompted: Select arc or circle

2. Select the arc or circle you want to annotate.
   The radial dimension displays dynamically.
3. Position the dimension line.

Creating Center Marks and Center Lines

1. Do one of the following
   - Click the *Center* tool button ( ) on *Circles* toolbar or the *Circles* flyout of the *Dimensions* toolbar.
   - Choose *Center* in the *Dimensions* menu.
   - Type *dimcenter* in the command bar, then press Enter.

You are prompted: Select arc or circle to dimension:

2. Select the arc or circle.
The center marks are created.

NOTE

- The *Center mark* setting of the dimension style controls whether center marks or center lines are created.
  The *Center mark size* settings of the dimension style controls the size of the center mark.
- The center mark settings are saved to the *DIMCEN* system variable.

Centerlines and Center Marks

Commands: CENTERLINE, CENTERMARK, CENTERREASSOCIATE, CENTERDISASSOCIATE, CENTERRESET

CenterLine command: creates a centerline entity associated with two selected lines.

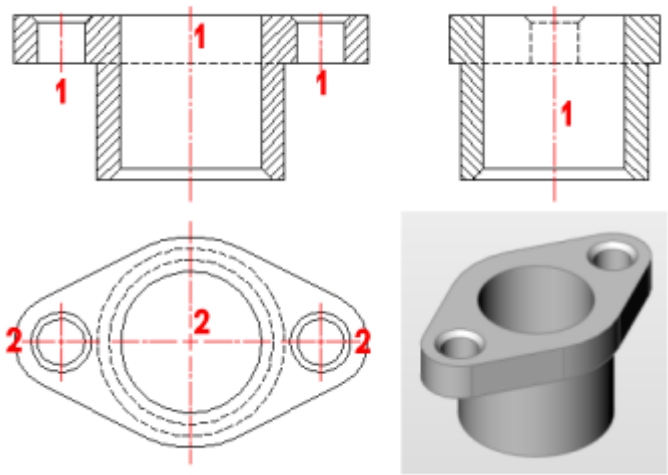
CenterMark command: creates a center mark entity associated with a selected circle or arc.

CenterReassociate command: recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc.

CenterDisassociate command: breaks the association of centerline and a center mark entities.

CenteReset command: resets centerline and center mark entities.

Centerlines and Center Marks are drawing reference entities to indicate axes of symmetry and to centers of circular openings. Centerlines and Center Marks are associative entities: when the referenced lines, circles or circular arcs are modified they are adjusted automatically.



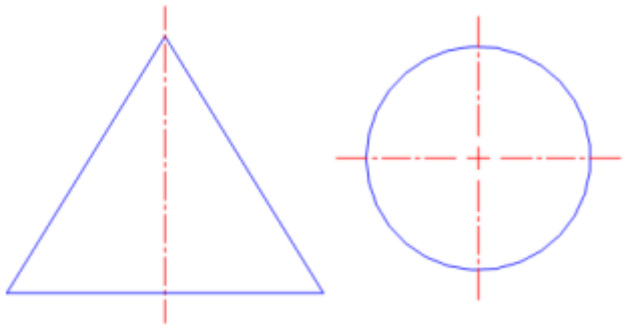
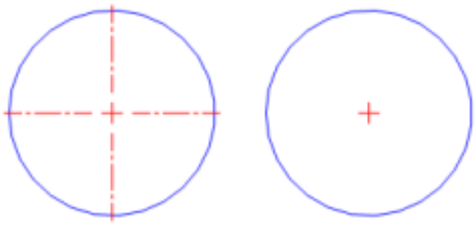
1. Centerline
2. Center Mark

System variables for center entities

The properties and geometry of centerline and center mark entities are controlled through system variables.

Center entities	
Center mark cross gap	0.05x
Center mark cross size	0.1x
Centerline extensions length	3.5
Default layer for center mark or centerline	.
Linetype file for center mark or centerline	iso.lin
Automatic extension for center mark or centerline	☑ Automatic extension

Name	Title	Description
CENTERCROSSGAP	Center mark cross gap	Defines the gap between the center mark and its centerlines. Type *centercrossgap* in the command line; prompts you: New value for CENTERCROSSGAP, or . for none/<"0.05x">: • Type a value to specify the absolute size of the gap in drawing units. • Type a value followed by x to set the size of the gap relative to the diameter of the circle or circular arc. The default value 0.05x creates a gap of 5/100 of the diameter. • . (dot) creates no gap.
CENTERCROSSSIZE	Center mark cross size	Defines the size of the associative center mark. Type *centercrosssize* in the command line; prompts you: New value for CENTERCROSSSIZE, or . for none/<"0.1x">: • Type a value to specify the absolute size of the center mark lines in drawing units. • Type a value followed by x to set the size of the center mark relative to the diameter of the circle or circular arc. The default value 0.1x creates center marks 1/10th of the diameter. • . (dot) creates center marks with zero length lines.
CENTEREXE	Centerline extensions length	Defines the length of the centerline extensions. The value is expressed in drawing units.

		
CENTERLAYER	Default layer for center mark or centerline	Specifies the default layer for new centerlines and center marks. Type *centerlayer* in the command line; prompts you: New value for CENTERLAYER, or . for none/<"">: • Type a name. If such layer not yet exists, the layer will be created when a centerline or centemark is created. • Type . (dot) to create centerlines and center marks on the current layer.
CENTERLTYPEFILE	Linetype file for center mark or centerline	Specifies the linetype file used to create centerlines and center marks. Default in imperial unit drawings: *default.lin* Default in metric unit drawings: *iso.lin*
CENTERLTYPE	Linetype for center mark or centerline	Specifies the linetype for newly created centerlines and center marks. Type *centerltype* in the command line; prompts you: New value for CENTERLTYPE, or . for none/<"CENTER2">: • Type a new linetype name or press Enter to accept the current name. • Type . (dot) for continuous lines.
CENTERLTSCALE	Linetype scale for center mark or centerline	Specifies the linetype scale for newly created centerlines and center marks. Type
CENTERMARKEXE	Automatic extension for center mark or centerline	Determines whether centerlines extend (left) or not (right) from new center marks. 

Ordinate Dimensions

Command: DIMORDINATE

An ordinate dimension annotates the perpendicular distance from an origin or base point (the origin of the current user coordinate system [UCS]). Ordinate dimensions consist of an x- or y-coordinate and a leader. An x-ordinate dimension measures distances along the x-axis, while a y-ordinate dimension measures distances along the y-axis.

As you select ordinate points, the program automatically determines whether the point is an x- or y-ordinate based on which direction you drag the second point. You can also specify whether the ordinate represents an x- or y-ordinate. Ordinate dimension text is always aligned with the ordinate leader lines, regardless of the text orientation specified by the current dimension style.

To create an ordinate dimension

1. Do one of the following:
   - Click the *Ordinate* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Ordinate* in the *Dimensions* menu.
   - Type *dimordinate* in the command bar, then press Enter.

You are prompted: Select point for ordinate dimension:

2. Specify the origin point for the ordinate dimension.
   The ordinate dimension displays dynamically.
3. Position the ordinate dimension.

NOTE Adjust the origin point of the current user coordinate system (UCS) before placing ordinate dimensions if necessary.

Leaders and Annotations

A leader consists of an arrowhead, a line, polyline or a spline and an annotation.

The general procedure to place an annotation is:

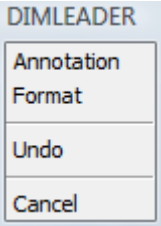
- Specify the position of the arrowhead.
- Create the leader line segments.
- Type the annotation text.

To Create a Leader and an Annotation

1. Do one of the following:
   - Click the *Leader* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Leader* in the *Dimensions* menu.
   - Type *leader* or *dimleader* in the command bar, then press Enter.

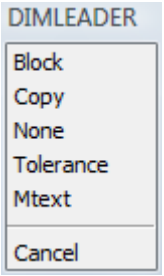
You are prompted: Start of leader

2. Specify the position of the arrowhead.
   You are prompted: Next point.
3. Specify the endpoint of the first leader segment.
   You are prompted: To point: Format/Undo/<Annotation>:
   A prompt menu displays:



4. Add more segments to the leader, then press Enter or right click to stop adding segments.
   You are prompted: First line of annotation text/<options>:
5. Do one of the following:

   - Type the first line of the annotation text, then press Enter.
     You are prompted: Next line of annotation text:
     Do one of the following:
     - Press Enter or right click to create the leader and annotation.
     - Keep adding lines to the annotation text, then press Enter or right click to create the leader and annotation.
     - Press Enter or right click to see the annotation text options.
       You are prompted: Dimension text options:
       Block/Copy/None/Tolerance/<Mtext>:
       A prompt menu displays:



   - Press the Esc key to abort the *Leader* tool.

To Create a Leader only

1. Do one of the following:
   - Click the *Leader* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Leader* in the *Dimensions* menu.
   - Type *dimleader* in the command bar, then press Enter.

You are prompted: Start of leader

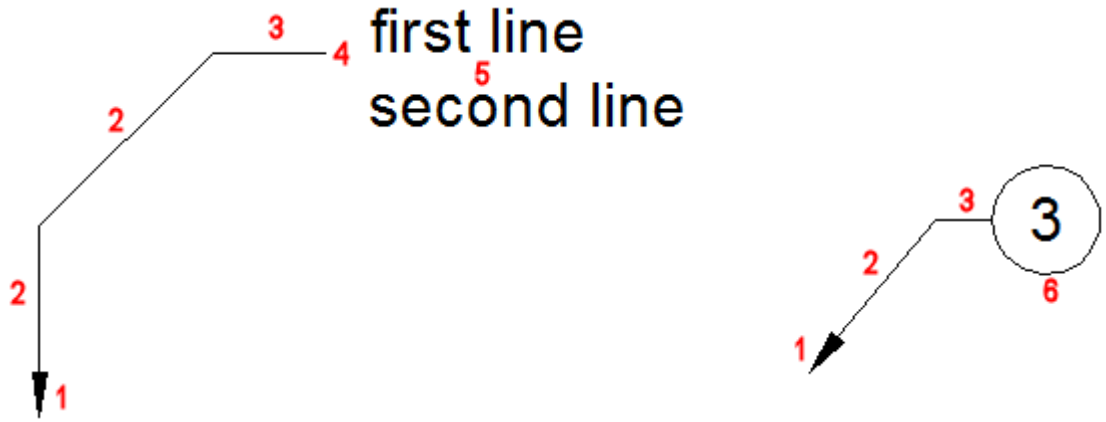
2. Specify the position of the arrowhead.
   You are prompted: Next point.
3. Specify the endpoint of the first leader segment.
   You are prompted: To point: Format/Undo/<Annotation>:
4. Add more segments to the leader, then press Enter or right click to stop adding segments.
   You are prompted: First line of annotation text/<options>:
5. Press Enter, then choose *None* in the prompt menu or type *N*.

Multileaders

Commands: MLEADER, MLEADERSTYLE, MLEADEREDIT and MLEADEREDITEXT

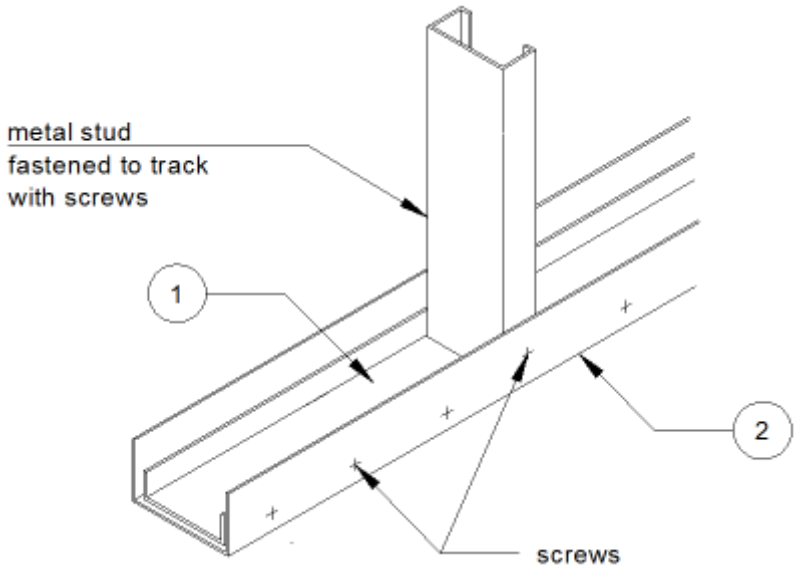
A multileader consists of:

- an arrowhead (1)
- one or more leader lines (2), which can be straight or curved
- optionally a horizontal landing (3) and landing gap (4)
- content: either a multiline text (5) or a block (6)



Multileaders are created using the current multileader style. Each drawing contains a *Default* leader style, which cannot be renamed or deleted; the *Default* leader style can be customized though. The *MLeader* command allows to define overrides on the current multileader style.

Multileaders can be created arrowhead first, landing first or content first.

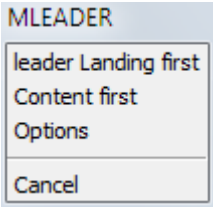


To create a multileader

1. (option) Launch the *MLeaderStyle* command, then make a multileader style current.
2. Do one of the following
   - Click the *Multileader* tool button ( ) on the *Dimensions* toolbar.
   - Choose *Multileader* in the *Dimensions* menu.
   - Type *multileader* in the command bar, then press Enter.

You are prompted: Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>:

A prompt menu displays:



3. (option) Select a placement option:

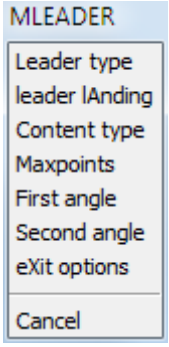
- arrowhead first
- landing first
- content first.

The most recently used placement option is repeated.

The default sequence is: arrowhead - landing - content.

4. (option) Choose *Options* in the prompt menu or type *O*, then press Enter, if you want to define overrides on the current multileader style.

You are prompted: Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/Second angle/eXit options] <eXit options>:
A prompt menu displays:



5. Specify a point to define the leader arrowhead location.
   You are prompted: Specify leader landing location:
6. Specify a point to define the leader landing location.

Depending on the content type:

- *MText*: The MText command is launched.
- *Block*: A block is inserted.
- *None*: The command is concluded.

Adding vertices to a leader line of a multileader

1. If necessary press the *F12* function key to activate the Quad cursor menu.
2. Pause the cursor over the leader line you want to add a vertex to.
   The Quad cursor menu displays, showing the most recently used command (*).
3. Move the cursor to the Quad cursor menu and expand the *2D Editing* command group.
4. Click the *Add Vertex* tool ( ).
   A new vertex is added dynamically.
5. Specify the position of the vertex.
6. (option) Pause the cursor of a leader line, then right click to add another vertex.

(*) Right click to repeat the most recently used Quad command.

Removing vertices from a leader line of a multileader

1. If necessary press the *F12* function key to activate the Quad cursor menu.
2. Pause the cursor over the leader line close to the vertex to be removed.
   The Quad cursor menu displays, showing the most recently used command (*).
3. Move the cursor to the Quad cursor menu and expand the *2D Editing* command group.
4. Click the *Remove Vertex* tool ( ).
   The vertex is removed.
5. (option) Pause the cursor of a leader line close to a vertex, then right click to remove the vertex.

Adding leader lines to a multileader

1. If necessary press the *F12* function key to activate the Quad cursor menu.
2. Pause the cursor over a leader line of the multileader entity.
   The Quad cursor menu displays, showing the most recently used command (*).
3. Move the cursor to the Quad cursor menu and expand the *2D Editing* command group.
4. Click the *Add Leader* tool ( ).
   A new leader line displays dynamically.
5. Specify the origin of the new leader line.
   Another new leader line displays dynamically.

4. Do one of the following:

   - Specify the origin of the new leader line.
   - Right click or press Enter to stop adding leader lines.

(*) Right click to repeat the most recently used Quad command.

Removing leader lines from a multileader

1. If necessary press the *F12* function key to activate the Quad cursor menu.
2. Pause the cursor over the leader line you want to remove.
   The Quad cursor menu displays, showing the most recently used command (*).
3. Move the cursor to the Quad cursor menu and expand the *2D Editing* command group.
4. Click the *Remove Leader* tool ( ).
   The leader line is removed.

(*) Right click to repeat the most recently used Quad command.

Editing dimensions

Commands:
DIMEDIT: Changes the wording and angle of dimension text; changes the angle of extension lines.
DIMTEDIT: Changes the position of dimension text.

Dimension entities can be edited in the following ways:

- use grips to edit dimensions
- edit the dimension text
- rotate dimension text
- reposition the dimension text
- restore the text position
- make the extension lines oblique

Using Grips to Edit a Linear Dimension

1. Select the dimension entity.

5 grips display:

- 2 origin point grips (1)
- 2 dimension line grips (2)
- 1 dimension text grip (3)

2. (option) Drag the origin point grips to adjust the dimension origin points.
   The dimension text is adjusted automatically.
3. (option) Drag dimension line grips to move the dimension line.
4. (option) Drag the text grip to move the dimension text.

NOTES

- Depending on the *Text movement* setting, the dimension line will move along with the text or the text will move independently. A leader will be drawn if the *Text movement* setting is set to *Move text, add leader*.
- If two adjacent dimensions (continued dimensions) are selected both, you can move their coinciding grips simultaneously.

Editing the Dimension Text

Dimension text can be edited:

- in the MText editor:
  - using the DDEDIT command
  - double clicking the dimension entity
- in the *Text override* field in the Properties Bar.
- using the *Edit Dimension Text* tool.

NOTES The following tags can be used to format the dimension text:

- type *\L* (backslash upper case L) to start underlined text, *\l* (backslash lower case L) to stop underlined text
- type *<>* to add the default dimension text
- type *[]* to add the alternate units, if alternate units is currently switched off in the dimension style.
- type *%%d* to add the degree symbol (°)
- type *%%c* to add the diameter symbol (Ø)
- type *%%p* to add the plus/minus symbol (±)

e.g. if you type *%%c\L<> cm* in the *Text override* field, dimension text displays as: **Ø24,43 cm**

To edit the dimension text in the MText editor

1. Double click the dimension entity.
   The *Text Formatting* toolbar displays.
2. Edit the dimension text.
3. Do one of the following:
   - Click the *OK* button on the *Text Formatting* toolbar.
   - Click outside the dimension text.

To edit the dimension text in the Properties Bar

1. Select the dimension entity.
2. In the Properties Bar, select the *Text override* field.
3. Type the new dimension text in the *Text override* field and press Enter.
4. Press the Esc key to stop editing the dimension entity.

To edit the dimension text using the Edit Dimension Text tool

1. Do one of the following:
   - Click the *Edit Dimension Text* tool ( ) in the *Dimensions* toolbar.
   - Choose *Edit Dimension Text* in the *Dimensions* menu.
   - Type *dimedit* in the command bar, press Enter, then type *new* and press Enter.

You are prompted: New dimension text <>:

2. Type the new dimension text and press Enter.

You are prompted: Select dimensions to replace with new text:

3. Select the dimensions and press Enter.

To Rotate the Dimension Text

1. Do one of the following:
   - Click the *Rotate Dimension Text* tool ( ) in the *Dimensions* toolbar.
   - Choose *Rotate Dimension Text* in the *Dimensions* menu.
   - Type *dimedit* in the command bar, press Enter, then type *R* and press Enter.

You are prompted: Angle of dimension text:

2. Type the rotation angle, then press Enter.

   You are prompted: Select entities to rotate text:
3. Select the dimensions and press enter.

To Reposition the Dimension Text

1. Do one of the following:
   - Click the *Reposition Dimension Text* tool ( ) in the *Dimensions* toolbar.
   - Choose *Reposition Dimension Text* in the *Dimensions* menu.
   - Type *dimtedit* in the command bar, then press Enter.

You are prompted: Select dimensions to reposition text:

2. Select the dimension.
   The dimension text moves dynamically.
3. Click to reposition the dimension text.

To Restore the Text Position

1. Do one of the following:
   - Click the *Restore Text Position* tool ( ) in the *Dimensions* toolbar.
   - Choose *Restore Text Position* in the *Dimensions* menu.
   - Type *dimedit* in the command bar, then press Enter.

You are prompted: Select dimensions to restore text to default position:

2. Select the dimensions and press enter.

To Make the Extension Lines Oblique

1. Do one of the following:
   - Click the *Make Oblique* tool ( ) in the *Dimensions* toolbar.
   - Choose *Make Oblique* in the *Dimensions* menu.
   - Type *dimedit* in the command bar, press Enter, then type *O* and press Enter.

You are prompted: Select dimensions to make oblique:

2. Select the dimensions and press enter.
   You are prompted: Enter obliquing angle:
3. Type the obliquing angle, then press Enter.

NOTE To restore non-rotated extension lines, execute the above procedure and type 0 (zero) when asked to enter the obliquing angle.

Geometric Tolerances

Geometric tolerances specify the maximum variation that is allowed in form or position from true geometry. The geometric tolerance is, in essence, the tolerance zone within which a surface or axis of hole or cylinder can lie which results in resulting feature being acceptable for proper function and interchangeability.

BricsCAD draws geometric tolerances using a feature control frame, which is a rectangle divided into compartments.

Each feature control frame consists of at least two compartments. The first compartment contains a geometric tolerance symbol that indicates the geometric characteristic to which the tolerance is applied, such as location, orientation, or form. For example, a form tolerance may indicate the flatness or roundness of a surface.

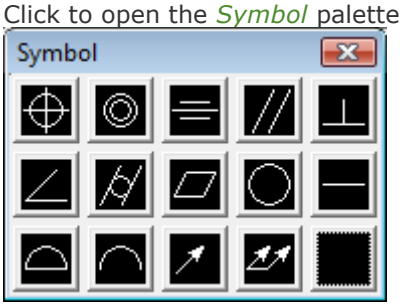
The second compartment contains the tolerance value. When appropriate, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol. The material conditions apply to features that can vary in size.

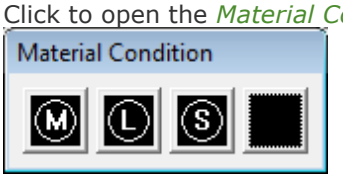
The tolerance value can then be followed by primary, secondary, and tertiary datum reference letters, along with the material conditions of each datum. Datum reference letters are generally used as reference tolerances to one of up to three perpendicular planes from which a measurement is made, although datum reference letters can also indicate an exact point or axis.

Geometric Tolerance Symbols

When two tolerances apply to the same geometry, you can also add a composite tolerance consisting of a primary tolerance value followed by a secondary tolerance value. To make a tolerance even more specific, it can also contain a projected tolerance consisting of a height value followed by a projected tolerance symbol. For example, you can use a projected tolerance to indicate the perpendicularity of an embedded part.

Geometric Tolerance Dialog

1. *Geometric Tolerance Symbol* buttons.
   Click to open the *Symbol* palette.
   

2. *Diameter Symbol* option buttons.
   Click to add the diameter symbol.
3. *Material Condition* buttons.
   Click to open the *Material Condition* palette.
   

To Create a Geometrical Tolerance Frame

1. Do one of the following
   - Click the *Tolerance* tool button (±) in the *Dimensions* toolbar.
   - Choose *Tolerance* in the *Dimensions* menu.
   - Type *tolerance* in the command bar, then press Enter.

The Geometric Tolerance dialog opens.

2. Compose the geometrical tolerance frame in the *Geometric Tolerance* dialog.
3. Click the *OK* button.
   The *Geometric Tolerance* dialog closes.
4. Click to insert the geometric tolerance frame in the drawing.

Working with Text

Text Variables

System variables can be set using the Settings dialog.
Some system variables which apply to text entities are:

Variable Name	Variable Title	Description
FONTALT	Alternate font	Defines the font which will be used if a text font cannot be found.
HIDETEXT	Hide text on hide	Specifies whether the HIDE command processes text entities.
MIRRTEXT	Mirror text	Determines whether text is mirrored by the MIRROR command or not.
MTEXTED	Multiline text editor	Sets the primary and secondary text editors to use for multiline text entities.
MTEXTFIXED	Multiline text fixed	Controls whether BricsCAD zooms, rotates and/or pans the view to fit the Mtext being edited.
QTEXTMODE	Quick text mode	Controls how text entities are displayed. If ON, displays text as a box.
TEXTANGLE	Text angle	Stores the angle of the last added text entity.
TEXTFILL	Text fill	Determines whether True Type fonts display either filled or outlined. Filled Text Outlined Text
TEXTQLTY	Text quality	Defines the smoothness of True Type fonts for plotting and rendering.
TEXTSIZE	Text size	Sets the default height for new text entities. TEXTSIZE has no effect if the current text style has a fixed height
TEXTSTYLE	text style	Stores the name of the current text style.
TSPACEFAC	Text space factor	Specifies the line spacing distance of multiline text measured as factor of the text height. Values between 0.25 and 4 are accepted.
TSPACETYPE	Text space type	Specifies the type of line spacing used for multiline text. At least: adjusts line spacing based on the tallest character(s) in a line. Exactly: used the specified line spacing, regardless of in individual character sizes.
TSTACKALIGN	Text stack align	Determines the vertical alignment of stacked text: bottom, center or top.
TSTACKSIZE	Text stack size	Specifies the percentage a which the stacked text fraction height is relative to the current height of the selected text. Values between 25 and 125 are accepted.

Working With Text Styles

When you add text to a drawing, it uses the current text style, which determines the font, size, angle, orientation and other properties of the new text entity.

Every drawing has a default text style, named *Standard*, which initially uses the *Arial* font. You cannot delete or rename the *Standard* style, but you can modify it.

If you change the *font* or *orientation* of an existing style, all existing text entities created using that style are automatically updated to reflect the change. Changing any other property has no effect on existing text.

You can create and use an unlimited number of additional text styles.

Annotative text styles create annotative text entities. In an annotative text style the *Height* defines the paper space height of the text; the model space height depends on the current annotation scale as defined by the CANNOSCALE and CANNOSCALEVALUE system variables.

Open the Text Style Explorer Dialog

Do one of the following:

- In the *Tools > Drawing Explorer* menu choose *Text Styles ...*
- Double click the *Style* field in the Status Bar.
- Type *style* in the command bar and press Enter.

The *Drawing Explorer - Text Styles* dialog opens.

Create a Text Style

1. Open the Text Style Explorer dialog.

2. Do one of the following:

   - Click the *New* tool button ( ) in the *Details* section of the *Drawing Explorer* dialog.
   - Select an existing style, then right click and choose *New* in the context menu.

A *NewStyle* text style is created, inheriting the properties of the currently selected style.

3. Type a name in the *Text Style Name* field, replacing the *NewStyle* default name.
4. Define the text style properties.
5. Close the *Text Style Explorer* dialog.

Edit a Text Style

1. Open the text style explorer dialog.
2. Select the text style.
3. Define the text style properties.
4. Close the *Text Style Explorer* dialog.

Make a Text Style Current

1. Open the text style explorer dialog.
2. Do one of the following:
   - When in *Detail view* ( ), click the blank tile in front of the text style name.
     A blue filled circle indicates the text style is set current.
   - When in *Detail view* ( ) or *Icon view* ( ), right click then choose *Set current* in the context menu.

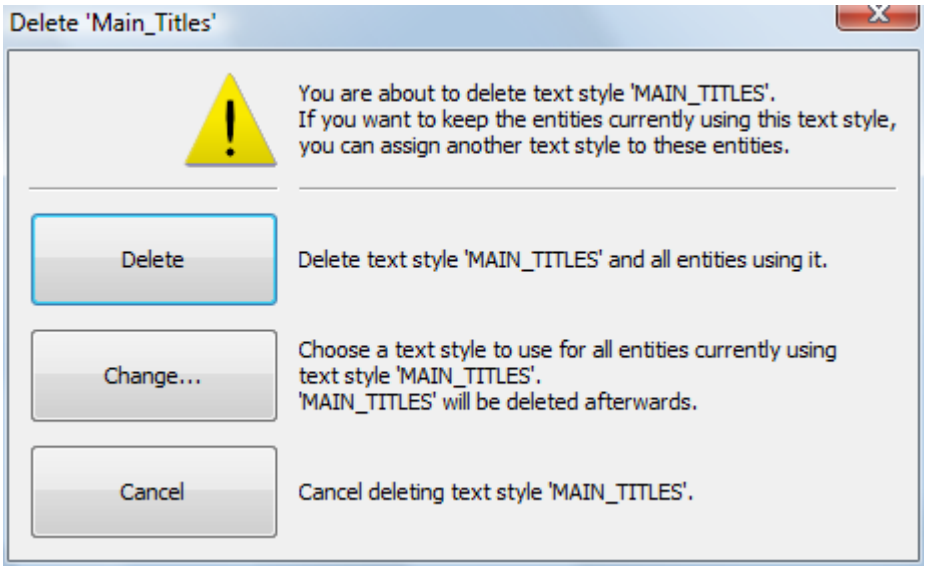
NOTE

- Alternatively you can right click the *Text style* field in the Status Bar, then choose the new current style. The current style is marked in the style list.
- When you use the *Style* option in the create text procedures, the selected style will become the current text style.

Delete a Text Style

1. Open the text style explorer dialog.
2. Select the text style.
3. Do one of the following:
   - Click the *Delete* tool button ( ) in the *Details* section of the *Drawing Explorer* dialog.
   - Right click, then choose *Delete* in the context menu.

If the text style is not used in the drawing or in a dimension style, the text style is deleted, else an alert box displays:



4. Do one of the following:

   - Click the *Delete* button to delete the text style and all entities using it.
   - Click the *Change...* button to delete the text style and assign a different text style to the entities, which currently use the selected style.
   - Click the *Cancel* button to cancel the delete procedure.

NOTE It is not possible to delete the current text style.

Creating text

The *Text* command creates single line text, while the *Mtext* command creates text entities which consist of multiple lines.

To Create Single Line Text

1. Do one of the following:
   - Click the *Text* tool button (A) on the *Draw* toolbar.
   - Choose *Text* in the *Text* submenu in the *Draw* menu.
   - Type *text* in the command bar, then press Enter.

You are prompted: Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>:

2. Specify the insertion point of the text.
   You are prompted: Height of text <current height>:
3. Do one of the following:

   - Press Enter or right click to accept the current text height.
   - Type the new text height in the command bar, then press Enter.
   - Define the new text height graphically by specifying two points in the drawing.

You are prompted: Rotation angle of text <current angle>

4. Do one of the following:

   - Press Enter or right click to accept the current angle.
   - Type the new angle in the command bar, then press Enter.
   - Define the now angle graphically by specifying two points in the drawing.

You are prompted: Text:

5. Type the text in the command bar, then press Enter.
   The text is placed in the drawing.
   You are prompted: Text:
6. Do one of the following:

   - Repeat step 5 to create another text entity.
     The new text is placed below the previous one.
   - Press Enter or right click to conclude the *Text* command.

NOTE If you have already created text, the most recent text entity highlights when you repeat the *Text* command. If you want to place the new text right below the previously created text, press Enter in step 2 of the above procedure.

To Create Multiline Text

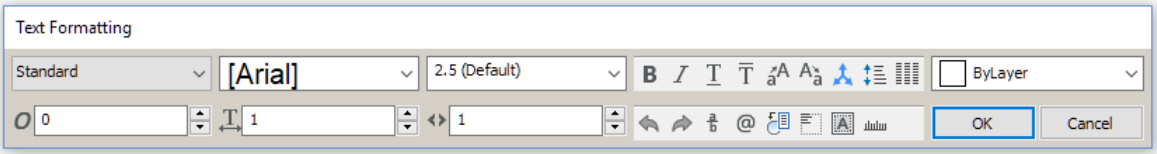
1. Do one of the following:
   - Click the *Multiline Text* tool button ( ) on the *Draw* toolbar.
   - Choose *Multiline Text* in the *Text* submenu in the *Draw* menu.
   - Type *mtext* in the command bar, then press Enter.

You are prompted: First corner for block of text:

2. Specify the first corner (insertion point of the multiline text).
   You are prompted: Justification/Rotation/Style/Height/Direction/Width/<Opposite corner for block of text>:
3. Do one of the following:

   - Specify the opposite corner of the text block to define the maximum width of the multiline text.
   - Type @ in the command bar, then press Enter for non-wrapped (zero width) text.
   - If Dynamic Dimensions are active, the height and width of the text field display dynamically.
     Type the height and width in the dynamic dimension fields. Press the TAB key to switch; press Enter to confirm.

The *Text formatting* toolbar opens right above the text block.



4. (option) Define the text properties.
5. Type your text.
6. To conclude the command and close the *Text formatting* toolbar do one of the following:

   - Click the *OK* button.
   - Press Ctrl+Enter.
   - Click outside the MText entity.

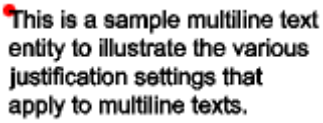
Text formatting toolbar tools and settings

Button or setting	Name	Description
Standard	Text Style	Sets the text style
[Arial]	Text Font	Sets the text font
2.5 (Default)	Text Height	Sets the text height
B	Bold	Creates bold text
I	Italic	Creates italic text
T	Underscore	Creates underscored text
T	Overscore	Creates overscored text
aA	Uppercase	Converts selected text to uppercase
Aa	Lowercase	Converts selected text to lowercase
	Annotative	Toggles the Annotative property
	Line Spacing	Sets the line spacing
	Columns	Arranges the text in columns
ByLayer	Color	Sets the color
O 0	Oblique Angle	Sets the oblique angle
T 1	Width Factor	Sets the width factor
1	Character Spacing	Sets the character spacing
	Undo	Undoes recent actions
	Redo	Redoes previously undone actions
a/b	Stacked Fraction	Creates stacked fractions
@	Special Character	Inserts special characters
	Field	Inserts a variable field
	Justify	Sets the text justification
A	Background Mask	Adds a colored background
	Ruler	Toggles the display of the ruler
OK	OK	Closes the *Text formatting* toolbar and applies the changes
Cancel	Cancel	Closes the *Text formatting* toolbar and leaves the Mtext entity unchanged

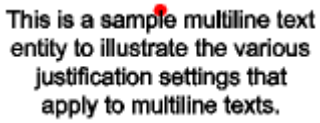
NOTE If the line length exceeds the *Width* of the multiline text, the text will be wrapped. Set the *Width* to zero for non-wrapped multiline text.

Multiline Text Justification

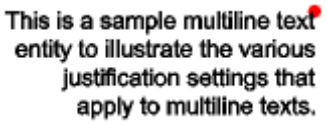
(The red dot indicates the insertion point of the multiline text entity)



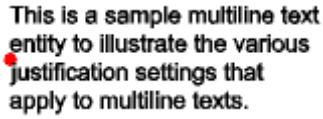
Top left



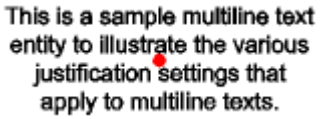
Top center



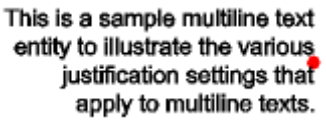
Top right



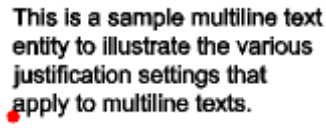
Middle left



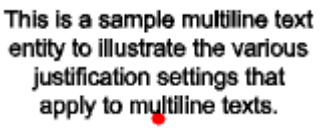
Middle center



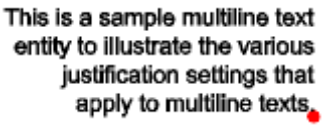
Middle right



Bottom left



Bottom center



Bottom right

Special Characters and Symbols

You can use control codes and unicode strings to include special characters in text entities.

To include control codes, as you type text, type two percent symbols (%%) followed by the special control code or character. A single percent sign is treated as a normal text character.

Special text characters

Control code	Unicode String	Result
%%d	\U+00B0	Draws a degree symbol (º)
%%p	\U+00B1	Draws a plus-or-minus symbol (±)
%%c	\U+2205	Draws a diameter symbol (Ø)

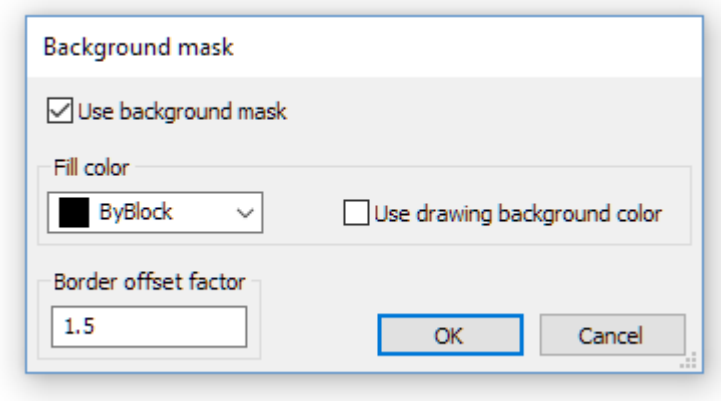
Symbols and Unicode strings

Name	Symbol (*)	Unicode String
Almost equal	≈	\U+2248
Angle	∠	\U+2220
Center line	℄	\U+2104
Delta	Δ	\U+0394
Electrical phase	ϕ	\U+0278
Identity	≡	\U+2261
Not equal	≠	\U+2260
Omega	Ω	\U+03A9
Squared	²	\U+00B2
Cubed	³	\U+00B3

(*) If supported by the text font.

To Add a Background Mask

1. When creating or editing a multiline text entity, click the *Background Mask* tool button ( ) on the *Text Formatting* toolbar.
   The *Background Mask* dialog displays:



2. Check the *Use background mask* option.
3. Do one of the following:

   - Choose a color in the *Fill color* drop-down list.
   - Choose *Select color...* in the *Fill color* drop-down list and choose an Index color or a True color in the *Select color* dialog.
   - Check the *Use drawing background color* option.

4. Define the *Border offset factor*, which defines the margin around the text for the background.
   The value is based on the text height and must be in the range 1-5. A factor of 1.0 exactly fits the multiline text entity.
5. Click *OK* to create the background mask.

Editing text

Command: DDEDIT

You can edit and modify a text entity as you would any other drawing entity. That is, you can delete, move, rotate, and scale text.

You can change the text color or layer in the Properties Bar.

To edit a multiline text, you can use the built-in text editor or choose an external program, such as Wordpad.

The MTEXTFIXED system variable controls whether the view is zoomed, rotated and/or panned to fit the edited Mtext.

The TEXTED setting controls the text edit behavior:

- 0 = in-place editor
- 1 = pop up dialog
- 2 = in-place editor with repeated input

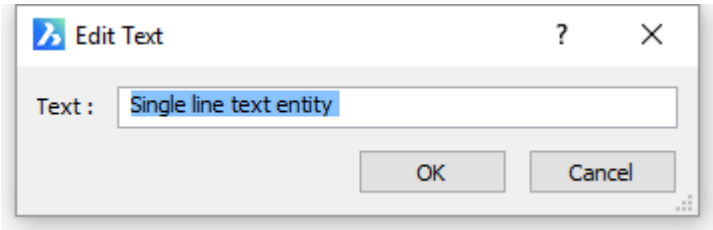
To edit a text entity

1. Do one of the following:
   - Type *ddedit* in the command bar.
     You are prompted: Select entity to modify:
   - Double click the text entity.
     Then go to step 3.

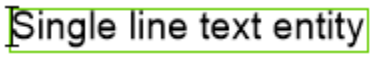
2. Do one of the following:

   - Click a single line text entity.
     Depending on the value of the TEXTED system variable:

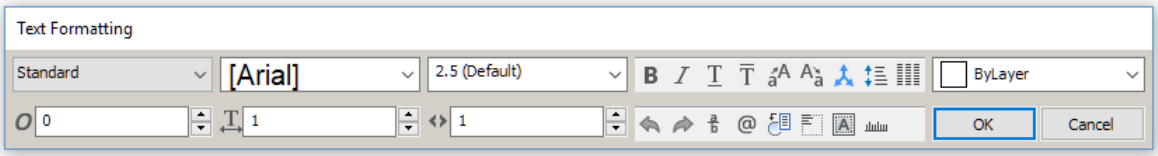
     TEXTED = 1: The *Edit Text* window opens.

     TEXTED = 0 or 2: The text is edited 'in place'.

   - Click a multiline text entity
     The *Text formatting* toolbar opens.

3. Edit the text.
4. Press the *OK* button on the *Edit Text* window or *Text Formatting* toolbar to apply the changes.

5. Do one of the following:

   - Select another text entity to edit.
   - Right click or press Enter to stop editing text entities.

To edit a selection of Mtext entities

1. Select the Mtext entities.
2. In the Properties bar select *Contents*.
3. Press the *Browse* button (...) on the right hand side of the *Contents* field

4. The *Text formatting* toolbar opens for the first Mtext entity in the selection.
5. Edit the Mtext,
6. Press the *OK* button or click in the drawing.
   The next Mtext entity in the selection can be edited now.
7. Repeat steps 5 and 6 until the last Mtext is edited.

To select a different multiline text editor

Edit the *MTEXTED* system variable under *Drawing/Drafting/Texts/Multiline Texts* in the Settings dialog.

or

1. Type *mtexted* in the command bar, then press Enter.
   You are prompted: New value for MTEXTED, or . for none/<current value>:
2. Type the file name and full path of the alternate text editor in the command bar, then press Enter.
   e.g. C:\Program Files\Windows NT\Accessories\Wordpad.EXE

NOTE To restore the built-in text editor, type a . (period) in step 2 of the above procedure, then press Enter.

Exploding text

Command: TXTEXP

The *TXTEXP* command explodes Text and Mtext entities into 2D polylines.

1. Type *txtexp* in the command bar, then press Enter.
   You are prompted: Select text to be exploded:
2. Select a single text entity or multiple text entities, then press Enter.
   The selected text(s) are exploded.

NOTE The solid fill of certain TTF fonts is removed when exploding such text, leaving the text outline only.

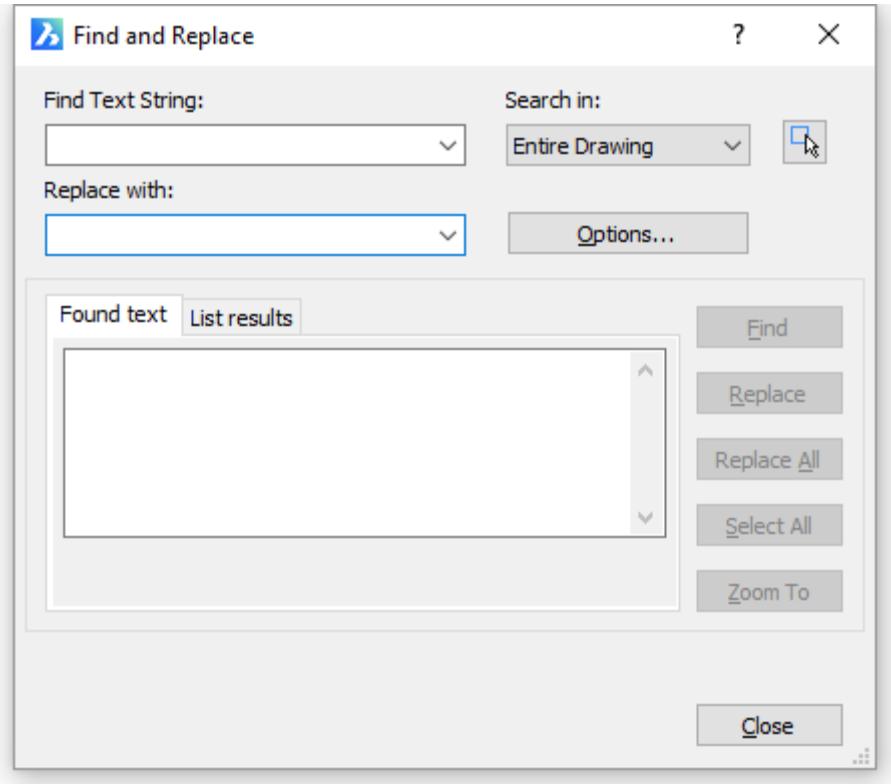
Finding and Replacing Text

The *FIND* command lets you specify the text you want to find. You can than find or replace the text. Optionally you can zoom in on the various instances of the text entities that contain the search text.

To Open the Find and Replace Dialog

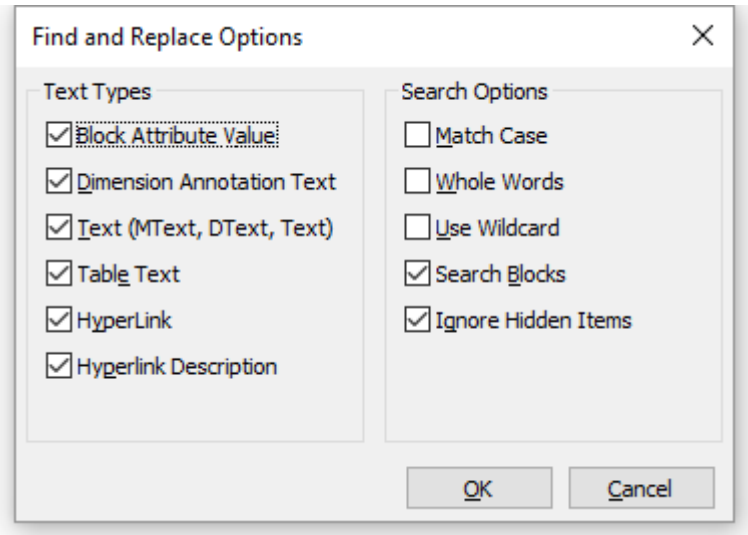
Do one of the following:

- Choose *Find...* in the *Edit* menu.
- Type *find* in the command bar, then press Enter.



To Set the Find and Replace Options

1. Open the Find and Replace dialog.
2. Click the *Options* button.



3. Set the *Find and Replace* options.
4. Click the *OK* button.

To Find Text

1. Open the Find and Replace dialog.
2. (option) Set the Find and Replace options.
3. Type the text in the *Find Text String* field.
4. (option) Click the *Select Entities* button ( ) to compose a selection set.
   - The *Find and Replace* dialog temporarily closes to let you select entities.
   - Right click to conclude the selection of entities.
     The *Search in* field reads *Current Selection* instead of *Entire Drawing*.
5. Click the *Find Next* button.
   The content of the first text entity that contains the search string displays in the *Search results* pane.
6. (option) Click the *Zoom To* button.
   The drawing is zoomed in on the text entity.
7. Do one of the following:
   - Repeat steps 7 and 8 to find more instances of the same search string.
   - Repeat steps 2 through 8 to find another text string.
   - Press the Esc key or click the *Close* button to stop.

NOTE If a selection is active when you launch the *FIND* command, only text entities in the selection set are searched.

To Replace Text

1. Open the Find and Replace dialog.
2. (option) Set the Find and Replace options.
3. Type the search text in the *Find Text String* field.
4. Type the replacement text in the *Replace with:* field.
5. (option) Click the *Select Entities* button ( ) to compose a selection set.
   - The *Find and Replace* dialog temporarily closes to let you select entities.
   - Right click to conclude the selection of entities.
     The *Search in* field reads *Current Selection* instead of *Entire Drawing*.
6. (option) Click the *Replace All* button.
   All instances of the search text are replaced by the replacement text.
7. (option) Click the *Find Next* button.
   The content of the first text entity that contains the search text displays in the *Search results* pane.
   - (option) Click the *Zoom To* button.
     The drawing is zoomed in on the text entity.
   - Do one of the following:
     - Click the *Replace* button.
     - Repeat this step to find another text entity that contains the search string.
   - (option) Repeat this step until no more occurrences of the search text are found in the drawing or selection set.

8. Do one of the following:

   - Repeat steps 2 through 7.
   - Press the Esc key or click the *Close* button to stop.

NOTE If a selection is active when you launch the *FIND* command, only text entities in the selection set are searched.

Checking Spelling

The spelling checker checks the spelling of text in the current drawing. You can check the spelling of one or more selected text entities or check the entire drawing.

During the spelling check, the *Spell* command matches the words in the drawing or the current selection set to the words in the current main dictionary and the current custom dictionary. If a word is not found in either dictionaries you can select the correct spelling in the suggestions field or you can add the word to the current custom dictionary. Custom dictionaries are used for discipline-specific words, such as medical or mechanical.

Dictionary Settings

Which dictionaries are used by the *Spell* command is defined by the *DCTMAIN* (main dictionary) and *DCTCUST* (custom dictionary) system variables.

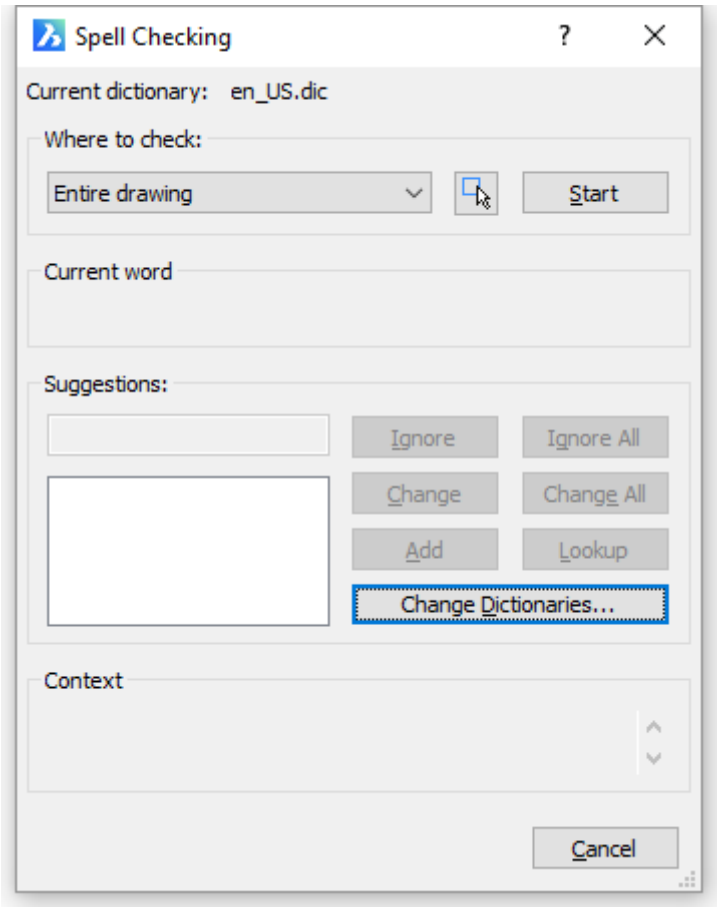
Dictionaries	
Custom spelling dictionary	C:\Users\Louis\AppData\Roaming\Bricsys\Bricscad\V9\en_US\support\Sample.cus
Main spelling dictionary	en_US.dic

NOTE We only deliver the English dictionary because of the license type on the dictionary files : the English version can be distributed freely ; all other languages are not free. However, you can download other languages and use them for free (see To add a main dictionary).

To Open the Spell Checking Dialog

Do one of the following:

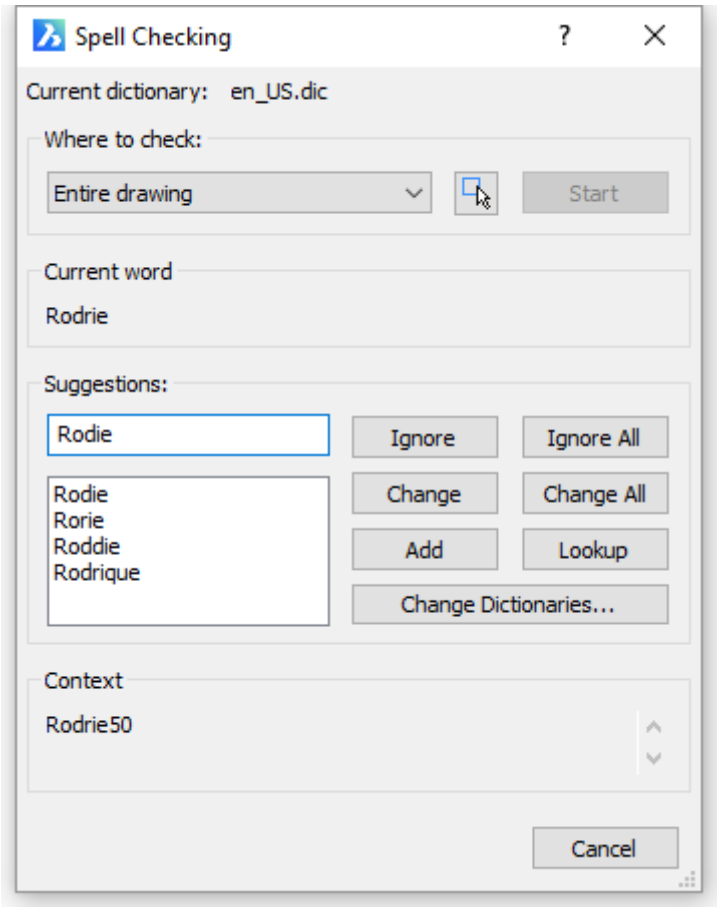
- Choose *Check Spelling* in the *Tools* menu.
- Type *spell* in the command bar, then press Enter.



To Check the Spelling

1. Open the Spell Checking dialog.
2. (option) Click the *Select* button ( ) to compose a selection set.
3. (option) Change the current dictionaries.
4. Click the *Start* button.

The first possibly misspelled word displays in the *Current word* section.
The content of the text entity in which the *Current word* is found displays in the *Context* section.



5. Do one of the following:

- Click the *Ignore* button to leave this instance of the *Current word* unchanged.
- Click the *Ignore All* button to leave all instances of the *Current word* unchanged.
- Accept the word in the *Suggestions* field or select a word in the *Suggestions* list, then click the *Change* button to replace this instance of the *Current word* by the word in the *Suggestions* field.
- Accept the word in the *Suggestions* field or select a word in the *Suggestions* list, then click the *Change All* button to replace all instances of the *Current word* by the word in the *Suggestions* field.
- Click the *Add* button to add the *Current word* to the current custom dictionary.
- Click the *Lookup* button to list words similar to the word in the *Suggestions* field.

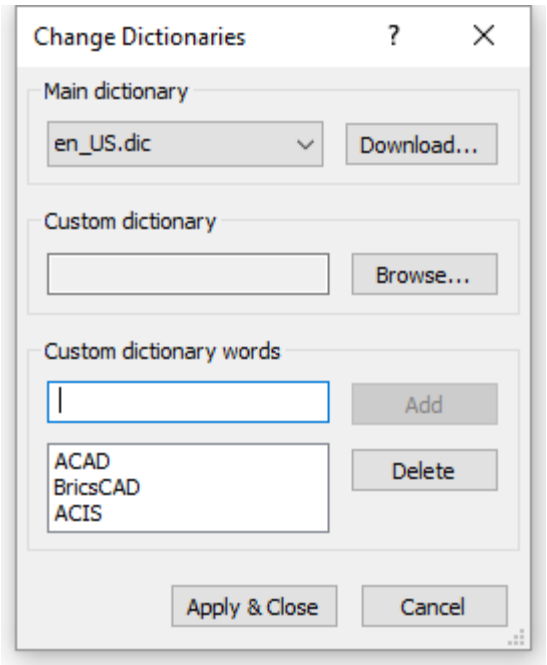
6. Repeat step 5 until the *Spelling check is complete* message displays.

NOTE If a selection is active when you launch the *Spell* command, only text entities in the selection set are checked.

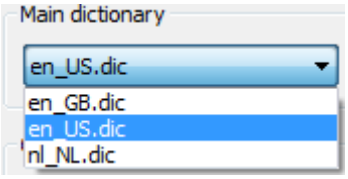
To Change Dictionaries

1. Open the Spell Checking dialog.
2. Click the *Change Dictionaries...* button.

The *Change Dictionaries* dialog opens.



3. Click the *Main dictionary* list button to change the main dictionary.



4. Select a new main dictionary.
5. (option) Click the *Browse...* button to select a new *Custom dictionary*.
6. (option) Type a word in the *Custom dictionary words* field, then click the *Add* button to add a word to the current custom dictionary.
7. Click the *Apply & Close* button.

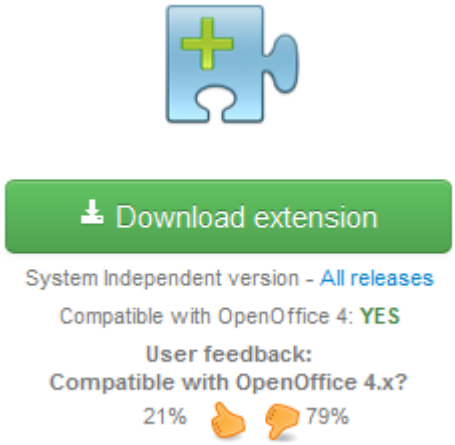
NOTE
- Custom dictionaries are best saved in the *Support* folder of the Roamable root folder.
- If you want to share the custom dictionary with MS Word, point to the MS Word custom dictionary in step 5 in the above procedure. By default, *CUSTOM.DIC* is the custom dictionary in MS Word, which can be found under *C:\Users\<user name>\AppData\Roaming\Microsoft\Proof\CUSTOM.DIC*.

To Add a New Main Dictionary

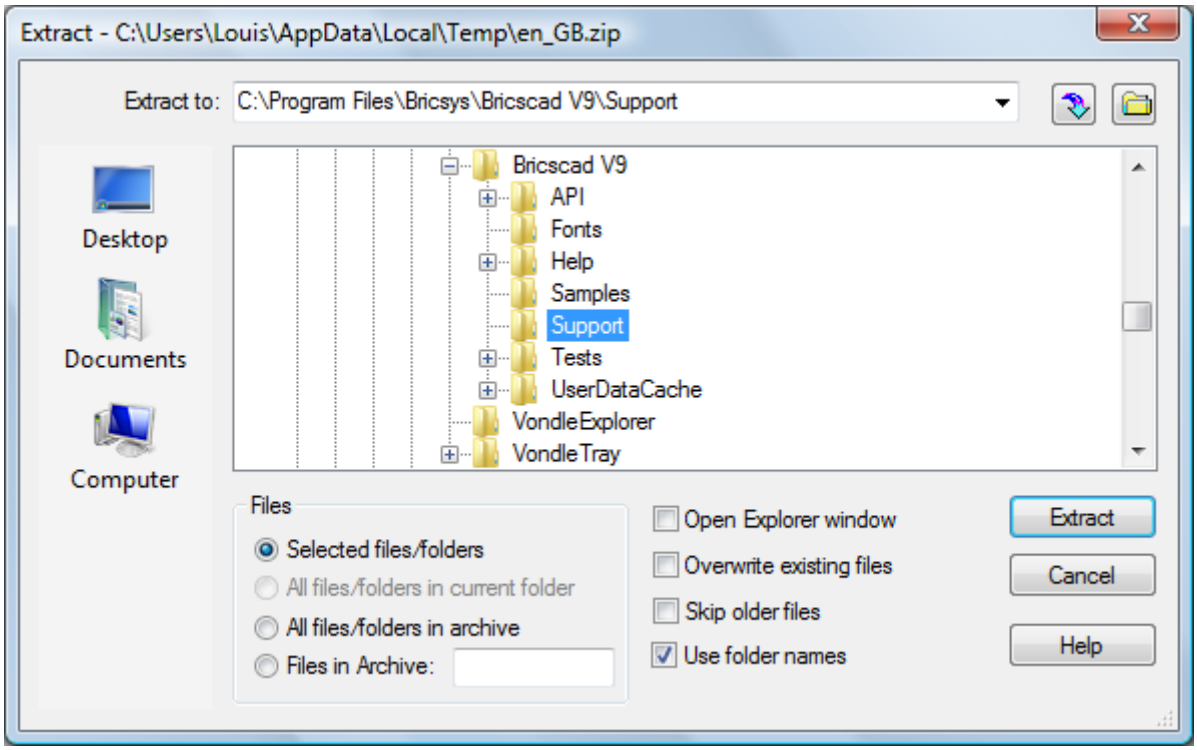
1. Open the Spell Checking dialog.
2. Click the *Change Dictionaries...* button.
3. Click the *Download...* button.

Your browser opens: *https://extensions.openoffice.org/dictionaries*

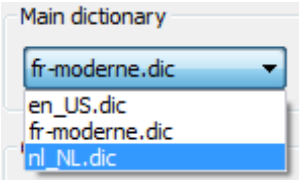
4. Scroll down to the language of your choice, then click the link to open the page of the selected language.
5. Click the *Download extension* button to start downloading the **.oxt* file.



6. Change the *oxt* extension to *zip*.
7. Extract the **.aff* and **.dic* files from the ZIP file to the folder *C:\Program Files\Bricsys\BricsCAD Vx\Support*.



The dictionary is now available in the *Main Dictionary* list on the *Change Dictionaries* dialog box.



To Add a New Custom Dictionary

1. Open a text editor, such as *Notepad*.
2. (option) Type the words you want to use in your custom dictionary.
   Each word must be on a separate line.
3. Save the file to the the *Support* folder of the Roamable root folder.
   Make sure the file extension is *.cus*.

Working with Fields

A field is a variable text that. When a field is updated it displays the latest value of the data source it refers to. Fields can be used to insert drawing properties, custom properties or object properties in multiline texts or tables or as an attribute in a block. Fields in attributes are especially useful in title blocks: using drawing, sheet set, subset and sheet properties title blocks can be filled out automatically.

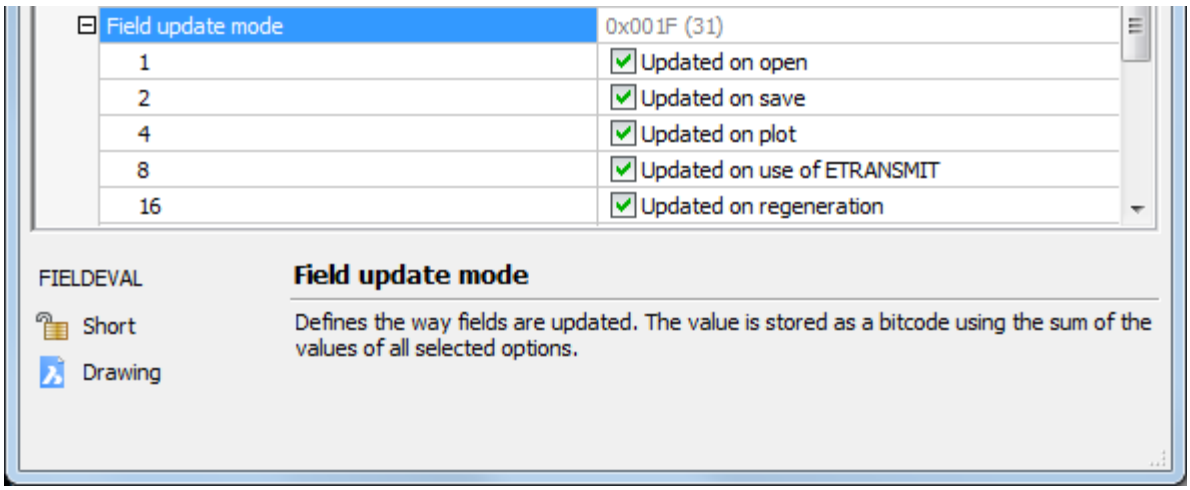
See the DWGPROPS command to learn more about default and custom drawing properties.

See the SHEETSET command to learn more about default and custom sheet set, subset and sheet properties.

If no value is available, a field displays hyphens (----).

If the FIELDDISPLAY system variable is ON, fields display on a light gray background which does not print.

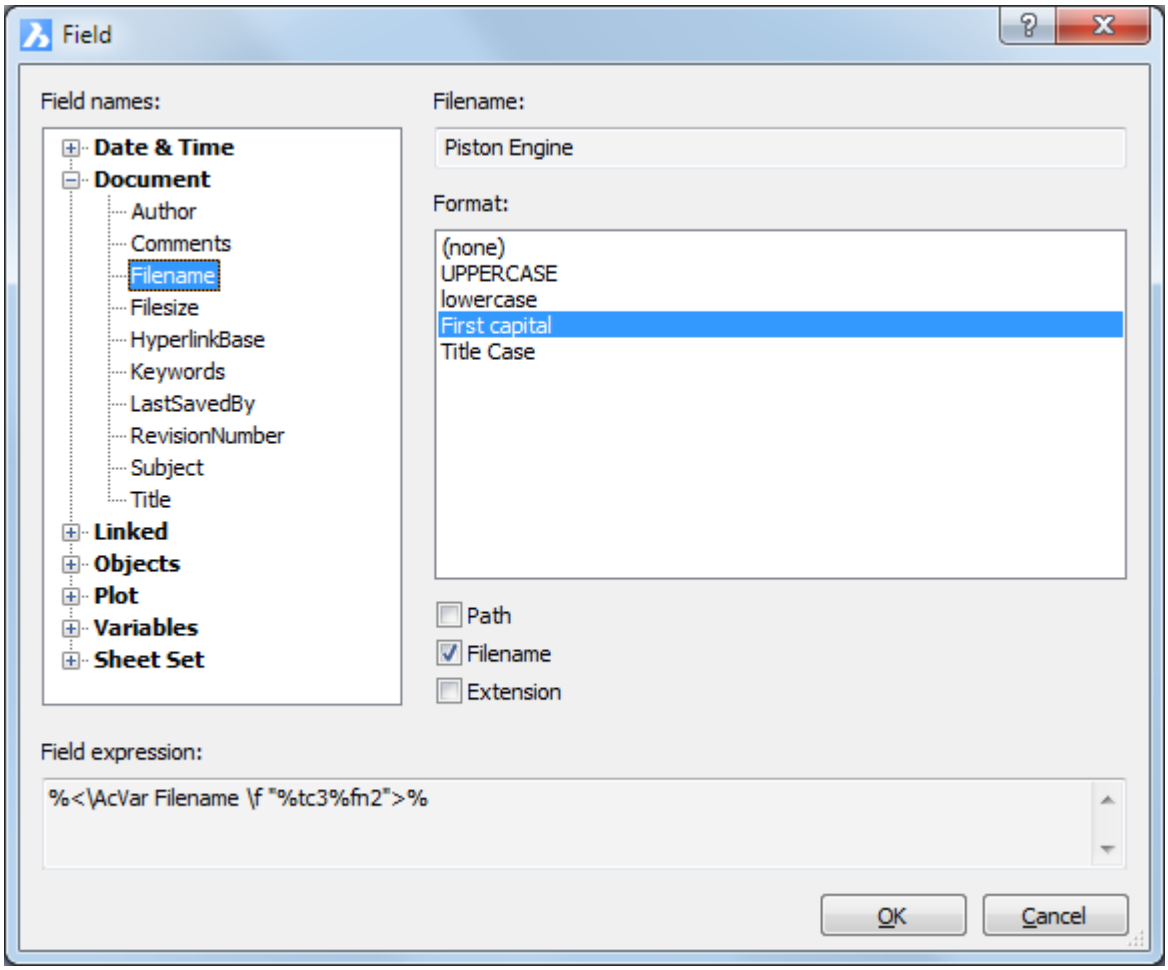
The FIELDEVAL system variable controls when fields are updated.



General Procedure to Create Fields

1. Do one of the following:
   - Click the *Insert Field* tool button ( ) on the *Insert* toolbar.
   - Choose *Insert Field...* in the *Insert* menu.
   - Type *field* in the command bar, then press Enter.
   - Click the *Field* tool button ( ) on the *Text Formatting* toolbar when creating or editing Mtext or a table cell content.

The *Field* dialog box displays.



2. Under *Field Names*, expand a group, then choose a field name.
   The settings for the selected field display.
3. Adjust the settings, then click the *OK* button.
   The *Field* dialog box closes.

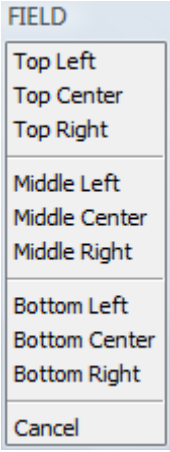
The field is created in the Mtext or table cell being edited

or

You are prompted: Specify start point or [Height/Justify]:
A context menu displays:

- (option) Type *H* or choose *Height* in the context menu to define the text height.

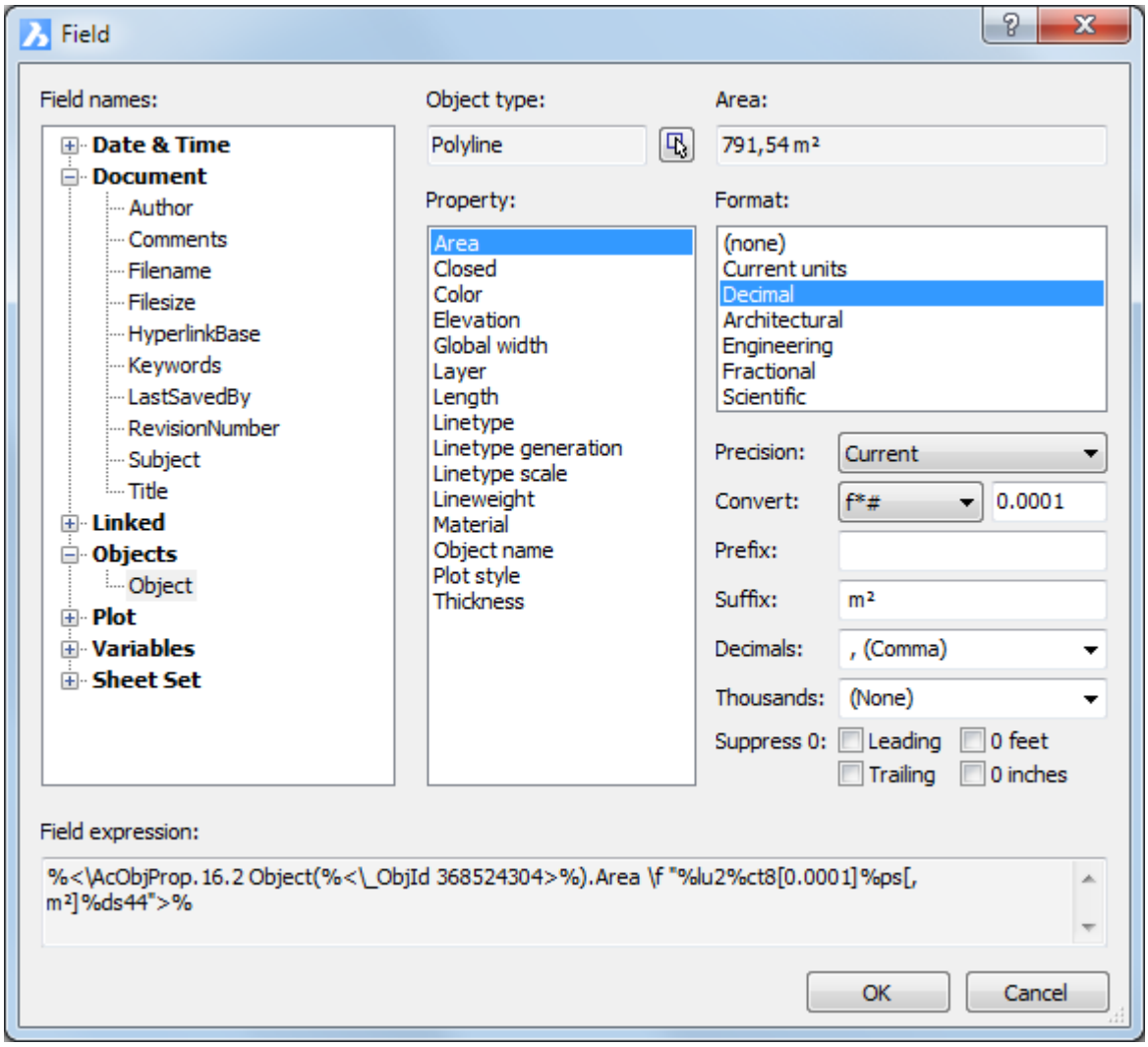
- (option- Type *J* or choose *Justify* in the context menu to set the text justification.



- Specify the insertion point of the field in the drawing.
  An *Mtext* entity, containing the field is created.

To Display an Entity Property in a Field

1. Launch the Field command.
2. Expand *Objects* in the *Field names* list and select *Object*.
3. Click the *Select Entity* button ( ) on the *Field* dialog box.
   The *Field* dialog box closes temporarily.
   You are prompted: Select Entity:
4. Click the entity you want to display a property of.
   The *Field* dialog box displays again.
5. Select the entity property in the *Property* list.



6. Set the display layout:

- *Format*: choose a unit format.
- *Precision*: select a precision in the list button
- *Convert*: this option allows to recalculate the field value.
    - Choose a formula in the list; e.g. f*#, where f is the factor you type in the Convert field and # is the field value.
    - Type a factor in the Convert field; e.g. 0.0001 to express the area of a polyline in square meters in a cm drawing.
- *Prefix*: type a text string
- *Suffix*: type a text string
- *Decimals*: select a decimal separator character
- *Thousands*: select a thousands separator character
- *Suppress 0*: allows to suppress leading and/or trailing zeros

8. Click the *OK* button.
9. Specify a point to place the field in the drawing.

To Edit a Field

1. Edit the table cell or Mtext entity that contains the field.
2. Do one of the following:
   - Double click the field.
   - Right click the field, then choose *Edit Field ...* in the context menu.
   - Select the field, then press *Ctrl + F*.

The *Field* dialog box displays.

3. Adjust the properties of the field.
4. Click the *OK* button to apply the changes.

To Update a Field

1. Do one of the following:
   - Click the *Update Field* tool button ( ) on the *Redraw / Regen* toolbar.
   - Choose *Update Fields...* in the *Tools* menu.
   - Type *updatefield* in the command bar.

You are prompted: Select field(s) to update:

2. Select the field(s) you want to update, then right click or press Enter.

The selected fields are updated.

Convert Field To Text

1. Edit the table cell or Mtext entity that contains the field.
2. Select the field.
3. Right click, then choose *Convert Field to Text* in the context menu.

Working with Tables

Commands: TABLE, -TABLE, TABLEDIT, TABLEEXPORT, TABLEMOD, TABLESTYLE

A table is an entity that is composed of rows and columns, much like a Microsoft Excel spreadsheet. A table entity can be created as an empty table or you can import data from a CSV (**C**omma **S**eparated **V**alues) file. The appearance of the table text and gridlines is controlled with a table style. Table styles are saved in the DWG file.

A default table consists of a *Title* cell (A), a row of *Header* cells (B) and one or more rows of *Data* cells (C).

A table can have one of two directions: *Down* (title row at the top) or *Up* (title row at the bottom).

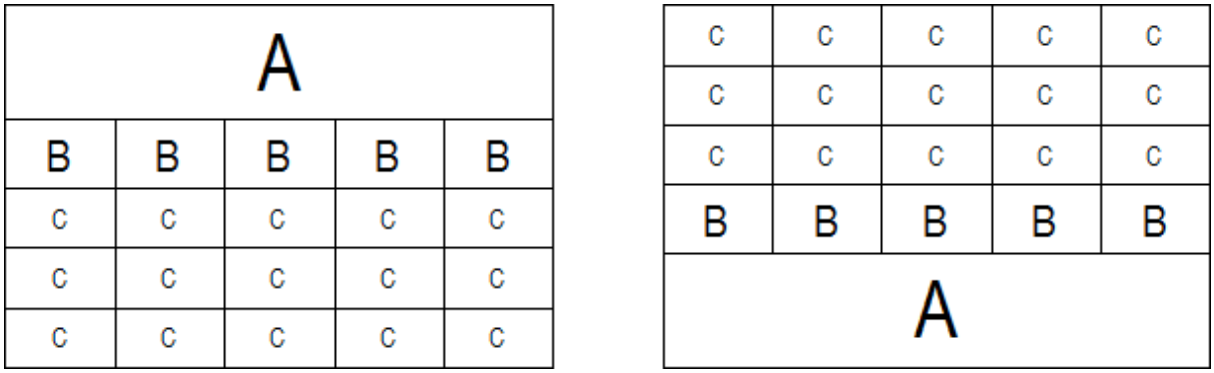


Table with a Down direction (left) and a Up direction (right).

Table Styles

The appearance of a table is controlled by a table style. Each drawing contains the *Standard* table style, which cannot be deleted. You can modify the *Standard* table style to fit your needs or create your own table styles. Table styles are created and edited in the *Table Styles* category of the Drawing Explorer dialog.

In a table there are 3 cell types: *Data*, *Header* and *Title*. Each cell type has its own cell style settings. By default all cells in a row share the same cell style settings, but you can edit the settings of each cell afterwards.

To open the Table Styles Explorer

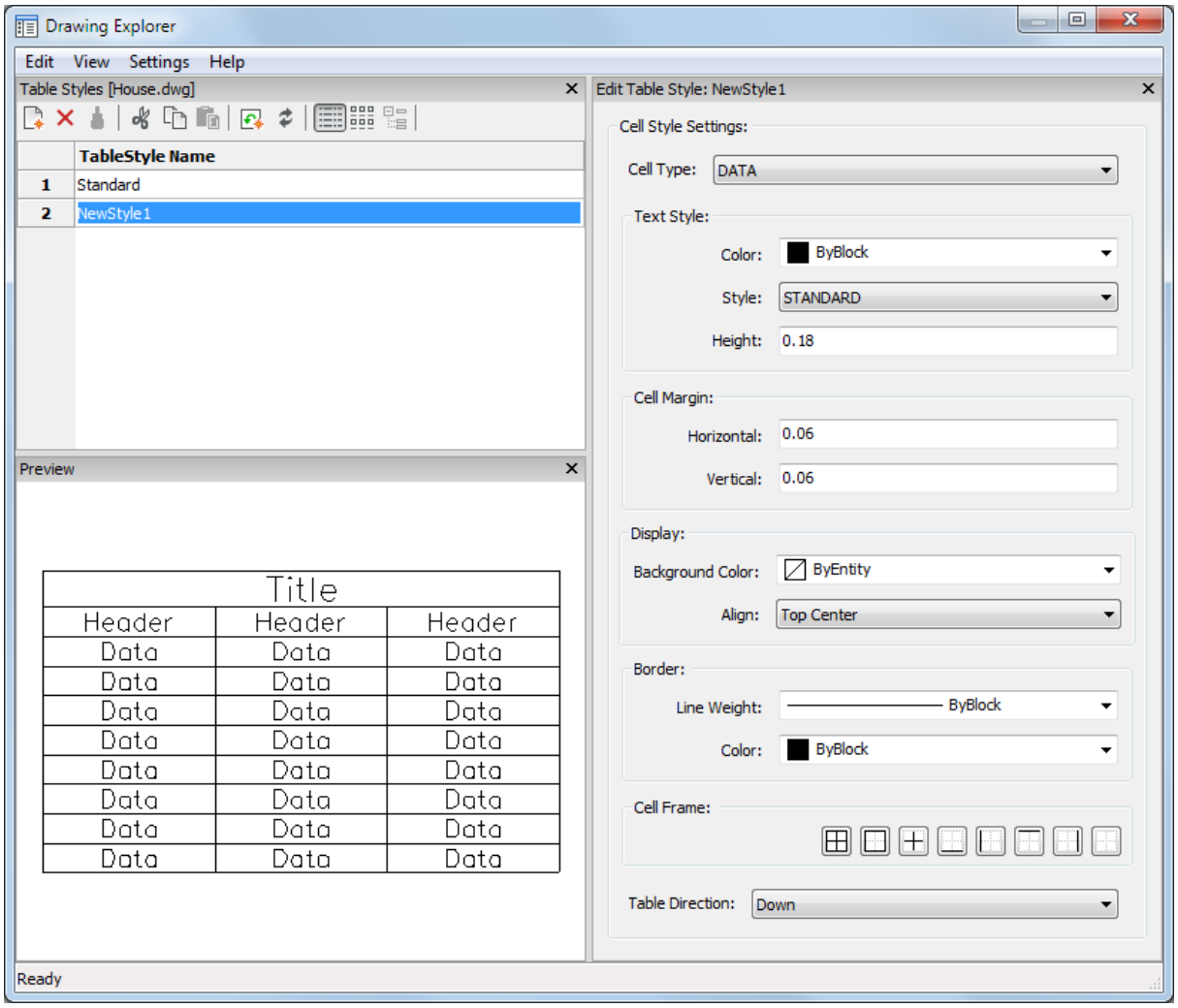
Do one of the following:

- Choose *Drawing Explorer > Table Styles...* in the *Tools* menu.
- Type *tablestyle* in the command bar, then press Enter
- Click the *Explore Table Styles* tool button ( ) on the Insert Table dialog.

To Create a Table Style

1. Open the Drawing Explorer - Table Styles dialog displays.
2. In the *Drawing Explorer - Table Styles* dialog do one of the following:
   - Choose *New* in the *Edit* menu.
   - Click the *New* tool button ( ).

A new style is created using default settings.



3. Rename the style, replacing the *NewStyle1* default name.
4. Select a cell type: *Data*, *Header* or *Title*.
5. Define the cell style settings for the selected cell type.
6. Repeat the previous steps for each cell type.

To Edit a Table Style

1. Open the Drawing Explorer - Table Styles dialog displays.
2. In the *Details* pane of the *Drawing Explorer - Table Styles* dialog, select the table style you want to edit.
3. Select a cell type: *Data*, *Header* or *Title*.
4. Edit the cell style settings for the selected cell type.
5. (option) Repeat the previous steps for another cell type.

To Apply a Table Style

To edit a table using the Properties bar

To Delete a Table Style

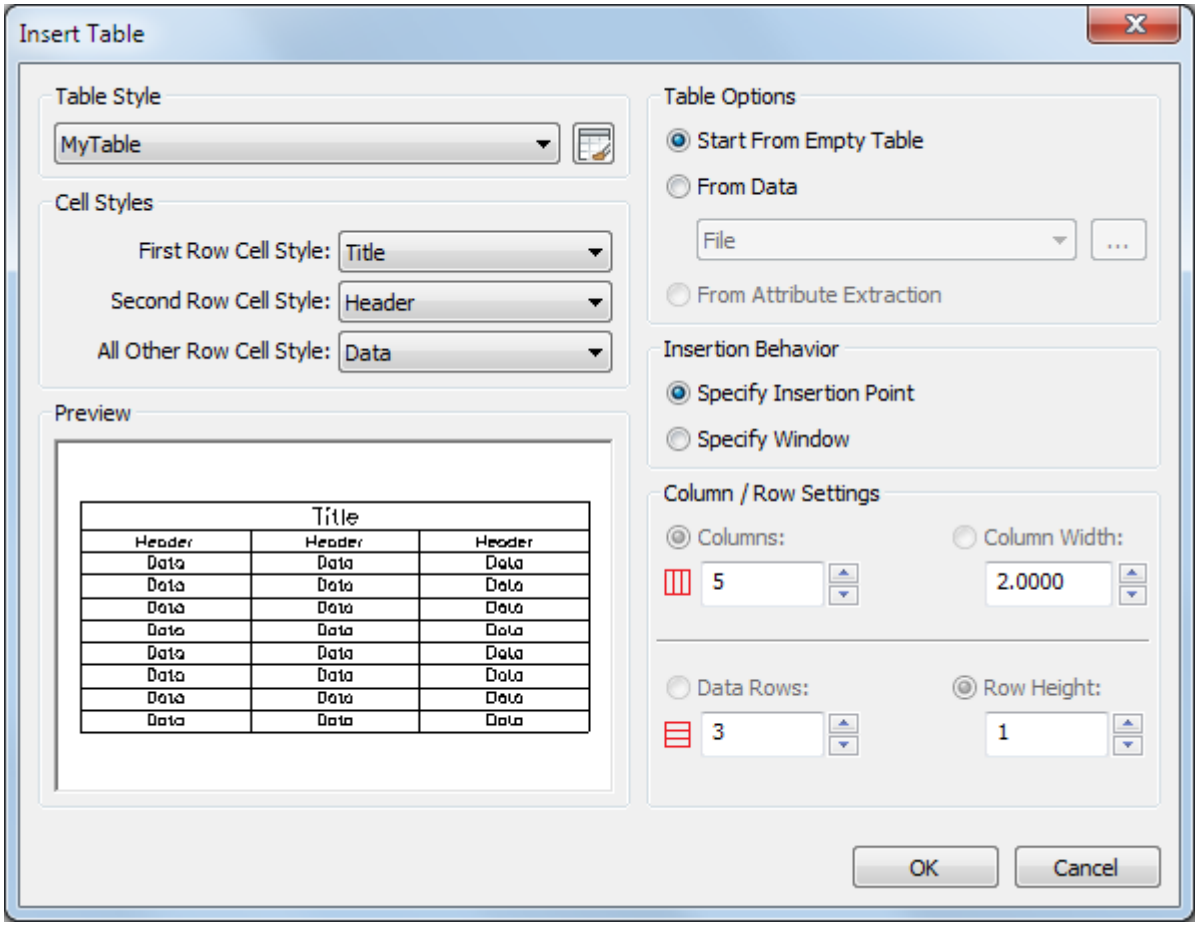
1. Open the Drawing Explorer - Table Styles dialog displays.
2. Select the table style you want to delete.
   Press and hold the Ctrl key to select multiple table styles.
3. Do one of the following:
   - Right click, then choose *Delete* in the context menu.
   - Click the *Delete* button in the *Details* menu on the *Drawing Explorer - Table Styles* dialog

NOTE It is not possible to delete a table style that is being used in the drawing.

To Create an Empty Table

1. Do one of the following:
   - Click the *Table* tool button ( ) on the *Draw* toolbar.
   - Choose *Insert Table...* in the *Draw* menu.
   - Type *table* in the command bar, then press Enter.

The *Insert Table* dialog displays:



2. Choose a Table Style from the *Table Style* list.
3. (option) Set the cell style for the first, second and other rows.
   By default the cell style for the first row is *Title*, the cell style for the second row is *Header* and the cell style for the other rows is *Data*.
4. Under *Table Options*, choose *Start from Empty Table*.

5. Under *Insertion Behavior* do one of the following:

   - Choose *Specify Insertion Point*, then specify the number of columns and the column width and number of data rows and the row height.
   - Choose Specify Window, then do one of the following:
     - Specify the number of columns and data rows.
     - Specify the column width and the row height.
     - Specify the column width and the number of data rows.
     - Specify the number of columns and the row height.

6. Click the *OK* button.
   Depending on your choice in step 5 you are prompted to specify an insertion point or a window.
   The table is created and the *Text Formatting* toolbar displays (see Create multiline text).
7. Do one of the following:

   - Start adding text in the table (see Edit the table content).
   - Click outside the table to close the *Text Formatting* toolbar.

To Create a Table from Data

When importing data from a CSV (**C**omma **S**eparated **V**alues) file, please make sure the list separator character on your system is the same as the one used in the CSV file (see Defining the list separator character).

1. Do one of the following:
   - Click the *Table* tool button ( ) on the *Draw* toolbar.
   - Choose *Insert Table...* in the *Draw* menu.
   - Type *table* in the command bar, then press Enter.

The *Insert Table* dialog displays.

2. Choose a Table Style from the *Table Style* list.
3. Under *Table Options*, choose *From Data*.
4. Click the Browse button ( ).
   The *Open* dialog displays.
5. Choose a CSV file, then press the *Open* button.
6. Click the OK button on the *Insert Table* dialog.
   The dialog box closes and the table is attached to the cursor.
7. Click a point in the drawing to place the table.
   The table is created and the *Text Formatting* toolbar displays (see Create multiline text).
8. Do one of the following:

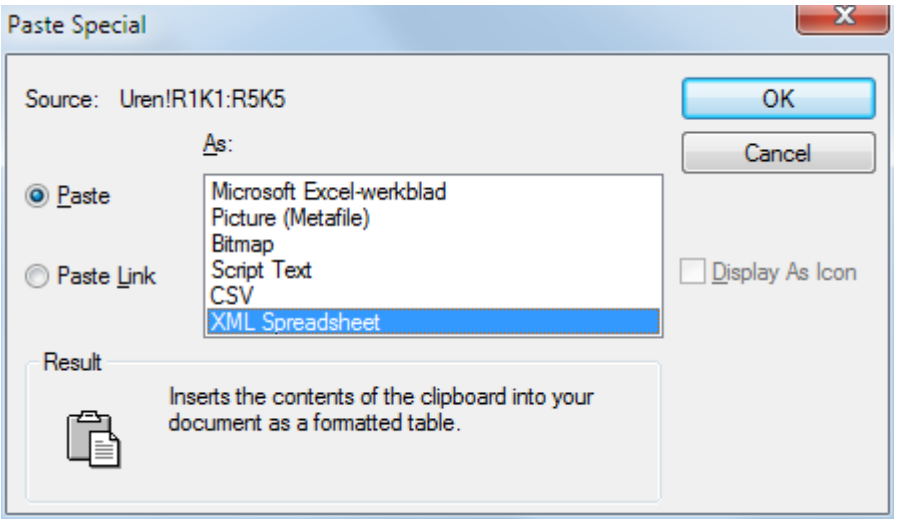
   - Start editing the text in the table (see Edit the table content).
   - Click outside the table to close the *Text Formatting* toolbar.

9. (option) Select the table and drag the grips to adjust the dimensions of the table.

To Create a Table from an Excel Spreadsheet

1. Open the spreadsheet in *Microsoft Office Excel*.
2. Select the cells you want to create a table from.
3. Copy the selection to the clipboard:
   - Press *Ctrl + C*.
   - Right click and choose *Copy* in the context menu.

4. In BricsCAD, choose *Paste special...* in the *Edit* menu.
   The *Paste Special* dialog box displays:

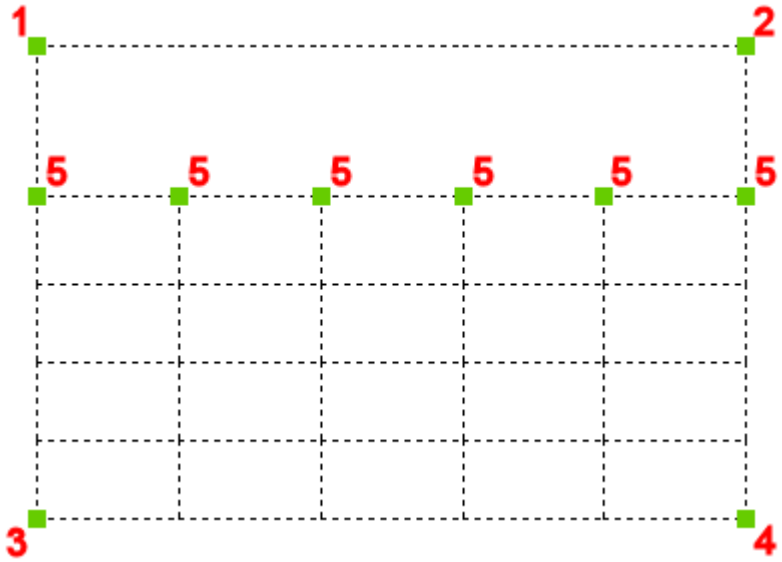


5. Choose *XML Spreadsheet* in the *As* option list.
6. Click in the drawing to insert the table.

Editing a Table Using Grips

When a table is selected, multiple grips display:

1. Moves the table.
2. Modifies the width uniformly.
3. Modifies the height uniformly.
4. Modifies the width and height uniformly.
5. Modifies the width of a column.

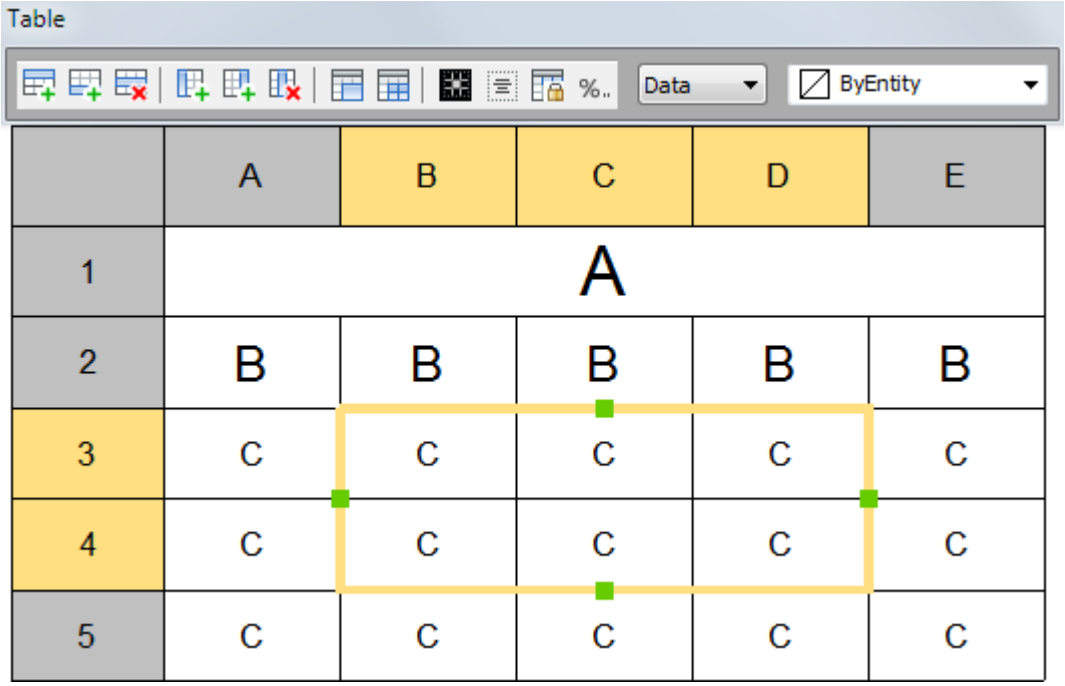


Editing a Table Using the Table Toolbar

1. To open the *Table* toolbar, do one of the following:
   - Click the cell you want to edit.
   - Press and hold the left mouse button then define a rectangle to select multiple cells

Column letters and row numbers display.
The column(s) and row(s) of the selection hightlight.

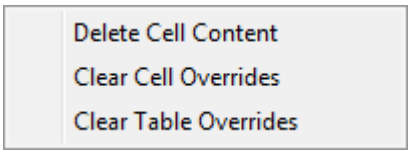
A rectangle with four handles is drawn around the selection.



2. Do one of the following:

- Click a cell to select it.
- Press and hold the left mouse button, then drag the cursor to select multiple cells.

3. (option) Drag the grips of the selection to modify the dimensions.
4. (option) Click a tool in the Table toolbar.
5. (option) Right click to display a context menu:



- *Delete Cell Content*: deletes the text in the selected cells.
- *Clear Cell Overrides*: resets all cell properties of the selected cells to the Cell Style settings as defined in the Table Style.
- *Clear Table Overrides*: resets all cell properties of all cells to the Cell Style settings as defined in the Table Style.

Table tools

Icon	Tool name	Description
	Insert row above	Inserts an empty row above the selection.
	Insert row below	Inserts an empty row below the selection.
	Delete row	Deletes the row(s) of the selection.
	Insert column left	Inserts an empty column at the left of the selection.
	Insert column right	Inserts an empty column at the right of the selection.
	Delete column	Deletes the column(s) of the selection.
	Merge	Merges the selected cells.
	Unmerge	Splits previously merged cells.
	Border style	Sets the border style of the selection (not implemented yet).
	Text alignment	Sets the text alignment of the selection.
	Lock / Unlock	Locks / Unlocks the selection.
%..	Set cell format	Sets the cell format of the selection (not implemented yet)
Title	Set cell style	Sets the style of the selected cells: *Title*, *Header* or *Data*.
ByEntity	Set cell background	Sets the background color of the selection.

6. (option) Repeat steps 3 and 4 to continue editing the table.
7. (option) Double click a cell to edit the cell text.
8. (option) Do one of the following:

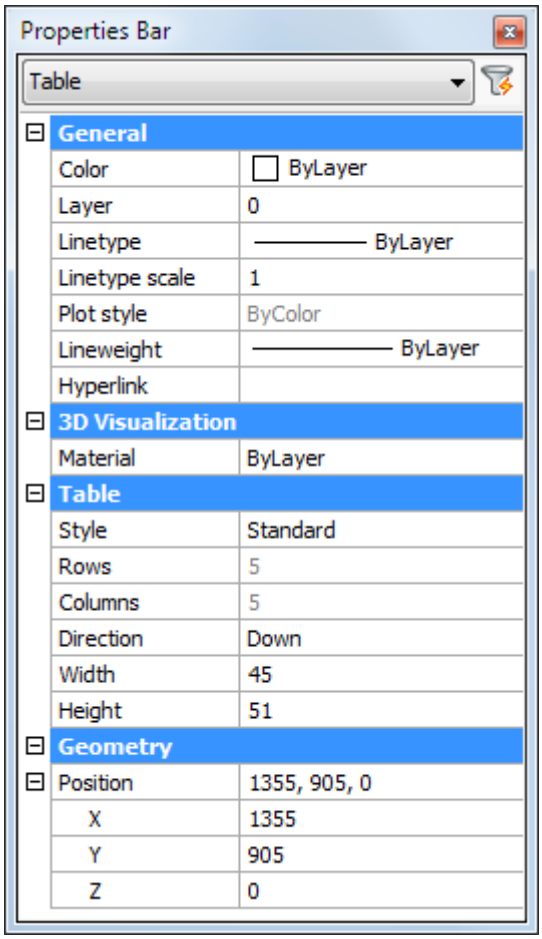
   - Click another cell.
   - Press and hold the left mouse button, then drag the cursor to select multiple cells.

9. Click outside the table to close the *Table* toolbar.

Editing a Table in the Properties Bar

1. Click one of the grid lines.

The properties of the table display in the Properties bar.

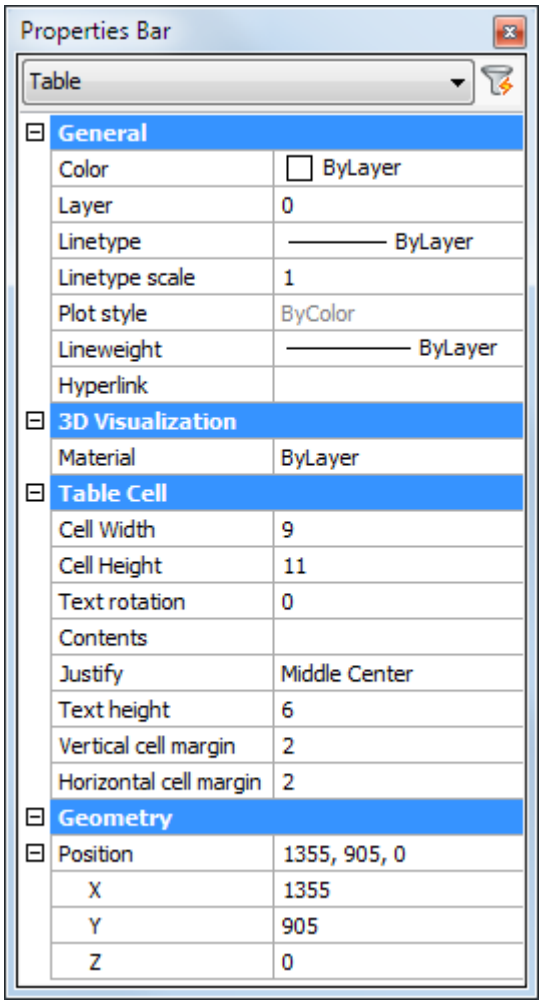


2. (option) Apply a different Table style.
3. (option) Change the table direction.
4. (option) Modify the width and/or height of the table.
5. (option) Modify the vertical and/or horizontal cell margins.
6. Press the Esc key to stop editing the table.

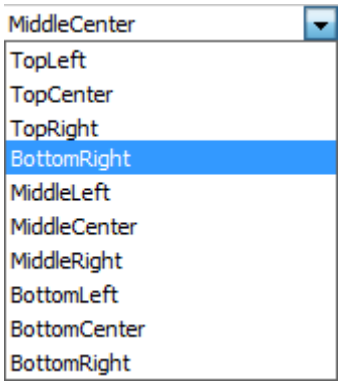
Editing a Cell in the Properties Bar

1. Click the cell.

The properties of the cell display in the Properties bar.



2. (option) Modify the width and/or height of the cell.
   The width and height of the column and/or row of the cell are changed.
3. (option) Modify the text rotation.
   The options are: 0°, 90°, 180° and 270°.
4. (option) Edit the cell content.
5. (option) Choose a different text alignment option.



Evaluating Formulas

Currently formulas are supported partially. There are more complex formulas which are not supported yet, but in most cases formulas in Excel sheets will remain functional when the Excel sheet is imported as a table. Editing and creating formulas through the field dialog is not supported yet, but in-place editing/creating is possible. E.g.: if you type =SUM(A2:A4) in a cell, it will be interpreted as a formula, and the sum of the valuess in the cells A2 through A4 is calculated.

To Edit the Table Content

1. Do one of the following:
   - Double click the cell you want to edit.
   - Type *tabledit* in the command bar, then press Enter and click the cell you want to edit.

The *Text Formatting* toolbar displays above the selected cell (see Create multiline text).

2. Edit the text in thale cell.
   (option) Press Alt + Enter to create a second line in the same cell.
3. To edit another cell:

   - Press the TAB key to go to the next cell in the same row.
   - Press Shift + TAB to go to the previous cell in the same row.
   - Press Enter to go the next cell in the same column.
   - Press Shift + Enter to go to the previous cell in the same column.
   - Press one of the Arrow keys.

4. Click to stop editing the table content.

To Export a Table

When exporting data to a CSV (**C**omma **S**eparated **V**alues) file, the list separator character off your system is used in the CSV file (see Defining the list separator character).

1. Type *tableexport* in the command bar, then press Enter.
   You are prompted: Select table to export:
2. Click the table you want to export the data of.
   The Export Data dialog opens.
3. Select a folder to save the data in.
4. Type a name in the *File name* field.
5. Click the *Save* button.
   The data in the selected table are saved as a CSV (**C**omma **S**eparated **V**alues) file.

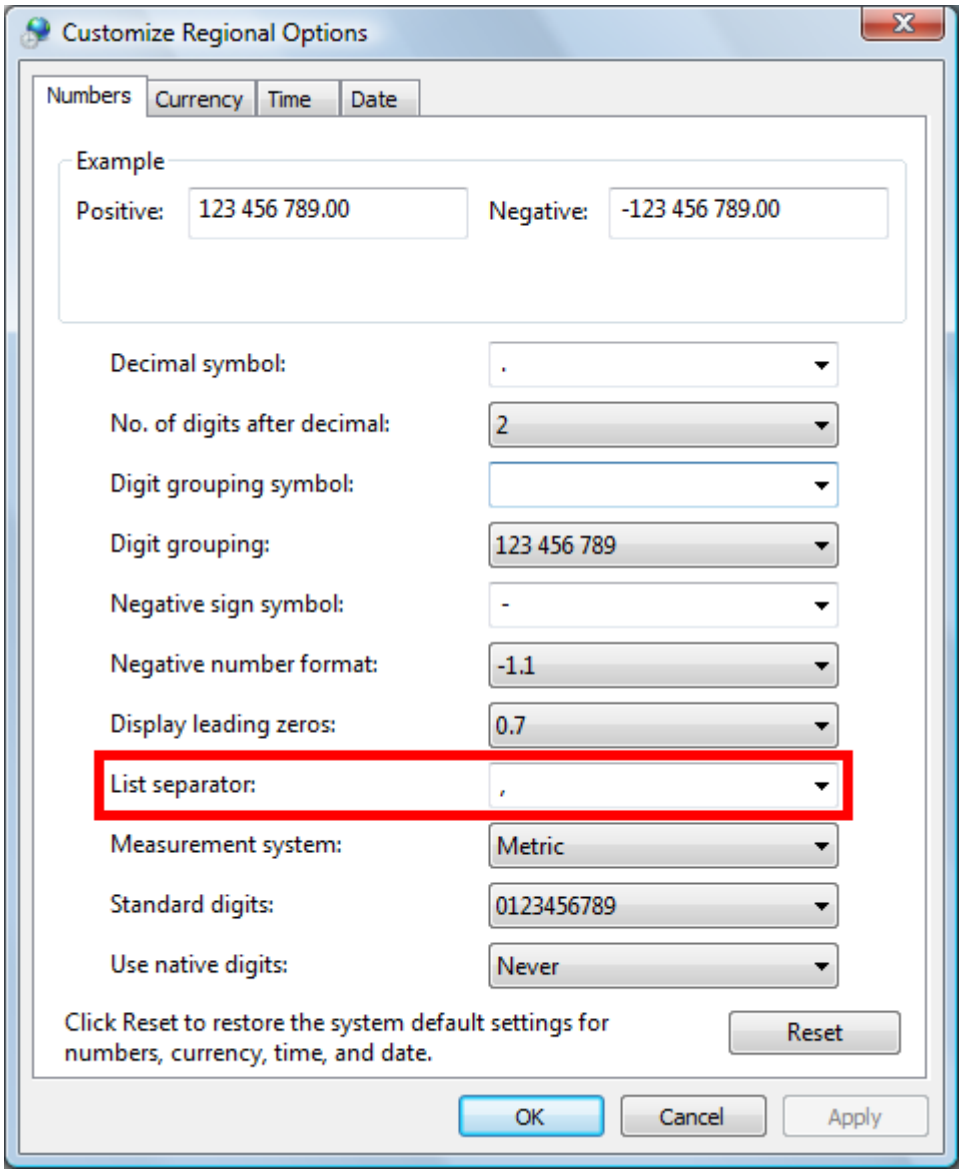
Defining the list separator character

1. Click the Start button in the Windows Taskbar, then choose Control Panel.
2. Do one of the following:
   - On the Control Panel Home window, choose: Clock, Language and Region > Regional and Language Options.
   - On the Control Panel Classic View window, double click the Regional and Language Options icon.

The Regional and Language Options dialog box displays.

3. On the Regional and Language Options dialog box click the Customize this format... button.

The Customize Regional Options dialog box displays:



4. On the Numbers page of the Customize Regional Options dialog box, go to List separator, then click the down arrow and select either comma (,) or semicolon (;).
5. Click the OK button on the Customize Regional Options and Regional and Language Options dialog boxes.

Annotative Entities

Annotations are all entities in your drawing which are not a model or geometry, such as dimensions, texts, symbols, etc. A common problem with annotations is that they are often scale dependent. Some annotations must be scaled to the plot scale, or are meant to be shown at certain scales only.

Annotative styles and entities change the way in which they are displayed (size, location, and orientation) depending on the current scale factor and the viewport.

The following entities can have the annotative property:

- text and multiline text
- hatches
- blocks and attributes
- dimensions and tolerances
- leaders and multileaders

The following styles can have the annotative property:

- text styles (see the STYLE command)
- dimension styles (see the DIMSTYLE command)
- multileader styles (see the MLEADERSTYLE command)

NOTES

- Entities that are created using an annotative style, get the current annotation scale assigned automatically.
- When an annotative style is applied to a non-annotative entity, the annotative property is not assigned automatically.

Commands

ANNORESET: Resets all scale representations of the selected annotative entities to the location of their current scale representation, thus synchronizing all scale representations.

ANNOUPDATE: Updates the selected annotative entities to match their current style definition.

OBJECTSCALE: Adds or removes supported scales for annotative entities through a dialog box.

-OBJECTSCALE: Adds or removes supported scales for annotative entities at the command line.

SCALELISTEDIT: Edits the scale list of a drawing, used in *Plot Scale* section of the *Print* and *Page Setup* dialogs and the *Standard scale* property of a paper space viewport. Defines the available annotation scales for annotative entities.

-SCALELISTEDIT: Edits the scale list of a drawing at the command line.

System Variables and User Preferences

ANNOALLVISIBLE: Toggles the display of annotative enitities that do not support the current annotation scale.

ANNOTATIVEDWG: Specifies whether or not the drawing behaves as an annotative block when inserted into another drawing.

CANNOSCALE: Sets the name of the current annotation scale for the current space. Only scales that exist in the current scale list are accepted. The CANNOSCALE setting is saved with model space and with each layout viewport in the various layouts. When you create an annotative entity the current annotative scale is applied automatically.

CANNOSCALEVALUE: Displays the value of the current annotation scale. E.g. the value of the 1:2 scale is 0.5.

HPANNOTATIVE: Controls whether new hatch entities are annotative.

MSLTSCALE: If ON, linetypes in model space are scaled to the current annotation scale. A regen is needed when you toggle the value of MSLTSCALE. This variable is the equivalent of the PSLTSCALE (Paperspace linetype scale) variable in paper space.

MSOLESCALE: Controls the size of an OLE object containing text when pasted in model space. The MSOLESCALE system variable only affects the initial scaling. Existing OLE objects are not affected if the variable is modified.

SAVEFIDELITY: Controls whether the current display of the drawing is preserved when opened in a program that does not support annotative entities.

SELECTIONANNODISPLAY: Controls whether all scale representations display when an annotative entity is selected.

Setting the Current Annotation Scale

To set the current annotation scale, do one of the following:

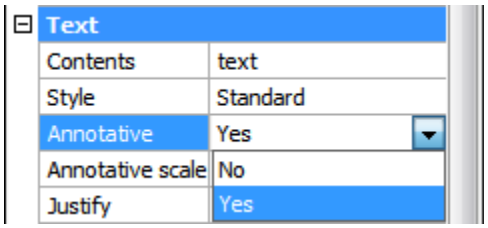
- Click the *Annotation Scale* field under *Misc* in the Properties Bar, then press the down-arrow button and select a scale in the list.
- Right click the *Annotation Scale* field in the Status Bar, then select a scale in the list. If the *Annotation Scale* field is not available, click the down arrow at the right hand side of the Status Bar and choose *Annotation Scale* in the list.

NOTE Only scales that exist in the *Scale List* of the drawing are available. Use the SCALELISTEDIT command to edit the *Scale List*.

Making an Annotation Entity Annotative

Annotation entities which are created using a non-annotative style can be made annotative using the following procedure:

1. Select the annotation entity.
2. In the *Properties Bar*, click the *Annotative* property.
3. Select *Yes*.



The current annotation scale is applied to the selected entity.

Annotation Scales

Commands: OBJECTSCALE and SCALELISTEDIT

Annotation scales are used to calculate the scaling factor for annotative entities. It defines the height of annotative text entities or the overall scaling of annotative blocks. In general the annotation scale allows to create annotative entities in model space at the correct paper space size automatically.

- Texts, MTexts, Dimensions, MLeaders: The current annotation scale is applied upon creation if the current style is annotative.
- Hatches and Gradient Fills: The current annotation scale is applied if the HPANNOTATIVE system variable is ON.
- Blocks: The current annotation scale defines the scaling of annotative blocks upon insertion.

Annotative entities can support multiple annotation scales. When the SELECTIONPREVIEW system variable is active or when using the Quad cursor menu, an icon displays next to the highlighted entity indicating whether a single annotation scale ( ) applies to an annotative entity or multiple ( ).

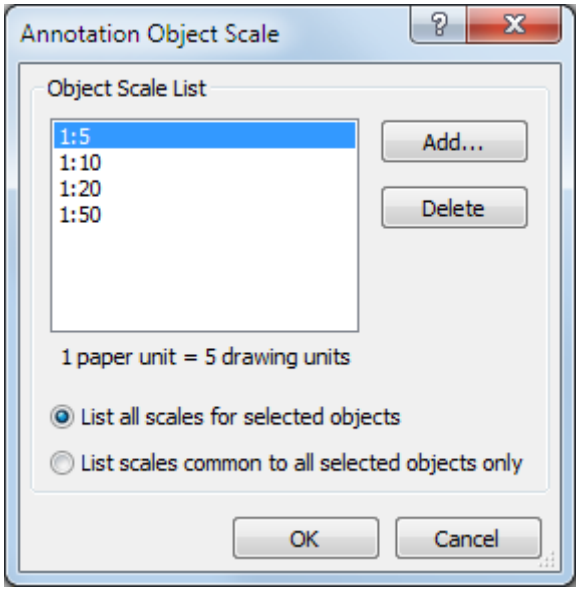
To add or remove annotation scales to annotative entities

Use the following procedure to:

- Check which annotation scales currently apply.
- Add annotation scales.
- Remove annotations scales.

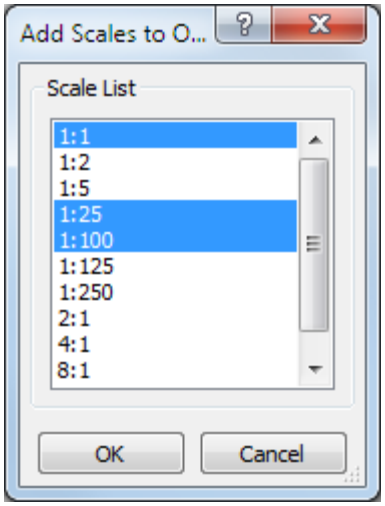
1. Select the annotative entity or entities.
2. In the Properties Bar, select the *Annotative Scale* field.
3. Click the *Browse* button (...) of the *Annotative Scale* field.

The *Annotation Object Scale* dialog displays.



4. (option) If multiple entities were selected, choose whether to list all scales for the selection or the common scales only.
5. (option) Do one of the following:

   - Select one or more scales in the list, then click the *Delete* button.
     The selected scale(s) are removed from the list.
   - Click the *Add...* button, then select one or more scales in the list and click the *OK* button.
     The selected scale(s) are added are added to the list.



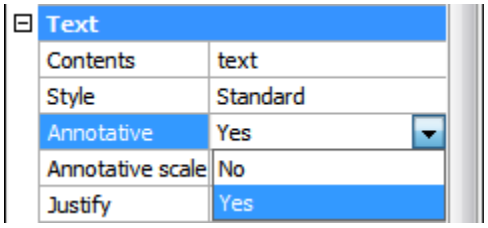
6. Click the OK button to close the *Annotation Object Scale* dialog.
   The changes are applied to the selection set.

NOTE Only scales that exist in the *Scale List* of the drawing are available. Use the SCALELISTEDIT command to edit the *Scale List*.

Adding or Removing Annotation Scales to Annotative Entities

Annotation entities which are created using a non-annotative style can be made annotative using the following procedure:

1. Select the annotation entity.
2. In the *Properties Bar*, click the *Annotative* property.
3. Select *Yes*.



The current annotation scale is applied to the selected entity.

Annotative Texts

Commands: TEXT, MTEXT and STYLE

Text entities are created using the current text style:

- If the text style is annotative, the text entity gets the annotative property automatically and the current annotation scale is assigned.
- If the text style is not annotative, you need to make the text entity annotative manually.

Annotative texts have two height properties:

- *Model text height*: the display height on the screen in model space, with respect to the current annotation scale.
- *Paper text height* : the height on paper if printed on the supported annotation scale.

To make a text entity annotative

1. Select the text entity.
   The properties of the selected text entity display in the Properties Bar.
2. In the Properties Bar, select the *Annotative* property.
3. Choose *Yes* from in the options list.
   The current annotation scale is assigned.
   The text is not scaled: the current text height becomes the *Model text height*.

To remove the annotative property

1. Select the text entity.
   The properties of the selected text entity display in the Properties Bar.
2. In the Properties Bar, select the *Annotative* property.
3. Choose *No* from in the options list.
   If the text entity supports multiple annotation scales, the text is scaled back to its initial scale.

Annotative Hatches

Commands: HATCH, -HATCH, GRADIENT

The HPANNOTATIVE system variable controls whether new hatch or gradient entities are annotative.

HPANNOTATIVE is OFF by default. The system variable is not saved and is reset to its default value when BricsCAD closes.

To set the HPANNOTATIVE system variable

Do one of the following:

- In the *Settings* dialog, under *Drafting / Entity Creation / Hatches*.
- Check the *Annotative* option on the *Hatch and Gradient* dialog box of the HATCH or GRADIENT command.
- Choose the *Annotative* option in the command prompt of the -HATCH command.

To make a hatch or gradient entity annotative

1. Select the hatch or gradient fill.
   The properties of the selected entity display in the Properties Bar.
2. In the Properties Bar, select the *Annotative* property.
3. Choose *Yes* from in the options list.
   The current annotation scale is applied to the selected hatch or gradient entity and the hatch is scaled accordingly.

To remove the annotative property

1. Select the hatch or gradient fill.
   The properties of the selected entity display in the Properties Bar.
2. In the Properties Bar, select the *Annotative* property.
3. Choose *No* from in the options list.

Annotative Blocks

Commands: BLOCK and EXPBLOCKS

The annotative property of a block is either set when creating the block definition or afterwards, in the *Drawing Exlorer / Blocks* dialog.

Annotative blocks can have an optional property to always match the orientation of a layout, independent from the rotation of the viewport content.

To make a block definition annotative

1. Open the Drawing Explorer / Blocks dialog.
2. If necessary, choose Detail View.
   Annotative block definitions have the *Annotative* icon ( ) displayed.
3. Click in the *Annotative* field of a block definition to toggle the annotative property on/off.

To match the orientation of a block to a layout

1. Open the Drawing Explorer / Blocks dialog.
2. If necessary, choose Detail View.
3. Click the checkbox in the *Match Orientation* field of a block definition to toggle the *Match Orientation* property on/off.

Annotative Dimensions

Dimension entities are created using the current dimension style:

- If the dimension style is annotative, the dimensions get the annotative property automatically and the current annotation scale is assigned.
- If the dimension style is not annotative, you need to make the dimensions annotative manually.

NOTE The *Dim Scale Overall* (DIMSCALE) setting is not editable for annotative dimension styles, and is neglected for annotative dimensions.

It is recommended to set *Dim Scale Overall* = 1 for annotative dimension styles.

To make a dimension entity annotative

1. Select the dimension entity.
   The properties of the selected dimension entity display in the Properties Bar.
2. In the Properties Bar, select the *Annotative* property.
3. Choose *Yes* from in the options list.
   The current annotation scale is assigned.
   The dimension entity is scaled with respect to the current annotation scale.

To remove the annotative property

1. Select the dimension entity.
   The properties of the selected dimension entity display in the Properties Bar.
2. In the Properties Bar, select the *Annotative* property.
3. Choose *No* from in the options list.
   The dimension entity is scaled with respect to the current value of the *Dim Scale Overall* setting.

Modifying Entities

Entity Modification Commands

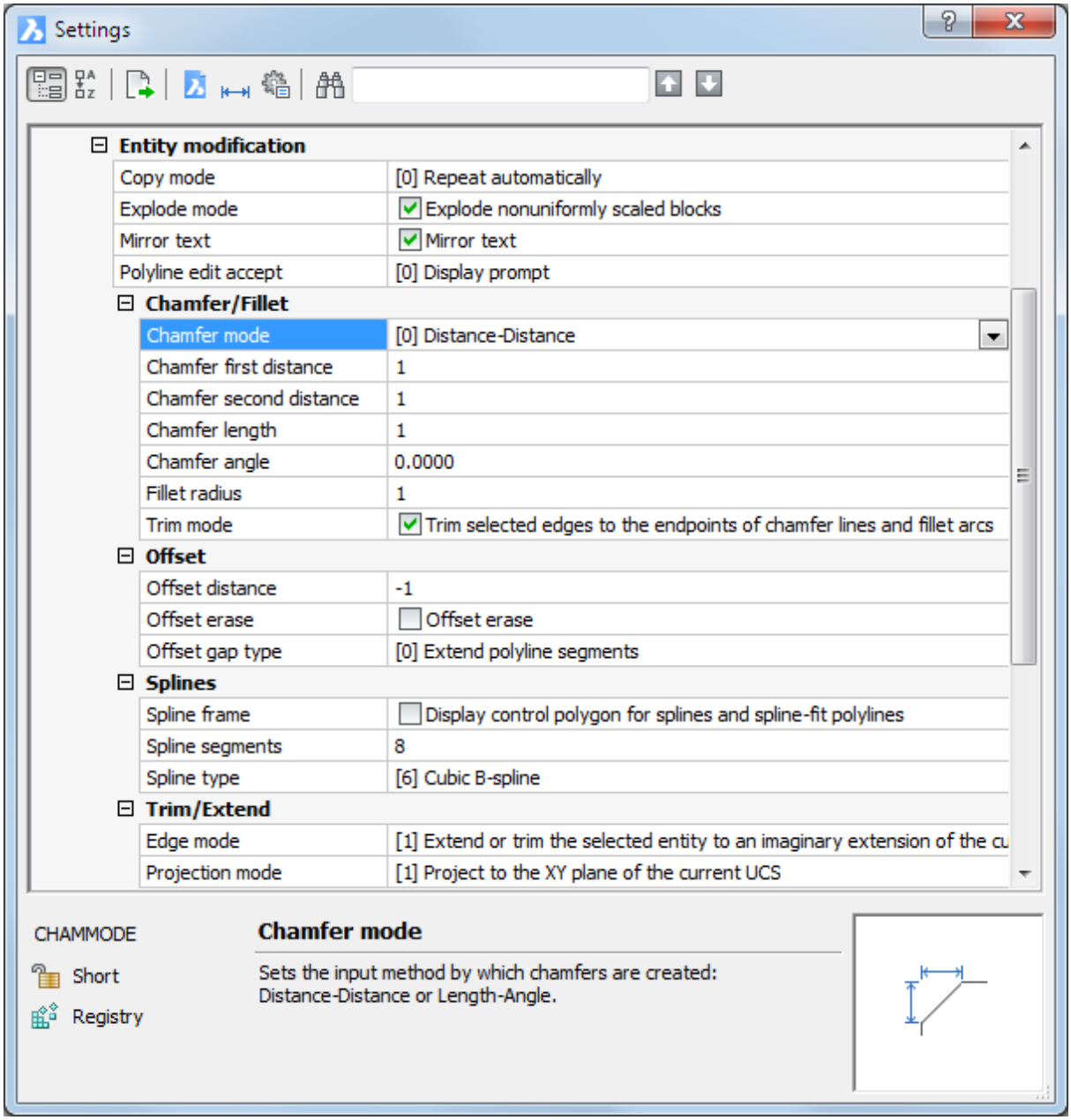
All entity modification tools can be found either on the *Modify* toolbar or in the *Modify* menu.

Icon	Tool name	Command	Description
	Manipulator	MANIPULATE	Launches the Manipulator widget; rotates, moves, mirrors and scales entities. Keep the Ctrl-key pressed while starting an action to manipulate a copy of the selected entities.
	Move	MOVE	Moves entities to another location in the same drawing or into another drawing.
	Copy	COPY	Draws a duplicate or multiple duplicates of the selected entities.
	Offset	OFFSET	Creates a parallel or offset copy of curves and lines.
	Scale	SCALE	Changes the scale of existing entities, either enlarging them or reducing them proportionately in x, y, and z directions.
	Rotate	ROTATE	Rotates entities around a specified point.
	3D Rotate	ROTATE3D	Rotates entities around a three-dimensional axis.
	Mirror	MIRROR	Moves or copies the reflected image of entities about a line.
	3D Mirror	MIRROR3D	Moves or copies the reflected image of entities about a plane.
	Array	ARRAY	Creates multiple copies of entities in one of two symmetrical patterns: rectangular (rows and columns) or polar (circular).
	3D Array	3DARRAY	Creates multiple copies of entities in three dimensions. Entities are arrayed in a three-dimensional rectangular (rows, columns and levels) pattern or a two-dimensional polar (circular) pattern in three-dimensional space.
	Break	BREAK	Splits an entity into two entities.
	Join	JOIN	Joins two entities (two or more lines or two or more arcs) into one entity.
	Trim	TRIM	Erases the portions of selected entities that cross a specified boundary.
	Align	ALIGN	Aligns selected entities with other entities in three-dimensional space.
	Extend	EXTEND	Lengthens lines, arcs, polylines or rays to meet another entity.
	Edit Length	LENGTHEN	Changes the length of a line, polyline, freehand entity or arc.
	Stretch	STRETCH	Moves a portion of a drawing while retaining connections to other parts of the drawing.
	Chamfer	CHAMFER	Creates a chamfer, or a beveled edge, at the intersection of two 3D solids, lines, rays, or infinite lines.

	Fillet	FILLET	Creates a fillet, or rounded corner, at the intersection of 3D solids, two lines, rays, or infinite lines.
	Measure	MEASURE	Divides a selected entity into segments by placing markers (points or blocks) at specified intervals along its length or circumference.
	Divide	DIVIDE	Places markers (points or blocks) along a selected entity. The markers evenly divide the entity into the specified number of equal parts.
	Hatch Edit	HATCHEDIT	Edits hatches and gradient fills in a dialog box
	Edit Polyline	PEDIT	Edits a two-dimensional or three-dimensional polyline, or a polygon mesh.
	Spline Edit	SPLINEDIT	Edits a two-dimensional or three-dimensional spline.
	Flatten	FLATTEN	Projects 2D and 3D entities onto the XY-plane of the current view.
	Explode	EXPLODE	Ungroups a block, polyline, polyface mesh, solid, or hatch, creating separate entities for each element.
	Change	CHANGE	Changes the location, size, color, elevation, layer, linetype, linetype scale, lineweight, and three-dimensional thickness of entities.
	Properties	PROPERTIES	Opens the BricsCAD Properties Bar (if not already open).

Entity Modification Settings

Entity modification settings are found in the *Entity modification* settings group of the *Drawing* settings category in the Settings dialog.



Match Properties

Command: MATCHPROP

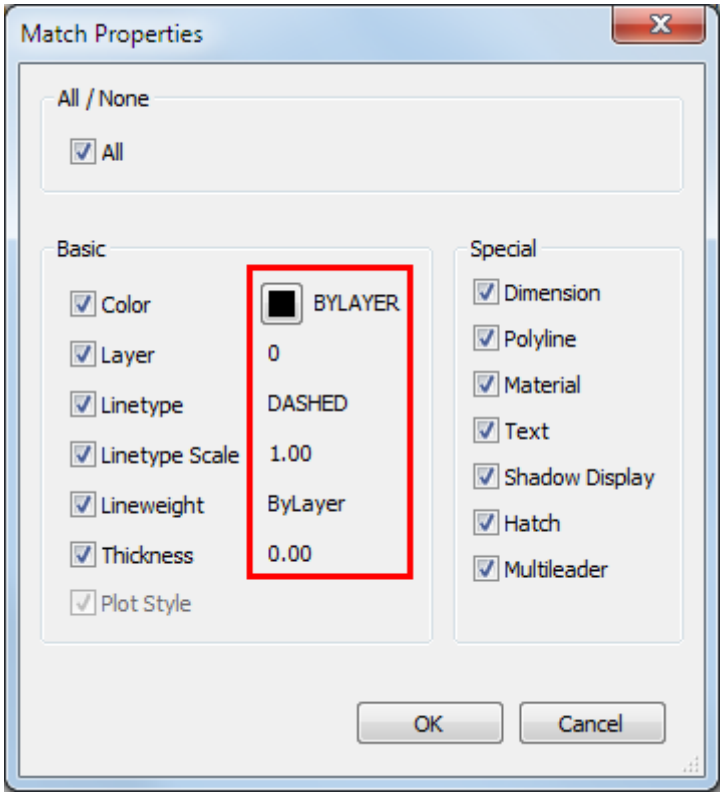
The *Matchprop* command assigns the properties of one entity to one or more other entities (short for "match properties").

To copy properties between entities

1. Do one of the following:
   - Click the *Match* ( ) tool button on the *Standard* toolbar.
   - Type match in the command bar then press Enter.

You are prompted:
Select entity to copy properties from:

2. Select the source entity.
   You are prompted:
   Active Settings: 'Color' 'Layer' 'Linetype' 'Linetype Scale' 'Lineweight' 'Thickness' 'Plot Style' 'Dimension' 'Polyline' 'Material' 'Text' 'Shadow Display' 'Hatch' 'Multileader'
   Select entities to copy properties to or [Settings]:
3. (option) Type *S*, then press Enter.
   The *Match Properties* dialog box displays:



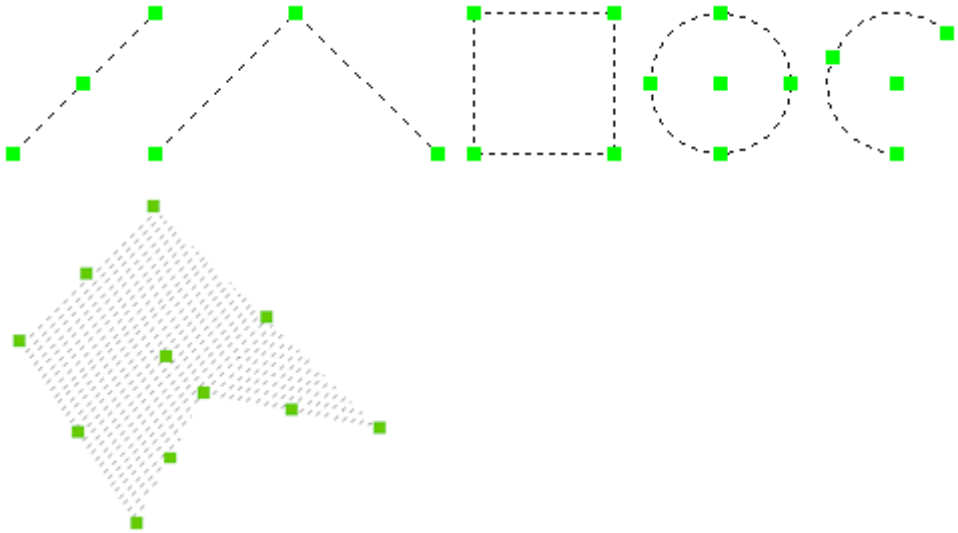
The entity properties you are about to copy display in the Match Properties dialog.
Adjust the settings, then click the *OK* button

4. Select the target entities.
   You are prompted: Select entities to copy properties to or [Settings]:
5. Do one of the following:

   - (option) Repeat step 3 to adjust the settings, then select more target entities.
   - Right click or press Enter to stop.

Grips editing

Grips display as small colored squares at key-points on selected entities.



Whether grips display or not and the appearance of grips is controlled through a series of grips settings.

When the display of grips is turned on, grips can be dragged using the mouse in order to stretch, move, mirror, rotate or scale entities.

The *Copy* option of the grips editing commands allows to create multiple copies of the entities being edited.

To select a grip, move the mouse over it, then click. Selected grips display in a different color, which is defined by the *HOTGRIP* system variable. A selected grip is called a hot grip. By default hot grips display in red.

The *midpoint* grip on lines and infinite lines, the *startpoint* grip on rays, the *center* grip on circles, arcs, ellipses and elliptical arcs, the grip on points and the *insertion point* grip of texts, mtexts, blocks and Xrefs can be used to quickly move such entities.

NOTE When DRAGSNAP is ON, dragged entities display at the current entity snap location.

Selecting Multiple Grips

1. Select all entities you want to edit.
2. Press and hold the Shift key, then click the first grip.
   The selected grip displays in red.
3. Keep the Shift key pressed, then click more grips.
   All selected grips display in red.

4. Release the Shift key, then click the hot grip you want to use as the base point.
   The hot grips move simultaneously.

Grip Edit Command Options

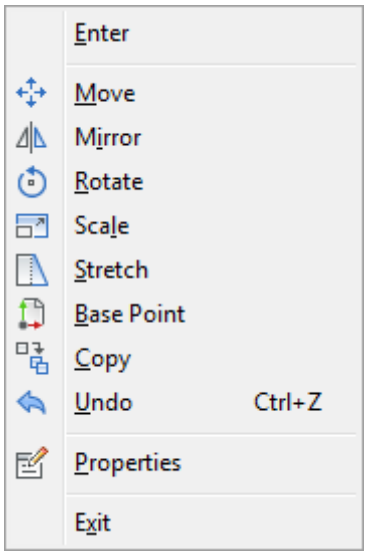
Keyboard	Prompt Box	Description
B	Base Point	Prompts you to specify a new base point, other than the selected grip.
C	Copy	Copies the selected entities and keeps the originals. By default, multiple copies can be created. Press Enter to stop making copies.
U	Undo	If the *Copy* option is chosen, deletes the latest copy.
R	Reference	(Applies to *Grip Rotate* only) Allows to specify a reference angle. Type the angle in the command bar or specify two points.
X	Exit	Aborts the grip edit command.

The grips editing context menu

The grips editing commands can either be chosen by repeatedly hitting the space bar or the Enter key or in a context menu.

To display the context menu, do the following:

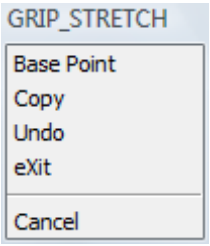
1. Select the entities you want to manipulate, then drag one of the grips.
2. Right click.



To Stretch Entities Using Grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point.
   All selected grips move simultaneously.

The *GRIP_STRETCH* prompt box displays:



You are prompted: Enter new point: Base point/Copy/Undo/eXit:

4. Do one of the following to define the displacement point:

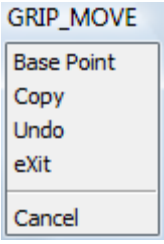
   - Click in the drawing.
   - Enter the coordinates, then press Enter.
   - Key-in the distance, then press Enter.
     Use ORTHO or Polar Tracking to control the angle.

To Move Entities Using Grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point for the move.
   The *Grip Stretch* command is launched.
4. Do one of the following:

   - Right click, then select *Move* in the context menu.
   - Hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Move* is active.

The *GRIP_MOVE* prompt box displays:



You are prompted: Enter move point: Base point/Copy/Undo/eXit:

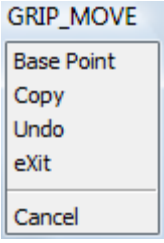
5. Do one of the following to define the displacement point:

   - Click in the drawing.
   - Enter the coordinates, then press Enter.
   - Key-in the distance, then press Enter.
     Use ORTHO or Polar Tracking to control the angle.

To copy entities using grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point for the move.
   The *Grip Stretch* command is launched.
4. Do one of the following:
   - Right click, then select *Move* in the context menu.
   - Hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Move* is active.

The *GRIP_MOVE* prompt box displays:



You are prompted: Enter move point: Base point/Copy/Undo/eXit:

5. Do one of the following

   - Choose *Copy* in the context menu.
   - Type *C*, then hit the *Space bar* or *Enter* key.

6. Do one of the following to define the displacement point:

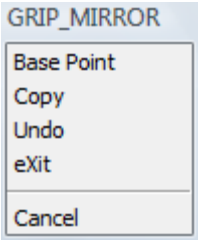
   - Click in the drawing.
   - Enter the coordinates, then press Enter.
   - Key-in the distance, then press Enter.
     Use ORTHO or Polar Tracking to control the angle.

7. Repeat the previous step to create more copies or press the Space bar or Enter key to stop.

To Mirror Entities Using Grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point for the mirroring.
   The *Grip Stretch* command is launched.
4. Do one of the following:
   - Right click, then select *Mirror* in the context menu.
   - Repeatedly hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Mirror* is active.

The *GRIP_MIRROR* prompt box displays.



You are prompted: Enter second point: Base point/Copy/Undo/eXit.

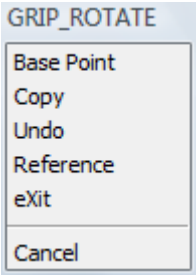
4. Specify a second point to define the mirror axis.

The selection is mirrored.

To Rotate Entities Using Grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point for the rotation.
   The *Grip Stretch* command is launched.
4. Do one of the following:
   - Right click, then select *Rotate* in the context menu.
   - Repeatedly hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Rotate* is active.

The *GRIP_ROTATE* prompt box displays.



You are prompted: Enter rotation angle: Base point/Copy/Undo/eXit.

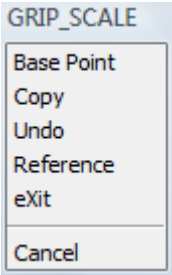
5. Do one of the following to rotate the selection:

   - Specify a point to define the rotation angle.
   - Type the rotation angle in the command bar, then press Enter

To Scale Entities Using Grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point for the scaling.
   The *Grip Stretch* command is launched.
4. Do one of the following:
   - Right click, then select *Scale* in the context menu.
   - Repeatedly hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Scale* is active.

The *GRIP_SCALE* prompt box displays.



You are prompted: Enter scale factor: Base point/Copy/Undo/eXit.

5. Type the scale factor in the command bar, then press Enter to scale the selection.

Selecting Entities

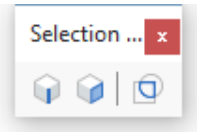
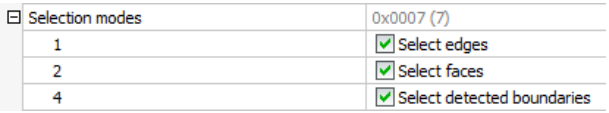
Selection settings sit in the *Selection* settings group of the *Program Options* settings category in the Settings dialog.

Selection	
Pick add	☑ Add selected entities to current selection set
Pick automatic	0x0003 (3)
negative	☐ Disables window selection, saving earlier value
1	☑ Draws a selection window if mouse is not over any entity when clicked
2	☑ Draws a selection window if mouse is clicked over an entity and dragged
Pick box	4
Pick drag	[0] Draw selection window using two points
Pick first	[1] First select entities, then issue a command
Pick style	0x0001 (1)
1	☑ Group selection
2	☐ Associative hatch selection (Not yet supported)
Match options for SELECTSIMILAR	0x0082 (130)
Selection area	
Selection area	☑ Show selection area effects
Selection area opacity	25
Crossing area color	91
Window area color	150
Selection preview	
Selection preview display	0x0003 (3)
1	☑ When no commands are active
2	☑ When a command prompts for entity selection
Selection preview effect	[2] Dashed and thickened lines
Selection preview filter	0x0005 (5)
1	☑ Exclude entities on locked layers
2	☐ Exclude entities in xrefs
4	☑ Exclude tables
8	☐ Exclude multiline text entities
16	☐ Exclude hatch entities
32	☐ Exclude entities in groups
Selection modes	0x0000 (0)
1	☐ Select edges
2	☐ Select faces
4	☐ Select detected boundaries
Delay to preview selection	30
Constraint Bar Display	0x0003 (3)
1	☑ Displays constraint bars when geometrical constraints are added.
2	☑ Displays hidden constraint bars when constrained entities are selected.
Dynamic Constraint Mode	☐ Display hidden dimensional constraints when the constrained entities are selected.

Name	System Variable	Description
Highlight	HIGHLIGHT	Determines wether or not entities highlight on screen when selected.
Image highlight	IMAGEHLT	Determines whether the entire raster image highlights or only the raster image frame when selected.
Pick add	PICKADD	Controls whether subsequent selections replace the current selection set or add to it. *ON*: The selection set is extended if you select additional entities. Press and hold the Shift key to remove entities from the selection set. *OFF*: You cannot add entities to a selection set. The newly selected entity or entities replace the existing

		selection set. However, if you press and hold the *Shift* key, you can add entities. If you select an entity that was already selected while pressing the *Shift* key, this entity is removed from the selection set.
Pick automatic	PICKAUTO	Controls automatic window selection (inside or crossing) when selecting entities. The value is stored as a bitcode using the sum of the selected options. • negative: Disables window selection, saving current value. • 1 = draws a selection window if the mouse is over a blank area of the drawing when clicked. • 2 = draws a selection window if the mouse is over an entity when dragged.
Pick box	PICKBOX	Defines the size of the small square at the end of the selection cursor ( ) . If you select an entity by clicking the *Pick Box* must touch or overlap the entity. The default size of the *Pick Box* is 3.
Pick drag	PICKDRAG	Controls the method of drawing a selection window. *ON*: Allows to define a selection window by dragging:press and hold the left mouse button to define the first corner of the rectangle, then move the mouse to define the size of the selection window and release the mouse button to define the opposite corner. *OFF*: Define the selection window by clicking two opposite corners.
Pick first	PICKFIRST	Controls whether you select objects before or after you issue a modification command. *ON*: Allows to first compose a selection set, then launch a modification command. *OFF*: You must first start the command, then compose the select entities.
Pick style	PICKSTYLE	Controls the selection of groups and associative hatches. • 0 = No group selection, nor associative hatch selection. • 1 = Group selection: if a member of a selectable group is selected, all members of the group are selected. • 2 = Associative hatch selection: the hatch and its boundary are selected, no matter what is picked: the hatch or the boundary (not supported yet). • 3 = Group selection and Associative hatch selection.

Match options for SELECTSIMILAR	SELECTSIMILARMODE	Controls which properties must match for an entity of the same type to be selected with the SELECTSIMILAR command. ⊟ Match options for SELECTSIMILAR — 0x0082 (130) 1 — ☐ Color 2 — ☑ Layer 4 — ☐ Linetype 8 — ☐ Linetype scale 16 — ☐ Lineweight 32 — ☐ Plot style 64 — ☐ Object style 128 — ☑ Name
Selection area	SELECTIONAREA	Controls the display of selection area effects.
Selection area opacity	SELECTIONAREAOPACITY	Controls the transparency of the selection area: the higher the value the more opaque the selection area.
Crossing area color	CROSSINGAREACOLOR	Specifies the color for crossing selection areas.
Window area color	WINDOWAREACOLOR	Specifies the color for window selection areas.
Selection preview display	SELECTIONPREVIEW	Controls when entities highlight when the cursor hovers over them: • 0 = never; • 1 = when no commands are active; • 2 = when a command prompts for entity selection; • 3 = both, when no commands are active and when a command prompts for entity selection. When the QUAD cursor menu is active, SELECTIONPREVIEW is neglected, unless a command prompts for entity selection.
Selection preview effect	PREVIEWEFFECT	Controls the appearance of highlighted entities: 0=dashed lines; 1=thickened lines; 2=dashed and thickened lines (not yet supported).
Selection preview filter	PREVIEWFILTER	Controls which entity types are excluded from selection previewing. The options are: • Entities on locked layers • Entites in Xrefs • Tables • Mtext entities • Hatches • Groups
GL Selection Highlight Style	HIGHLIGHTEFFECT	Specifies the highlight method: 0=dashed lines; 1=use different color; 2=use thickened line; 3=use different color and thickened line.
Selection Highlight color	HIGHLIGHTCOLOR	Specifies the highlight color to be used when HIGHLIGHTEFFECT is 1 or 3.
Selection Modes	SELECTIONMODES	Defines which subentities should highlight in selection preview. The variable is stored as a bitcode using the sum

		of the values of all selected options. Click the buttons on the *Selection Modes* toolbar to set the value of SELECTIONMODES. A pressed button indicates the option is selected.   • 1 = Enable detection of 3D solid edges ( ) • 2 = Enable detection of 3D solid faces ( )* • 4 = Enable boundary detection ( ). Controls whether closed boundaries in XY-plane of the current coordinate system or on the face of 3D solids highlight when no commands are active. The BOUNDARYCOLOR system variable specifies the color for highlighting detected boundaries. * Hold down the Ctrl-key to invert the status of the *Select Faces* option: if ON allows to select 3D solids, if OFF allows to select solid faces. Repeatedly pressing the TAB key, while keeping the cursor at the same position, highlights all included subentity types one by one. See also window selection.
Constraint Bar Display	CONSTRAINTBARDISPLAY	Controls the visibility of a geometric constraint at its creation or selection.
Dynamic Constraint Mode	DYNCONSTRAINTMODE	Controls the visibility of hidden dimensional constraints when the constrained entity is selected. When the cursor is over an entity with a dimensional constraint, the blue constraint glyph displays.
	User Preference	
Preview selection delay	PreviewDelay	Specifies how many milliseconds to wait before highlighting the (sub)entities under the cursor.

Grips Settings

If the *Grips* system variable is different from zero, grips display on selected entities.
Grips settings sit under *Display/Viewing* in the *Drafting* settings group of the *Drawing* settings category in the Settings dialog.

Grips can be used to manipulate entities (see Grips Editing)

Grips	
Grips	[1] Turn on endpoint and midpoint grips
Grips in blocks	☐ Assigns grips to entities within the block
Grip color	72
Dynamic grip color	140
Selected grip color	240
Hover grip color	150
Grip size	4
Grip object limit	100
Grip tips	☑ Display grip tips
Grips attraction	☑ Enable grips attraction
Grips attraction distance	4

Name	System Variable	Description
Grips	GRIPS	The value of GRIPS can be 0, 1 or 2 • 0: No grips. • 1: Display grips. Polyline endpoint and midpoint grips. • 2: Display grips. No midpoint grips on polyline segments.
Grips in blocks	GRIPBLOCK	Toggles the display of grips on entities in blocks on/off.
Grip color	GRIPCOLOR	Sets the color of unselected grips. Default value is green (index color no. 72).
Selected grip color	GRIPHOT	Sets the color of selected grips, so called 'hot grips'. Default value is red (index color no. 240).
Hover grip color	GRIPHOVER	Sets the color of an unselected grip when the cursor pauses over it. Default value is blue (index color no. 150).
Grip size	GRIPSIZE	Sets the size of the grip box, expressed in pixels. Default value = 4.
Grip object limit	GRIPOBJLIMIT	Suppresses the display of grips if the number of selected objects exceeds the value of GRIPOBJLIMIT. If set to 0 (zero) grips always display.
Grip tips	GRIPTIPS	(Not supported yet) Determines whether grip tips display when the cursor hovers hover grips on custom entities or dynamic blocks that support grip tips.
Grips attraction	EnableAttraction (*)	Toggles the magnetism of grips on/off.
Grips attraction distance	AttractionDistance (*)	Sets the grips attraction distance: if the cursor is within this distance the cursor is attracted by the grip and jumps to it.

(*) User Preference

Selection Methods

Before you start to use modification commands, you need to know how to compose a selection set. If the *PICKFIRST* variable is *ON* you can build the selection set before starting the command, else you are prompted to select the entities after you launched the modification command.

Using QUICK SELECT you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set.

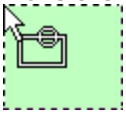
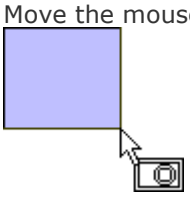
The SELECTIONPREVIEW and PREVIEWEFFECT system variables facilitate the selection procedure.

Composing a Selection Set Before (pre- pick)

To compose a selection set before launching a modification command do one or more of the following:

- Click an entity.
  The entity highlights and grips show.

  NOTE If the Quad cursor menu is active you must press and hold the Ctrl key while selecting entities.
  Click again to unselect.

- Window selection:
  - Click to define the first corner of a selection window.
    Move the mouse to the left to define a *Crossing Window*:
    
    All entities which overlap the window or are completely inside the window are added to the selection set.
    The selection window displays in dashed lines with a green background.
  - Click to define the first corner of a selection window.
    Move the mouse to the right to define a *Window-Inside*:
    
    All entities which are completely inside the window are added to the selection set.
    The selection window displays in continuous lines with a blue background.
  - Pressing the Ctrl key during window selection allows to select faces or edges of 3D solids and surfaces.
    The icon next to the cursor indicates the selection mode:
    - : Solids (Default)
    - : Faces
    - : Edges

  Depending on the value of the TIPS system variable, the Hotkey Assistant widget displays to indicate the current selection mode:

- Press an hold the *Shift* key, then use one of the above methods to select entities that you want to *remove* from the selection set.

NOTE If the *PICKADD* system variable is *OFF*, you cannot add entities to a selection set.

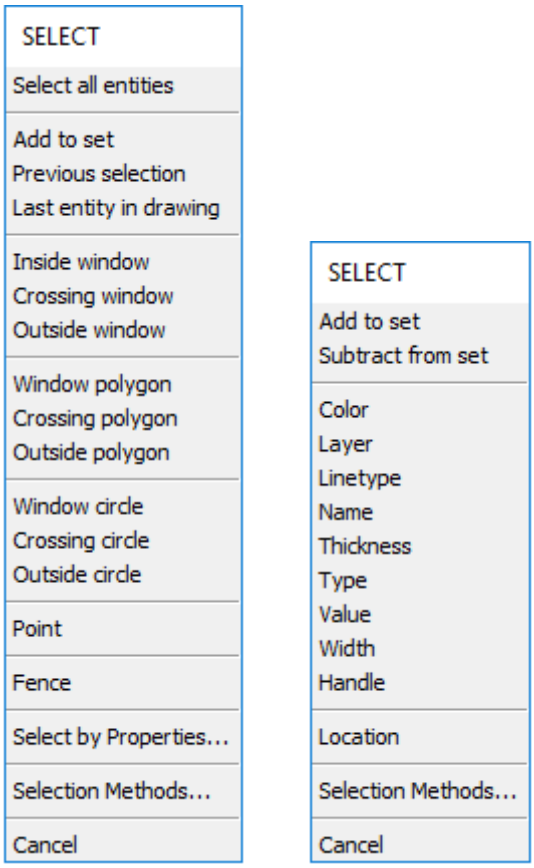
Composing a Selection Set Afterwards (post-pick)

Command: SELECT

If you launch a modification command when no selection set is active, you are prompted to select entities. BricsCAD provides a range of selection methods to let you compose your selection set easily.

The various selection methods are:

- *Add to set*: Selected entities are added to the selection set.
- *Subtract from set*: Selected entities are removed from the selection set.
- *Picking*: Place the pickbox over a part of the object and click.
- *Select all entities*: Select all entities in the entire drawing, including entities which are on hidden layers. Entities on frozen layers are not selected.
- *Previous selection*: Re-use the previous selection set.
- *Last entity in drawing*: Select the most recent entity.
- *Window*: All entities which are inside, overlap or are completely outside a window are selected. You can choose between a *rectangle*, *polygon* or a *circle*.
- *Fence*: All entities that cross a multi-segment line are selected.
- *Point*: Click a point to select all closed entities which enclose this point.
- *Select by Properties*: Displays the *Select by Properties* context menu to select entities by *Color*, *Layer*, *Linetype*, *Name*, *Thickness*, *Type*, *Value*, *Width* or *Handle*.
- *Undo*: Undoes the last selection action.
- *Location*: Displays the *Select Location* context menu.



Select Location and Select By Properties context menus

Selecting Overlapping Entities

When entities are close together or lie on top of each other it is difficult or sometimes impossible to select the entities needed.

Cycling through the overlapping entities, highlighting each entity one by one, helps you to select such entities.

Make sure SELECTIONPREVIEW is ON and/or the Quad cursor menu is active.

1. Place the cursor over the overlapping entities.
   One of the entities highlights.
2. Repeatedly hit the TAB key.
   The overlapping entities highlight one by one.
3. Click when the desired entity highlights.
4. Repeat the previous steps to select more entities.

NOTES

- Do not move the mouse while hitting the TAB key, because this will restart the highlight order.
- Hold down the Shift key to cycle the overlapping entities in reverse order.

Using Quick Select

TUTORIALS

Use smart filters to compose a selection set

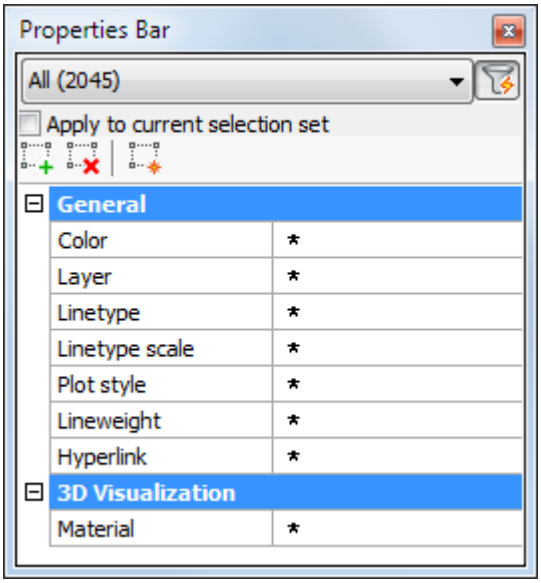
Using *Quick Select* you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set.

After composing the filter criteria you can choose to add the matching entities to a new selection set, add the entities to the current selection set or remove them from the current selection.

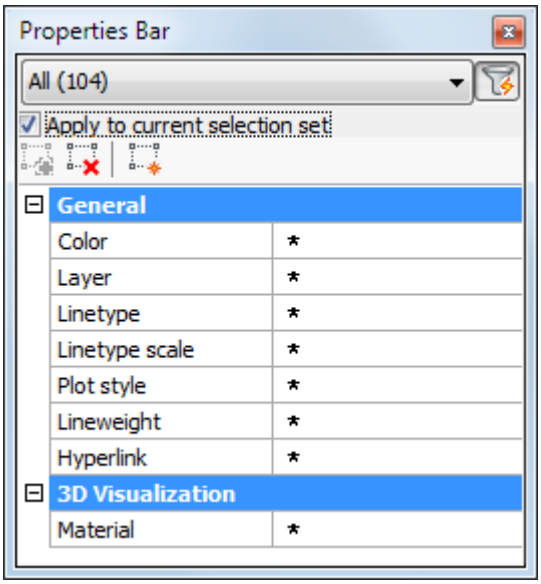
Quick Select applies to the entire drawing or to a previously created selection.

To create a selection set

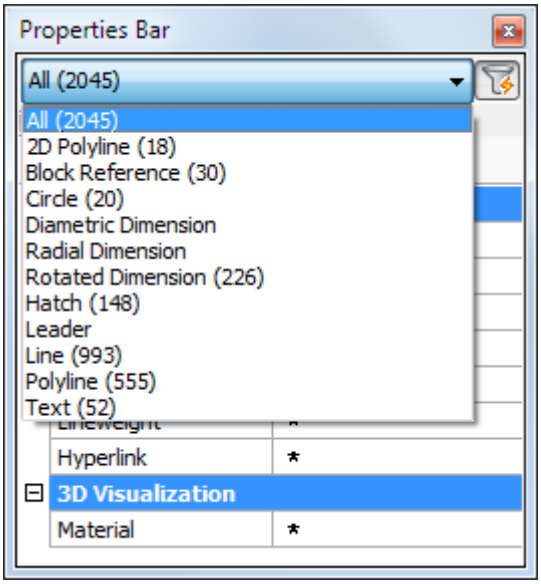
1. Click the *Quick Select* tool button ( ) on the Properties bar.
   The *Properties bar* mode is set to *Quick Select*.



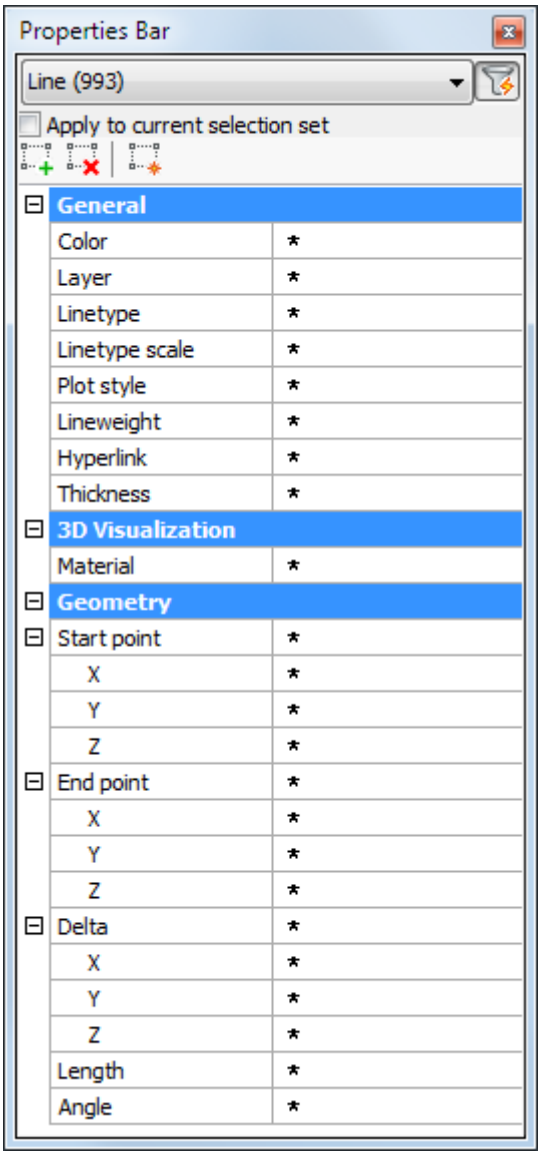
The *Apply to current selection set* option is checked if a selection set exists.



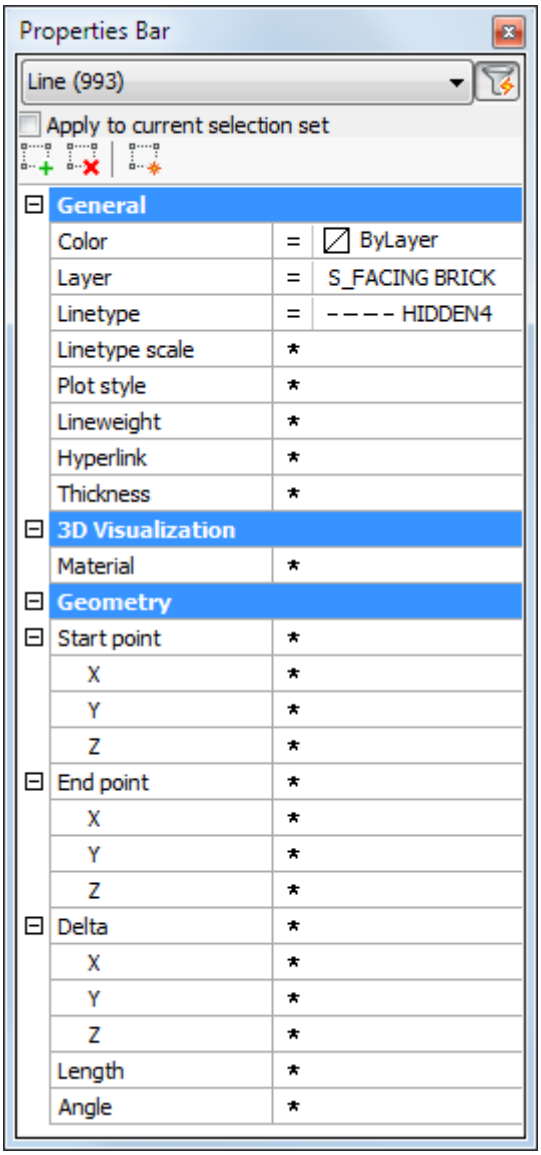
2. Click the *Selection List* button, to see a list of all entity types in the current drawing or current selection set (see previous step).
   A list of all entities in the drawing and the number of entities of each type displays.



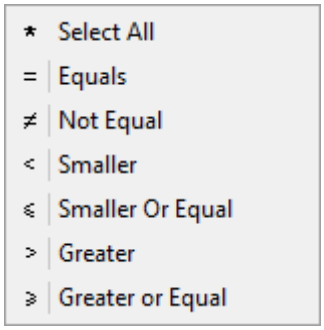
3. Select an entity type in the list, e.g. *Line*.
   Line properties show in the *Properties* Bar.


Properties Bar
Line (993)
Apply to current selection set
General
Color *
Layer *
Linetype *
Linetype scale *
Plot style *
Lineweight *
Hyperlink *
Thickness *
3D Visualization
Material *
Geometry
Start point *
X *
Y *
Z *
End point *
X *
Y *
Z *
Delta *
X *
Y *
Z *
Length *
Angle *

4. Specify the property filters, e.g. *Color*, *Layer* and *Linetype*.

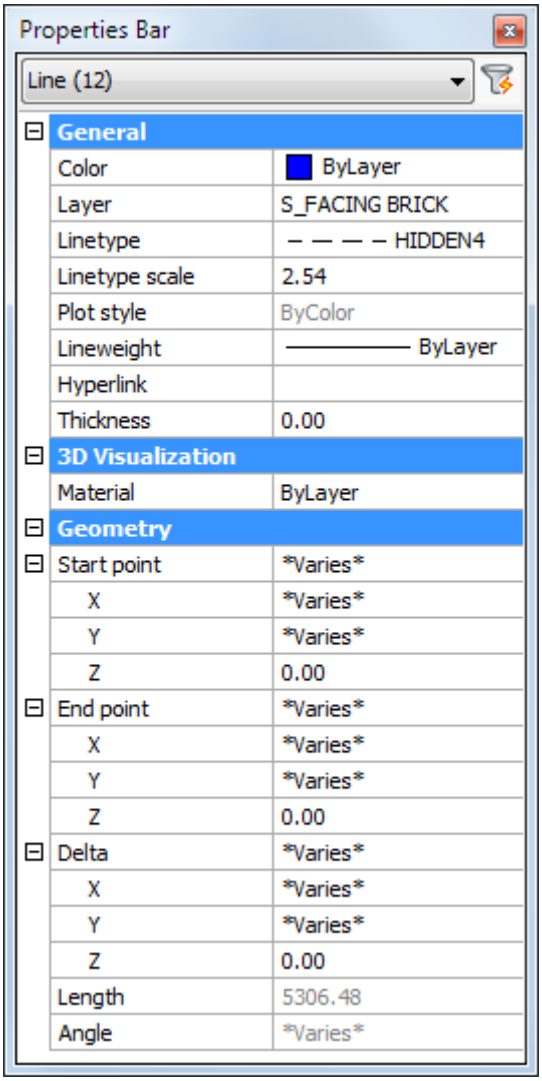


5. (option) Specify a different selection operator for each entity property:
   Click the = sign, then select an operator in the list.



7. Do one of the following:

- Click the *Add to new selection set* button ( ).
  The entities that match the selected criteria are selected.



- Click the *Add to the current selection set* button ( ).
  The entities that match the selected criteria are added to the current selection.
- Click the *Remove from current selection* set button ( ).
  The entities that match the selected criteria are removed from the current selection.

8. (option) Repeat the above procedure to further refine the selection set.

The Structure Panel

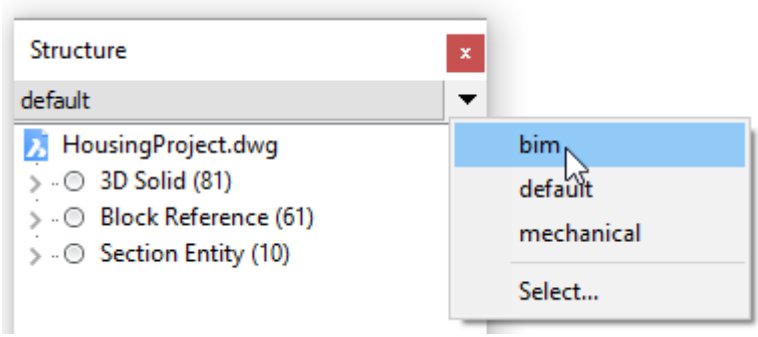
The dockable *Structure* panel displays a structured tree view of the drawing content. This structured view is fully configurable and configurations can easily be saved and recalled. The *.cst configuration files are stored in the Support folder by default. Entities selected in the structure tree can optionally be simultaneously highlighted and selected in the drawing, and vice versa, when selecting entities in the drawing they optionally can become selected and highlighted in the structure tree.

To open the Structure panel

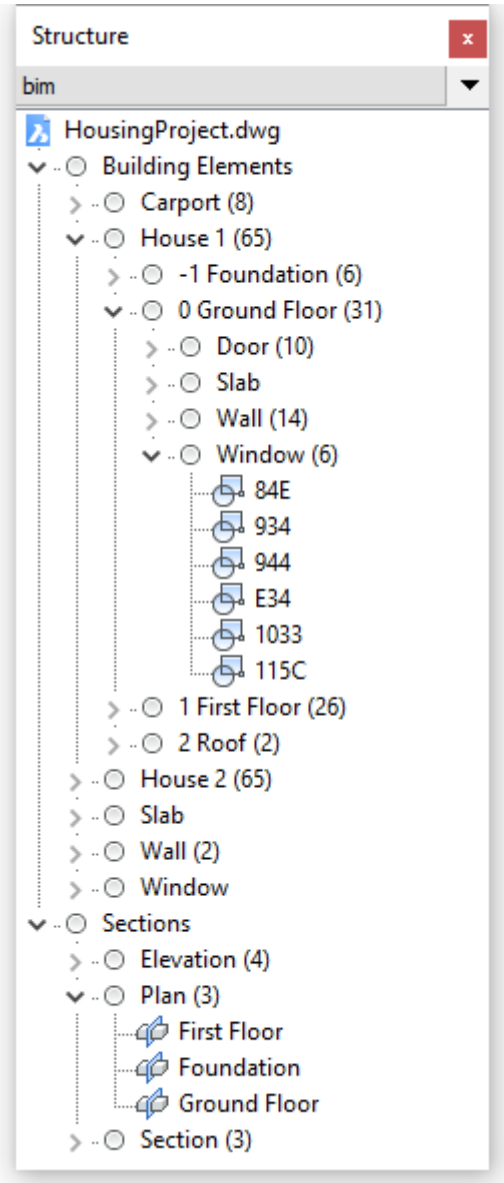
1. To open the *Structure* panel do one of the following:

- Place the cursor over a toolbar or ribbon panel, then right click and choose *Structure* in the context menu.
- Type *structurepanel* in the command bar, then press Enter.

2. Click the down-arrow button and load a configuration file in the list; e.g. *bim*.



The BIM model is organized in a tree-structure:



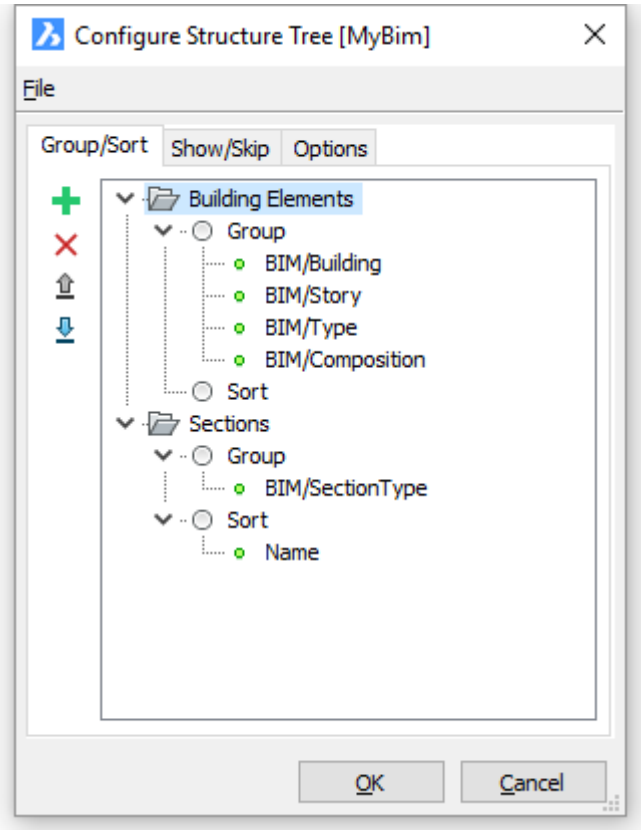
3. Do the following:

- Click the expand/collapse icon of a node to expand or collapse the node.
- Right click a node then choose *Expand All* or *Collaspe All* in the context menu to expand or collapse all branches.
  If you do this on the drawing name the whole tree structure expands or collapses.
- Select an entity in the tree. Depending on the selected option in the current configuration file, entities are either selected or highlighted in the model.
  e.g. if you click a story, all entities on that story are selected. You can then use IsolateObjects ( ) or HideObjects ( ) to isolate or hide the selected story in the model.
- Right click an entity, then choose *Zoom* in the context menu, to zoom onto the entities.
- Select an entity in the model: the entity is selected in the tree.
- ...

To configure the tree-structure on the Structure panel

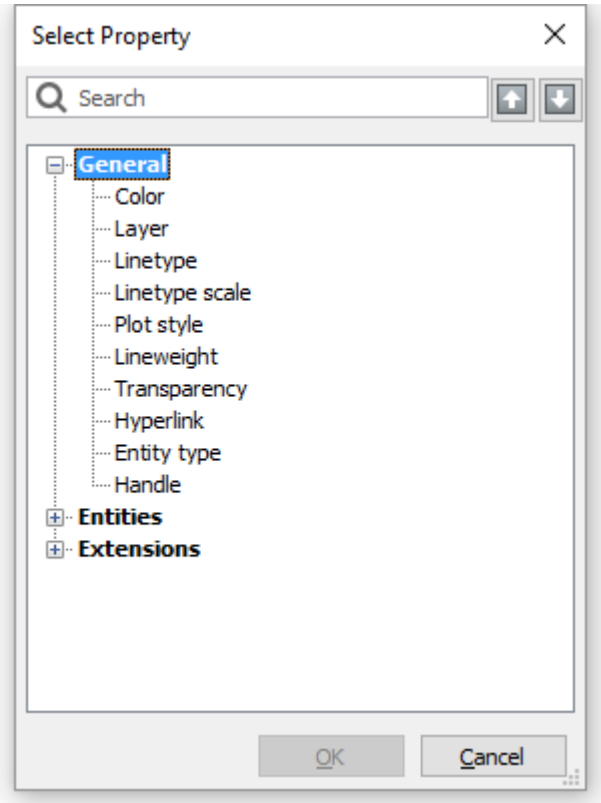
The tree structure on the *Structure* panel is a hierarchical set of rules. Each rule has a *grouping property* and a *sorting property*. The structure configurations are saved to *.cst files. By default *.cst files are saved in the *Support* subfolder of the Roamable Root folder (e.g. *C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V17x64\en_US\Support*). The name of the *. cst configuration file is saved in the drawing through the StructureTreeConfig user preference. BricsCAD searches for *.cst configuration files in the folders defined by the SRCHPATH configuration variable.

1. Right click a node, then choose *Configure* in the context menu.
   The *Configure Structure Tree* dialog box opens:



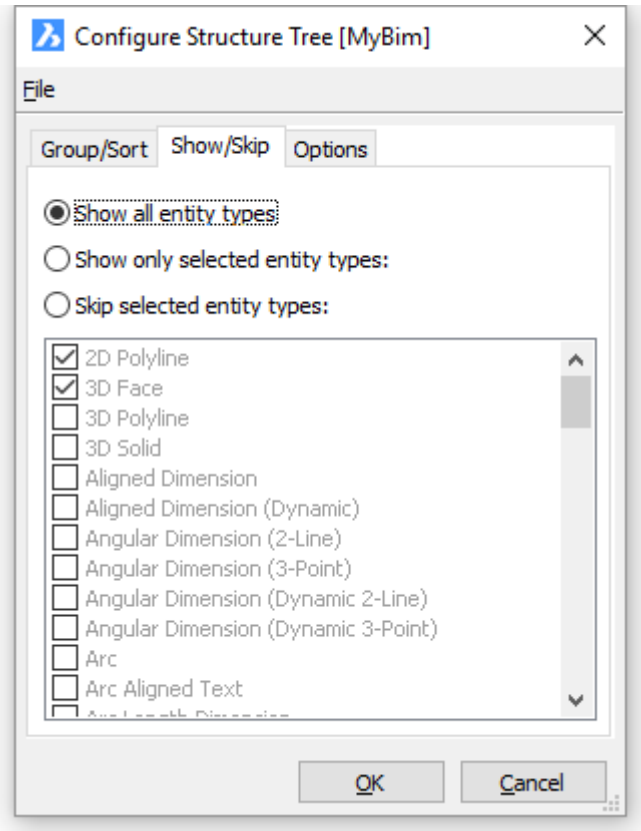
2. To add a rule, do the following:

- Select an existing rule, then click the *Add* icon (✚) or right click and choose *Add Rule* in the context menu.
  The new rule is added below the selected rule.
- Select the newly created group, then click the name or right click and choose *Rename* in the context menu and type a new name for the rule.
- Select the *Group* node of the rule, then click the *Add* icon (✚) or right click and choose *Add Grouping Property* in the context menu.
  The *Select Property* dialog box displays.



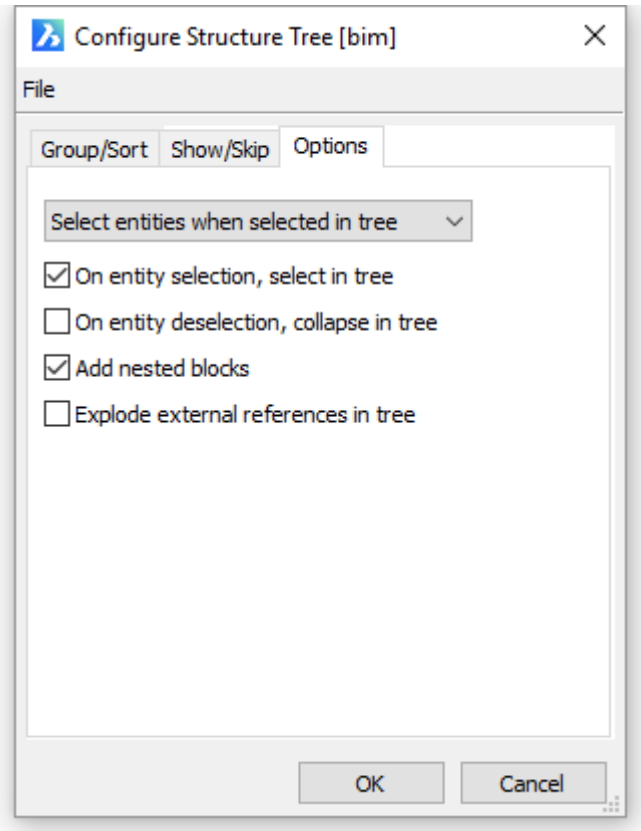
- Select a property, then double click or click the *OK* button.
  Optionally, type a search string in the *Search* field, then click the *Up* ( ) or *Down* ( ) arrow buttons to look up the properties containing the search string.
- Repeat the previous step to add more grouping properties.
- Select the Sort node, then click the *Add* icon ( ) or right click and choose *Add sorting property* in the context menu.
  The *Select Property* dialog box displays.
- Select a property, then double click or click the *OK* button.
  Optionally, type a search string in the *Search* field, then click the *Up* ( ) or *Down* ( ) arrow buttons to look up the properties containing the search string.
- Repeat the previous step to add more grouping properties.

3. Click the *Show/Skip* tab:



Select an option and the select the entity types to show or to exclude.

3. Click the *Options* tab:



- Choose what happens in the model when entities are selected in the structure tree:

  Ignore tree selection
  Highlight entities when selected in tree
  Select entities when selected in tree

- Select what happens in the structure tree when an entity is selected / deselected in the model:

  ☑ On entity selection, select in tree
  ☐ On entity deselection, collapse in tree
  ☑ Add nested blocks
  ☐ Explode external references in tree

4. (option) To move an item in the tree: select the item, then click the *Move Up* ( ) or *Move Down* ( ) button.
5. Do one of the following:

- Click the *OK* button to save the configuration and close the dialog box.
- Click the *File* menu on the dialog box and choose an option:

  Open...
  Save
  Save As...

  Open...: opens a file dialog to select another *.cst configuration file.
  Save: save the current configuration and keep the file dialog open.
  Save As...: save the current configuration under a different name and keep the file dialog open.

Rearranging Entities

Draw Order

Commands: DRAWORDER, HATCHTOBACK, TEXTTOFRONT

If new entities overlap existing entities, they will display and print on top of the previously drawn entities. The *Draw Order* tool can change the order in which overlapping entities are displayed and printed. You can move entities to the front or the back, or above or under another entity.

Using Draw Order

1. Do one of the following:
   - Click the *Draw Order* tool button ( ) on the *Draw Order* toolbar.
   - Choose *Draw Order* in the *Tools* menu.
   - Type *draworder* in the command bar.

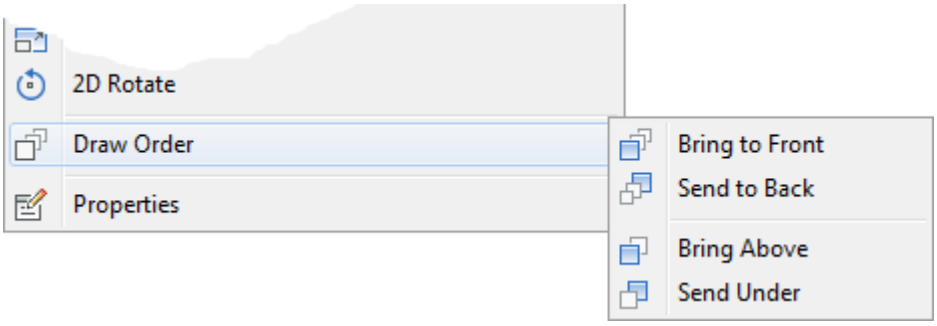
You are prompted: Select entities to change the draw order:

2. Select the entities, then right click or press enter.
   You are prompted: Change draw order: Above/Under/Clear/Front/<Back>:
3. Choose the appropriate draw order tool option.
4. Depending on your choice:

   - If you have chosen *Above* or *Under*, the command bar reads: Select reference entity: Identify the reference entity. The display of the selection set is changed with respect to the reference entity.
   - If you have chosen *Front* or *Back*, the display of the selection set is changed accordingly.

Using Draw Order tools

1. Select the entities you want to change the draw order of.
2. Do one of the following.
   - Click the appropriate tool button on the *Draw Order* toolbar.
   - Right click, then choose *Draw Order* in the context menu and select the appropriate tool.



Tool Button	Tool Name	Description
	Bring To Front	Places the selection set on top.
	Bring Above	Places the selection set above the reference entity. (*)
	Send Under	Places the selection set under the reference entity. (*)
	Send to Back	Moves the selection set to the back.
	Hatch to Back	Sends all hatch entities to the back.
	Text and Dimensions to Front	Brings all texts and/or dimension entities to front. Options: *Text / Dimensions / Both* can be chosen either in the command bar or in the prompt menu. The default option is *Both*.

(*) You are prompted to select the reference entity.

Manipulator

Command: MANIPULATE

The manipulator is a versatile grip tool to swiftly move, rotate, scale and mirror selected entities. Holding down the Ctrl-key when starting the action keeps the original entities, thus creating a copy. Upon selecting an entity, the manipulator is displayed at a user-controlled location and orientation. The location and orientation are controlled in a similar way as for a dynamic UCS. When multiple entities are selected simultaneously, the manipulator initially is placed at their geometric center. When a selection is built up step by step, the initial location of the manipulator is preserved. The manipulator can easily and directly be dragged to any desired location or orientation using its handles.

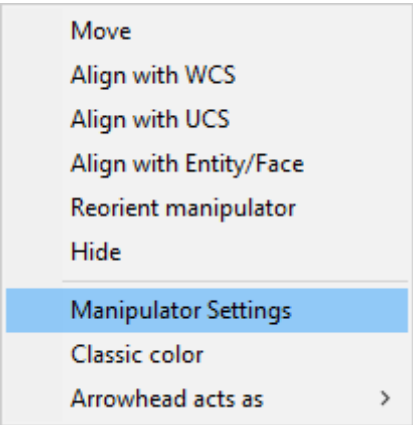
Depending on the operation type and if dynamic dimensions are active, a dynamic entry field is available to set the distance, angle or scale factor.

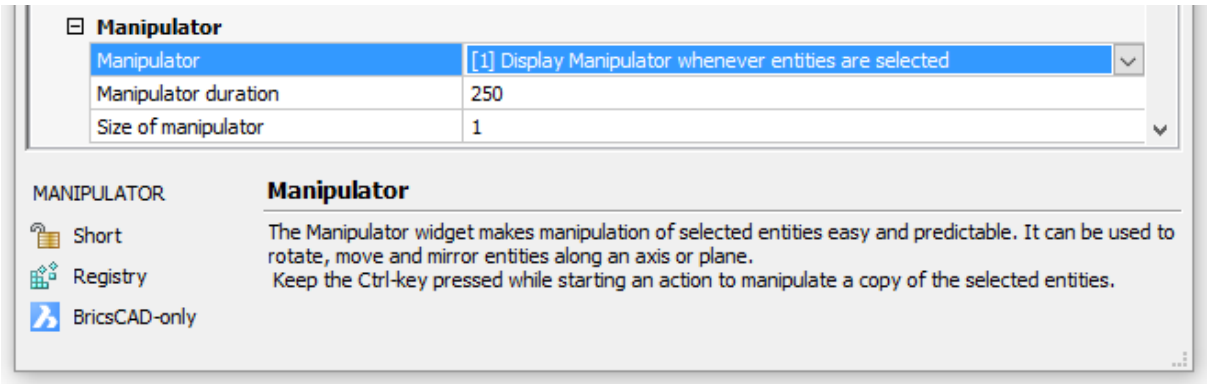
When the current view is parallel to one of the main axes of the coordinate system, as in a plan view, the manipulator is simplified to a 2D version.

The manipulator settings

To display the manipulator settings in the Settings dialog:

1. Right click the manipulator.
2. Choose Manipulator Settings in the context menu.





Name	Description
Manipulator	Controls the display of the manipulator widget. 0: The manipulator is not displayed. 1: Display the manipulator whenever entities are selected. 2: Display the manipulator is the left mouse button was pressed longer than the current value of MANIPULATORDURATION.
Manipulator duration	Specifies how long the left mouse button should be pressed during entity selection to trigger the display of the manipulator. The value is expressed in milliseconds.
Size of manipulator	Specifies the size of the manipulator relative to the default. Values between 0.5 and 2.0 are accepted.

To control the location and orientation of the manipulator

The initial location and orientation of the 3D manipulator tripod depends on:

- The selected entity type
- The point the entity is selected with

Entity Type	Selection point	Manipulator	
		Anchor point	Orientation
Line	Near endpoint	At endpoint	X-axis through the other endpoint
	Near midpoint	At midpoint	X-axis through the nearest endpoint
Polyline Line Segment	Near vertex	At vertex	X-axis through the other vertex of the selected line segment
	Near segment midpoint	At the segment midpoint	X-axis through the nearest vertex
Polyline Arc Segment	On the arc segment	At the arc center	X-axis through the nearest vertex
Arc	Random	At the arc center	X-axis through the nearest endpoint
Circle	Random	At the circle center	X-axis Parallel with the X-axis of the current coordinate system
Spline	Random	At the nearest endpoint	Through the other endpoint
Region	On a line segment	At the nearest endpoint of the line segment	X-axis through the other endpoint

	On a arc segment	At the center of the arc segment	X-axis through the nearest endpoint of the arc segment
Text, MText Block, Xref	Random	At the origin of the entity	X-axis at the rotation angle of the entity.
3D Solid* 3D Solid Face*	On a face of the solid	At the nearest vertex	X-axis along the nearest edge XY-plane in the selected face.
Multiple Entities	Random	At the geometric center of the selection set**	Parallel with the axes of the current coordinate system

* See Selection Modes to learn how to select a 3D solid face or the entire solid.

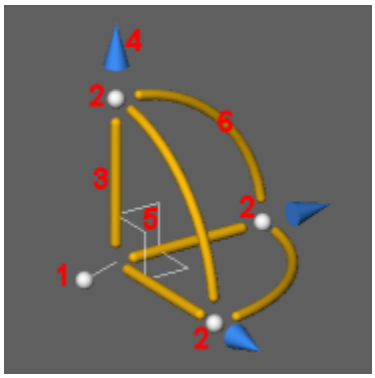
** The position of the manipulator does not change when more entities are added to the selection set.

To display the manipulator when multiple entities are selected

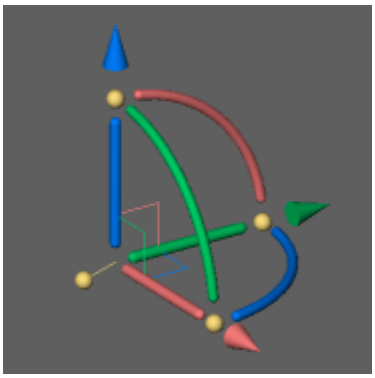
Do one of the following:

- Select the entities one by one or use window selection, then click to select the last entity
- Set the value of the MANIPULATOR system variable to 1: *Display the manipulator whenever entities are selected*.
- Compose the selection set, then launch the MANIPULATE command.

The manipulator layout



Default Manipulator



Classic Manipulator

4 handles: 1 anchor handle (1) and 3 axis handles (2).

12 command trigger zones: 3 axes (3), 3 arrowheads (4), 3 plane swatches (5) and 3 rotation arcs (6)

To move the manipulator

Do one of the following:

- Drag the manipulator by its anchor handle (1). Specify the new location using entity snaps. The anchor handle stands out to avoid overlap with entity grips.
- Place the cursor over the manipulator then right click and choose *Move* in the context menu. Specify the new location using entity snaps

To rotate the manipulator

Drag one of its axis handles (2) to rotate the manipulator. Pick a point or type a value in the dynamic entry field to specify the rotation angle.

To reorient the manipulator

1. Place the cursor over the manipulator, then right click and choose *Reorient* in the context menu.
   You are prompted: Specify origin of manipulator:
2. Pick a point.
   The anchor point of the manipulator is placed at the specified point.
   You are prompted: Point on X-axis or <Accept>:
3. Pick a point.
   The manipulator rotates about its X-axis.
   You are prompted: Point on the XY-plane with positive Y value or <Accept>:
4. Pick a point to define new position of the manipulator.

To align the manipulator with a coordinate system

Move the cursor over the manipulator, then right click and select an option in the context menu:

- **Align with WCS**: the manipulator axes are forced parallel with the corresponding axes of the World Coordinate System
- **Align with UCS**: if a UCS is currently the active coordinate system, the manipulator axes are forced parallel with the corresponding axes of the UCS.

To align the manipulator with the face of a solid

1. Place the cursor over the manipulator then right click and choose *Move* in the context menu.
2. Hover over the face of a solid.
   The XY-plane of the manipulator aligns with the solid face under the cursor.
3. Do one of the following:
   4. Snap to a point on the selected face or on one of its edges.
   5. Hit the Shift-key to lock the XY-plane to the selected face, then snap to a point outside the face.

Moving Entities

Commands: MOVE, CUTCLIP, PASTECLIP, PASTEORIG and PASTEBLOCK

The *Move* command moves entities in the drawing.

The *Cutclip* command copies entities to the *Clipboard*, and then erases them from the drawing.

The *Pasteclip* command pastes entities from the *Clipboard* into the current drawing (short for "paste clipboard").

The *Pasteorig* command pastes entities from the *Clipboard* into another drawing, using the coordinates of the source drawing.

The *Pasteblock* command pastes entities from the *Clipboard* as a block into the current drawing.

Use *Cutclip* then either *Pasteclip, Pasteorig* or *Pasteblock* to move entities to another drawing.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

NOTE When DRAGSNAP is ON, dragged entities display at the current entity snap location.

Moving entities in a drawing

The default method is to create a selection set and then specify a starting point (base point0 and an endpoint (displacement point) to define the relocation of the entities. You can also move the entities using a direction vector.

Some entities can be moved using grips. The grip you select depends on the type of entity. For example, to move a line entity, select the midpoint grip. To move a curved entity, such as an arc, circle, or ellipse, select the center point grip. Not all entities can be moved using grips.

1. Do one of the following:
   - Click the *Move* tool button ( ) on the *Modify* toolbar.
   - Choose *Move* in the *Modify* menu.
   - Type *move* or *M* in the command bar.

You are prompted: Select entities to move:

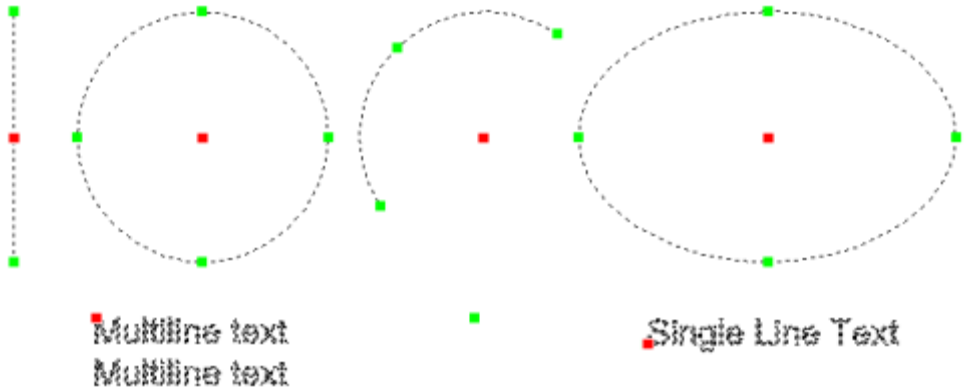
2. Select the entities, then right click or press Enter.
   You are prompted: Vector/<Base point>:
3. Specify the base point.
   The selection set is now attached to the cross hairs.
   You are prompted: Displacement point.
4. Do one of the following to specify the displacement point.

   - Click the displacement point.
   - Use Direct Distance Entry: type the displacement distance, then press Enter.
     The distance is measured in the cursor direction.
     Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

The selection set is moved.

Moving Entities Using Grips

1. Select the entity.
2. Click the grip.
   The selected grip displays in the *Selected Grip Color* (defined by the *GRIPHOT* system variable, default = RED).
   The entity moves with the cross hairs.



3. Click to relocate the entity.

NOTE The following entities can be moved using grips:

- lines, infinite lines and rays
- circles and circular arcs
- ellipses and elliptical arcs
- 3D solids
- text and multi-line text
- blocks

Moving entities between drawings

Entities can be moved between drawings in one of three ways:

Use *CUTCLIP* to cut the entities from the source drawing, then either *PASTECLIP, PASTEORIG* or *PASTEBLOCK* to paste the entities into the target drawing.

NOTE Use Ctrl + Tab (press and hold the Ctrl key, then press the Tab key) to cycle between open drawings.

1. Do one of the following:
   - Right click, then choose *Cut* in the context menu.
   - Choose *Cut* in the *Edit* menu.
   - Press *Ctrl + X* (press and hold the Ctrl key, then press X).

The entities are deleted in the source drawing and copied to the Clipboard.

3. Switch to the target drawing.
4. Do one of the following:

   - Right click, then choose *Paste* in the context menu.
   - Choose *Paste* in the *Edit* menu.
   - Press *Ctrl + V* (press and hold the Ctrl key, then press V).

The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.

5. Do one of the following:

   - Specify the displacement point.
   - Press Enter or choose *Paste to Original Coordinates* in the *Edit* menu to paste the selection set using the coordinates of the source drawing.

NOTE If you choose *Undo* in the *Edit* menu in the source drawing, the deletion of the selection set is undone.

Move a Selection as a Block

1. In the source drawing, select the entities you want to move.
2. Do one of the following:
   - Right click, then choose *Cut* in the context menu.
   - Choose *Cut* in the *Edit* menu.
   - Press *Ctrl + X* (press and hold the Ctrl key, then press X).

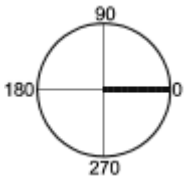
The entities are deleted in the source drawing and copied to the Clipboard.

3. (option) Switch to the target drawing.
4. Choose *Paste as a Block* in the *Edit* menu or press Ctrl+Shfit+V.
   The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.
5. Specify the insertion point.

Rotating Entities

You can rotate entities about a specified point at a specified rotation angle or by an angle referenced to a base angle. The default method rotates the entities using a relative rotation angle from their current orientation.

By default, angles start at 3 o'clock and increase in a counter-clockwise direction. If you want to rotate in a clockwise direction you can enter a negative angle by using a minus sign.



NOTES

- The *Angular Base* is controlled by the *ANGBASE* system variable, which in turn refers to the current UCS.
- The *Angle Direction* is controlled by the *ANGDIR* system variable, which sets the positive angle direction from angle 0 with respect to the current UCS.

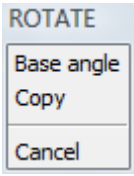
The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

Rotating a Selection Set

1. Do one of the following:
   - Click the *Rotate* tool button ( ) on the *Modify* toolbar.
   - Choose *Rotate* in the *Modify* menu.
   - Type *rotate* or *RO* in the command bar.

You are prompted: Select entities to rotate:

2. Select the entities, then right click or press Enter.
   You are prompted: Rotation point:
3. Specify the rotation point.
   You are prompted: Copy/Base angle/<Rotation angle>:
   A prompt menu displays:



4. (option) Choose *Copy* in the prompt menu or type *C*, then press Enter to create a rotated copy of the selection set.
5. Specify the rotation angle.
   The selection set is rotated.

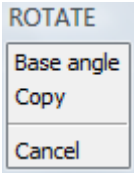
NOTE Use the *Center* option of the Array command if you want to keep the original entities.

Rotating a Selection Set in Reference to a Base Angle

1. Do one of the following:
   - Click the *Rotate* tool button ( ) on the *Modify* toolbar.
   - Choose *Rotate* in the *Modify* menu.
   - Type *rotate* or *RO* in the command bar.

You are prompted: Select entities to rotate:

2. Select the entities, then right click or press Enter.
   You are prompted: Rotation point:
3. Specify the rotation point.
   You are prompted: Copy/Base angle/<Rotation angle>:
   A prompt menu displays:



4. (option) Choose *Copy* in the prompt menu or type *C*, then press Enter to create a rotated copy of the selection set.
5. Do one of the following:

   - Choose *Base angle* in the prompt menu.
   - Type *B* in the command bar, then press Enter.

You are prompted: Base angle <0>:

6. To specify the base angle do one of the following:

   - Type the base angle in the command bar.
   - Click the rotation point again, then click a second point.

You are prompted: New Angle.

7. Do one of the following:

- Type the new angle in the command bar.
- Click a point to define the new angle.

Rotating in 3D

1. Do one of the following.

   - Click the *3D Rotate* tool button ( ) on the *Modify* toolbar.
   - Choose *3D Rotate* in the *Modify* menu.
   - Type *rotate3d* in the command bar.

You are prompted: Select entities to rotate:

2. Select the entities, then right click or press Enter.
   You are prompted: Select axis by [Entity/Last/View/Xaxis/Yaxis/Zaxis/2points] <2points>:
3. Press Enter to define the rotation axis by specifying two points.
4. Specify the first rotation axis point.
5. Specify the second rotation axis point.
   You are prompted: Rotation angle or [Base angle/Copy]:
6. Specify the rotation angle.
   The selection set is rotated clockwise, looking in the rotation axis direction.

Mirroring Entities

The *Mirror* command creates a mirror image of a selection set. The selection is mirrored about a mirror line, which you define by specifying two points. You can choose to either delete or retain the original entities.

The *Mirror3d* commands creates a mirror image of selected entities in three-dimensional space. In this case the selection is mirrored about a mirror plan, which can be defined by either specifying three points, selecting an existing two-dimensional planar entity, aligning the plane parallel to the xy, yz, or xz plane of the current UCS or aligning the plane with the current view. You can choose to either delete or retain the original entities.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

NOTE Whether text is mirrored or not by the *Mirror* command is controlled by the *Mirror Text* (*MIRRTEXT*) system variable.

Mirroring Entities about a Line

1. Do one of the following:

   - Click the *Mirror* tool button ( ) on the *Modify* toolbar.
   - Choose *Mirror* in the *Modify* menu.
   - Type *mirror* or *MI* in the command bar.

You are prompted: Select entities to mirror:

2. Select the entities then right click or press Enter.
   You are prompted: Start of mirror line.
3. Specify the start point of the mirror line.
4. Specify the endpoint of the mirror line.
   You are prompted: Delete the original entities? <N>
5. Do one of the following:

   - Press Enter to keep the original entities.
   - Type *Y*, then press Enter to delete the original entities.

- Choose *Yes-Delete entities* in the context menu.

Mirroring Entities about a Plane

1. Do one of the following:
   - Click the *3D Mirror* tool button ( ) on the *Modify* toolbar.
   - Choose *3D Mirror* in the *Modify* menu.
   - Type *mirror3d* in the command bar.

You are prompted: Select entities:

2. Select the entities then right click or press Enter.
   You are prompted: Define mirror plane by: Entity/Last/View/Zaxis/XY/YZ/ZX/<3points>:
3. Specify the first point of the mirror plane.
4. Specify the second point of the mirror plane.
5. Specify the third point of the mirror plane.
   You are prompted: Delete the original entities? <N>
6. Do one of the following:

   - Press Enter to keep the original entities.
   - Type *Y*, then press Enter to delete the original entities.
   - Choose *Yes-Delete entities* in the context menu.

Aligning Entities

The *Align* command lets you reposition a selection set with respect to an existing entity in the drawing. The selection set is moved and rotated in a single action. If necessary you can even scale the selection to fit in its new location.

The command can be used both in a 2D and 3D environment.

TUTORIALS

How to Align Entities

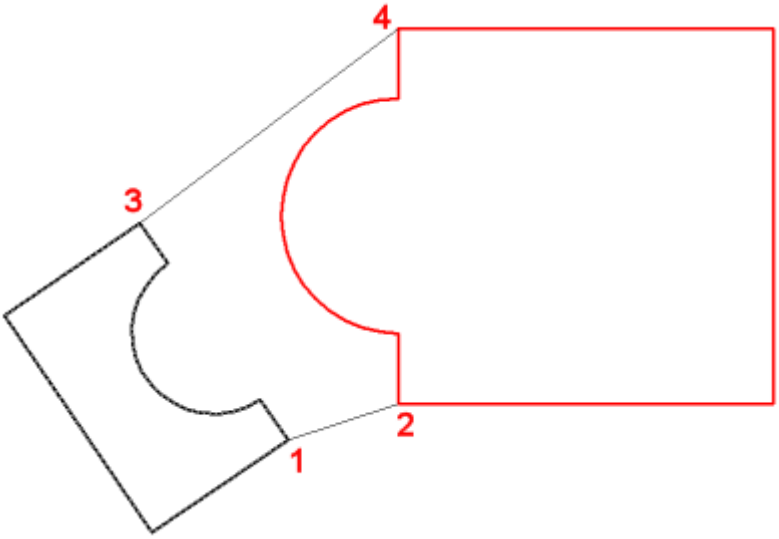
Aligning an Entity in 2D

1. Do one of the following:
   - Click the *Align* tool button ( ) on the *Modify* toolbar.
   - Choose *Align* in the *Modify* menu.
   - Type *align* in the command bar, then press Enter.

You are prompted: Select entities:

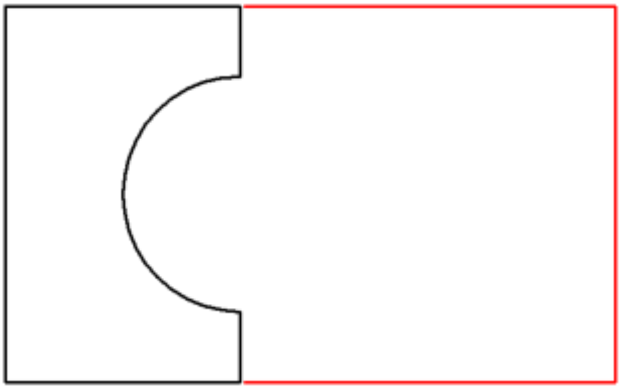
2. Select the entities, then right click or press Enter.
   You are prompted: Specify first source point:
3. Snap to the first source point in the selection set (1).
   You are prompted: Specify first destination point:
4. Snap to the first destination point on the reference entity (2).
   A witness line is drawn between the source point and the target point.

You are prompted: Specify the second source point:



5. Snap to the second source point in the selection set (3).
   You are prompted: Specify second destination point:
6. Snap to the second destination point on the reference entity (4).
   A witness line is drawn between the source point and the target point.
   You are prompted: Specify the third source point:
7. Right click to skip the third source point.
   You are prompted: Scale objects based on alignment points [Yes/No] <No>:
8. Do one of the following:

   - Press Enter if you don't want to scale the selection set.
   - Type Y and press Enter to scale the selection set.



Result of the Align procedure with scaled selection set.

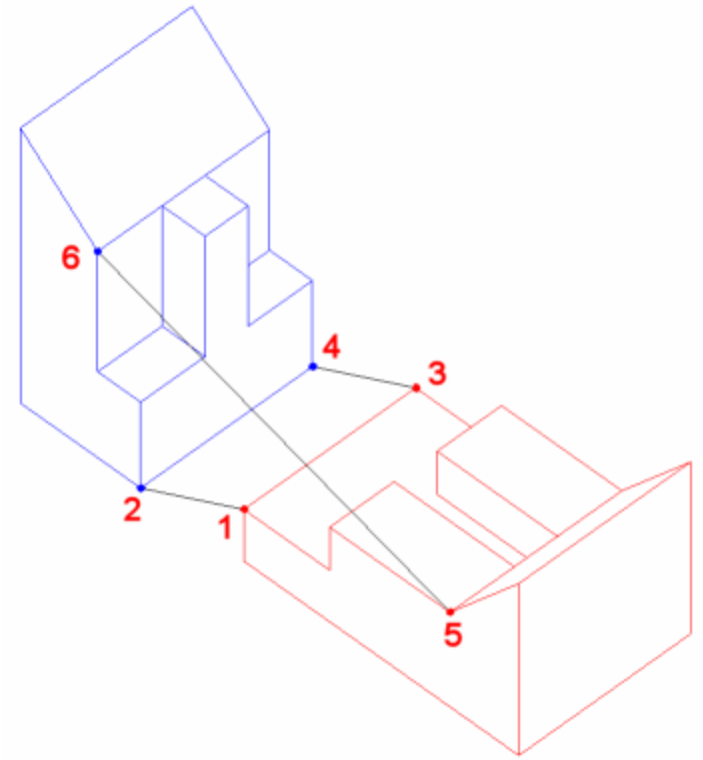
Aligning an Entity in 3D

1. Do one of the following:
   - Click the *Align* tool button ( ) on the *Modify* toolbar.
   - Choose *Align* in the *Modify* menu.
   - Type *align* in the command bar, then press Enter.

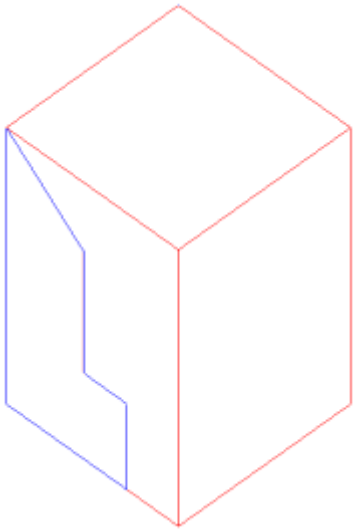
You are prompted: Select entities:

2. Select the entities, then right click or press Enter.
   You are prompted: Specify first source point:

3. Snap to the first source point in the selection set (1).
   You are prompted: Specify first destination point:
4. Snap to the first destination point on the reference entity (2).
   A witness line is drawn between the source point and the target point.
   You are prompted: Specify the second source point:



5. Snap to the second source point in the selection set (3).
   You are prompted: Specify second destination point:
6. Snap to the second destination point on the reference entity (4).
   A witness line is drawn between the source point and the target point.
   You are prompted: Specify the third source point:
7. Snap to the third source point in the selection set (5).
   You are prompted: Specify third destination point:
8. Snap to the third destination point on the reference entity (6).
   A witness line is drawn between the source point and the target point.
   The entity is moved.



Changing Entities

The *Change* command lets you change the insertion point and orientation of texts and block inserts.

The *Properties* option of the command can also change the layer, line type, line type scale, line weight, etc. but the BricsCAD Properties bar is much more versatile to do such things.

To Change Text Entities

1. Type *change* in the command bar, then press Enter.
   You are prompted: Select entities to change:
2. Select the text entities, then right click or press Enter.
   You are prompted: Change: Entities/Properties/<Change point>:
3. Right click or press Enter to accept the default option.
   You are prompted: New point for text, or Enter for no change:
   The first text entity is attached to the cursor.
4. Specify the new insertion point for the text or right click to accept the current location.
   You are prompted: New text style <current style>:
5. Type a new text style and press Enter or right click to keep the current style.
   You are prompted: New height <current height>:
6. Type a new height and press Enter or right click to keep the current height.
   You are prompted: New rotation angle <current angle>:
7. Type a new angle and press Enter or right click to keep the current angle.
   You are prompted: New text <current text>:
8. Type the new text and press Enter or right click to keep the current text.
9. (option) If multiple text entities were selected in step 2, steps 3 through 8 are repeated for each text.

NOTE The *Change* command cannot change Multi-Line text (Mtext) entities.

To Relocate Blocks

1. Type *change* in the command bar, then press Enter.
   You are prompted: Select entities to change:
2. Select the blocks, then right click or press Enter.
   You are prompted: Change: Entities/Properties/<Change point>:
3. Right click or press Enter to accept the default option.
   You are prompted: New point for block, or Enter for no change:
   The first block is attached to the cursor.
4. Specify the new insertion point or right click to accept the current orientation.
   You are prompted: New rotation angle <current angle>:
5. Do one of the following:

   - Right click to accept the current rotation angle.
   - Type a new rotation angle in the command bar, then press Enter.
   - Click to specify the new rotation angle graphically.

6. (option) If multiple blocs were selected in step 2, steps 3 through 5 are repeated for each block.

Copying Entities

Copying Entities in a Drawing

Commands: COPY, COPYCLIP, COPYBASE, PASTECLIP and PASTEBLOCK

You can duplicate entities within the current drawing using the *Copy* command. The default method is to create a selection set and then specify a starting point (base point) and an endpoint (second point) for the copy. You can also specify the displacement as a direction vector (X,Y,Z).

Depending on the *COPYMODE* system variable you can create multiple copies of the selection set or just one.

Copybase and *Pasteblock* pastes the selection as a block.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

To Copy a Selection Set

1. Do one of the following:
   - Click the *Copy* tool button ( ) on the *Modify* toolbar.
   - Choose *Copy* in the *Modify* menu.
   - Type *copy* or *CO* in the command bar.

You are prompted: Select entities to copy:

2. Select the entities, then right click or press Enter.
   If *Copy Mode* is *Multiple*:
   You are prompted: Enter base point: mOde/<Displacement>:
   If *Copy Mode* is *Single*:
   You are prompted: Enter base point: mOde/Multiple/<Displacement>:
3. Specify the base point.
   The selection set is now attached to the cross hairs.
   You are prompted: Enter second point: <Enter to use the base point as displacement>:

4. Do one of the following:

   - Click the second point.
   - Use Direct Distance Entry: type the displacement distance, then press Enter.
     The distance is measured in the cursor direction.
     Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.
   - Press <Enter>: the first point is interpreted as a relative X,Y,Z displacement.
     E.g. if the coordinates of the base point are 2,3,0, the selection set is copied 2 units in the X direction and 3 units in the Y direction from their current position.

The selection set is copied.

If *Copy Mode* is *Single*, the copy command is completed.

If *Copy Mode* is *Multiple*, you are prompted to create the next copy. See To make multiple copies.

To Make Multiple Copies

1. Do one of the following:
   - Click the *Copy* tool button ( ) on the *Modify* toolbar.
   - Choose *Copy* in the *Modify* menu.
   - Type *copy* or *CO* in the command bar.

You are prompted: Select entities to copy:

2. Select the entities, then right click or press Enter.
   If *Copy Mode* is *Multiple*:
   You are prompted: Enter base point: mOde/<Displacement>:

If *Copy Mode* is *Single*:
You are prompted: Enter base point: mOde/Multiple/<Displacement>:

3. (Option) If *Copy Mode* is *Single*, do one of the following:

   - Choose *Multiple* in the prompt menu
   - Type *M*, then press Enter

You are prompted: Enter base point: mOde/<Displacement>:

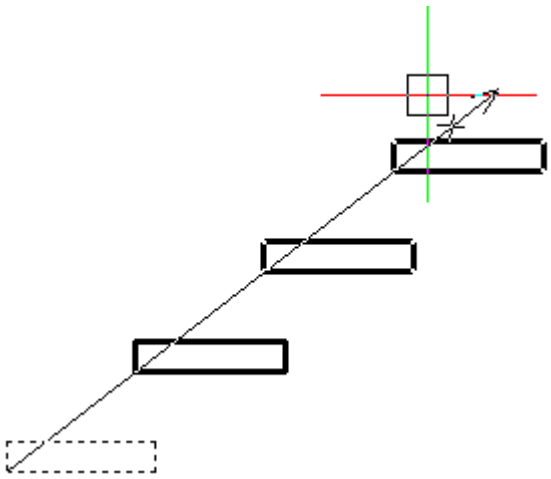
4. Specify the base point.
   The selection set is now attached to the cross hairs.
   You are prompted: Enter second point: <Enter to use the base point as displacement>:
5. Do one of the following to specify the second point.

   - Click the second point.
   - Type the displacement distance, then press Enter.
     The distance is measured in the cursor direction.
     Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

The first copy of the selection set is created.

You are prompted: Enter second point: Undo/Repeat/Exit:

6. (option) Repeat step 5 to create additional copies.
   Right click or press Enter to conclude the command.
7. (option) Choose the *Repeat* option.
   You are prompted: Select end point (<ENTER> to key in number of repeats)
   The number of repeats increases when you move the cursor away from the source selection set.



Do one of the following:

- Click when the desired number of copies displays.
- Press Enter or right click, then type the desired number of copies.

To Paste a Selection Set as a Block

1. Do one of the following:
   - Choose *Copy with Base Point* in the *Edit* menu.
   - Type *copybase* in the command bar, then press Enter.
   - Press and hold the *Ctrl* and *Shift* keys, then press *C*.

You are prompted: Select base point:

2. Specify the base point (origin point).
   You are prompted: Select entities to copy to the clipboard:
3. Select the entities.
4. Press Enter to conclude the selection of entities.
5. Do one of the following:

   - Choose *Paste as Block* in the *Edit* menu.
   - Type *pasteblock* in the command bar, then press Enter.
   - Press and hold the *Ctrl* and *Shift* keys, then press *V*.

The selection set is attached to the cursor.

The command window reads: Select insertion point.

6. Specify the insertion point.
   The selection is pasted as a block.
7. (option) Repeat steps 5 and 6 to paste another copy as a block.

NOTE If you paste multiple copies of the same selection set, each block is named differently.

Copying Entities Between Drawings

Commands: COPYCLIP, COPYBASE, PASTECLIP, PASTEORIG and PASTEBLOCK

Entities can be copied between drawings in several ways:

- *Copyclip* and *Pasteclip* pastes the selection using a default base point.
- *Copyclip* and *Pasteorig* pastes the selection using the coordinates of the source drawing.
- *Copybase* and *Pasteclip* pastes the selection using a user-defined base point.
- *Copybase* and *Pasteblock*: pastes the selection as a block in the target drawing.

NOTE Use Ctrl + Tab (press and hold the Ctrl key, then press the Tab key) to cycle between open drawings.

TUTORIALS

Copy Between Drawings

Using Copy and Paste

1. In the source drawing, select the entities you want to Copy.
2. Do one of the following:
   - Right click, then choose *Copy* in the context menu.
   - Choose *Copy* in the *Edit* menu.
   - Press *Ctrl + C* (press and hold the Ctrl key, then press C).

The entities are copied to the Clipboard.

3. Switch to the target drawing.
4. Do one of the following:

   - Right click, then choose *Paste* in the context menu.
   - Choose *Paste* in the *Edit* menu.
   - Press *Ctrl + V* (press and hold the Ctrl key, then press V).

The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.

5. Do one of the following:

   - Specify the displacement point.
   - Press Enter to paste the selection set using the coordinates of the source drawing.

Using Copy and Pasteorig

1. In the source drawing, select the entities you want to Copy.
2. Do one of the following:
   - Right click, then choose *Copy* in the context menu.
   - Choose *Copy* in the *Edit* menu.
   - Press *Ctrl + C* (press and hold the Ctrl key, then press C).

The entities are copied to the Clipboard.

3. Switch to the target drawing.
4. Do one of the following:

   - Choose *Paste to Original Coordinates* in the Edit menu.
   - Type *pasteorig* in the command bar, then press Enter.

The selection is pasted using the coordinates of the source drawing.

Using Copybase and Paste

1. In the source drawing do one of the following:
   - Choose *Copy with Base Point* in the *Edit* menu.
   - Type *copybase* in the command bar, then press Enter.
   - Press and hold the *Ctrl* en *Shift* keys, then press *C*.

You are prompted: Select base point:

2. Specify the base point (origin point).
   You are prompted: Select entities to copy to the clipboard:
3. Select the entities.
4. Press Enter to conclude the selection of entities.
5. Switch to the target drawing.
6. Do one of the following:

   - Choose *Paste* in the *Edit* menu.
   - Press *Ctrl + V* (press and hold the Ctrl key, then press V).

The selection set is attached to the cursor.

You are prompted: Select insertion point:

7. Specify the insertion point to paste the selection set.
8. (option) Repeat step 7 to create more copies of the selection set.

Paste as a Block

1. In the source drawing do one of the following:
   - Choose *Copy with Base Point* in the *Edit* menu.
   - Type *copybase* in the command bar, then press Enter.
   - Press and hold the *Ctrl* en *Shift* keys, then press *C*.

You are prompted: Select base point:

2. Specify the base point (origin point).
   You are prompted: Select entities to copy to the clipboard:
3. Select the entities.
4. Press Enter to conclude the selection of entities.
5. Switch to the target drawing.
6. Do one of the following:

   - Choose *Paste as Block* in the *Edit* menu.
   - Type *pasteblock* in the command bar, then press Enter.
   - Press and hold the *Ctrl* and *Shift* keys, then press *V*.

The selection set is attached to the cursor.

The command window reads: Select insertion point.

6. Specify the insertion point.
   The selection is pasted as a block.
7. (option) Repeat steps 5 and 6 to paste another copy as a block.

NOTE If you paste multiple copies of the same selection set, each block is named differently.

Making Parallel Copies

Command: OFFSET

The *Offset* command creates a copy of linear entities and aligns them parallel to the original entities at a specified distance. You can make parallel copies of arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays and infinite lines.

Depending on the selected options in the *Selection Modes* toolbar, the *Offset* command allows to create a parallel copy of:

- an edge of a 3D solid if the *Select Edges* option ( ) is set.
- the edges of a 3D solid face if the *Select Faces* option ( ) is set.
- detected boundaries if the *Select Detected Boundaries* option ( ) is set.

Making parallel copies of curved entities creates larger or smaller curves, depending on which side of the original entity you place the copy. For example, placing a parallel copy of a circle outside the circle creates a larger concentric circle; positioning the copy inside the circle creates a smaller concentric circle.

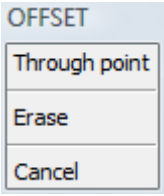
NOTE The *OFFSETGAPTYPE* settings variable controls how potential gaps between segments are treated when closed polylines are offset.

Making a parallel copy at a specified distance

1. Do one of the following:
   - Click the *Offset* tool button ( ) on the *Modify* toolbar.
   - Choose *Offset* in the *Modify* menu.
   - Type *offset* in the command bar.

You are prompted: Offset: Through point/Erase/<Distance> <current distance>:

A prompt menu displays:



2. (Option) Choose *Erase* in the prompt menu or type *E*, then press Enter.
   Respond *Yes* at the command prompt to erase the source entity.
3. Do one of the following:

   - Press Enter to accept the current distance.
   - Type a new distance in the command bar.
   - Define a new distance by specifying two points.

You are prompted: Select Entity.

4. Select the entity.
   You are prompted: Both sides/<Side for parallel copy>:
5. Click the side for the parallel copy.
   The parallel copy is created.
   You are prompted: Select Entity.
6. Do one of the following:

   - Repeat steps 4 and 5 to create more parallel copies.
   - Right click or press Enter to stop.

Making a parallel copy through a point

1. Do one of the following:
   - Click the *Offset* tool button ( ) on the *Modify* toolbar.
   - Choose *Offset* in the *Modify* menu.
   - Type *offset* in the command bar.

You are prompted: Offset: Through point/<Distance> <current distance>:

2. Choose *Through point* in the context menu or type *T*, then press Enter.
   You are prompted: Select entity:
3. Select the entity.
   You are prompted: Through point:
4. Specify the point to create the parallel copy through.
   The parallel copy is created.
   You are prompted: Select entity:
5. Do one of the following.

   - Repeat steps 3 and 4 to create more parallel copies.
   - Right click or press Enter to stop.

Making a parallel copy of a 3D solid edge

Make sure the Quad is active and the *Select edges* button ( ) in the *Selection Modes* toolbar is pressed.

1. Move the cursor over the edge of the solid.
   When the edge highlights, choose *Parallel* ( ) in either the *2D Editing* or the *Direct Modeling* command group in the *Quad*.
   The parallel copy is created dynamically.
2. Do one of the following:
   3. Type a distance in the dynamic entry field.
   4. Snap to a point.

Making a parallel copy a 3D solid face

Make sure the Quad is active and the *Select Faces* button ( ) in the *Selection Modes* toolbar is pressed.

1. Move the cursor over the face of the solid.
   When the face highlights, choose *Parallel* ( ) in either the *2D Editing* or the *Direct Modeling* command group in the *Quad*.
   The parallel copy is created dynamically.
2. Do one of the following:
3. Type² a distance in the dynamic entry field.
4. Snap to a point.

Making a parallel copy of a detected boundary

Make sure the Quad is active and the *Select Detected Boundaries* button ( ) in the *Selection Modes* toolbar is pressed.

1. Move the cursor inside the boundary.
   When the boundary highlights, choose *Parallel* ( ) in either the *2D Editing* or the *Direct Modeling* command group in the *Quad*.
   The parallel copy is created dynamically.
2. Do one of the following:
3. Type a distance in the dynamic entry field.
4. Snap to a point.

Arraying Entities

Commands: ARRAY , -ARRAY, ARRAYCLASSIC, ARRAYRECT, ARRAYPATH, ARRAYPOLAR, ARRAYEDIT, 3DARRAY

The *Array* commands copy a selection set in a rectangular or polar (circular) pattern or along a linear entity. For a rectangular array you must specify the number of rows and columns and also the spacing between subsequent rows and columns. To create a polar array you will be prompted to specify the center point of the array, the rotation step and the number of items in the array or the angle to fill. You can choose to rotate the selection set about the center point or to keep its original orientation.

The *ArrayClassic* command creates a static polar or rectangular array of entities through a dialog box.

The *-Array* command creates a static polar or rectangular array of entities.

The *3DArray* command constructs static 3D rectangular arrays and rotated polar arrays

The *Array* command distributes entity copies in an evenly spaced associative rectangular, polar or path array.

The *ArrayRect* command associatively distributes entity copies into any number of rows, columns, and levels.

The *ArrayPolar* command associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.

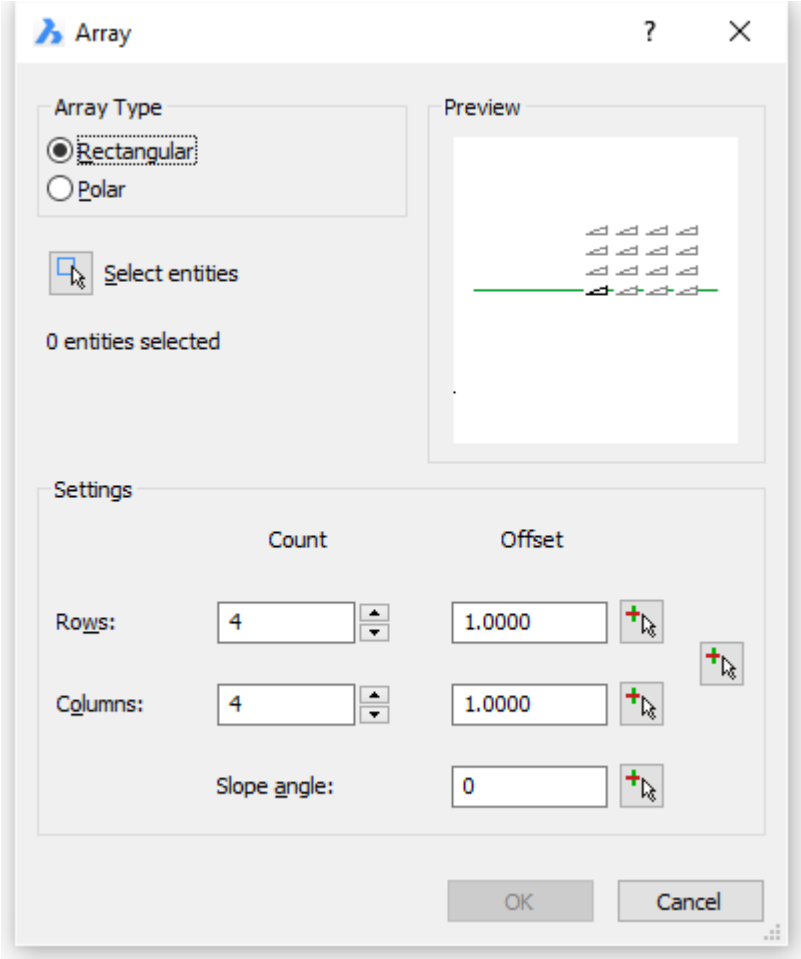
The *ArrayPath* command associatively distributes entity copies evenly along a path into multiple rows, and levels.

The *ArrayEdit* command allows editing associative arrays and their source entities.

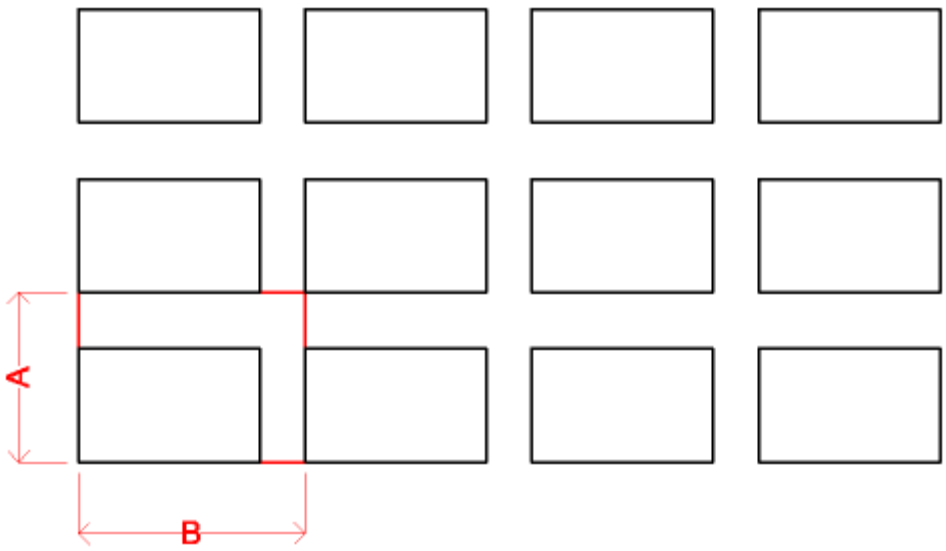
Creating a Rectangular Array

1. Type *arrayclassic* in the command bar, then press Enter.

A dialog box displays:



2. Under *Array Type* choose *Rectangular*.
3. Click the *Select Entities* button ( ).
   The *Array* dialog box closes temporarily to let you select entities.
4. Select the entities, then right click or press Enter.
   The *Array* dialog box reopens.
5. Fill out the *Settings* fields or press the *Pick Points* buttons ( ) to define the settings graphically.
   The *Array* dialog box closes temporarily to let you pick point in the drawing.



Row offset (A) and column offset (B) can be defined by the spacing rectangle (red).

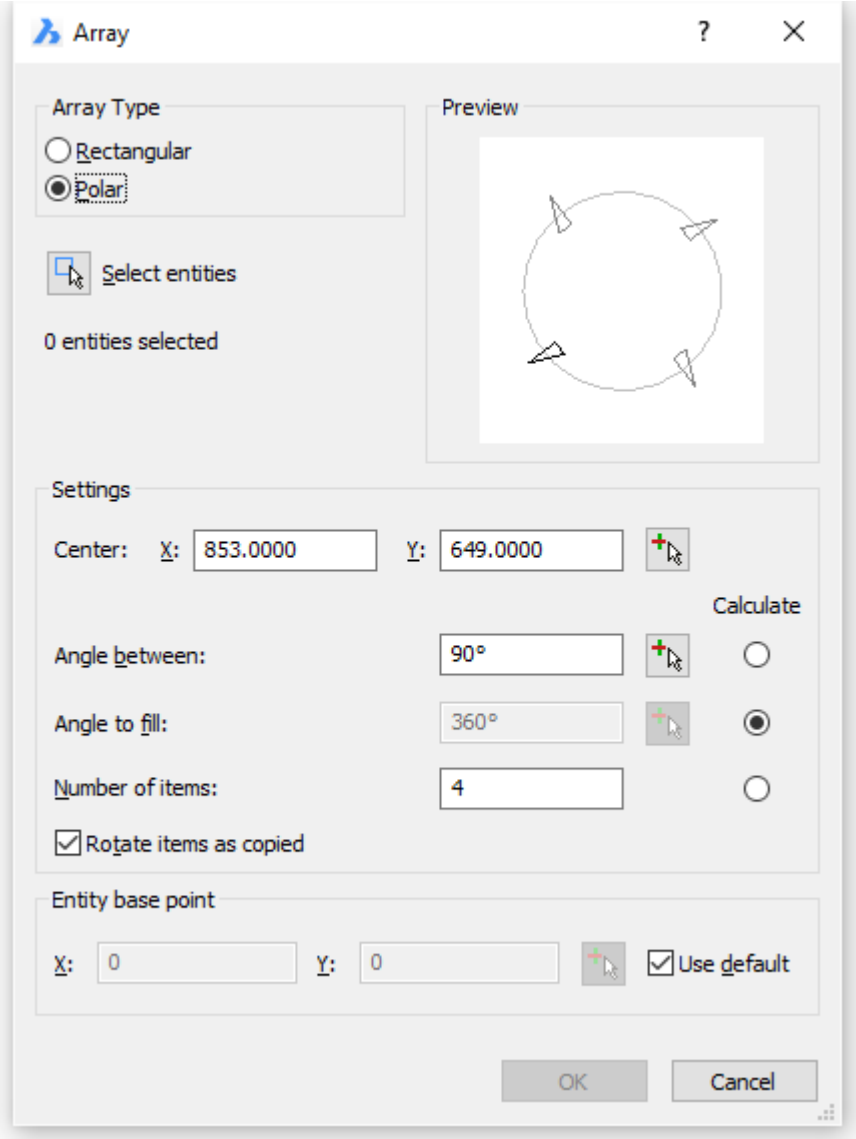
6. Click the OK button to create the array.

NOTE Positive values in steps 4 and 5 are measured along the positive X- and Y-axis of the current UCS. Negative values are measured in the opposite direction.

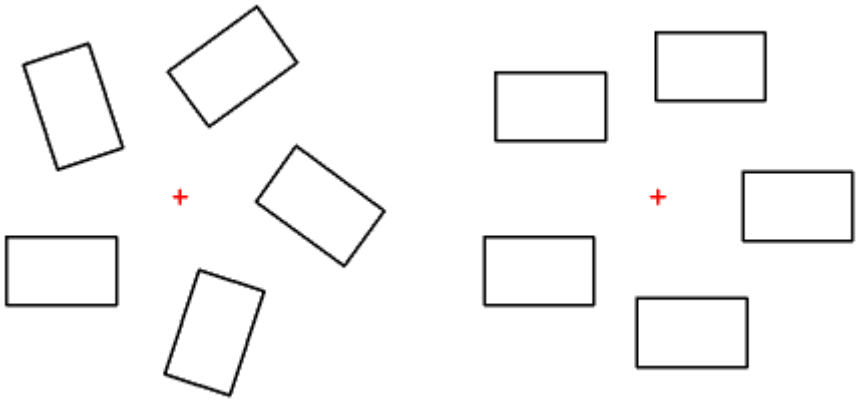
Creating a Polar Array

1. Type *arrayclassic* in the command bar, then press Enter.

A dialog box displays:

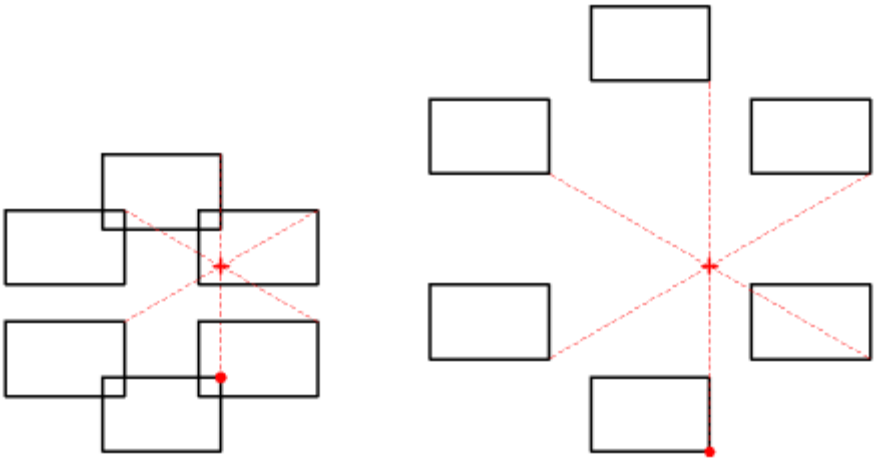


2. Under *Array Type* choose *Polar*.
3. Click the *Select Entities* button ( ).
   The *Array* dialog box closes temporarily to let you select entities.
4. Select the entities, then right click or press Enter.
   The *Array* dialog box reopens.
5. Choose a *Calculate* option: *Angle between*, *Angle to fil*l or *Number of items*.
   The settings field of the selected option is dimmed.
6. Fill out the *Settings* fields or press the *Pick Points* buttons ( ) to define the settings graphically.
   The *Array* dialog box closes temporarily to let you pick point in the drawing.
7. (option) Check the *Rotate items as copied* option.



Entities rotated (left) or not (right) about the center point of the array.

8. (option) Define an *Entity base point*.



Polar array with base point (dot) and center point (cross).

Creating a 3D Rectangular Array

1. Do one of the following:
   - Click the *3D Array* tool button ( ) on the *Modify* toolbar.
   - Choose *3D Array* in the *Modify* menu.
   - Type *3darray* in the command bar, then press Enter.

You are prompted: Select entities to array:

3. Select the entities, then right click or press Enter.
   You are prompted: Type of array: Polar/<Rectangular>:
4. Choose *Rectangular* in the prompt menu or type *R* in the command bar and press Enter.

You are prompted: Number of rows in the array <1>:

4. Type the number of rows in the command bar, then press Enter.
   You are prompted: Number of columns <1>:
5. Type the number of columns in the command bar, then press Enter.
   You are prompted: Number of levels <1>:
6. Type the number of levels in de command bar, then press Enter.
   You are prompted: Vertical distance between rows:

7. To define the distance between rows of the array, do one of the following:

   - Type the distance in the command bar and press Enter.
   - Specify the distance by clicking two points.

You are prompted: Horizontal distance between columns:

8. To define the distance between the columns of the array, do one of the following:

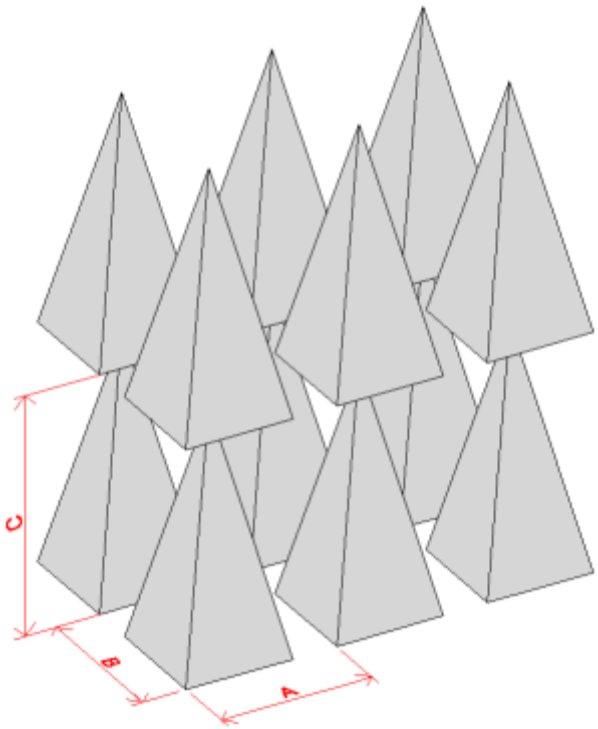
   - Type the distance in the command bar and press Enter.
   - Specify the distance by clicking two points.

You are prompted: Depth between levels:

9. To define the distance between the levels of the array, do one of the following:

   - Type the distance in the command bar and press Enter.
   - Specify the distance by clicking two points.

The array is created.



Rectangular 3D Array:row spacing A, column spacing B, level spacing C

NOTE Positive values in steps 7, 8 and 9 are measured along the positive X-, Y- and Z-axis of the current UCS. Negative values are measured in the opposite direction.

Creating a 3D Polar Array

1. Do one of the following:
   - Click the *3D Array* tool button ( ) on the *Modify* toolbar.
   - Choose *3D Array* in the *Modify* menu.
   - Type *3darray* in the command bar, then press Enter.

You are prompted: Select entities to array:

3. Select the entities, then right click or press Enter.
   You are prompted: Type of array: Polar/<Rectangular>:
4. Choose *Polar* in the prompt menu or type *P* in the command bar and press Enter.

You are prompted: ENTER to specify angle between items/<Number of items to array>:

4. Type the number of items you want in the array, then press Enter.
   You are prompted: Angle to array (+ for ccw, - for cw) <360>:
5. Do one of the following:

   - Type the angle to array, then press Enter.
   - Press Enter to create a 360° array.

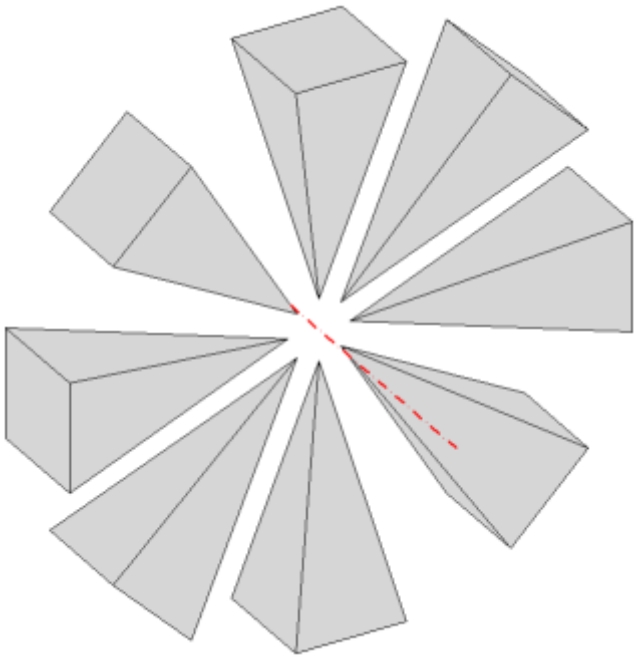
You are prompted: Rotate entities around the array? No/<Yes>

6. Do one of the following:

   - Press Enter to rotate the entities about the array axis.
   - Type N and press Enter to keep the original orientation of the entities.

You are prompted: Center of polar array.

7. Specify the first point of the array axis.
   You are prompted: Specify second point along central axis of array:
8. Specify the second point of the array axis.
   The array is created.



Polar 3D array about a horizontal axis (red).

Resizing Entities

Extending Entities

Command: EXTEND and TRIM

The *Extend* command lets you extend entities to a boundary, which is defined by one or more other entities.

If the *Edge Mode* setting (*EDGEMODE*) is *On*, you can extend entities to an implied edge of the boundary entities.

If the boundary entity is not in the same plane as the entity you want to extend, the *Projection Mode* setting (*PROJMODE*) lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).

When extending entities, first select the boundary edges, and then specify the entities to extend, selecting them either one by one or using the fence selection method.

The following entities can be extended: arcs, lines, two-dimensional polylines, rays.

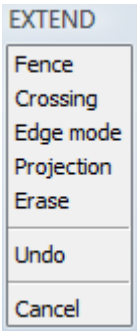
Boundary entities can be: arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, layout viewports.

To extend entities

1. Do one of the following:
   - Click the *Extend* tool button ( ) on the *Extend/Stretch* flyout of *Modify* toolbar.
   - Choose *Extend* in the *Modify* menu.
   - Type *extend* or *EX* in the command bar, then press Enter.

You are prompted: Select boundary entities for extend <ENTER to select all>:

2. Select the boundary entities, then press Enter or right click.
   You are prompted: Fence/Crossing/Edge mode/Projection/eRase/Undo/<Select entity to extend or shift-select to trim>:
   A prompt menu displays.



3. Click the entity you want to extend near the end that can make the extension.
   The entity is extended.
   or
   Press and hold the shift key to trim the entity.
4. Repeat step 3 to extend more entities.
5. Right click or press Enter to conclude the *Extend* command.

NOTES

- Entities which are selected when you launch the *Extend* command will be used as boundary entities.
- If you select an entity near the end that cannot make an extension to one of the boundary entities, the *Extend* command is aborted.

Trimming Entities

Command: TRIM and EXTEND

The Trim command lets you clip or trim entities by cutting entities.

If the *Edge Mode* (*EDGEMODE*) system variable is *On*, you can trim entities by an implied edge of the cutting entities.

If the cutting entity is not in the same plane as the entity you want to trim, the *Projection Mode* (*PROJMODE*) system variablele defines how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).

When trimming entities, you first select the cutting edges, and then specify the entities to trim, selecting them either one by one or using the fence selection method.

The following entities can be trimmed: lines, two- and three dimensional polylines, arcs, circles, ellipses, elliptical arcs, splines, rays and infinite lines.

Cutting entities can be: lines, splines, polylines, arcs, circles, elliptical arcs, ellipses, rays, infinite lines, layout viewports.

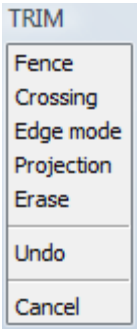
Entities in a block can be selected as cutting entities.

To trim entities

1. Do one of the following:
   - Click the *Trim* tool button ( ) on the *Modify* toolbar.
   - Choose *Trim* in the *Modify* menu.
   - Type *trim* or *TR* in the command bar, then press Enter.

You are prompted: Select cutting entities for trim <ENTER to select all>:

2. Select the cutting entities, then press Enter or right click.
   You are prompted: Fence/Crossing/Edge mode/Projection/eRase/Undo<Select entity to trim or shift-select to extend>:
   A prompt menu displays.



3. Click the entity you want to trim, the part of the entity that you click will be removed.
   The entity is trimmed.
   or
   Press and hold the shift key to extend the entity.
4. Repeat step 3 to trim more entities.
5. Right click to conclude the *Trim* command.

NOTES Entities which are selected when you launch the *Trim* command will be used as cutting entities.

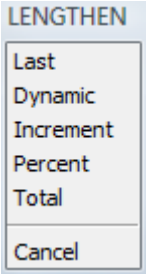
Changing the Length of an Entity

The *Lengthen* command lets you change the length of lines, open polylines and arcs. You can also modify the included angle of arcs.

1. Do one of the following:
   - Click the *Lengthen* tool button ( ) on the *Modify* toolbar.
   - Choose *Lengthen* in the *Modify* menu.
   - Type *lengthen* in the command bar, then press Enter.

You are prompted: Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>:

A prompt menu displays:



2. (option) Click an entity.
   The current length of the selected entity displays in the command bar.
   In case an arc is selected, the current length and the included angle display in the command bar.
3. Do one of the following:

   - Choose *Dynamic* in the prompt box.
   - Type *DY* in the command bar, then press Enter.

You are prompted: Mode/<Select entity to change>:

4) Click the entity near the end you want to change.
   The length of the entity changes dynamically.
5) Click to define the new length.

To Modify the Included Angle of an Arc

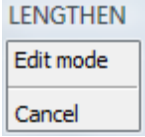
1. Do one of the following:
   - Click the *Edit Length* tool button ( ) on the *Modify* toolbar.
   - Choose *Edit Length* in the *Modify* menu.
   - Type *editlen* in the command bar, then press Enter.

You are prompted: Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>:

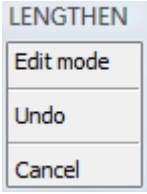
2. Choose *Total* in the prompt menu or type *T* and press Enter.
   You are prompted: Angle/<Enter total Length (0.00)>:
   A prompt menu displays:

3. Choose *Angle* in the prompt menu or type *A* and press Enter.
   You are prompted: Enter total angle <00° 0' 0">:
4. Type the new angle in the command bar and press Enter.
   You are prompted: Mode/<Select entity to change>:

A prompt menu displays:



5. Click the arc at the end you want to lengthen or shorten.
   The arc is modified.
   You are prompted: Mode/Undo/<Select entity to change>:
   A prompt menu displays:



6. Do one of the following:

   - Select another arc.
   - Choose *Undo* in the prompt menu or type *U* and press Enter to undo the previous action.
   - Choose *Edit mode* in the prompt menu or type *M* and press Enter to choose a different *Edit Length* command option.
   - Choose *Cancel* in the prompt menu to conclude the *Edit Length* command.

NOTE In the Properties bar you can edit the *start angle*, *end angle* and *radius* of an arc.

Stretching Entities

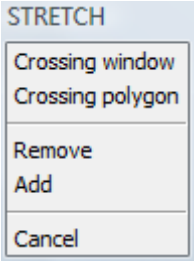
The size and shape of entities can be changed by stretching them. You select an area in your drawing using either a rectangular window or a polygon, then you specify a base point and a displacement point. All points and nodes inside the selected area will be moved over the specified distance. As a result, entities that cross the window or polygon boundary are stretched; those completely within the window or polygon are simply moved.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

1. Do one of the following:
   - Click the *Stretch* tool button ( ) on the *Extend/Stretch* flyout of the *Modify* toolbar.
   - Choose *Stretch* in the *Modify* menu.
   - Type s*tretch* or *S* in the command bar and press Enter.

You are prompted: Select entities to stretch by crossing-window or crossing-polygon:

A prompt menu displays:



2. (option) Choose *Crossing window* in the *Stretch* prompt menu, then define the stretch area by a rectangular window.

3. (option) Choose *Crossing polygon* in the *Stretch* prompt menu, then define the stretch area by a polygon.
4. (option) Repeat steps 2 and 3 to expand the stretch area.
5. (option) Choose *Remove* in the *Stretch* prompt menu to select entities in the stretch area that must not be stretched.
6. (option) Choose *Add* in the *Stretch* prompt menu to add previously remove entities to the selection of entities that must be stretched.
7. Right click to conclude the selection of entities.
   You are prompted: Base point of displacement.
8. Specify the base point.
   The selection stretches dynamically.
   You are prompted: Second point of displacement.
9. Do one of the following to specify the second displacement point.

   - Click the displacement point.
   - Use Direct Distance Entry: type the displacement distance, then press Enter.
     The distance is measured in the cursor direction.
     Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

The selection is stretched.

NOTES To add or remove entities from the selection set in steps 5 and 6 you can use any selection method: picking, window inside or crossing window.

Stretching Entities using Grips

Click the entity you want to stretch.
The entity grips display.

Do one of the following:

Click a grip to activate it.
The grip is attached to the drawing cursor.

Press and hold the Shift key to select multiple grips, then release the Shift key and move one of the selected grips.

Click to relocate the grip.
The grip is released from the drawing cursor.

NOTES

- Use Ortho Mode, Polar Tracking or Snap Tracking to constrain the movement of the drawing cursor.
- If you select two (or more) entities with coinciding grips, the shared grips move simultaneously.

Scaling Entities

The *Scale* command resizes a selection set in relation to a base point. You can specify the scale factor by selecting a base point and a length or by typing an explicit scale factor in the command bar. The scale factor can also be defined with respect to a base scale factor. E.g. when the base scale factor is 2 and the new scale is 3, the new size is 3/2 of the original. The base scale and the new scale can also be defined graphically in the drawing.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

Scaling an Entity Using a Base Scale

Select the entity.

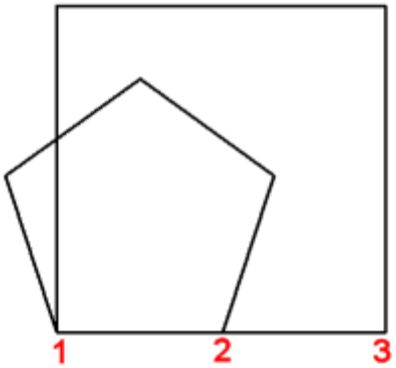
Do one of the following:

Click the *Scale* tool button ( ) on the *Modify* toolbar

Choose *Scale* in the *Modify* menu.

Type *scale* or *SC* in the command bar.

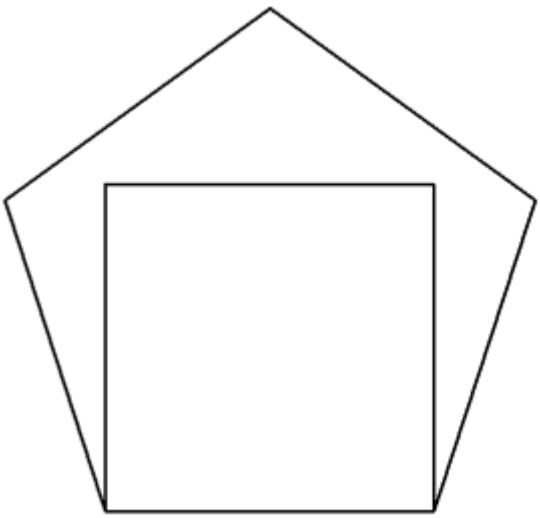
You are prompted: Base point:
Specify the base point (1).
You are prompted: Base Scale/<Scale Factor>:



Choose *Base* in the prompt menu of type *B* or *R* and press Enter.
You are prompted: Base scale <1>:

Click point 1, then point 2 to define the base scale.
The pentagon scales dynamically.

Click point 3.
The edge of the pentagon now equals the side of the square.



To Scale a Selection Set

Do one of the following:

Click the *Scale* tool button ( ) on the *Modify* toolbar

Choose *Scale* in the *Modify* menu.

Type *scale* or *SC* in the command bar.

You are prompted: Select entities to scale:
Select the entities, then right click or press Enter.
You are prompted: Base point:
Specify the base point for the scaling.
The selection set scales dynamically.
You are prompted: Base scale/<Scale Factor>:
Type the scale factor in the command bar and press Enter.
The selection set is scaled.

NOTE The length of the vector between the base point and the current cursor position is used as the dynamic scale factor in step 3.

Breaking Entities

Command: BREAK

The *Break* command remove a portion of an entity, thus breaking it into two parts.

You can break arcs, circles, ellipses, lines, polylines, rays and infinite lines. Breaking a circle converts it to an arc. A ray is broken into a ray and a line, an infinite line is broken into two rays.

When breaking entities, you must specify two points for the break. By default, the point you use to select the entity becomes the first break point; however, you can use the First option to select a break point different from the one that selects the entity.

Do one of the following:

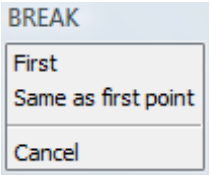
Click the *Break* tool button ( ) on the *Modify* toolbar.

Choose *Break* in the *Modify* menu.

Type *break* or *BR* in the command bar, then press Enter.

You are prompted: Select entity to break:

Click the entity you want to break.
You are prompted: First break point/<Second break point>:
A prompt menu displays:



Click a second point on the entity.
The entity is broken. The portion between the two break points is removed.

Joining Entities

Command: JOIN

The *Join* command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.

The type of the resulting entity depends on the types of the input entities and on their coplanarity.

To join colinear lines

When you join colinear lines, the farthest endpoints remain at their existing locations; BricsCAD draws a new line between these points.

Do one of the following:

Click the *Join* tool button ( ) on the *Modify* toolbar.

Choose *Join* in the *Modify* menu.

Type *join* in the command bar, then press Enter.

You are prompted: Select source entity or multiple entities to join at once:

Select the lines.
You are prompted: Select entities to join:

Right click or press Enter.
The lines are joined.

To join two arcs

Coplanar arcs, sharing midpoint and radius, are joined counterclockwise, therefore the result depends on the selecting order.

Do one of the following:

Click the *Join* tool button ( ) on the *Modify* toolbar.

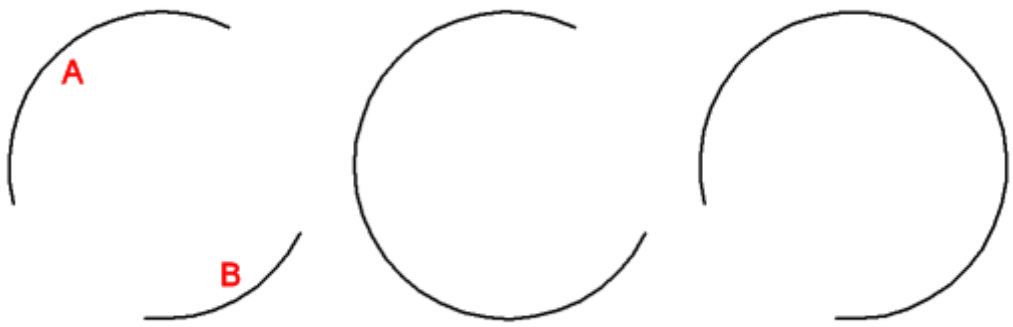
Choose *Join* in the *Modify* menu.

Type *join* in the command bar, then press Enter.

You are prompted: Select source entity or multiple entities to join at once:

Select the first arc.
You are prompted: Select entities to join:

Select the second arc and press Enter.
You are prompted: 2 entities joined into 1 arc



Joining two arcs: start situation (left) and result when selecting A first (middle) or B first (right).

NOTE If more arcs share the same center point and radius, you can select them in step 3. Press Enter after selecting the last arc.

Chamfering and Filleting

The *Chamfer* command connects two non-parallel entities by extending or trimming them and then joining them with a line to create a beveled edge.

In BricsCAD you can choose between two chamfer methods:

distance-distance: specify how far to trim the entities back from their intersection

distance-angle: specify the length of the chamfer and the angle it forms along the first entity.

The following entities can be chamfered: lines, polylines, rays and infinite lines. When chamfering a polyline, you can create a chamfer between two polyline segments or you can chamfer the entire polyline.

Chamfering Using the Distance Distance Method

Do one of the following:

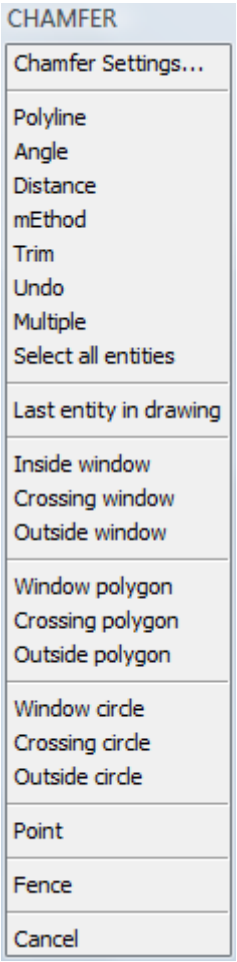
Click the *Chamfer* tool button ( ) on the *Chamfer/Fillet* flyout of the *Modify* toolbar.

Choose *Chamfer* in the *Modify* menu.

Type *chamfer* or *CHA* in the command bar, then press Enter.

You are prompted: Chamfer (<current chamfer settings>):
Settings/Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/<Select first entity>:

A prompt menu displays:



(option) Choose *Chamfer Settings ...* in the prompt menu or type *S* and press Enter.
In the *Settings* dialog window:

Specify the *Chamfer first distance*.

Set the *Chamfer second distance*.

Set the *Chamfer mode* to *Distance-Distance.*

Chamfer/Fillet	
Chamfer first distance	2.0000
Chamfer second distance	3.0000
Chamfer length	2.5000
Chamfer angle	30
Chamfer mode	Distance-Distance
Fillet radius	0.5000
Trim mode	☑ Trim selected edges to the endpoints of chamfer lines and fillet arcs

Close the *Settings* dialog window.
You are prompted: Chamfer (<current chamfer settings>):
Settings/Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/<Select first entity>:
Select the first entity or polyline segment.
The command window reads: Select second entity.

Select the second entity or polyline segment.
The chamfer is created.

Chamfering Using the Length Angle Method

Do one of the following:

Click the *Chamfer* tool button ( ) on the *Chamfer/Fillet* flyout of the *Modify* toolbar.

Choose *Chamfer* in the *Modify* menu.

Type *chamfer* or *CHA* in the command bar, then press Enter.

You are prompted: Chamfer (<current chamfer settings>): Settings/Polyline/<Select first entity>:

A prompt menu displays.
Choose *Chamfer Settings ...* in the prompt menu or type *S* and press Enter.

In the *Settings* dialog window:

Specify the *Chamfer length*.

Set the *Chamfer angle*.

Set the *Chamfer mode* to *Length-Angle.*

Chamfer/Fillet	
Chamfer first distance	0.5000
Chamfer second distance	0.5000
Chamfer length	2.5000
Chamfer angle	30
Chamfer mode	Length-Angle
Fillet radius	0.5000
Trim mode	☑ Trim selected edges to the endpoints of chamfer lines and fillet arcs

Close the Settings dialog window.
You are prompted: Chamfer (<current chamfer settings>): Settings/Polyline/<Select first entity>:

Select the first entity or polyline segment.
The command window reads: Select second entity.

Select the second entity or polyline segment.
The chamfer is created.

Chamfering All Vertices of a Polyline

Do one of the following:

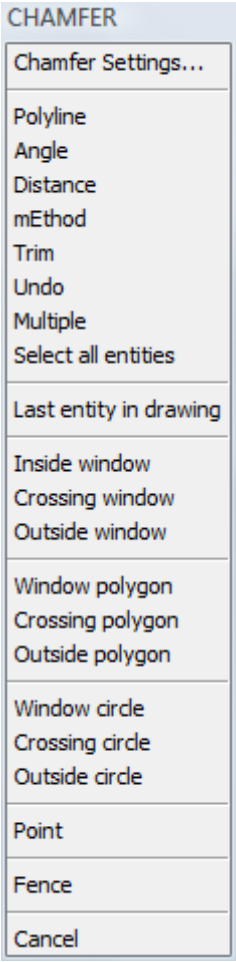
Click the *Chamfer* tool button ( ) on the *Chamfer/Fillet* flyout of the *Modify* toolbar.

Choose *Chamfer* in the *Modify* menu.

Type *chamfer* or *CHA* in the command bar, then press Enter.

You are prompted: Chamfer (<current chamfer settings>): Settings/Polyline/<Select first entity>:

A prompt menu displays:



(option) Adjust the *Chamfer Settings*.

Choose *Polyline* in the prompt menu or type *P* and press Enter.
You are prompted: Select 2D polyline to chamfer:

Select a polyline.
All vertices of the selected polyline are chamfered.

NOTE When the chamfer method is *distance-angle*, the direction of the polyline defines which is the first entity of a vertex. See drawing rectangles for more information about the direction of closed polylines.

Filleting Entities

The *Fillet* command connects two entities with an arc of a specified radius to create a rounded edge.

You can fillet pairs of line segments, straight polyline segments, arcs, circles, rays, and infinite lines. You can also fillet parallel lines, rays, and infinite lines.

When filleting a polyline, you can fillet multiple segments between two selected segments or you can fillet the entire polyline.

Filleting Two Entities or Polyline Segments

Do one of the following:

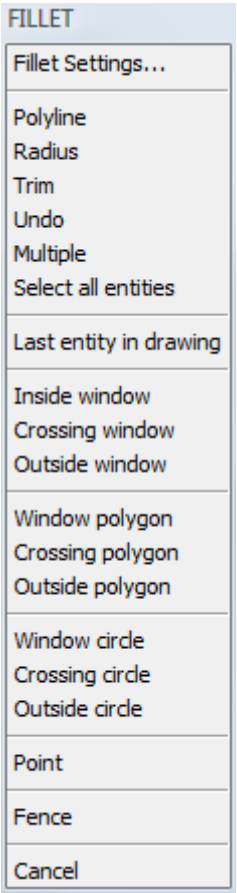
Click the *Fillet* tool button ( ) on the *Chamfer/Fillet* flyout of the *Modify* toolbar.

Choose *Fillet* in the *Modify* menu.

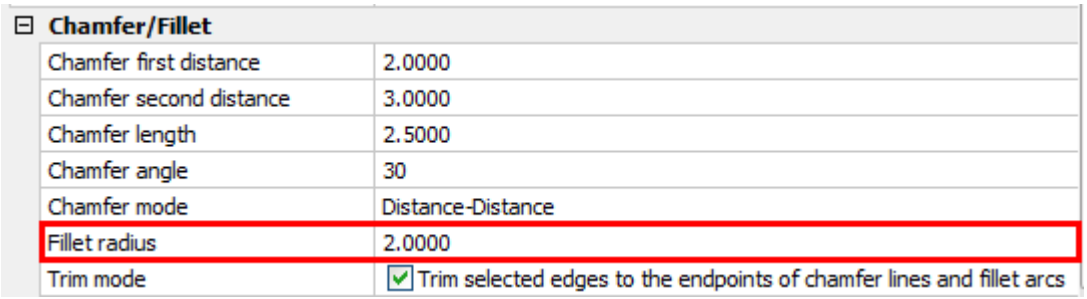
Type *fillet* or *F* in the command bar, then press Enter.

You are prompted: Fillet (<current fillet settings>):
Settings/Polyline/Radius/Trim/Undo/Multiple/<Select first entity>:

A prompt menu displays:



(option) Choose *Fillet Settings ...* in the prompt menu or type *S* and press Enter.
Specify the *Fillet radius* in the *Settings* dialog window:



Chamfer/Fillet	
Chamfer first distance	2.0000
Chamfer second distance	3.0000
Chamfer length	2.5000
Chamfer angle	30
Chamfer mode	Distance-Distance
Fillet radius	2.0000
Trim mode	Trim selected edges to the endpoints of chamfer lines and fillet arcs

Close the Settings dialog window.
You are prompted: Fillet (<current fillet settings>):
Settings/Polyline/Radius/Trim/Undo/Multiple/<Select first entity>:

Select the first entity or polyline segment.
The command window reads: Select second entity.

Select the second entity or polyline segment.
The fillet is created.

Filleting all Vertices of a Polyline

Do one of the following:

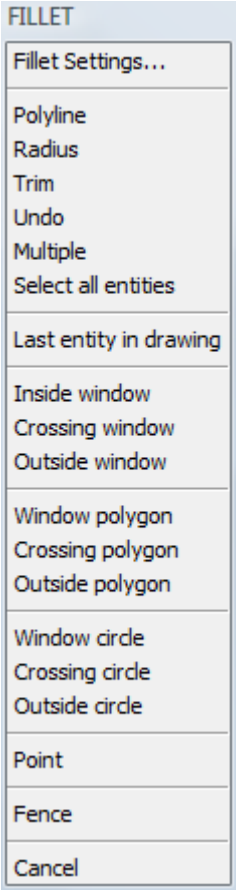
Click the *Fillet* tool button ( ) on the *Chamfer/Fillet* flyout of the *Modify* toolbar.

Choose *Fillet* in the *Modify* menu.

Type *fillet* or *F* in the command bar, then press Enter.

You are prompted: Fillet (<current fillet settings>): Settings/Polyline/<Select first entity>:

A prompt menu displays:



(option) Adjust the *Fillet Settings*.

Choose *Polyline* in the prompt menu or type *P* and press Enter.
You are prompted: Select 2D polyline to chamfer:

Select a polyline.

All vertices of the selected polyline are filleted.

Filleting two Parallel Lines

Do one of the following:

Click the *Fillet* tool button ( ) on the *Chamfer/Fillet* flyout of the *Modify* toolbar.

Choose *Fillet* in the *Modify* menu.

Type *fillet* or *F* in the command bar, then press Enter.

You are prompted: Fillet (<current fillet settings>):
Settings/Polyline/Radius/Trim/Undo/Multiple/<Select first entity>:

Select the first entity (line or ray).
The command window reads: Select second entity:

Select the second entity (line, ray or infinite line).
The fillet is executed at the endpoint closest to the point were the first entity is selected.
The length of the second entity is adjusted.

Editing Polylines

Command: PEDIT

The PEDIT command modifies any type of two-dimensional or three-dimensional polyline: such as rectangles, polygons and donuts, as well as three-dimensional entities such as pyramids, cylinders and spheres.

Editing a polyline can be:

- opening or closing a polyline;
- changing its overall width or the widths of individual segments;
- converting a polyline with straight line segments into a flowing curve or an approximation of a spline;
- editing individual vertices, adding, removing or moving vertices;
- adding new segments to an existing polyline;
- reversing the direction or order of the vertices;
- merging a series of polylines, lines or arcs, which are connected endpoint to endpoint, into a single polyline;
- turning a line or an arc into a polyline;

To modify a polyline, you first select the polyline, and then select a polyline editing option. The available options vary depending on whether the selected polyline is a two-dimensional or three-dimensional entity.

Converting an entity into a polyline

Do one of the following:

Click the *Edit Polyline* tool button ( ) on the *Modify* toolbar.

Choose *Edit Polyline* in the *Modify* menu.

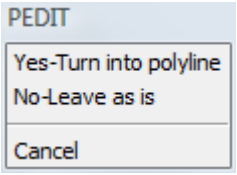
Type *editpline* or *pedit* in the command window, then press Enter.

You are prompted: Select polyline to edit.

Select a line or arc.

You are prompted: The entity selected is not a polyline. Turn it into one? <Y>

A prompt menu displays:



Do one of the following:

Press Enter to accept the default option.

Choose *Yes-Turn into polyline* in the prompt menu.

Press Enter to conclude the *Edit Polyline* command.

NOTE The *Explode* tool turns a single segment polyline back into a line or arc.

Opening and Closing Polylines

Command: PEDIT

When you close a polyline, a straight polyline segment is drawn from the last vertex of the polyline to the first vertex. Opening a polyline removes the closing segment.

When you select a polyline for editing, the prompt menu displays either the *Open* or *Close* option, depending on whether the polyline you select is closed or open.

The following entities are created as closed polylines:

rectangles

polygons

donuts

revision clouds

You can open or close a polyline using the *Edit Polyline* tool or in the BricsCAD Properties bar.

To close or open a polyline

Do one of the following:

Click the *Edit Polyline* tool button ( ) on the *Modify* toolbar.

Choose *Edit Polyline* in the *Modify* menu.

Type *editpline* or *pedit* in the command window, then press Enter.

You are prompted: Select polyline to edit.

Click the polyline you want to close or open.
You are prompted: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.

To close (or open) a polyline do one of the following:

Type *C* (or *O*) in the command bar, then press Enter.

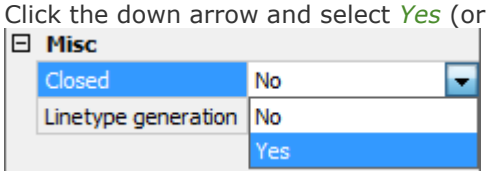
Choose *Close* (or *Open*) in the prompt menu.

Opening and closing a polyline using the BricsCAD Properties bar

Select the polyline.
The current properties of the selected polyline display in the BricsCAD Properties bar.

Select *Closed* in the *Misc* settings category in the *BricsCAD Properties* bar.

Click the down arrow and select *Yes* (or *No*).



Joining Polylines

Commands: PEDIT and JOIN

Using the *Join* option of the *Edit Polyline* tool you can add an arc, a line or a polyline entity to an existing open polyline, forming one continuous polyline entity.

To join an entity to a polyline, that entity must already share an endpoint with an end vertex of the selected polyline.

When you join an entity to a polyline, the *width* of the new polyline segment depends on the width of the original polyline and the type of entity you are joining to it:

A line or an arc inherits the width from the polyline segment to which it is joined.

A polyline joined to a tapered polyline retains its own width values.

A polyline joined to a uniform-width polyline inherits the width from the polyline to which it is joined.

The *Join* command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.

The type of the resulting entity depends on the types of the input entities and on their coplanarity.

To join an arc, line, or polyline to an existing polyline

Do one of the following:

Click the *Edit Polyline* tool button ( ) on the *Modify* toolbar.

Choose *Edit Polyline* in the *Modify* menu.

Type *editpline* or *pedit* in the command window, then press Enter.

You are prompted: Select polyline to edit.

Select the parent polyline.
You are prompted: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.

Choose *Join* in the prompt menu or type *J* and press Enter.
You are prompted: Select entities to join:

Select the entities to join.
The number of already selected entities displays in the command bar.

Right click or press Enter to stop selecting entities and add the selection to the polyline.

Do one of the following:

Continue editing the selected polyline.

Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

NOTE You can select multiple entities in step 3 on condition they form one chain with the parent polyline.

To Apply a Uniform Width to Polyline

Do one of the following:

Click the *Edit Polyline* tool button ( ) on the *Modify* toolbar.

Choose *Edit Polyline* in the *Modify* menu.

Type *editpline* or *pedit* in the command window, then press Enter.

You are prompted: Select polyline to edit.

Select the polyline.
You are prompted: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.

Choose *Width* in the prompt menu or type *W* and press Enter.
You are prompted: Enter new width for all segments:

Type a new width in the command bar and press Enter.
The new width is applied to the entire polyline.

Do one of the following:

Continue editing the selected polyline.

Choose *Cancel* in the prompt menu or press Enter or right click to conclude the *Edit Polyline* tool.

To Taper a Polyline Uniformly along its Length

Do one of the following:

Click the *Edit Polyline* tool button ( ) on the *Modify* toolbar.

Choose *Edit Polyline* in the *Modify* menu.

Type *editpline* or *pedit* in the command window, then press Enter.

You are prompted: Select polyline to edit.

Select the polyline.
You are prompted: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.

Choose *Taper* in the prompt menu or type *T* and press Enter.
You are prompted: Enter new starting polyline width <current starting width>:

Type a new starting width in the command bar and press Enter.
You are prompted: Enter new ending polyline width <current ending width>:

Type a new ending width in the command bar and press Enter.
The polyline is tapered uniformly along its length.

Do one of the following:

Continue editing the selected polyline.

Choose *Cancel* in the prompt menu or press Enter or right click to conclude the *Edit Polyline* tool.

Editing a Polyline Segment

The *Edit vertices* option of the *Pedit* command modifies individual polyline vertices.

When you select this option, the program switches into a special vertex editing mode and places an X on the first vertex. The X indicates the vertex you are editing. The *Next* and *Previous* options move the X to the next or previous vertex. You can edit only one vertex at a time.

When editing vertices, you can modify the polyline in the following ways:

- Convert a straight polyline segment into a curve.
- Break a polyline into two separate polylines.
- Insert a new vertex in a polyline.
- Move a vertex in a polyline.
- Delete vertices in a polyline.
- Change the width of a polyline segment.

The *Peditext* command is used in *2D Editing* command group of the Quad cursor menu

Editing a Polyline Using the Quad

Move the cursor over the polyline.
If currently ON, the TIPS widget displays:

The available commands in the Quad apply to the entire polyline.

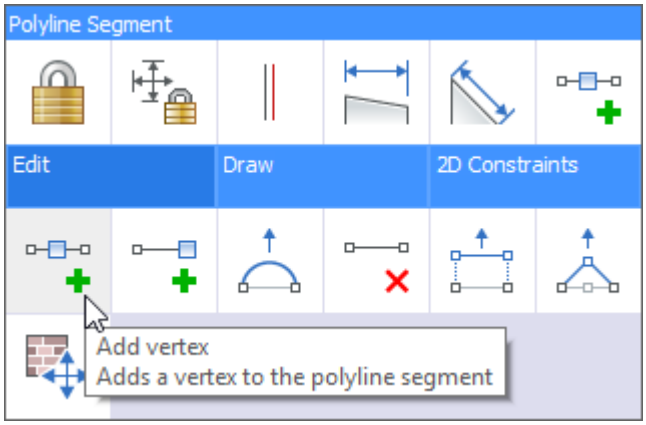
Press the Ctrl-key to switch to segment mode.

The available commands in the Quad apply to the segment under the cursor.

When the Quad cursor menu displays, do one of the following:

Right click to execute the default *Quad* command.

Move the cursor to the *Quad* and move to the *Edit* command group.



Select a command:

Add vertex: adds a vertex to the segment under the cursor.

Add vertex at end: adds as a new vertex at the end of the polyline.
This option is only available if the cursor is on one of the endpoints of a polyline.

Adjust Bulge: Modifies the bulge factor of the segment under the cursor.

Delete segment: deletes the segment under the cursor.

Stretch segment: moves the segment under the cursor.

Stretch vertex: moves the vertex closest to the cursor.

Starting the Polyline Vertex Editing Mode

Do one of the following:

Click the *Edit Polyline* tool button ( ) on the *Modify* toolbar.

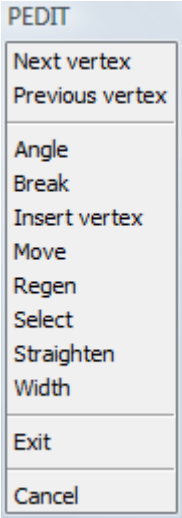
Choose *Edit Polyline* in the *Modify* menu.

Type *editpline* or *pedit* in the command window, then press Enter.

You are prompted: Select polyline to edit.

Select the polyline.
You are prompted: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.

Choose *Edit vertices* in the prompt menu or type *E* and press Enter.
You are prompted:
Next/Previous/Angle/Break/Insert/Move/Regen/SElect/Straighten/Width/eXit/<Next>:
The *Editpline* prompt menu changes:



A triangle indicates the start point of the polyline. The first vertex of the polyline is selected.

Converting a Straight Polyline Segment into a Curve

Start the polyline vertex editing mode.

Select the start vertex of the segment you want to convert.
Choose *Next vertex* / *Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.

Choose *Angle* in the prompt menu or type *A* and press Enter.
The prompt menu closes.
You are prompted: Included angle for segment (>0 is ccw, 0 is straight, <0 is cw) <current angle>:

Type a new included angle for the segment and press Enter.
The direction of an arc segment is as follows:

positive angles: counter clockwise

negative angle: clockwise direction

0° for straight segments

Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.

Do one of the following:

Continue editing the selected polyline.

Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Breaking a Polyline into two Separate Polylines

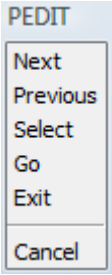
Start the polyline vertex editing mode.

Select the start vertex where you want to break the polyline.
Choose *Next vertex* / *Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.

Choose *Break* in the prompt menu or type *B* and press Enter.

You are prompted: Next/Previous/Select/Go/eXit/<Next>:

The prompt menu changes:



(option) Do one of the following to select a second vertex:

Choose *Next vertex* / *Previous vertex* in the prompt menu.

Choose *Select* in the prompt menu or type *S* and press Enter.
You are prompted to select a vertex.

Choose *Go* in the prompt menu or type *G* and press Enter.
The polyline s broken into two polylines.
If a second point is selected in step 4 the segment(s) between the selected vertices is (are) deleted.

Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.

Do one of the following:

Continue editing the selected polyline.

Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Inserting a New Vertex

Start the polyline vertex editing mode.

Select the start vertex of the segment where you want to insert a vertex.
Choose *Next vertex* / *Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.

Choose *Insert vertex* in the prompt menu or type *I* and press Enter.
You are prompted: Location for new vertex:

Specify the location for the new vertex.
The new vertex is inserted.

Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.

Do one of the following:

Continue editing the selected polyline.

Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Moving a Vertex

Start the polyline vertex editing mode.

Select the vertex you want to move.
Choose *Next vertex* / *Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.

Choose *Move* in the prompt menu or type *M* and press Enter.
You are prompted: New location for vertex.

Specify the new location for the selected vertex.
The vertex is moved.

Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.

Do one of the following:

Continue editing the selected polyline.

Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Deleting Vertices

Start the polyline vertex editing mode.

Select the vertex before the first vertex you want to delete.
Choose *Next vertex* / *Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.

Choose *Straighten* in the prompt menu or type *S* and press Enter.
You are prompted: Straighten: Next/Previous/Select/Go/eXit/<Next>:

Do one of the following to select the vertex after the last vertex you want to delete:

Choose *Next vertex* / *Previous vertex* in the prompt menu.

Choose *Select* in the prompt menu or type *S* and press Enter.
You are prompted to select a vertex.

Choose *Go* in the prompt menu or type *G* and press Enter.
The vertices between the selected vertices are removed.
A straight segment is drawn between the selected vertices.

Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.

Do one of the following:

Continue editing the selected polyline.

Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Changing the Width of a Polyline Segment

Start the polyline vertex editing mode.

Select the start vertex of the segment you want to change the width of.
Choose *Next vertex* / *Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.

Enter starting width <current width>:

Do one of the following:

Type the new width in the command bar and press Enter.

Click to define the width graphically.

You are prompted: Enter ending width <current width>:
Do one of the following:

Type the new width in the command bar and press Enter.

Click to define the width graphically.
Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.
Do one of the following:

Continue editing the selected polyline.

Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

NOTE The new width is applied when you conclude the *Edit Polyline* tool in step 6.

Curving and Decurving Polylines

Command: PEDIT

The *Fit* or *Spline* options of the *Edit Polyline* tool convert a multi-segment polyline into a smooth curve. The *Fit* option creates a smooth curve connecting all the vertices. The *Spline* option computes a smooth curve that is pulled toward the vertices but passes through only the first and last vertices.

The *Decurve* option removes *Fit* or *Spline* curves and arcs, leaving straight segments between the vertices.

To fit a curve to a polyline

Do one of the following:

Click the *Edit Polyline* tool button ( ) on the *Modify* toolbar.

Choose *Edit Polyline* in the *Modify* menu.

Type *editpline* or *pedit* in the command window, then press Enter.

You are prompted: Select polyline to edit.
Select the polyline.
You are prompted: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.
Do one of the following:

Choose *Fit* in the prompt menu of type *F* and press Enter.

Choose *Spline* in the prompt menu or type *S* and press Enter.

Original polyline *After applying Fit* *After applying Spline*

Do one of the following:

Continue editing the selected polyline.

Choose *Done* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

NOTES

- Use the *Decurve* option in step 3 of the above procedure to restore the original polyline.
- The *SPLINESEGS* (Spline Segments) settings variable controls the number of segments (lines or arcs) to be generated for spline-fit polylines.

Setting the Linetype Mode

Command: PEDIT

The *Linetype mode* option of the *Edit Polyline* tool lets you change the way how a dashed linetype is applied to a multi-segment polyline.

Do one of the following:

Click the *Edit Polyline* tool button ( ) on the *Modify* toolbar.

Choose *Edit Polyline* in the *Modify* menu.

Type *editpline* or *pedit* in the command window, then press Enter.

You are prompted: Select polyline to edit.

Select the polyline.

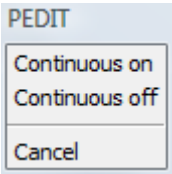
You are prompted: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:

A prompt menu displays.

Choose *Linetype* mode in the prompt menu or type *L* and press Enter.

You are prompted: Linetype continuous along polyline: ON/OFF <current setting>:

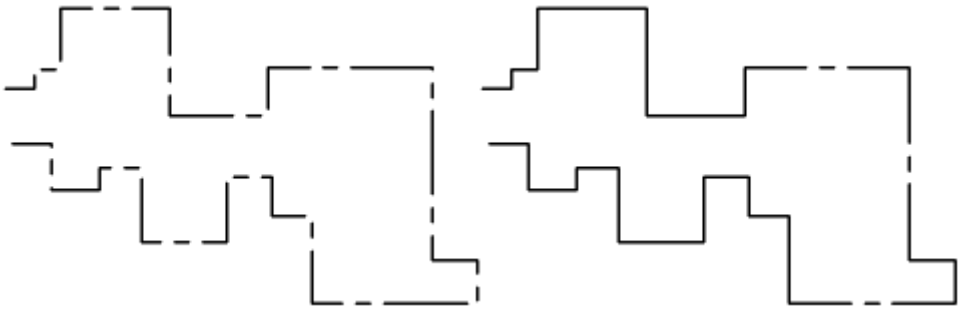
The *Editpline* prompt menu changes:



Do one of the following:

Choose *Continuous on* or *Continuous off* in the prompt box.

Type *ON* or *OF* in the command bar, then press Enter.



The same polyline with Continuous On (left) and Continuous Off (right)

Do one of the following:

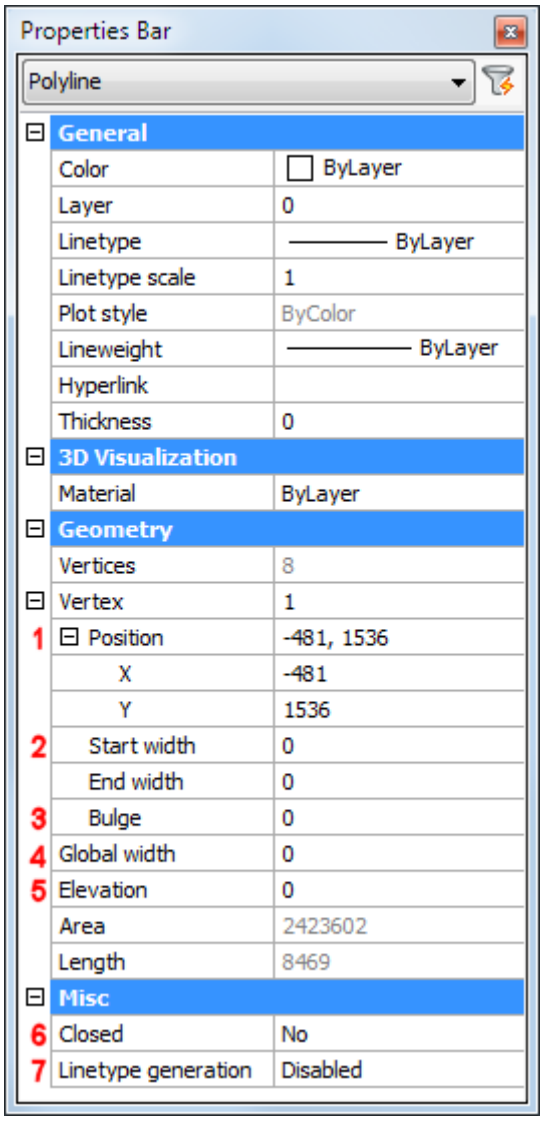
Continue editing the selected polyline.

Choose *Cancel* in the prompt menu or press Enter or right click to conclude the *Edit Polyline* tool.

Editing Polylines in the Properties Bar

When you select polyline, its current properties display in the BricsCAD Properties bar.
In the *BricsCAD Properties bar* the following polyline properties can be edited:

Move vertices (1)
Change the width of polyline segments (2)
Convert straight segments into a curve (3)
Change the global width (4)
Change the Elevation (5)
Open / close the polyline (6)
Change the Linetype generation mode (7)

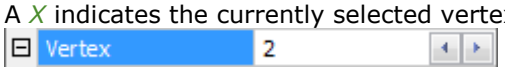


To move polyline vertices

Select the polyline.

Click the *Vertex* field in the *Properties bar*, then press the arrow buttons to select the vertex you want to move.

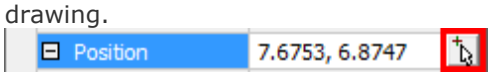
A *X* indicates the currently selected vertex in the drawing.



Do one of the following:

Adjust the *X-* and/or *Y-field* and press Enter.

Click the *Vertex Position* field and press the *Position* button to move the vertex in the drawing.

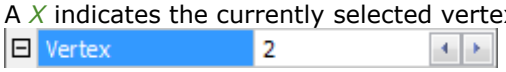


Continue editing the polyline or press the Esc key to stop.

To change the width of polyline segments

Select the polyline.

Click the *Vertex* field in the *Properties bar*, then press the arrow buttons to select the start vertex of the segment you want to modify.
A *X* indicates the currently selected vertex in the drawing.



Type a new width in the *Start width* field and press Enter.

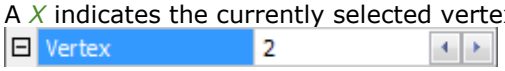
Type a new width in the *End width* field and press Enter.

Continue editing the polyline or press the Esc key to stop.

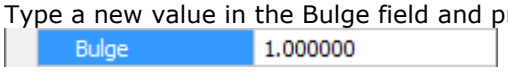
To convert straight segments into a curve

Select the polyline.

Click the *Vertex* field in the *Properties bar*, then press the arrow buttons to select the start vertex of the segment you want to modify.
A *X* indicates the currently selected vertex in the drawing.



Type a new value in the Bulge field and press Enter.

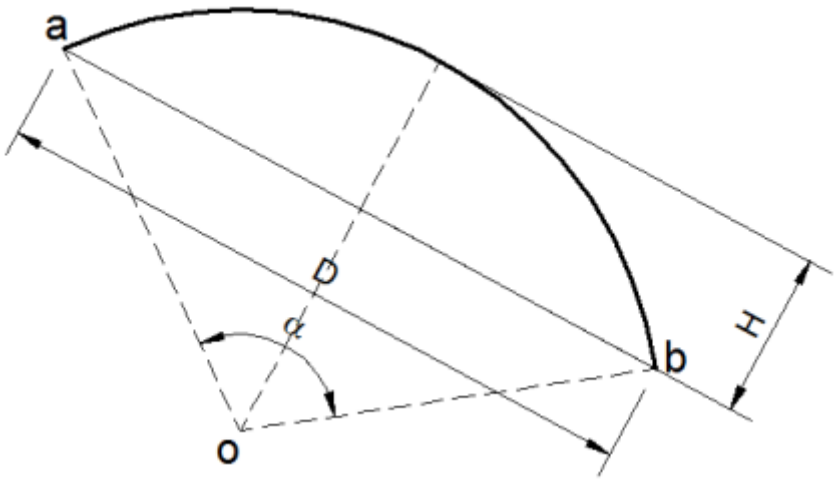


Continue editing the polyline or press the Esc key to stop.

NOTE The bulge is the tangent of one fourth the included angle for an arc segment.

A positive bulge creates a counter clockwise arc, while a negative bulge creates an arc with a clockwise direction.

Straight segments have a zero bulge.



Bulge = 2H/D

Bulge = tg α/ 4

Converting Entities

Exploding Entities

Commands: EXPLODE and TXTEXP

The *Explode* command converts complex entities, such as blocks, polylines, solids or dimensions into their component parts.

Exploding a polyline or dimension reduces it to a collection of individual line and arc entities that you can then modify individually.

Blocks are converted to the individual entities, possibly including other, nested blocks that composed the original entity.

Multiline texts explode into single line text entities.

In general exploding entities will have no visible effect in the drawing, except for:

If the original polyline had a width, the width information is lost when you explode it. The resulting lines and arcs follow the centerline of the original polyline.

If you explode a block containing attributes, the attributes are lost, but the original attribute definitions remain.
Colors and linetypes assigned *BYBLOCK* may appear different after exploding an entity, because they will adopt the default color and linetype.

The *TxtExp* command explodes text into polylines (short for "text explode").

To explode entities

Do one of the following:

Click the *Explode* tool button ( ) on the *Modify* toolbar.

Choose *Explode* in the *Modify* menu.

Type *explode* in the command bar, then press Enter.

The command window reads: Select entities to explode.

Select the entities, then right click or press Enter.

NOTE If you select the entities first, then launch the *Explode* tool, the selected entities are exploded immediately.

Creating Regions

Command: REGION

The *Region* command converts closed entities into a solid entity.

You can create regions from closed entities, such as polylines, polygons, circles, ellipses, closed splines and donuts.

Creating regions typically has no visible effect on a drawing. However, if the original entity had a width or lineweight, that information is lost when you create the region.

Do one of the following:

Click the *Region* tool button ( ) in the *Draw* toolbar.

Choose *Region* in the *Draw* menu.

Type *region* in the command bar, then press Enter.

You are prompted: Select objects:

Select the entities, then right click or press Enter.
The command bar displays how many regions have been created.

Measuring and Dividing Entities

Measuring Entities

Command: MEASURE

The *Measure* command places markers - points or blocks - at a specified interval along the length or circumference of an entity. The Measure tool starts placing markers at the closes endpoint to where you select the entity.

You can measure lines, polylines, arcs, circles, ellipses, elliptical arcs and splines. You cannot measure rays and infinite lines.

You measure a circle along its circumference starting from the *Angle Base* value as defined in the *Settings* dialog. If *Angle Base* is set to zero, a circle is measured starting at 3 o'clock (east).

To measure an entity using points

Do one of the following:

Click the *Measure* tool button ( ) on the *Modify* toolbar.

Choose *Measure* in the *Modify* menu.

Type *measure* in the command bar, then press Enter.

You are prompted: Select entity to measure:

Click the entity you want to measure.
You are prompted: Block/<Segment length>:

To define the measurement interval, do one of the following:

Type the measurement interval and press Enter.

Specify two points in the drawing.

Points are placed along the selected entity at the specified interval.

NOTE It might be necessary to adjust the Point display mode settings to display the points correctly.

To place blocks a specified interval along an entity

Do one of the following:

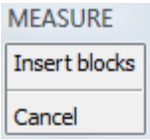
Click the *Measure* tool button ( ) on the *Modify* toolbar.

Choose *Measure* in the *Modify* menu.

Type *measure* in the command bar, then press Enter.

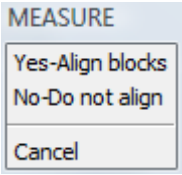
You are prompted: Select entity to measure:

Click the entity you want to measure.
You are prompted: Block/<Segment length>:
A prompt menu displays:



Choose *Insert blocks* in the prompt menu or type B and press Enter.
You are prompted: Name of block to insert:

Type the name of the block in the command bar and press Enter.
You are prompted: Align blocks with entity? <Y>:
A prompt menu displays:



Do one of the following:

Choose *Yes-Align blocks* in the prompt menu or press Enter to align the blocks with the selected entity.

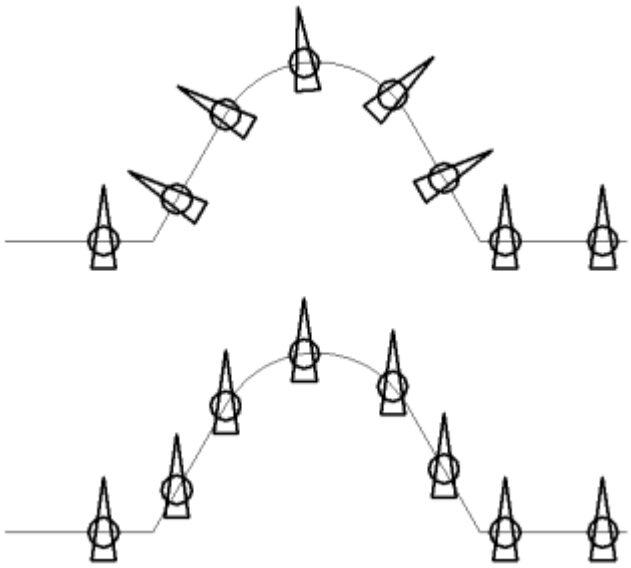
Choose *No-Do not align* in the prompt menu or type N and press Enter to place the blocks not rotated.

You are prompted: Segment length:
To define the measurement interval, do one of the following:

Type the measurement interval and press Enter.

Specify two points in the drawing.

The blocks are placed along the selected entity at the specified interval.



Blocks aligned (top) or not aligned (bottom)

Dividing Entities

Command: DIVIDE

The *Divide* command places markers - points or blocks - along a selected entity. The markers evenly divide the entity into the specified number of equal parts.

You can divide lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

Do one of the following:

Click the *Divide* tool button ( ) on the *Modify* toolbar.

Choose *Divide* in the *Modify* menu.

Type *divide* in the command bar, then press Enter.

You are prompted: Select entity to divide:

Click the entity you want to divide.
You are prompted: Blocks/<Number of Segments>:

Type the number of segments in the command bar and press Enter.

Points are placed along the selected entity to divide the entity.

NOTE It might be necessary to adjust the Point display mode settings to display the points correctly.

To divide an entity using blocks

Do one of the following:

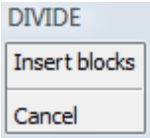
Click the *Divide* tool button ( ) on the *Modify* toolbar.

Choose *Divide* in the *Modify* menu.

Type *divide* in the command bar, then press Enter.

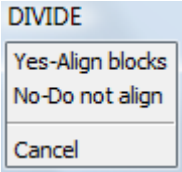
You are prompted: Select entity to divide:

Click the entity you want to divide.
You are prompted: Blocks/<Number of Segments>:
A prompt menu displays:



Choose *Insert blocks* in the prompt menu or type B and press Enter.
You are prompted: Name of block to insert:

Type the name of the block in the command bar and press Enter.
You are prompted: Align blocks with entity? <Y>:
A prompt menu displays:

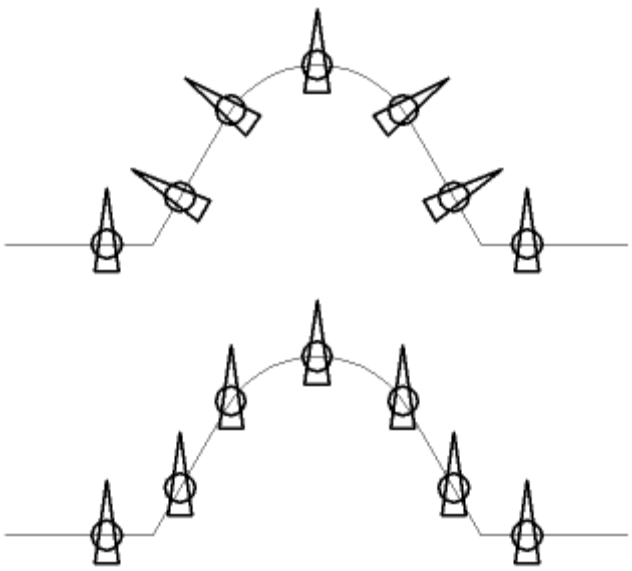


Do one of the following:

Choose *Yes-Align blocks* in the prompt menu or press Enter to align the blocks with the selected entity.

Choose *No-Do not align* in the prompt menu or type N and press Enter to place the blocks not rotated.

The blocks are placed along the selected entity to divide the entity.



Blocks aligned (top) or not aligned (bottom)

Blocks, Attributes and External References

Working with Blocks

Commands: BLOCK, -BLOCK, WBLOCK, PASTEBLOCK and SECTIONPLANETOBLOCK

The *Block* command groups entities into a block using a dialog box. The *-Block* commands prompts you in the command bar.

The *Wblock* command writes blocks and other drawing parts to a separate drawing file (short for "write block").

The *Pasteblock* command pastes entities from the *Clipboard* as a block in the current drawing (see Paste a selection as a block).

The *SectionPlaneToBlock* command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

TUTORIALS

Creating blocks quickly on the fly

General procedure to create a block

Specify the name.

Specify the insertion point.
The insertion point is the base point for the block and serves as the reference point when you later insert the block into a drawing.

Select entities.

To Create a Block in a Drawing

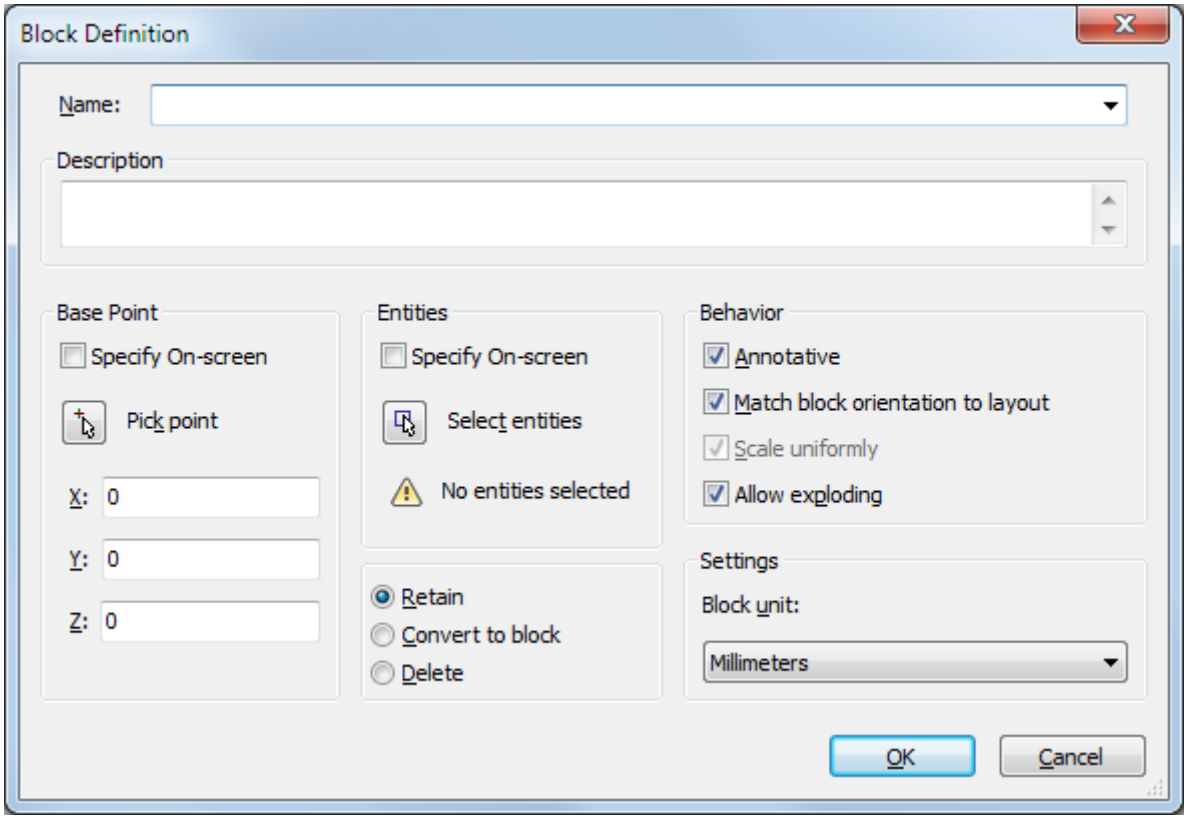
Do one of the following:

Click the *Create Block* tool button ( ) on the *Tools* toolbar.

Choose *Create Block* in the *Tools* menu.

Type *block* in the command bar, then press Enter.

The *Block Definition* dialog displays:



Type a name for the new block in the *Name* field.

Do one of the following to specify the *Base Point*:

Check the *Specify On-screen* option.
You will be prompted to specify the base point after clicking the *OK* button.

Click the *Pick point* button ( ).
The *Block Definition* dialog closes temporarily to let you pick a point in the drawing.

Type the coordinates of the base point in the X-, Y- and Z-field.

Do one of the following to select the entities to be included in the block definition:

Check the *Specify On-screen* option.
You will be prompted to select the entities after clicking the *OK* button.

Click the *Select entities* button ( ).
The *Block Definition* dialog closes temporarily to let select the entities in the drawing.

Set the block properties:

Annotative: creates an annotative block.

Match block orientation to layout: if checked, the orientation of an annotative block is preserved in a rotated viewport.

Scale uniformly: if checked, the X-, Y- and Z-scaling cannot be set differently.

Allow exploding: if not checked, prevents the block from being exploded.

Block unit: set the insertion unit for the block (see also: To insert a block).

Set the behavior of the selected entities after the block is created:

Retain: The selected entities remain in the drawing.

Convert to block: The selected entities are replaced by the newly created block.

Delete: The selected entities are deleted.

Click the *OK* button to create the block.

NOTE If the *Delete* option in step 6 was set, the OOPS command brings back the erased block entities.

To Create a Block Using the Blocks Explorer

Do one of the following:

Choose Blocks... in the Settings menu.

Type explblocks in the command bar, then press Enter.

The Drawing Explorer - Blocks dialog opens.
In the *Drawing Explorer* dialog do one of the following:

Click the *New* button ( ) on the *Details* toolbar.

Choose *New* in the *Edit* menu.

The *Drawing Explorer* dialog closes.
You are prompted: Insertion point for new block:
Specify the insertion point of the new block.
You are prompted: Select entities for block.
Select the entities for the new block.
Right click to create the block.
The selected entities are removed from the drawing.
The *Drawing Explorer* dialog reopens.
The *Name* field of the new block is selected.
Type a name in the *Name* field, replacing the default *NewBlock1* name.
Close the *Drawing Explorer* dialog.
(option) Type *oops* in the command bar, then press Enter to bring back the selected block entities.

To Save a Block to a Separate File

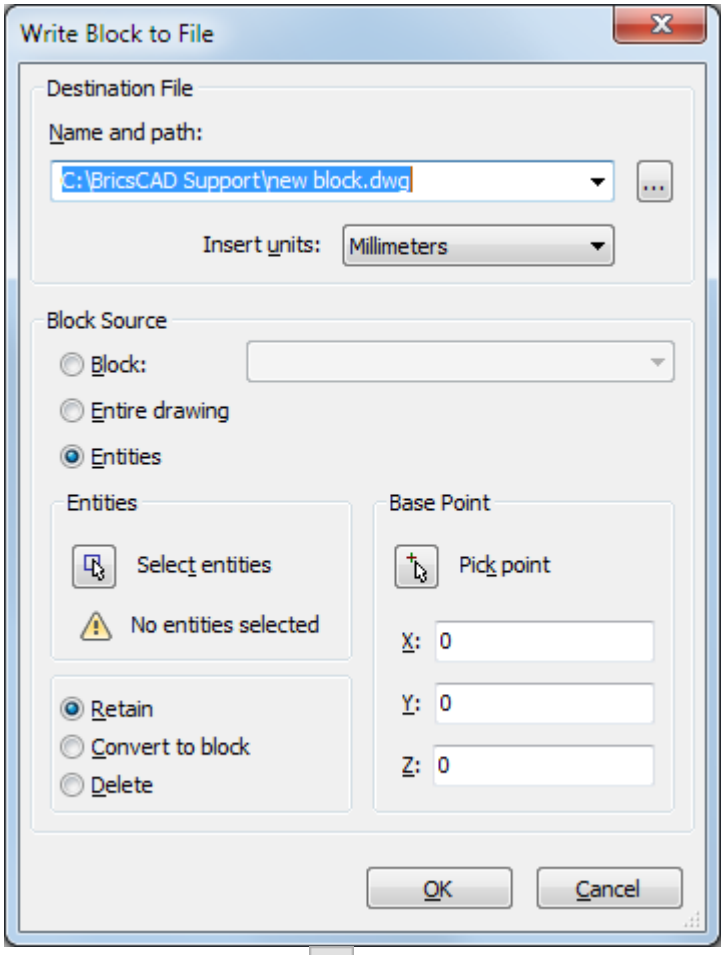
Do one of the following:

Click the *Save Block...* tool button ( ) on the *Tools* toolbar.

Choose *Save Block...* in the *Tools* menu.

Type *wblock* in the command bar, then press Enter.

The *Write Block to File* dialog opens.



Click the browse button (...) next to the *Name and path* field.
The *Save Block* dialog opens.

On the *Save Block* dialog, select a folder and type a name in the *File name* field, then click the *Save* button.
The *Save Block* dialog closes.

Insert units: set the insertion unit for the block (see also: To insert a block).

To select the source entities for the new block, do one of the following:

Click the *Block* radio button, then select a block from the drop-down list.

Click the *Model* space radio button, to select all the entities in the model space of the current drawing.

Click the *Entities* radio button, then click the *Select Entities* button ( ).
The *Write Block to File* dialog temporarily closes to let you select entities in the drawing.
Right click to stop selecting entities.

Click the *Pick point* button ( ) to define the *Base point* of the new drawing
or
Type the coordinates of the base point in the X-, Y- and Z-field.
Set the behavior of the selected entities after the block is created:

Retain: The selected entities remain in the drawing.

Convert to block: The selected entities are replaced by the newly created block.

Delete: The selected entities are deleted.

Click the *OK* button to create the drawing file.

NOTE If the *Delete* option in step 5 was set, the OOPS command brings back the erased block entities.

Inserting Blocks

Commands: INSERT, -INSERT, INSERTALIGNED and MINSERT

The *Insert* command lets you insert blocks through a dialog box.

The *-Insert* command inserts blocks by prompting in the command bar. BricsCAD will lookup the block name in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the *SRCHPATH* system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'

The *Insertaligned* command inserts a block entity, with easy alignment on existing entities. The *Insertaligned* command is similar to the *Insert* command, but during the placement the block will be dynamically aligned with existing lines, polylines, arcs or circles. The block is aligned with the entity to which the cursor snaps. If no entity is snapped, the block is aligned with the X-axis of the WCS.

The *Minsert* command Inserts a block as a rectangular array; combines the *-Insert* and *Array* commands (short for "multiple insertion").

You can choose to select an existing block definition or to insert an entire drawing as a block. When you insert a drawing, a new block definition is created in the current drawing. If you change the original drawing file, those changes have no effect on the current drawing unless you redefine the block by reinserting the changed drawing. When a block is inserted in a drawing, it is treated as a single entity.

TUTORIALS

Manage blocks with the Drawing Explorer

General procedure to insert a block:

Specify a block definition or drawing file.

Specify the insertion point.

Specify the scale.

Specify the rotation angle.

NOTES

- When inserting a drawing as a block, the base point, as defined by the *INSBASE* system variable, of the inserted drawing is the origin point of the block. The *INSBASE* system variable is set by the BASE command.
- If a block contains attributes you will be prompted to fill out the text for each attribute in the command bar.
- If the *DRAGOPEN* system variable is set to zero (OFF), drawings can be inserted as a block by dragging them from the *Windows Explorer* dialog into the current drawing; if *DRAGOPEN* = 1 (ON), the drawing will be opened instead.
- The *INSUNITS* system variable defines a drawing units value for automatic scaling when inserting blocks or attaching Xrefs.

In the *Insert Block* dialog:

The *Unit* field under *Block Unit* is set by the *INSUNITS* (Insertion Units) system variable in the source drawing or the *Block Unit* property of a block definition (see To create a block in a drawing).

The *Factor* field expresses the relation between the value of the *INSUNITS* variable in the source drawing and the target drawing. E.g. if *INSUNITS* is Millimeters in the source and the target, the value of the *Factor* field is 0.1.

If *INSUNITS* in the source drawing is *Unspecified*, *INSUNITSDEFSOURCE* is used instead.

If *INSUNITS* in the target drawing is *Unspecified*, *INSUNITSDEFTARGET* is used instead.
The values of *INSUNITSDEFSOURCE* and *INSUNITSDEFTARGET* are saved in the registry and therefore apply to all drawings in which *INSUNITS* is unspecified (unitless).

To Insert a Block

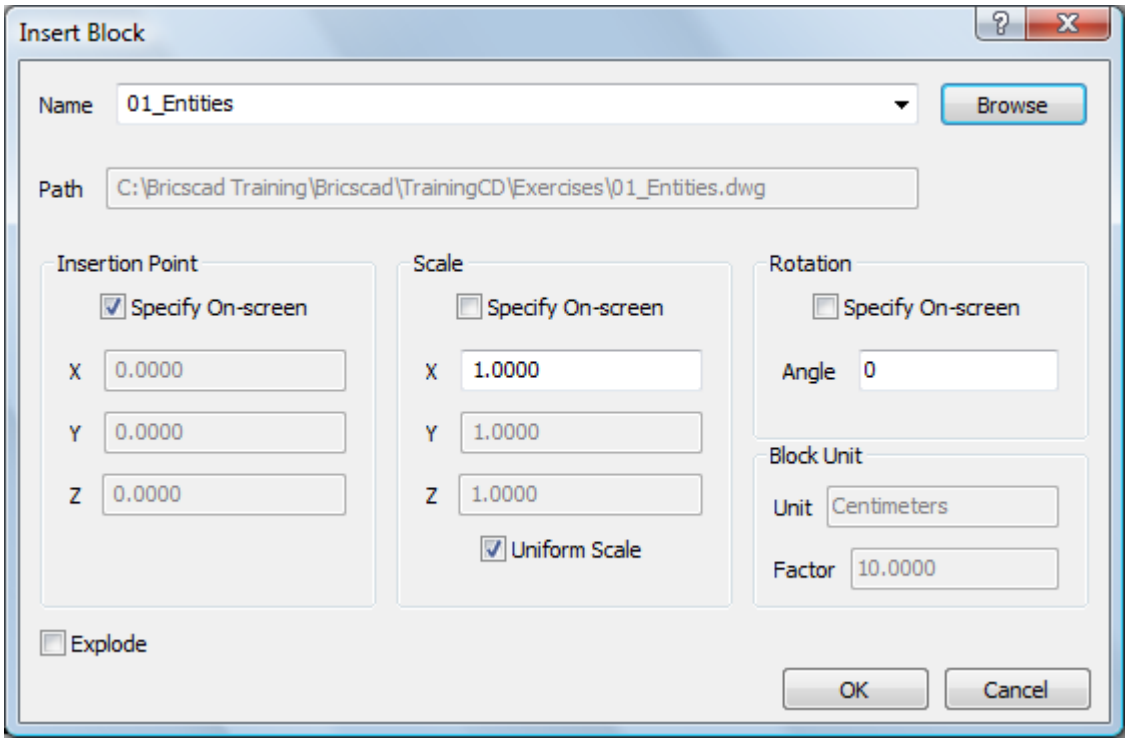
Do one of the following

Click the *Insert Block...* tool button ( ) on the *Insert* toolbar.

Choose *Insert Block...* in the *Insert* menu.

Type *insert* in the command bar, then press Enter.

The *Insert Block* dialog opens.



In the *Insert Block* dialog, do one of the following:

Select an existing block definition in the *Name* list.

Click the *Browse* button and select a drawing file.

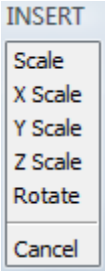
(option) Check the Explode option to explode the block after insertion.

(option) Uncheck the *Specify On-Screen* option, then specify the *Insertion Point* by keying in the coordinates in the *X*, *Y* and *Z* fields.

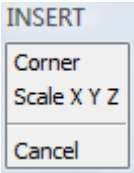
(option) Uncheck the *Specify On-Screen* option, then specify the *Scale* by keying in the scaling factors in the *X*, *Y* and *Z* fields.
Check the *Uniform Scale* option to equal the *X*, *Y* and *Z* scale factors.

(option) Check the *Specify On-Screen* option to specify the Rotation angle when inserting the block.

Click the *OK* button.
You are prompted: Insertion point for block.
A prompt menu displays.



Specify the insertion point.
You are prompted: Corner/XYZ/X scale factor <1.000000>:
A prompt menu displays.



Right click or press Enter to accept the default *X scale factor*.
You are prompted: Y scale factor: <Equal to X scale (1.000000)>:

Right click or press Enter to set the *Y scale factor* equal to the *X scale factor*.
The block is inserted.

To Insert a Block Using the Blocks Explorer

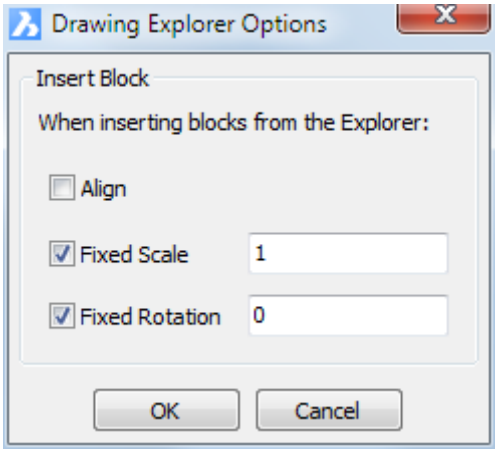
Open the Drawing Explorer - Blocks dialog.

(optional) Choose either *Detail View* ( ) or *Icon View* ( ).

(option) Edit the insert options:

Choose *Options...* in the *Settings* menu or select a block, then right click and choose *Options ...* in the context menu.

Set the insert options in the *Drawing Explorer Options* dialog box.



Do one of the following:

Select a block, then do one of the following:

Click the *Insert Block* button ( ) on the *Details* toolbar.

Right click and select *Insert* from the context menu

When in *Icon View*, double click the block.

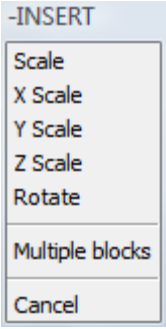
To select an external drawing, do one of the following:

Click the *Insert External* tool button ( ) on the *Details* toolbar.

Choose *Insert External* in the *Edit* menu.

Right click, then choose *Insert External* in the context menu.

The *Drawing Explorer - Blocks* window closes.
You are prompted: Multiple blocks/Scale/<Insertion point for block>:
A prompt menu displays



Follow the instructions in the command bar to insert the block.
The *Drawing Explorer - Blocks* window reopens.
(option) Insert more blocks.
Close the *Drawing Explorer - Blocks* window.

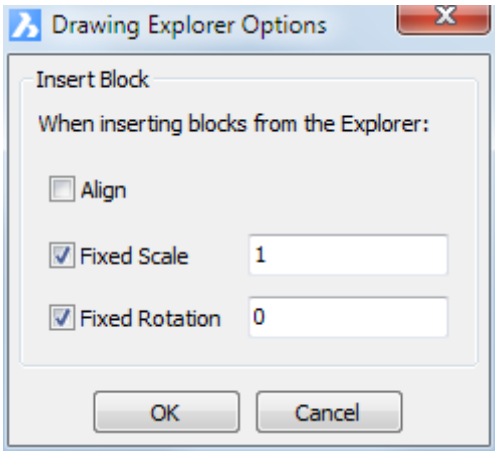
Inserting blocks from another drawing

Open the Drawings Explorer.
In the *Drawings* pane, click the *Folders* tab.
(option) If not yet available in the *Local Folders* tree, add the folder of the drawing.
Expand the folder.
Click the *Expand* icon (+) in front of the drawing.
The *Blocks* icon of the drawing displays.

Click the *Blocks* icon.
The blocks in the drawing display in the *Details* pane.
If necessary, click the *Icon View* button ( ) in the *Details* toolbar to see thumbnail images of the blocks.
(option) Edit the insert options:

Choose *Options...* in the *Settings* menu or select a block, then right click and choose *Options ...* in the context menu.

Set the insert options in the *Drawing Explorer Options* dialog box.



The available options are:

Align: If checked, launches the InsertAligned command to insert the block.
When the *Align* option is checked, the *Fixed Scale* and *Fixed Rotation* options are not available.

Fixed Scale: If checked, allows to insert the block at a fixed scale.

Fixed Rotation: If checked, allows to insert the block at a fixed rotation.

To insert a block, do one of the following:

Click the *Insert Block* button ( ) on the *Details* toolbar.

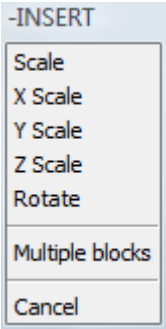
When in Icon View, double click the thumbnail image of the block.

When in Detail view, double click the number in front of the block name.

The *Drawing Explorer* dialog closes temporarily to let you insert the block in the drawing.

You are prompted: Multiple blocks/<Insertion point for block>:

A prompt menu displays



Follow the instructions in the command bar to insert the block.
The *Drawing Explorer* window reopens.

(option) Insert more blocks.

Close the *Drawing Explorer - Blocks* window.

To Insert a Block Aligned with an Entity

Make sure the appropriate Entity Snaps are active.

Type *insertaligned* or *insal* in the command bar, then press Enter.
You are prompted: ? to list blocks in drawing/~ to open the file dialog/<Block to insert> <name>:

Do one of the following:

Press Enter or right click to insert the most recently placed block again.

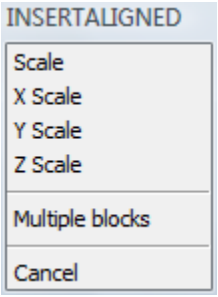
Type the name of an existing block definition in the command bar, then press Enter.

Type the name of an external block file in the command bar, then press Enter.

Type ~ (tilde) then press Enter to open the file dialog.

You are prompted: Multiple blocks/<Insertion point for block>:

A prompt menu displays.



Snap to an entity.
The block is aligned with the entity.

Click to define the insertion point.
When you move the crosshairs, the block is mirrored about the selected entity or a about a line tangent to the selected entity.

Click to specify the mirroring of the block.

NOTE If you type the name of block in step 3, BricsCAD will lookup the block in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the *SRCHPATH* system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'

NOTE The block is aligned as follows: The block is rotated so that its local Z-axis is parallel with the Z-axis of the active UCS, and its local X-axis should be parallel with the tangent to the entity at the position of the snap point. When you click a point on an entity the block is aligned with the entity. Then, by moving the mouse pointer around the insertion point, you can choose how the block should be mirrored. The origin point of the block lies on the entity.

To Insert Multiple Instances of a Block in a Rectangular Array

Type *minsert* in the command bar, then press Enter.
You are prompted:
? to list blocks in drawing/~ to open the file dialog/<Block to insert>:

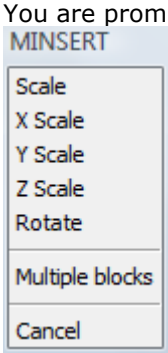
Do one of the following:

Type ~ (press and hold the *alt gr* key, then press the ~ key), then press Enter.
The *Insert Block* file dialog displays.
Select the drawing file you want to insert and press the *Open* button (or double click the file name).

Press Enter to accept the <Block to insert>.

Type a block name, then press Enter.
Optionally, type ? to display search for existing block definitions.

The block is attached to the cursor.
You are prompted: Multiple blocks/<Insertion point for block>:



Specify the insertion point.
You are prompted: Corner/XYZ/X scale factor <1.00>:

To define the X scale factor, do one of the following:

Press Enter to accept the default .

Type the X scale factor, then press Enter.

Click to define the X scale factor graphically.

You are prompted: Y scale factor: <Equal to X scale (current X scale)>:

Repeat step 5 to define the Y scale factor.
You are prompted: Rotation angle for block <0>:

Do one of the following:

Press Enter to set the rotation angle to 0°.

Type a rotation angle, then press Enter.

Click to define the rotation angle graphically.
You are prompted: Number of rows in the array <1>:

Type the number of rows, then press Enter or press Enter for one row.
You are prompted: Number of columns <1>:

Type the number of columns, then press Enter or press Enter for one column.
You are prompted: Vertical distance between rows, or spacing rectangle:

Type the row spacing distance, then press Enter.
You are prompted: Horizontal distance between columns:

Type the column spacing distance, then press Enter.
The block array is created as a single entity.

NOTES

- If you choose *Multiple blocks* in step 3, you are prompted to create multiple instances of the block array.
- It is not possible to explode a *MInsert Block* entity..
- If you type the name of block in step 2, BricsCAD will lookup the block in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the *SRCHPATH* system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'

Redefining Blocks

To redefine a block that was created in the current drawing, you create a new block using the same name.

If the block was inserted from a separate drawing, reinsert that block.

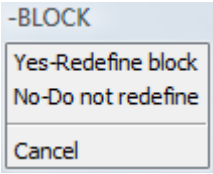
When a block definition is redefined all instances of that block in the current drawing are updated automatically.

See also: Editing blocks and external references.

To Redefine an Internal Block Definition

Type *-block* in the command bar, then press Enter.
You are prompted: Name for new block, or ? to list existing blocks:

Type the name of the block in the command bar and press Enter.
You are prompted: The block <name> already exists. Do you want to redefine it? <N>.
A prompt menu displays.



To redefine the block definition, do one of the following:

Type *Y*, then press Enter.

Choose *Yes-Redefine block* in the prompt menu.

You are prompted: Insertion point for new block:

Specify the insertion point of the new block.
You are prompted: Select entities for block.

Select the entities for the new block.

Right click to create the block.
The selected entities are removed from the drawing.
All existing instances of the block are updated.

(option) To bring back the selected block entities, type *oops* in the command bar, then press Enter.

To Reload an External Drawing as a Block

Do one of the following

Click the *Insert Block...* tool button ( ) on the *Draw* toolbar.

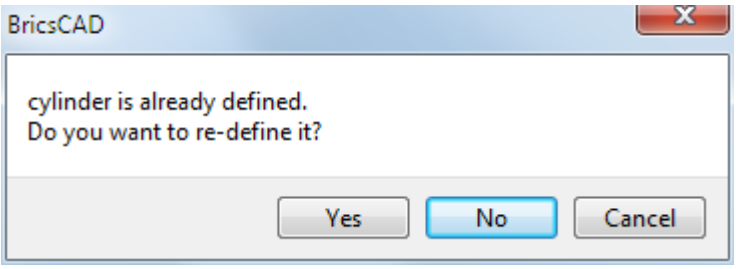
Choose *Insert Block...* in the *Draw* menu.

Type *insert* in the command bar, then press Enter.

The *Insert Block* dialog opens.

On the *Insert Block* dialog click the *Browse* button, then select the external drawing file.

Click the *Insert* button.
An alert box displays.



Click the *Yes* button to redefine the existing block definition.
You are prompted: Insertion point for block:

Continue with step 3 of the Inserting Blocks procedure.

Exploring Blocks

The *Explblocks* command opens the *Blocks* section of the *Drawing Explorer* dialog box (short for "explorer blocks").

In the *Blocks Explorer* you can:

- *Create* new blocks ( )
- *Delete* blocks ( )
- *Purge* unused block definitions ( )
- *Insert* blocks ( ) in the current drawing
- *Insert* external block files ( )
- *Save* a block as a (new) drawing ( )
- *Cut* ( ) or *Copy* ( ) a block definition, then *Paste* ( ) the block definition in another drawing
- *Rename* a block.

Open the Blocks Explorer

To open the *Blocks Explorer* do one of the following:

Choose *Drawing Explorer > Blocks...* in the *Tools* menu.

Type *expblocks* in the command bar.

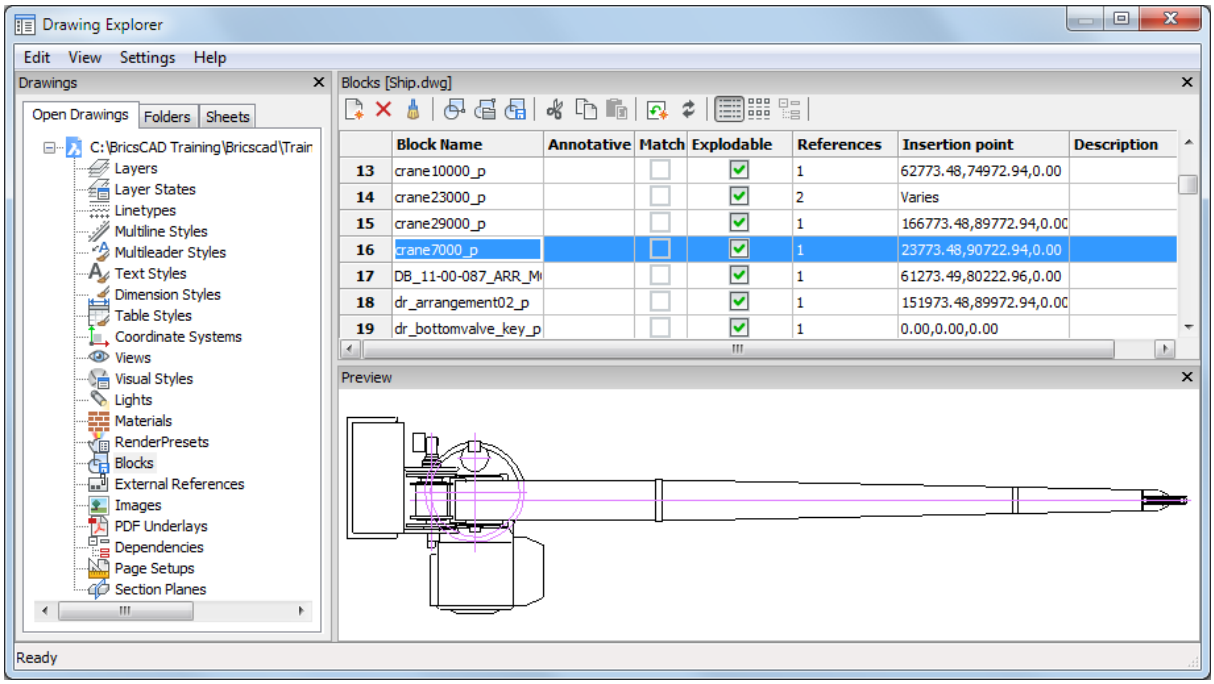
The Blocks Explorer display options

You can choose between *Detail View* (list) and *Icon View* (thumbnails) to see the blocks in the current drawing.

Open the Detail View of the Blocks Explorer

Click the *Detail View* button ( ) in the *Drawing Explorer* toolbar. The *Detail View* button is now pressed indicating the block details are displayed.

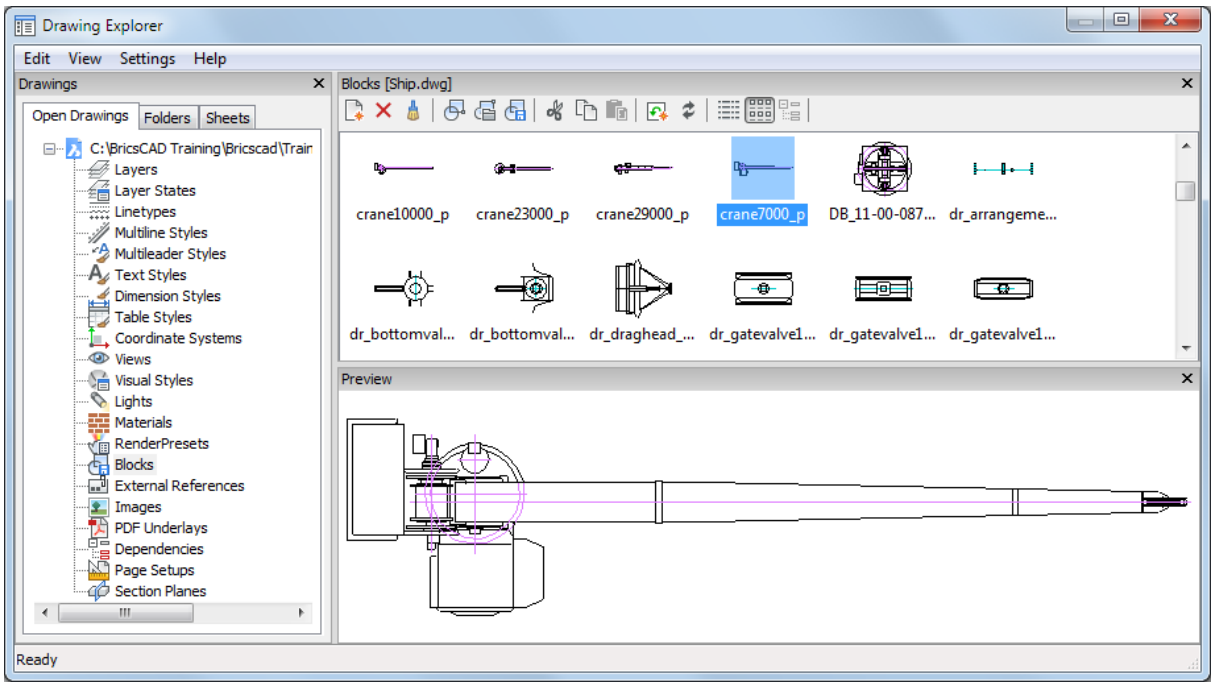
A preview of the selected block displays in the *Preview* sub-window of the *Drawing Explorer*.



Detail view of the Blocks Explorer

Open the Icon View of the Blocks Explorer

Click the *Icon View* button (⊞) in the *Drawing Explorer* toolbar. The *Icon View* button is now pressed indicating the block icons are displayed.
A preview of the selected block displays in the *Preview* sub-window of the *Drawing Explorer*.



Icon View of the blocks in the current drawing

To rename a block

Open the Blocks Explorer dialog.

Select the block, then click the *Block Name* field.

Type a new name in the *Block Name* field and press Enter.

Exploding Blocks

Command: EXPLODE

If the *Explodable* property of a block is set, you can explode an inserted block to its original component entities. Exploding a block affects that single instance of the block only. The original block definition remains in the drawing and you can still insert additional copies of the original block. If you explode a block that contains attributes, the attributes are lost, but the original attribute definitions remain.

Do one of the following:

Click the *Explode* tool button ( ) on the *Modify* toolbar.

Choose *Explode* in the *Modify* menu.

Type *explode* in the command bar, then press Enter.

You are prompted: Select entities to explode.

Select the block(s).

Press Enter or right click to explode the blocks.

NOTE Exploding dissociates component entities to their next simplest level of complexity: nested blocks or polylines in a block become blocks or polylines again.

Setting the Explodable property of a block

Do one of the following:

Choose *Drawing Explorer > Blocks...* in the *Tools* menu.

Type *expblocks* in the command bar, then press Enter

The *Drawing Explorer - Blocks* dialog displays.

If necessary, click the *Detail View* tool button ( ) in the *Drawing Explorer - Blocks* toolbar.

Click the checkbox in the *Explodable* field of a block to toggle the property.

Dynamic Blocks

BricsCAD partly supports dynamic blocks, you can:

- insert dynamic blocks from your existing dynamic blocks libraries
- edit the custom properties in the Properties Bar
- grip edit dynamic blocks

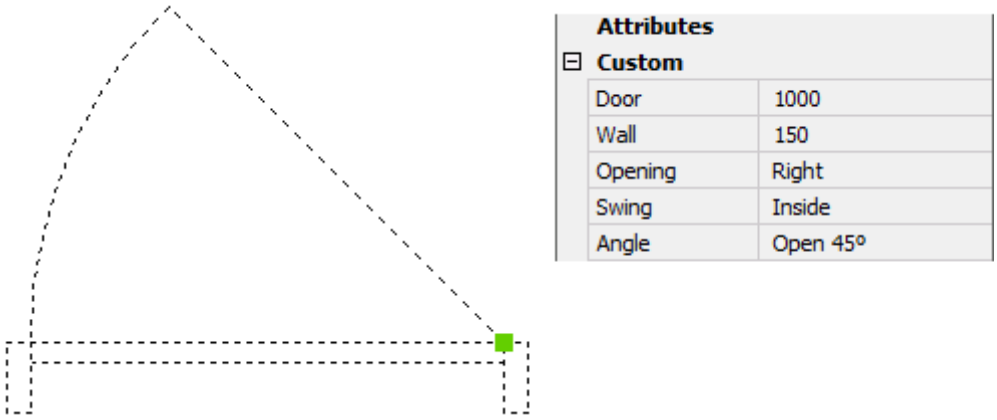
The creation of new dynamic blocks is not possible yet.

TUTORIALS

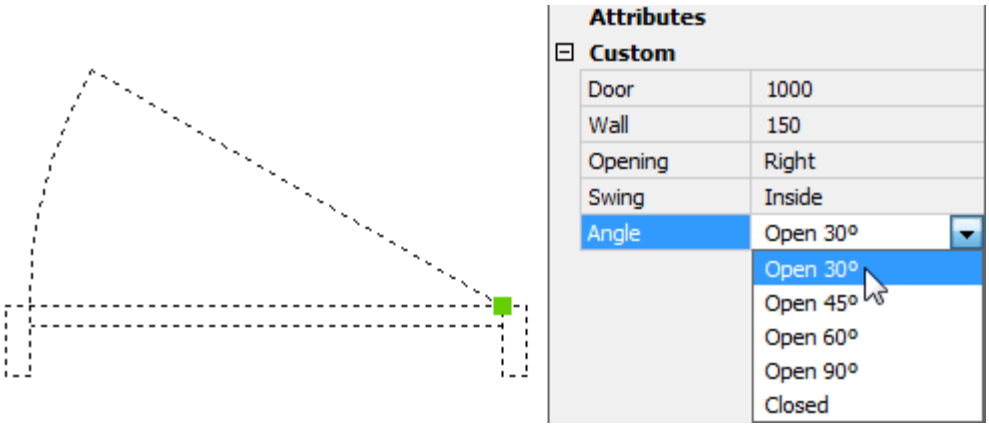
Dynamic Grips and Tips

Edit a Dynamic Block

Select the dynamic block in the drawing.
The properties of the selected block display in the Properties Bar.



Under *Attributes > Custom*: edit the properties needed.
The dynamic block is updated automatically.



Grip Editing Dynamic Blocks

Select the dynamic block in the drawing.
The following grip types display:

- Point
- XY
- Linear
- Polar
- Rotation
- Flip
- Visibility

Drag a grip to edit the block.
In the example below:

- Rotate the block (1): drag the grip to rotate the block dynamically or type a value in the dynamic entry field.
- Show/hide entities (2): click to select an optin.
- Flip (3): click to mirror the block
- Edit the length (4): select the grip to dynamically edit the length or type a value in the dynamic entry field.

Working with attributes

An attribute is a particular entity that you can save as part of a block definition. Attributes consist of text-based data. You can use attributes to track such things as part numbers and prices. Attributes have either fixed or variable values. When you insert a block containing attributes, the program adds the fixed values to the drawing along with the block, and you are prompted to supply any variable values.

After you insert blocks containing attributes, you can extract the attribute information to a separate file and then use that information in a spreadsheet or database to produce a parts list or bill of materials. You can also use attribute information to track the number of times a particular block is inserted into a drawing.

Attributes can be visible or hidden. Hidden attributes are neither displayed nor plotted, but the information is still stored in the drawing and written to a file when you extract it.

General procedure to work with attributes

Define the attributes.

Create a block.

You can attach attributes to a block. Include the attributes when the program prompts you to select the entities to be included in the block definition. If a block contains attributes, the program prompts you each time you insert the block, so you can specify different values for the attributes each time you insert it into a new drawing.
Insert the block in a drawing.

(option) Edit the attributes in a block.

Extract attribute information.

The *Attdef* command defines how a block's attributes will be created through a dialog box.

The *-Attdef* command creates block's attribute definitions at the command prompt.

General procedure to define attributes:

specify the characteristics of the attribute, including its name, prompt, and default value;

specify the location

define the text formatting

optionally: set the attribute flags (hidden, fixed, validate or predefined).

To Define an Attribute

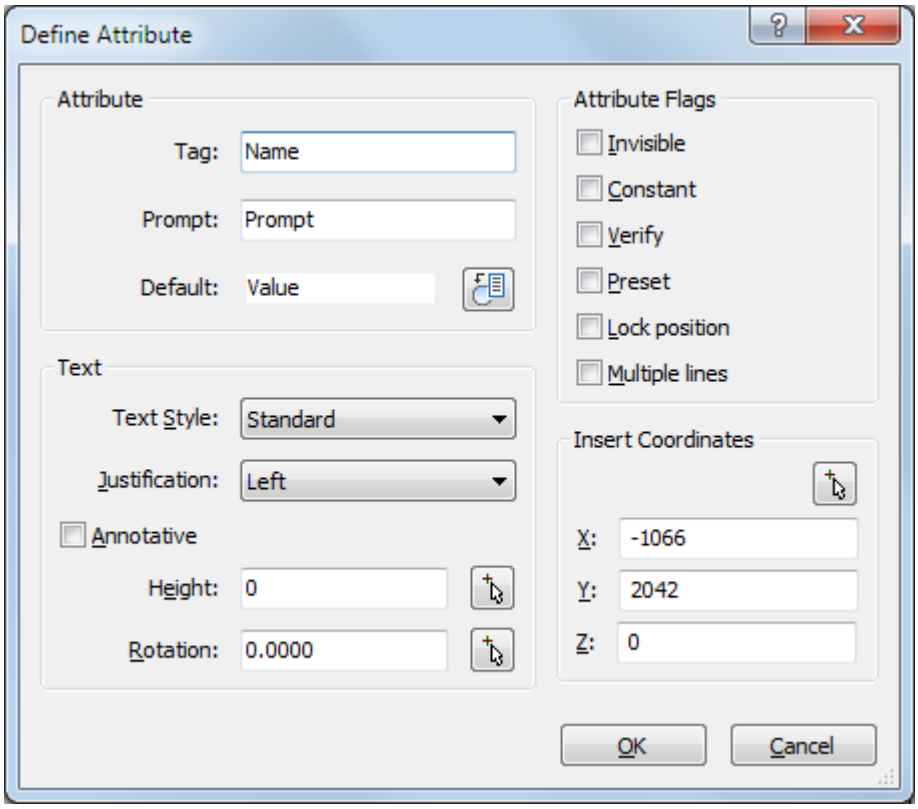
Do one of the following:

Click the *Define Attributes...* tool button ( ) on the *Tools* toolbar.

Choose *Define Attributes...* in the *Tools* menu.

Type *attdef* in the command bar, then press Enter.

The *Define Attribute* dialog opens.



Define the Attribute:

Type a name in the *Tag* field.
The *Tag* Identifies each occurrence of an attribute in the drawing. The name you type in the *Tag* field can be any combination of characters except spaces. Lowercase letters are automatically converted to uppercase.

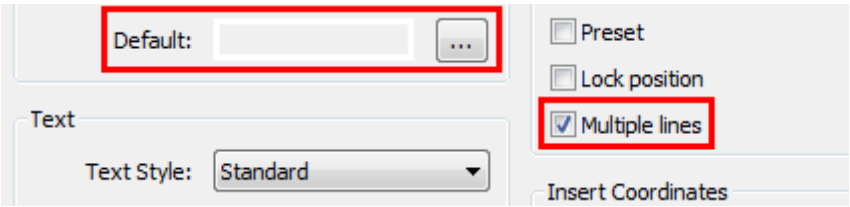
Type a prompt in the *Prompt* field.
The attribute prompt displays when you insert a block containing the attribute. If a prompt is omitted, the *Tag* is used as the prompt instead.

Do one of the following:

Type a default value in the *Default text* field.
For variable attributes, the default value is replaced by the actual value when you insert a block containing the attribute.

Click the *Insert Field* button ( ) to assign a field value to the attribute (see Using sheet set and sheet properties in a title block).

If the *Multiple lines* flag is checked, the *Default text* field is dimmed.



Click the button next to the *Default text* field.
The *Define Attribute* dialog temporarily closes to let you create a Mtext entity.

Do one of the following:

Specify the x-, y-, and z-coordinates for the attribute insertion point.

Click the *Pick Point* button ( ) to specify the insertion point in the drawing.
You are prompted: Select insertion point.
The *Define Attribute* dialog closes temporarily.

(option) Set the *Invisible* attribute flag.
Attributes with the *Invisible* flag set do not display nor print.
If the *ATTMODE* (Attribute Display Mode) variable is set to *2*, all attributes display, including *Hidden* attributes.

(option) Set the *Constant* attribute flag.
Attributes with the *Constant* flag set cannot be edited.

(option) Set the *Verify* attribute flag.
Attributes with the *Verify* flag set must be explicitly validated when a block containing such attributes is inserted.

(option) Set the *Preset* attribute flag.
When inserting a block you are not prompted to define attributes of which the *Preset* flag is set. You can edit attribute afterwards though.

(option) Set the *Lock Position* attribute flag to lock the position of the attribute within the block reference.
Unlocked attributes can be moved relative to the rest of the block using grip editing and multiline attributes can be resized.

Define the *Text* properties: *Text Style*, *Justification*, *Annotative*, *Text height* and *Rotation*.

Click the *OK* button to create the attribute in the drawing.
The *Define Attribute* dialog closes.

(option) Right click to reopen the *Define Attribute* dialog, then repeat steps 2 and 3 to create another attribute.

Sheet Set Properties in a Title Block

Create the sheet set.

Define the custom sheet set and sheet properties.

Open a sheet (layout) from the sheet set.

Create the title block source entities in the paper space layout:

- Create the borders, line work and fixed texts.
- Create attributes with fields that refer to sheet set, subset and sheet properties (see To define an attribute).

See the SHEETSET command to learn more about default and custom sheet set, subset and sheet properties.

Custom sheet set properties:

Sheet Set Custom Properties	
Architect	Bricsys & Partners
ArchitectAddress	Bellevue 5/201
ArchitectCity	9050 Gent
ArchitectPhone	09 244 01 91
Contractor	Robert Stone ltd.
ContractorAddress	123 Main Street
ContractorCity	New Town
Owner	Carpenter
OwnerAddress	1 Church Road
OwnerCity	New Town
OwnerTitle	Mr. & Mrs.
SiteAddress	48 Brick Lane
SiteCity	New Town
SiteReference	Section A no. 123

Sheet properties and sheet custom properties:

Sheet	
Number	08
Title	Elevation South
Description	South elevation drawing
Publish	☑ Include for Plot/Publish
Layout	LAYOUT1 (C:\BricsCAD Trainir
Revision Number	1
Revision Date	15/08/2013
Issue Purpose	Final Design
Category	Elevation
Sheet Custom Properties	
Date	12/08/2013
PaperSize	A2
Printer	pdf
Remark	none
Scale	1/50

Create a block from the title block source entities.

Copy the block to drawing in one of your block folders (see To insert a block form another drawing).

Delete the title block source entities.

The title block can be inserted in the other sheets of the sheet set and in any other sheet set where the same custom sheet set and sheet properties exist. All fields that refer to default or custom sheet set, subset or sheet properties are filled out automatically. If no value is available, a field displays hyphens (----) (see also Placing blocks with attributes).

Editing Attribute Definitions

Command: DDEDIT

The *Ddedit* command edits single-line text, multi-line text, attribute definitions, and attribute text (short for "dynamic dialog editor").

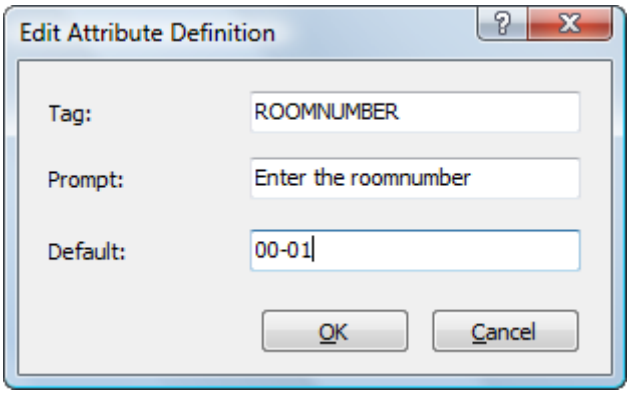
You can edit an attribute definition only before it is saved as part of a block definition.

Do one of the following:

Type *ddedit* in the command bar, then press Enter and select the attribute.

Double click the attribute.

The *Edit attribute definition* dialog opens.



(option) Type a new name in the *Tag* field.
Lowercase letters are converted to uppercase automatically.

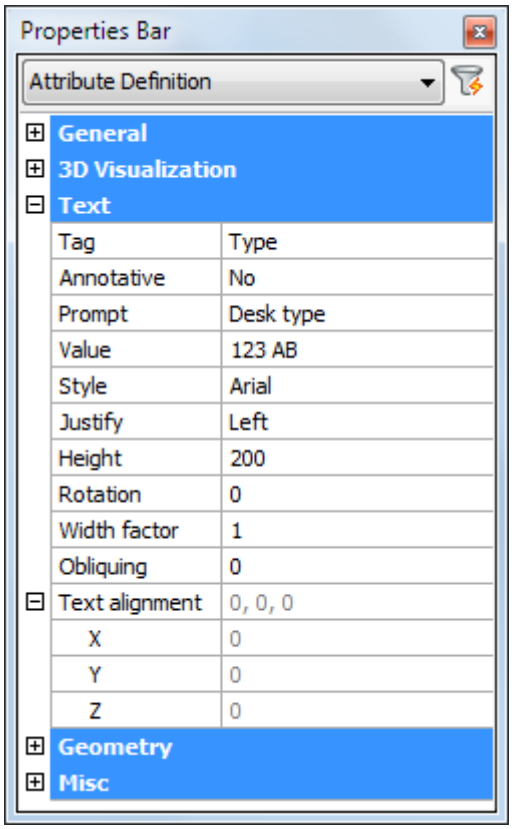
(option) Type a new prompt in the *Prompt* field.
If you leave the *Prompt* field empty, the attribute *Tag* will be used as the prompt instead.

(option) Type a new text in the *Default* field.

Press the *OK* button to save your changes.

To edit an attribute definition

Click the attribute.
The attribute properties display in the Properties Bar.



Modify the attribute properties in the *Properties Bar* fields.

Press the Esc key to stop editing.

NOTE Make sure the Properties Bar is open before starting this procedure.

Using the Attribute Editor

Command: EATTEDIT

The *Eattedit* command edits the attributes in blocks. You are prompted to select a block with editable attributes. When you select such block, the *Attribute Editor* dialog displays. If the selected block does not contain attributes or if the selected entity is not a block you are prompted to select another block.

Do one of the following

Click the *Edit Block Attributes...* tool button ( ) on the *Attributes* toolbar.

Choose *Edit Block Attributes...* in the *Tools | Attributes* menu.

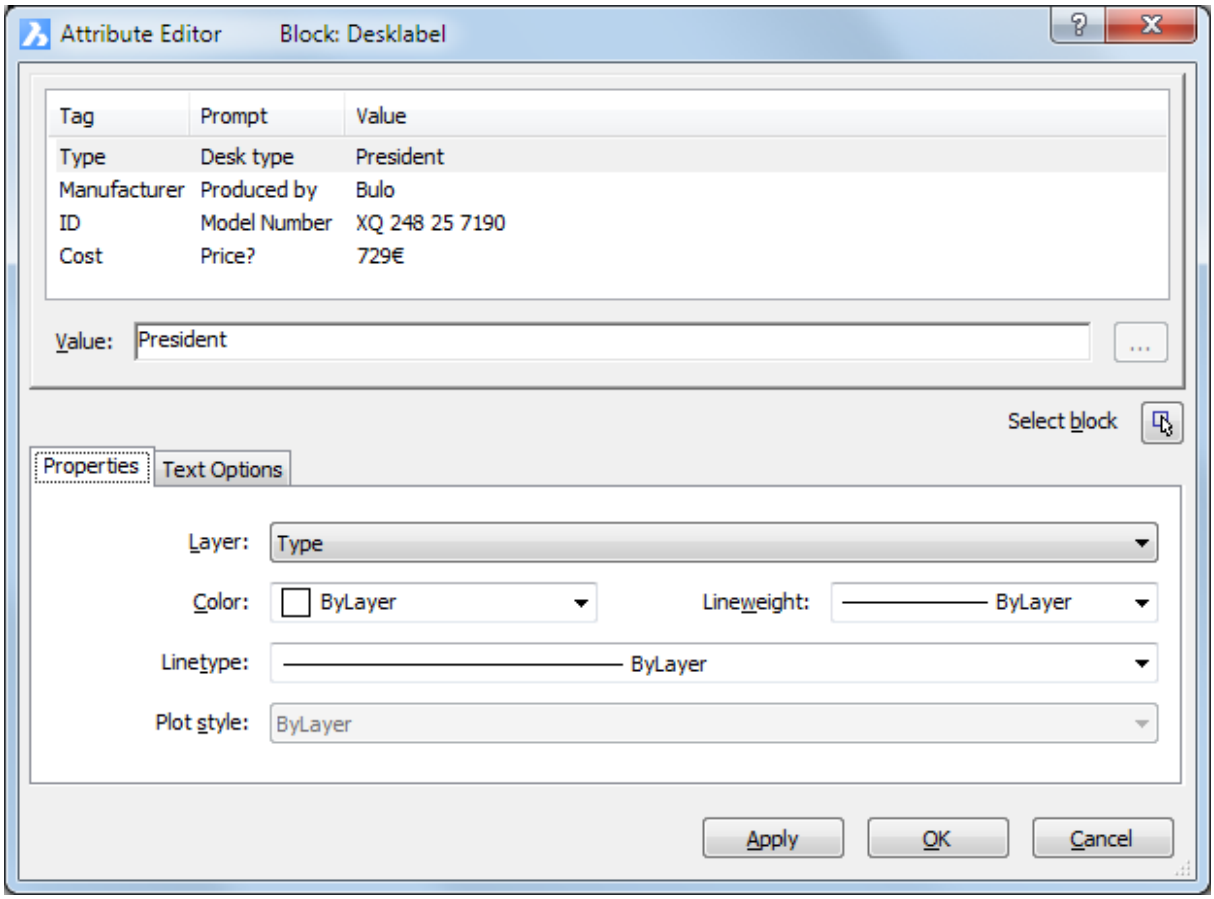
Type *eattedit* in the command bar, then press Enter.

You are prompted: Select a block:

or

Double click a block containing attributes, then go to step 3.

Select a block with attributes.
The *Attribute Editor* dialog opens.

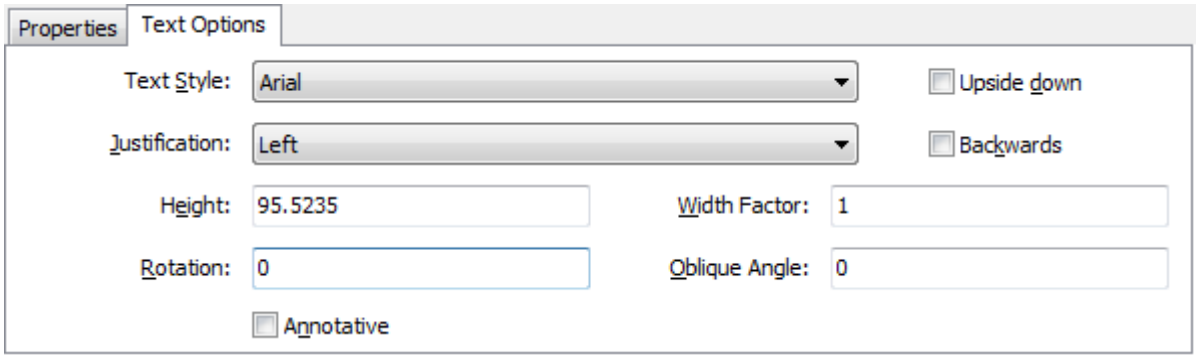


Select an attribute in the list.
The current value of the selected attribute displays in the *Value* field.

(option) Type a new value in the *Value* field.

(option) Click the *Properties* tab to modify *Layer*, *Color*, *Lineweigth* and/or *Linetype*.

(option) Click the *Text Options* tab to edit the text properties.



(option) Click the *Apply* button to apply the changes.

(option) Repeat steps 3 through 6 to edit another attribute.

Do one of the following:

Click the *OK* button to close the *Attribute Editor* dialog and apply the changes.

Click the Apply button to apply the changes and keep the *Attribute Editor* dialog open.

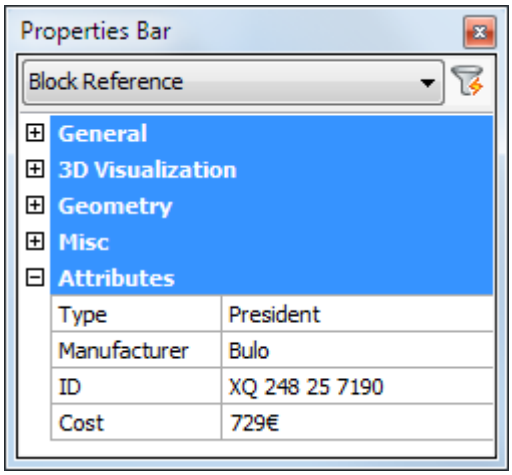
Click the *Cancel* button to close the *Attribute Editor* dialog. Changes that are not applied yet are not saved.

Click the *Select* button ( ) to select another block.
The *Attribute Editor* dialog temporarily closes to let you select another block in the drawing.

Editing attributes in the Properties bar

Select a block containing attributes.
The current value of the attributes display in the Properties bar.

Click the attribute you want to edit.
The settings field of the selected attribute is active.



Type a new value, then press enter or select another attribute.

Placing Blocks with Attributes

Commands: INSERT and -INSERT

When you place a block which contains attributes, you will be prompted to define the attributes in the command bar or through a dialog box.

If the ATTDIA system variable is ON, attributes display in a dialog box.

If the ATTDIA system variable is OFF, you are prompted in the command bar to define the attributes.

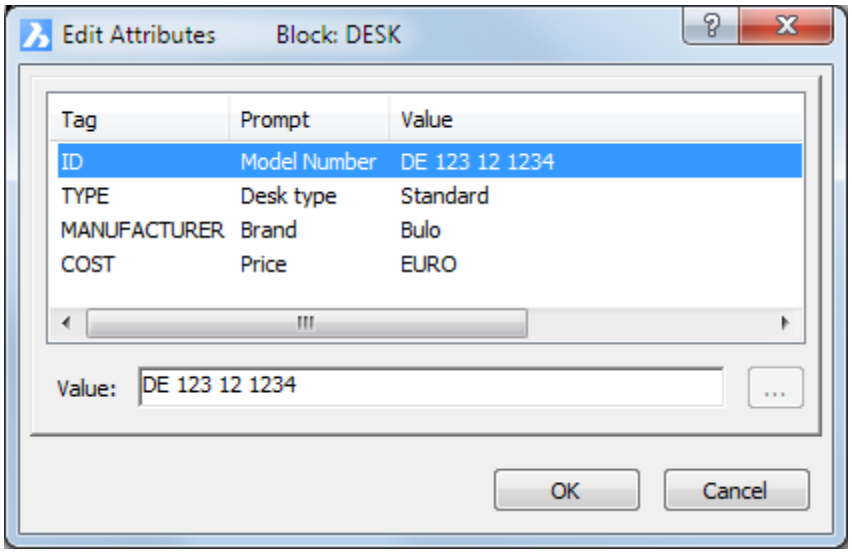
To place a block with attributes

Launch the Insert Block... tool.

Position the block in the drawing.
All block entities are placed in the drawing, except the attributes.

You are prompted: <Attribute Prompt> <Default Text>:

A dialog box displays:



Do one of the following:

In the command bar: The command bar reads: <Attribute Prompt> <Default Text>:

Press Enter or right click to accept the default text.

Type a new text in the command bar, then press Enter and define the next attribute.

In the *Edit Attributes* dialog box:

Select an attribute in the list.
The current value displays in the *Value* field.

Type a new value in the *Value* field.

Press *OK* when all attributes are properly defined.

Extracting Attribute Information

Command: ATTEXT and - ATTEXT, DATAEXTRACTION

The *Attext* command copies data from attributes to a text file via a dialog box.

The *-Attext* command copies data from attributes to a text file via the command line.

The text file can the be imported in a database or a spreadsheet.

You can save the file in any of the following formats:

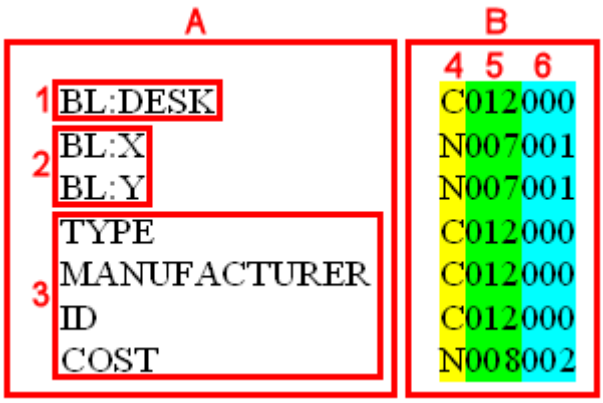
Comma Delimited Format (CDF): Contains one line for each instance of a block, with individual attribute fields separated by commas. Character string fields are enclosed with single quotation marks. You must specify a template file when extracting to a CDF file.

Space Delimited Format (SDF): Contains one line for each instance of a block. Each attribute field has a fixed length; there are no separators or character string delimiters. You must specify a template file when extracting to a SDF file.

Drawing Exchange Format (DXF): Creates a subset of a standard DXF file (a *.dxx file) containing all the information about each block, including the insertion points, rotation angles, and attribute values. No template file is required.

Before extracting attributes to a CDF or SDF file, you must create a template file. The template file is an ASCII text file that specifies the attribute data fields to be written in the extract file. Each line of the template file specifies one attribute field. BricsCAD recognizes 15 different fields, which contain elements such as the block name, the x-, y-, and z-coordinates of its insertion point, the layer on which it is inserted, etc. You can include any of these fields. The template file must include at least one attribute name.

Each line in the template file must start with the field name. Block name and insertion-point values must begin with BL:. The next nonblank character must be either a C (indicating a character string field) or an N (indicating a numeric field). This character is then followed by three digits indicating the width of the field (in characters). The final three digits indicate the number of decimal places (for numeric fields). In the case of character fields, the last three digits must be zeros (000). A typical template file is similar to the one shown here:



A. Field name
B. Field format
1. Block name (must be preceded by *BL:*)
2. Coordinate fields (must be preceded by *BL:*)
3. Attribute tags
4. Indicates the field data type: *C* for character fields, *N* for numeric fields
5. Field width (number of characters)
6. Number of decimal places for numeric fields, *000* for character fields

To extract attribute information

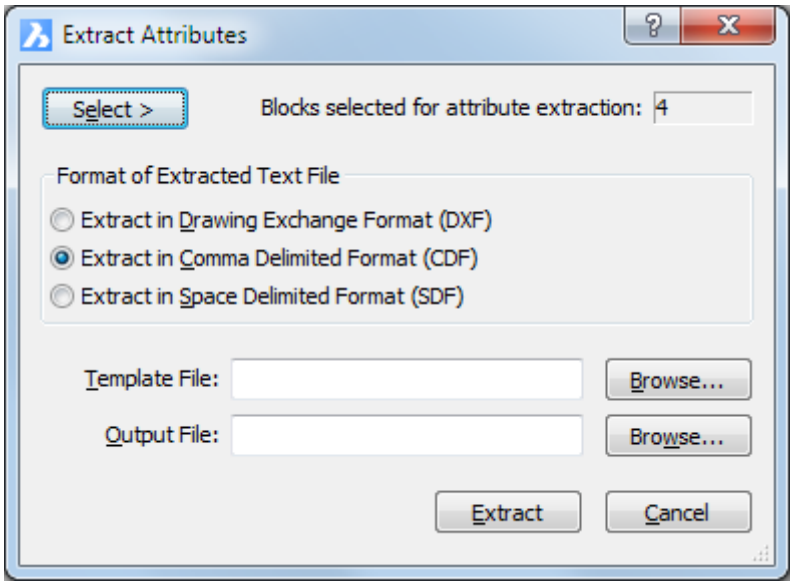
Do one of the following:

Click the *Extract Attributes* tool button on the *Tools* toolbar.

Choose *Extract Attributes* in the *tools* menu.

Type *attext* in the command bar, then press Enter.

The *Extract Attributes* dialog opens.



Click the *Select* button.
The *Extract Attributes* dialog closes temporarily.

Select the blocks, then press Enter or right click.
The *Extract Attributes* dialog reopens.

Select the text file format: *DXF* (drawing exchange format), *CDF* (comma delimited format) or *SDF* (space delimited format).

Select the *Template file*.

Specify the *Output file* name.

Click the *Extract* button.
You are prompted: <number> records extracted.

Working with external references

You can link entire drawings to the current drawing as external references. Unlike inserting a drawing as a block, in which you add all the entities from the separate drawing into the current drawing, external references attach a pointer to the external file. The entities in the external reference appear in the current drawing, but the entities themselves are not added to the drawing. Thus, attaching an external reference does not significantly increase the size of the current drawing file.

External references provide additional capabilities not available when you insert a drawing as a block. When you insert a drawing as a block, the entities are stored in the drawing. Any changes you make to the original drawing are not reflected in the drawing in which you inserted it. When you attach an external reference, however, any changes you make to the original drawing file are reflected in the drawings that reference it. These changes appear automatically each time you open the drawing containing the external reference. If you know that the original drawing was modified, you can reload the external reference anytime you're working on the drawing.

External references are useful for assembling master drawings from component drawings. Use external references to coordinate your work with others in a group. External references help reduce drawing file size and ensure that you are always working with the most recent version of a drawing.

Commands: XREF, -XREF, XATTACH and XCLIP

The *Xref* command attaches DWG files to the current drawing through the Drawing Explorer (short for "external reference").

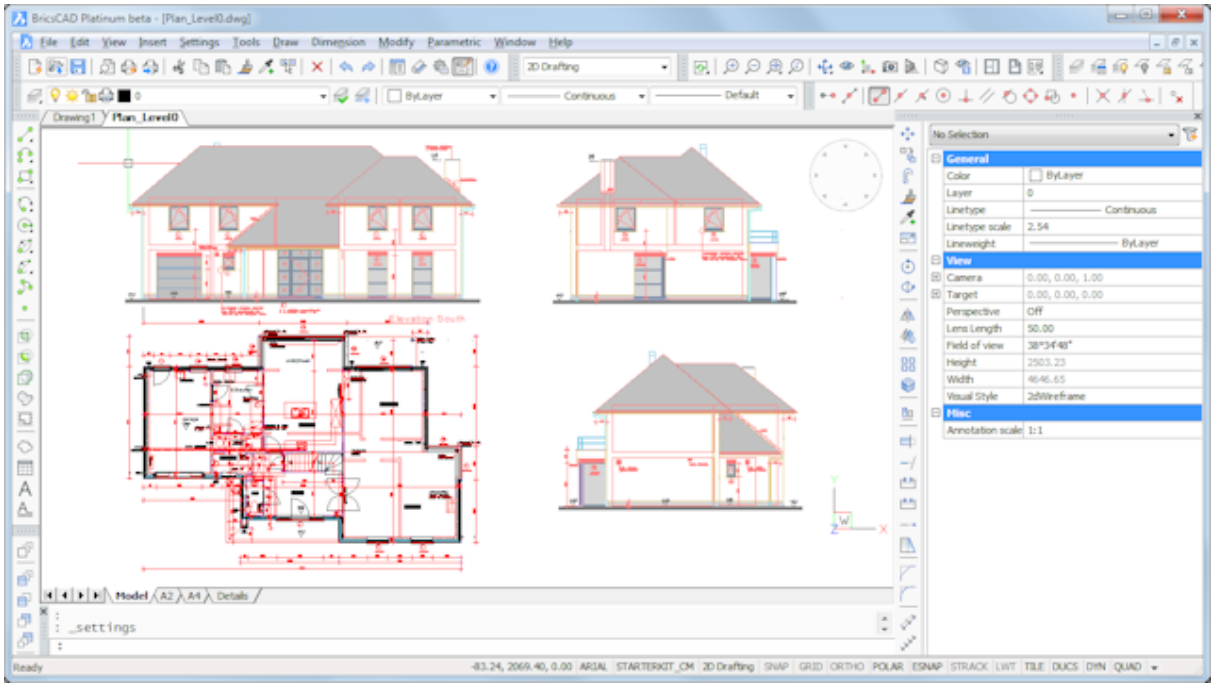
The *-Xref* command attaches DWG files to the current drawing through the command bar.

The *Xattach* command attaches externally-referenced drawings through a dialog box (short for "external attach").

The *Xclip* command clips externally-referenced drawings, and adjust front and rear clipping planes (short for "xref clip").

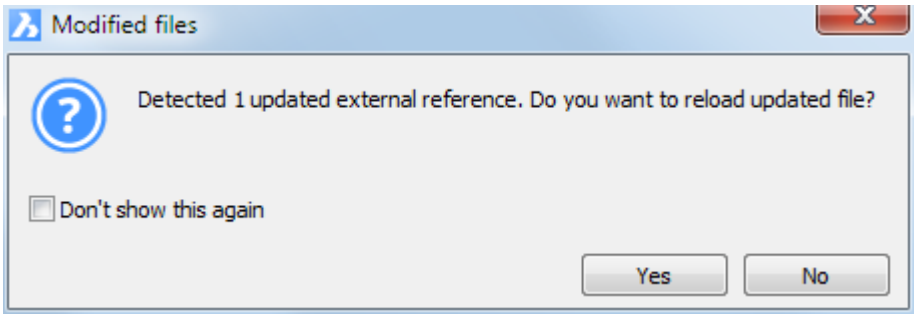
The *XDWGFADECTL* system variable controls the fading of exnternal references. Values between 0 (no fading) and 90 are accepted.

The fading factor affects the screen display only, not the printing.



External References (elevations) display faded: XDWGFADECTL + 50

The *XNOTIFYTIME* system variable defines the frequency to check for modified externally referenced resources: xrefs, images and pdf attachments if *XREFNOTIFY, IMAGENOTIFY* and/or *PDFNOTIFY* is ON.



Command: XATTACH

The *Xattach* command attaches a drawing as an external reference (xref). The attached xref appears as an insert in the current drawing. Like blocks, attached xrefs can be nested. The attached drawing is based on the most recently saved version.

The origin point used when attaching an external reference is controlled by the *INSBASE* (insertion base point) settings variable. Use the BASE command to define the *INSBASE* variable in the external reference drawing.

To attach an external reference

Do one of the following

Click the *Attach Xref...* tool button ( ) on the *Insert* toolbar.

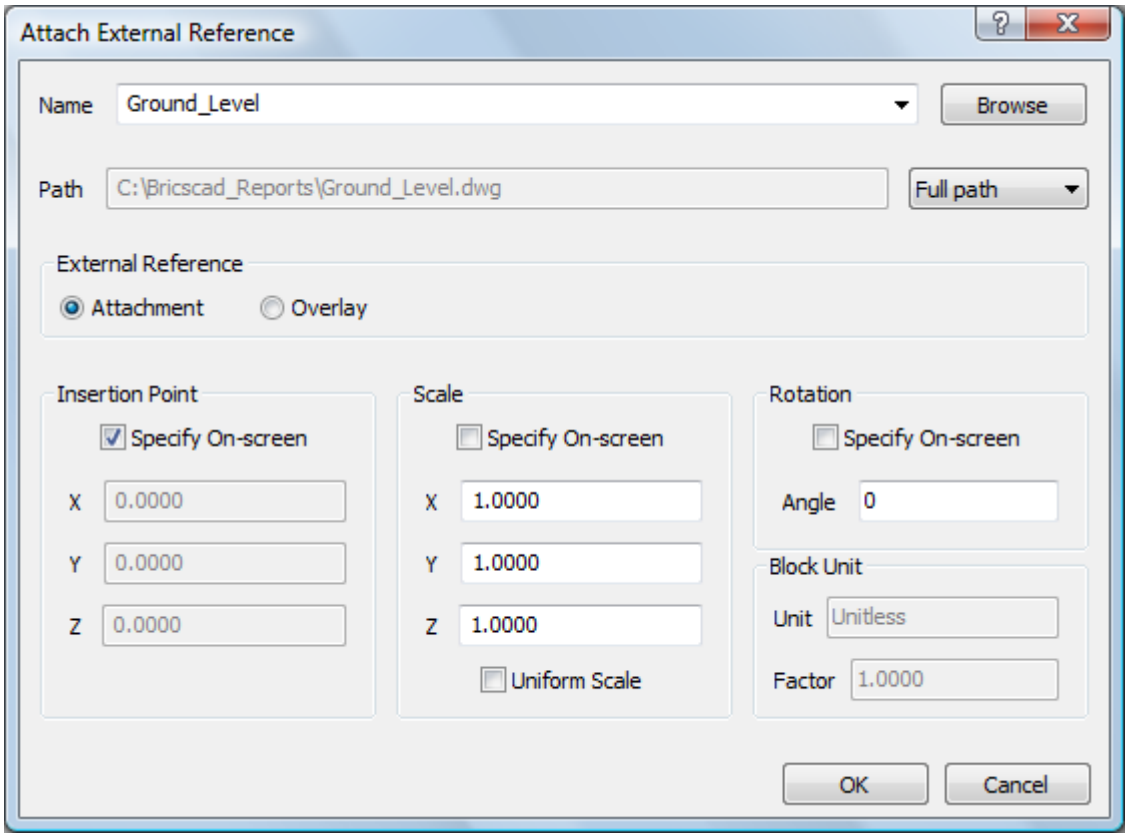
Choose *Attach Xref...* in the *Insert* menu.

Type *xattach* in the command bar, then press Enter.

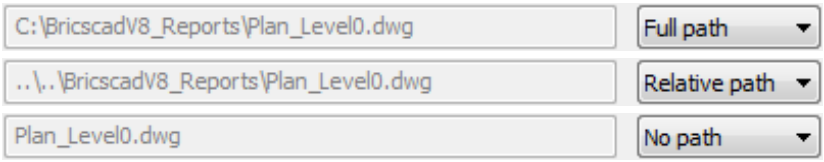
Click the *Attach Xref* tool button ( ) on the *Details* toolbar on the Drawing Explorer / Xrefs window.

The *Attach External Reference* dialog window displays.

Select the drawing you want to attach, then press the *Open* button on the *Open* dialog window. The *Attach External Reference* dialog window opens:



The full path to the selected drawing displays in the *Path* field.
(option) Select the *Path Type*:



Choose the *External Reference Type*:

Attachment: If the current drawing is attached as an xref to another drawing, an attachment becomes a nested xref.

Overlay: An overlay is ignored when the drawing to which it is attached is then attached as an xref to another drawing.

Choose whether to specify the *Insertion Point* on screen or in the X-, Y-, Z-fields.

Choose whether to specify the *Scale* on screen or in the X-, Y-, Z-fields.
Check the *Uniform Scale* option to specify the scale in the X-field.

Choose whether to specify the *Rotation* angle on screen or in the *Angle* field.

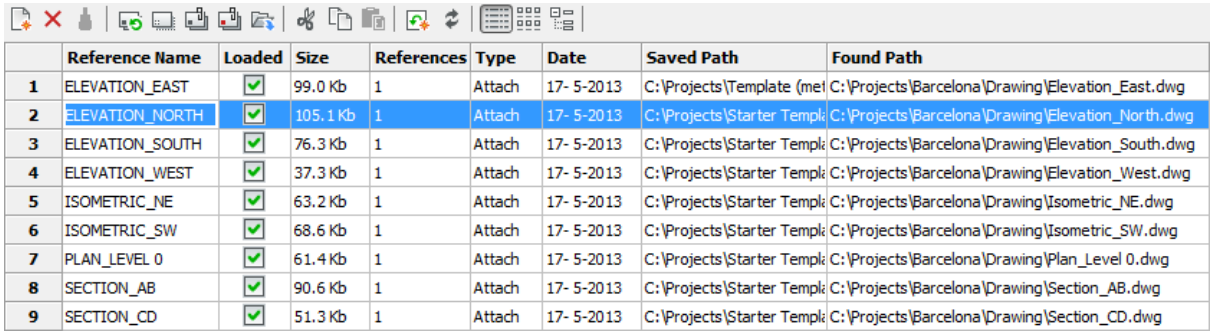
Click the *OK* button to attach the Xref.
Depending on the options in steps 5, 6 and 7 you are prompted to specify the insertion point, scale and/or rotation angle.

NOTE The *Unit field* under *Block Unit* is set by the *INSUNIT* (Insertion Units) system variable in the external reference. The *Factor* field expresses the relation between the value of the *INSUNIT* system variable in the xref and the parent drawing.

Managing Xrefs

Command: XREF

The *Xref* command opens the *External References* section of the *Drawing Explorer* dialog box.



	Reference Name	Loaded	Size	References	Type	Date	Saved Path	Found Path
1	ELEVATION_EAST	✓	99.0 Kb	1	Attach	17- 5-2013	C:\Projects\Template (mel	C:\Projects\Barcelona\Drawing\Elevation_East.dwg
2	ELEVATION_NORTH	✓	105.1 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templa	C:\Projects\Barcelona\Drawing\Elevation_North.dwg
3	ELEVATION_SOUTH	✓	76.3 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templa	C:\Projects\Barcelona\Drawing\Elevation_South.dwg
4	ELEVATION_WEST	✓	37.3 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templa	C:\Projects\Barcelona\Drawing\Elevation_West.dwg
5	ISOMETRIC_NE	✓	63.2 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templa	C:\Projects\Barcelona\Drawing\Isometric_NE.dwg
6	ISOMETRIC_SW	✓	68.6 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templa	C:\Projects\Barcelona\Drawing\Isometric_SW.dwg
7	PLAN_LEVEL 0	✓	61.4 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templa	C:\Projects\Barcelona\Drawing\Plan_Level 0.dwg
8	SECTION_AB	✓	90.6 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templa	C:\Projects\Barcelona\Drawing\Section_AB.dwg
9	SECTION_CD	✓	51.3 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templa	C:\Projects\Barcelona\Drawing\Section_CD.dwg

The *Drawing Explorer - External References* dialog allows to:

Attach a drawing as an external reference ( ).

Detach an external reference ( ).

Reload an external reference ( ).

Unload an external reference ( ).

Bind an external reference ( ).

Insert an external reference ( ).

Modify the attachment type: *Attach* or *Overlay*.

Modify the *Saved Path* of an xref.

Modify the *Found Path* of an overlay.

To detach external references

Select the xref(s) you want to detach.
Press and hold the Ctrl key to select multiple xrefs or to unselect a selected xref.

Click the *Detach Xref* tool button ( ) on the *Details* toolbar.
The selected xref(s) are detached. You are not prompted to confirm the detachment.

To unload external references

Do one of the following:

Click the *Loaded* column of a loaded Xref.

Select the Xref(s), then either click the *Unload Xref* tool button ( ) on the *Details* toolbar or click the *Loaded* column of one of the selected Xrefs.

To reload external references

Do one of the following:

Click the *Loaded* column of an unloaded Xref.

Select the Xref(s), then either click the *Reload Xref* tool button ( ) on the *Details* toolbar or click the *Loaded* column of one of the selected Xrefs.

Editing Blocks and External References

Commands: REFEDIT, -REFEDIT, REFSET and REFCLOSE

The *Refedit* command edits block references and externally-referenced drawings (Xrefs) through a dialog box (short for "reference editor"). *Refedit* is referred to as *in-place reference editing*. *RefEdit* either works on a portion of the reference or on the entire drawing or all entities in the block.

The *-Refedit* command edits block references and externally-referenced drawings through the command bar.

The *Refset* command adds and removes entities from the reference being edited.

The *Refclose* command closes the reference editor.

The *XFADECTL* system variable sets the fading level for the entities which are not in the working in the current refedit session. Values between 0 (no fading) and 90 are accepted.

To edit Xrefs or blocks

Do one of the following:

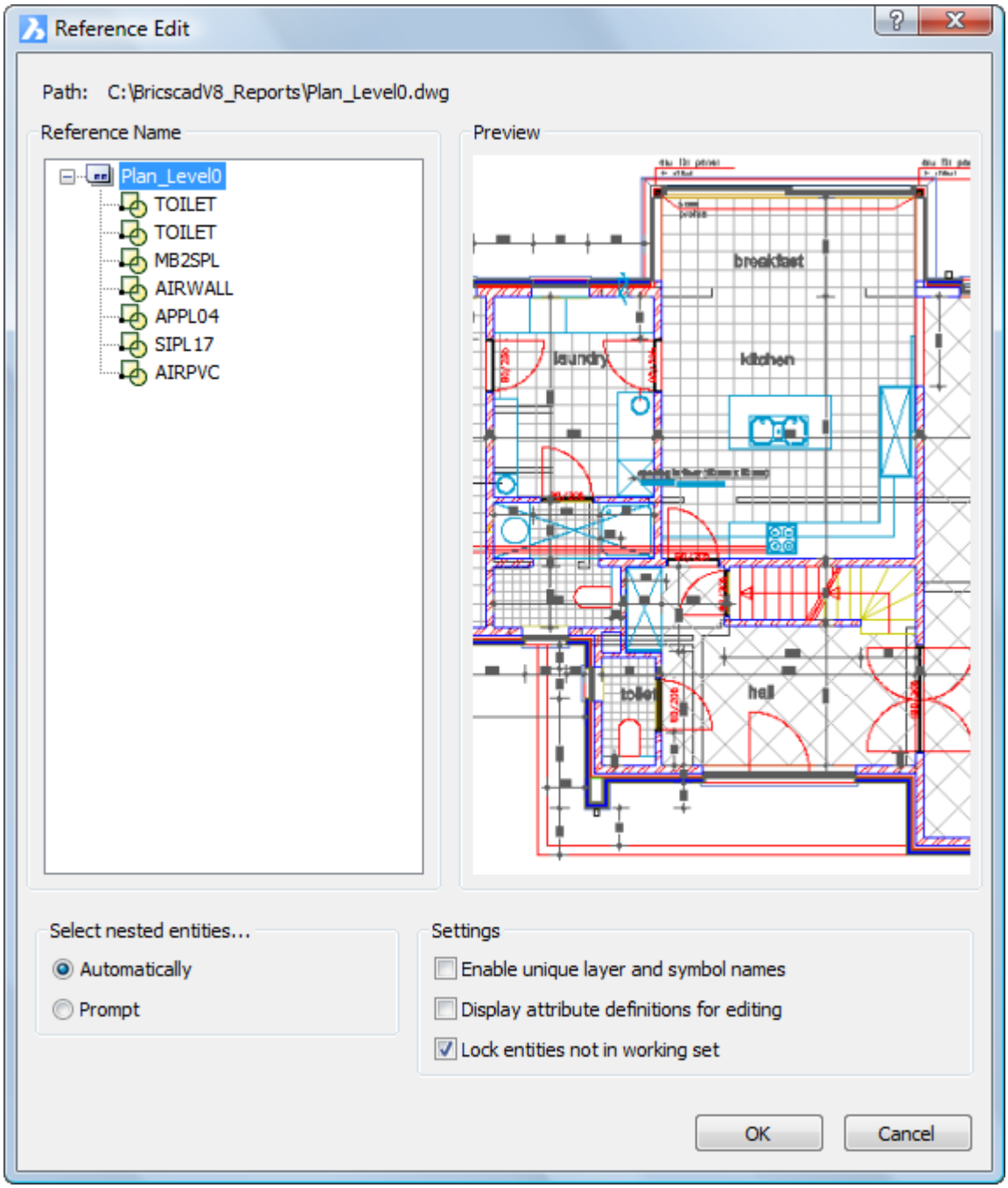
Click the *Refedit* tool button ( ) on the *Refedit* toolbar.

Type *refedit* in the command bar, then press Enter.

You are prompted: Select reference:
Click the Xref or block.

Double click the Xref or block.

The *Reference Edit* dialog opens.



The *Reference Edit* dialog helps you to visually identify the reference to edit and controls how the reference is selected.

Path: Location of the selected reference; if the reference is a block, no path is displayed.

Reference Name: The name of the currently selected reference and any references, that are nested within the selected reference.

Preview: Displays the reference as it was last saved in the drawing. Please note that the preview image is not updated when changes are saved back to the reference.

(option) If multiple nested references display in the *Reference name* tree, select a reference to modify.
Click the expand/collapse button to show/hide nested references.
You can edit only one reference a time.

Select one of the following to *Select nested entities ...*:

Automatically: All the entities and nested blocks in the selected reference are included in the reference editing session.

Prompt: After you close the *Reference Edit* dialog, you are prompted to select the enties and nested blocks in the reference that you want to edit. Only these entities will be temporarily extracted and be made available for modification within the context of the current drawing.

This temporary working set of entities can be edited and then saved back to update the reference being edited. To distinguish 'current drawing' from 'editing set' BricsCAD displays all the entities of the current drawing as faded. The fading level is controlled through the *XFADECTL* settings variable, of which the default value is set to *50*.

Check the *Settings*:

Enable unique layer and symbol names:

If selected, named objects in xrefs are altered, adding a *$#$* prefix.

If cleared, the names of layers and other named objects remain the same as in the reference drawing.

Display attribute definitions for editing: Controls whether attribute definitions in blocks are extracted and displayed during reference editing.

Lock entities not in working set: Entities in the reference being edited that are not included in the temporary working set (see step 5) cannot be manipulated, nor entities in the host drawing or other Xrefs. This behavior is similar to objects on locked layers.

Click the *OK* button to start editing the working set (see step 5).
The *Reference Edit* dialog closes.

(option) Click the *Add to Refedit* tool button ( ) to add entities to the working set.
The selected entities will be removed from the host drawing and added to the reference being edited when the working set is saved back.

(option) Click the *Remove from Refedit* tool button ( ) to remove entities from the working set.
The selected entities will be removed from the reference being edited when the working set is saved back. These entities are added to the host drawing.

Do one of the following:

Click the *Refclose and Save* button ( ) to save the changes back to the Xref source drawing or the block definition in the current drawing.

Click the *Refclose and discard* button ( ) to stop the refedit session without saving. The source drawing or block definition is left unchanged.

Working with Groups

To Create a Group

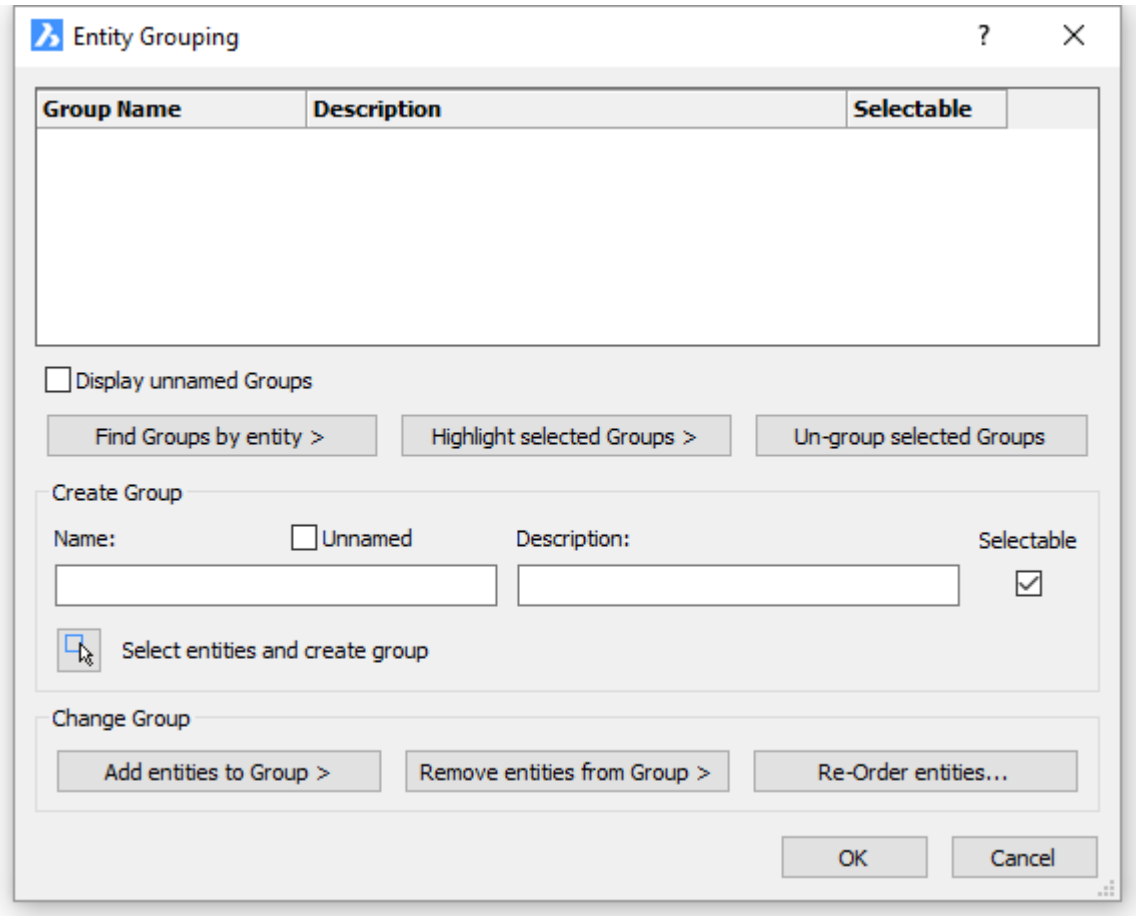
Do one of the following:

Click the *Group* tool button ( ) on the *Tools* toolbar.

Choose *Group* in the *Tools* menu.

Type *group* in the command bar, then press Enter.

The *Entity Grouping* dialog window opens:



Do one of the following:

Type a name in the *Name* field.

Check the *Unnamed* option to create an unnamed group.

(option) Type a description in the *Description* field.
(option) Check / uncheck the *Selectable* option.
Click the *Select entities and create group* button.
The *Entity Grouping* dialog temporarily closes to let you select entities.
Select the entities you want to include in the new group.
Right click to conclude the selection of entities.
The *Entity Grouping* dialog reopens.
The newly created group is added.
Click the *OK* button to close the *Entity Grouping* dialog

To Modify a Group

Do one of the following:

Click the *Group* tool button ( ) on the *Tools* toolbar.

Choose *Group* in the *Tools* menu.

Type *group* in the command bar, then press Enter.

The *Entity Grouping* dialog window opens.

Select the group you want to modify.

(option) Double click in the *Name* field to edit the group name.

(option) Double click in the *Description* field to edit the group description.

(option) Click in the *Selectable* column to toggle the *Selectable* option on/off.

(option) Click the *Add entities to group* button to add new entities to the group.
The *Entity Grouping* dialog temporarily closes to let you select the new group entities.

(option) Click the *Remove entities from group* button to remove entities from the group.
The *Entity Grouping* dialog temporarily closes to let you select the entities.

Click the *OK* button to close the *Entity Grouping* dialog

To Ungroup Entities

Do one of the following:

Click the *Group* tool button ( ) on the *Tools* toolbar.

Choose *Group* in the *Tools* menu.

Type *group* in the command bar, then press Enter.

The *Entity Grouping* dialog window opens.

Select the group or groups you want to delete.

Click the Un-group selected groups button.
The selected groups are removed.

NOTE When you ungroup entities, the entities remain in the drawing but the group is deleted from the drawing.

To Change the Order of Entities

Do one of the following:

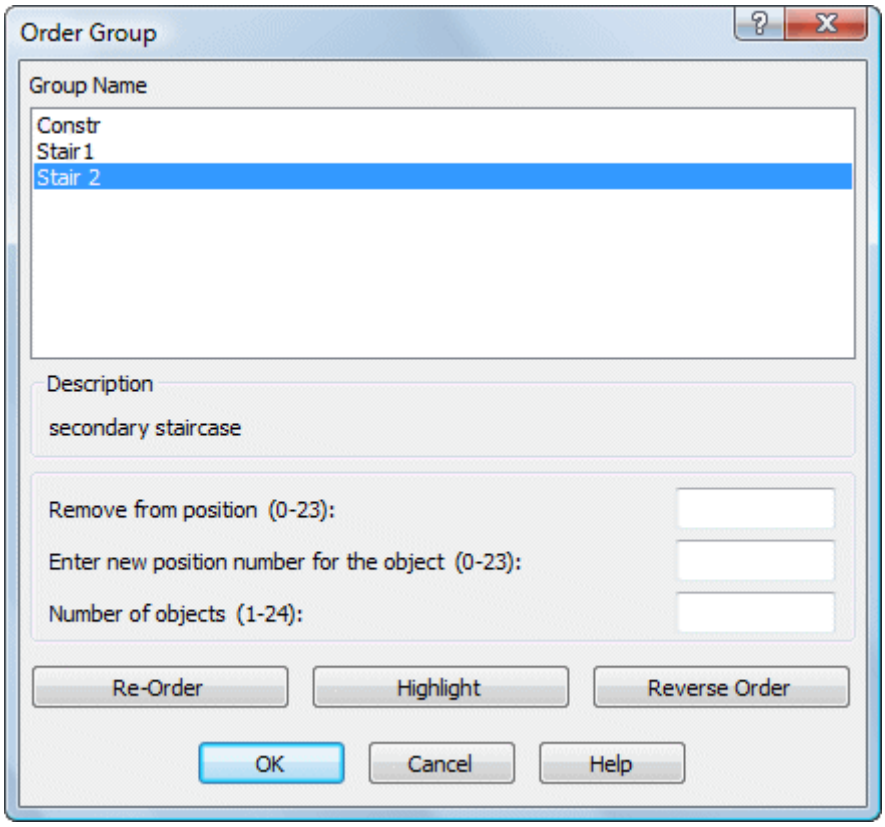
Click the *Group* tool button ( ) on the *Tools* toolbar.

Choose *Group* in the *Tools* menu.

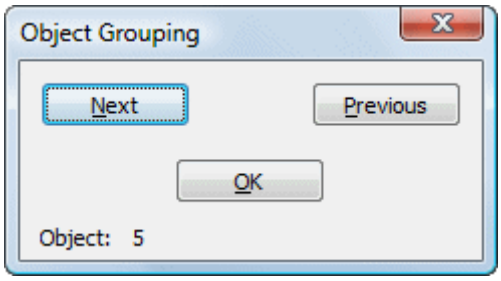
Type *group* in the command bar, then press Enter.

The *Entity Grouping* dialog window opens.

Click the *Re-Order entities* button.
The *Order Group* dialog opens:



(option) Click the *Highlight* button to see the current position of each entity.
The *Object Grouping box* opens.



(option) Click the *Reverse Order* button to reverse the order of all entities in the group.

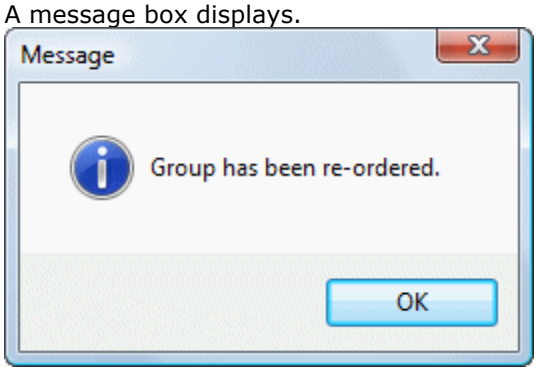
(option) To change the order of a single entity do the following:

In the *Remove from position* field type the current position of the entity you want to move.

In the *Enter new position number for the object* field type the new position.

(option) Type the number of objects you want to reorder in the *Number of objects* field.

Click the Re-Order button.
A message box displays.



Working with Underlays

Do one of the following:

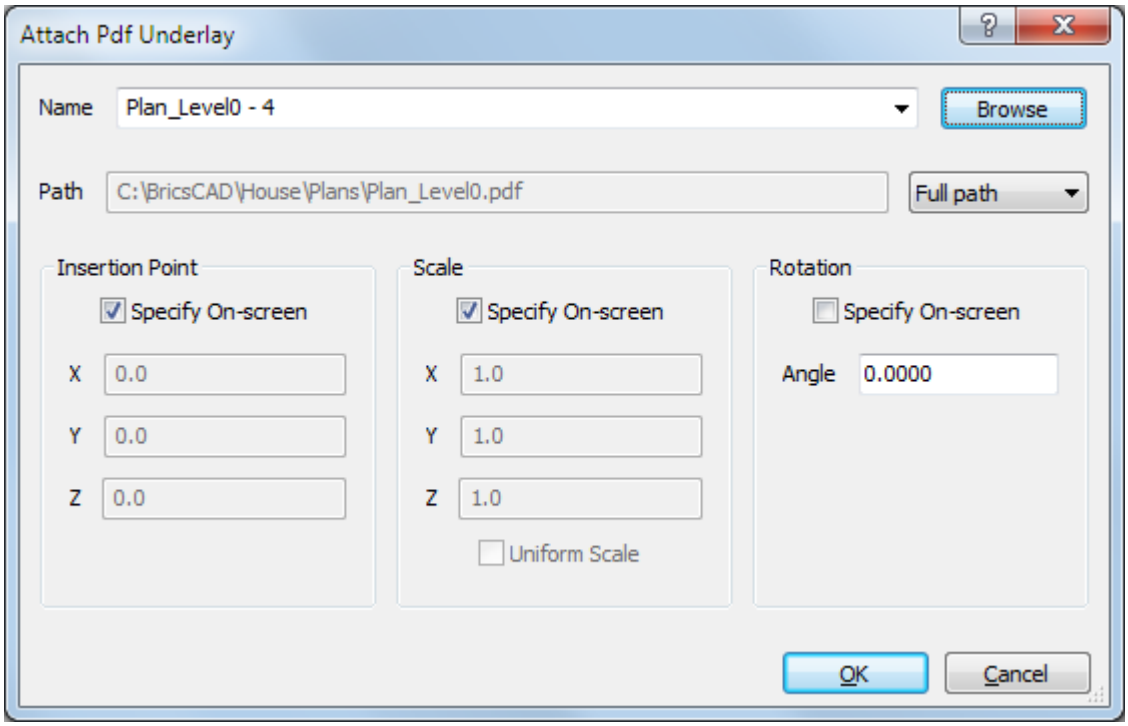
Click the *Attach pdf* tool button ( ) on the *Insert* toolbar.

Click the *New* tool button ( ) on the *Details* toolbar of the Drawing Explorer - PDF Underlays dialog.

Type *PDFattach* in the command bar, then press Enter.

The *Select PDF Underlay File* dialog opens.

Select the file, then double click the file or click the *Open* button on the *Select PDF Underlay File* dialog.
The *Attach PDF Underlay* dialog opens.



In case of a multi-page PDF, click the down arrow in the *Name* field, then select the page of the PDF you want to load.

Click the *Path* option button, then select either:

Full path: The full path will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" displays at the insertion point of the PDF overlay.

Relative path: The relative path, with respect to the folder of the parent drawing, will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" displays at the insertion point of the PDF overlay.

No path: Only the folder of the parent drawing is searched when the drawing is loaded. If the overlay file is not found there, "Missing or invalid reference" displays at the insertion point of the PDF overlay.

Choose whether you want to specify the *Insertion Point*, *Scale* and the *Rotation* angle on-screen or not.

Click the *OK* button.
The *Attach PDF Underlay* dialog closes.

Depending on the insertion options chosen in the previous step you are prompted to specify the *Insertion Point*, *Scale* and/or *Rotation* angle.

Inserting a PDF Underlay

If you want to insert a second instance of a PDF that is already attached or another page of such underlay do the following:

Choose *Drawing Explorer > PDF Underlays...* in the *Tools* menu.
The *Drawing Explorer - PDF Underlays* dialog opens.

Select the PDF underlay.

Do one of the following:

Right click and choose Insert in the context menu.

Choose insert in the Edit menu on the *Drawing Explorer - PDF Underlays* dialog.

The *Drawing Explorer - PDF Underlays* dialog closes.

Continue with steps 3 through 7 of the previous procedure.

Controlling the Display of a PDF Underlay

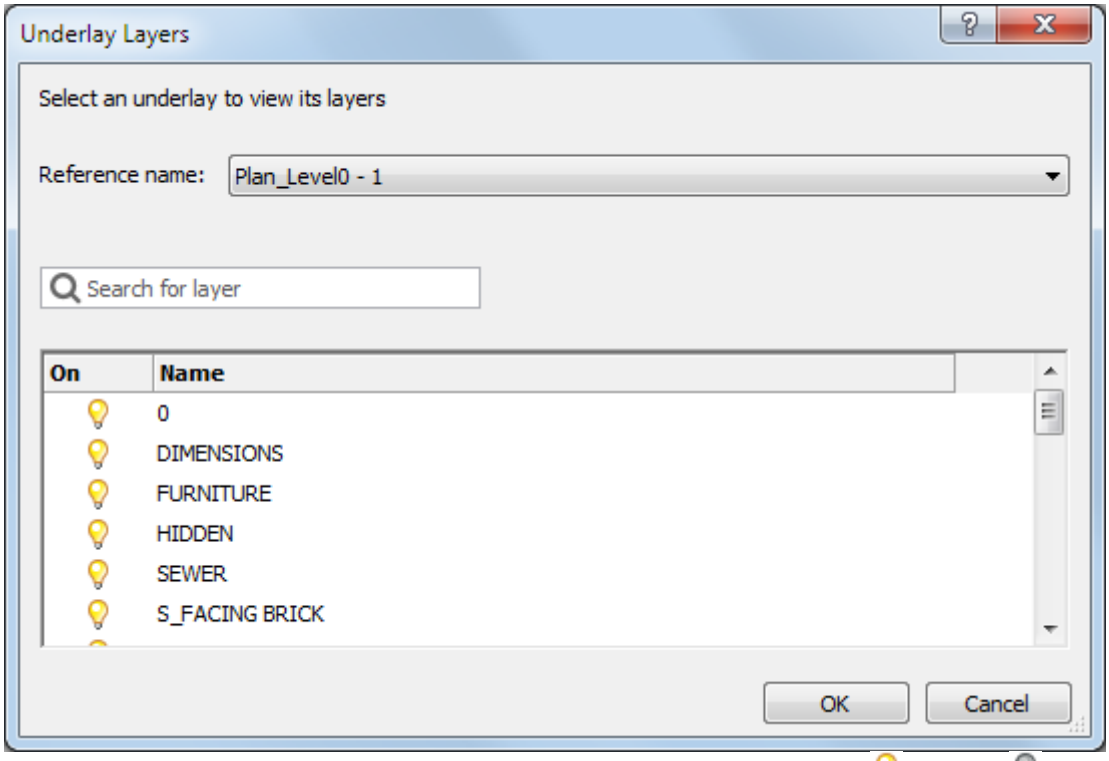
Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
The properties of the underlay display in the Properties Bar.

Under *Misc*, click *Show underlay* in the Properties Bar, then select *Yes* or *No*.

Setting the Layer Display in a PDF Underlay

Type *PDFlayers* in the command bar, then press Enter.
You are prompted: Select PDF underlay.

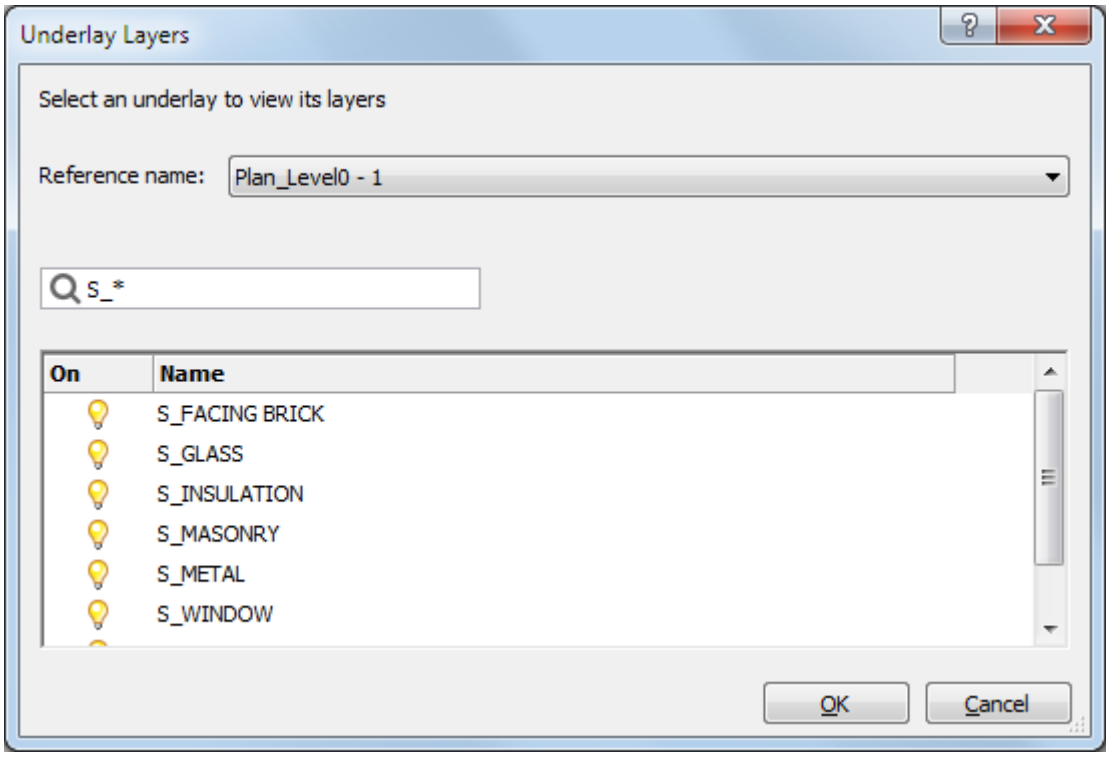
Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
The *Underlay Layers* dialog opens.



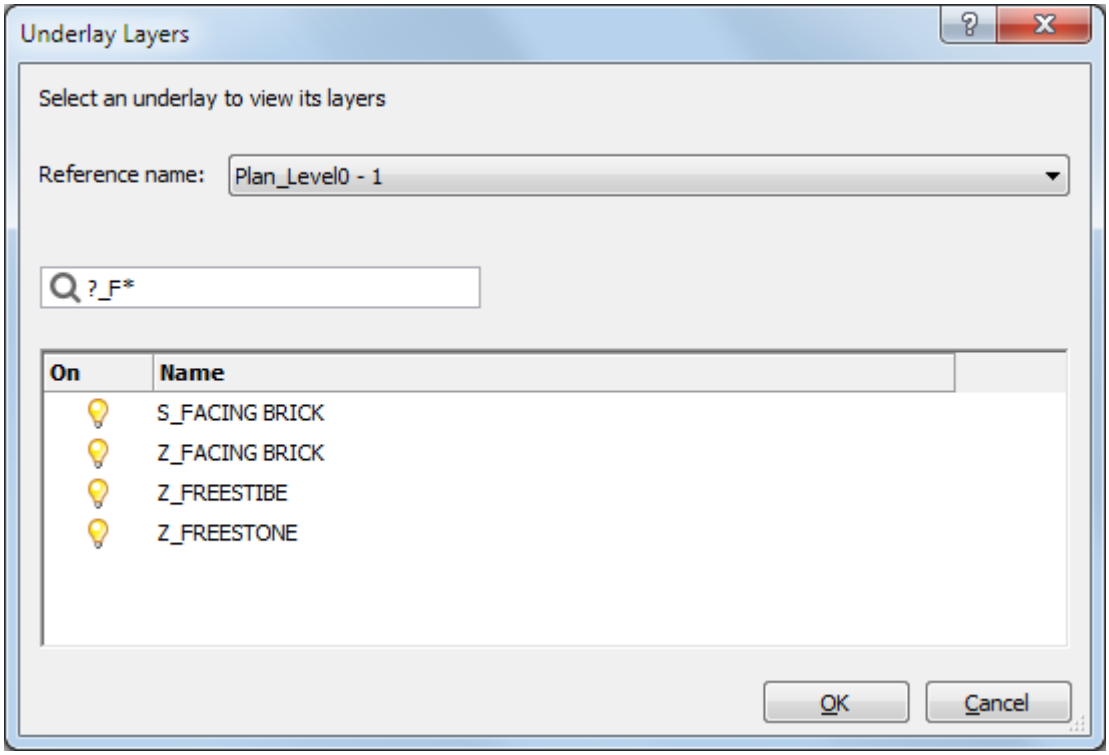
The icon in the *On* column indicates the current state of the layers On ( ) or Off ( ).

Click the icon in the *On* column to toggle the display of a layer.

(option) Click the *Search for layer* field, then type a layer name to search for a layer.
Use wildcard characters (? or *) to limit the number of layers in the list:



Layernames starting with 'S_' only.



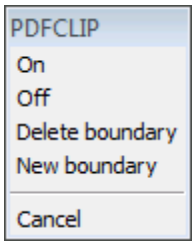
Layernames of which the second and third character is '_F' only.

Do one of the following:

Clipping a PDF Underlay

Type *pdfclip* in the command bar, then press Enter.
You are prompted: Select PDF underlay:

Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
You are prompted: Enter PDF clipping option [ON/OFF/Delete/New] <New>:
A prompt menu displays:



Do one of the following:

Choose *New boundary* in the prompt menu.

Press Enter to accept the *New* default command option.

You are prompted: Enter PDF clipping type [Polygonal/Rectangular] <Rectangular>:

Do one of the following:

Press Enter to accept the Rectangular default command option to define a rectangular clipping boundary.

Type P, then press Enter to define a polygonal clipping boundary.

Define the clipping boundary.
The defining points must lie inside the frame of the PDF underlay. If you click outside the PDF underlay the point is placed on the PDF underlay frame.

To Toggle the Display of the Clipped Part

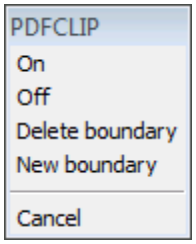
Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
The properties of the underlay display in the Properties Bar.

Under *Misc*, click *Show clipped* in the Properties Bar, then select *Yes* or *No*.

Deleting a Clipping Boundary

Type *pdfclip* in the command bar, then press Enter.
You are prompted: Select PDF underlay:

Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
You are prompted: Enter PDF clipping option [ON/OFF/Delete/New] <New>:
A prompt menu displays:



Do one of the following:

Choose *Delete boundary* in the prompt menu.

Type D in the command bar, then press Enter.

The clipping boundary is deleted and the complete PDF underlay displays.

To adjust the display settings of a PDF underlay

The PDFADJUST command allows to adjust the *Fade*, *Contrast* and *Monochrome* settings of a PDF-underlay in the command bar.

Select the PDF underlay, then adjust the display settings under *Underlay Adjust* the Properties Bar:

Underlay Adjust	
Contrast	0
Fade	0
Monochrome	No
Adjust colors for	Yes

Managing Images

Open the Images Explorer

To open the *Images Explorer* do one of the following:

Choose *Drawing Explorer > Images* in the *Tools* menu.

Click the *Image Management* tool button ( ) in the *Images* toolbar.

Type *image* in the command bar, then press Enter.

Select *Images* in the *Open Drawings* sub-window of the Drawing Explorer.

In the *Images Explorer* you can:

See a preview of the attached images

Place images ( ) or ( )

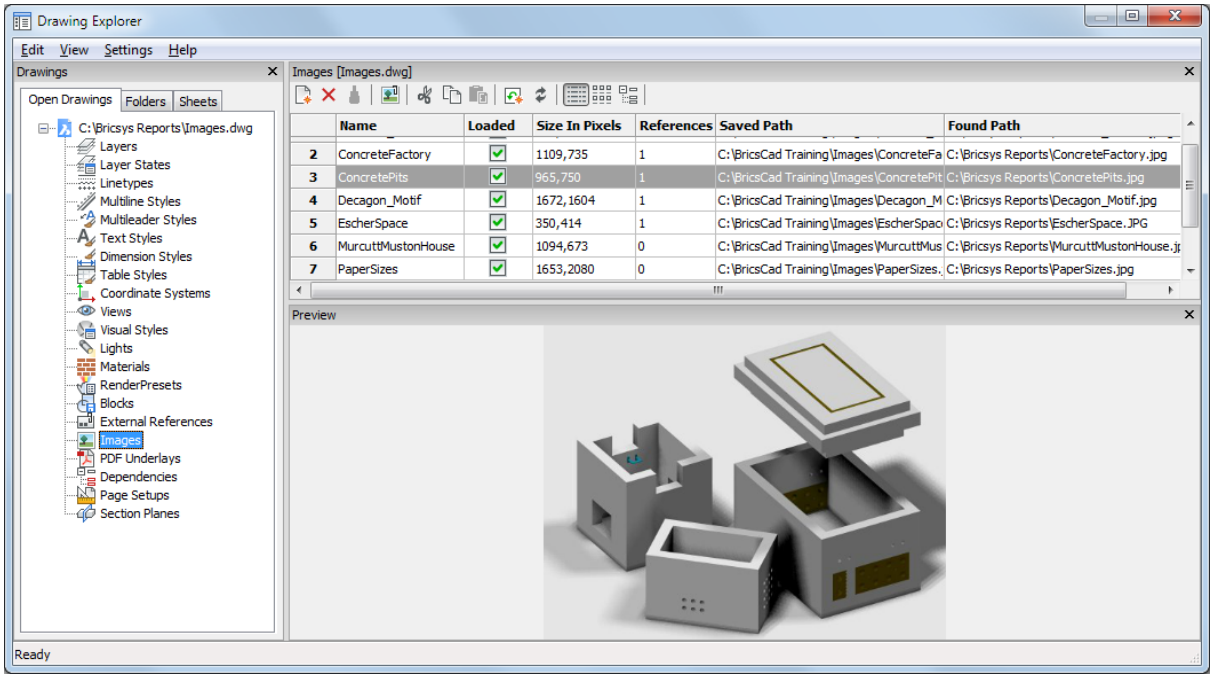
Load / Unload inserted images

Detach images ( )

Open the Detail View of the Images Explorer

Click the *Detail View* button ( ) in the *Drawing Explorer* toolbar. The *Detail View* button is now pressed, indicating the image details are displayed.

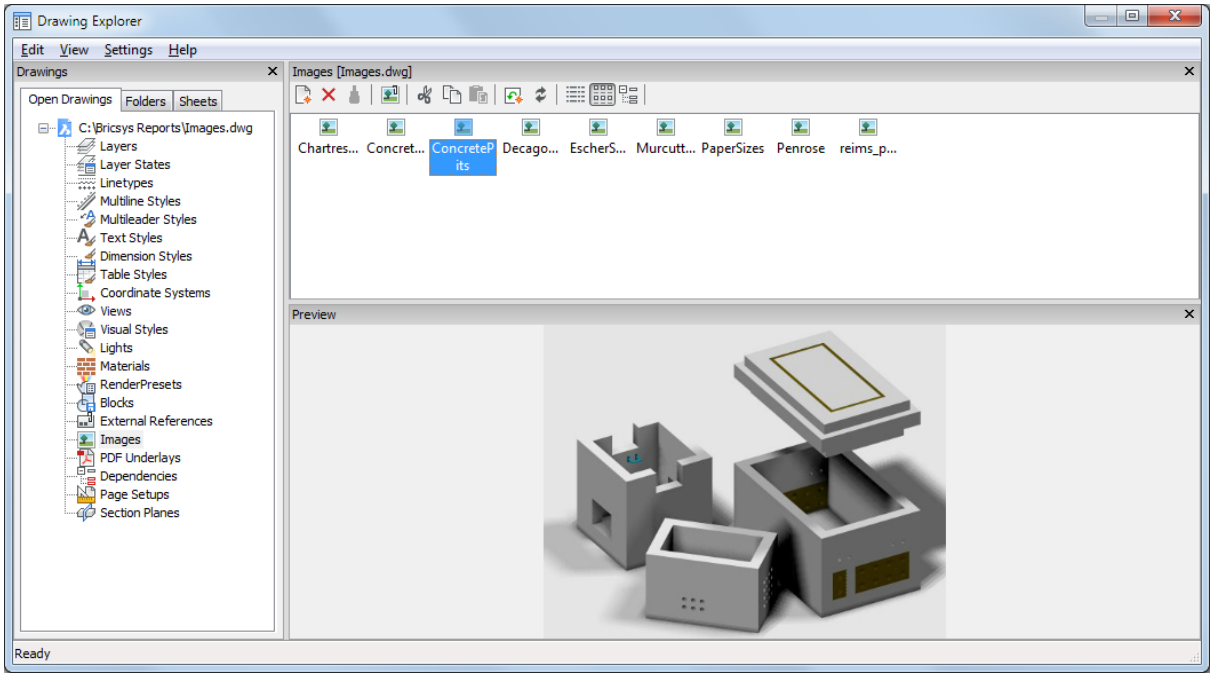
The selected image displays in the *Preview* sub-window of the *Drawing Explorer*.



Detail View of the images in the current drawing

Open the Icon View of the Images Explorer

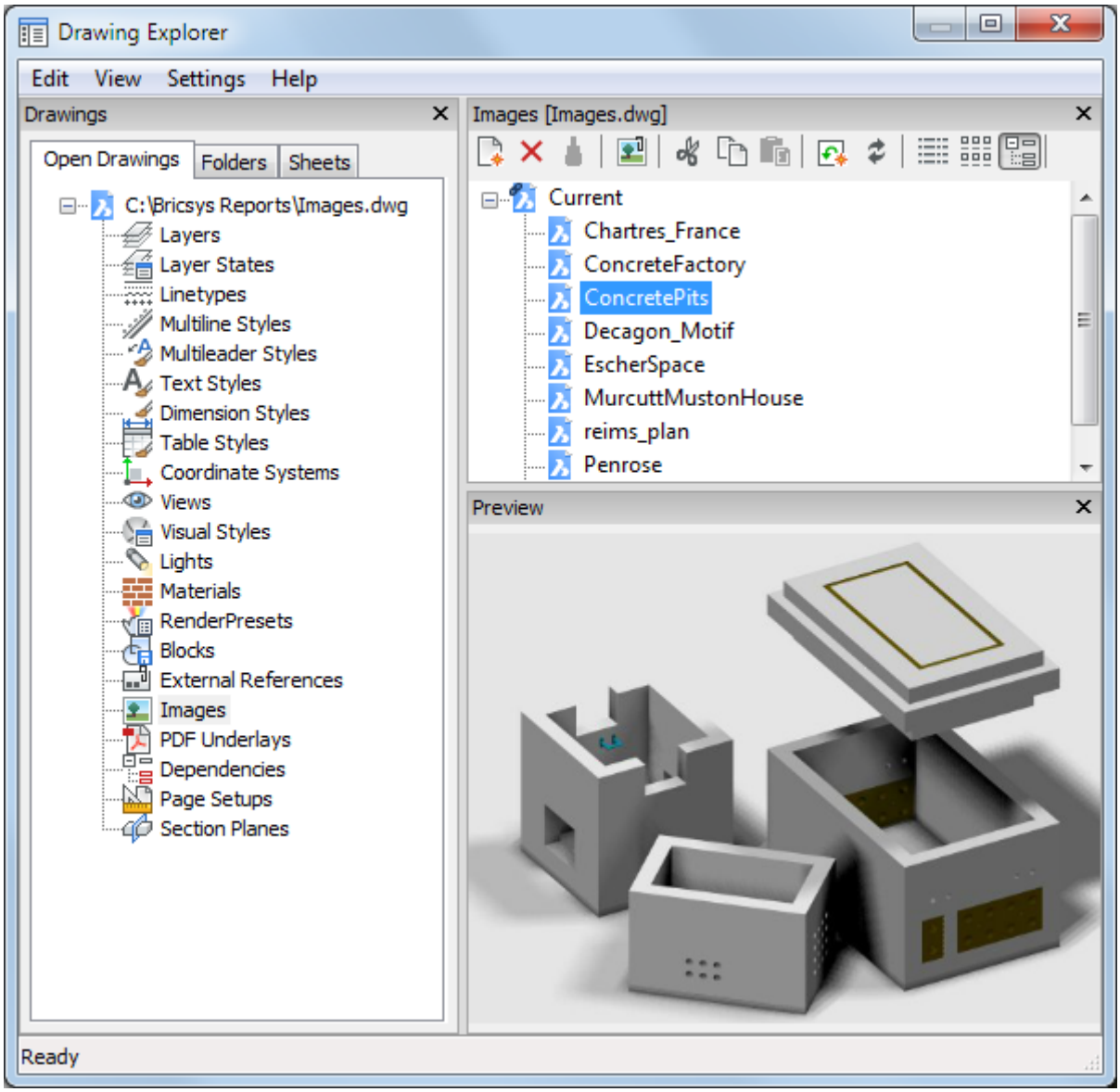
Click the *Icon View* button ( ) in the *Drawing Explorer* toolbar. The *Icon View* button is now pressed, indicating the image icons are displayed.
The selected image displays in the *Preview* sub-window of the *Drawing Explorer*.



Icon View of the images in the current drawing

Open the Tree View of the Images Explorer

Click the *Tree View* button ( ) in the *Drawing Explorer* toolbar. The *Tree View* button is now pressed, indicating the image tree is displayed.
The selected image displays in the *Preview* sub-window of the *Drawing Explorer*.



Tree View of the images in the current drawing

Placing Images in a Drawing

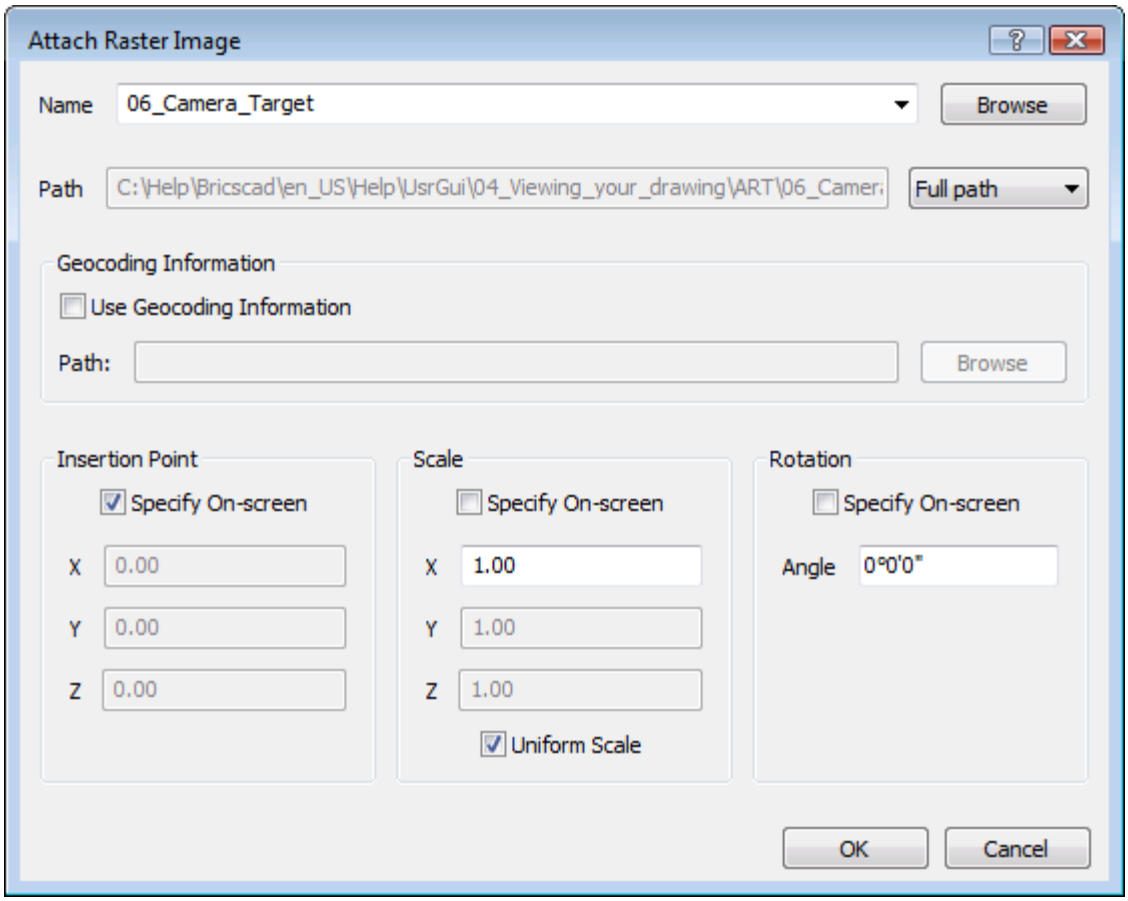
Do one of the following:

Launch the *Image Explorer*, then click the *New* tool button ( ) in the *Drawing Explorer - Images* toolbar.

Launch the *Image Explorer*, select a raster image file in the list, then click the *Insert* tool button ( ) in the *Drawing Explorer - Images* toolbar.

Click the *Attach Raster Image* tool button ( ) in the *Insert* toolbar.

The *Attach Raster Image* dialog opens.



Do one of the following:

Accept the currently selected image file.

Click the down arrow button in the *Name* field to select a different raster image file in the list.
Raster images which are already attached are listed.

Click the *Browse* button to select a new raster image file in the *Select Raster Image* dialog.

(option) Check the *Use Geocoding Information* option, then click the *Browse* button to open an appropriate world file to be used with the selected raster image.
If the *Use Geocoding Information* option is checked, the *Insertion Point*, *Scale* and *Rotation* settings fields are not available.

(option) Do the following:

Specify an *insertion point* or select the *Specify on-screen* check box.

Specify *scale* or select the *Specify on-screen* check box.

Specify *rotation* angle or select the *Specify on-screen* check box.

Click the *OK* button.

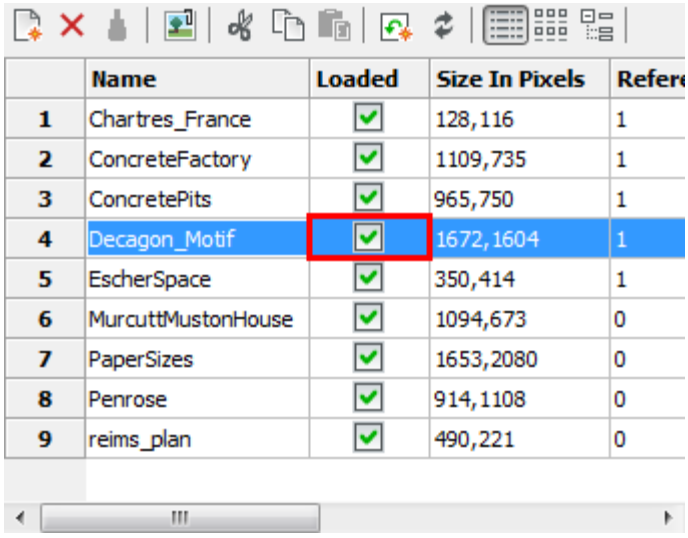
If none of the *Specify on-screen* options is selected, the image is inserted at the desired insertion point, scale and rotation angle;
else
You are prompted to specify the insertion point, scale and/or rotation angle on-screen.

Load/Unload Images

When an image is attached and then inserted in the drawing, you can unload the image to temporarily remove it. Unloaded images are still inserted in the drawing, but they no longer display. If the *Imageframe* setting is ON, the frame around the image still displays though.

Launch the *Image Explorer*.

For the image you want to load or unload, click the check box in the *Loaded* column.



	Name	Loaded	Size In Pixels	Refere
1	Chartres_France	✓	128,116	1
2	ConcreteFactory	✓	1109,735	1
3	ConcretePits	✓	965,750	1
4	Decagon_Motif	✓	1672,1604	1
5	EscherSpace	✓	350,414	1
6	MurcuttMustonHouse	✓	1094,673	0
7	PaperSizes	✓	1653,2080	0
8	Penrose	✓	914,1108	0
9	reims_plan	✓	490,221	0

If the *Regen On/Off* button ( ) is not pressed, click the *Regen* tool ( ) in the *Images Explorer* toolbar.

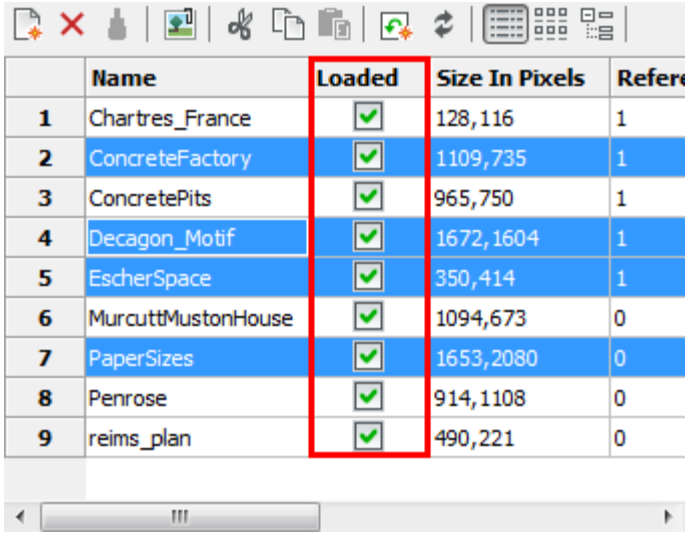
NOTE It is not necessary to select the image first in step 2.

Load / Unload multiple images

Launch the *Image Explorer*.

Select the images you want to load or unload.

For one of the selected images, click the check box in the *Loaded* column.



	Name	Loaded	Size In Pixels	Refere
1	Chartres_France	✓	128,116	1
2	ConcreteFactory	✓	1109,735	1
3	ConcretePits	✓	965,750	1
4	Decagon_Motif	✓	1672,1604	1
5	EscherSpace	✓	350,414	1
6	MurcuttMustonHouse	✓	1094,673	0
7	PaperSizes	✓	1653,2080	0
8	Penrose	✓	914,1108	0
9	reims_plan	✓	490,221	0

All selected images will be loaded or unloaded simultaneously.

If the *Regen On/Off* button ( ) is not pressed, click the *Regen* tool ( ) in the *Images Explorer* toolbar.

NOTE Press and hold the Ctrl key to select multiple images.

Removing images

Since an image is first attached, then inserted in the drawing, you can delete an image, without detaching it.

Detach Images

Launch the *Image Explorer*.

Select the image(s) that you want to detach.

Do one of the following

Click the *Delete* tool button (✕) in the *Images Explorer* toolbar.

Right click and select *Delete* in the context menu.

NOTE Press and hold the Ctrl key to select multiple images.

Image Transparency

Do one of the following:

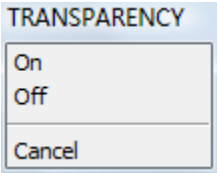
Click the *Image Transparency* tool button ( ) on the *Images* toolbar.

Choose *Image Transparency* in the *Images* menu.

Type *transparency* in the command bar, then press Enter.

You are prompted: Select images.

Select the images you want to set the *Transparency* property of.
You are prompted: Enter transparency mode [ON/OFF]<current mode>:
A prompt menu displays.



Do one of the following:

Press Enter to accept the current mode.

Choose *On* in the prompt menu or type *ON* in the command bar, then press Enter to switch the *Transparency* property of the selected images on.

Choose *Off* in the prompt menu or type *OFF* in the command bar, then press Enter to switch the *Transparency* property of the selected images off.

Transparency OFF (left) and Transparency ON (right)

Image Frame Setting

Click the *Image Frame* tool ( ) in the *Images* toolbar.
You are prompted: Enter image frame setting <1>:

Do one of the following:

Type 1 + Enter to display the image frame.

Type 0 + Enter to hide the image frame.

To Clip an Image

Make sure image frames display.

Do one of the following:

Click the *Clip Image* tool button ( ) on the *Image* toolbar.

Type *imageclip* in the command bar, then press Enter.

You are prompted: Select image:

Click the image frame.
You are prompted: Enter image clipping option [ON/OFF/Delete/New] <New>:

Press Enter to accept the *New Boundary* default option.
You are prompted: Enter image clipping type [Polygonal/Rectangular] <Rectangular>:

Press Enter to accept the *Rectangular* default option.
You are prompted: Specify first corner

Click a point to define the first corner of the clipping rectangle.
The clipping rectangle displays dynamically.
You are prompted: Specify opposite corner:

Click a point to define the clipping rectangle.
The image is clipped.

To Toggle the Clipping Boundary of an Image

Make sure image frames display.

Click the image frame.
The properties of the image display in the Properties Bar.

Under *Misc / Show clipped* choose *Yes* or *No*.

Printing and Plotting

The *Plot* command plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines.

The *Print* command prints drawings to plotters and files through a dialog box.

The *Qprint* command prints the drawing with the default plot configuration, without displaying the Print dialog box (short for "quick print").

The *PageSetup* command creates and edits page setups for plotting drawings in the Drawing Explorer.

TUTORIALS

Printing a layout

Creating page setups and layouts

A standard print procedure consists of the following steps:

Choose either a Layout tab or the Model tab.

Open the Print dialog.

Select a Page Setup name. (1)

The following steps are optional and only necessary if you want to use overrides on the selected page setup definition.

Choose a Printer or Plotter Configuration. (2)

Choose a Plot style table. (3)

Choose a Paper Size. (4)
The image reflects the position and orientation of the drawing on the selected paper size.

Choose the Paper Units. (5)

Choose the Drawing Orientation.(6)

Define the Plot offset. (7)

Specify the Area to plot. (8)

Define the Plot Scale. (9)

Check the Plot Options. (10)

When printing model space, choose a Shade Plot mode. (11)
This option is disabled when printing a paper space layout.

Check/uncheck the *Print to file* option (12)

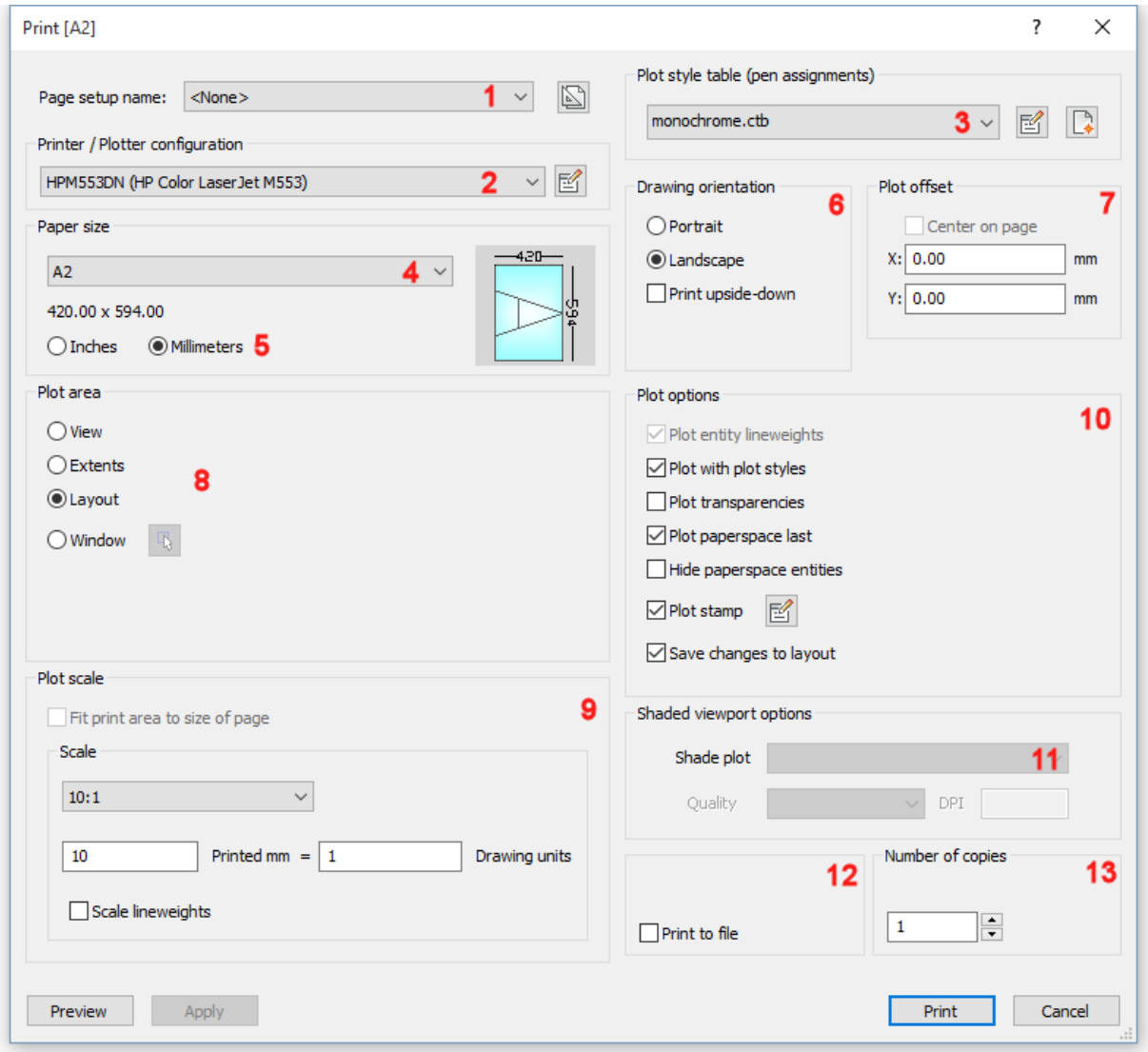
Specify the Number of copies. (13)

Click the *Preview* button to see a preview of the plot output.

When in a layout, click the *Apply* button to update the layout.

Click the *Print* button to start printing.

NOTE	All print settings and options are saved in the *Model* and each of the *Layouts*.



To Open the Print Dialog

Do one of the following:

Choose *Print...* in the File menu.

Type *print* in the command bar, then press Enter.

Press *Ctrl + P*

Select the *Model* tab or a *Layout* tab, then right click and choose *Print* in the context menu.

NOTE	If you click the *Print* ( ) button in the *Standard* toolbar, the *Qprint* command is launched, which will start printing the currently selected view tab using previously saved print settings.

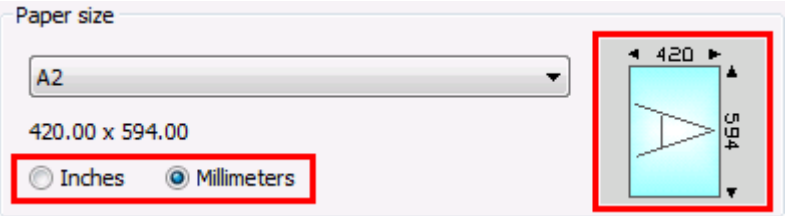
To Choose the Paper Size

Press the down arrow, then select the paper size in the *Paper Size* list.

The available sizes depend on the selected printer.

To Choose the Paper Units

Select either *Inches* or *Millimeters*.



The paper size image indicates the size, position and orientation of the plot area on the selected paper size.

NOTE	The *Scale* and *Plot offset* setting units are changed according to the chosen *Paper units*.

To Choose the Paper Orientation

Select either *Portrait* or *Landscape*.

- *Portrait*: The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- *Landscape*: The drawing or layout x-axis is aligned with the longest edge of the selected paper size.

The paper size image changes accordingly.

When in a layout, click the *Apply* button to update the display.

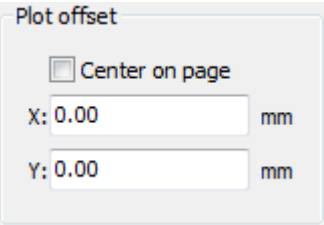
To Define the Plot Offset

Select either *Center on page*

or

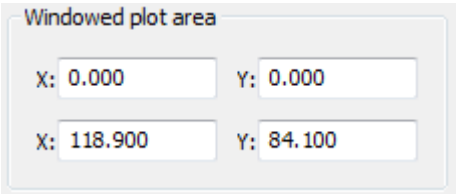
type the plot offset in the *X:* and *Y:* fields.

The offset is calculated from the bottom left corner of the paper.



When in a layout, click the *Apply* button to update the display.

To Specify the Area to Plot

Option	Result
View	Prints the portion of the drawing that displays in the current viewport or a saved view.
Extents	Prints all entities in the drawing
Limits	Prints to the limits defined in the drawing This option is only available when printing in Model space.
Layout	Prints the current layout. This option is only available when printing in Paper space.
Window	Prints the portion of the drawing contained in a user defined window, maintaining the aspect ratio of the windowed area to the drawing. Click the *Select print area* button ( ), then specify the two corners of the print area in the drawing. or Type the the X- and Y-coordinates of the corners of the print area in the *X:* and *Y:* fields. 

To Define the Plot Scale

NOTE The SCALELISTEDIT command allows to edit the list of available scales in the *Print* dialog and the *Standard Scale* property of a paper space viewport.

... when printing in paper space:

If you print in paper space, the final scale of the plot output is the product of the viewport scale and the plot scale.

In order to keep control over the scale of the plot output:

Specify the width and height of the viewports in drawing units.

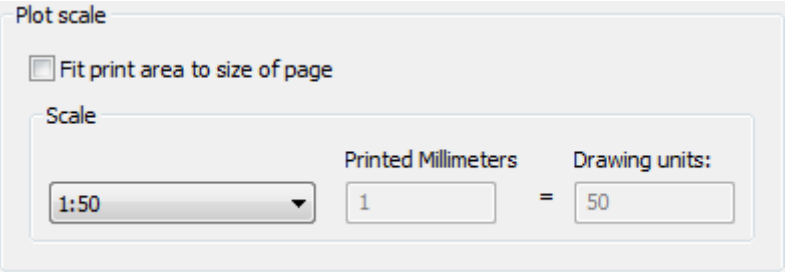
Set the scale of each viewport to the scale you want in the plot output.

The *plot scale* reflects the relationship between paper units and drawing units. (see table below)

Paper units	Drawing units	Plot scale	Printed Millimeters or Inches	Drawing Units
mm	mm	1:1	1	1
mm	cm	10:1	10	1
mm	m	1000:1	1000	1
inches	inches	1:1	1	1
inches	feet	12:1	12	1

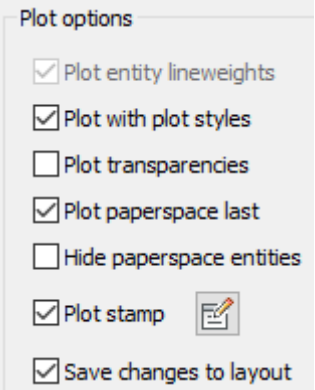
... when printing in model space:

If you print in model space, the scale of the plot output equals the plot scale, on condition your drawing units equal the paper units. Otherwise you need to multiply the plot scale with respect to the relationship between paper units and drawing units. E.g. if you want to plot at a 1/50 scale in a drawing of which 1 DU = 1 cm, the *plot scale* = 1/5, if 1 DU = 1 m the *plot scale* = 20/1 or 1/0.05



To Set the Plot Options

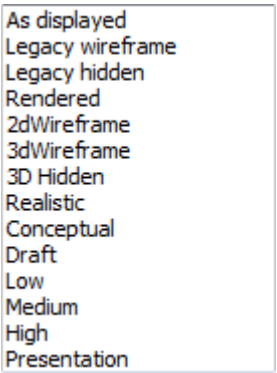
Click to toggle the *plot options* On / Off.



Option	Result
Plot object lineweights	If *Plot with plot styles* is off, object lineweights are plotted.
Plot with plot styles	Settings of the assigned STB or CTB file are used.
Plot transparencies	Plots entities using the value of the Transparency property.
Plot paperspace last	Entities in Model space are plotted first, paperspace entities last.
Hide paperspace entities	Removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.
Plot stamp	Adds a header and / or footer. Click the *Edit plot stamp* button ( ) to define the plot stamp.
Save changes to layout	All changes made in the *Print* dialog are saved in the layout. This option toggles the *Save changes to layout* variable

To Set the Shade Plot Mode

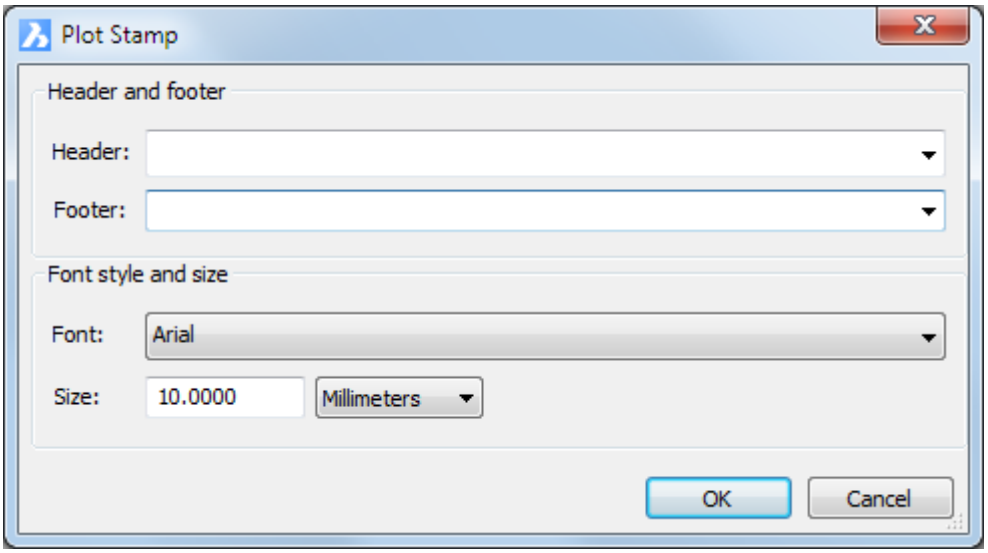
Under *Shade viewport options*, click the list button, then select an option:



NOTE This option applies to model space printing only.
The *Shade Plot* mode of a paper space viewport is defined in the Shade Plot property of the viewport.

To Define the Plot Stamp

Click the *Edit plot stamp* button ( ).
The *Plot stamp* dialog opens.



In the *Header* field, do one of the following:

Type the header text.
Click the down arrow button, then choose a variable in the drop down list.
Type one or more variable names.
Leave the field empty.

In the *Footer* field, do one of the following:

Type the footer text.
Click the down arrow button, then choose a variable in the drop down list.
Type one or more variable names.
Leave the field empty.

Select a font in the *Font* list button.
Enter a value in the *Font Size* field
Set the unit type to *Millimeters* or *Inches*.
Click the *OK* button to confirm.

The plot stamp settings are saved to a series of user preferences in the current user profile, which can be edited in the Settings dialog under *Program Options / Plot and Publish / Plot Stamp*:

Plot stamp	
Header	
Footer	
Font Style	Arial
Font Size	10
Units	[1] Milimeters

Variables

Name	Description
$Name	Adds the name of the current user, as definied in the *User Name* field of the License Information.
$Company	Adds company name of the current user, as definied in the *Company* field of the License Information.
$Date	Adds the current date.
$DateTime	Adds the current date and time.
$DwgName	Adds the file name and path of the drawing.

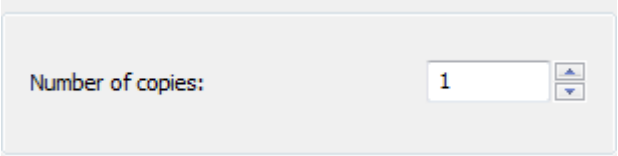
NOTE	The header and footer text consists of three parts (left, center and right), separated by commas. By default the header and footer text is centered. The header and footer text can be outlined left center and/or right by separating the text with commas. e.g. *$Name,$Company,$Date* places your name left, company name centered and the current date at the right. Type a space before one of the commas to leave one of the parts empty, thus outlining the header or footer left or right. *$Company, ,* ($Company,<space>,<space>) outlines the company name left. , ,$Company (<space>,<space>,$Company) outlines the comapny name right.

To Specify the Number of Copies

Do one of the following:

Type the desired number in the *Number of copies* field.

Click the up-arrow and the down-arrow buttons to specify the number of copies.



To See a Preview of the Plot Output

Do one of the following:+

Click the *Preview* button on the *Print* or *Page Setup* dialogs.

Choose *Print Preview...* in the *File* menu.

Type *preview* in the command bar, then press Enter.

(option) Use the mouse scroll wheel to zoom. Double click the middle mouse button to zoom extents.

To close the *Print Preview* window, do one of the following:

Click the *Close* button to return to the drawing window.

Click the *Print Settings...* button to open the Print dialog.

Click the *Print* button to start printing.

Plotter Configurations

TUTORIALS

Plotter configuration

Configurations for Windows system printers are stored in plotter configuration files (.PC3 files). Unlike in AutoCAD®, PC3 plotter configuration files in BricsCAD™ apply to Windows system printers only.

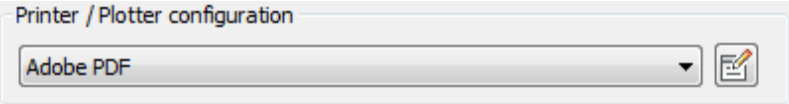
In a plotter configuration file, you override one or more settings of the system printers installed on your computer. You can configure BricsCAD for many plotting/printing devices and keep multiple configurations for each single device.

Each plotter configuration file contains information such as:

- The device driver and model
- The output port to which the device is connected
- Various device-specific settings (not supported on the Linux platform).

PC3 files are saved in the *Plotconfig* subfolder of your Roamable root folder.

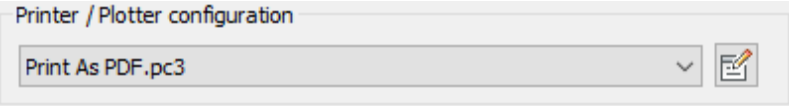
After creating a PC3 file, it is available in the list of plotter configuration names in the *Printer / Plotter configuration* list on the *Print* or *Page Setup* dialog boxes.



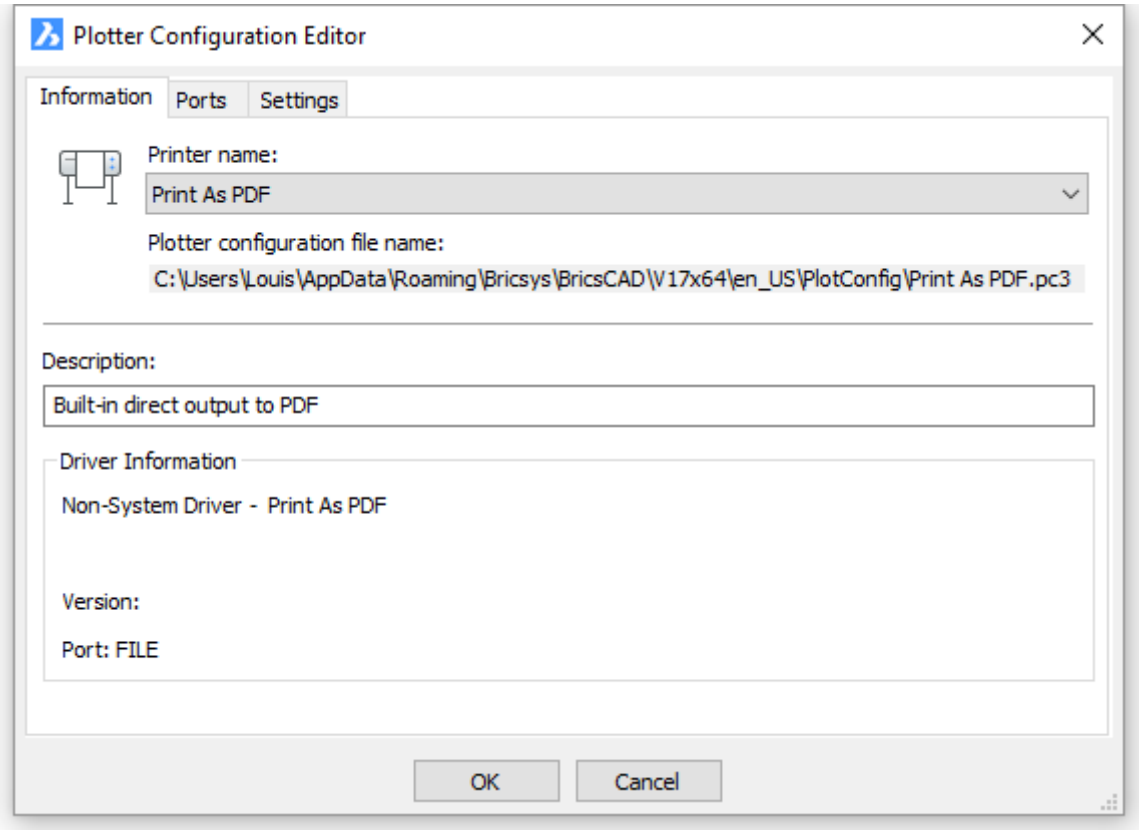
To Configure the Built-in PDF Printer

Open the Print dialog.

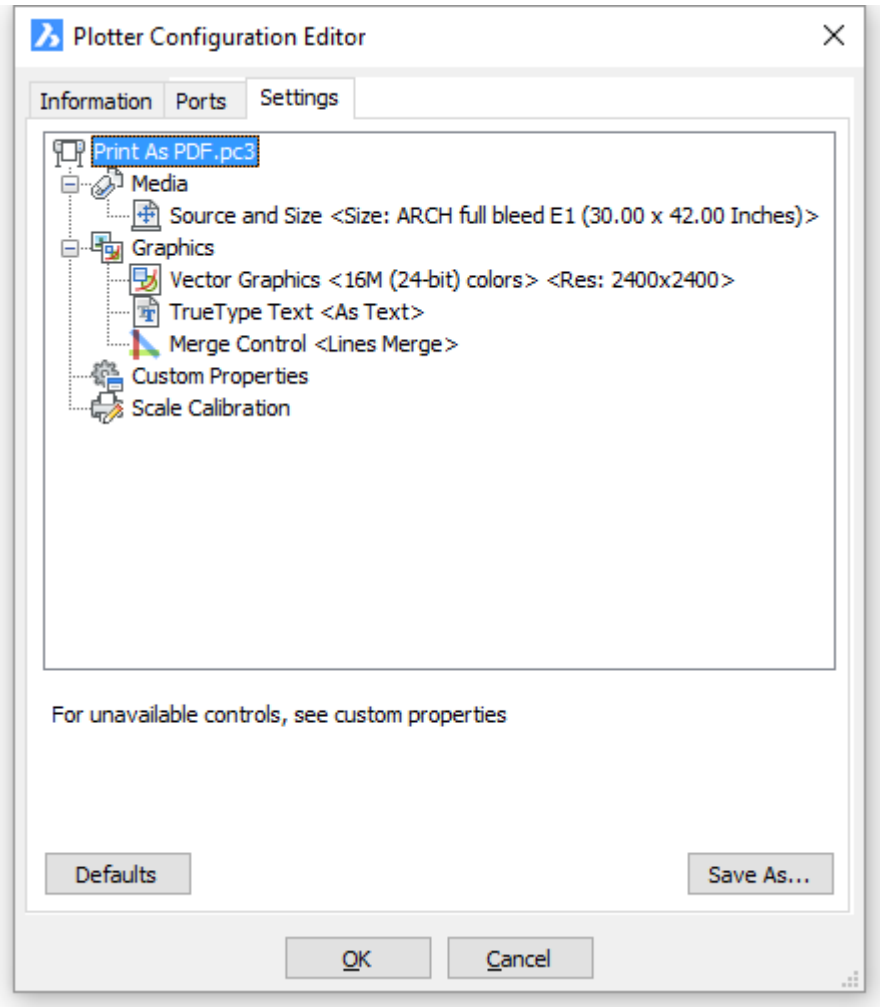
Click the *Printer/Plotter configuration* list button and choose *Print As PDF.pc3* in the list.



Click the *Edit Plotter Configuration* button ( ).
The *Plotter Configuration Editor* dialog displays withe the *Print As PDF.pc3* configuration file loaded.
The default path for plotter configuration files (*.pc3) is:
C:\Users\<User_Name>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\PlotConfig


Plotter Configuration Editor
Information
Ports
Settings
Printer name:
Print As PDF
Plotter configuration file name:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V17x64\en_US\PlotConfig\Print As PDF.pc3
Description:
Built-in direct output to PDF
Driver Information
Non-System Driver - Print As PDF
Version:
Port: FILE
OK
Cancel

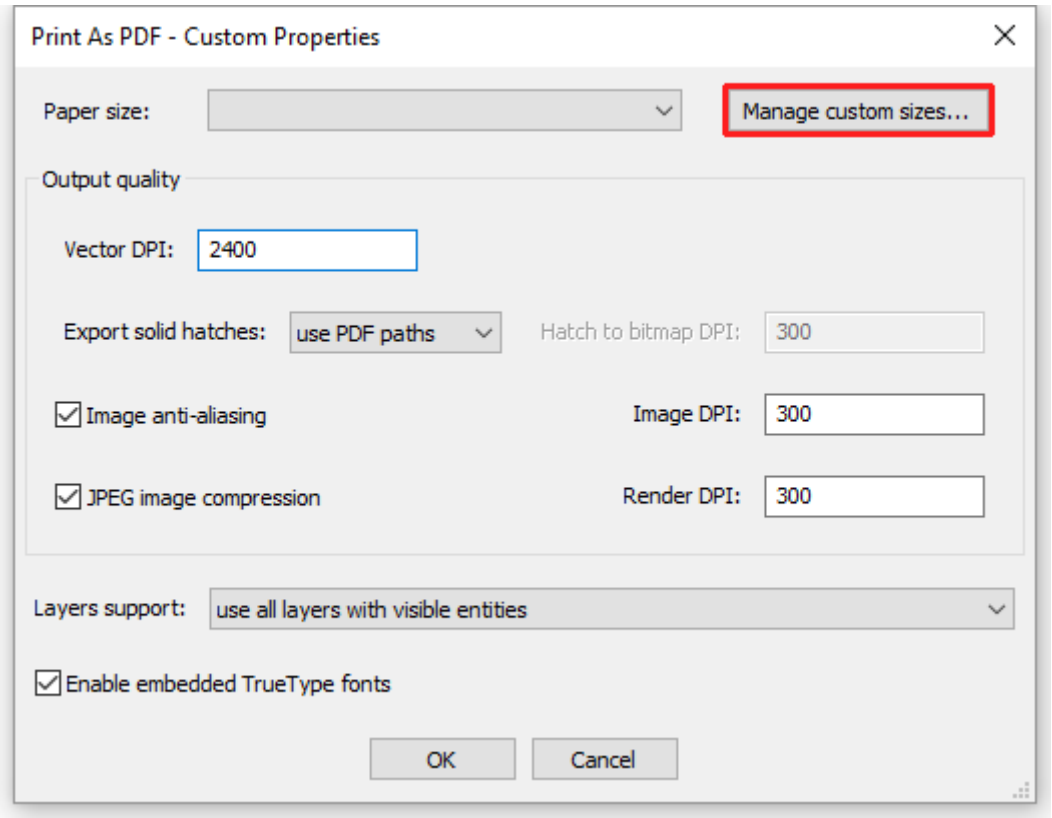
Click the *Settings* tab on the *Plotter Configuration Editor* dialog.



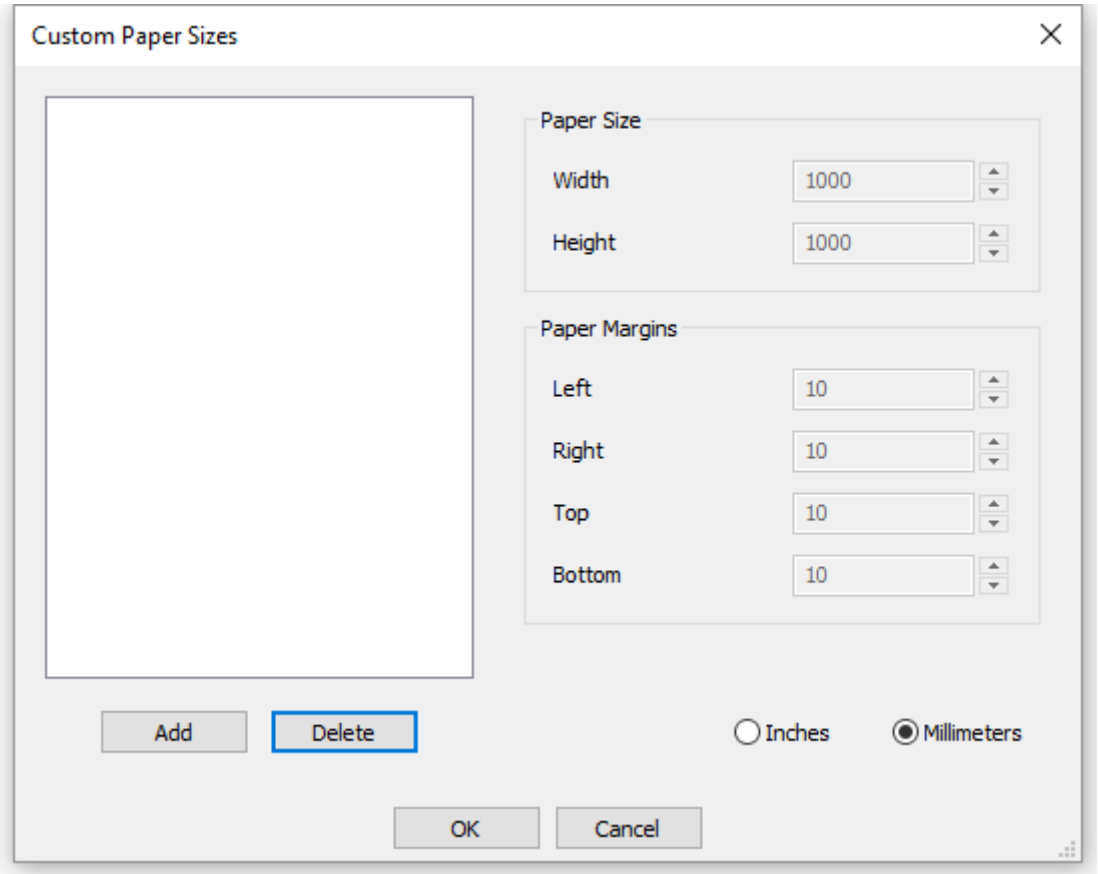
(Option) Edit the *Graphics* settings.

(Option) Execute the *Scale Calibration*.

Click the *Custom Properties* button.
The *Print As PDF - Custom Properties* dialog displays.



(Option) Adjust the *Output quality* and *Layers support* settings.
Click the *Manage Custom sizes ...* button.

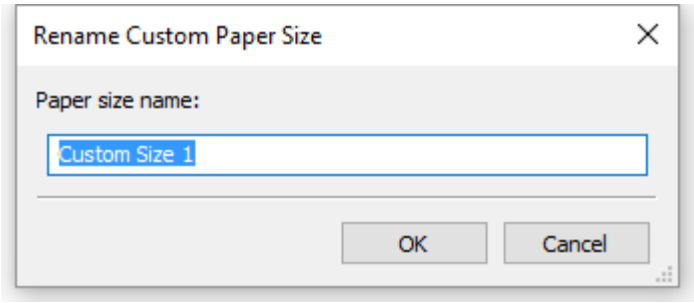


To create a custom paper size do the following:

Click the *Add* button.
A new paper size is added.

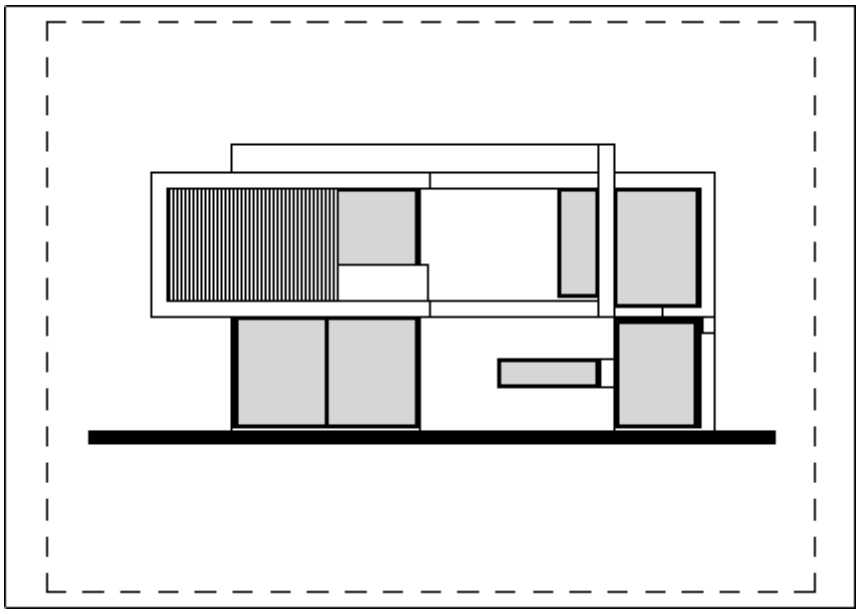
Double click the newly created papersize.

Type a name for the new paper size, then click the *OK* button.



Define *Width* an *Height* for the new paper size.

Define the *Paper Margins*.
Paper margins define the printable area of paper sheet.
If the DISPPAPERMARGINS system variable is *ON*, dashed lines indicate the paper margins on the paper sheet* in a layout.

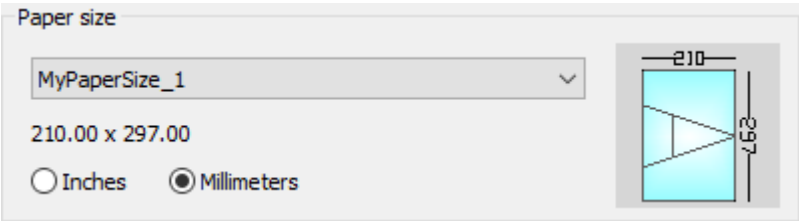


* The paper sheet background displays if the DISPPAPERBKG system variable is *ON*.

Click the *OK* button to create the custom paper size.

Click the *OK* button on each of the previously opened dialogs.

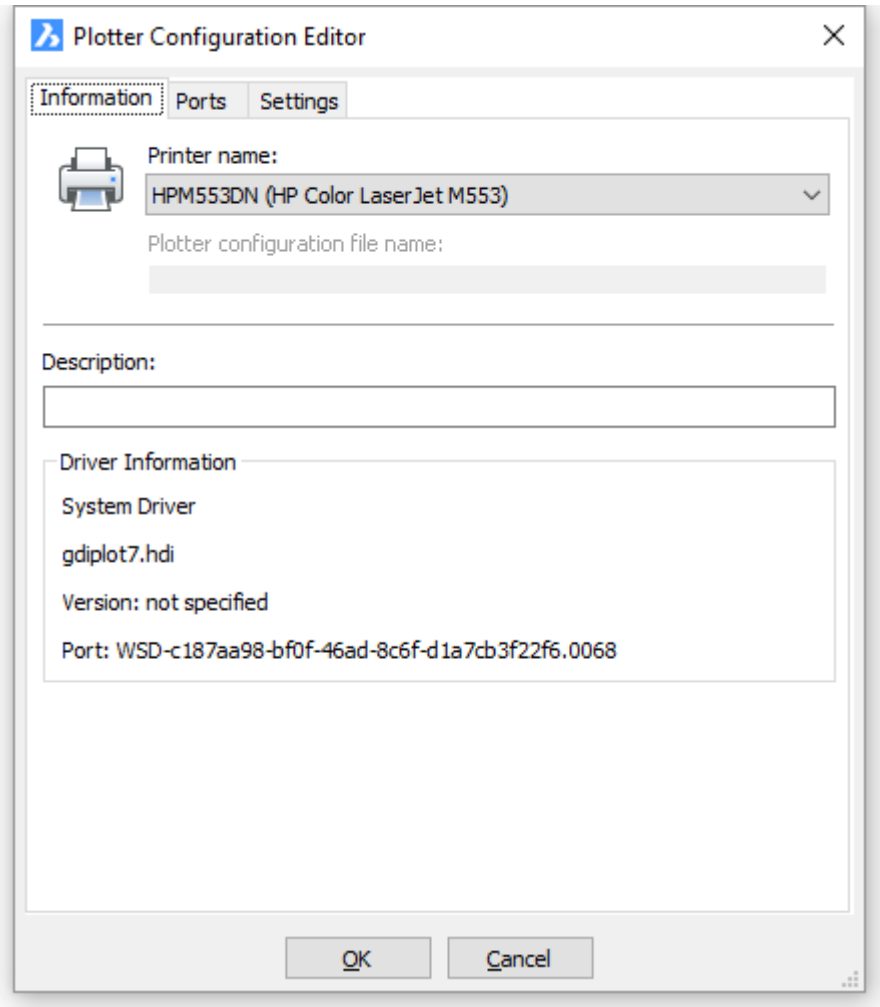
The custom paper size(s) can now be selected in the *Paper Size* list button.



To Create a Plotter Configuration File

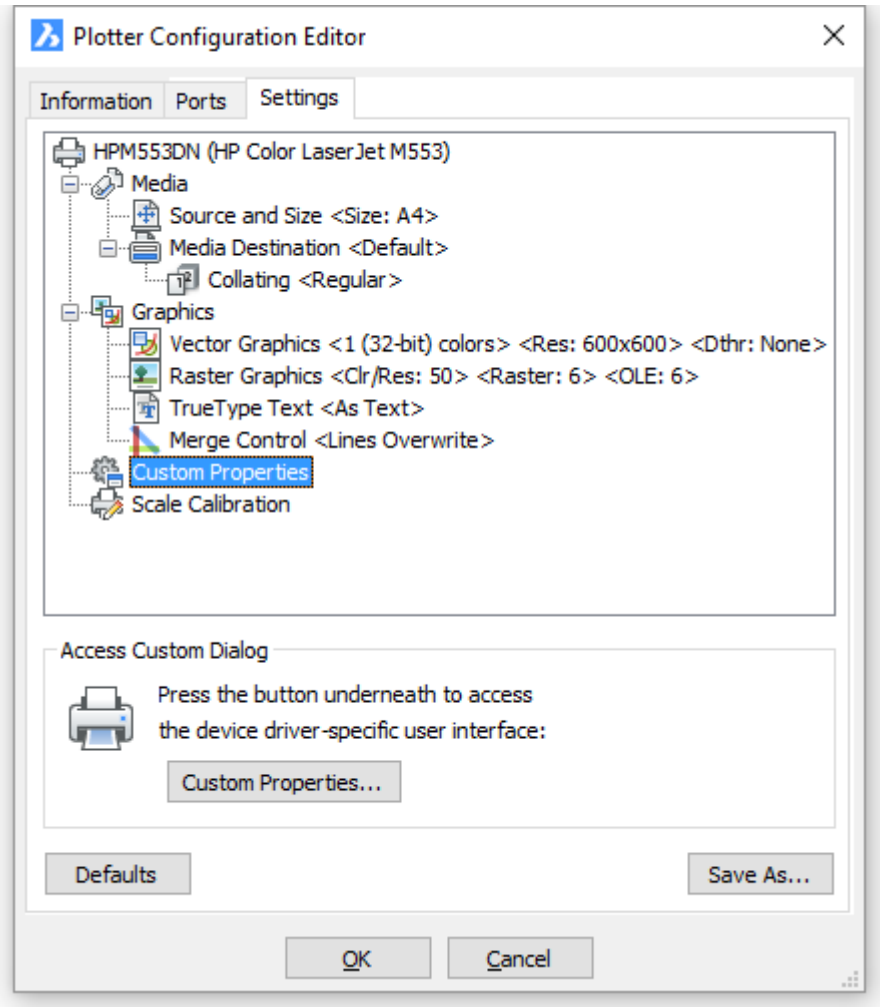
Choose *Plotter Manager...* in the *File* menu, then double click *Create a Plotter Configuration* shortcut.

The *Plotter Configuration Editor* dialog opens.



In the *Printer name* list, select the printer you want to create a plotter configuration (PC3) for.

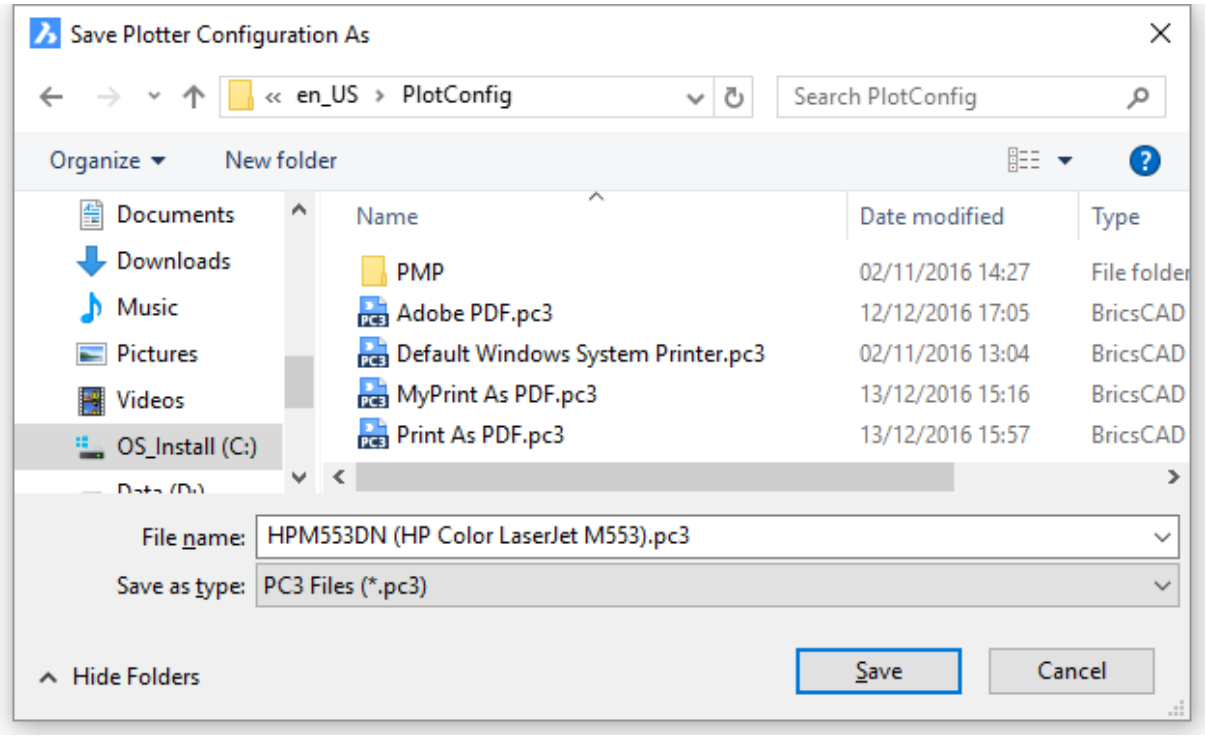
Click the *Settings* tab on the *Plotter Configuration Editor* dialog.



Click the *Custom Properties...* button.
The *<Selected Printer> Properties* dialog opens.
The layout of this dialog is different for each printer.
Custom Properties are not supported on the Linux platform.

On the *Properties* dialog of the selected printer, adjust the printer settings, then click the *OK* button to confirm.
The *<Selected Printer> Properties* dialog closes.

Click the *OK* button on the *Plotter Configuration Editor* dialog.
The *Save Plotter Configuration As* file dialog displays.



(option) Adjust the PC3 file name.
The default name is *<Selected Printer>.pc3*.

Click the *OK* button on the *Changes to a Printer Configuration File* dialog.
The PC3 file is created.

NOTE If you want to create multiple plotter configurations for the same printer, you must adjust the default name in step 7. If you accept the default name, the existing plotter configuration will be overwritten.

To Edit a Plotter Configuration

Do one of the following:

Choose *Page Setup* in the *File* menu.

Type *pagesetup* in the command bar, then press Enter.

Select a plotter configuration (PC3) in the *Printer / Plotter configuration* list.

Click the *Edit Plotter Configuration* button ( ) on the *Print* or *Page Setup* dialogs.
The *Plotter configuration editor* dialog opens.

Proceed in the Create a plotter configuration procedure starting from step 3.

NOTE If you select a system printer in step 2, a new plotter configuration is created for the selected printer.

When you click the *Edit Plotter Configuration* button ( ) in step 3, the *Properties* dialog of the selected printer opens first.

After clicking the *OK* button on the *Properties* dialog the *Plotter configuration editor* dialog opens.

You can then complete the Create a plotter configuration procedure.

To Assign a Plotter Configuration

Choose *Model* or the *Layout* to which you want to assign a specific plotter configuration file.
Do one of the following:

Choose *Page Setup* in the *File* menu.

Type *pagesetup* in the command bar, then press Enter.

Choose one of the following from the *Printer / Plotter configuration* list:

A system printer

A configuration file (PC3 file)

None (uses Default)

Click the *OK* button.

NOTE

- When BricsCAD™ cannot find the plotter configuration file assigned to the drawing, it changes the configuration file assigned to your drawing to *None*.
- The *None* printer device has its own set of paper sizes that can be stored in the drawing very much the same as for any 'real' printer. If a layout is opened and the stored printer is absent, the printer switches to *None* while the paper size is maintained. The missing printer and the previous paper size are indicated upon opening the *Print* dialog. The previous paper size can be saved in the drawing,
- PREVIOUS PAPER SIZE: drawings created by other parties are often set up for a printer that is not available in your office. In such case, BricsCAD resets the printer device to *None* , which results in using the default printer. Instead of resetting the paper size to a default, BricsCAD shows the Previous Paper size, in order to inform you about the intended paper size and allow to choose a similar size on any of the available printers.

Plot Style Tables

TUTORIALS

Control the Color and Lineweight with Plot Style Tables

The *Plot Style Table Editor* can be opened, even if BricsCAD is not open.

To open the Plot Style Table editor

Do one of the following:

Click the *Edit Plot Style* button ( ) on the *Print* dialog.

Choose *Plot Style Manager...* in the File menu, then double click a *CTB* or *STB* file.

In a *Windows Explorer* window, double click a *CTB* or *STB* file.

The *Plot Style Table Editor* displays the plot styles contained in the specified plot style table. The Plot Style Table Editor includes the following tabs:

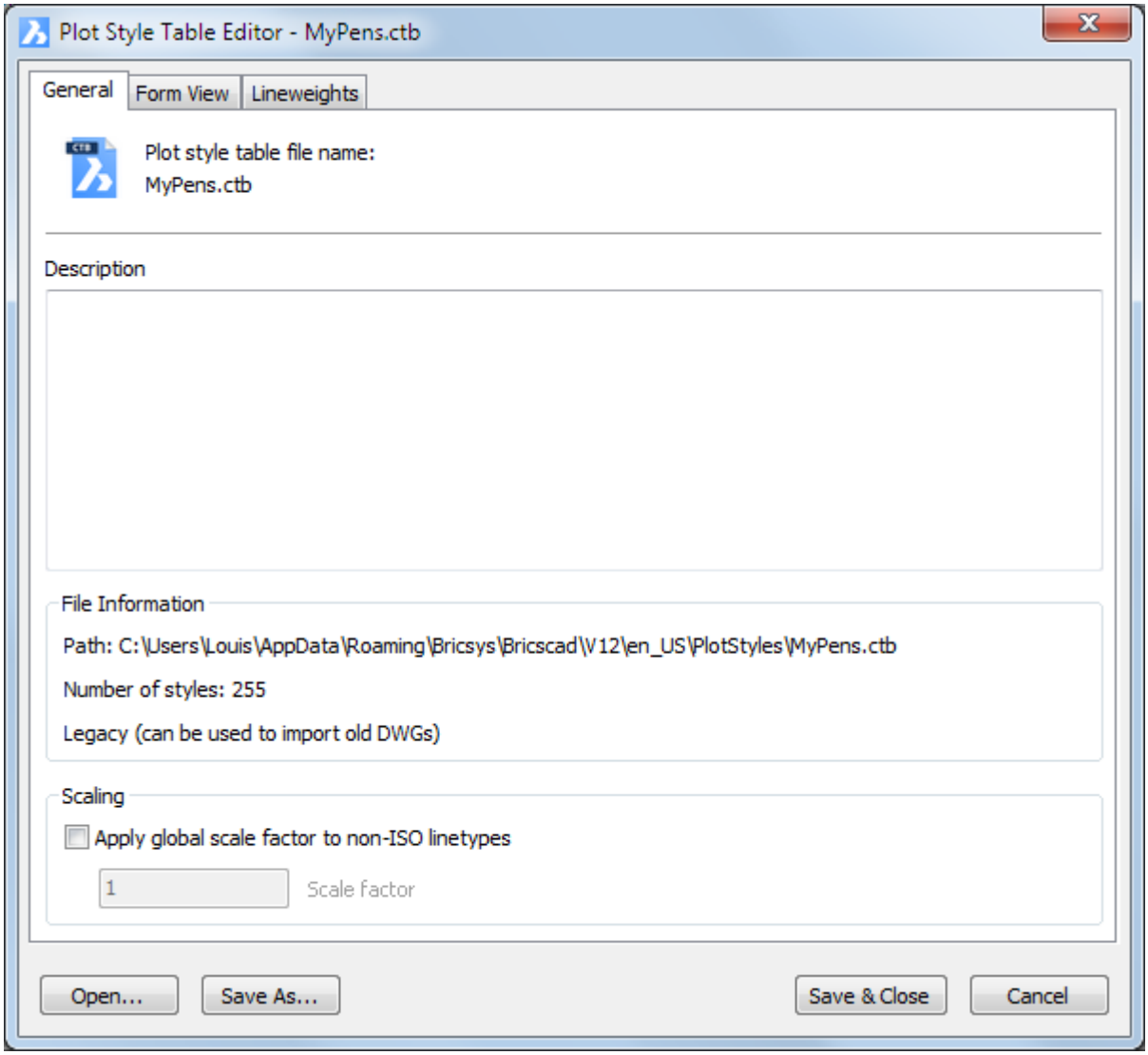
The *General* tab

The *Form View* tab

the *Lineweights* tab

The General tab

The *General* tab lists general information about the plot style table.

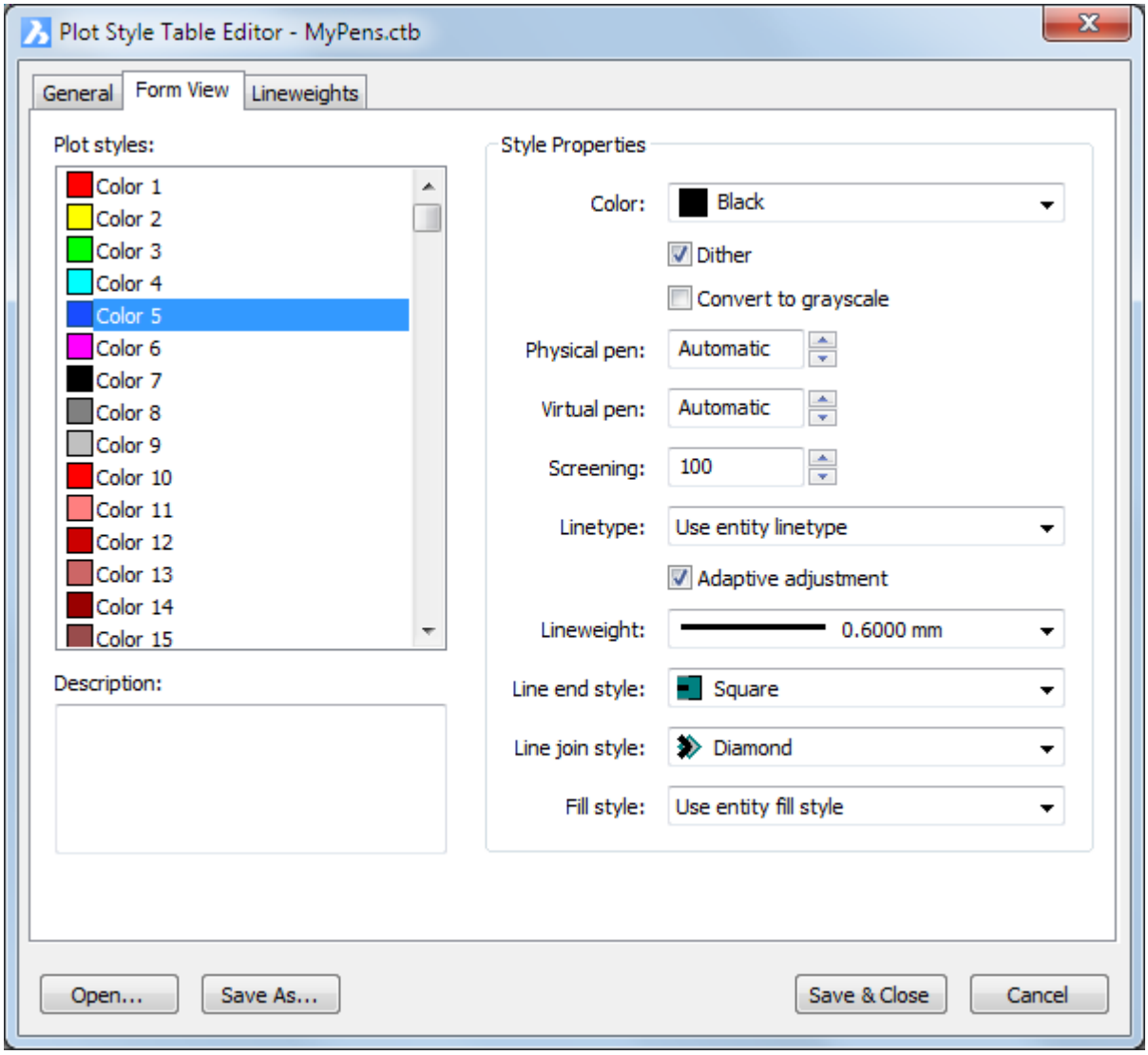


Name	Description
Plot style table file name	Displays the name of the plot style table file you are editing.
Description	Displays a description you want to include about the plot style table.
File Info	Displays information about path, number of plot styles and the version number of the Plot Style Table Editor. Note: A color-dependent plot style table always contains 255 plot styles.
Apply global scale factor to non-ISO linetypes	Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.
Scale factor	Specifies the factor by which to scale non-ISO linetypes and fill patterns.

The Form View tab

The *Form View* tab lists the plot styles contained in the plot style table and their settings. Plot styles are style overrides for your drawings that occur during plotting.

The first plot style in a *Named Plot Style table* is *NORMAL* and represents an entity's default properties (no plot style applied). You cannot modify or delete the *NORMAL* style.



Property	Description
Plot styles	Displays the names of plot styles contained in the selected plot style table. **Color-dependent plot style table:** • Always contains 255 styles (1 for each color). • They are tied to entity color. • You cannot add nor delete a plot style. • You cannot rename a plot style. **Named plot style table:** • Contains one or more plot styles. • You can add or remove plot styles, except for the *Normal* style, which cannot be deleted. • Plot styles can be renamed, except for the *Normal* style, which cannot be renamed.
Description	Displays a description for the selected plot style.
Color	Specifies the plotted color for an entity. The default setting for plot style color is *Use Object Color*. If you assign a plot style color, the color overrides the entity's color in the plot output. • Choose *Other...* to select one of 255 colors from the *Select Color* dialog box. • Choose *True color* to select a color from the *Color* dialog box. If the plot device does not support the color you have specified, the nearest available color will be used. When you have a monochrome device, *black* will be used.
Dither	Depending on the capabilities of your plotter, dithering approximates the colors with dot patterns. When this option is not active, the colors are mapped to the nearest color, resulting in a smaller range of colors when plotting. Dithering is available only whether you select the object's color or assign a plot style color. By default, this option is active. Setting it to Off has no effect in BricsCAD.
Grayscale	Depending on the capabilities of your plotter, converts the colors to grayscale in the plot output.
Pen #	Specifies a pen to use when plotting entities that use this plot style. You can select a pen number from 1 to 32.
Virtual pen #	Specifies a virtual pen number between 1 and 255. Choose *Automatic* to specify that BricsCAD should make the virtual pen assignment from the BricsCAD Color Index. The virtual pen setting in a plot style is used only by non-pen plotters and only if they are configured for virtual pens (select *255 Virtual Pens* under *Color Depth* under *Vector Graphics* on the *Settings* tab in the *Plotter Configuration Editor*).
Screening	Specifies the color intensity of the plot on the paper. The valid range is 0 through 100. If you select 100 the drawing will plotted with its full color intensity. In order for screening to work, the *Dithering* option must be active.
Linetype	Displays a list with an example and a description of each linetype. The default setting for plot style linetype is *Use Object Linetype*. The selected linetype overrides the entity's linetype in the plot output.

Adaptive	Adjusts the scale of the linetype to complete the linetype pattern. You can activate this property if a complete linetype pattern is more important than a correct linetype scaling.
Lineweight	Displays a sample of the lineweight as well as its numeric value. You can modify an existing lineweight if the one you need is not available. The default setting for plot style lineweight is *Use Object Lineweight*. The selected lineweight overrides the entity's lineweight in the plot output.
Line end style	Displays a list of several end styles for the lines. By default, the option *Use Object End Style* is selected. If you select a line end style, the object's line end style will be overridden in the plot output.
Line join style	Displays a list of several join styles for the lines. By default, the option *Use Object Join Style* is selected. If you select a line join style, the object's line join style will be overridden in the plot output.
Fill style	Displays a list of several fill styles. By default, the option Use *Object Fill Style* is selected. If you select a fill style, the object's fill style will be overridden in the plot output.

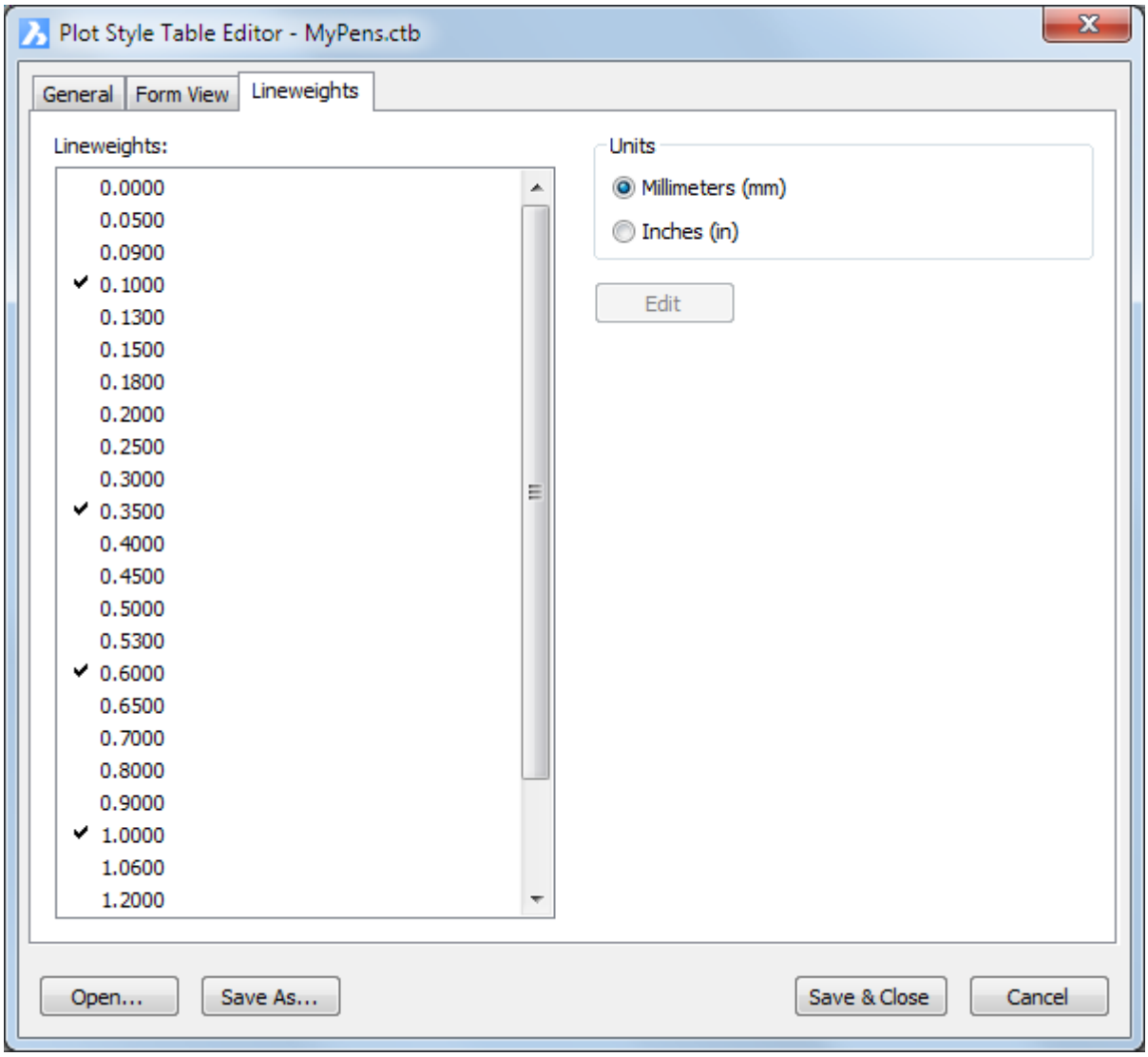
NOTE Objects which are created in *true color* are always plotted using their own object properties (color, lineweight, ...).

The Lineweights tab

The *Lineweights* tab lists the available lineweights.

Lineweights with a tick are used in the *Lineweight* field on the *Form View* tab.

To edit a lineweight, select the lineweight in the list, then click the *Edit* button.



Assigning Plot Style Tables

In BricsCAD you can assign plot style tables to model space and to each of the layouts in a drawing. By assigning different plot style tables to each layout in your drawing, you can control how entities in the layout are plotted. The plot style table affects both model space and paper space entities. To plot the drawing without applying plot style properties, choose 'None (use Default)' from the list of plot style tables.

In STB-type drawings, each entity in the drawing either is assigned a plot style explicitly or *By Layer*.

Missing plot style table

If a plot style table is not found, the text "(missing)" is added to the plot style table name. When you print the drawing, the settings of the default plot style file Default.ctb or Default.stb will be used instead.

To Assign a Plot Style Table

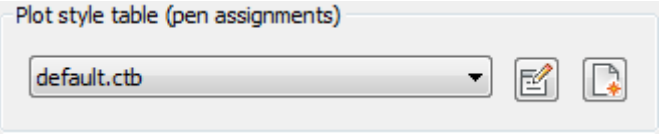
Choose *Model* or the *layout* to which you want to assign a different plot style table.

Do one of the following:

Choose *Print...* in the *File* menu.

Type *print* in the command bar, then press Enter.

Select a *Plot style table* from the list box in the *Plot Style Table (pen assignments)* section.



Click the *OK* button.

To Toggle the Plot Style Table on/off

When you turn off plot style tables, entities print according to their own properties. However, all of the plot style information is saved so you can easily turn on plot styles again. Actual plot style table files are not deleted, and for drawings that use named plot style tables, entities and layers retain their assigned plot styles.

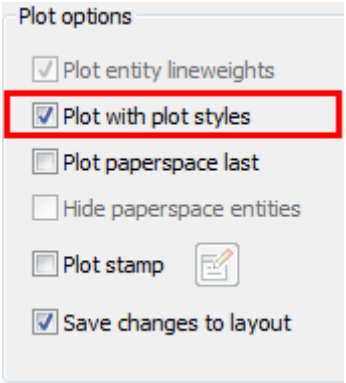
Choose *Model* or the *layout* in which you want to turn the plot style table on/off.

Do one of the following:

Choose *Print...* in the *File* menu.

Type *print* in the command bar, then press Enter.

Click the Plot with plot styles plot option.



The Plot Styles Manager

The *Plot Styles Manager* is a window that lists plot style table files (CTB and STB files).

The Plot Styles Manager allows you to:

Create Color-dependent Plot Style (CTB) and Named Plot Style files (STB) using the *Create a Plot Style Table* wizard.

Edit existing Plot Style Table files.

To Open the Plot Styles Manager

Do one of the following:

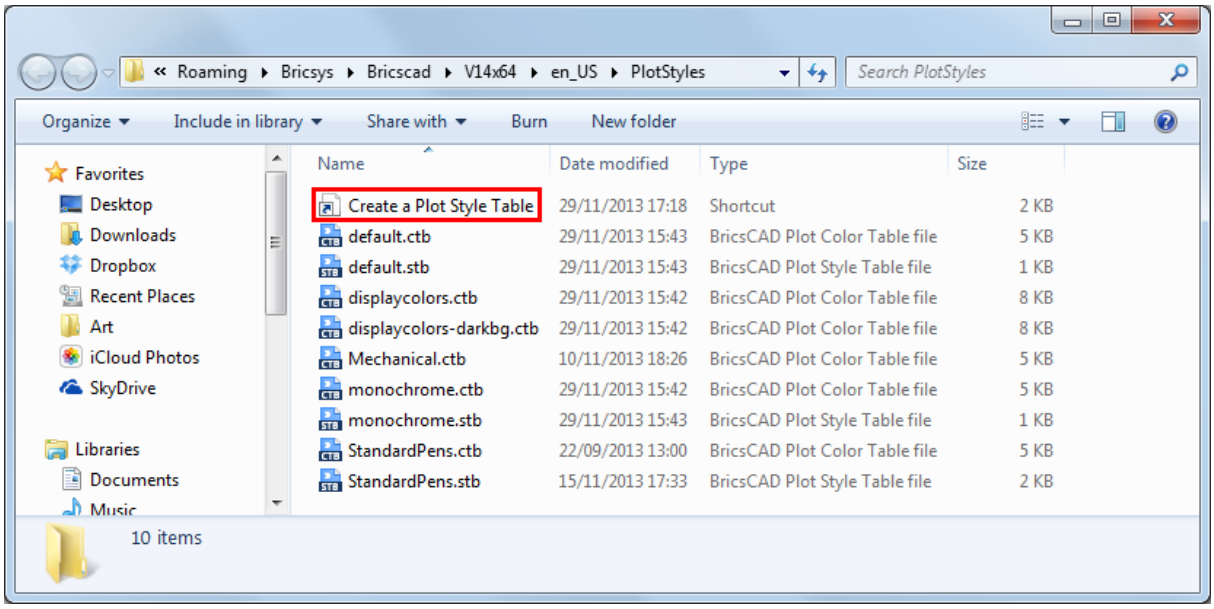
Choose *Plotstyle manager...* in the *File* menu.

Type *stylesmanager* in the command bar then press Enter.

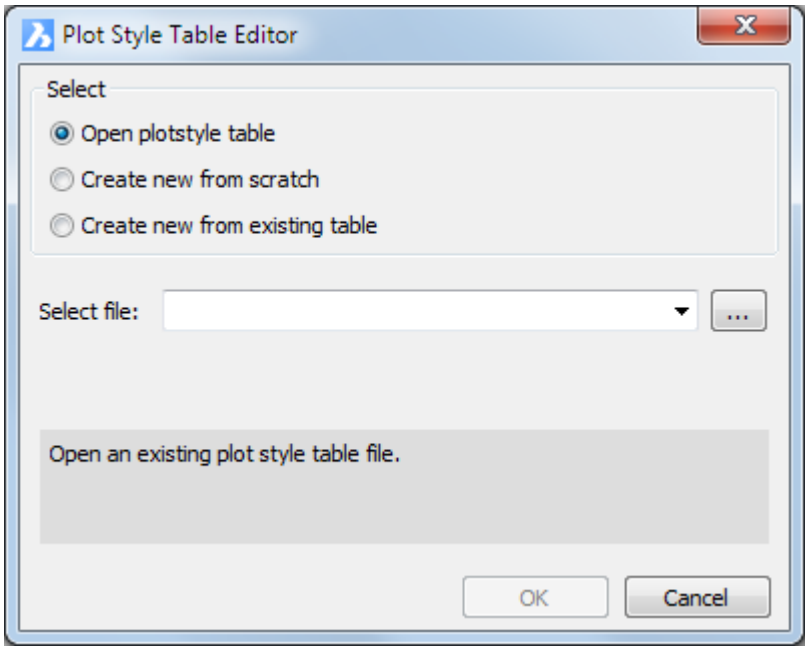
To Create a New Plot Style Table

Open the Plot Styles Manager.

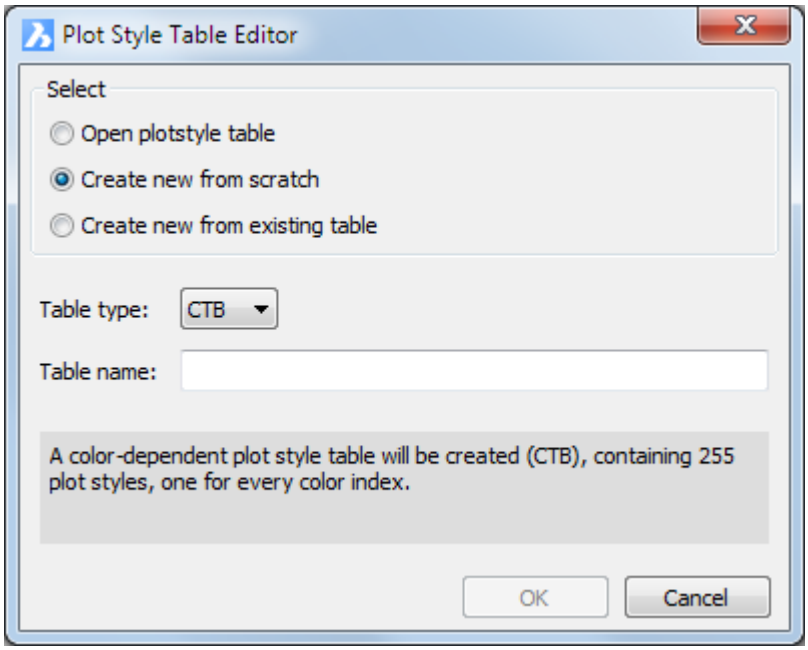
Double click the *Create a Plot Style Table* shortcut.



The *Add Plot Style Table Editor* dialog opens.



Choose *Create new from scratch.*



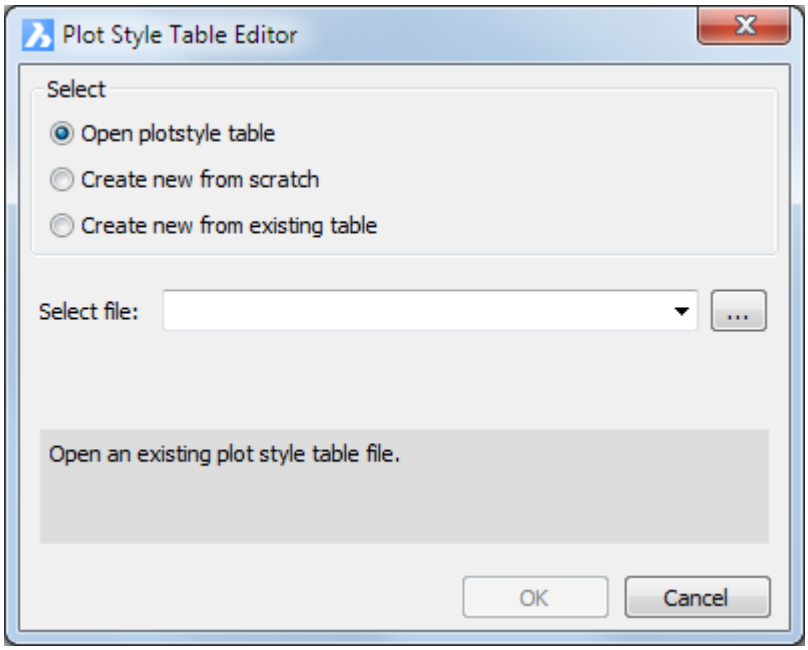
Do one of the following:

Select either *CTB* or *STB* from the *Table type* selection button.

CTB (**C**olor-dependent Plot Style **T**a**b**le) - bases plots on entity colors. Plotting parameters, such as pen thickness and linetype, are matched to the color of entities. This system is older and less flexible than named plot styles, but is the default in all new drawings. Creates a CTB file.

STB (Named Plot **S**tyle **T**a**b**le) - bases plots on plot styles. Plotting parameters can be given to every entity and every layer. This system is new and more flexible, but may be incompatible with other software that reads DWG files. Creates an STB file.

Choose *Create from existing table*, then choose a file in the drop down list box or click the browse button to select an existing CTB or STB file.



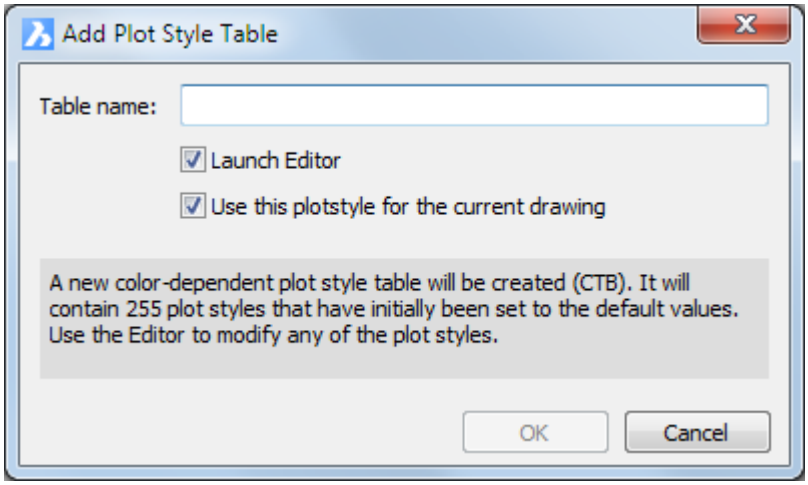
Type a name in the *Table name* field, then click the *OK* button.

The Plot Style Table Editor dialog displays.

To Create a New Plot Style Table in the Print Dialog

Open the Print dialog.

Click the *Create new plot style* button ( ) on the *Print* dialog.
The *Add Plot Style Table* dialog displays.



Type a name in the *Table name* field, then click the *OK* button.
If the Launch Editor option is chosen, the Plot Style Table Editor dialog displays.
If the *Use this plotstyle for the current drawing* option is chosen, the new plot style table is used in the current layout.

NOTE The above procedure creates a CTB file in CTB-type drawing and a STB file in a STB-type drawing.

To Edit a Plot Style Table

Open the Plot Styles Manager.

Double click the plot style table you want to edit.

The Plot Style Table Editor dialog displays.

To Edit a Plot Style Table in the Print Dialog

Open the Print dialog.

Select a plot style table from the *Plot Style Table* list button.

Click the edit *Edit plot style* button ( ).

The Plot Style Table Editor dialog displays.

Batch Plotting

Commands: PUBLISH, -PUBLISH, SHEETSET

The *Publish* command allows to print a sheet list (= a list of model space or paper space layouts) to a printer. A sheet list can be saved to a DSD (Drawing Set Description) file.

The *SheetSet* command manages sheet sets and allows to print, publish or etransmit a sheet set as a whole or a selection of sheets. A sheet set is saved to a DST file (Sheet Set Data) file.

Rendering

Commands: RENDER, SHADEMODE, RENDERPRESETS, ANIPATH

Rendering is the process of generating an image from a 3D model. The model contains the geometry, viewpoint, texture, lighting and shading information. The result is a digital image which can be saved as a raster image file: bmp, jpeg or png.

Rendering in BricsCAD is based on the Redway3d rendering engine. By default the rendering engine will try to leverage the hardware on the graphics card. If experiencing problems, please switch off the *RenderUsingHardware* user preference. It is recommended to install the most recent drivers for your graphics card. The *IsRedSdkHardwareRenderingSupported* user preference reports if Red SDK hardware rendering is fully supported and the initial value of *RenderUsingHardware* user preference is set accordingly.

The *RenderPresets* command allows to create and edit render presets, and to set the current render preset through a dialog box.

Setting the Render Preferences

Open the Settings dialog.

Under *Program Options* expand the *Rendering* settings category.

Rendering	
Current material	
Lighting units	[2] Enable photometric lighting using International units (lux)
Default lighting	[0] Default lighting is used only in case no other lights are switched on
Linear contrast	0
Linear brightness	0
Render using hardware	[1] Prefer hardware (faster)
Real world scale	☑ Render real-world scale materials
Sky status	[0] No sky
Texture map path	C:\ProgramData\Bricsys\RenderMaterials\textures;
Render material directory path	C:\ProgramData\Bricsys\RenderMaterials\materials\
Render material static directory path	C:\Program Files\Bricsys\BricsCAD V17 en_US\RenderMaterialStatic\
Tile mode light synch	☑ Synchronize lighting

Current material: Specifies the material for new entities. By default the *Global* material is used.

Lighting units:

0: No lighting units are used. Enables generic lighting.

1: Enables American lighting units: Foot-Candles

2: Enables International lighting units: Lux.

Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 or 2.

Default lighting: A distant light that follows the view direction. This setting can be different per viewport.

0: Default lighting is used if no lights are switched ON.

1: Default lighting is used instead of the light sources defined in the drawing.

Linear contrast: Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light.

Linear brightness: Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport.

Render using hardware: Choose *User software only (slower)* if hardware rendering fails. It might be necessary to restart BricsCAD after changing the setting.

Real world scale: Controls the rendering of materials with units set to real-world scale

Sky status:Specifies whether sky illuminiation is computed at render time. (Not implemented yet)

Texture map path: Defines the search path for texture map images. In the BricsCAD program folder exist three subfolders under *Textures*, each containing a number of texture files of the same name. Images in folder 1 have a size of 256 x 256 pixels, folder 2 contains images of 512 x 512 pixels, images in folder 3 have a size of 1024 x 1024 pixels. If the *Diffuse map* setting of a material uses the image name only (not path), you can control the quality of a rendered image by setting the TextureMapPath user preference to folder 1, 2 or 3.

Render material directory path: Specifies the path of the folder where the render materials for the Material Browser are installed.

Render material static directory path: Specifies the path of the folder which contains read-only resources for predefined materials, such as their previews.

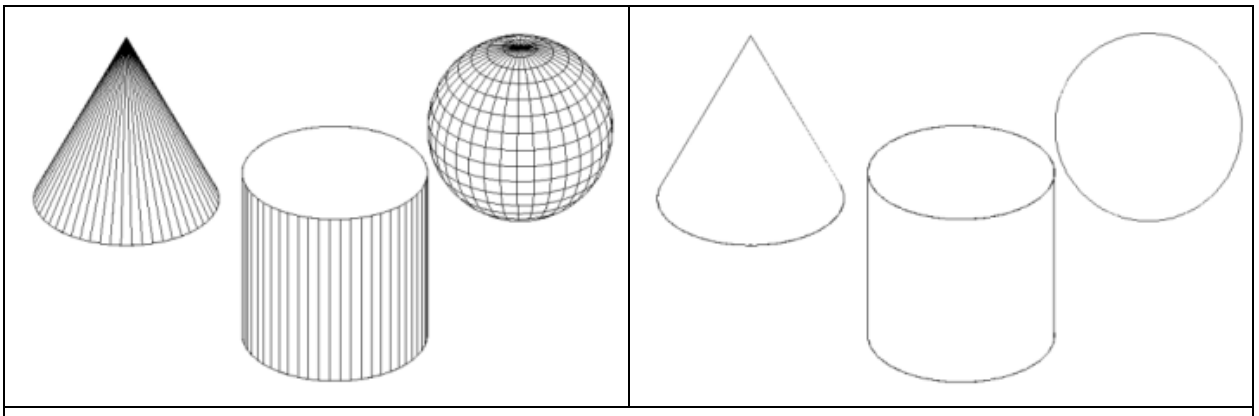
Tile mode light synch: Controls the synchronization of lighting in all model space viewports. (Internal use only)

Render Tools

Icon	Tool Name	Keyboard	Description
	Render	render	Photorealistic rendering using materials and lights
	Hide	hide	Hidden line view (*). If the previous shademode option was *2Dwireframe*, wireframe representation is restored if you zoom or pan. If the previous shademode option was other than *2Dwireframe,* the hidden line representation is kept until a different shade mode is chosen.
	Shade	shade	Shades the entities between the polygon faces. Curved surfaces are segmented.
	2D Wireframe	shademode + 2	Wireframe representation
	3D Wireframe	shademode + 3	Wireframe representation
	Hidden	shademode + H	Hidden line view (*)
	Flat	shademode + F	Shades the entities between the polygon faces. Curved surfaces are segmented
	Gouraud	shademode + G	Shades the entities between the polygon faces. Curved surfaces are smoother and more realistic
	Flat + Edges	shademode + L	Combines flat shading and wireframe lines and curves
	Gouraud + Edges	shademode + O	Combines gouraud shading and wireframe lines and curves
	Materials	materials	Opens the Drawing Explorer - Materials
	Material Map	materialmap	Allows to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.
	Light List	lightlist	Opens the Drawing Explorer - Lights
	Sun Properties	sunproperties	Opens the sun properties editor
	Geographic Location	geographiclocation	Opens the Geographic Location dialog.

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

(*) Set the DISPSILH system variable On to display silhouette lines instead of a polygon mesh for curved surfaces.



Hidden line of a cone, a cylinder and a sphere with DISPSILH Off (left) and On (right).

Creating a rendered image of a model

The rendered image displays in the current viewport, in a separate window or can be exported to a file.

The background of a rendered view can be controlled by the Background property of the current named view.

The ANTIALIASRENDER system variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5.

Note: high anti-alias values incur a high calculation cost.

NOTE The values of the lighting system variables influence the rendering result.

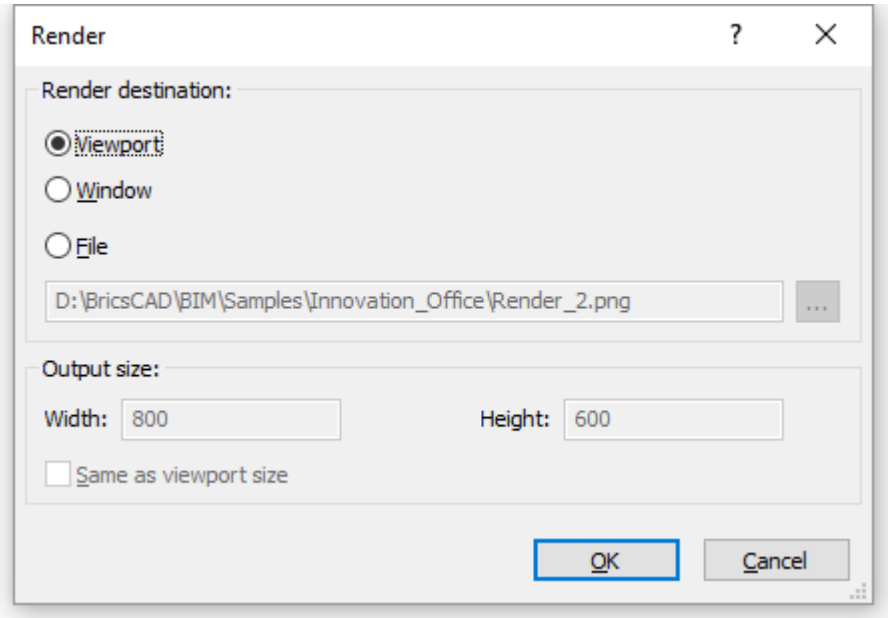
To Render a Viewport

Do one of the following

Click the *Render* tool button ( ) on the *Rendering* toolbar.

Type *render* in the command bar, then press Enter.

The *Render* dialog displays:



Click the *Viewport* option.

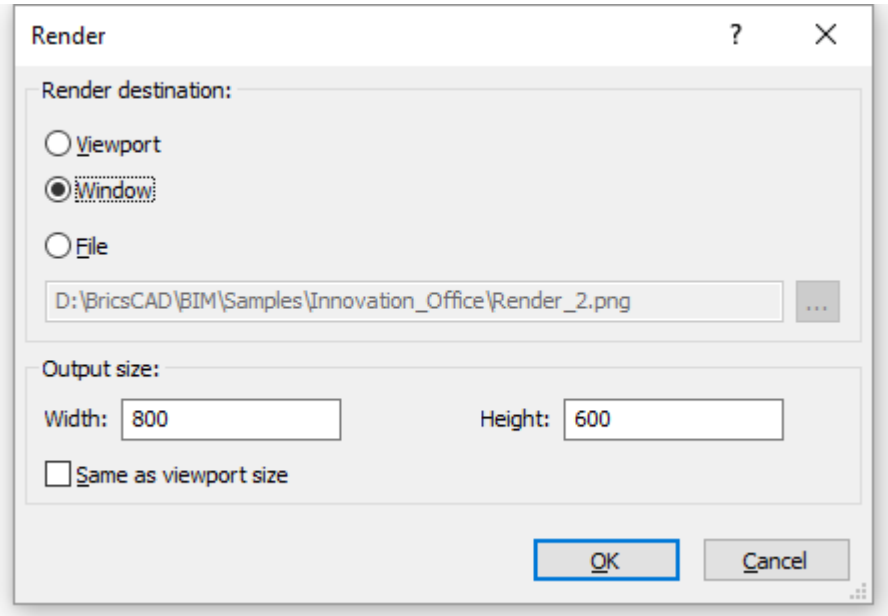
Click the *OK* button.
The view is rendered.

To Render in a Window

Do one of the following

Click the *Render* tool button ( ) on the *Rendering* toolbar.

Type *render* in the command bar, then press Enter.
The *Render* dialog displays:



Click the *Window* option.

Click the *OK* button.
The view is rendered in a popup window.

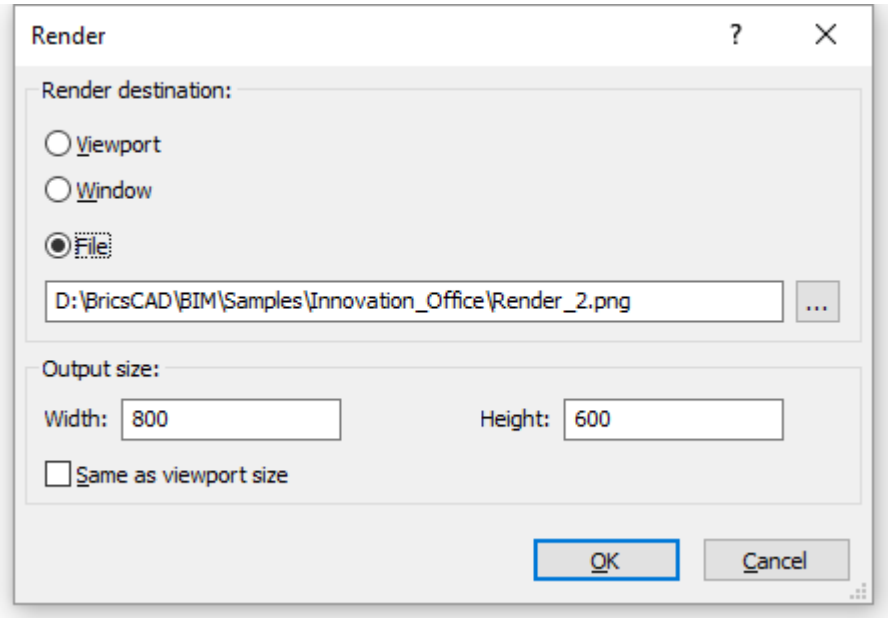
To Save a Rendering to a File

Do one of the following

Click the *Render* tool button ( ) on the *Rendering* toolbar.

Type *render* in the command bar, then press Enter.

The *Render* dialog displays.
Click the *File* radio button (1) on the *Render* dialog.



Click the *File* option.

Click the *Browse* button ( ), at the right hand side of the *File* field.
The *Render output file* dialog displays.

Select a folder and type a name for the output file in the *File name* field.

Click the *Save as type* button to select the file type.
The options are: *bmp*, *jpg* and *png*.

Click the *OK* button
The *Render Output File* dialog closes.

Do one of the following:

Type the desired resolution for the output size in the *Width* and *Height* fields.

Check the *Same as viewport* size option to set the outsize to the size of the current viewport.

Click the *OK* button to create the file.

Render Materials

Command: MATERIALS, MATBROWSEROPEN, MATBROWSERCLOSE, MATERIALMAP

The *Materials* command creates materials and edits their properties through the Drawing Explorer.

The *MatBrowserOpen* command opens the materials browser, which shows an overview of available high-resolution render materials.

The *MatBrowserClose* command closes the materials browser.

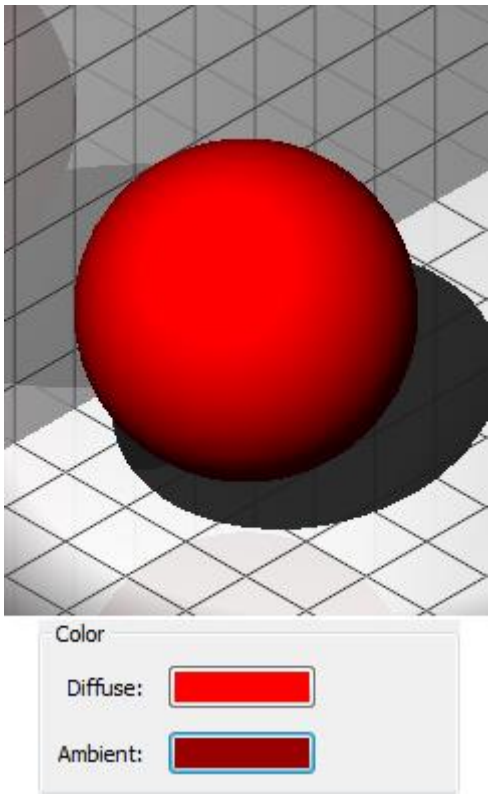
The *MaterialMap* command allows to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.

Material definitions are saved in the drawing. Each drawing contains the *Global* material. It is not possible to delete or rename the *Global* material. You can edit the properties of the *Global* material though.

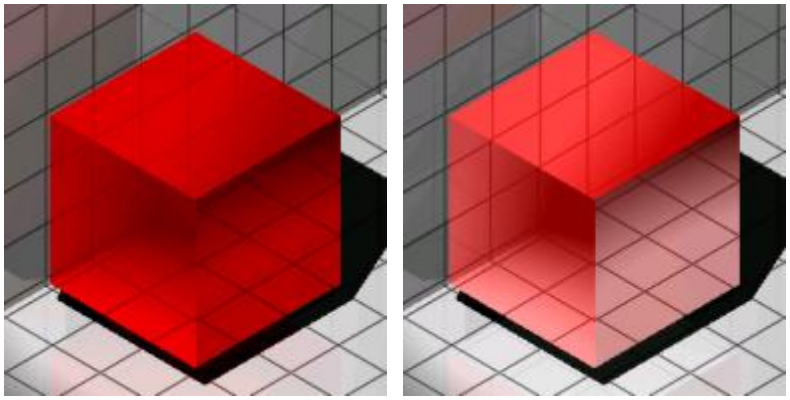
Glossary

Diffuse color: the color that the object reveals under pure direct, white light. It is perceived as the color of the object itself.

Ambient color: the color of the object in the shadow, under ambient light rather then direct light.



Highlights: highly reflective highlights in the color of the material.



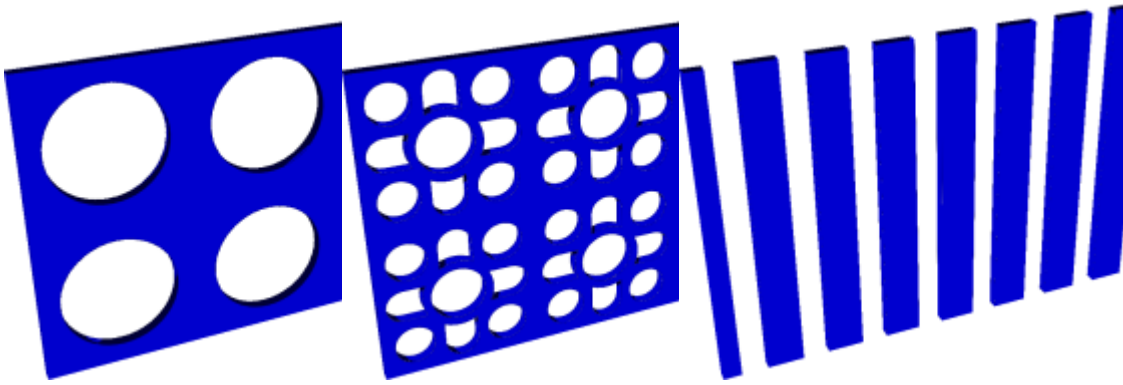
Metallic Highlights (left) and Non-metallic Highlights (right)

Shading: how the color and brightness of a surface varies with lighting.

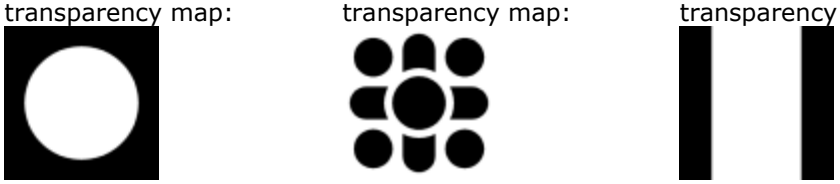
Texture-Mapping: adds detail to a surface which is not included in the 3D model itself.

Diffuse map: adds a structure or pattern to a surface to simulate a material.

Transparency map: sets the transparency of a texture from 0% (black) to 100% (white) and anything between (grey scales).

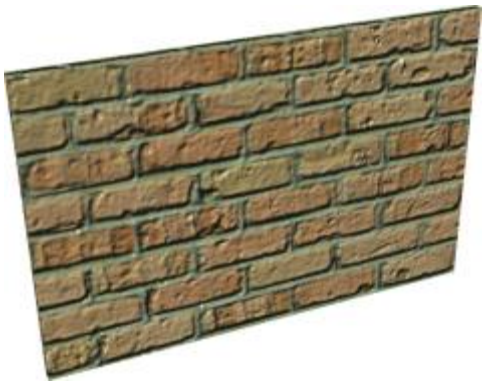


transparency map: transparency map: transparency map:



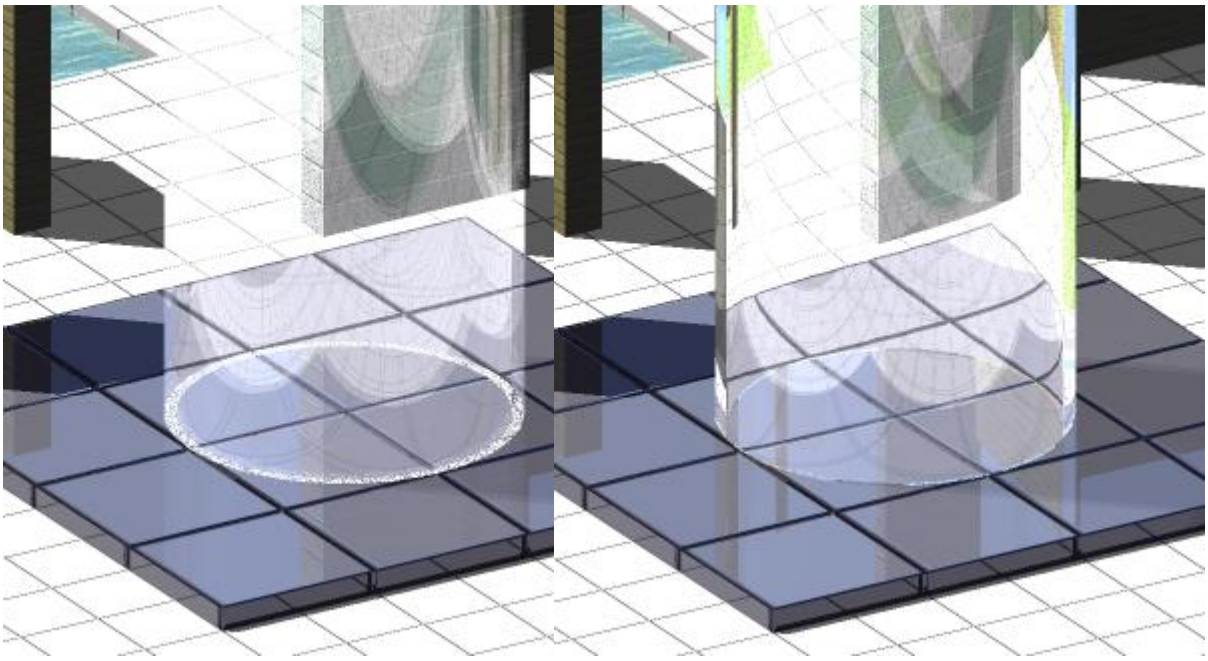
Bump map: simulates a certain roughness or bumpiness on surfaces.

brick pattern: diffuse map only



brick pattern: diffuse map and bump map applied

Reflectivity: the property of a surface to reflect light.
Transparency: sharp transmission of light through solid objects.
Translucency: scattered transmission of light through solid objects.
Refraction: bending of light when passing through transparent objects.



Refraction index = 1.000 Refraction index = 2.000

self-illumination: it appears as if a surface is emitting its own light. No light is cast on other objects.

Defining Materials

See the Materials command.

Assigning Materials

Materials can be assigned to a layer and to entities. If the *Material* property of an entity is *BYLAYER*. The material assigned to the layer of the entity will be used.

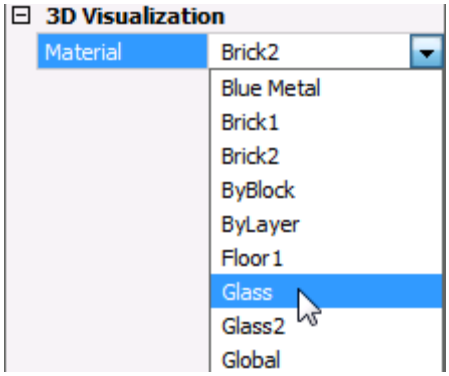
By default new entities get the *BYLAYER* material setting.

To assign a material to entities

Method 1: Using the Properties Bar.

Select the entities or subentities (e.g. a faces a 3D solids).

In the Properties bar, under *3D Visualization > Material*, click the down arrow and select the material in the *Material* list.



Method 2: Using the Render Materials panel

To assign a material to a single entity:

Select the material in the Render Materials panel.

Press and hold the left mouse button, then drag the material onto the entity.

To assign a material to multiple entities:

Select the entities.

Select the material in the Render Materials panel.

Press and hold the left mouse button, then drag the material onto one of the selected entities.

To assign a material to a layer

Do one of the following

Click the *Explore Layers* tool button ( ) in the *Entity Properties* toolbar.

Choose *Layers > Layers...* in the *Settings* menu.

Type *layer* in the command bar, then press Enter.

The *Drawing Explorer - Layers* dialog opens.

In the *Drawing Explorer - Layers* dialog click the *Detail view* tool button ( ).

Click the *Material* column of the layer you want to assign a material to.

Click the down arrow and select the material in the *Material* list.

Lights

Commands: DISTANTLIGHT, GEOGRAPHICLOCATION, LIGHT, LIGHTLIST, POINTLIGHT, SPOTLIGHT, SUNPROPERTIES, WEBLIGHT

Lighting is the key to make the viewer believe to look at a realistic scene.

Glossary

Ambient light: light from an unspecified source.

Back light: light that adds depth and dimension by creating highlights.

Distant light: light that is cast evenly throughout a scene with shadows all in the same direction.

Fall-off: the attenuation or decrease in brightness the further from the light source.

Fill light: the light that brightens dark areas and softens shadows from the main light.

Point light: light that shines in all directions from a central spot.

Spot light: focusable light that is aimed in one area.

Photometric web: a 3D representation of the light intensity distribution of a light source.

Web light: approximates real-world light distribution using a 3D representation of the light intensity. Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 (American lighting units) or 2 (International lighting units).

Attenuation: the fall-off in brightness the further from the light source.

System Variables

LIGHTINGUNITS (Lighting units):

0: No lighting units are used. Enables generic lighting.

1: Enables American lighting units: Foot-Candles

2: Enables International lighting units: Lux.

DEFAULTLIGHTING (Default lighting): A distant light that follows the view direction. This setting can be different per viewport.

0: Default lighting is used if no lights are switched ON.

1: Default lighting is used instead of the light sources defined in the drawing.

LINEARCONTRAST (Linear contrast): Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light.

LINEARBRIGHTNESS (Linear brightness): Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport.

General Properties

The general properties are common to all lights:

Light	
Name	SpotHigh
Type	Spot
On/Off	Off
Shadows	On
Intensity factor	500
Filter color	RGB:255,255,255
Plot glyph	No
Glyph display	Auto

Name: a user defined name for the light.

Type: defines the type of light: spot, point, web or distant.

On/Off: specifies whether the light is turned on or off.

Shadows: specifies whether the light casts shadows or not. Turning shadows off to increases performance.

Intensity factor: multiplies the *Lamp Intensity* property, the result is the *Resulting Intensity*.

Filter color: defines the color of the light.

Plot glyph: if on, the light glyph is plotted.

Glyph display: controls the display of the light glyph

Photometric Properties

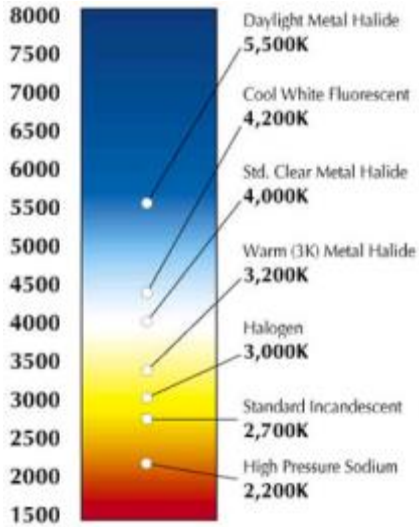
LIGHTINGUNITS = 1 (American units) or 2 (International units).

Photometric Properties	
Lamp intensity	1500.000 Cd
Resulting intensity	750000
Lamp color	D65White
Resulting color	RGB:255,255,255
Geometry	
Attenuation	
Type	Inverse Square
Use limits	Yes
Start limit offset	0 mm
End limit offset	20000.0 mm

Lamp intensity: Defines the brightness of the light. The lamp intensity is expressed in candela (cd) which is the SI base unit of luminous intensity: the power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function)

Resulting intensity: the product of the lamp intensity and and intensity factor.

Lamp color: defines the inherent color of the light or color temperature in Kelvin or standards.



color temperatures

D65White
Fluorescent
Cool White
White Fluorescent
Daylight Fluorescent
Incandescent
Xenon
Halogen
Quartz
Metal Halide
Mercury
Phosphor Mercury
High Pressure Sodium
Low Pressure Sodium

lamp color list

Resulting color: this is the final color of the light: a combination of the lamp color and the filter color.

Photometric web: If the light type is *Web*, *Photometric Web* and *Web offsets* are available (not implemented yet).

Geometry Properties

Position: controls the location of the light.

Target: defines the target point for point lights, spotlights and weblights

Targeted: switches the *target* property on/off.

Attenuation Properties

Attenuation	
Type	Inverse Linear
Use limits	Yes
Start limit offset	0 mm
End limit offset	20000.0 mm

In the real world, an object appears darker if it is farther away from the light source. The attenuation properties define how the light diminishes over distance. Attenuation does not apply to photometric lights.

An alternative method to control the distance a light shines is the use of limits, which define the points from where a light starts to shine and where it stops. Using limits decreases the time needed to calculate the illumination of a scene.

Attenuation properties apply to spot lights and point lights.

Type: the options are *none*, *inverse linear* and *inverse square*

None: no attenuation. The distance to the light source has no influence.

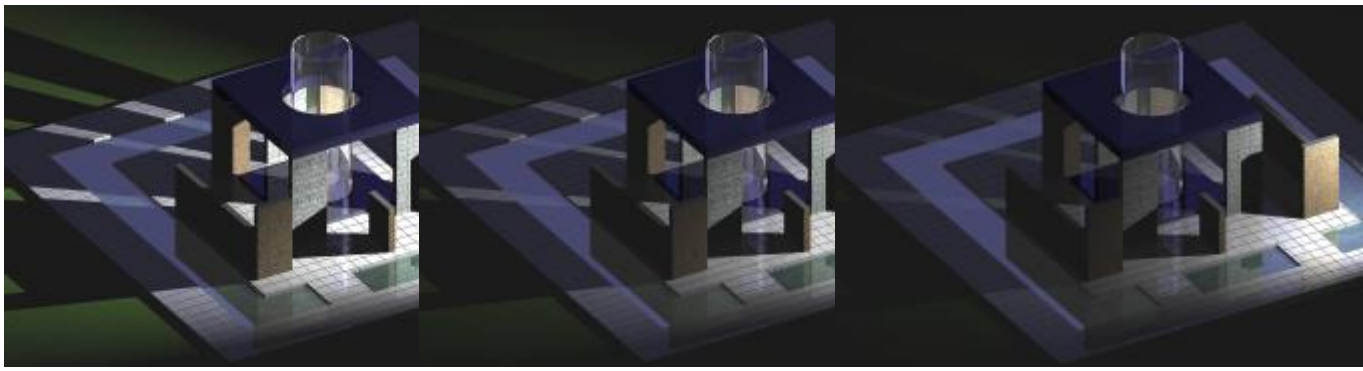
Inverse Linear: the attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, light is half as strong; at a distance of 4 units, light is one quarter as strong.

Inverse Square: the attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, light is one quarter as strong; at a distance of 4 units, light is one sixteenth as strong.

Use Limits: if this property is set to Yes, the *Start limit offset* and *End limit offset* fields become active.

Start limit offset: defines the point where the light starts to shine, measured from the center of the light.

End limit offset: defines the point where the light stops to shine, measured from the center of the light.



No attenuation | Inverse linear attenuation | Inverse square attenuation

The *Attenuation* properties apply to generic lights only (LIGHTINGUNITS = 0).

Inverse square attenuation is applied automatically for photometric lights (LIGHTINGUNITS = 1 or 2).

Rendered Shadow Details

Rendered Shadow Details	
Type	Soft (shadow map)
Map size	256
Softness	1

Type: Sets the shadow type.

Sharp (raytraced): generates sharp edged shadows.

Soft (shadow map): generates realistic shadows with soft edges.

Soft (sampled): Not supported yet

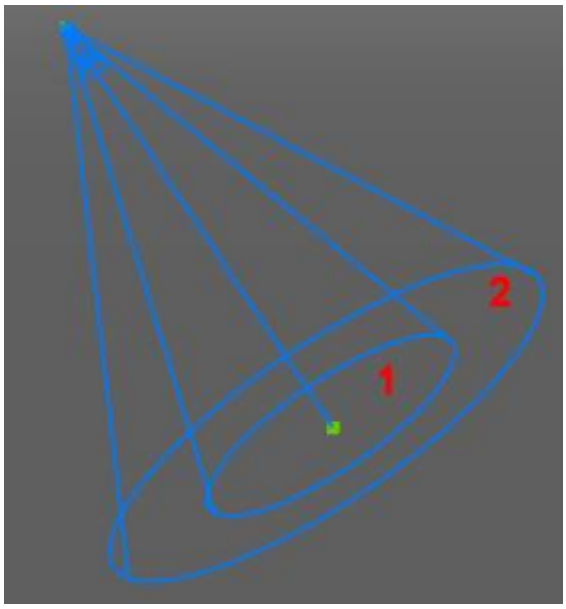
Map size: specifies the amount of memory for shadow calculation. Click the field, then select a value in the drop-down list (64/128/256/512/1024/2048/4096).

Softness: sets the softness of the shadow map. Enter a value between 1 and 10.

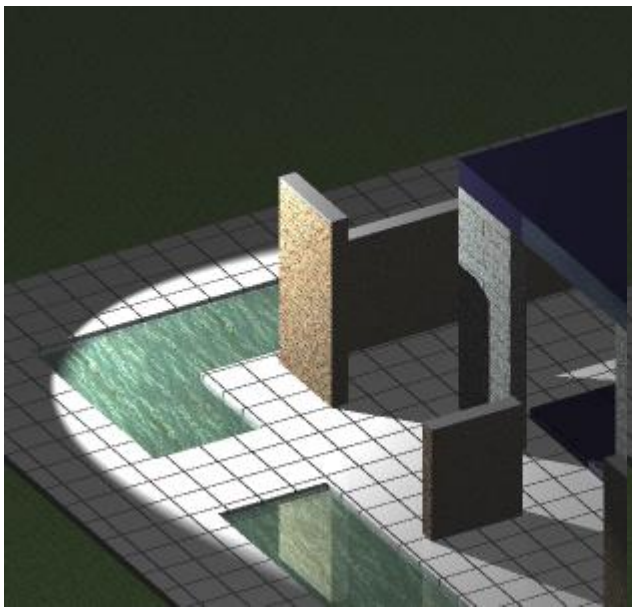
Spotlight Properties

Hotspot angle	45
Falloff angle	50

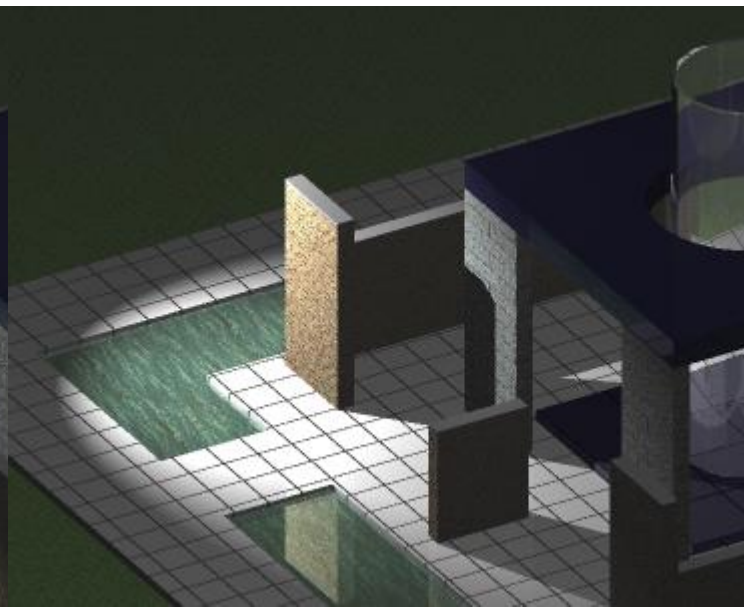
When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot = 1) that is surrounded by an area of lesser intensity (falloff = 2).



Hotspot angle: angle of the central light cone (defines the hotspot)
Falloff angle: angle of the full light cone



Hotspot angle = 55, Falloff angle = 60



Hotspot angle = 30, Falloff angle = 60

The difference between the hotspot angle and the falloff angle defines the area of lesser light intensity.

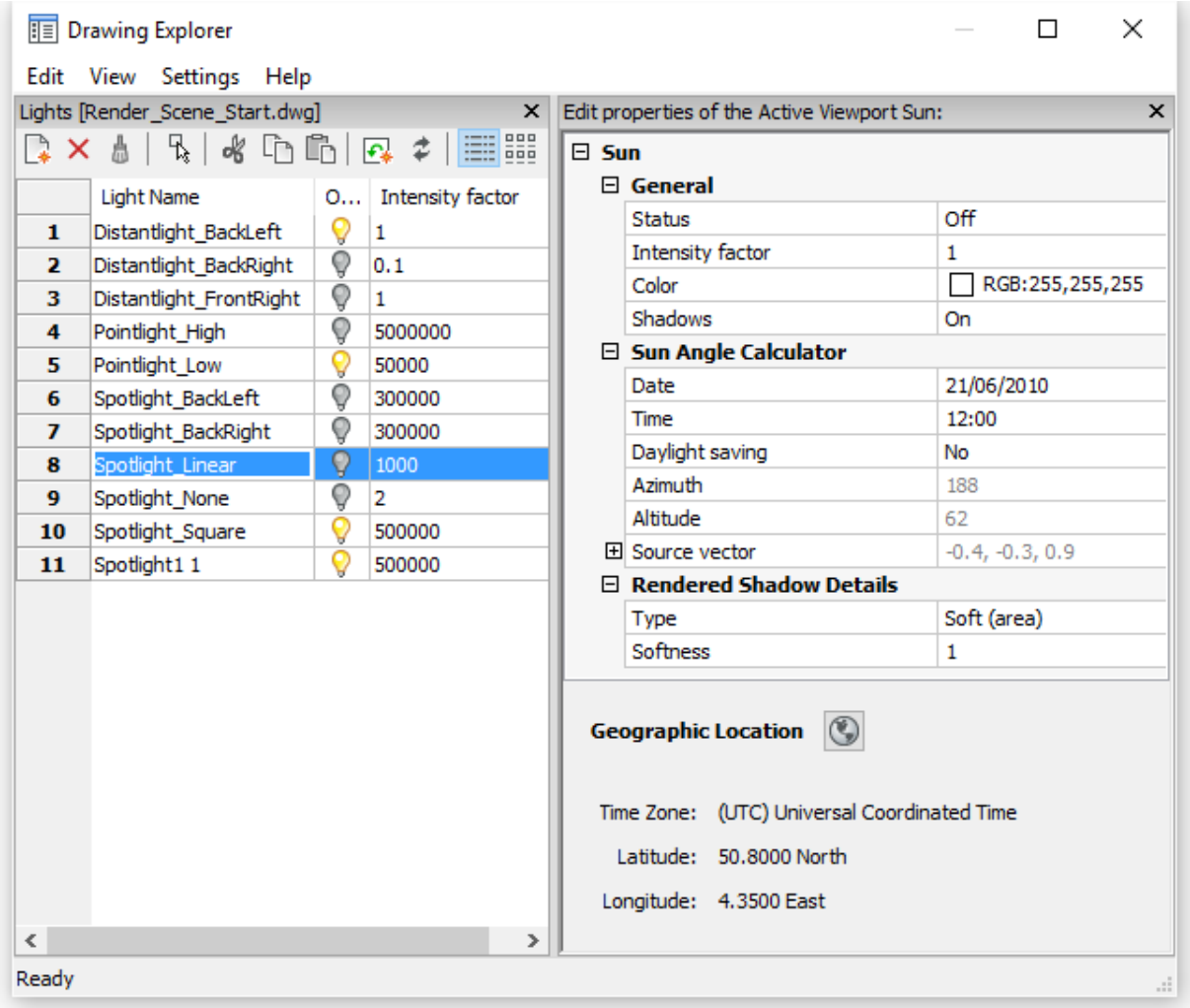
If the falloff angle and the hotspot angle are almost equal the edge of light cone is rather sharp (left).

The greater the difference between both angles, the softer the edge of the light cone (right).

Open the Light List

Do one of the following:

Click the *Light List...* button ( ) on the *Lights* toolbar.
Choose *Drawing Explorer > Lights* in the *Tools* menu.
Choose *Light List..* in the *View | Rendering | Lights* menu
Type *lightlist* or *LL* in the command bar, then press Enter.



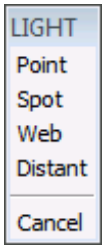
Icon	Tool name	Description
	New light	Closes the *Drawing Explorer - Lights* dialog, then guides you to the creation of a new light in the drawing.
	Delete light	Deletes the selected light.
	Select in drawing	Closes the *Drawing Explorer - Lights* dialog and selects the light's glyph in the drawing. The properties of the light display in the Properties Bar.
	Light ON	Click to switch light off.
	Light OFF	Click to switch light on.

To Define a Light

Type *light* in the command bar, then press Enter.

You are prompted: Enter light type [Point/Spot/Web/Distant] <Point>:

A prompt menu displays:



Choose a light type in the prompt menu.

Click the tool button of the light type you want to create on the *Lights* toolbar:

New spot light

New point light

New distant light

New web light

Do one of the following:

Specify the source position in the drawing.

Type the coordinates of the source position in the command bar and press Enter.

Press Enter to accept the default coordinates.

You are prompted: Specify target position <0,0,-10>:

Do one of the following:

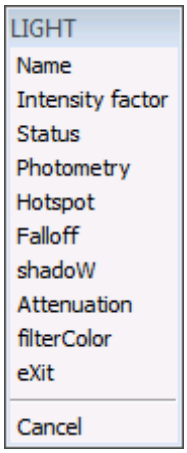
Specify the target position in the drawing.

Type the coordinates of the target position in the command bar and press Enter.

Press Enter to accept the default coordinates.

You are prompted: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/shadoW/Attenuation/filterColor/<eXit>:

A prompt menu displays:



Do one of the following:

Select *Name* in the prompt box.

Type *N* in the command bar and press Enter.

You are prompted: Enter light name <Spotlight1>:

Do one of the following:

Type a name in the command bar and press Enter.

Press Enter to accept the default name.

You are prompted: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/shadoW/Attenuation/filterColor/<eXit>:

Do one of the following:

Click a light property in the prompt menu.

Type the capitalized letter of a light property in the command bar and press Enter.

You are prompted to define the selected property in the command bar.

Do one of the following to conclude the creation of the light:

Choose Exit in the prompt menu.

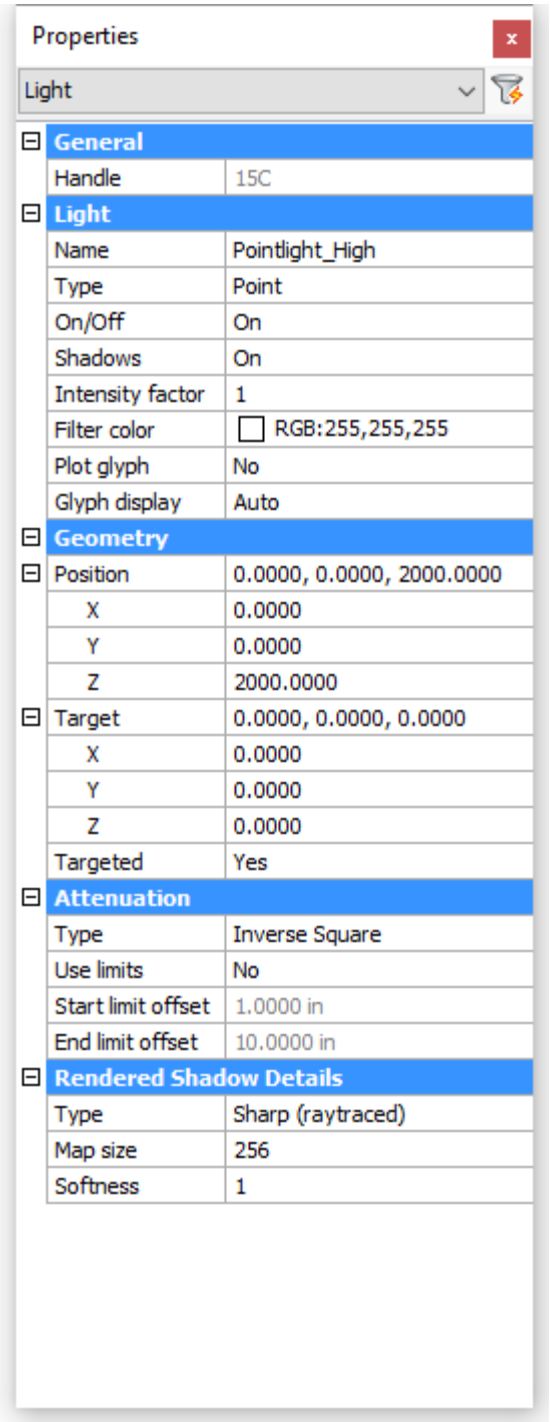
Press Enter.

NOTE It is not possible to define a web light if the *LIGHTINGUNITS* system variable is zero (No lighting units).

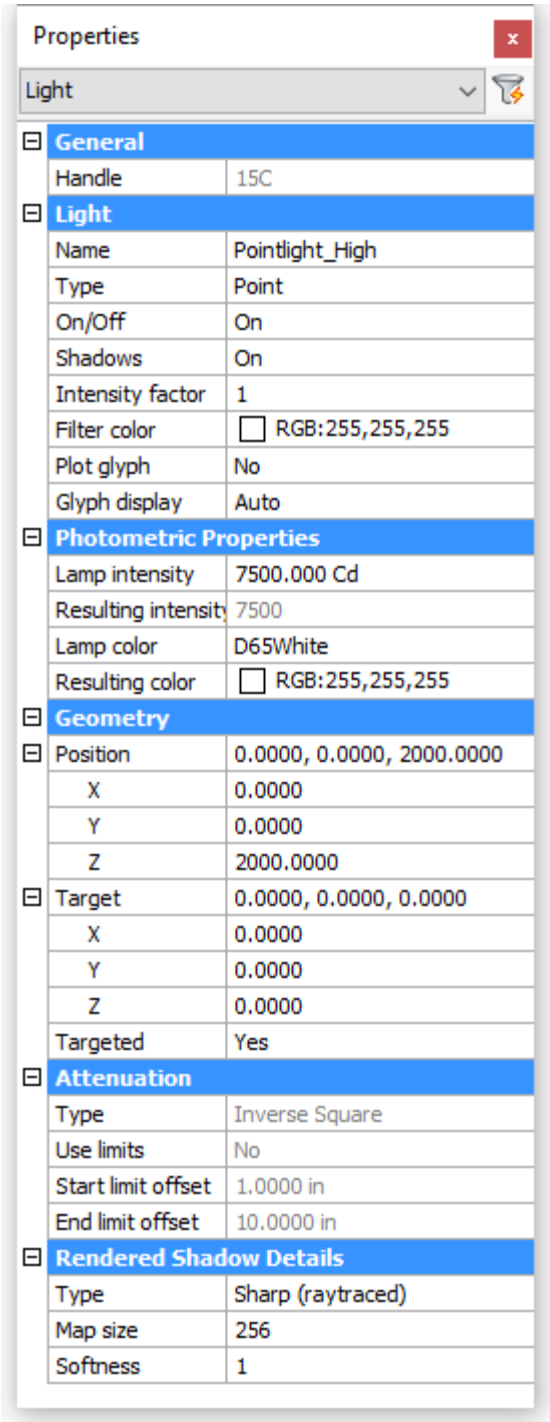
To Edit a Light

Select the light in the drawing.
If the Properties bar is not open yet, double click the light.

The properties of the selected light display in the Properties Bar:



Generic light properties



Photometric light properties

To edit a property:

Expand the settings group if needed.

Select the property

Edit the selected property.

To Define the Sun Properties

Do one of the following:

Click the *Sun Properties...* tool button ( ) on the *Render* toolbar.

Type *sunproperties* or *sun* in the command bar, then press Enter.

The *Drawing Explorer - Lights* dialog displays, showing the sun properties in the *Editor* pane.

Sun	
General	
Status	Off
Intensity factor	1
Color	RGB:157,143,106
Shadows	On
Sun Angle Calculator	
Date	09/21/2010
Time	15:00
Daylight Saving	No
Azimuth	223.3884
Altitude	30.6176
Source vector	-0.7, -0.5, 0.5
Rendered Shadow Details	
Type	Sharp
Softness	1

To edit a property:

Expand the settings group if needed.

Select the property

Edit the selected property.

NOTE When LIGHTINGUNITS = 1 or 2, the *Color* property of the sun cannot be edited.

To Define the Geographic Location

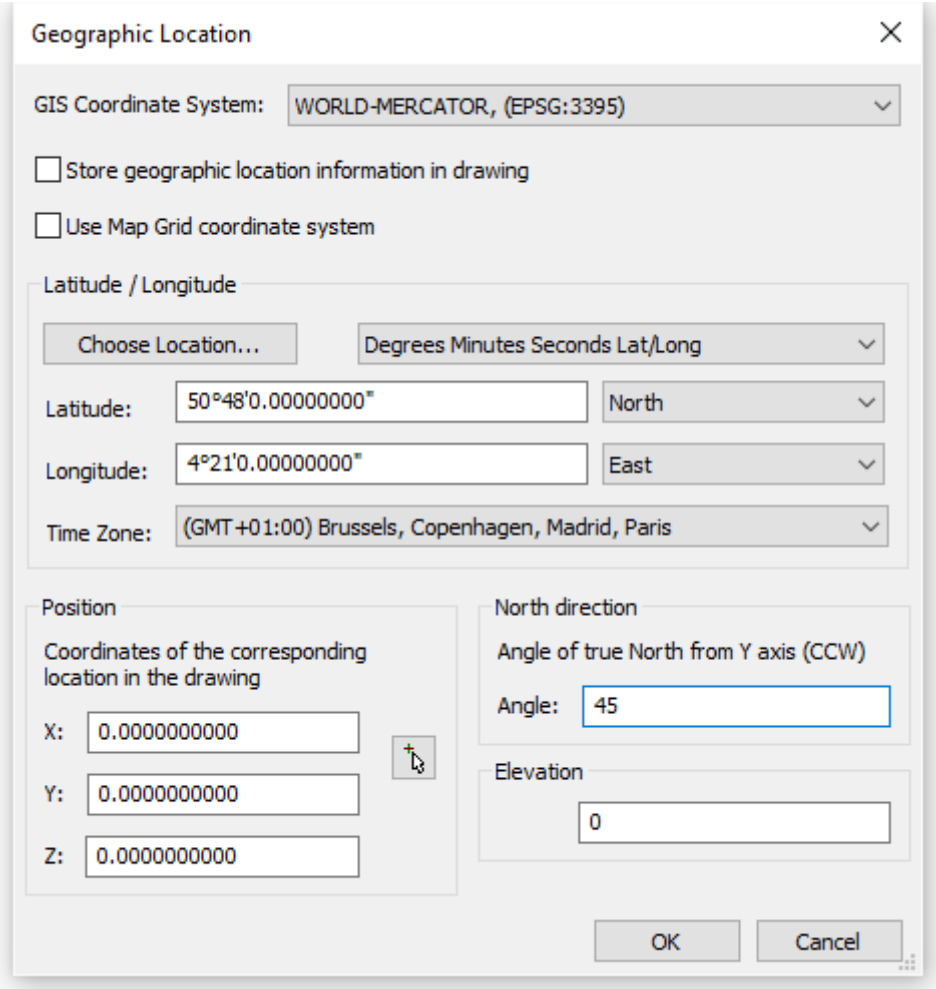
Do one of the following:

Click the *Geographic location...* tool button ( ) on the *Render* toolbar.

Click the *Geographic location...* tool button ( ) on *Editor* pane of the *Drawing Explorer - Lights* dialog.

Type *geographiclocation* or *geo* in the command bar, then press Enter.

The *Geographic Location* dialog displays:



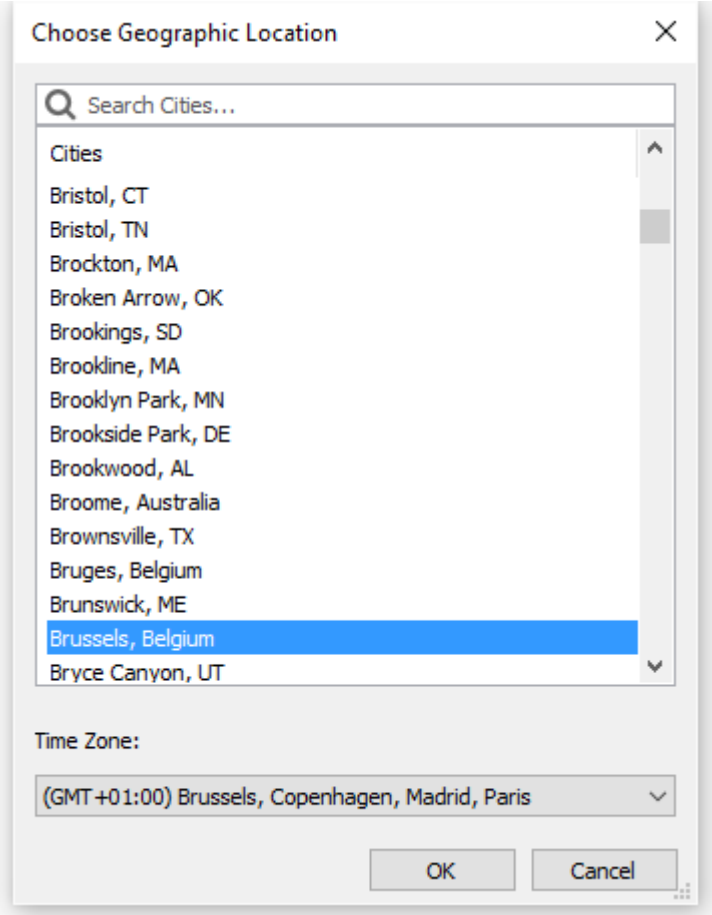
(option) Check the *Store geographic location information in drawing* option.

(option) Check the *Use Map Grid coordinate system* option.

Do one of the following:

Type the latitude and longitude of the location in the *Latitude* and *Longitude* fields and select the appropriate hemisphere for each setting.

Click the *Choose location...* button, then select a city and timezone and click *the* OK button.



(option) If you did not select a time zone in the previous step, choose a time zone in the *Time Zones* list.

(option) Specify the *Coordinates of the corresponding location in the drawing*.

Define the direction of the north.

Click the *OK* button.

Digitizing Tablet

The Tablet command configures and calibrates a digitizing tablet and toggles tablet mode.

This command requires that Wintab32.dll from Wacom Technology, Corp. be installed on the computer. Download Wintab32.dll from the internet, then place the file in your system directory: C:\Windows\System32.

A digitizing tablet, also called a graphics tablet, a graphics pad or drawing tablet, is a computer input device that allows you to hand-draw images and graphics, similar to the way one draws images with a pencil and paper. In a CAD program a digitizing tablet can be used to trace paper drawings into a drawing file or to launch drawing commands from a digitizing tablet overlay.

Using a digitizing tablet in BricsCAD

In order to prepare BricsCAD for using a digitizing tablet do the following:

Install the appropriate driver for your tablet.

Make sure Wintab32.dll exists in C:\Windows\System32.

Download the tablet overlay and CUI files.

Print the BricsCAD tablet overlay.

Load the overlay CUI file.

Start BricsCAD, then initialize the tablet.

Configure the tablet: part of the tablet surface is used as the menu area, another part as the screen pointing area.

Calibrate the tablet: a calibrated tablet can be used to trace the geometry of a paper drawing or image into a drawing.

To download the tablet overlay and CUI files

Download the file Tablet.zip.

The ZIP file contains the following files:

overlay.png: an image of the tablet overlay

overlay(A3).pdf: to print the tablet overlay on a A3 paper size.

overlay(cm).dwg: a DWG in which the *overlay.png* is attached as an image.
The drawing contains two layouts to print the overlay using decimal units.

overlay(inch).dwg: a DWG in which the *overlay.png* is attached as an image.
The drawing contains two layouts to print the overlay using imperial units.

tablet.cui: a partial cui file to be used with a tablet.

tablet(acadLike).cui: an AutoCAD-like partial cui file to be used with a tablet.

To load the overlay CUI file

Extract the overlay CUI files to the *Support* folder of the *roamable root* folder (see BricsCAD User Files).
E.g. C:\Users\[user name]\AppData\Roaming\Bricsys\BricsCAD\V[version nummer]\en_US\Support\.

Load *Tablet.cui* as a partial CUI (see To load a partial CUI file).
The next time you start BricsCAD, *Tablet.cui* will be loaded automatically.

NOTES Please make sure the *overlay.png* image file is extracted to the same folder as the DWG's.

It might be necessary to repeat the above procedure after upgrading BricsCAD.

To switch the tablet ON / OFF

Do one of the following:

Click the *TABLET* field in the Status Bar.

Press the *F4* function key to toggle the tablet mode.

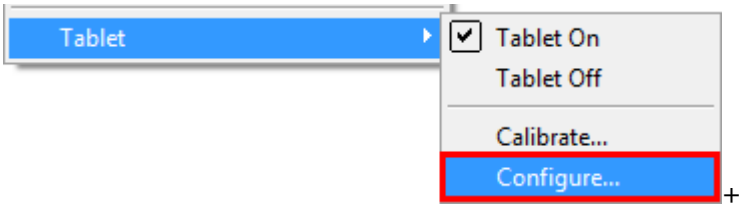
Choose *Tablet On* or *Tablet Off* in the *Settings* menu.

Type *tablet* in the command bar, press Enter, then type *ON* or *OFF* ;
or
click *ON* or *OFF* in the prompt box.

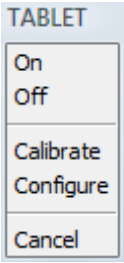
To configure the tablet

Do one of the following:

Choose *Tablet* in the *Settings* menu, then choose *Configure...* in the flyout menu.



Type tablet in the command bar and press Enter.
You are prompted: Tablet: ON/OFF/CALibrate/ConFiGure/<On>:
a prompt menu displays:

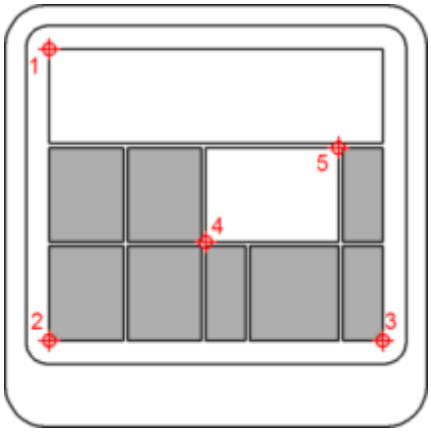


Type *cfg* in the command bar and press Enter or choose *Configure* in the prompt menu.

You are prompted: Do you want to align the tablet overlay? Yes/No/<No>:

Type *Y* in the command bar and press Enter or choose *Yes* in the prompt menu.
You are prompted: Digitize upper left corner of the overlay:

Click on the mark in the upper left corner (1) of the digitizing tablet overlay.



You are prompted: Digitize the lower left corner of the overlay.

Click on the mark in the lower left corner (2) of the digitizing tablet overlay.
You are prompted: Digitize the lower right corner of the overlay.

Click on the mark in the lower right corner (3) of the digitizing tablet overlay.
You are prompted: Digitize the lower left corner of the screen pointing area:

Click on the mark at position (4) on the of the digitizing tablet overlay.
You are prompted: Digitize the upper right corner of the screen pointing area:

Click on the mark at position (5) on the of the digitizing tablet overlay.
You are prompted: Tablet configured.

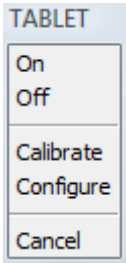
To calibrate the tablet

To calibrate the digitizing tablet you must specify at least two points. The more additional points you specify, the more accurate the digitizing process. Additional points are particularly recommended when digitizing a non-orthogonal image, such as an aerial photograph.

Do one of the following:

Choose *Tablet* in the *Settings* menu, then choose *Calibrate...* in the flyout menu.

Type tablet in the command bar and press Enter.
You are prompted: Tablet: ON/OFF/CALibrate/ConFiGure/<On>:
a prompt menu displays:



Type *cal* in the command bar and press Enter or choose *Calibrate* in the prompt menu.

You are prompted: Digitize point #1:

Click a point on the digitizing tablet .
You are prompted: Enter coordinates for point #1:

Do one of the following:

Click a point in the BricsCAD drawing window.

Enter the coordinates (x,y) in the command bar.

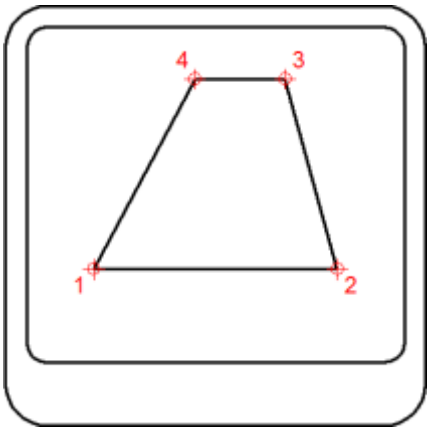
You are prompted: Digitize point #2:

Repeat step 3 to specify the following calibration point.
You are prompted: Digitize point #3 (or ENTER to end):

Do one of the following:

Repeat step 4 to specify an additional calibration point.
(You can specify up to 10 calibration points).

Press Enter to conclude the calibration process.
You are prompted:
if 2 calibration points are specified: Tablet calibrated.
if 3 or more than 4 calibration points are specified: Select transformation type Orthogonal/Affine:
if 4 calibration points are specified: Select transformation type Orthogonal/Affine/Projective:



Select the transformation type.
Transformation refers to the calculation of the points on the screen that correspond to points you digitize on the tablet.

Orthogonal (left): To be used with accurate paper drawings. Orthogonal transformation maintains all angles and preserves relative distances. If only two calibration points are specified orthogonal transformation is generated automatically.

Affine (middle): Maintains parallel lines, but not necessarily the angles between intersecting lines.

Projective (right): Does not maintain parallel lines, nor angles.

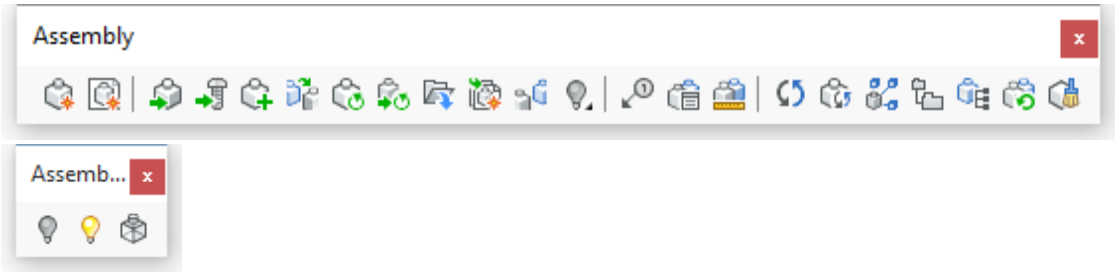
Mechanical

Commands and Toolbars

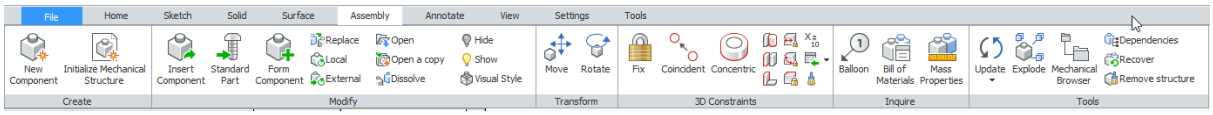
Icon	Command	Description
	BmNew	Creates a new mechanical component as a new drawing file
	BmMech	Convert the current drawing into a mechanical main component.
	BmInsert	Inserts an existing mechanical component into the current drawing.
	BmHardware	Inserts a standard hardware part as a mechanical component in the current drawing.
	BmForm	Creates a new mechanical component and inserts it into the current drawing.
	BmReplace	Replaces a component insert.
	BmLocalize	Switches external components to local components.
	BmExternalize	Switches local components to external components.
	BmOpen	Opens the source drawing of an external mechanical component.
	BmOpenCopy	Opens a copy of a component insert as a new drawing.
	BmDissolve	Dissolves a mechanical component inserted into the current drawing.
	BmHide	Hides a mechanical component or subcomponent in the current drawing.
	BmShow	Shows a previously hidden mechanical component or subcomponent in the current drawing.
	BmVStyle	Applies a visual style to a mechanical component
	BmBom	Inserts the Bill-of-Materials (BOM) table in the current drawing
	BmMassProp	Computes mass properties for the current model using densities assigned to its components.
	BmUpdate	Reloads all referenced components from external files and updates BOM tables.
	BmUpdate	Reloads all selected referenced components from external files and updates BOM tables.
	BmExplode	Creates an exploded representation for an assembly by moving components to make them all visible, stores it in a new block and inserts the block in the Model Space.
	MechanicalBrowserOpen	Toggles the display of the *Mechanical Browser* window.
	BmDependencies	Lists all files, containing component definitions inserted in the assembly, in the command window.
	BmRecover	Recovers a broken mechanical structure
	BmUnmech	Converts the current mechanical component into a plain drawing.
	BmXConvert	Converts X-Hardware solids in the current drawing to mechanical components.

Tools to work with mechanical components are available on the *Assembly* toolbar and its *Assembly Visualization* flyout toolbar, the *Assembly* menu and on the *Assembly* tab of the *Mechanical* workspace Ribbon:

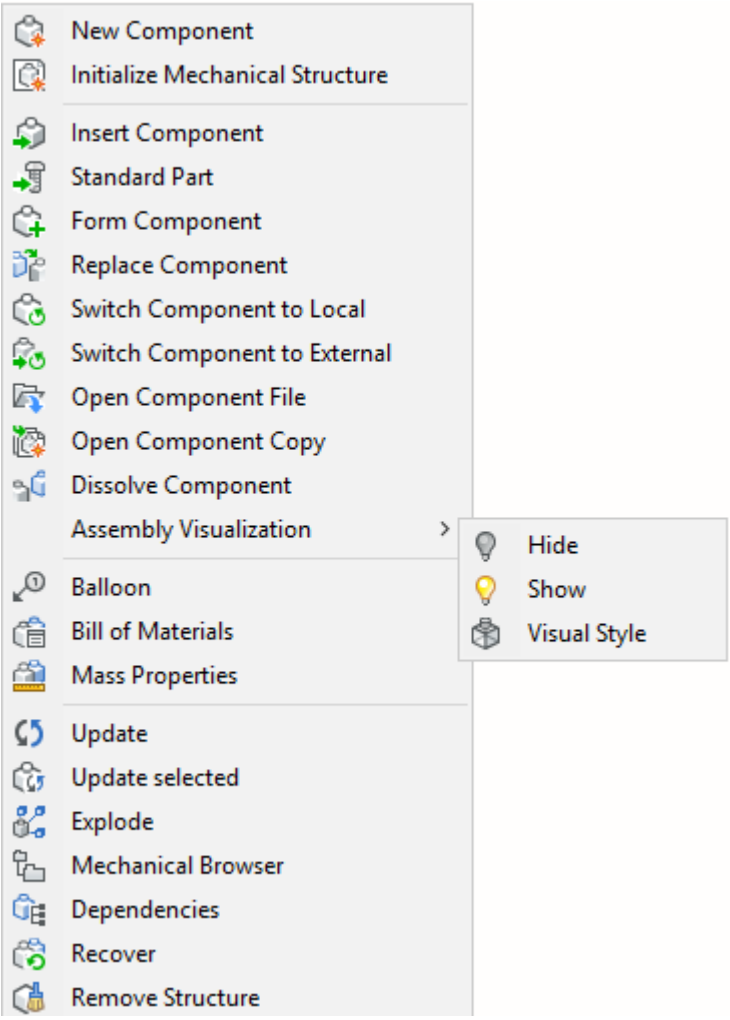
Toolbars



Ribbon



Menu



Components and Inserts

Definitions

Assembly modeling consists in using mechanical components to group geometric entities in hierarchies, representing the structure of a designed product. Assembly modeling is commonly used in Mechanical Computer-Aided Design (MCAD) applications, because it simplifies the design of complex products containing thousands and millions of geometric entities.

A mechanical component is a named group of entities. A mechanical component can be any .dwg file or a standard hardware part. Users can define their own components. A component can be inserted once or several times into one or more other components. If the component is modified, all instances of the component will reflect these changes automatically. Components cannot be self-referencing.

A component insert is a named entity, of which the default name is composed of the name of the component and the serial number of the insert.

An assembly is a mechanical component that contains other components.

An assembly with inserted components (which in turn can be assemblies with other inserted components) forms a tree that is called a mechanical structure of the drawing. This tree is displayed in the Mechanical Browser.

Users can create mechanical components from scratch or add them to an existing .dwg

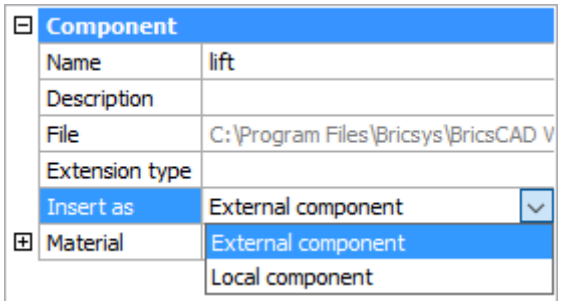
External and local components

Mechanical components can be either local or external. A local component is stored in the .dwg file with the assembly. An external component is stored in a separate .dwg file. All inserts of external components are updated automatically when the corresponding .dwg file is modified. If you use external components in your assembly, always remember to transfer all related files when you want to share your model. The ETRANSMIT command creates a package of a drawing file and all its dependencies, such as: external references, images, font files, plot configuration files, plot style tables and font map files.

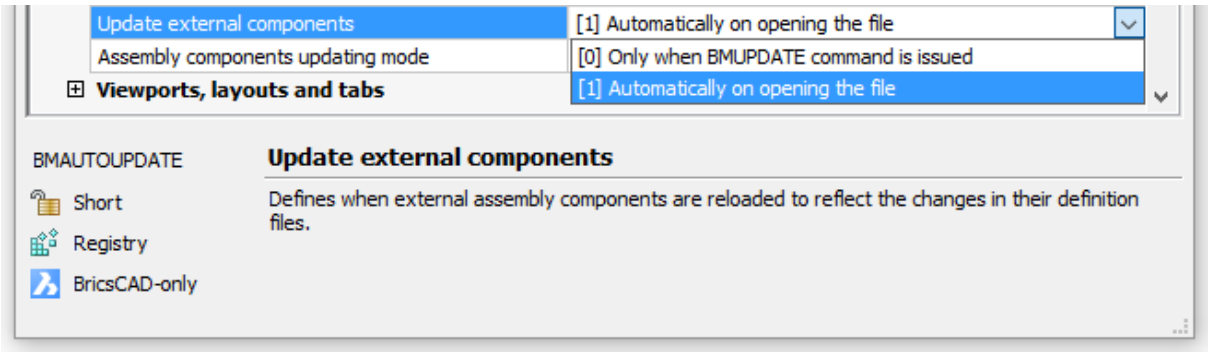
You can always convert a local component to an external one and vice versa.

To edit a local component it needs to be converted to an external component using the BMEXTERNALIZE command. Alternatively you can use the BMOPENCOPY command to open a copy of a component, edit and save it, and then use the *BmReplace* command to replace the original with the copy.

You can choose the default insertion type for your component: *Local* or *External*. To change the type select the root node in the Mechanical Browser and select *Local* or *External* from *Insert as* properties:



Depending on the value of BMAUTOUPDATE, inserts of external components are updated automatically on opening the assembly document or manually with the BmUpdate command. You can change this value in the Settings dialog:



Converting Components

Do one of the following:

Click the *Switch Component to External* ( ) or *Switch Component to Local* tool on the *Assembly* ribbon tab. The tools are also available on the *Assembly* toolbar and in the *Assembly* menu.

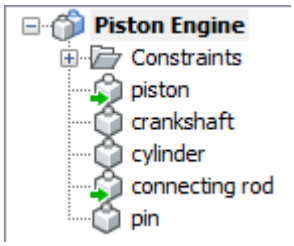
Right click on the component insert in the Mechanical Browser and select *Switch to External* or *Switch to Local* in the context menu.

Hover your mouse cursor over a component insert in a drawing window and select *Switch to External* ( ) or *Switch to Local* ( ) in the *Assembly Modeling* command group in the Quad.

Type *BmExternalize* or *BmLocalize* in the command bar.

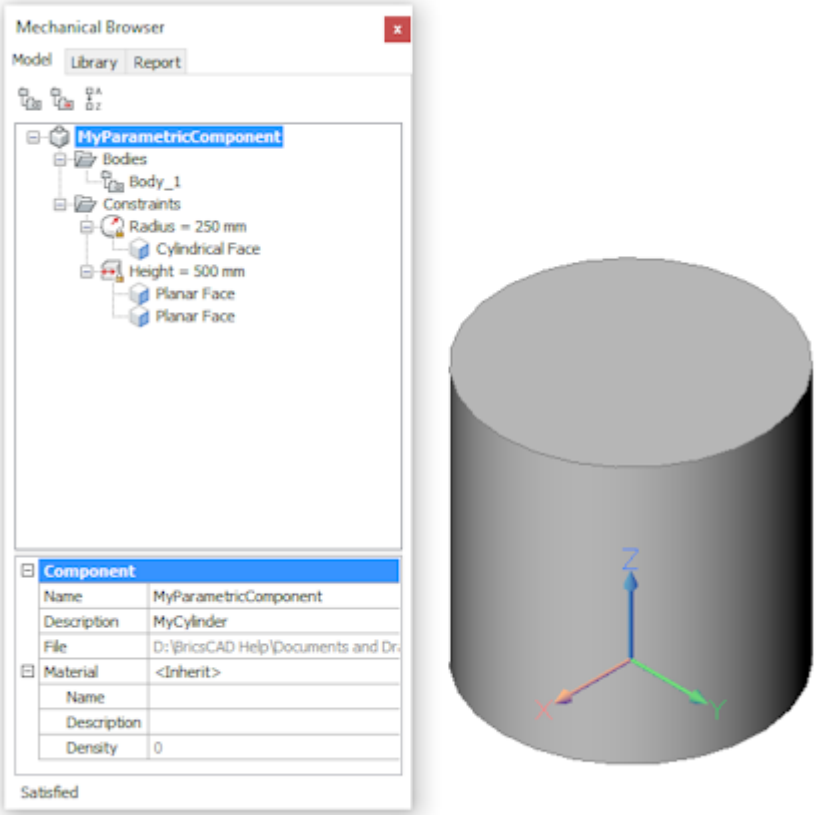
When you convert an external component to local, the .dwg file with the component definition is not deleted, but its future changes will not affect your assembly.

You can always distinguish between inserts of local components ( ) and inserts of external components ( ) by their icons in the Mechanical Browser:



Parametric Components

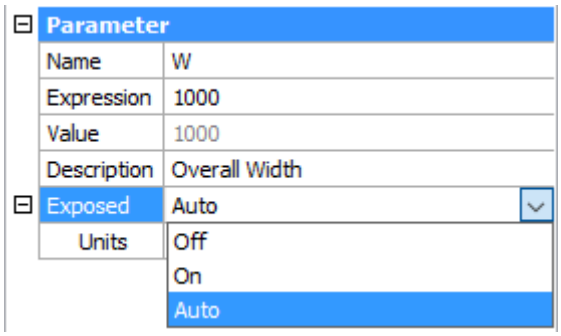
BricsCAD allows users to make parametric changes of inserted components at assembly level. A component is considered as parametric if it contains at least one 2D or 3D dimensional constraint.



NOTES BricsCAD automatically maintains associativity between the definition of a parametric component and its inserts. So you can open the parametric component for editing, modify its geometry, save it, and then execute the BmUpdate command to update the mechanical structure of your assembly see the result.

If you need to go back to the default value of a component' parameter, just edit the *Expression* field and delete its content.

You can control whether a particular component parameter will be available for editing in a drawing containing the inserts of this components (at the assembly level). To do that, select the corresponding parameter in the Mechanical Browser and change the value of the *Exposed* field in the properties grid of the parameter:



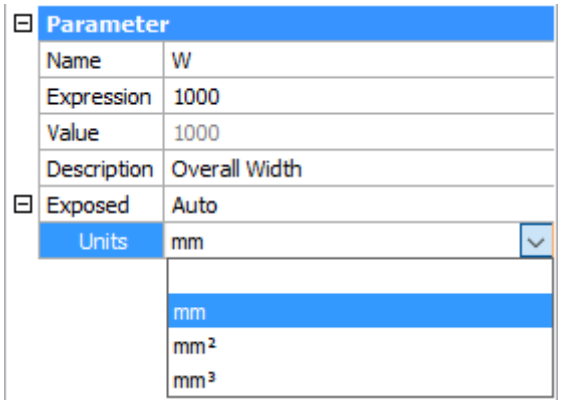
The options are:

Off:Hides the parameters at the assembly O

On: Makes the parameter visible at the assembly level.

Auto: Hides all parameters which values are controlled by expressions, and exposes all other parameters.

You can associate units with a parameter. If you assign a value, directly or via an expression, to a component parameter at the assembly level, it is automatically converted from drawing units of the assembly drawing to the units of the parameter. Values assigned to unitless parameters are not converted.

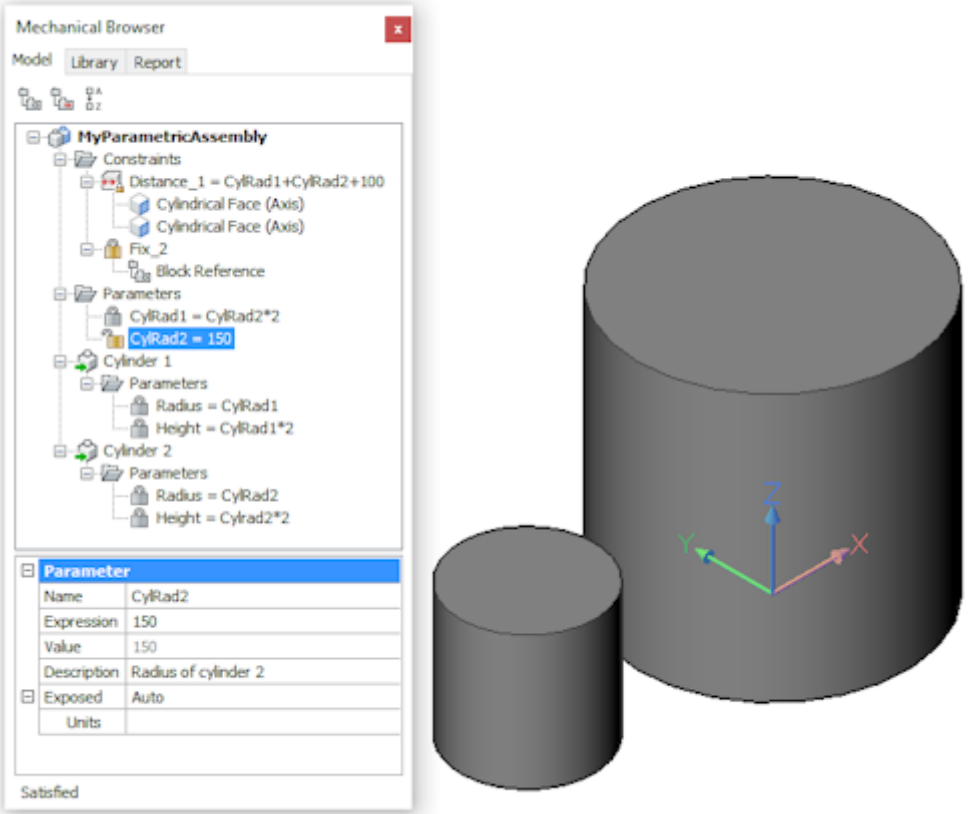


By default all parameters are unitless except parameters of dimensional constraints and parameters, which are used as expressions for dimensional constraints. BricsCAD automatically assigns drawing units to such parameters.

To edit the parameters of a parametric component insert

When a parametric component is inserted in a model, you can edit the parameters either in the Mechanical Browser or in the Properties Bar.

The component insert immediately updates according to the new value of a parameter. Please note that this does not modify the definition of the parametric component, which is a separate drawing file. As a result, multiple inserts of the same parametric component can exist in the same model, with different values for each of their parameters:



To edit a parameter in the Mechanical Browser

Expand the *Parameters* node of the component.

Select the parameter.

In the *Parameter* section, type a new value in the *Expression* field .

Parameter	
Name	Radius
Current value	150 mm
Expression	CylRad2

To edit a parameter in the Properties Bar

Select the component in the drawing.

Expand the *Parameters* settings group.

Type a new value for one of the available parameters.

Parameters	
Name	Cylinder 2
File	MyParametricComponent.dwg
Radius	150 mm
Height	300 mm

NOTES In the Properties Bar always the current value displays. In case this value is controlled through an expression, the expression will be overwritten by the newly entered value.

If multiple parametric components are selected, the shared parameters can be edited simultaneously. The settings fields of parameters that currently have different values read *Varies*.

Parameters	
Name	*Varies*
Radius_1	30
Distance_2	*Varies*

Replacing Components

Component inserts can be replaced with another component using the BmReplace command. Only top-level inserts can be replaced.

Do one of the following:

Click *Replace Component* tool ( ) on the *Assembly* ribbon tab. The tool is available on the *Assembly* toolbar and in the *Assembly* menu.

Right click on the component insert in the Mechanical Browser and select *Replace&ldots;* or *Replace all inserts&ldots;* in the context menu

Hover over a component insert in a drawing window and select the *Replace* tool ( ) in the Quad menu.

Type *BmReplace* in the command bar

A *File open* dialog displays.

Select a .dwg file for the replacement component and press the *Open* button.
You are prompted: Replace file for all similar inserts [Yes/No] <Yes>:

Do one of the following:

Yes: Replace all inserts of the same name.

No: Replace the selected insert only.

NOTE Parameter values which are shared between the replacement component and the replaced component are copied automatically, which allows intelligent replacement of parametric components.

Components Blocks and Xrefs

Drawings with blocks and external references can be converted to mechanical assemblies: block references are converted to inserts of local components, and external references are converted to inserts of external components.

Do one of the following:

Click the *Initialize Mechanical Structure* tool ( ) in the ribbon tab. The tool is also available in the *Assembly* toolbar and in the *Assembly* menu.

Type *BmMech* in the command bar

You are prompted: Name for new mechanical component <test>:

Type a name for the mechanical component or press Enter to accept the current name or the default *Component_1* name.
You are prompted: Convert blocks and external references to mechanical components? [No/Yes] <Yes>:

Press Enter to convert blocks and external references to mechanical components.

Mechanical components can be converted back to blocks and external references.

Do one of the following:

Click The *Remove Structure* tool ( ) on Assembly ribbon tab. The tool is also available in the *Assembly* toolbar and in the *Assembly* menu.

Type *BmUnmech* in the command bar.

You are prompted:
Warning: this command removes all mechanical structure from the current document. Proceed? No/<Yes>:

Press Enter to:

Convert external components to External References (Xrefs).

Convert local components to blocks.

A feature is a physical constituent of a part, which has engineering significance. Examples of features are holes, ribs, slots, pockets, forms. Being applied to a 3D solid, a feature removes and/or adds some volumes to it. You can create your own features using BricsCAD component extension technology.

An extended component has BC_SUBTRACT and/or BC_UNITE layers. When you insert such a component on a face of any 3D solid using the BmInsert command, this target solid is modified as follows:

all 3D solids located on BC_SUBTRACT layer of the component are subtracted from the target 3D solid;

all 3D solids located on BC_UNITE layer of the component are united with the target 3D solid.

All faces created after these Boolean operations form a component based feature, which is updated when you change the component insert &endash; by moving it along a target 3d solid or by modifying its parameters (if the extended component was parametric). When the insert is deleted, the feature is deleted as well.

Some examples of parametric component-based features are included into your BricsCAD installation. You can use them as samples for creating your own features.

Assigning a Physical Material

A mechanical component may have an assigned physical material.

To assign a material:

Select the root component in the Mechanical Browser.

Select *Material* property and press the *Browse* button (...).

Component	
Name	support_final
Description	
File	C:\Program Files\Bricsys\E
Extension type	
Insert as	External component
Material	Aluminium ... x
Name	Aluminium
Description	
Density	2700.00 kg/m³

In the *Physical Materials* dialog select a material in the *In Project* or the the *In Library* list. If a material from the *In Library* list is chosen, it will be copied to the *In Project* database automatically.

Click the *OK* button to save changes and to assign the selected material to the component.

To remove the material from the component:

Select the root component in the Mechanical Browser.

Select *Material* property and press the *Clear* button (X)

If a mechanical component does not have an assigned material, it will inherit the material from its parental component. If the parental component also does not have an assigned material, the material of the closest component with an assigned material in the hierarchy will be used. It means that you can assign a material to several components having a common parent at once by assigning the material to their common parent component.

Inserting a component with a physical material

If you insert a mechanical component into a document, all materials used in this component will be automatically copied to the document material library. Only materials assigned to the components will be copied. However, if the current document already has a material with the same name, this material will be used instead. It means that the properties of the material defined in the document library will be used rather than properties of the material from the inserted component.

NOTE Materials which are copied to the document material library will not be erased in Undo operations.

Organizing Geometry

There are two main ways to design mechanical products: top-down and bottom-up approaches. It is also possible to add mechanical structure to existing geometry (designed in BricsCAD or imported from another source).

BmNew command is called to create a new document for a new mechanical component.

Bottom-Up Design

With bottom-up approach users start to design a product with detailing its low-level component. These components are then grouped into higher-level components. The process is continued till the very top component (corresponding to the product itself) will be designed.

The main command for bottom-up design is **BmInsert**, which is used to insert an existing component into the current drawing.

Top-Down Design

With top-down approach users start to design a product with defining (but not detailing) its first-level components. Each component is then refined in more details by defining its sub-components. The process is continued till all low-level components will be defined and detailed.

BmForm command allows to create a new subcomponent for an existing component.

Adding Mechanical Structure to Existing Geometry

BricsCAD users can add component hierarchy to any existing drawing created in BricsCAD or imported from a different CAD format.

BmForm converts a selected set of geometric entities to a subcomponent of the current drawing.

Changing Mechanical Structure

During design process users may realize than a particular component should be moved up or down in the component hierarchy.

BmForm command forms a new subcomponent for the current drawing that will contain a selected set of component.

BmDissolve deletes a component and moves all its subcomponents to the level of the dissolved component.

Using Standard Hardware

Standard hardware (screws, nuts, washers, bearings, etc.) can be considered as a mechanical component.

BricsCAD supplies a huge library of standard hardware (more than 30,000 details of different size). Users can significantly increase the design performance by using standard parts.

BmHardware inserts a standard part into the existing component.

Assembly Constraints

Assembly constraints are ordinary **3D Constraints**, which link faces and edges of bodies that represent the components of an assembly. 3D Constraints can link not only top-level components, but also any their subcomponents.

Component Visibility

A particular component of a complex product can be invisible due to other components surrounding it. For example, a piston is invisible inside a cylinder block. To make such component visible users may hide some other components.

Users can hide any subcomponent inserted into the current drawing or any its subcomponent on any level of nesting.

There is also a possibility to make a component transparent to visualize it in wireframe model.

BmHide hides a particular subcomponent.

BmShow unhides a hidden subcomponent.

Mechanical Browser

TUTORIALS

Introduction to Standard Parts
Working with Standard Parts

Commands: MECHANICALBROWSEROPEN, MECHANICALBROWSERCLOSE, REPORTPANELOPEN, REPORTPANELCLOSE, STANDARDPARTSPANELOPEN, STANDARDPARTSPANELCLOSE

The Mechanical Browser allows navigation on the hierarchy of mechanical components for the current drawing and calling different commands for the inserts. 2D and 3D Constraints also display in the Mechanical Browser.

With the mechanical browser you can:

click a component to highlight its geometric entities in the drawing window

click a component to change its name

right click a component to run a command for it

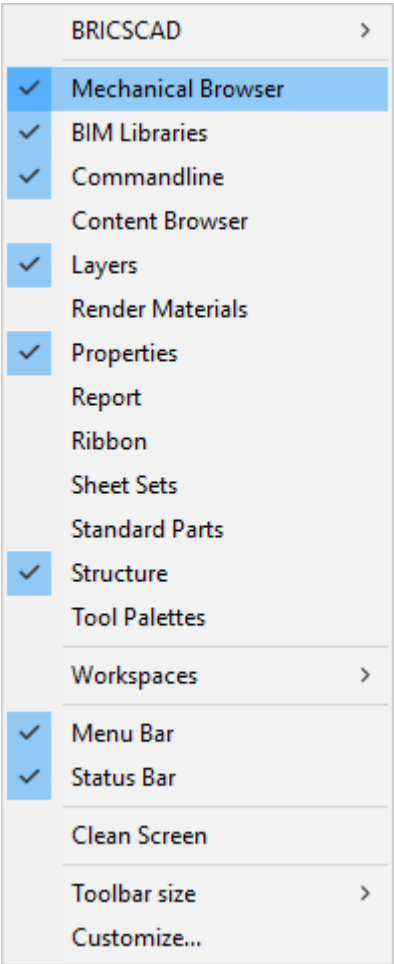
edit 2D and 3D constraints

To open the Mechanical Browser:

Do one of the following

Choose *Mechanical Browser* in the *Assembly* menu.

Place the cursor on a toolbar, then right click and choose *Mechanical Browser* in the context menu.



Bill of Materials

Command: BMBOM

Engineering Bill of Materials (BOM) is a list of unique mechanical components used in a drawing. The list is represented as a BricsCAD table entity (see Tables), and can be placed at any location in the drawing area. Each entity has its serial number (starting with 1), name, and the number of occurrences.

Bill of materials NV-ENT-Railway_inspection_car		
No	Part	Quantity
1	BBRG-00x40 MM	4
2	ISO 1035-FLx1.5 X 15x10	4
3	ISO 2009xM8 X 1x10	8
4	ISO 4014xM10 X 1.25x100	2
5	ISO 4014xM16 X 2x100	8
6	ISO 4034xM10 X 1.25	2
7	ISO 4034xM16 X 2	8
8	NV-AX-005-01-axis_2	1
9	NV-AX-010-01-axis_1	1
10	NV-BRK-005-01-bracket	4
11	NV-RD-005-01-rod_1	1
12	NV-RD-005-01-rod_1	1
13	NV-SBS-005-01-platform	1
14	NV-SI-005-01-bench	2
15	NV-SU-005-01-bearing	1
16	NV-WH-005-01-wheel	4

There are two options for BOM: listing top-level components only or bottom-level (parts) only.

By default the BOM table contains only three columns: number of a component, its name, and the number of inserts of this component in the assembly. The Configure option of the BmBom command allows to add more columns: Description, Density, Volume, Mass, Material, Thickness (for sheet metal parts) and Parameters (for inserts of parametric components).

Multiple inserts of the same parametric component are grouped in separate BOM rows depending on the values of their parameters.

Bill of Materials Bookshelf					
No.	Component	Quantity	L	T	W
1	Frame Board	2	360.0	20.0	200.0
2	Frame Board	2	700.0	20.0	200.0
3	Shelf Board	2	266.7	2.5	200.0
4	Shelf Board	2	466.7	2.5	200.0

3D Comparison

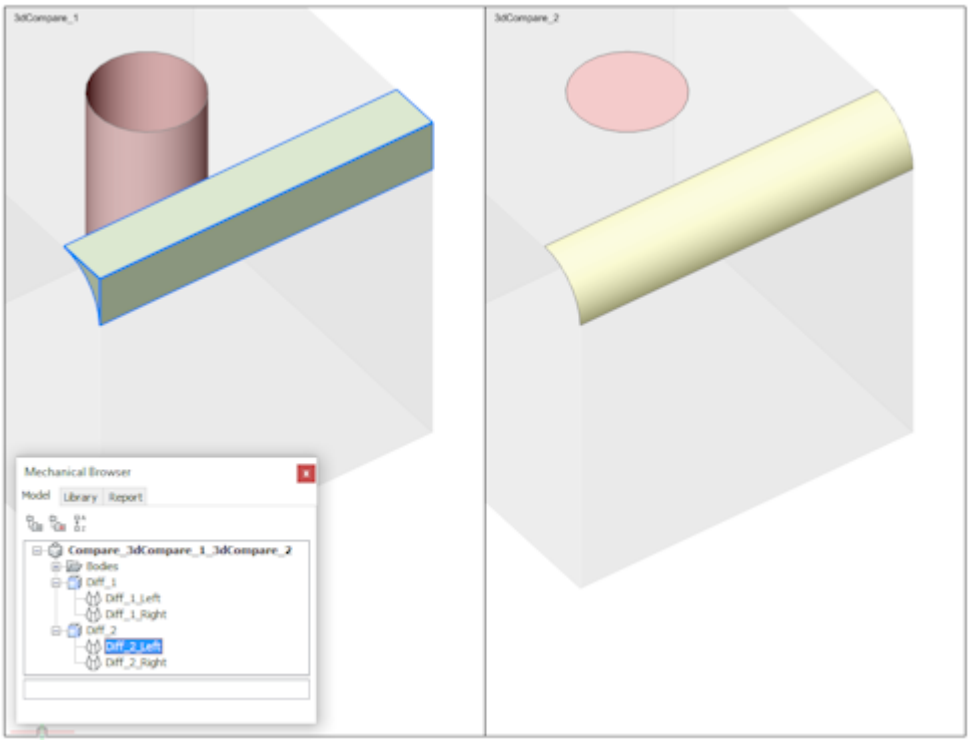
Command: 3DCOMPARE

You can compare two 3D models, i.e. 3D solids and surfaces stored in two .dwg files, including ones imported in BricsCAD from other file formats. The tool efficiently processes large models and reports differences in the Mechanical Browser. Several visualization modes are supported.

Definitions

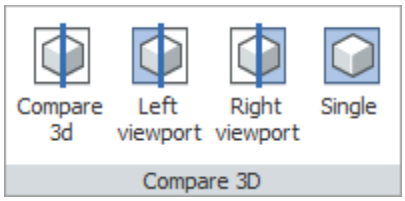
A 3D model is a .dwg file with one or more 3D solids or surfaces. When comparing two 3D models, BricsCAD first tries to automatically align them in 3D space to obtain the maximal similarity. Then all faces of these 3D solids and surfaces are compared geometrically with a given tolerance. If for every face in the first file an equal face in the second file exists and vice versa, such models are considered as equal. However, if for a particular face in one file there is no equal face found in another file, or if only part of this face is equal to a part of a face in the other file, then a difference is detected. Therefore a difference is a full face or a part of a face of a 3D solid or a surface, for which there are no equal parts of full faces found in the other file. Two or more differences are grouped together if they are geometrically adjacent to each other.

A difference node is created in Mechanical Browser for every group of adjacent differences. If you expand this node, you can see the particular differences it consists of. When you select the difference in the Mechanical Browser the corresponding feature highlights in the drawing.



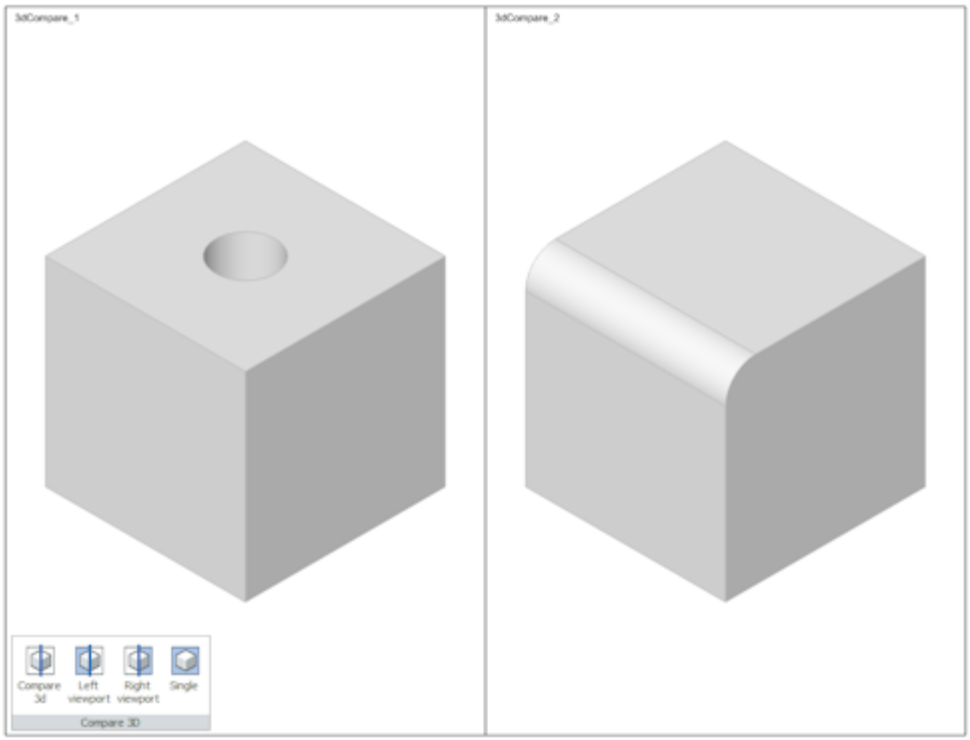
Toolbars

The 3D Comparison procedure can be accessed using the tools on the *Compare 3D* ribbon panel on the *Tools* ribbon tab



Visualization

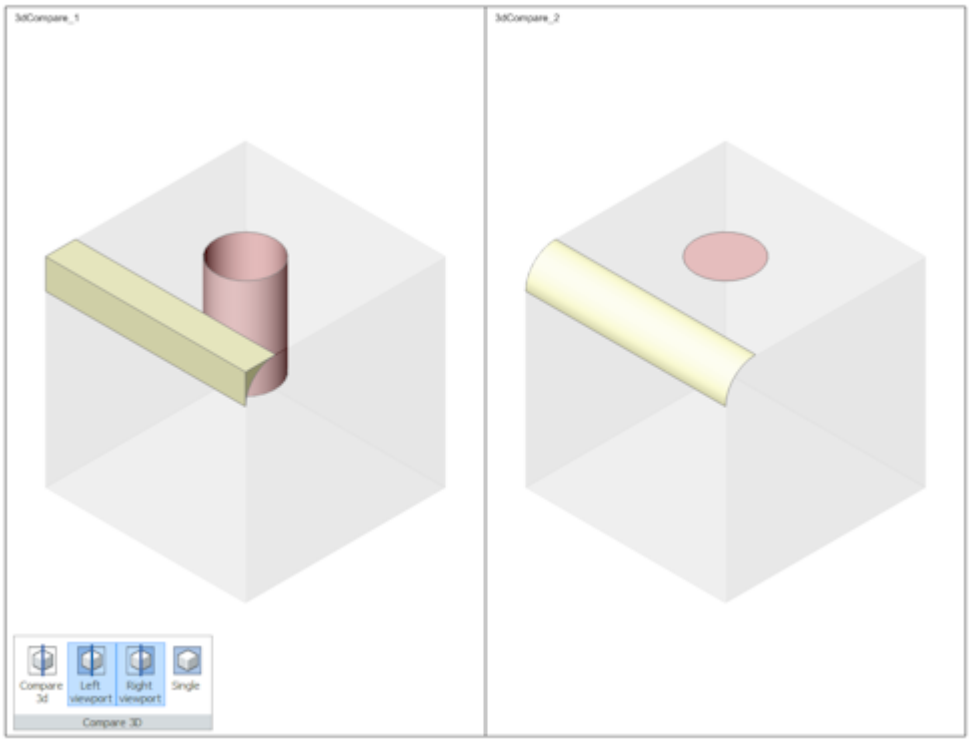
The 3DCOMPARE command creates a new .dwg document named Compare_<File_1>_<File_2>.dwg. The compared models are attached as external references to this drawing, A special layout named *Comparison*. in which two viewports exist, one for each of the two models.



The display is controlled through the 3DCOMPAREMODE system variable. If the value of 3DCOMPAREMODE is different from 0 (zero), the corresponding model becomes semi-transparent, while the differences are displayed using different colors (each difference is painted in its individual color). Using the Mechanical Browser context menu you can zoom in on a particular difference.

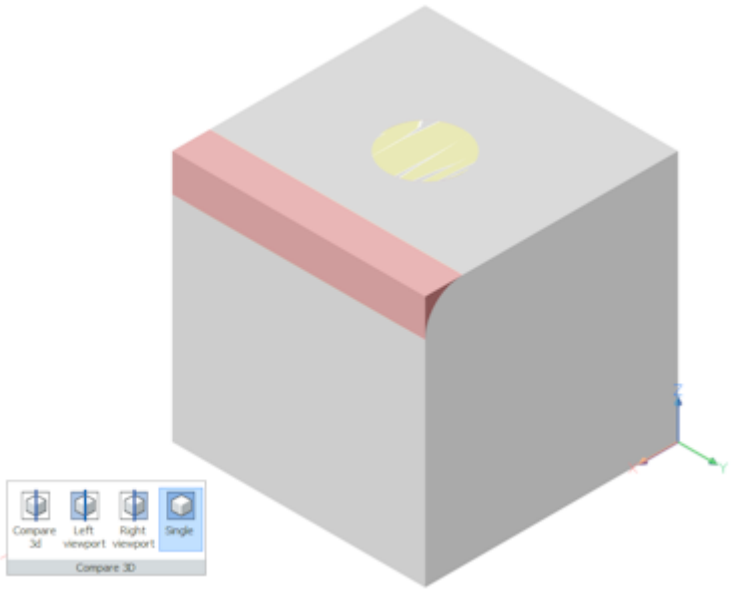
To display the differences, do one of the following:

Click the *Left Viewport* ( ) and/or the *Right Viewport* ( ) tool button.



Double click in one of the viewports, then use the mouse/keyboard shortcuts to manipulate the display (e.g. zoom in/out or rotate). The display in the other viewport is synchronized automatically, allowing to compare the same part in both models in detail.

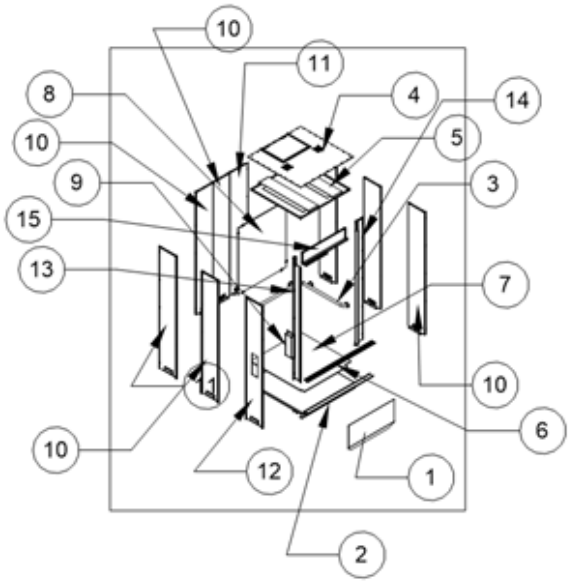
Click the *Single* ( ) tool button to see the differences in model space.



Exploded Representations

The exploded representation functionality allows to create associative exploded representations of assemblies without modifying the assembly itself. Exploded representations are stored in dedicated blocks which can be edited and inserted according to the user's needs. It is possible to generate drawing views using exploded representations as well.

Exploded representations are associative. Each part in an exploded representation is linked with the corresponding part in assembly. It is possible to update exploded representations to the current state of the assembly as well as to place balloon annotations on corresponding drawing views using the BMBALLOON command.



Bill of Materials Cabin_assembly		
No.	Component	Quantity
1	Apron	1
2	Bottom_assembly	1
3	Cabin_Handrail_assembly	1
4	Cabin_roof_assembly	1
5	Ceiling_light	1
6	Doorstep	1
7	Floor	1
8	Mirror	1
9	Panel_buttons	1
10	Wall_1	6
11	Wall_2	2
12	Wall_3	1
13	Wall_L	1
14	Wall_R	1
15	Wall_top	1

Levels of the Exploded Representation

Exploded representations contain references to the components which makes them quite lightweight. You can choose between two levels of the exploded representation: top or bottom.

Top Level: The exploded representation contains references to top level components. Top level components are components inserted directly into the main assembly. They can consist of several parts and/or subassemblies but are not parts of any subassembly by themselves.

Bottom level: The exploded representation contains references to bottom level components. Bottom level components are basic parts, which do not contain other parts or subassemblies.

NOTE If *Bottom Level* is selected and there are some middle-level components containing not only inserts of components but also other entities, such entities will be lost in the exploded representation, because it is not possible to reference them correctly. Those entities should be converted into corresponding bottom-level components first.

Algorithms

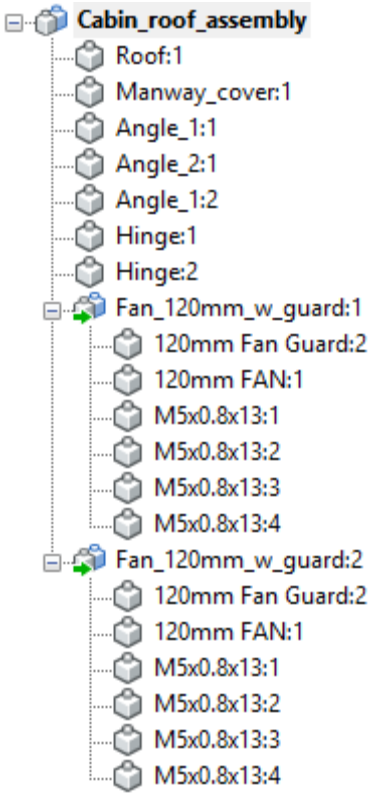
BricsCAD provides two algorithms to automatically create exploded representations and a Manual mode to create customized exploded representations.

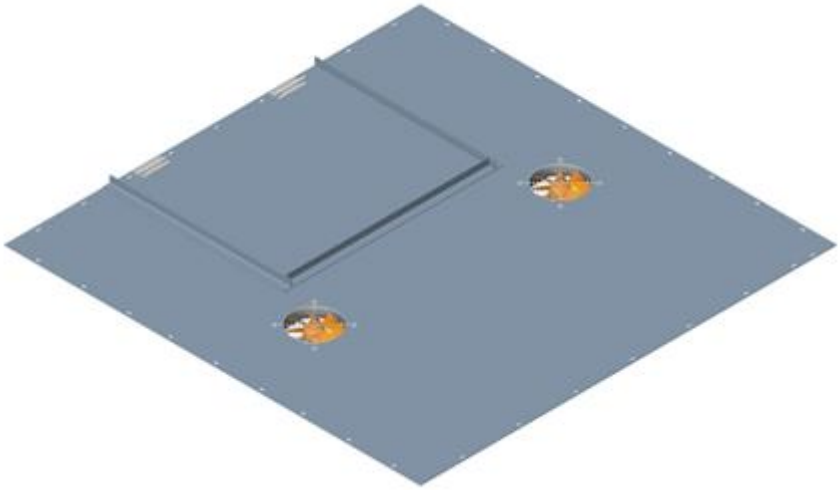
The *Table by Level* algorithm creates a table of parts where each table row contains parts which are on the same level in the assembly tree. It means that the first row corresponds to parts inserted in the assembly, the second row contains subassemblies which contain parts only, the third row contains subassemblies of subassemblies, etc.

The *Table by Type* algorithm creates a table where parts of the same type are grouped in same rows.

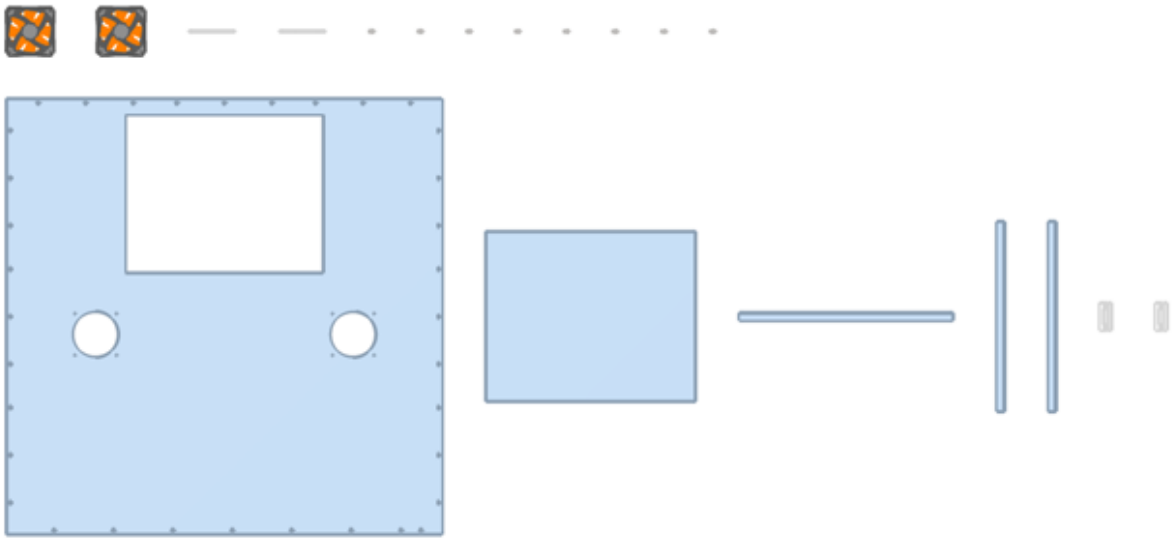
The *Manual* algorithm creates an exact copy of the assembly which is ready for custom editing. In this mode, it is possible to create a custom exploded representation without changing the main assembly.

Example: the Cabin Roof assembly:





Structure tree in the Mechanical Browser panel *Assembly*



'Table by Level' Exploded Representation

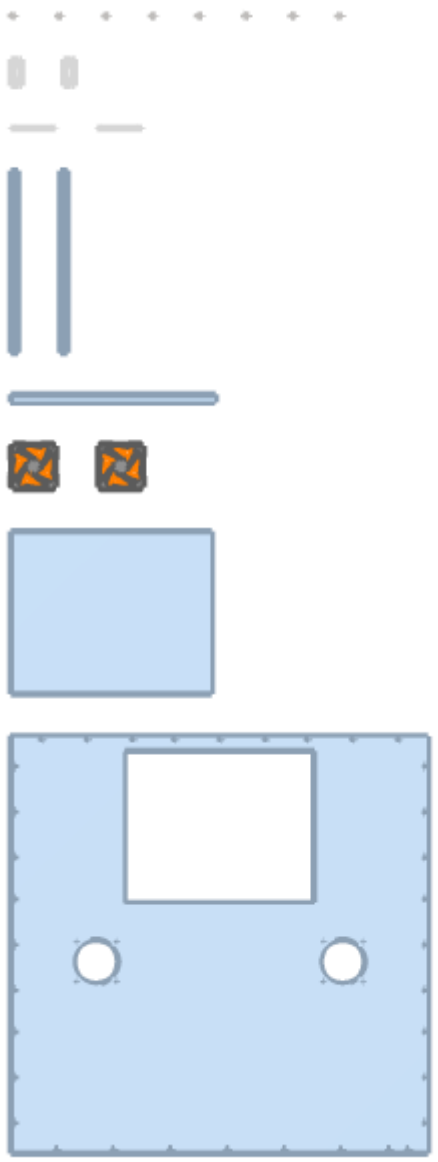


Table by Type' Exploded Representation

Creating an Exploded Representation

The BMEXPLODE command allows to create an exploded representation of the current assembly. Note that it is necessary to have at least one insert of a part in the assembly for the command to work.

To launch the BMEXPLODE command, do one of the following:

- Click the *Explode* tool button ( )on *Tools* panel of the *Assembly* ribbon tab
- Click the *Explode* tool button ( )on the *Assembly* toolbar
- Select *Explode* in the *Assembly* menu
- Type *bmexplode* in the command bar.

You are prompted: Select explosion algorithm [Table by Level/Table by Types/Manual/Settings]:

(Option) Set the *Level* and the *Name* of the exploded representation using the *Settings* option.

Select the algorithm.

You are prompted: Select position of the exploded body:

Select the position to insert the exploded representation.

Generating Drawing Views

Select the exploded representation in the model space

Launch the VIEWBASE command and select the exploded representation.
You are prompted: Enter new or existing layout name to make current <Layout1>:

Accept the default layout name or type a new name.

Position the generated drawing views of the exploded representation in the paper space layout.

NOTE To be able to use balloons (see BMBALLOON command) with drawing views of exploded representations, the GENERATEASSOCVIEWS system variable must be ON.

Updating an Exploded Representation

Execute the BMUPDATE command to synchronize exploded representations with the current state of the assembly.

If necessary, adjust the exploded representation. Such adjustments, for example, may be required because of addition and/or deletion of some parts.

NOTE The position of parts in exploded representations created in *Manual* mode are not updated automatically, even if no editing has been done.

Assembly Mass Properties

BmMassProp Command

An assembly can consist of multiple parts made from different materials. To correctly compute the mass, center of gravity (centroid), and inertia of such assemblies, BricsCAD supports individual densities for components.

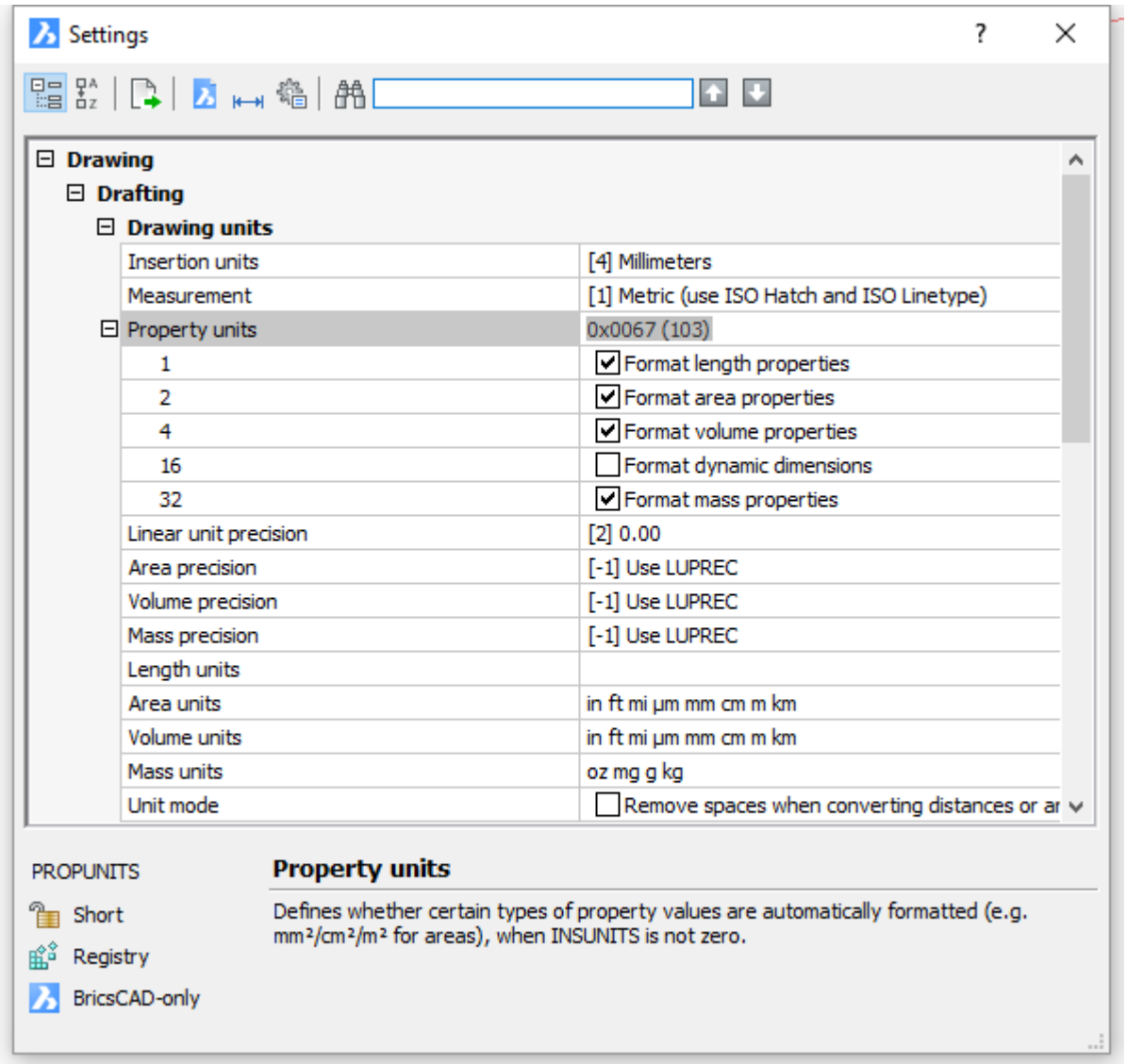
A physical material for a mechanical component is defined in the Material property in the Mechanical Browser.

By default, a component inherits the density of the assembly, in which it was inserted into or formed from.

To Set the Mass Units

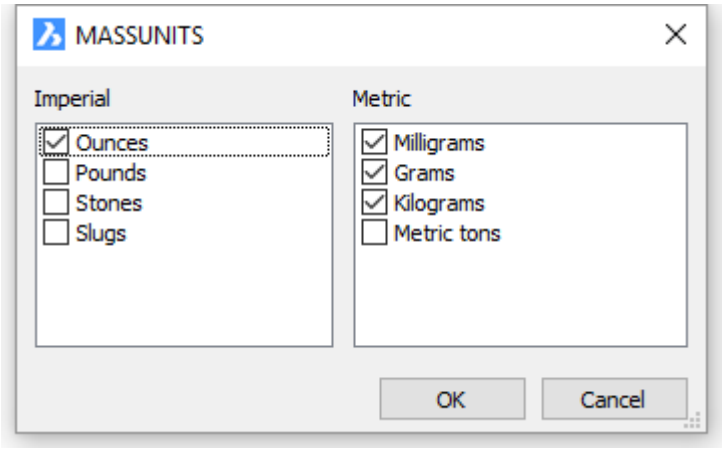
Open the Settings dialog box.

Expand the *Property units* settings under *Drawing / Drafting / Drawing* units.



Check the *Format mass properties* checkbox to enable formatting of mass values with appropriate units.

Select the *Mass units* field, then click the *Browse* button (...) and specify the desired mass units in the MASSUNITS dialog box.



Specify the desired precision in the *Mass precision* field.

Close the *Settings* dialog box.

NOTE The *Mass units* setting affects the mass values only . Other mass properties such as density or moments of inertia will be formatted in SI units for the metric system and in imperial units for the imperial system, regardless of the value of the *Mass units* setting.

To Compute the Mass Properties

Execute the BmMassProp command.

You can compute mass properties for the entire model or for a selection of 3D solids and/or subcomponents. The values of mass, volume, centroid, moments of inertia, products of inertia, radii of gyration, principal monents and X-Y-Z directions about centroid are reported in the command bar.

Optionally you can align the User Coordinate System with the inertia axes, as a result the center of masscentroid of the component lies at the origin of UCS, which can be useful to investigate the behavior of a product under gravity force.

Parametric Modeling

BricsCAD provides a rich toolset to control 2D and 3D geometric model via parameters of different kinds, including:

global parameters, which includes parameters of 2D/3D dimensional constraints, user-defined parameters, and configuration parameters of design table,

local parameters, which includes parameters of arrays and constrained entities inside arrays, and parameters of inserts of parametric components.

All parameters and their current values are displayed under different sections in the Mechanical Browser. Parameter values can be linked together via expressions. When the value of a parameter is changed, the geometric model is updated automatically. This toolset allows BricsCAD users to add parametric behavior to any geometry and easily explore design variants.

Note that in BricsCAD there are no parent-child dependencies between geometric elements: for example, if you change a sketch used to create an extruded 3d solid, the solid is not changed accordingly. But you can always create any kind of dependencies using parameters and expressions.

Global and local parameters

Commands: -PARAMETERS, MECHANICALBROWSEROPEN, CLEANUNUSEDVARIABLES

Parameters in BricsCAD can be global and local. Local parameters are attached to a particular entity: an associative array or a mechanical component. Global parameters are not attached to a particular entity; they are subdivided into three groups:

parameters of 2D dimensional constraints (see User Guide/Drawing Accurately/Parametric Constraints/Using Dimensional Constraints to learn how to use 2D dimensional constraints),

parameters of 3D dimensional constraints (see User Guide/Creating Entities/Direct Modeling/3D Constraints to learn more about 3D dimensional constraints),

user-defined parameters,

configuration parameters of design tables.

To learn how to create 2D dimensional constraints, please read User Guide/Drawing Accurately/Parametric Constraints/Using Dimensional Constraints.

To learn how to create 3D dimensional constraints, see User Guide/Creating Entities/Direct Modeling/3D Constraints.

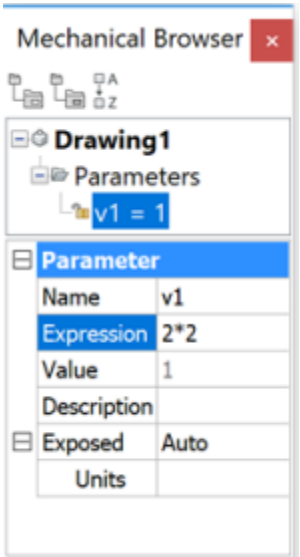
To create a user-defined parameter, follow the procedure described in User Guide/Drawing Accurately/Parametric Constraints/Using Dimensional Constraints/Creating a New Parameter.

To create a configuration parameter for a design table, see User Guide/Creating Entities/Parametric Modeling/Design Tables.

Setting an expression to a parameter

You can set an expression to any parameter, being it global or local. A simplest expression is just a number of a name of a global parameter (local parameters cannot be referenced by names in expressions). A more complex expression can also include standard operators and functions (see below).

To set an expression to a parameter, click on the parameter name in the Mechanical Browser, and fill out the *Expression* field:

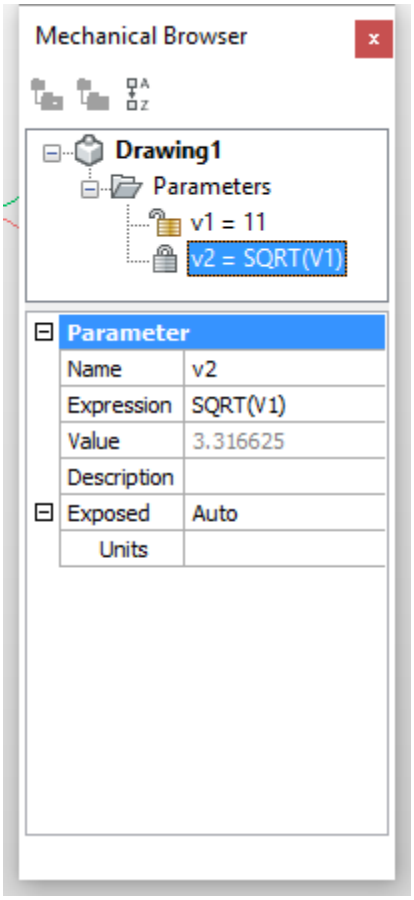


NOTE You can also set an expression to a parameter in command line using the -PARAMETERS command.

You can use operators and functions to compose an expression.

Driving and driven parameters

In expressions, you can use not only numbers, but also the names of global parameters. Such expressions are called non-constant. If a non-constant expression is set to a parameter, its icon, an open padlock ( ) in the Mechanical Browser changes to the closed padlock ( ):



This means that the parameter becomes driven by other (driving) parameters: its value will be automatically recalculated when the values of the driving parameters will be changed.

You can create a chain of parameters, where every next element is driven by the previous ones. The length of such chain is not limited. Obviously you cannot create a loop, where the next element in a chain drives one of the previous elements, because it can result in infinite evaluation of expressions in the loop. BricsCAD automatically detects such expressions and therefore does not accept expressions, which lead to a parametric loop.

Parameters of Associative Arrays

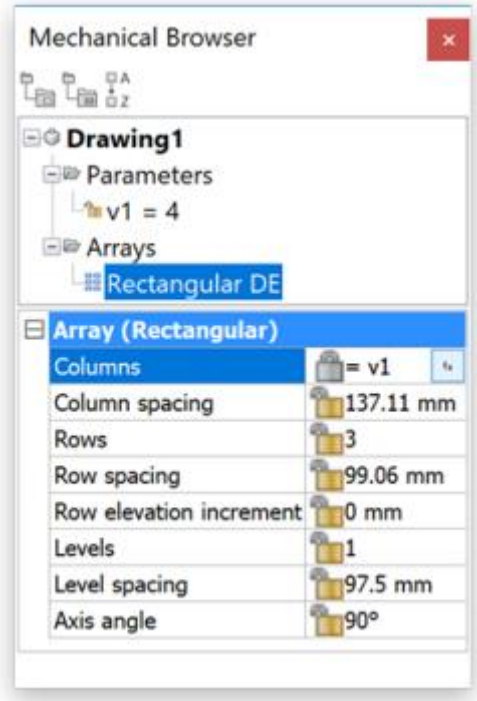
Commands: ARRAY, ARRAYRECT, ARRAYPOLAR, ARRAYEDIT.

Properties of associative arrays that can be controlled by an expression:

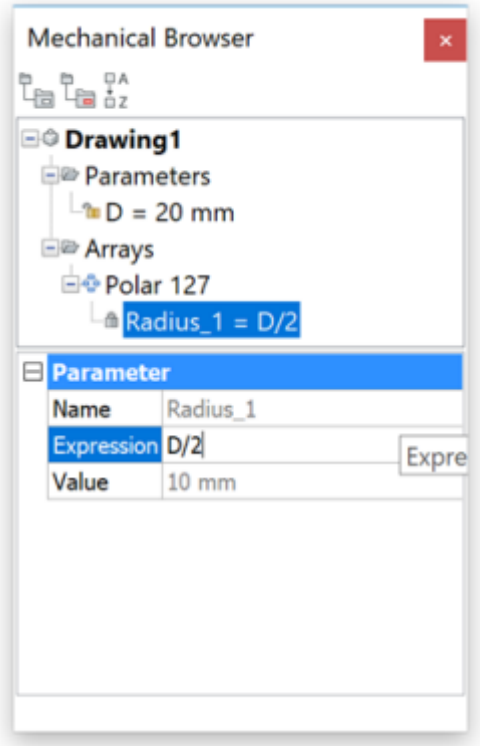
Rectangular arrays: Number of columns, Column spacing, Number of rows, Row spacing, Row elevation increment, Number of levels, Level spacing, Axis angle.

Polar arrays: Radius, Number of Items, Angle between items, Fill angle, Number of rows, Row spacing, Row elevation increment, Number of Levels, Level spacing.

To set an expression to any of these properties, select the corresponding array entity, either in the drawing area or directly in the Mechanical Browser and enter the expression in the corresponding properties field in the Mechanical Browser or in the Properties bar.



If an associative array contains entities constrained with 2D or 3D dimensional constraints, then parameters of these constraints can also be evaluated with expressions. These nested parameters are listed in the Mechanical Browser, and if you select any of them, you can set an expression to it:

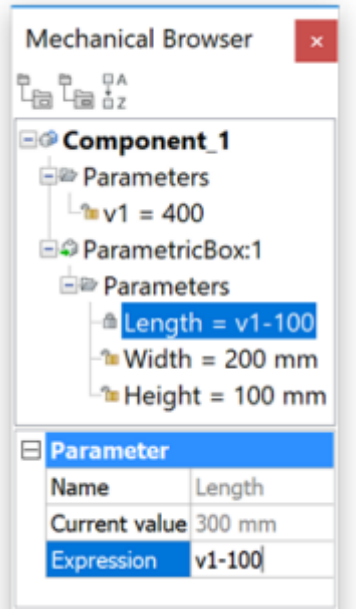


Parameters of Mechanical Components

Commands: -BMPARAMETERS, BMUPDATE,

The BMUPDATEMODE system variable controls whether external assembly components are always reloaded or only in case they are modified.

If you insert a mechanical component or just a plain .dwg file into your drawing with BMINSERT command, and the insert contains global exposed parameters, then such parameters become local in the current drawing. They are associated with the corresponding component insert and are visible in the Mechanical Browser under the name of this insert. Click such parameter to edit:



Design Tables

Commands: DESIGNTABLE, -DESIGNTABLEEDIT

A design table allows BricsCAD users to evaluate several global parameters at once using the value of one configuration parameter. Design tables can be created from scratch in BricsCAD or imported from a .csv file (a comma-separated values text file), which can be created with any spreadsheet software or a text editor.

Creating a Design Table from a .csv file

The expected format of the .csv file is as follows:

Size,Length,Width,Height
Small,300,200,100
Medium, 450,300,150
Large,600,400,200

The first row lists the parameter names separated by a comma. The first name should not coincide with the name of any global parameter persistent in your drawing. It is called the configuration name. Other names in the first row can (and usually do) coincide with the names of global parameters in your drawing. The next rows list possible values for these parameters, which can either be strings or numbers.

Importing such .csv file into BricsCAD using te DESIGNTABLE command results in creating a special global parameter by the name of the first name in the first row in the .csv file (= Size in the above example), which is called the configuration parameter. It will drive the values of other parameters in the design table, which are then called configuration-driven parameters.

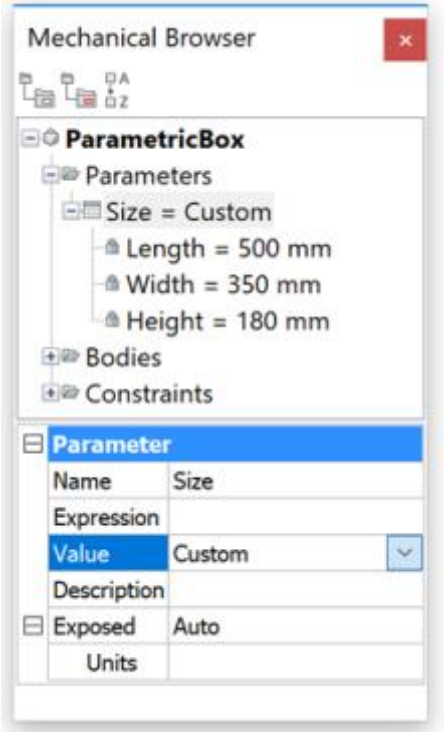
NOTE BricsCAD does not keep the association with the .csv file, which was used to create a design table: if this file has been modified then the design table is not updated automatically. Use the -DESIGNTABLEEDIT command to replace a design table using a new or modified .csv file.

Direct Lookup in a Design Table

To evaluate a configuration parameter:

Select it in the Mechanical Browser and type the desired value in its *Expression* field.

Select a value in the drop-down list of the *Value* field:



Changing the value of a configuration parameter will result in automatic changing the values of all configuration-driven parameters of the corresponding design table. The values of these parameters are taken from the row in the design table, of which the first element coincides with the current value of the configuration parameter.

Reverse Lookup in a Design Table

Parameters, which are driven by a configuration parameter via a design table, can be evaluated independently. To do so, select the driven parameter and apply an expression to it. If the value of this expression matches one of the values for this parameter in the corresponding design table, then the configuration parameter and all driven parameters of the design table will be automatically set to the values specified in the corresponding row in the table. If several configurations are possible for a particular value of a driven parameter, BricsCAD selects the one, which is closest to the current values of other parameters.

If you enter an expression of which the result is not available in the design table, or if you enter expressions for two or more design table parameters, of which the values cannot be found in a single configuration (= the values are not found in the same row in the design table), then the values of the configuration parameter are changed to 'Custom', and the values of the other driven parameters are not changed. This case corresponds to a configuration, which does not exist in the design table.

Adding a New Configuration to a Design Table

You can add any 'Custom' configuration to a design table under a particular name. To do so, right click the configuration parameter in the Mechanical Browser and select *Save current configuration* in the context menu. The configuration will be added to the design tableas 'Configuration_N', where 'Configuration' is the name of the configuration parameter, and 'N' is an index number, which is one more than the current number of configurations in the design table.

If you want to save a configuration in the design table under a different name, launch the &endash;DESIGNTABLEEDIT command.

Creating an Empty Design Table

Using the procedure described above (adding a new configuration to a design table), it is possible to start your design with an empty design table, and then fill it with desired values.

To do so, use the 'Empty Design Table' tool, which asks you:

The name of a new design table (which is also the name of the corresponding configuration parameter),

The names of global parameters, which will be driven by this design table.

Using a Parametric Component with Design Tables

You can expose both configuration parameters and configuration-driven parameters to be uses in assemblies. If you then insert such a component into your drawing, its parameters will include the exposed parameters of design tables, for which drop-down lists of admissible values will be available. Both direct and reverse lookup actions are supported for such parameters.

Other Actions with Design Tables

The &endash;DESIGNTABLEEDIT command allows to:

Delete a design table.

Replace a design table.

Link a new parameter to the design table or unlink an existing one.

Add new configurations, remove an existing one, or export a design table into a .csv file.

Global and Local Parameters

Commands: -PARAMETERS, MECHANICALBROWSEROPEN, CLEANUNUSEDVARIABLES

Parameters in BricsCAD can be global and local. Local parameters are attached to a particular entity: an associative array or a mechanical component. Global parameters are not attached to a particular entity; they are subdivided into three groups:

parameters of 2D dimensional constraints (see User Guide/Drawing Accurately/Parametric Constraints/Using Dimensional Constraints to learn how to use 2D dimensional constraints),

parameters of 3D dimensional constraints (see User Guide/Creating Entities/Direct Modeling/3D Constraints to learn more about 3D dimensional constraints),

user-defined parameters,

configuration parameters of design tables.

To learn how to create 2D dimensional constraints, please read User Guide/Drawing Accurately/Parametric Constraints/Using Dimensional Constraints.

To learn how to create 3D dimensional constraints, see User Guide/Creating Entities/Direct Modeling/3D Constraints.

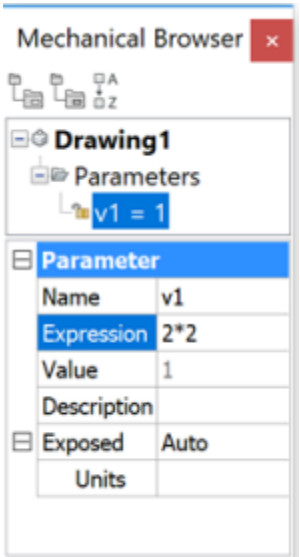
To create a user-defined parameter, follow the procedure described in User Guide/Drawing Accurately/Parametric Constraints/Using Dimensional Constraints/Creating a New Parameter.

To create a configuration parameter for a design table, see User Guide/Creating Entities/Parametric Modeling/Design Tables.

Setting an Expression to a Parameter

You can set an expression to any parameter, being it global or local. A simplest expression is just a number of a name of a global parameter (local parameters cannot be referenced by names in expressions). A more complex expression can also include standard operators and functions (see below).

To set an expression to a parameter, click on the parameter name in the Mechanical Browser, and fill out the *Expression* field:

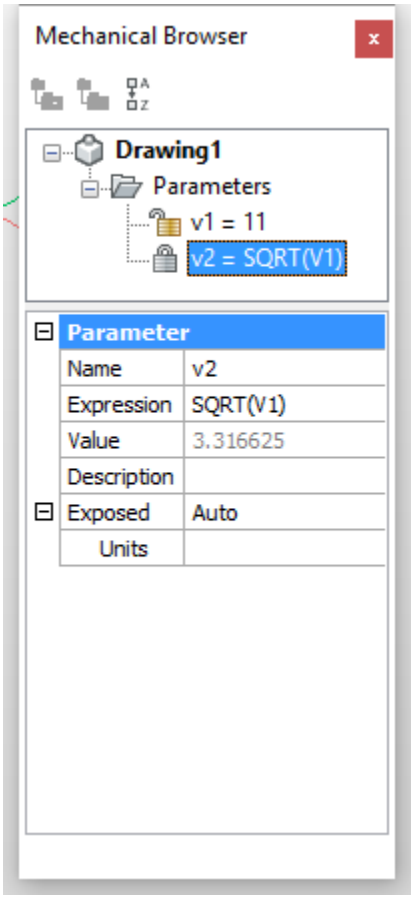


NOTE You can also set an expression to a parameter in command line using the -PARAMETERS command.

You can use operators and functions to compose an expression.

Driving and Driven Parameters

In expressions, you can use not only numbers, but also the names of global parameters. Such expressions are called non-constant. If a non-constant expression is set to a parameter, its icon, an open padlock ( ) in the Mechanical Browser changes to the closed padlock ( ):



This means that the parameter becomes driven by other (driving) parameters: its value will be automatically recalculated when the values of the driving parameters will be changed.

You can create a chain of parameters, where every next element is driven by the previous ones. The length of such chain is not limited. Obviously you cannot create a loop, where the next element in a chain drives one of the previous elements, because it can result in infinite evaluation of expressions in the loop. BricsCAD automatically detects such expressions and therefore does not accept expressions, which lead to a parametric loop.

Parameters and Associative Arrays

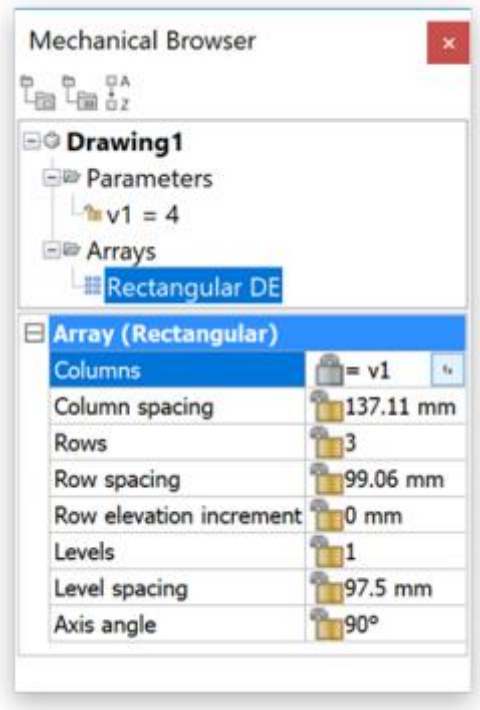
Commands: ARRAY, ARRAYRECT, ARRAYPOLAR, ARRAYEDIT.

Properties of associative arrays that can be controlled by an expression:

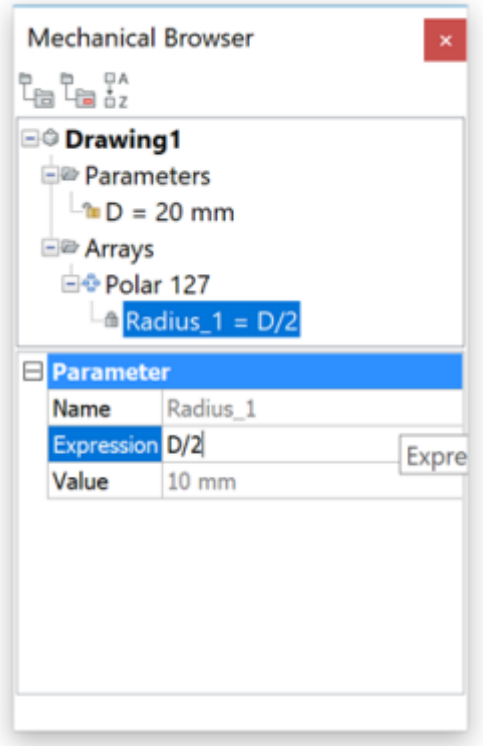
Rectangular arrays: Number of columns, Column spacing, Number of rows, Row spacing, Row elevation increment, Number of levels, Level spacing, Axis angle.

Polar arrays: Radius, Number of Items, Angle between items, Fill angle, Number of rows, Row spacing, Row elevation increment, Number of Levels, Level spacing.

To set an expression to any of these properties, select the corresponding array entity, either in the drawing area or directly in the Mechanical Browser and enter the expression in the corresponding properties field in the Mechanical Browser or in the Properties bar.



If an associative array contains entities constrained with 2D or 3D dimensional constraints, then parameters of these constraints can also be evaluated with expressions. These nested parameters are listed in the Mechanical Browser, and if you select any of them, you can set an expression to it:


Mechanical Browser
Drawing1
Parameters
D = 20 mm
Arrays
Polar 127
Radius_1 = D/2
Parameter
Name
Radius_1
Expression
D/2
Expre
Value
10 mm

MCAD Tutorial

Assembly Design and Kinematic Analysis Tutorial

The source files for this tutorial are installed in the ***[BricsCAD Installation folder]\Samples\Mechanical\piston*** folder.

Assembly design is a typical application where constraints are commonly used. BricsCAD Platinum allows users to assemble mechanisms and run kinematic analysis.

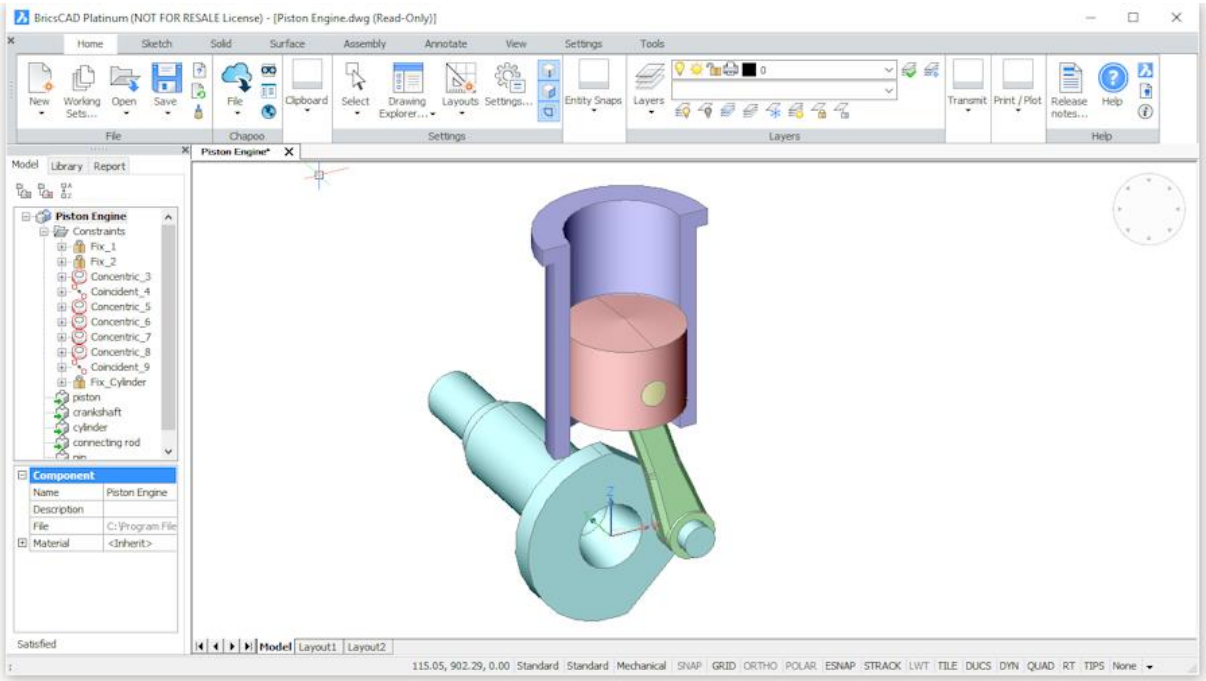
This tutorial shows you how to start working with assemblies in BricsCAD. You will learn how to assemble the different parts in a 3D model, which lets you control the final result.

There are two approaches in MCAD (Mechanical Computer-Aided Design) to build an assembly: top-down and bottom-up.

In the top-down approach, you start with an empty assembly and the geometry of each component is then created, one by one, in the assembly.

In the bottom-up approach, each component is first created as a single entity. All components are then inserted into the assembly. The position of each component is controlled through 3D constraints.

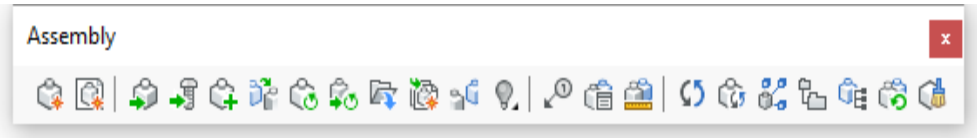
We will now assemble a simplified - in comparison with a real-world engine - model of a piston engine.



Step 1: Preparing for the Exercise

You need a *BricsCAD Platinum* license for this tutorial.

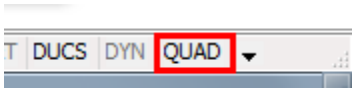
Make sure the *Asssembly* and *3D Constraints* toolbars are open.



To open a toolbar: move the cursor over an open toolbar, then right click and select the required

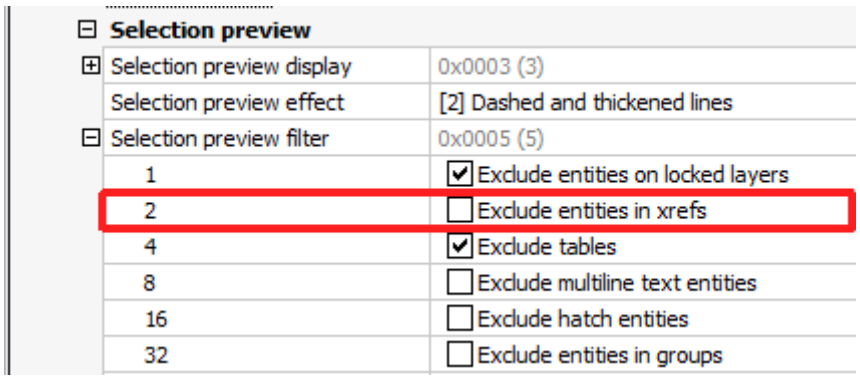
toolbars from the toolbar list.
Toolbars that are already open are marked in the list.

Make sure that the *Quad* cursor menu is active.



Click the *Quad* field in the Status bar if necessary.

Make sure that the *Exclude entities in Xrefs* option of the PREVIEWFILTER system variable is switched off.



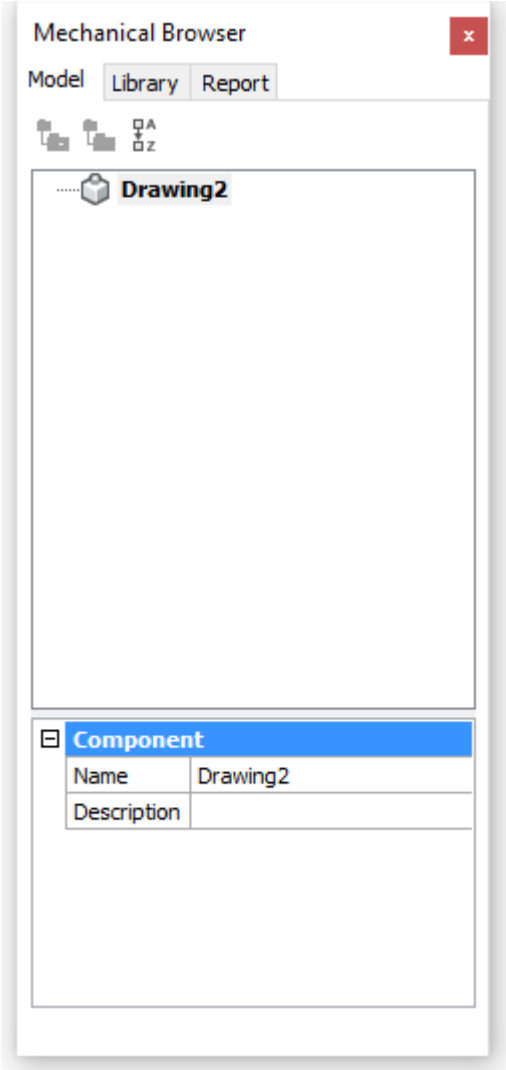
Open the *Mechanical Browser* dialog box.
Do one of the following:

Click the *Mechanical Browser* tool button ( ) on the *Mechanical | Tools* panel in the ribbon.

Click the *Mechanical Browser* tool button ( ) on the *Mechanical* toolbar.

Choose *Mechanical Browser* in the *Mechanical* menu.

Type *bmbrowser* in the command bar.



Make sure *Selection Modes* = 3
Do one of the following:

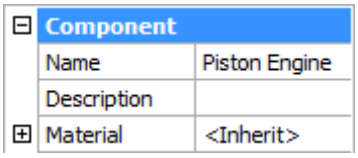
Type *selectionmodes* in the command bar, then type *3*.

Set the *Selection Modes* toolbar as indicated below.

Step 2: Creating the Assembly Drawing

Click the *New Component* tool button ( ) on the *Mechanical | Create* panel in the ribbon.
A new drawing of the name *Component_1.dwg* is created and *Component_1* is added in the *Mechanical Browser*.

In the bottom part of the *Mechanical Browser*, click the *Name* field and rename the main component: *Piston Engine*.

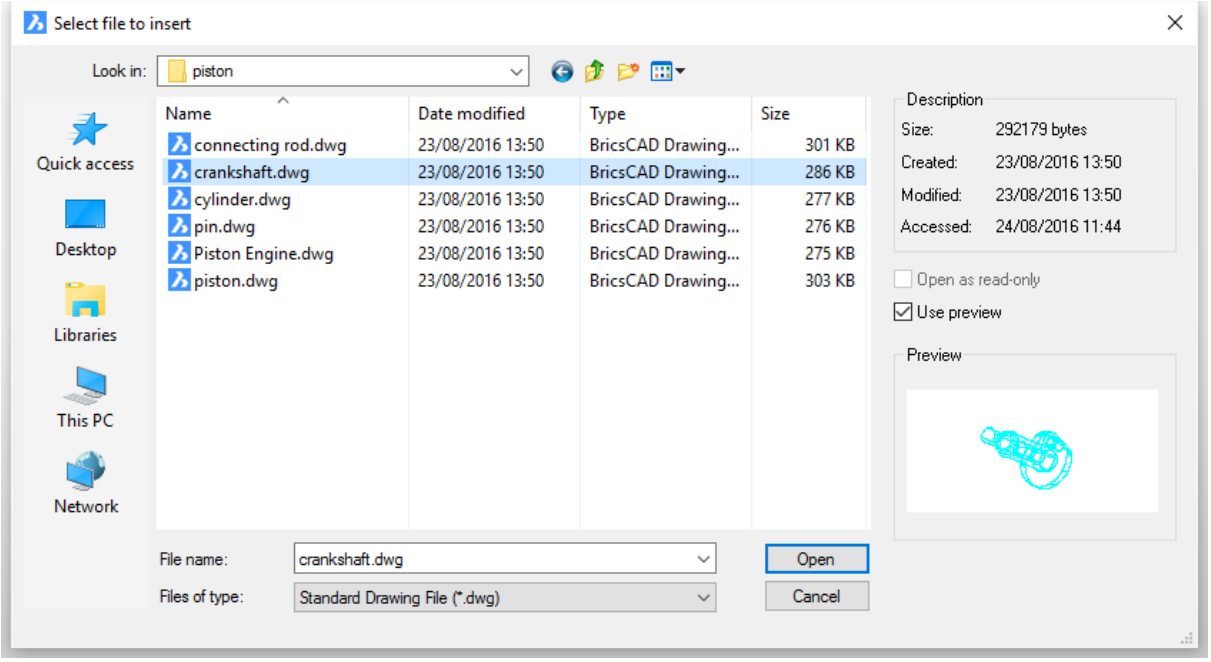


The drawing name automatically matches the main component name.

Save the drawing.

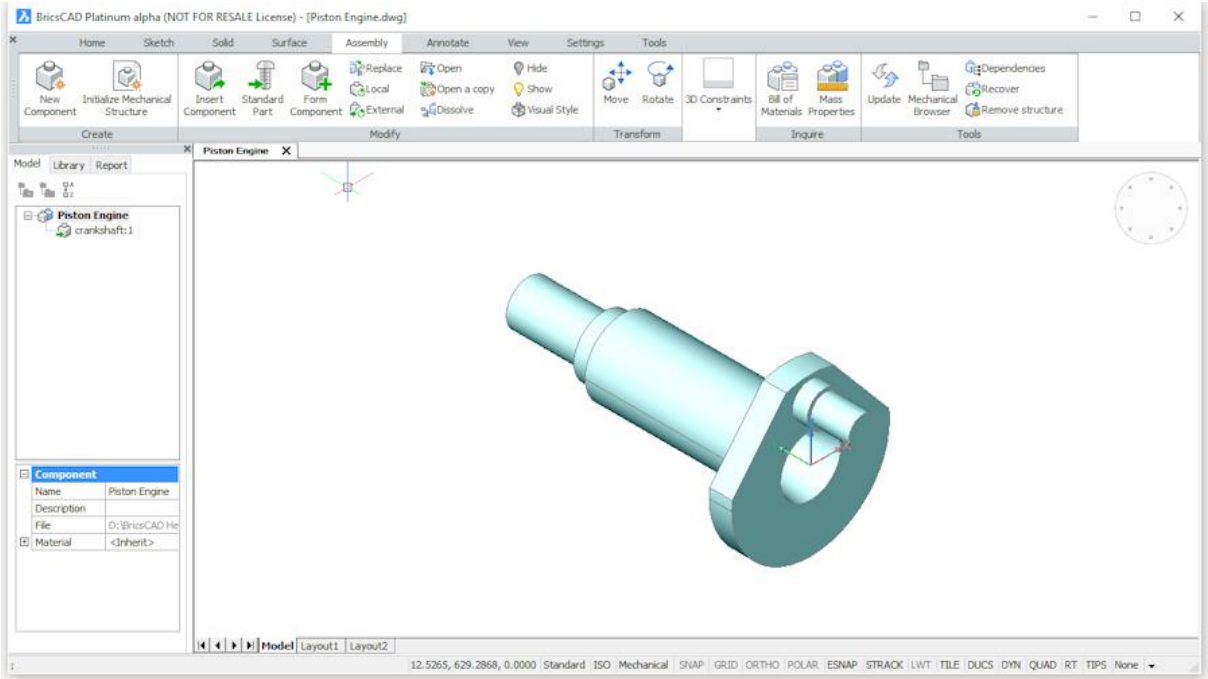
Step 3: Adding the Components

Click the *Insert Component* tool button ( ) on the *Mechanical | Modify* panel in the ribbon.
The *Insert Mechanical Component* dialog displays:
Open the *[BricsCAD Installation folder]\Samples\Mechanical\piston* folder.



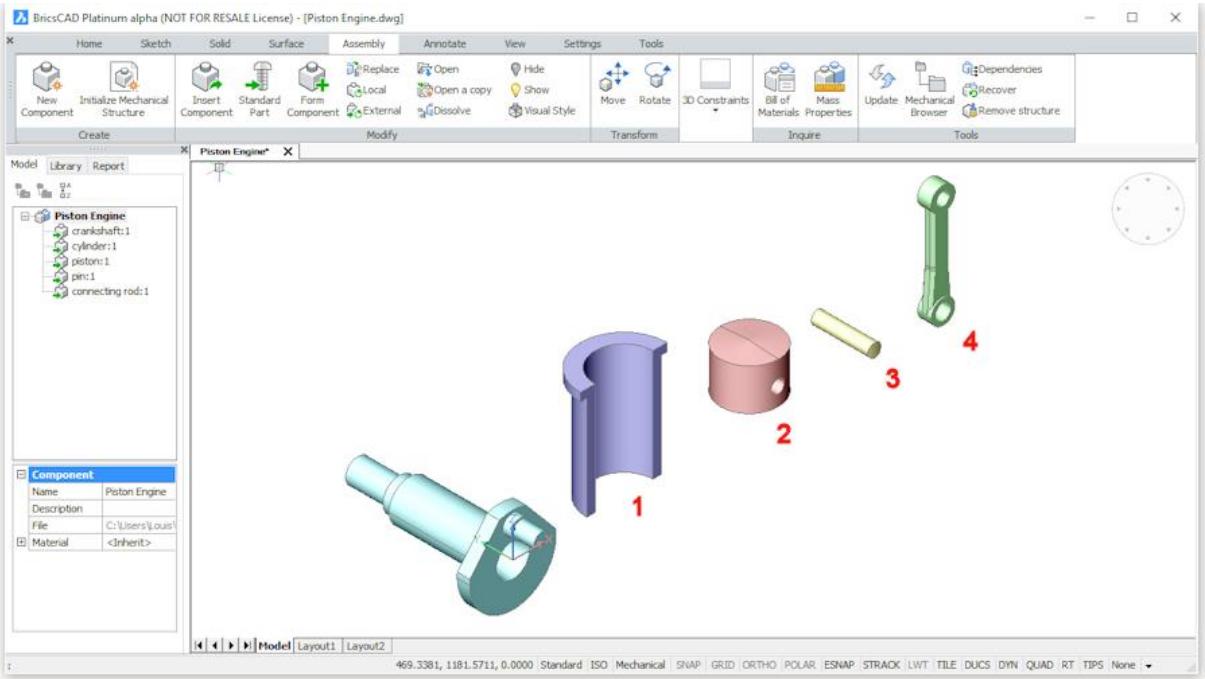
Select the *crankshaft.dwg*, then double click or click the *Open* button.
The component is attached to the cursor with its origin point.
Dynamic dimensions display: Distance from the origin of the current coordinate system.

We will consider the crankshaft as the anchor component of the assembly, and therefore insert it at the origin (0,0,0).
Please make sure the insertion point is exactly at 0,0,0, otherwise the kinematic analysis in step 5 might fail.



Use the Look From widget to adjust the view orientation.

Repeat the previous steps to insert the other components: *cylinder* (1), *piston* (2), *pin* (3) and *connecting rod* (4).
Avoid visual intersection of the components.

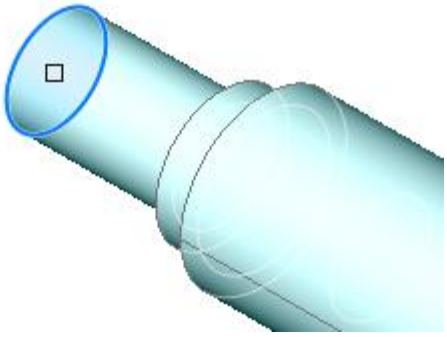


Step 4: Positioning the Components

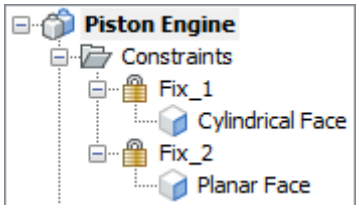
Click the *Fix* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted: Select an edge, face or 3D solid:

Move the cursor over the largest cylindrical face of the crankshaft. Click when the face highlights.

Repeat the DmFix3d command.
Move the cursor over the currently obscured circular face of the crankshaft.
Hit the TAB key until the face highlights and click when the face highlights.



The position of the selected faces of the crankshaft is now locked, as indicated in the *Constraints* tree in the *Mechanical Browser*.

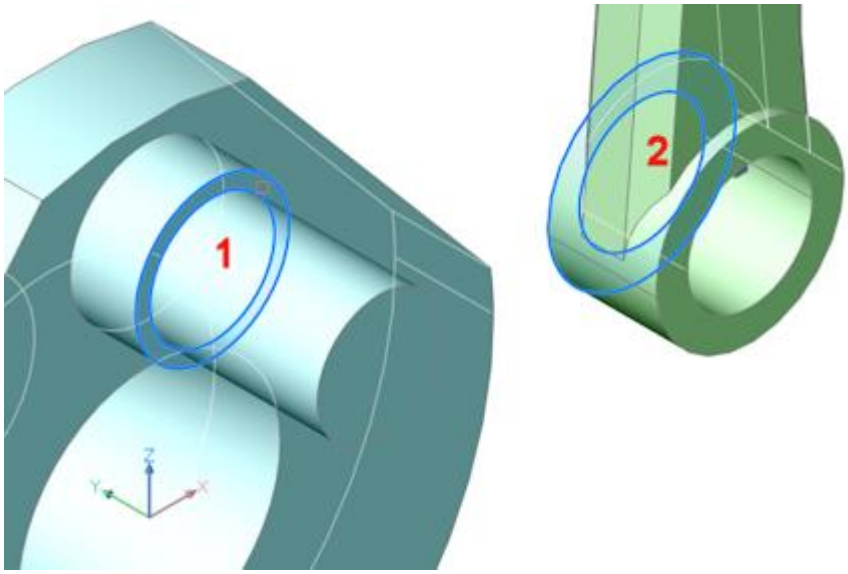


Clicking a constraint highlights the solid, face or edge it applies to.

Place the connecting rod near the crankshaft.

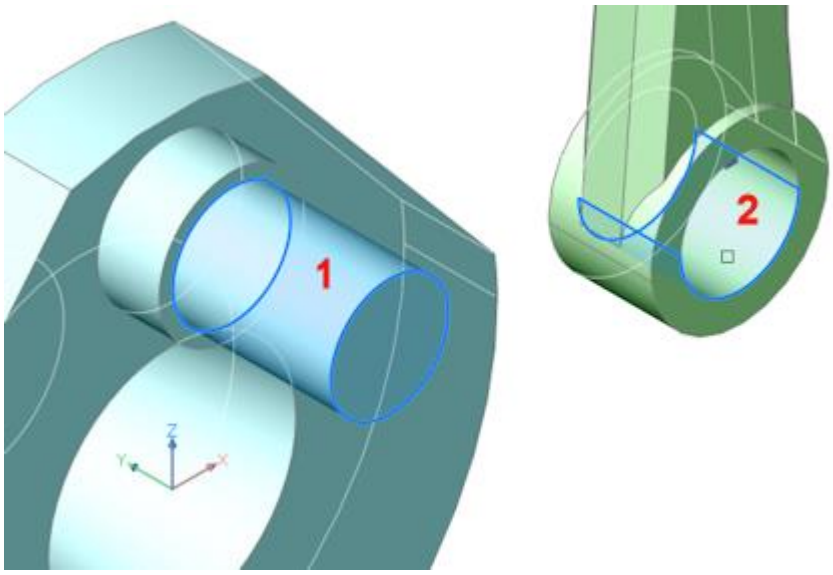
Click the *Coincident* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted:
Select a pair of subentities or [Autoconstrain] <Autoconstrain>:

Move the cursor over two planar faces of the crankshaft (1) and the connecting rod (2) as indicated in the image below.
Click when the correct face highlights. Hit the TAB key to select the obscured face of the connection rod and click when the face highlights.

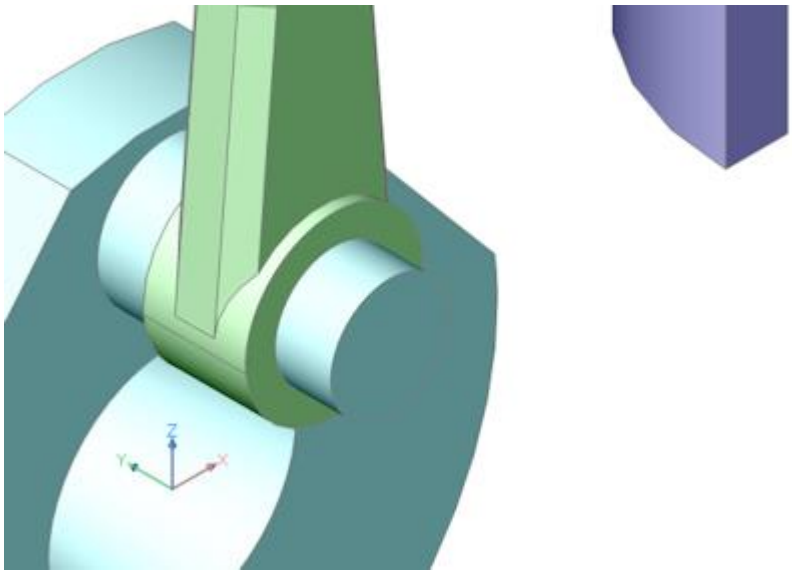


Click the *Concentric* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted:
Select the first entity:

Move the cursor over two cylindrical faces of the crankshaft (1) and the connecting rod (2) as indicated in the image below.
Click when the first face highlights, then select the second face.

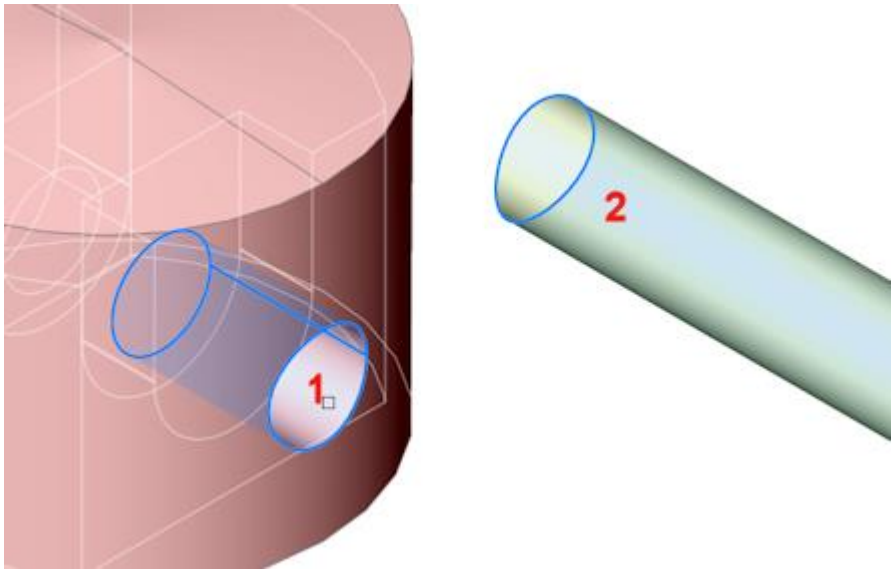


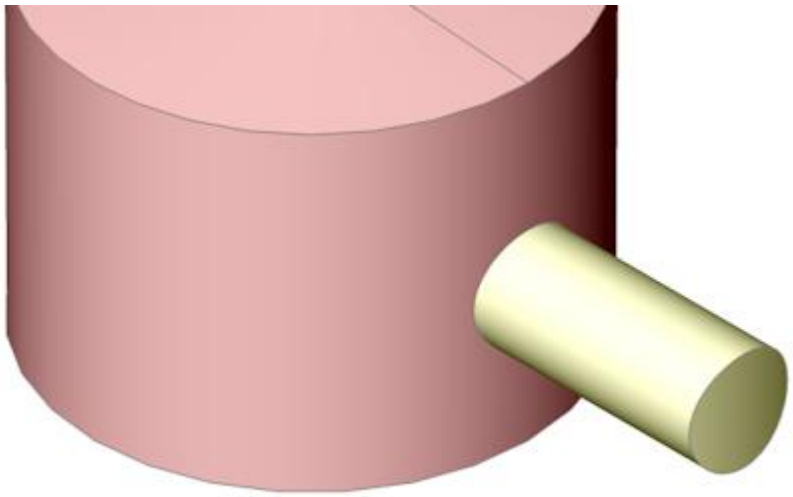
The connecting rod is automatically moved to the right position according to our design intent.



Click the *Concentric* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted:
Select the first entity:

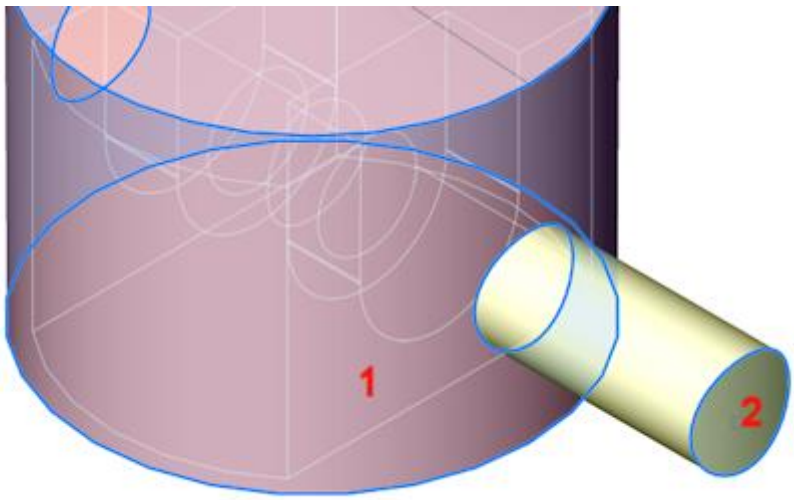
Move the cursor over two cylindrical faces of the piston (1) and the pin (2) as indicated in the image below.
Click when the first face highlights, then select the second face.

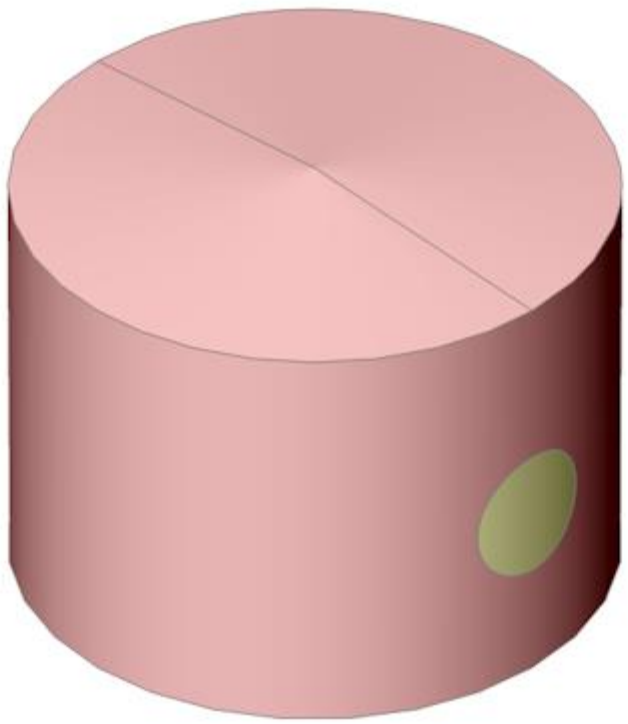




Click the *Tangent* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted:
Select the first entity:

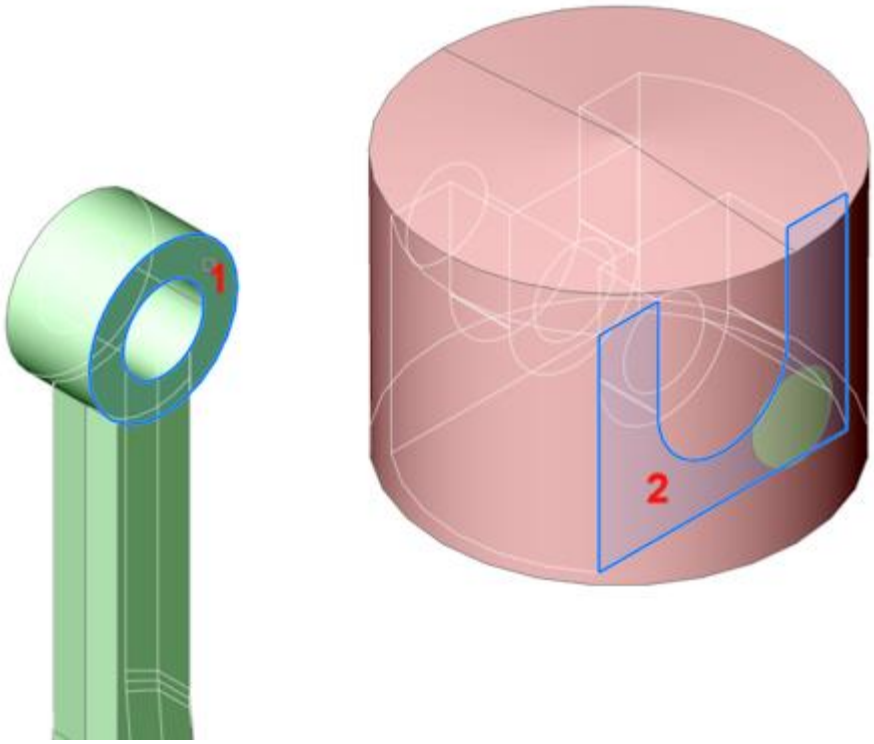
Move the cursor over two cylindrical faces of the piston (1) and the pin (2) as indicated in the image below.
Click when the first face highlights, then select the second face.





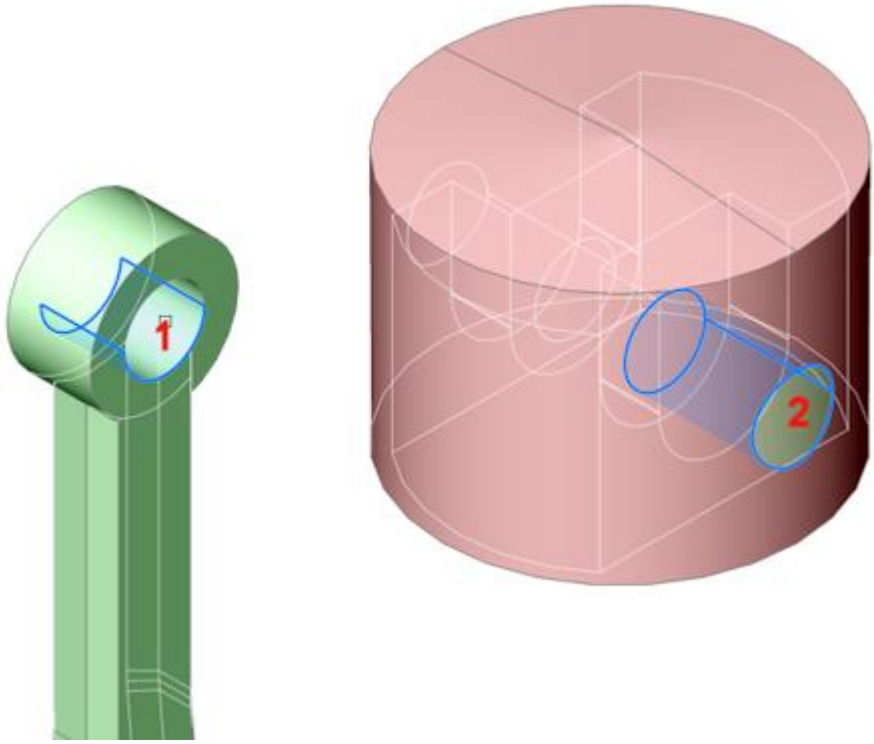
Click the *Coincident* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted:
Select a pair of subentities or [Autoconstrain] <Autoconstrain>:

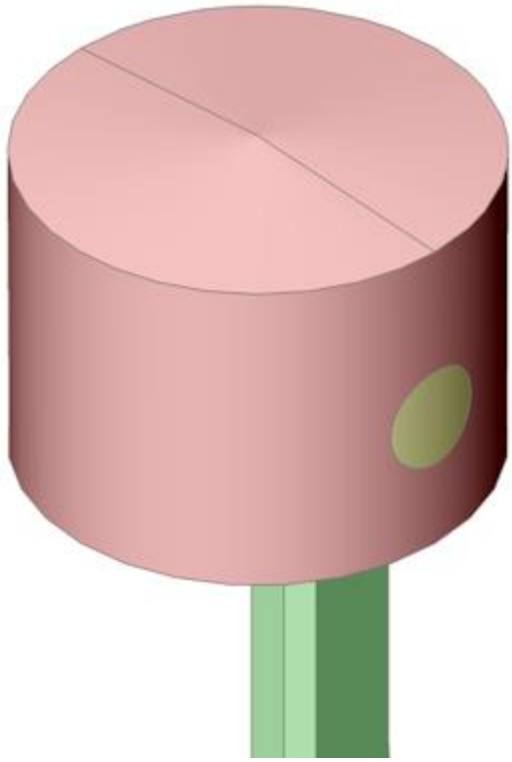
Move the cursor over two planar faces of the connecting rod (1) and the piston (2) as indicated in the image below.
Click when the first face highlights, then select the second face.
Hit the TAB key to select the interior face of the piston.



Click the *Concentric* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted:
Select the first entity:

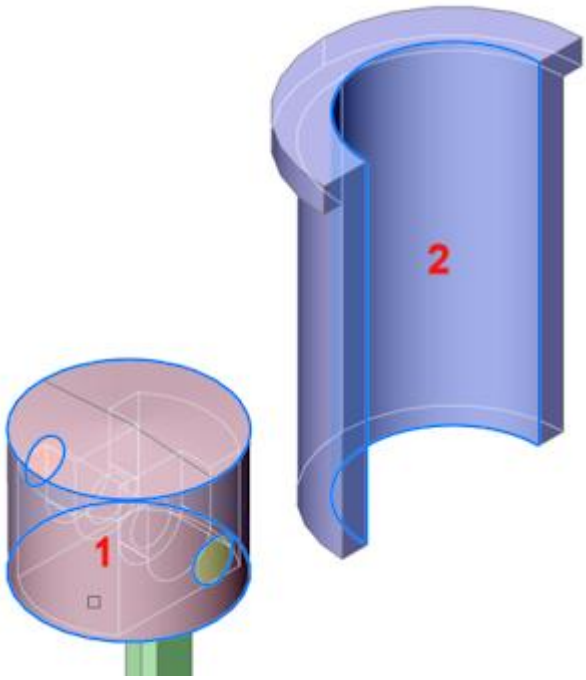
Move the cursor over two cylindrical faces of the connecting rod (1) and the pin (2) as indicated in the image below.
Click when the first face highlights, then select the second face.

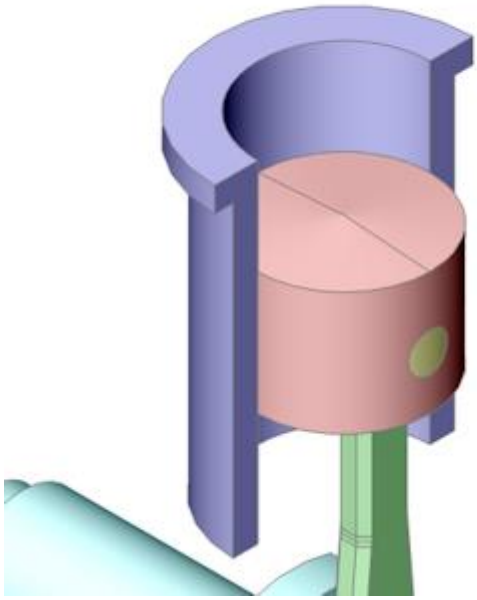




Click the *Concentric* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted:
Select the first entity:

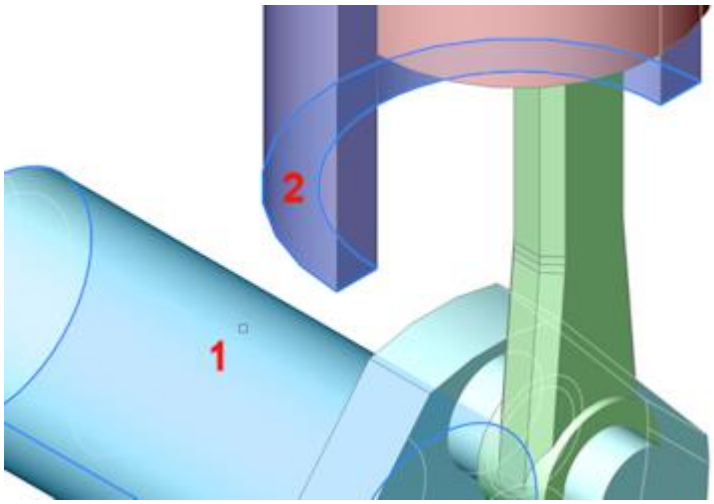
Move the cursor over two cylindrical faces of the piston (1) and the cylinder (2) as indicated in the image below.
Click when the first face highlights, then select the second face.





Click the *Distance* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted:
Select a first entity or specify [measurement Mode]:

Move the cursor over the cylindrical faces of the crankshaft (1) and the planar bottom face of the cylinder (2) as indicated in the image below.
Click when the first face highlights, then select the second face.
Hit the TAB key to select the bottom face of the cylinder.



Type 120 in the dynamic distance field.

(optional) Edit the *Expression* field of the distance constraint in the *Mechanical Browser* panel to adjust the vertical position of the cylinder.

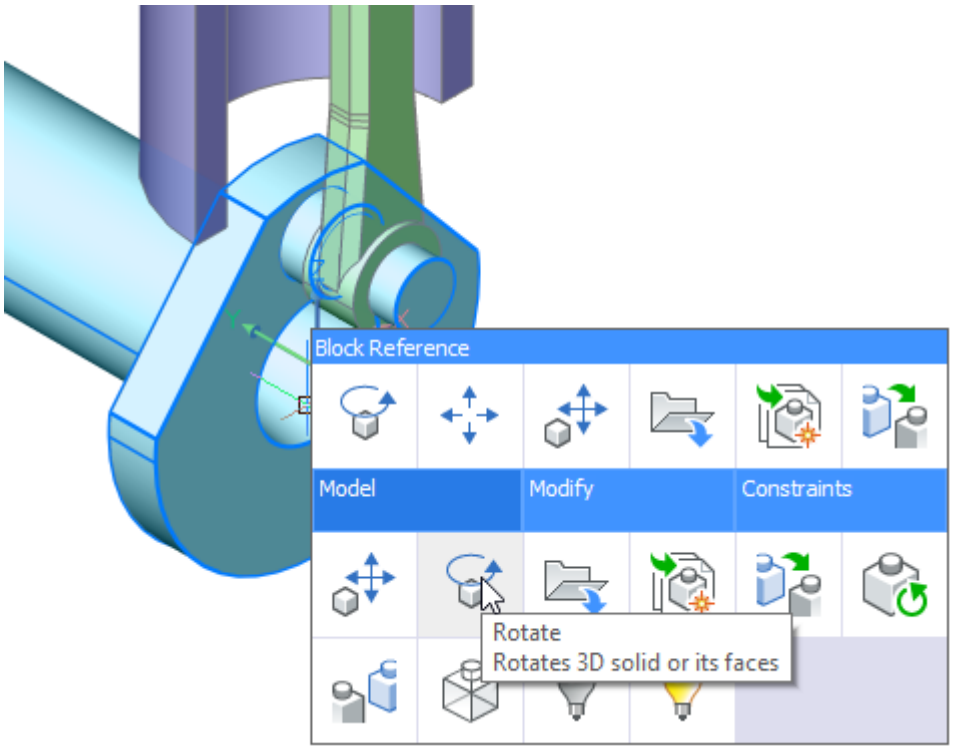
Click the *Fix* tool button ( ) on the *Solid | 3D Constraints* panel in the ribbon.
You are prompted: Select an edge, face or 3D solid:

Click the outside cylindrical face of the cylinder.

Step 5: Kinematic Analysis

In this assembly the crankshaft is the driving component. To run the kinematic analysis we will rotate the crankshaft around its axis, which in this case is the Y-axis of the WCS.

Move the cursor over the crankshaft, then choose *Rotate* ( ) in the *Model* command group in the *Quad* cursor menu.



You are prompted: Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>:

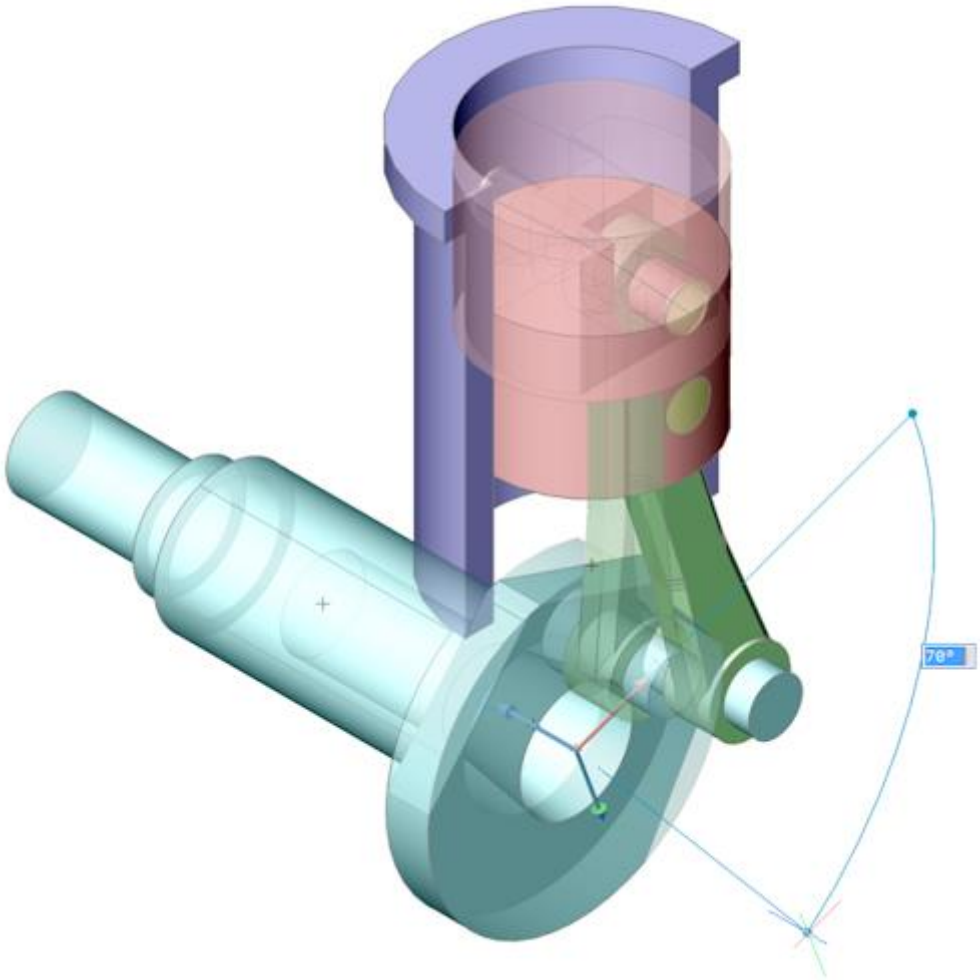
Do one of the following:

Type *Y*, then press Enter.

Choose *Y axis* in the prompt menu.

You are prompted: Pick start point in the rotation plane:

Pick a point in the drawing, then move the mouse to define the rotation angle.
The piston and the connecting rod move dynamically along with the rotation of the crankshaft, while the cylinder stays at its fixed position.



BricsCAD Communicator

Commands: COMMUNICATORINFO, EXPORT, IMPORT

TUTORIALS

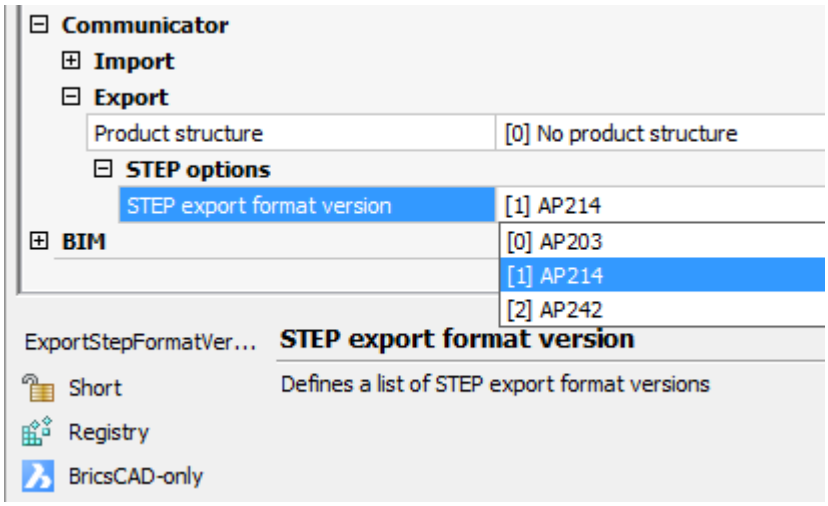
Assembly design using BricsCAD Communicator

BricsCAD Communicator is an optional plug-in for the Pro and Platinum editions of BricsCAD that imports and exports 3D data between major CAD formats. Its high quality translations facilitate the re-use of 3D CAD data, thereby improving productivity and reducing project costs.

Supported file formats

Export

STE, STP, STEP: Standard for Exchange of Product data. The following export protocols can be specified: AP203, AP214, AP242.



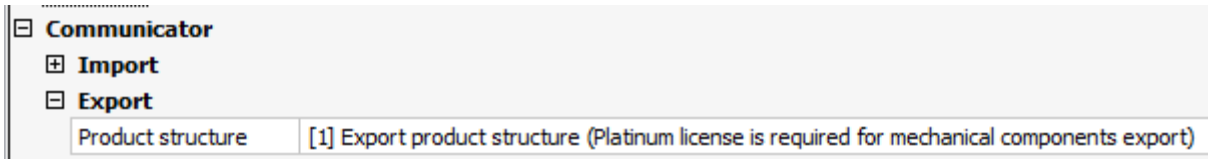
IGS, IGES: Initial Graphics Exchange Specification

CATIA V4 and V5:

VDA: VDA-FS

3D PDF

You can export BricsCAD assembly with mechanical components as a STEP file with product structure. To do that set the desired value in Settings dialog:



Import

ACIS

ASM, PRT: Creo Elements / Pro Engineer

CATIA V4/V5/V6

IAM, IPT: Inventor

IGS, IGES: Initial Graphics Exchange Specification

NX

PARASOLID

PAR, PSM: Solid Edge

Pro/E/Creo

PRT: NXSTE, STP, STEP: Standard for Exchange of Product data

SLDASM, SLDPRT: Solidworks

VDA: VDA-FS

XCGM

When you import an assembly file, you can get its content as a plain geometry (a nonhierarchical set of entities) or map its product structure into native blocks or mechanical components.

PMI (product and manufacturing information) is supported for the following file formats: ACIS, CATIA V5, NX, Pro/E / Creo, SolidWorks, XCGM (Communicator license is required). PMI annotations are imported as graphical elements grouped in blocks.

If the assembly contains hidden parts, you can decide how to represent them in BricsCAD: import and hide, import and set visible, or just do not import.

The desired behavior can be changed in the Settings dialog:

Communicator	
Import	
Product structure	[0] None
Product and manufacturing information	☑ Import
Perform stitching	☐ Perform stitching
Repair model on import	☑ Repair model on import
Perform simplification	☐ Perform simplification
Hidden parts	[0] Import and hide
Creo options	
Alternate search paths	
IGES options	
Perform stitching	☑ Perform stitching
Perform simplification	☑ Perform simplification
Inventor options	
Alternate search paths	
NX options	
Alternate search paths	
SolidEdge options	
Alternate search paths	
SolidWorks options	
Map SolidWorks Y to BricsCAD Z axis	☑ Map axis
Alternate search paths	
Export	
Product structure	[0] No product structure
STEP options	
STEP export format version	[1] AP214

You can also set other import options to automatically repair, stitch and simplify imported geometry. If you import a 3D model from SolidWorks, it makes sense to set an option to map Y axis into Z (because in SolidWorks the vertical axis is Y).

For SolidWorks and other native CAD formats it is possible to specify alternate search path, which will be used to look for parts absent in the imported assembly folder.

BIM

TUTORIALS

BricsCAD BIM tutorials

Instead of creating a set of dedicated BIM primitives like walls, slabs, columns, etc..., each with their own restricted behavior and properties, we opted for offering maximum flexibility. Any 3D Solid can be used in a BIM model, whether it is created and edited by using existing BricsCAD modeling tools, new powerful BIM commands, or even imported from 3rd party software or edited by 3rd party tools.

At any stage one can decide to classify 3D Solid, or other entities, as Building Elements of type Wall, Slab, Column, etc... Classification can be done automatically and manually.

The automatic classification tool analyzes and classifies entire buildings in a few seconds.

Compositions can be assigned to the various solids of the BIM model. Compositions can be composed of either a single ply or a multiply material. Compositions and materials are saved in a project database, which is either embedded in the BIM model file or in an external library.

IFC import/export ensures optimal cooperation with other disciplines and applications.

Villa PM on BricsCAD by courtesy of Architrend Architecture - Italy

Creating Walls

In BricsCAD walls are created using the PolySolid command.

When drawing walls it is recommended to use Dynamic Dimensions to specify the length of the wall and Polar Tracking to control the direction.

TUTORIALS

Creating Walls

Creating a Single Wall

Do one of the following:

Click the *PolySolid* tool button ( ) on the *3D Solids* toolbar.

Choose *Polysolid* in the *Model > 3D Solids* menu.

Type *polysolid* in the command bar.

You are prompted: Start point or Height/Width/Justification/separateSolids/Dynamic/<Object>:
Specify the start point of the wall.
You are prompted: Arc/Distance/<Next point>:
The footprint of the wall displays dynamically.
Move the cursor in the desired direction until a polar tracking line displays.
The current length displays in the *Length* dynamic entry field,
Optionally hit the Shift key to lock the polar tracking line. Locked tracking lines turn red. Hit the shift key again to unlock.
Do one of the following:

Specify the endpoint.

Type a value in the *Length* dynamic entry field, then press Enter.

The footprint of the next wall displays dynamically.
Right click or press Enter.
The height of the wall displays dynamically in the *Height* field..
The default width of the wall is indicated in the *Width* field.
The current justification of the wall is indicated in the Tips widget.

Repeatedly hit the Ctrl key to cycle the justification options: *Right*, *Center* or *Left*.
The justification of the wall is adjusted accordingly.
(option) Hit the TAB key, then enter a value in the *Width* field then hit the TAB key again.
Do one of the following to define the height of the wall:

Press Enter to accept the default height.

Type a value in the *Height* field, then press Enter.

NOTE The previously used width and height of polysolids is saved to the *PSOLWIDTH* and *PSOLHEIGHT* system variables, which are saved in the drawing. The default values are used the next time the Polysolid command is executed.

Creating a Chain of L-Connected Walls

Do one of the following:

Click the *PolySolid* tool button ( ) on the *3D Solids* toolbar.

Choose *Polysolid* in the *Model > 3D Solids* menu.

Type *polysolid* in the command bar.

You are prompted: Start point or Height/Width/Justification/separateSolids/Dynamic/<Object>:

Specify the start point of the wall.
You are prompted: Arc/Distance/<Next point>:
The footprint of the first wall displays dynamically.

Move the cursor in the desired direction until a polar tracking line displays.
The current length displays in the *Length* dynamic entry field,
Optionally hit the Shift key to lock the polar tracking line. Locked tracking lines turn red. Hit the shift key again to unlock.

Do one of the following:

Specify the endpoint.

Type a value in the *Length* dynamic entry field, then press Enter.

The footprint of the next wall displays dynamically.

Repeat steps 3 and 4 to create the next walls.
The footprints of all walls display.

Right click or press Enter to stop adding walls.
The height of the walls displays dynamically in the *Height* field.
The default width of the walls is indicated in the *Width* field.
The current justification of the walls is indicated in the Tips widget.

Repeatedly hit the Ctrl key to cycle the justification options: *Right*, *Center* or *Left*.
The justification of all walls is adjusted accordingly.

(option) Hit the TAB key, then enter a value in the *Width* field then hit the TAB key again.

Do one of the following to define the height of the walls:

Press Enter to accept the default height.

Type a value in the *Height* field, then press Enter.

NOTES The width and height of polysolids is saved to the *PSOLWIDTH* and *PSOLHEIGHT* system variables, which are saved in the drawing. The previously used values are remembered the next time the *PolySolid* command is executed.

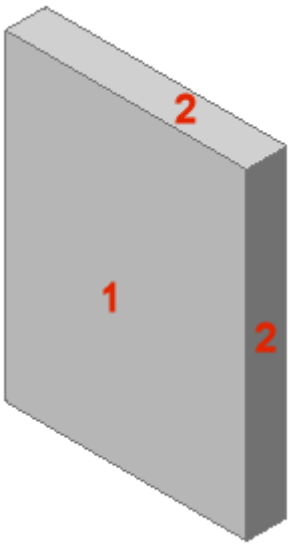
If *Separate Solids* is *OFF*, a single multisegment wall is created. If *Separate Solids* is *ON* a chain of bisector L-connected solids is created.
The *Separate Solids* can be chosen in the initial command prompt of the *PolySolid* command. The setting is remembered the next time the *PolySolid* command is executed.

Connecting Solids

Since solids, as any other BIM entity, are created as native 3D solids, any of the 3D Solids Editing commands or Direct Modeling commands can be used to modify solids.

The BimConnect command ( ) creates an L-connection (bisector or parallel) between two solids.

The BimDrag command ( ):



drags major faces (1):

: Drag Solid = move the entire solid

: Drag Face = modify the thickness of the solid

: Copy Solid = creates a parallel copy of a solid or copies part of a solid, defined by a detected boundary (if the *Boundary Detection* option of the SELECTIONMODES system variable is checked).

Drags minor faces (2):

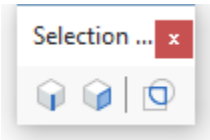
: Drag face = modify the length or height dynamically or by specifying a value

: Connect = connect to the major face of another solid. Creates T-connections between solids; connects solids to floor and roof slabs.

The procedures below use the Quad cursor menu.

Creating L-Connections

Make sure SELECTIONMODES = 0



Select the first solid.
The solid highlights.

Hover over the second solid.

When the solid highlights, choose *BimConnect* ( ) in the *BIM* command group of the Quad.
The selected solids are connected: the default connection type is *L-bisector*.
If TIPS is ON, the *Tips* widget displays.

You are prompted: Accept connection <Ok>/Next

Do one of the following:

Press Enter or right click to accept the default *L-bisector* connection.

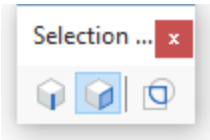
Hit the Ctrl key to cycle between *L-parallel* and *L-bisector* connections and press Enter or right click when the desired connection displays.
The 3D model updates dynamically when cycling the L-connection types.

NOTE L-connections can also be created between between non-vertical solids such as roof slabs.

Creating T-Connections

The below procedure allows to connect minor faces of a solid to the major faces of other solids, regardless what the orientation of the faces is. As a result you can use this option of the BIMDRAG command to T-connect solids, to connect the top or bottom face of a solid to a roof or floor slab or to connect the side faces of a slab to solids.

Make sure the *Select Faces* option of the SELECTIONMODES system variable is selected.



Move the cursor over the minor face of the first solid at the end that needs to be connected.
Hit the TAB key to highlight obscured faces.
Click when the face highlights.

Repeat the previous step to select more faces.
To select the last face, go to the next step.

Move the cursor over the face.
If necessary, hit the TAB key to highlight an obscured face.

When the face highlights, choose *BimDrag* ( ) in the *BIM* command group of the Quad.
The selected face moves dynamically with the cursor.
If TIPS is ON, the *Tips* widget displays.

: Drag dynamically.

: Connect

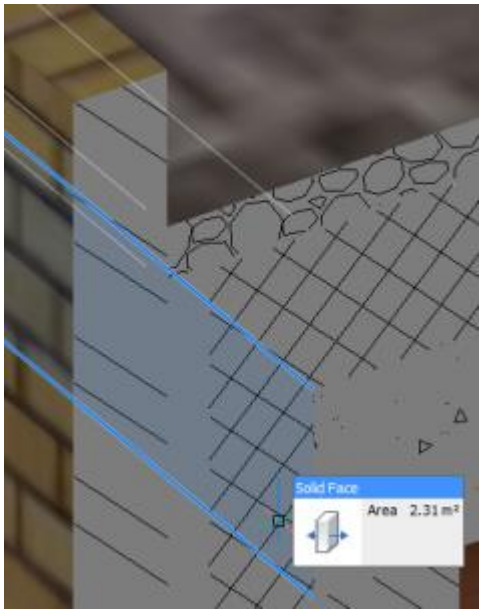
Press the Ctrl key.
Prompts you: Select one or more 3d solids to connect <ENTER to auto-select>:

Select the solid(s) to connect to.
The selected face is connected to the nearest face of the selected solid(s).

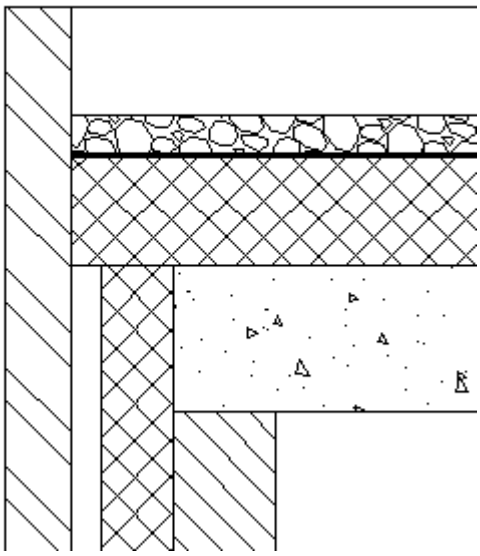
Right click or press Enter to accept the connection.

TIP Hover over the minor face of the solid, then choose *Connect with Nearest* ( ) in the *Model* command group of the Quad.

Creating T-connections ply-by-ply



3D Model



Calculated Section

To control the connection of composition plies between solids and and between solids and slabs, you can execute BimDrag on a ply in a clipped BimSection if *Display Composition* of the solids is *ON*.

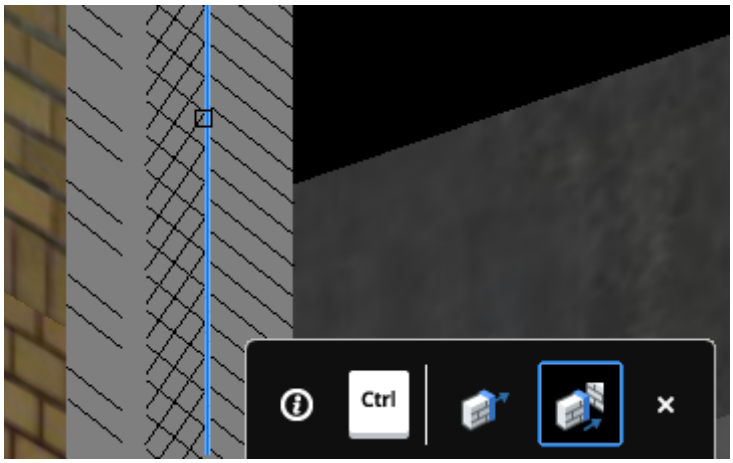
To create a T-connection ply-by-ply in a section, do the following:

Place the cursor over the face of the ply you want to connect. If necessary, repeatedly press the TAB-key to select an obscured face.

When the ply face highlights, choose *BimDrag* ( ) in the Quad.

Hit the Ctrl-key once.
Prompts you: Select one or more 3d solids to connect <ENTER to auto-select>:

Move the cursor over the sectioned multiply solid you want to connect to. The lines between the plies highlight.



Click when the desired line highlights.

Right click to confirm.
The ply of the first solid is connected to the ply of the second solid. The connected ply is subtracted from the second solid.

Modifying the Thickness

Make sure the *Select Faces* option of the SELECTIONMODES system variable is selected.

Move the cursor over a major face of the solid.

When the face highlights, choose *BimDrag* ( ) in the *BIM* command group of the Quad.
If TIPS is ON, the *Tips* widget displays.

: Drag Solid

: Drag Face = modifying the thickness of the solid

If necessary, hit the Ctrl key to select Major Faces = OFF ( ).
The selected face of the solid is dragged.
The dynamic dimension field displays the relative displacement of the face (= with respect to its original position).

(option) To modify the thickness incrementally, move the face in the desired direction, then type a value and press Enter.

(option) To set the new overall thickness of the solid, hit the TAB key, then type a value and press Enter.
When hitting the TAB key, most likely, the other major face of the solid is selected as the reference face for the dynamic dimension. However, if a major face of another solid highlights, hit the TAB key again.

NOTES When hitting the TAB key, most likely, the other major face of the solid is selected as the reference face for the dynamic dimension. However, if a major face of another solid highlights, hit the TAB key again. Repeatedly hitting the TAB key cycles all parallel major faces of all solids in the current view.

If the thickness of solid, that has a composition with a locked thickness applied, is modified, the solid solid displays in red. Execute the BimUpdateThickness command to re-apply the composition to such solid.

Modifying the Height of a single wall

Make sure the *Select Faces* option of the SELECTIONMODES system variable is selected.

Move the cursor over the top face of the wall.
If necessary, hit the TAB key to highlight an obscured face.

When the face highlights, choose *BimDrag* ( ) in the *BIM* command group of the Quad.
The selected face moves dynamically with the cursor.
If TIPS is ON, the *Tips* widget displays.

: Drag dynamically.

: Connect

The dynamic dimension fields displays the incremental height.

Do one of the following:

Specify a point in the model, e.g. snap to a point.

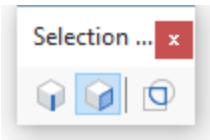
Type a value in the dynamic dimension field and press Enter.

Hit the TAB key to use another reference face, e.g. the bottom face of the solid, then type a value in the dynamic dimension field.

Modifying the height of multiple walls

Using the SelectAlignedFaces command (Quad > Select:) it is easy to select the top face of all solids that lie in the same horizontal plane.

Make sure the *Select Faces* option of the SELECTIONMODES system variable is selected.



Move the cursor over the top face of one of the walls.
If necessary, hit the TAB key to highlight an obscured face.

When the face highlights, choose *Select Aligned Faces* ( ) in the *Solid/Face Selection* command group in the Quad.
All faces lying in the plane of the selected face are selected.

Choose *BimDrag* ( ) in the *BIM* command group of the Quad.
The selected faces move dynamically with the cursor.
If TIPS is ON, the *Tips* widget displays.

: Drag dynamically.

: Connect

The dynamic dimension fields displays the incremental height.

Do one of the following:

Specify a point in the model, e.g. snap to a point.

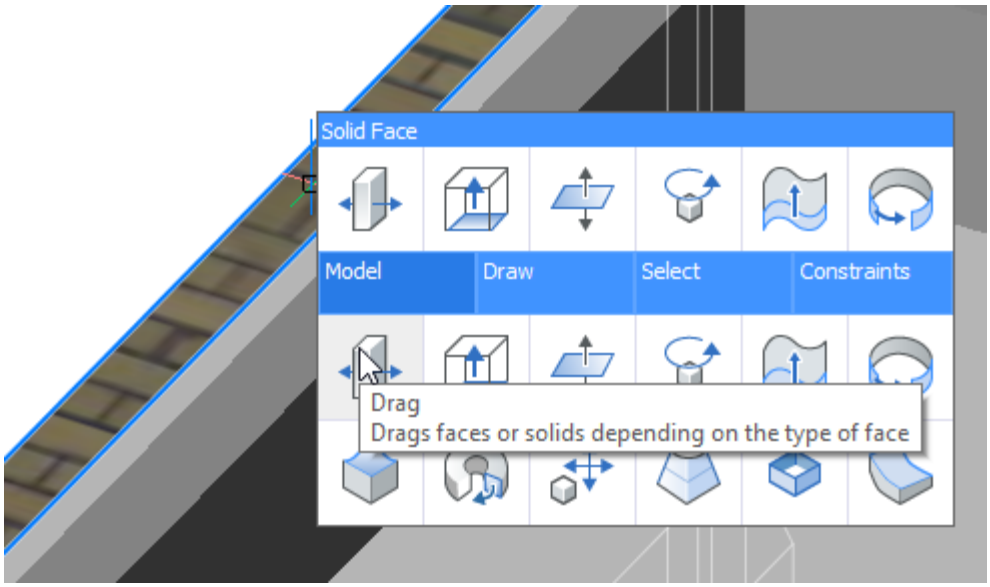
Type a value in the dynamic dimension field and press Enter.

Hit the TAB key to use another reference face, e.g. the bottom face of the solid, then type a value in the dynamic dimension field.

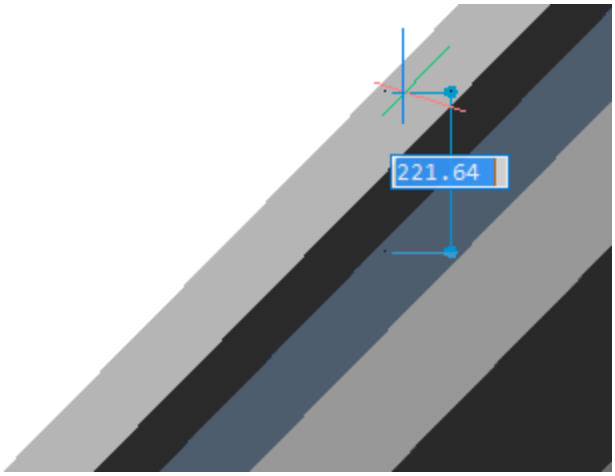
Modifying the height ply-by-ply

When the Display Composition property of a solid is set, and the *Select Faces* option of the SELECTIONMODES system variable is selected, the height or length of the composition plies can be modified separately.

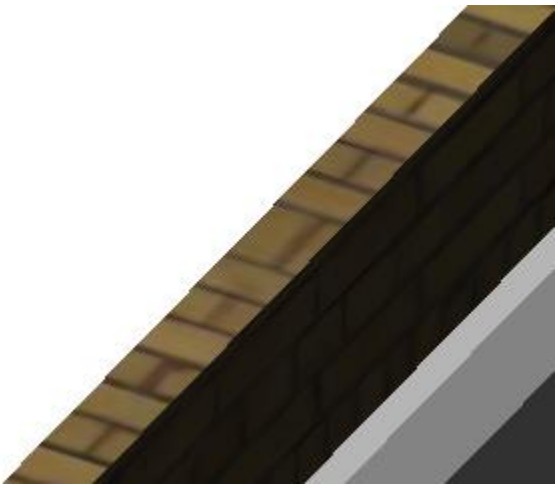
Move the cursor over the ply you want to edit and choose *BimDrag* ( ) in the *Model* command group of the Quad when the ply highlights.



Type value in the dynamic distance field.



Press Enter to accept the distance.



Setting the Distance between Parallel Solids

Make sure the *Select Faces* option of the SELECTIONMODES system variable is selected.

Move the cursor over a major face of the solid to be moved.

When the face highlights, choose *BimDrag* ( ) in the *BIM* command group of the Quad.
You are prompted: Select point for dragging or [Switch to copy solid] <drag solid>:
If TIPS is ON, the *Tips* widget displays.

The enitre solid moves.
The dynamic dimension field displays the relative displacement of the solid (= with respect to the original position of the selected face).

Repeatedly hit the TAB key until the desired reference face on a parallel solid highlights.
Press and hold the Shift key while hitting the TAB key to cycle reference faces in reversed order.
The dynamic entry field displays the distance with respect to the currently highlighted reference face.

Type a value in the dynamic entry field, then press Enter.

Creating a Parallel Copy of a Solid

Make sure the *Select Faces* option of the SELECTIONMODES system variable is selected.

Move the cursor over a major face of the solid to be copied.

When the face highlights, choose *BimDrag* ( ) in the *BIM* command group of the Quad.
You are prompted: Select point for dragging or [Switch to copy solid] <drag solid>:
If TIPS is ON, the *Tips* widget displays.

Hit the Ctrl key twice to select the *Copy Solid* option ( ).
You are prompted: Select point for dragging or [Switch to drag face] <copy solid>:

Do one of the following:

Specify a point.

Repeatedly hit the TAB key until the desired reference face on a parallel solid highlights.
Press and hold the Shift key while hitting the TAB key to cycle reference faces in reversed order.
The dynamic entry field displays the distance with respect to the currently highlighted reference face.
Type a value in the dynamic entry field, then press Enter.

Creating a Partial Copy of a Solid

Make sure the *Enable boundary detection* option of the SELECTIONMODES system variable is selected.

Move the cursor over the part of the solid to be copied.

When the face highlights, choose *BimDrag* ( ) in the *BIM* command group of the Quad.
You are prompted: Select point to copy entity:

Do one of the following:

Specify a point.

Repeatedly hit the TAB key until the desired reference face on a parallel solid highlights.
Press and hold the Shift key while hitting the TAB key to cycle reference faces in reversed order.
The dynamic entry field displays the distance with respect to the currently highlighted reference face.
Type a value in the dynamic entry field, then press Enter.

Creating Slabs

Creating Floor Slabs

Make sure that:

The *Enable Boundary Detection* option of the SELECTIONMODES system variable is ON.

The bottom faces of the surrounding walls are in the XY-plane of the current coordinate system.
If necessary move the origin of the coordinate system.

Dynamic dimensions are ON.

Move the cursor inside the surrounding walls.

When the boundary highlights, choose *Extrude* ( ) in the *Model* command group in the Quad.
The slab is extruded dynamically.

Do one of the following:

Type the thickness of the slab in the dynamic dimension field.

Pick a point.

Creating Pitch Roof Slabs

Make sure that:

Dynamic UCS is ON.

Dynamic dimensions (DYN) is ON; it is recommended to have the *Tracking dynamic dimensions* option of the DYNMODE system variable set.

Entity Snap Tracking (STRACK) is ON.

Endpoint entity snap ( ) is ON.

Launch the POLYSOLID command.
You are prompted: Start point or [Height/Width/Justification/Entity/Separate solids/Dynamic] <Entity>:

Move the cursor over the outside face of a wall which is perpendicular to the roof slab you want to create.

When the face highlights, hit the Shift key to lock the dynamic UCS.

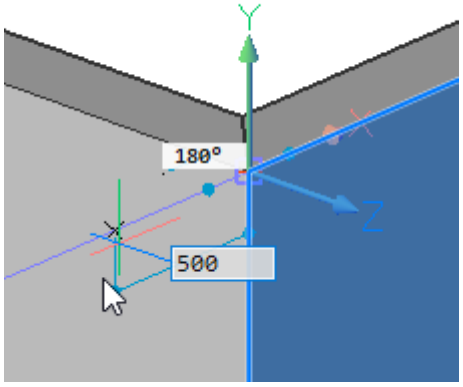
Do one of the following:

To create the roof at the top edge of the wall, sna to the exterior top corner of the wall.

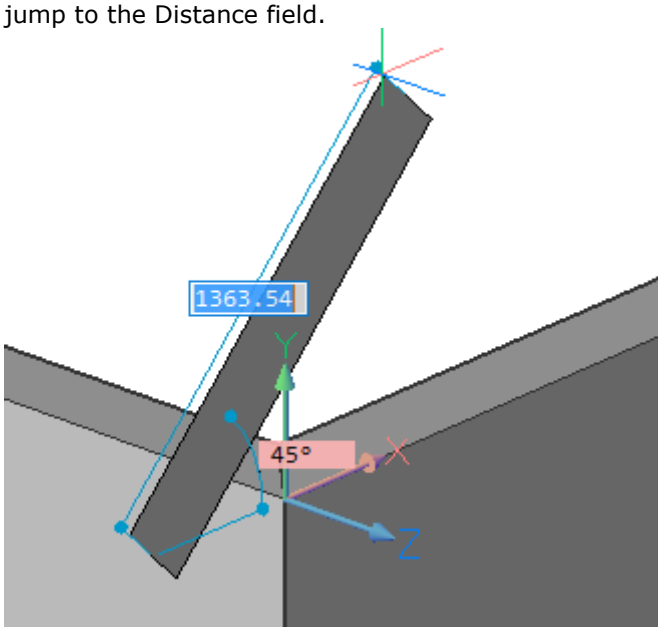
To create a roof overhang (eave):

Move the cursor over the exterior top corner of the wall to require a snap tracking point.

When a small red cross indicates the tracking point is acquired, move the cursor to the outside and type the overhang distance in the dynamic entry field.

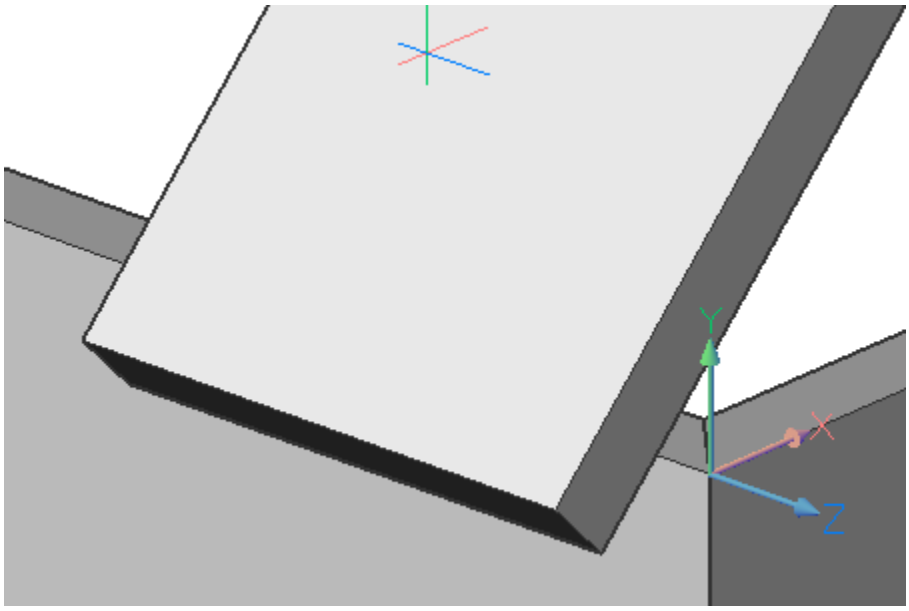


Hit the TAB key to activate the Angle field, then type the pitch angle and hit the TAB key again to jump to the Distance field.



Specify the the height of the roof slab, the press Enter twice: the first Enter creates the the footprint of the roofslab, the second Enter interrupts the creation a second solid.
The roof slab is extruded dynamically.

Specify a point to define the length of the extrusion.



Repeat the previous steps to create more roof slabs.
Use BIMDRAG to connect the roof slabs.

Windows and Doors

Commands: BIMINSERT, BIMWINDOWUPDATE, BIMREPOSITION, BIMWINDOWCREATE

The *BimInsert* command inserts windows and doors in any face of a 3D solid. Using Dynamic UCS, a window will automatically align with the 3D Solid Face it is inserted on. The *BimInsert* command also allows to insert a copy of an insert.

The *BimWindowUpdate* command updates the opening made by a window or door in a solid, in case the definition changed in such way that the opening was not correctly updated automatically.

The *BimReposition* command allows to reposition an insert in the face of a solid.

The BimWindowCreate command creates a fully parametric window based on a closed contour.

Windows and doors are native .dwg files. A 3D Solid in the window drawing on a layer named *BIM_SUBTRACT* will be used to automatically create an opening in the 3D solid, keep the window in its position in the wall, and keep the opening associative with the window. All solids in the window/door model that extend the subtractor solid and overlap the wall solid, e.g. a sill or a lintel, will be subtracted from the wall solid also.

TUTORIALS

Inserting Windows

Inserting Windows and Doors

To control the position of the window in the face of the solid (wall), you can use the following drawing aids:

Dynamic Dimensions: During placement of a window or door on a 3D Solid face, dynamic dimensions are created, starting from the middle of each edge of the bounding rectangle of the opening to the nearest parallel edge on the face of the 3D Solid. Make sure Dynamic Dimensions are active by checking the DYN field in the status bar and Dynamic UCS is active by checking the DUCS field in the status bar.
The Dynamic UCS aligns the window to the face of the solid. The edge of the face by which the cursor enters the face defines the orientation of the X-axis. Hit the Shift key to temporarily lock the UCS, which allows to use reference points outside the face of the solid. Hit the shift key again to unlock.

Temporary Tracking points: Click the mouse wheel or press the *Temporary Tracking Points* tool button ( ) on the *Entity Snaps* toolbar to start specifying temporary points.

OSNAPZ: If this system variable is ON, the Z-value of any entity snapping point is replaced by the current value of the ELEVATION system variable. If ELEVATION=0, entity snapping points are forced to lie in the face of the solid. OSNAPZ is toggled by the *Ignore entity snap elevation* tool button ( ) on the *Entity Snaps* toolbar. The state of the tool button (pressed or not) indicates the current value of OSNAPZ.

Entity snap to negative Z-values: Press the *Entity snap to negative-z* tool button ( ) on the *Entity Snaps* toolbar to enable entity snaps on points that lie behind the selected face. If OSNAPZ is ON, the point is projected onto the selected face.

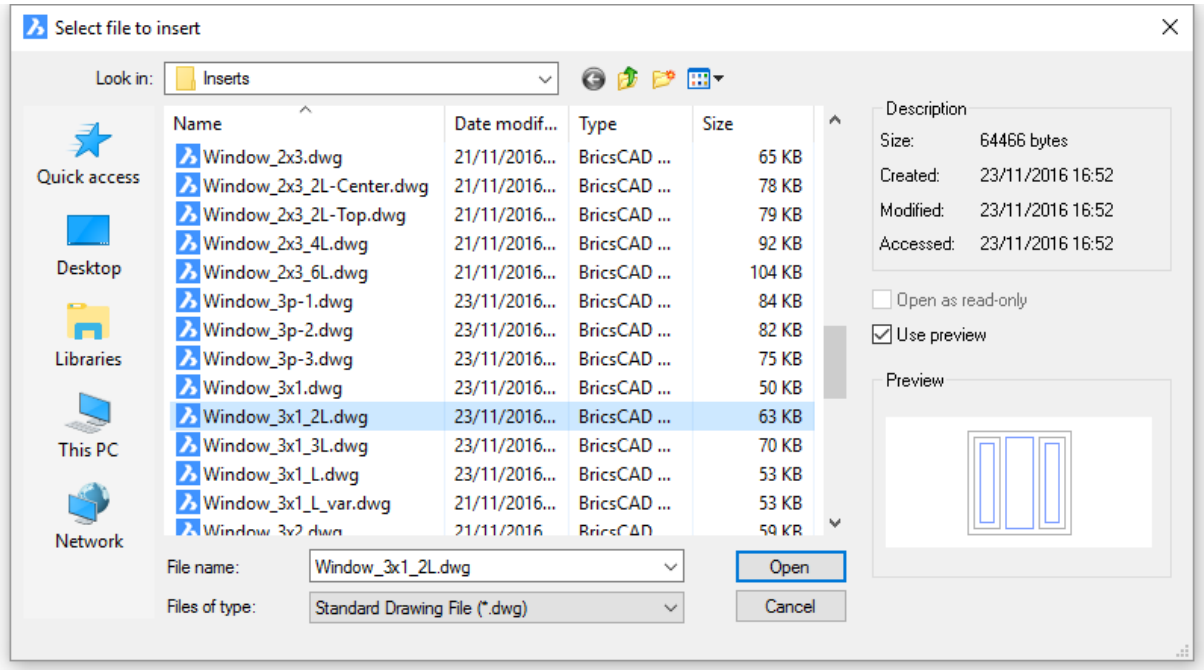
Do one of the following:

Click the *BimInsert* tool button ( ) on the *BIM* toolbar.

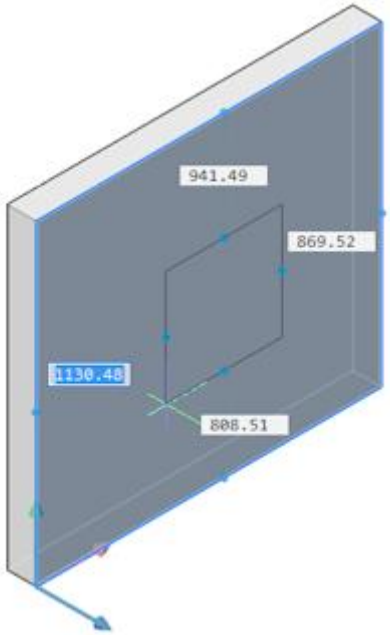
Choose *BimInsert* in the *BIM* menu.

Type *biminsert* in the command bar.

A file dialog box displays:

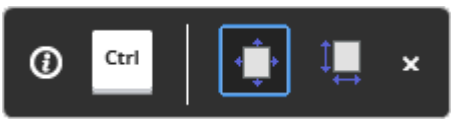


Select file, then either double click or click the *Open* button.
The selected insert is attached to the cursor.
When you move over a face of solid, the insert aligns with the face under the cursor.



The distances from the start-, end-, top- and bottom face of the wall to the placement rectangle of the insert display dynamically.

If TIPS is *ON*, the *Tips* widget displays at the bottom of the screen:



(option) Specify the size of the insert.

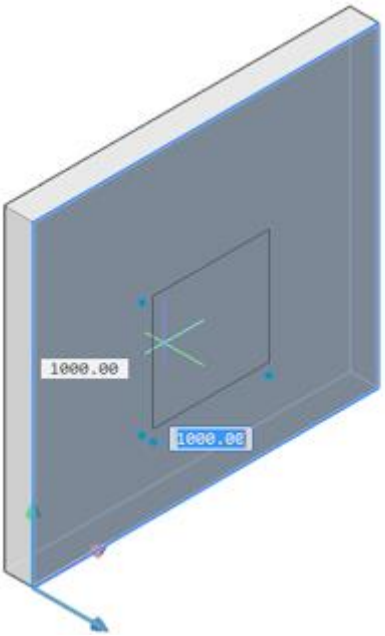
Do one of the following to switch to the *Edit inserted entity* mode ( ).

Hit the Ctrl-key.

Type *E*, then pres Enter.

Choose *Edit inserted entity* in the prompt menu.

The *Width* and *Height* field indicate the current dimensions of the insert.



Specify the width and height of the insert.
Hit the TAB key to switch between the dynamic dimension fields.

When done, do one of the following to switch to the *Select insertion point* mode ( ):

Hit the Ctrl-key.

Type *D*, then press Enter.

Choose *Done* in the prompt menu.

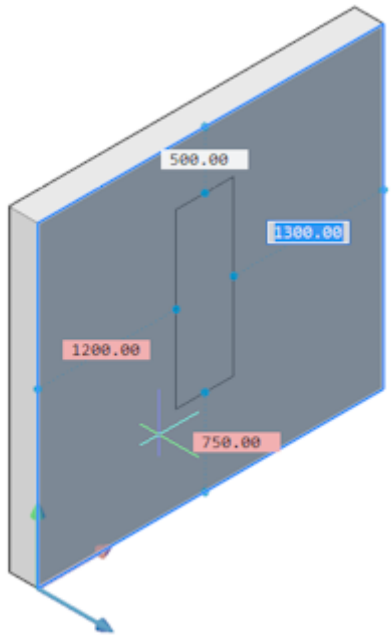
The distances from the start-, end-, top- and bottom face of the wall to the placement rectangle of the insert display dynamically.

Use one of the following methods to insert the window/door:

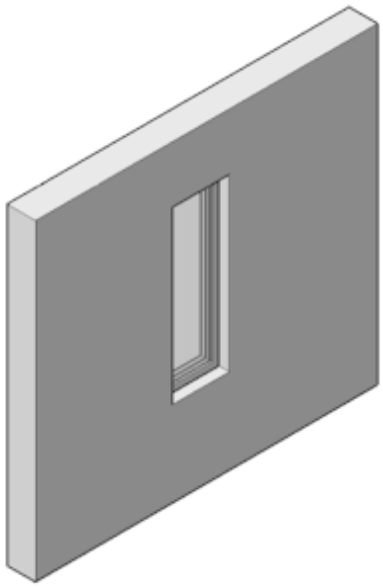
Method 1: Use dynamic dimensions fields:

Hit the TAB key to select one of the horizontal dynamic dimension fields, then type a value and hit the TAB key to lock the dimension.
The locked dimension turns red.

Hit the TAB key to select one of the vertical dynamic dimension fields, then type a value and hit the TAB key to lock the dimension.
The locked dimension turns red.



Press Enter or click to place the window.



Method 2: Specify a point

Snap to a point using an Entity Snap or any other method to specify a point in the drawing (e.g. Entity snap tracking or Temporary tracking points).
If OSNAPZ = ON, the point will be projected onto the face of the solid.

Click the *Entity snap to negative-z* button ( ) on the *Entity Snaps* toolbar to snap to points that lie behind the selected face.

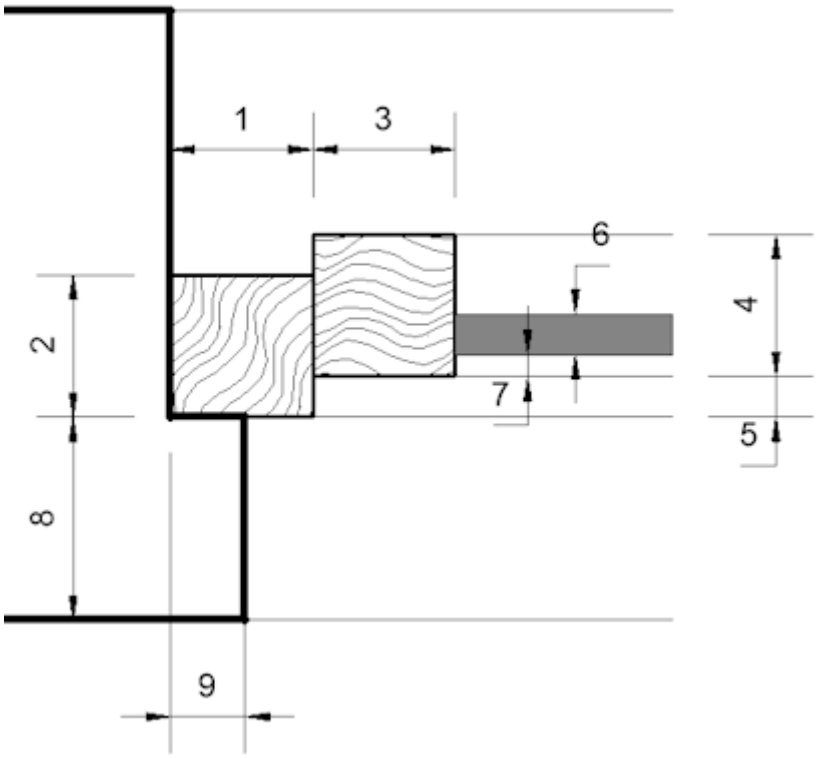
Editing Parameters

If parameters are defined in the window/door source drawing, these parameters can be used to edit each instance of a window/door separately.

Select the door/window in the drawing.
The current values of the various parameters display in *Parameters* section of the Properties Bar.

Parameters	
Name	Window_3x1_2L:1
File	
W	1000 mm
H	1000 mm
Rebate	0 mm
PlacementDepth	100 mm
FrameThickness	60 mm
FrameDepth	60 mm
LeafThickness	60 mm
LeafDepth	60 mm
PaneDepth	20 mm
PaneOffset	20 mm
LeafOffset	20 mm

In the sample windows, the following parameters are available:



W: Width = width of the opening in the wall*; for interior doors the W parameter defines the width of the door leaf.

H : Height = height of the opening in the wall*; for interior doors the H parameter defines the height of the door leaf.

1: FrameThickness

2: FrameDepth

3: LeafThickness

4: LeafDepth

5: LeafOffset

6: PaneDepth

7: PaneOffset

8: Placement Depth = distance between the face of the wall and the fixed window frame

9: Rebate

* If the Rebate = 0 (zero), the overall width and height of the window equals the opening in the wall.

Select a parameter, then type a value in the settings field.

NOTE If multiple frames or doors are selected they can be edited simultaneously.
Shared parameters only display in the Properties Bar.
The settings field of parameters which have different values, reads *Varies*.

Moving an insert

The following methods are available to move an insert in the face of a solid.

The BIMREPOSITION command repositions an insert in the face of a solid.

The BIMDRAG command allows to move an insert vertically or horizontally, if applied to an inner face of the opening made by the insert.

When, for some reason, the opening in the solid is not healed automatically, the BIMWINDOWUPDATE command recalculates the opening for the selected insert(s).

Repositioning an Insert

Do one of the following:

Click the *Reposition Insert* tool button ( ) on the *BIM* toolbar.

Choose *Reposition Insert* in the *BIM* menu.

Type *bimreposition* in the command bar.

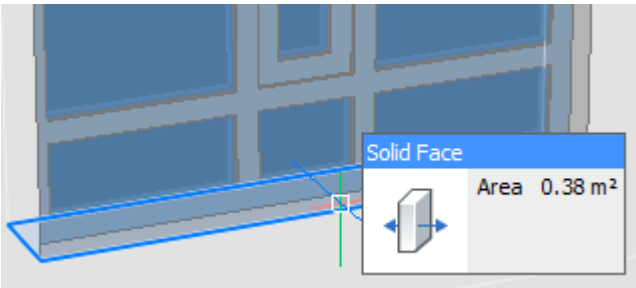
Hover over an insert and choose *Reposition Insert* ( ) in the Quad.

(Except when launching the command from the Quad)
Select an insert in the model.
You are prompted: Select insertion point or [Edit inserted entity] :

Move the cursor over the outside face of the wall, then press the Shift-key to lock the dynamic UCS to the face of the wall.
Proceed with the Inserting Windows and Doors procedure from step 2 on.

Moving an Insert Vertically

Move the cursor over the top or bottom inner face of the opening made by the insert.



When the face highlights, choose *BimDrag* ( ) in the *BIM* command group in the Quad.
The insert moves vertically.

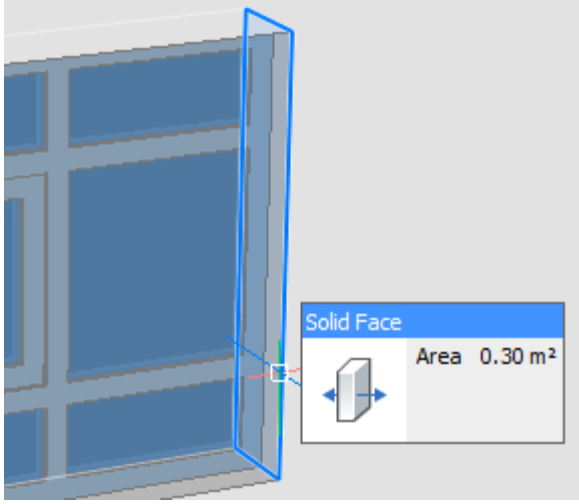
Do one of the following:

Specify a point.

Type a distance in the dynamic dimension field.
Optionally hit the TAB key to select a different reference face.

Moving an Insert Horizontally

Move the cursor over the left or right inner face of the opening made by the insert.



When the face highlights, choose *BimDrag* ( ) in the *BIM* command group in the Quad.
The insert moves horizontally.

Do one of the following:

Specify a point.

Type a distance in the dynamic dimension field.
Optionally hit the TAB key to select a different reference face.

Copying Inserts

Copying an insert in a solid

Use the Copy command to place copies of an insert in the same solid. The *Repeat* option of the copy command can create a series of copies at the same interval.

Make sure that the movement vector for the copy is parallel to the face of the wall. If the orientation of the wall, in which the source window or door is placed, is not parallel to either the X- or Y axis of the current UCS, use the *Face* option of the UCS command to align the UCS.

Inserting a copy of an insert

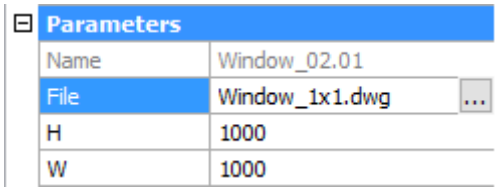
Hover over the insert. When the insert highlights, choose *BimInsert* ( ) in the *BIM* command group in the Quad.
You are prompted: Select insertion point or [Edit inserted entity] :

Proceed with the Inserting Windows and Doors procedure from step 2 on.

Replacing Inserts

Select the insert.
The properties of the selected insert display in the Properties Bar.

Select the File property in the Parameters group.



Click the *Browse* button ( ).
A *File* dialog displays.

Select the replacement file for the insert, then double click or click the *Open* button on the *File* dialog.
The insert is replaced.

Creating Windows and Doors

The creation of custom BricsCAD BIM inserts, such as windows and doors, consists of the following steps:

Creating the 3D geometry.

Optionally add 2D entities.

Add 3D constraints and parameters to modify the dimensions of the insert.

The inserts are created in the XY plane of the WCS.

Creating the 3D Geometry

The 3D geometry is composed of:

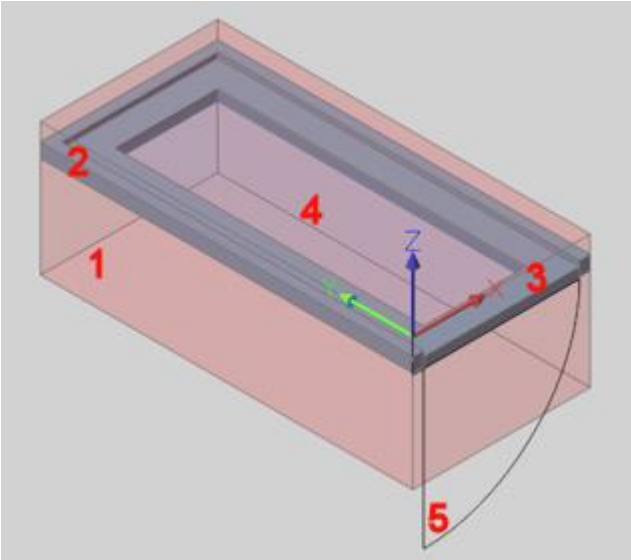
The subtractor solid (1)

The window/door frame (2)

The opening leaf(s) (3)

(Glass) panes (4)

2D stuff: opening indicators, 2D representation (5)



Step 1: Create the file

Choose *New* in the *File* menu.
The *Select Template* dialog displays.

Select the *BIM-Window-metric.dwt* or *BIM-Window-imperial.dwt* template drawing.
The Bim-Window templates contain the layers needed to create the geometry of the window or door.

Set the *BIM_SUBTRACT* layer current.

Create a closed polyline (e.g. a rectangle) that represents the shape of the opening for the insert.

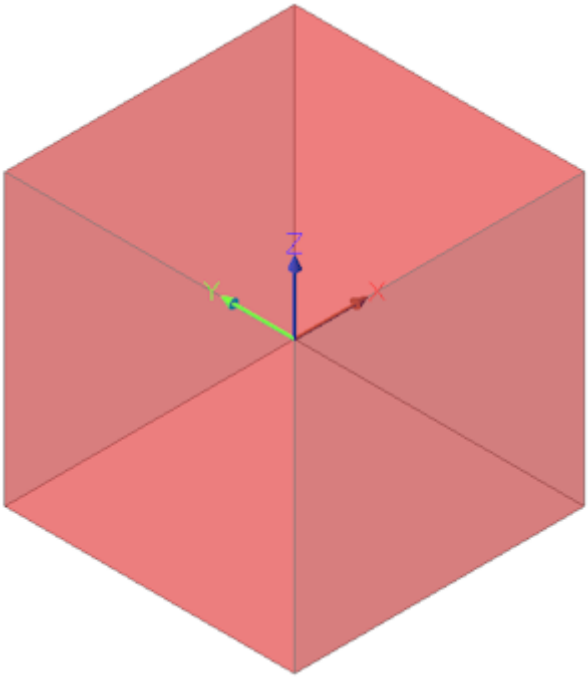
Save the file in *..\Support\Bim\Windows* subfolder of the Roamable Root folder.
By default this is:
C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\V16x64\en_US\Support\Bim\Windows

Step 2. Create the subtractor solid

The subtractor solid will be used to create the opening for the insert.

Create the outline of the solid, e.g. a rectangle, in the XY plane of the World Coordinate System (WCS).
The lower left corner of the rectangle lies at the origin (0,0,0) of the WCS.

Use the DMEXTRUDE command to extrude the rectangle in the negative Z direction.
The extrusion height must be higher than the largest width of the wall solids you want to place the insert in.



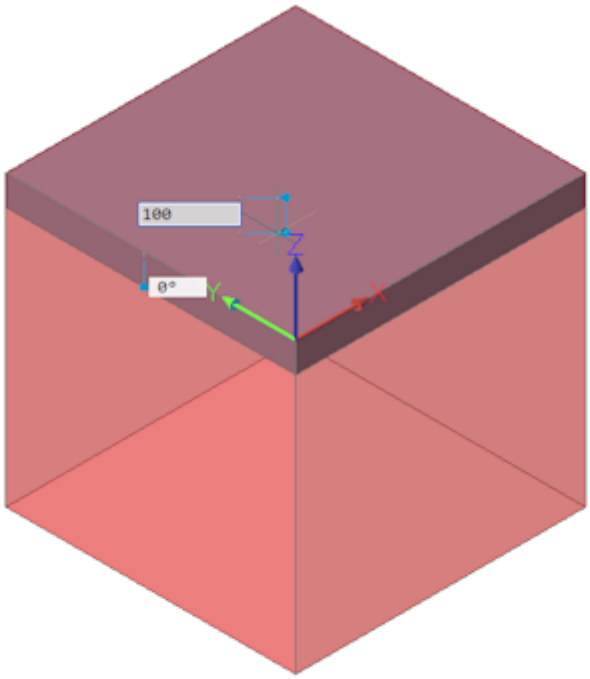
Step 3: Create the fixed frame

The outline of the fixed frame coincides or is parallel (in case of a rebate) with the outline of the subtractor solid.

Set the *WINDOW_FRAME* or *DOOR_FRAME* layer current.

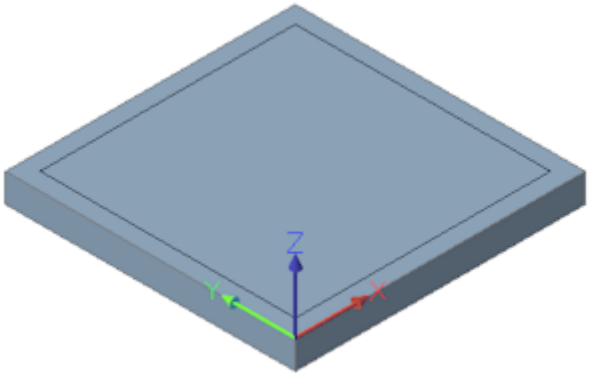
Use the DMEXTRUDE command to extrude the top face of the subtractor solid.
Hit the Ctrl-Key to select the *Create* option.

Create the extrusion in the negative Z-direction.
The height of the extrusion equals to the thickness of the fixed frame.



Switch the display of the *BIM_SUBTRACT* layer off.

Use the OFFSET command ( ) to create a parallel offset of the outline of the fixed frame. The distance of the offset equals the width of the fixed frame face.

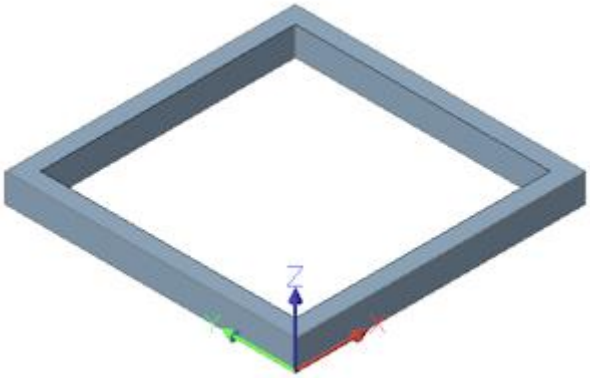


Use the DMEXTRUDE command to create an opening in the fixed frame.

Step 4: Create the opening leaf

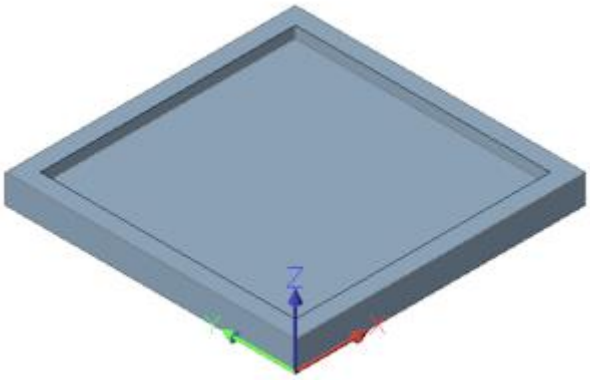
The outline of the opening leaf coincides with the inside faces of the fixed frame.

Use the UCS command to set the XY-plane at the bottom (inside) face of the fixed frame.

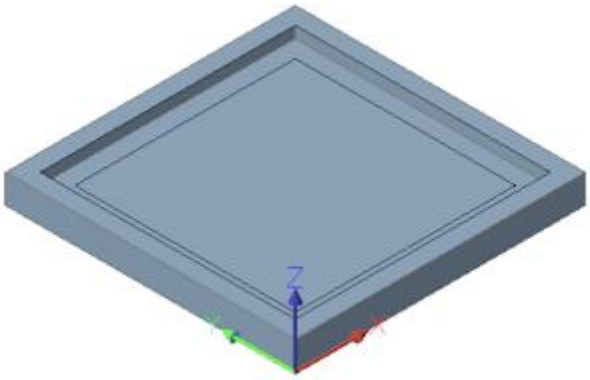


Set the *Enable Boundary Detection* option on the *Selection Modes* toolbar.

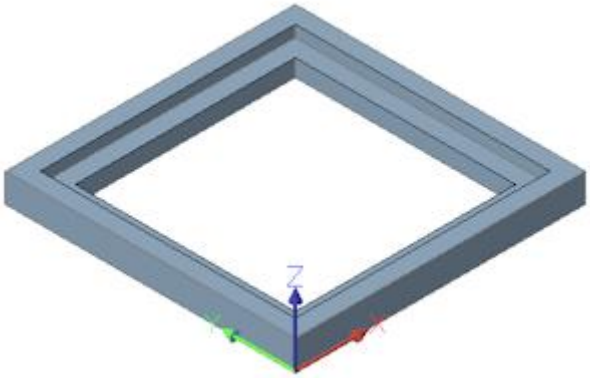
Use the DMEXTRUDE command to extrude the inside boundary of the fixed frame.
The height of the extrusion equals the thickness of the opening leaf.



Use the OFFSET command ( ) to create a parallel offset of the outline of the opening leaf.
The distance of the offset equals the width of the opening leaf face.



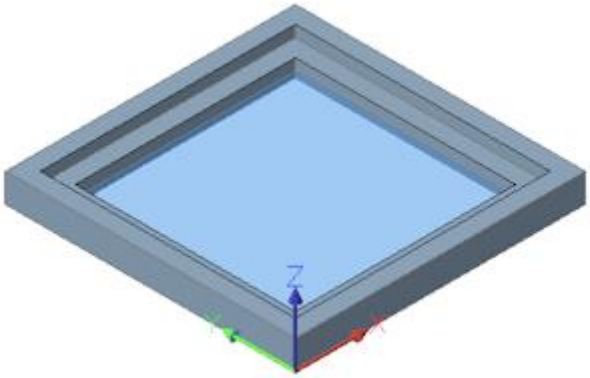
Use the DMEXTRUDE command to create an opening in the opening leaf.



Step 5: Create the glass panes

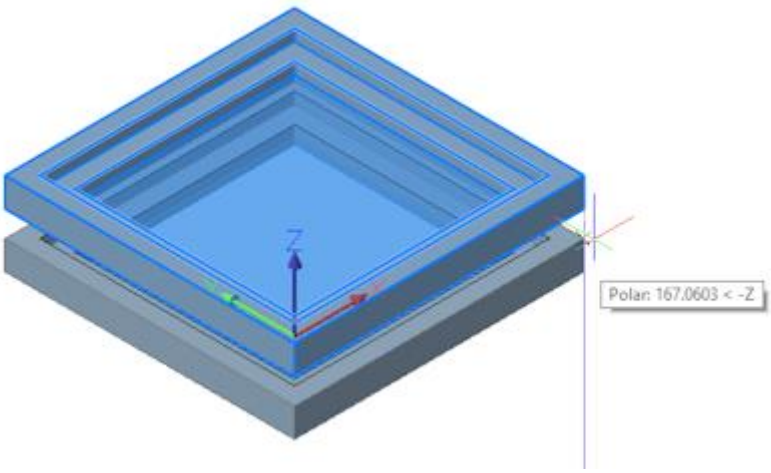
Set the *GLASS* layer current.

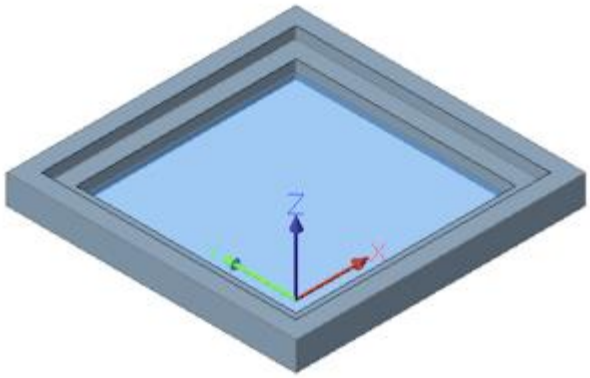
Use the DMEXTRUDE command to extrude the inside boundary of the opening leaf.
The height of the extrusion equals the thickness of the glass pane.



Restore the WCS (World Coordinate Sytem).

(option) Move the solid in the negative Z-direction if the outside face of the fixed frame does not coincide with the outside face of the solid (wall) the insert will be placed in.





Adding 2D Entities

Optionally, windows and doors drawings can contain 2D symbols. These 2D symbols will either replace the actual 3D Solid section of the window or door, or be added to the 3D Solid section, depending on their layer:

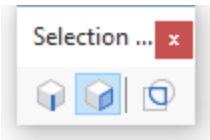
symbols on a layer with prefix *BRX_2D_* will be used to replace the geometry resulting from the actual section,

symbols on a layer with prefix *BRX_2D+_* will be added to the section result. Only those 2D entities in a plane parallel to the section plane are considered.

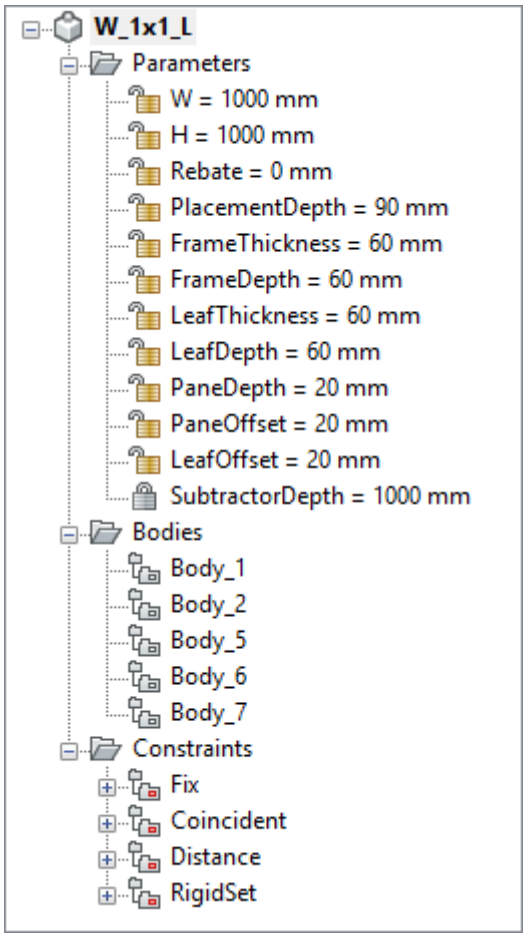
Adding 3D Constraints

See 3D constraints.

Choose *Enable selection of 3D solid faces* in the *Selection Modes* toolbar.



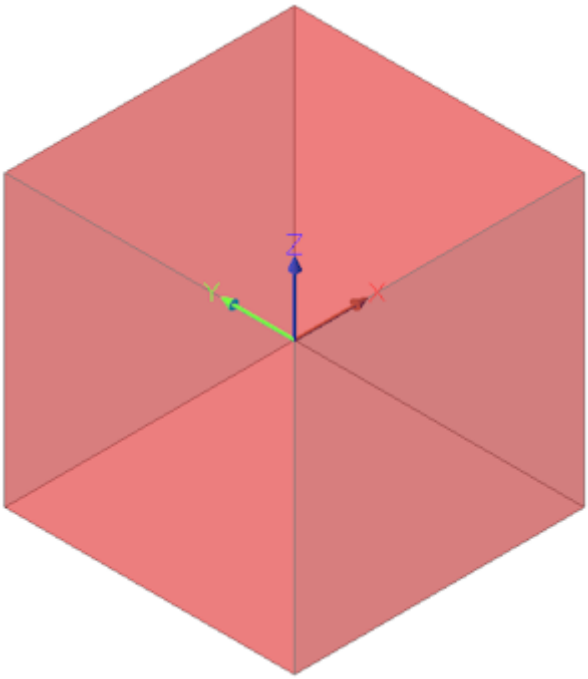
Open the Mechanical Browser to control and manage the 3D constraints and parameters.



The window we created in the above procedure is composed of three solids: subtractor, fixed frame, opening leaf, glass pane.

To apply Fix constraints

A FIX ( ) constraint is applied to the top, front and left face of the subtractor solid.



Hover over the face you want to apply the constraint to.

Choose *Add Fixation* ( ) in the *3D Constraints* command group in the Quad.

To apply Coincident constraints

Coincident constraints are applied to all coinciding faces of the composing solids, e.g. side faces of the fixed frame and the subtractor solid.

Select the first face
The face hightlights.

Hover over the second face.
Hit the TAB key to highlight an obscured face.

When the second face highlights, choose *Add Coincidence* ( ) in the *3D Constraints* command group in the Quad.

To apply Distance constraints

Distance constraints are used to control the overall dimensions (width and height) of the insert and the dimensions of the face and thickness of the fixed frame and opening leaf and the thickness of the glass pane.

Select the first face
The face hightlights.

Hover over the second face.
Hit the TAB key to highlight an obscured face.

When the second face highlights, choose *Add Distance* ( ) in the *3D Constraints* command group in the Quad.
You are prompted: Enter distance value:

Do one of the following:

Press Enter or right click to accept the current value.

Type a value in the command bar.

Using Parameters

Parameters are used to control:

- the width and height of the frame.
- the face and thickness of the fixed frame and the opening leaf.
- the thickness of the glass pane.
- the depth of the opening leaf with respect to the fixed frame.
- the depth of the fixed frame with respect to the outside face of the wall.

...

The value of the parameters can be edited for each insert separately (see Editing the parameters of an insert).

To create a parameter

Right click the Component name in the Mechanical Browser.

Choose *Add New Parameter* in the context menu.

Select the parameter, then edit its properties in the settings grid.

Parameter	
Name	H
Expression	1000
Value	1000
Description	Window Height
Exposed	Auto
Units	mm

Name: Parameter name

Expression: Type a value or formula

Value: The current value of the parameter.

Description: An optional description of the parameter.

Exposed: Controls whether the parameter is available in the properties grid when the window is inserted in a model.
Choose an option:

Off: not available

On: always available

Auto: available only if the Expression field contains a value, not an expression.

Units: Specifies whether the parameter is linear, areal or volumetric.
Select the field, then click the arrow button and select an option.

Using parameters

Select a distance constraint.

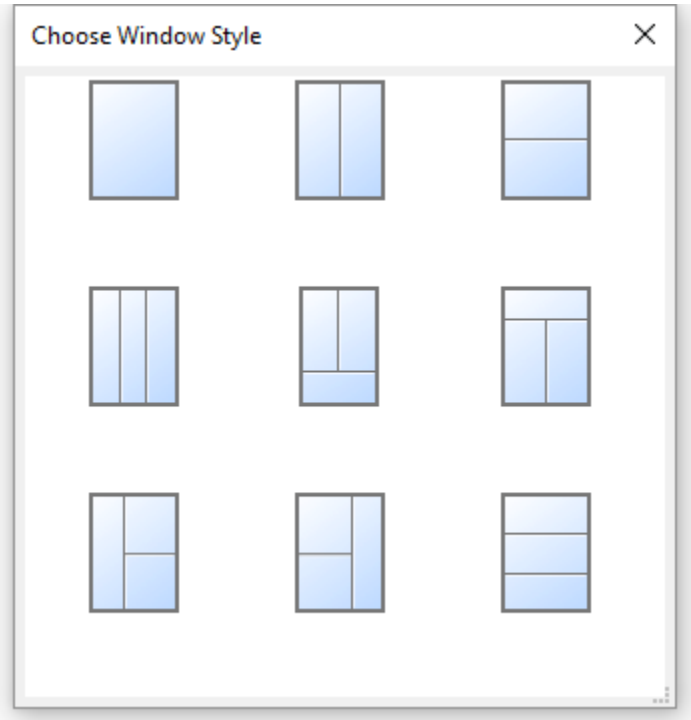
Use the parameters to compose an expression in the *Expression* field in the settings grid of the constraint.

Constraint	
Type	Distance
Name	Distance_8
Expression	W+2*Rebate
Value	1000 mm
Enabled	Yes
Exposed	Auto

Creating Standard Windows

The *BimWindowCreate* command allows to create standard windows from a closed 2D entity, a boundary or an existing opening in a solid.

The window style is selected in the *Window Style* dialog box:



To create a window using a closed 2D entity.

Closed polylines, circles, ellipses and closed splines are accepted.

Do one of the following:

Click the *BimWindowCreate* tool ( ) on the *BIM* toolbar, then select a single or multiple closed 2D entities.

Move the cursor over a closed 2D entity, then select *BimWindowCreate* tool ( ) in the *Model* command group in the Quad.

The *Window Style* dialog displays.

Pick a standard window style in the *Window Style* dialog.
The window is created.

(option) Edit the window parameters in the Properties Bar or the Mechanical Browser panel.

Parameters	
Name	Triple Panel Top Window:1
File	
Width	1200
Height	1500
LiningThickness	60
LiningDepth	60
MullionThickness	60
TransomThickness	60
FirstMullionOffset	0.5
FirstTransomOffset	0.5
GlazingDepth	30
GlazingOffset	30
InsertionDepth	50
LeftPanelThickness	60
LeftPanelDepth	60
RightPanelThickness	60
RightPanelDepth	60
TopPanelThickness	60
TopPanelDepth	60

To create a window using a boundary.

Make sure the *Enable Boundary Detection* option on the *Selection Modes* toolbar is selected.

Move the cursor inside an enclosed area in the XY-plane of the current UCS.

When the boundary hightlights, select *BimWindowCreate* tool ( ) in the *Model* command group in the Quad.
The *Window Style* dialog displays.

Pick a standard window style in the *Window Style* dialog.
The window is created.

(option) Edit the window parameters in the Properties Bar or the Mechanical Browser panel.

To create window in an existing opening.

Make sure the *Enable Selection of 3D Solid Faces* and *Boundary Detection* options on the *Selection Modes* toolbar are selected.

Move the cursor over the face of the solid and choose *Align UCS* ( ) in the *Model* command group in the Quad.
You are prompted: Enter an option [Flip/Xrotate/Yrotate] <accept>:

Press Enter to accept the UCS.

Move the cursor inside the opening.
The boundary of the opening highlights.

Select *BimWindowCreate* tool ( ) in the *Model* command group in the Quad.
The *Window Style* dialog displays.

Pick a standard window style in the *Window Style* dialog.
The window is created.

(option) Edit the window parameters in the Properties Bar or the Mechanical Browser panel.

Editing Solids

Splitting Solids

Command: BIMSPLIT

The *BimSplit* command either splits segmented solids into separated solids automatically or allows to split a solid using a selection of cutting faces.

To split a solid automatically

Make sure that the value of SELECTIONPREVIEW is 2 or 3.

Do one of the following:

> Click the *BimSplit* tool button ( ) on the *BIM* toolbar.
>
> Choose *Split* in the *BIM* menu.
>
> Type *bimsplit* in the command bar, then press Enter.

You are prompted: Select solid for auto splitting or cutting faces for manual splitting:

Place the cursor over a face of the solid.
The face highlights.

Without moving the mouse, hit the TAB key until the complete solid highlights, then click.
You are prompted:
Entities in set: 1
Select solid for auto splitting or cutting faces for manual splitting:

Right click.
The selected solid is split.
The number of parts is reported in the command window.

Using the Quad

Make sure the *Select Faces* option ( ) of the SELECTIONMODES system variable is switch off.

Place the cursor over a face of the solid.

When the solid highlights, choose *Split* ( ) in the *BIM* command group of the Quad.
The solid is split.
The number of parts is reported in the command window.

To split a solid using cutting faces

Make sure that the value of SELECTIONPREVIEW is 2 or 3.

Do one of the following:

> Click the *BimSplit* tool button ( ) on the *BIM* toolbar.
>
> Choose *Split* in the *BIM* menu.
>
> Type *bimsplit* in the command bar, then press Enter.

You are prompted: Select solid for auto splitting or cutting faces for manual splitting:

Place the cursor over a face of the solid.
Hit the TAB key to highlight an obscured face.
When the face highlights, click to select the face.
You are prompted:
Entities in set: 1
Select solid for auto splitting or cutting faces for manual splitting:

(option) Repeat step 4 to select more faces.

Right click to stop selecting faces.
The solid is split using the selected faces as cutting planes.
The number of parts is reported in the command window.

BIM Database

Project Database and Library Database

A BIM Project consists of dwg entities, classified as Building Elements, to which Compositions can be attached. All Compositions being used in a project, are stored in the Project database. The Project database can be stored inside a dwg file, or in a separate file with extension *.bimlib:*

When the Project database is embedded in a dwg file, that dwg file contains the complete BIM Project.

When the Project database is stored as an external file, the BIM Project can consist of multiple dwg files, all using the same Project database.

To reuse compositions and building materials across multiple BIM Projects, a Library database can be specified, in addition to the Project database. The Library database is a *.bimlib* file, usually stored in a location which is common to several projects. When Compositions and Building Materials from the Library database are used in a BIM Project, they are automatically imported in the Project database. To maintain and extend BIM databases, users can drag Building Materials and Compositions from one database to the other, provided that they have write access to the destination database.

NOTE Sample databases are installed in the *Support\BIM* subfolder the Roamable root folder. When a BricsCAD update is installed, the BricsCAD User File Manager will prompt you whether to overwrite the sample databases or not. Therefore it is recommended to rename your library databases and/or save them at a different location. The location of the library databases is defined in the *Library DB* tab of the BIM Project Info dialog.

Materials and Compositions

A BIM database consists of Building Materials and Compositions. A Building Material contains information about one specific material: its appearance, cost, manufacturer, etc. A Composition contains information about the structure of a building element, by defining Composition layers. Each Composition layer refers to a Building Material, and has a certain thickness.

Flexibility

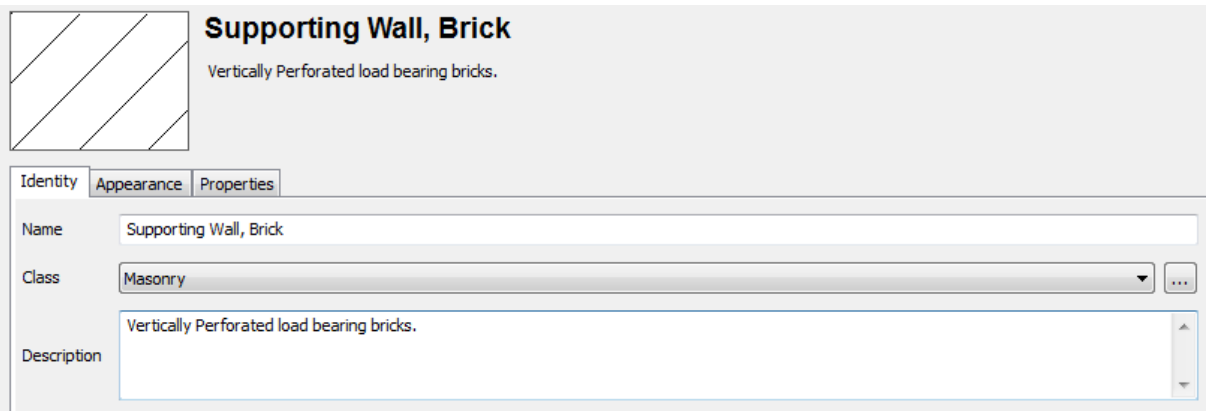
The same flexibility you have in modelling is available in attaching building information. Although Compositions are grouped into *Wall*, *Slab*, *Roof* and *Generic* Compositions, any type of Composition can be attached to any dwg entity. When attached to a 3D Solid, with specific geometrical properties, the thickness of the Composition is used to define the thickness of walls and slabs, and the cross section will show the wall or slab structure by applying the hatch patterns from the various layers on the section geometry. When a 3D Solid is assigned a Composition with a fixed thickness or a minimal thickness, and this thickness can not be applied to the geometry, the section appears in red to indicate the incompatibility between the selected Composition and the geometry of the solid.

BIM Compositions panel

All BIM database features are accessible from the dockable BIM Libraries panel.

Materials

A Material definition describes the appearance and properties of a Building Material.



Composition layers

A *Composition* describes the structure of a building element by an ordered set of composition layers. Each composition layer has the following fields:

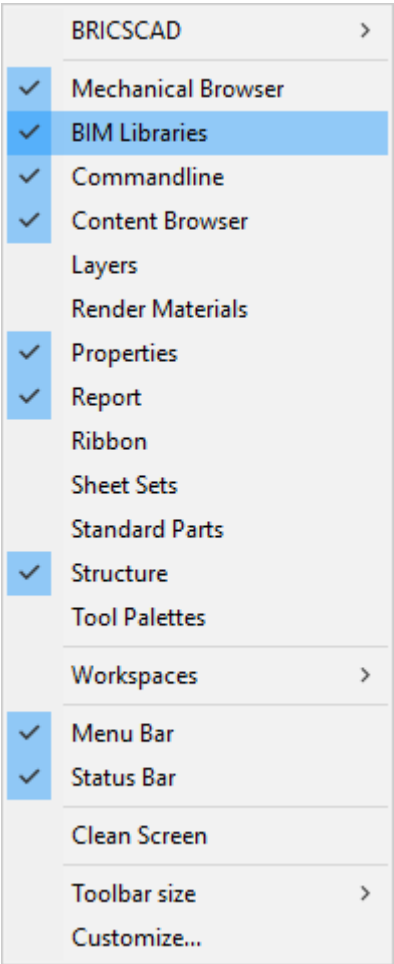
Pattern and Material Name: The Building Material for this layer.

Function: One of (None, Structure, Substrate, Insulation, Finish1, Finish2, Membrane). Currently this field serves for information purposes only.

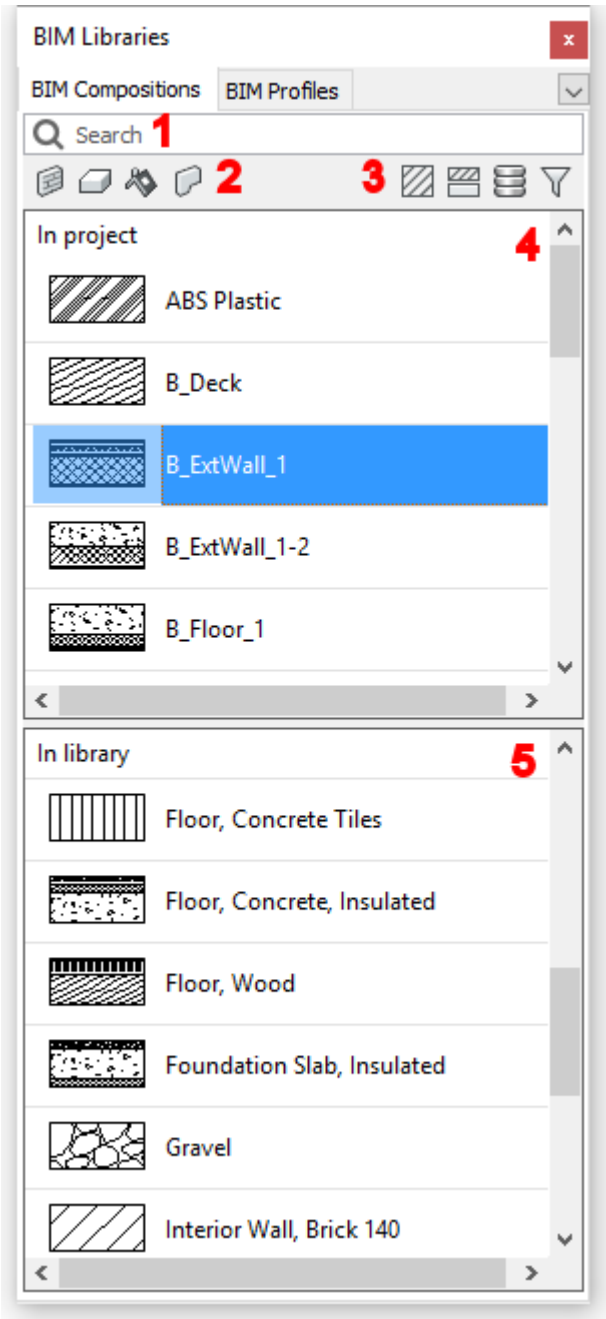
Thickness: Defines the thickness of the layer.

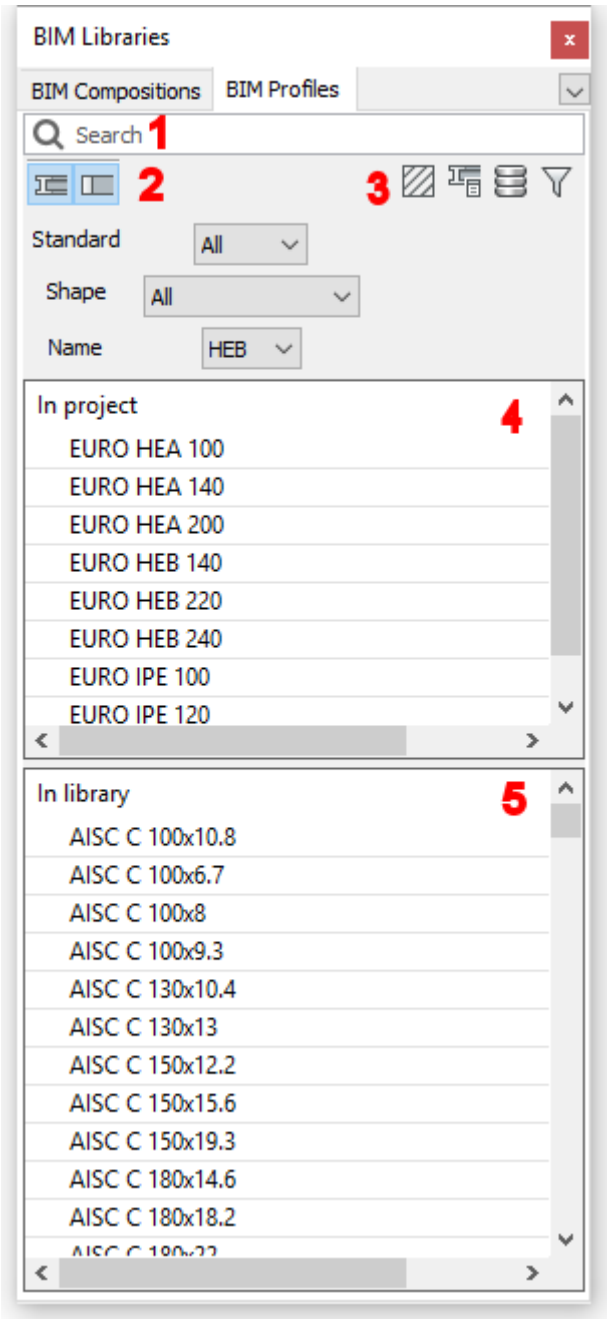
To open the BIM Libraries panel

Right click on any toolbar and choose *BIM Libraries* in the context menu.



The *BIM Libraries* panel is a dockable panel.


BIM Libraries
BIM Compositions
BIM Profiles
Search
1
2
3
In project
4
ABS Plastic
B_Deck
B_ExtWall_1
B_ExtWall_1-2
B_Floor_1
In library
5
Floor, Concrete Tiles
Floor, Concrete, Insulated
Floor, Wood
Foundation Slab, Insulated
Gravel
Interior Wall, Brick 140



Search field

Selection buttons

Click a button to add the selected category to the *In Project* and *In library* compositions or profiles list. Click again to remove the selected category.

Toolbar

Compositions or profiles of the selected categories in the *Project* database.

If the Filter button ( ) is pressed, the current filter is applied to the list.

Compositions or profiles of the selected categories in the *Library* database.

If the Filter button ( ) is pressed, the current filter is applied to the list.

The *BIM Libraries* panel allows to:

: Open the Physical Materials editor

: Open the Compositions editor

: Open the Profiles editor

: Open the BIM Project Info dialog

: If pressed, lists the compositions that match at least one the selected tags in the *Filter* tab of the BIM Project Info dialog.

: If pressed, includes *Wall* compositions in the list

: If pressed, includes *Slab* compositions in the list

: If pressed, includes *Roof* compositions in the list

: If pressed, includes *Generic* compositions in the list

: If pressed, includes *Structural* profiles in the list

: If pressed, includes *Generic* profiles in the list

Drag and drop compositions from the *Project* or *Library* database onto solids in the model.

Drag and drop compositions between the *Project* or *Library* database.

To copy compositions between the Project and Library databases.

The following procedure can be used in both the BIM Libraries panel and the Compositions editor.

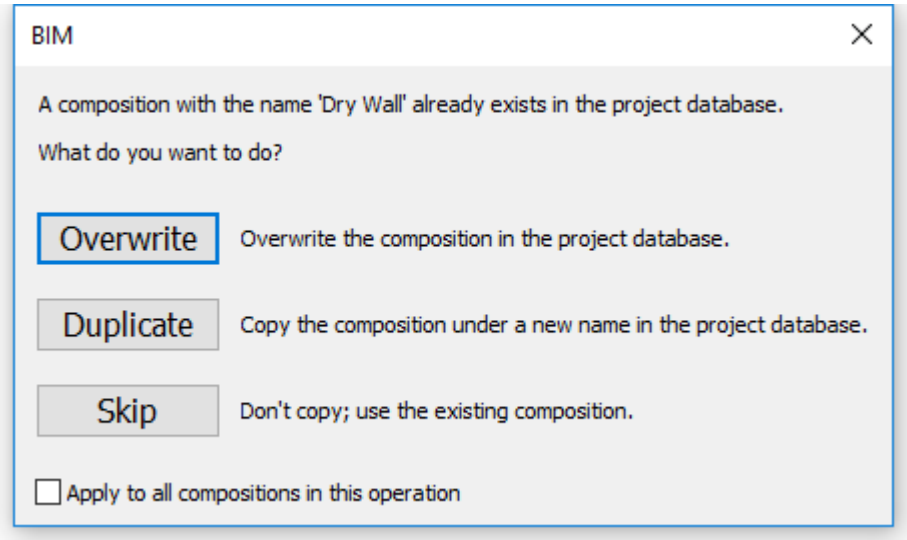
Select the compositions(s) in the source database.
Press and hold the Ctrl-key to select multiple compositions.

Do one of the following:

Drag the selection to the other database.

Right click and choose *Copy to Library* in the context menu.

If the value of the *CopyOverwrite* user preference is 0 (zero) a dialog box displays if the composition already exists in the target library:
Optionally check the *Apply to all compositions* in this operation option.



To Open the BIM Project Info Dialog

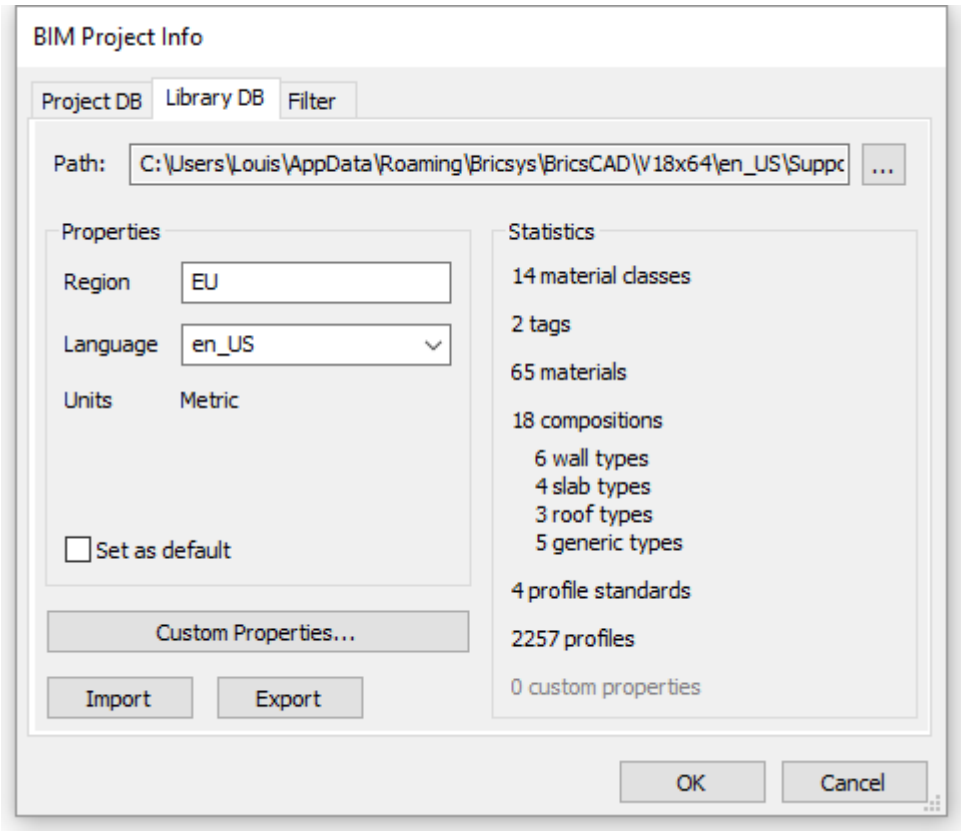
Click the *Project and Library Information* tool button ( ) on the BIM Compositions panel.

The *BIM Project Info* dialog has three tabs:

Project DB: The location, properties and content of the Project database.

Library DB: The location, properties and content of the Library database

Filter: The currently selected composition tags used to filter the compositions list on the BIM Compositions panel and the Building Materials editor.



To Set the Library Database

Select the *Library DB* tab page on the *BIM Project Info* dialog box.

Click the *Browse* button (...) at the right-hand side of the *Path* field.
A file dialog displays.

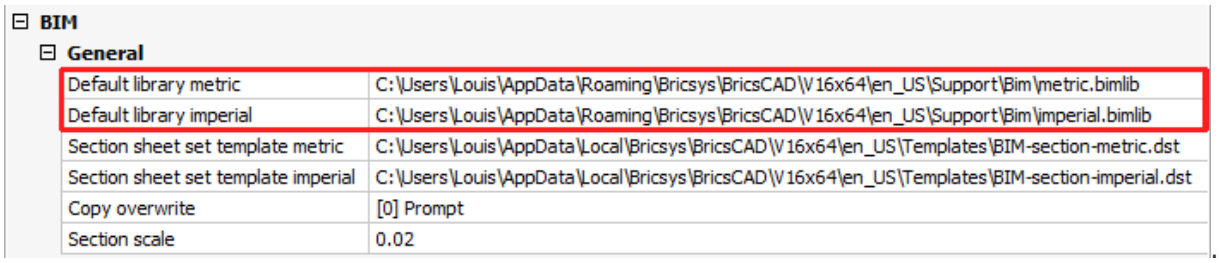
Select a BIM database file (*.bsyslib), then press the *Open* button.

(option) Check the *Set as default* option to set the selected database as default library database for new projects

NOTE The name and location of the default library databases is saved to the *DefaultLibraryMetric* and *DefaultLibraryImperial* user preferences.

Default libraries are used in new projects

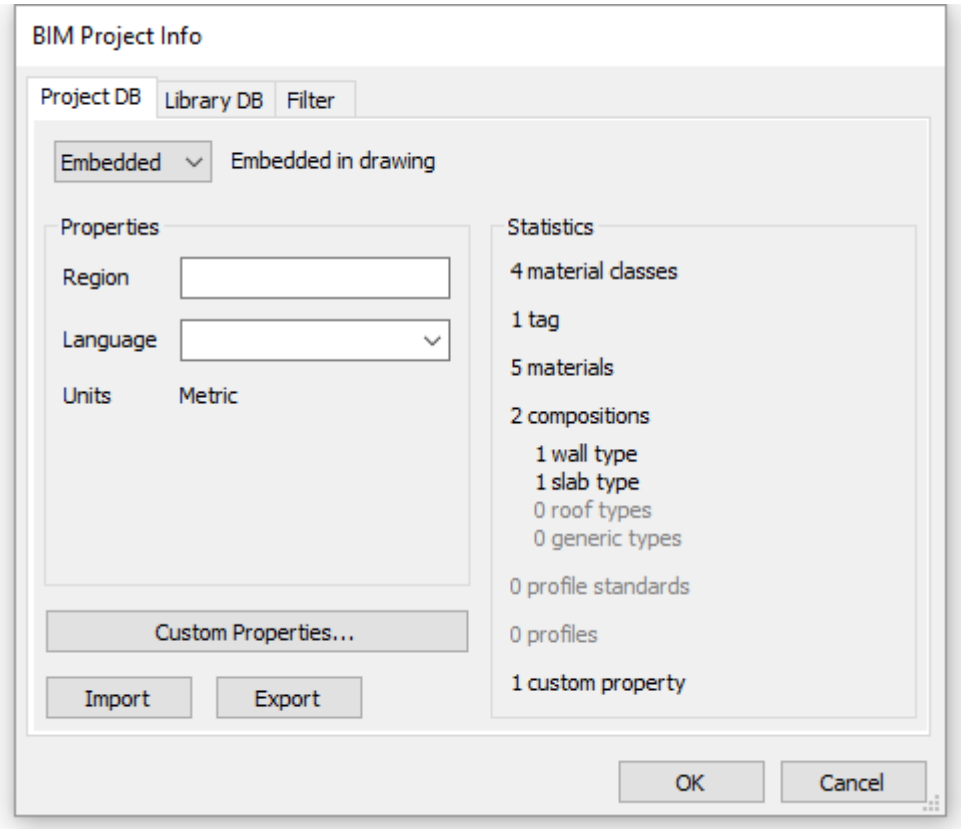
You can edit the user preferences in the Settings dialog under *BIM/General*.



⊟ **BIM**	
⊟ **General**	
Default library metric	C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V16x64\en_US\Support\Bim\metric.bimlib
Default library imperial	C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V16x64\en_US\Support\Bim\imperial.bimlib
Section sheet set template metric	C:\Users\Louis\AppData\Local\Bricsys\BricsCAD\V16x64\en_US\Templates\BIM-section-metric.dst
Section sheet set template imperial	C:\Users\Louis\AppData\Local\Bricsys\BricsCAD\V16x64\en_US\Templates\BIM-section-imperial.dst
Copy overwrite	[0] Prompt
Section scale	0.02

To Set the Project Database

Select the *Project DB* tab page on the *BIM Project Info* dialog box.



(option) Select *Embedded* to save the project library in the drawing file.

(option) Select *External*, then:

Click the *Browse* button ( ) at the right-hand side of the *Path* field.
A file dialog displays.

Select a BIM database file (*.bsyslib), then press the *Open* button.

To Set the Project Filter

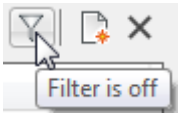
To each Building Material or Composition, any number of tags can be attached. Any string can be entered as a tag.

Each of the tags can be used as a filter on the database. There is a button at the top of each list ( ), to switch the filter on or off. If the filter is on, only Compositions and Building Materials that match at least one of the filter tags will be listed.

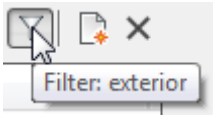
The *Filter* button ( ) has three states:

If the project filter is empty, the button is not active (*BIM Compositions* panel) or is not available (*Materials* and *Compositions* editors).

The project filter is defined, but not active.



The project filter is defined and active.



A tooltip shows the current project filter.

To set the project filter

Physical Materials

The *Physical Materials* dialog box allows to:

- Create new materials
- Edit materials
- Copy materials between the Project and Library databases
- Create single-ply compositions
- Delete materials

To Open the Physical Materials Editor

Do one of the following:

Type *blmaterials* in the command bar.

Click the *Physical Materials* button ( ) on the Compositions panel.

Click thee *Browse* button ( ) of the *Material* property of a component in the Mechanical Browser.

Component	
Name	support_final
Description	
File	C:\Program Files\Bricsys\E
Extension type	
Insert as	External component
Material	Aluminium ... x
Name	Aluminium
Description	
Density	2700.00 kg/m³

Adding Physical Materials

Physical materials can be created in either the Project or the Library database.
A new material can be created from scratch or as a copy of an existing material.

Adding a physical material from scratch

Click in either the *In Project* or the *In Library* material list.
Do one of the following:

Click the *Add Material* button ( ) on the *Physical Materials* dialog.
Right click and choose *Add Material* in the context menu.

A material named *New* is added in selected database.
Edit the newly created material.

Adding a physical material as a copy of an existing material

Select a material in either the *In Project* or the *In Library* material list.
Do one of the following:

Click the *Add Material* button ( ) on the *Physical Materials* dialog.
Right click and choose *Add Material* in the context menu.

A material named *<existing_material>_1* is added in selected database.
Edit the newly created material.

Editing Physical Materials

The various settings of a physical material definition are divided in three settings groups under three tabs on the Physical Materials dialog:

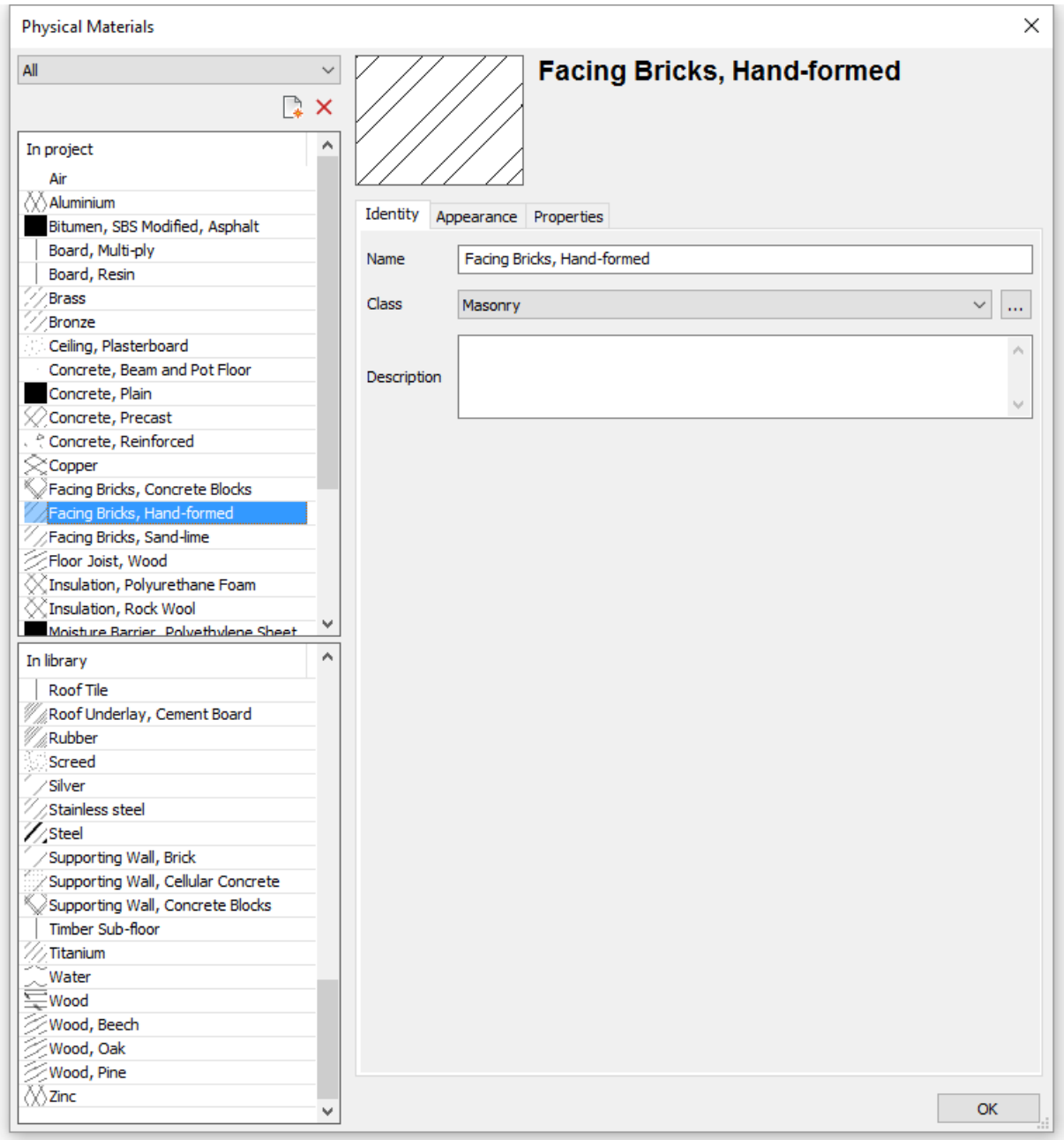
Identity: Name, Class and description of the material.

Appearance: The BIM Materials layers the composition is composed of.

Properties: Properties of the buidling material such as *Cost*, *Manufacturer*, *Thickness*, ...

Select the material in either the *In Project* or the *In Library* list on the Physical Materials dialog.

On the *Identity* tab page:

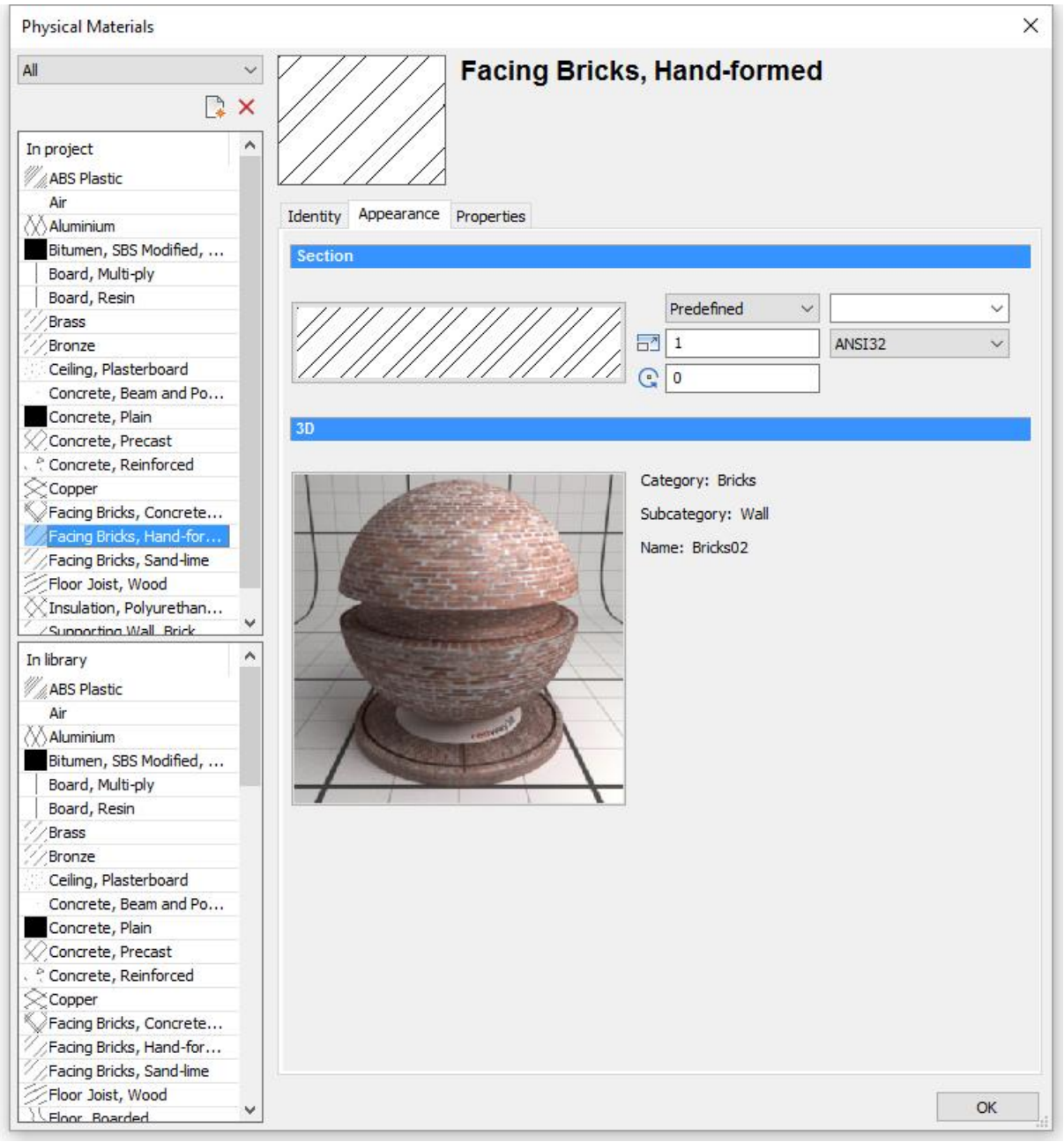


Type a name for the material.

Select a *Class* from the drop-down list.

(option) Type a description.

On the *Appearance* tab page:



***Section* settings:**

Click the *Hatch Type* button, then select the type of the hatch: *None*, *Solid*, *Linear*, *Predefined* or *Custom*.

Do one of the following to select a predefined or custom hatch pattern

Click the *Hatch* preview, then select the pattern in the *Hatch Pattern* dialog box.

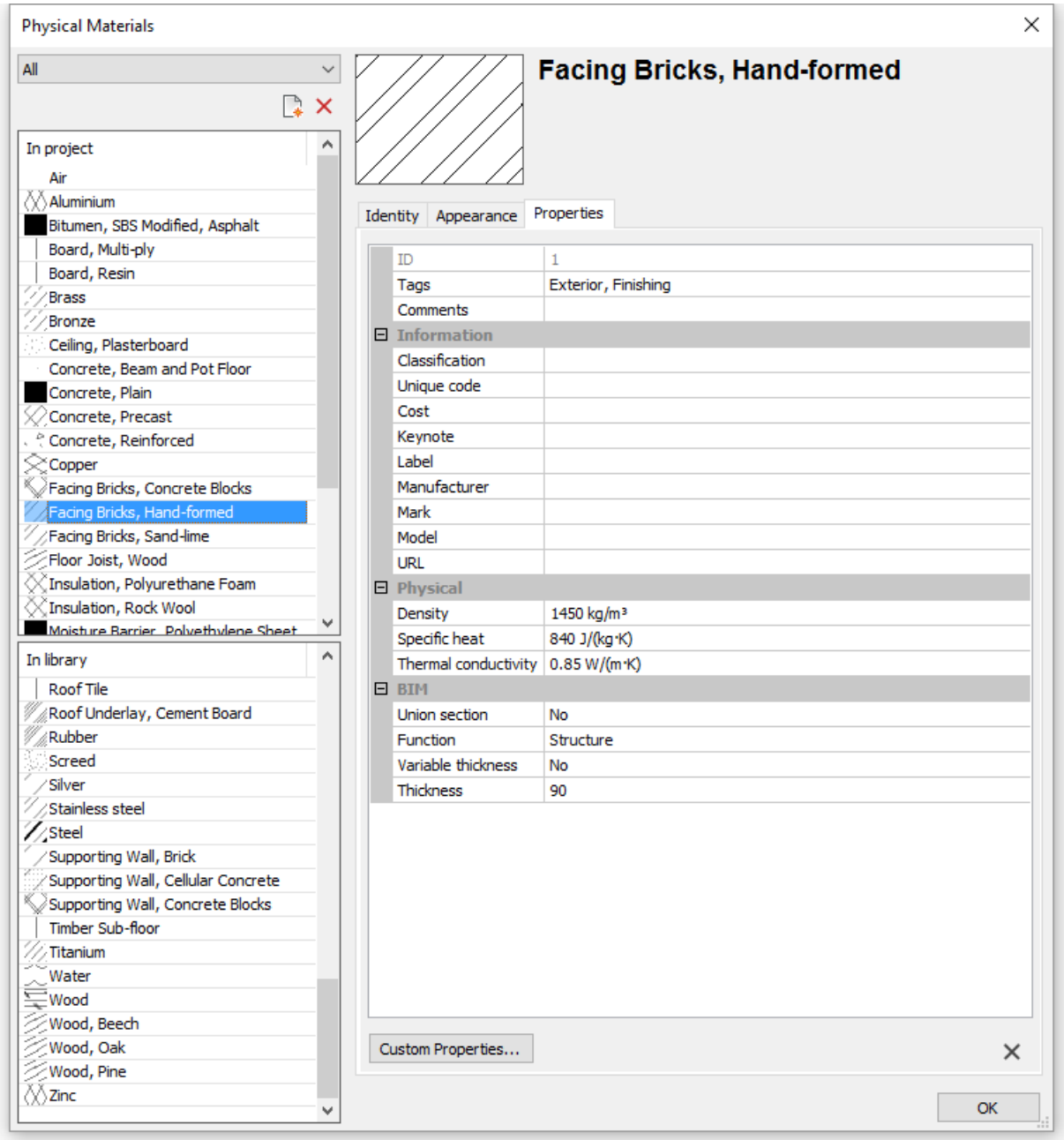
Click the *Hatch Pattern Name* button, then select the pattern in the list.

Define the *Scale*.
The final pattern scale in a section is the product of the *Scale* setting of the physical material and the value of the Hatch Scale property of the section plane entity.

Set the *Rotation* angle.

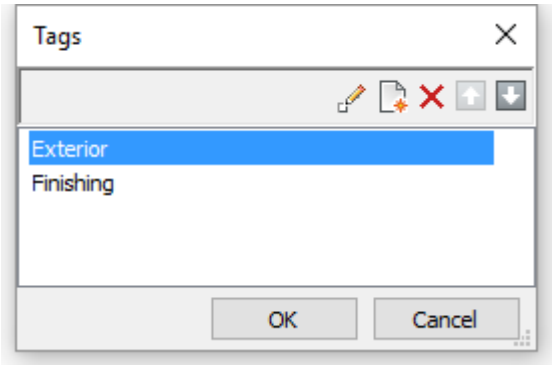
In the *Layer* field, either type a name or click the arrow button and select a layer in the list.

***3D* settings:**
Double click the preview image, then select a material in the *Render Material* dialog box.
On the *Properties* tab page:



Edit the *Tags* field: tags are separated by commas.
Tags are case sensitive.

Click the *Browse* button ( ) to open the *Tags* dialog.



Click the *New Item* button ( ) to add a new tag at the bottom of the list.

Select a tag in the list then click a tool button:

: edit the selected tag

: delete the tag

: move the tag up

: move the tag down

(option) Expand the *Information* properties and fill out the necessary fields.

(option) Expand the *Physical* properties and edit physical properties of the material.

Expand the *BIM* properties.

Union Section: Select *Yes* to remove connection lines in sections between different solids of this material. Choose *No* to keep the connection lines.

Select a *Function* from the drop-down list.

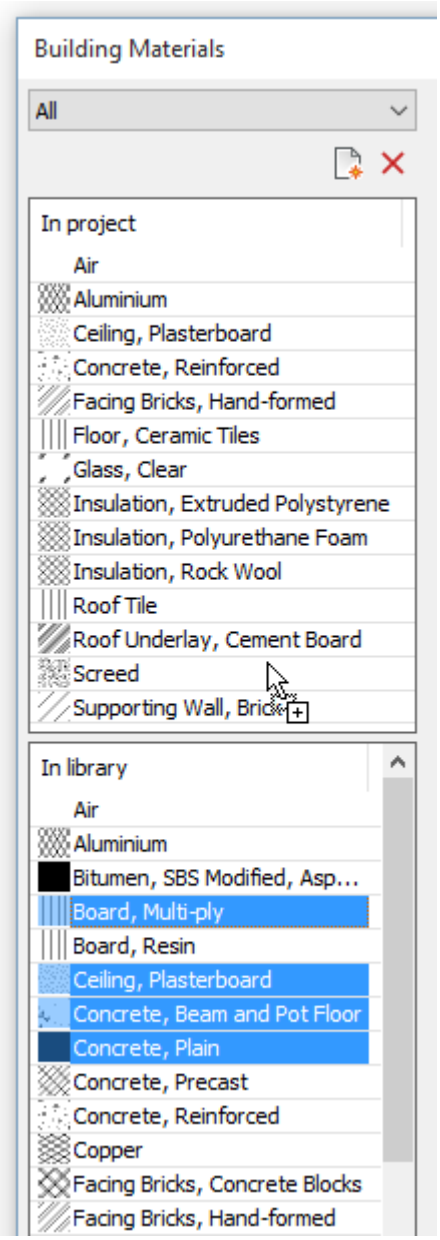
Choose *Yes* or *No* whether or not to allow a variable thickness.

Type a default thickness in the *Thickness* field.

Click the *OK* button.

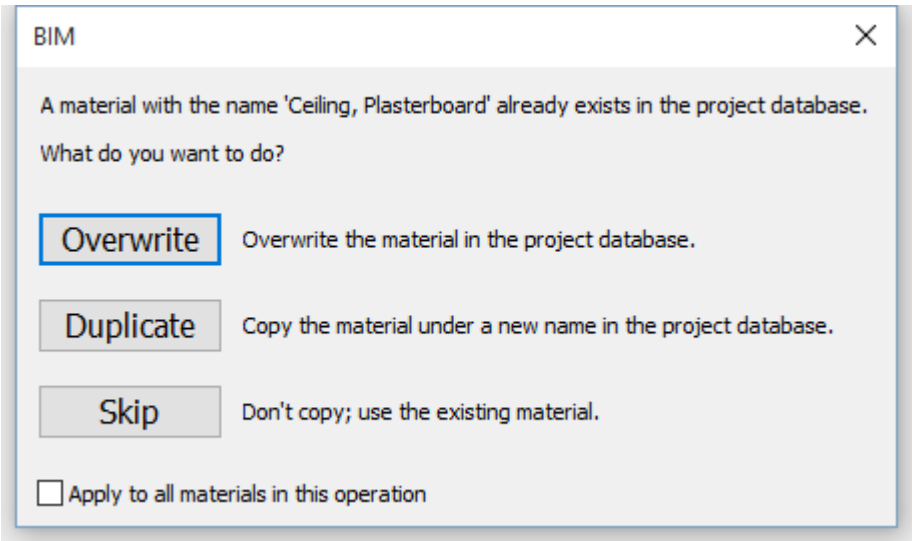
To copy materials between the Project and Library databases.

Select the material(s) in the source database.
Press and hold the Ctrl-key to select multiple materials.

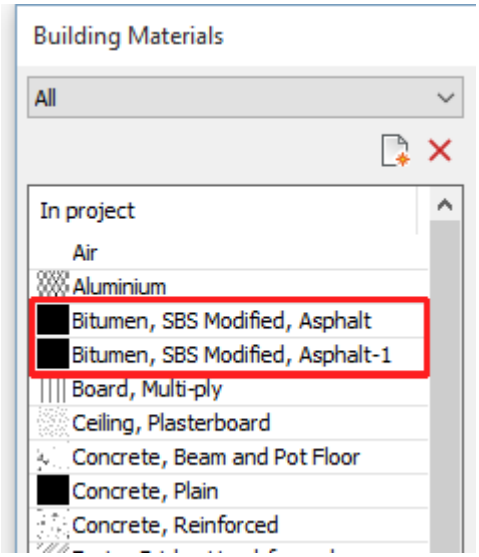


Drag the materials to the target database.
The selection set is copied.

If one or more materials already exist in the target database, an alert box displays:



Click the appropriate button to either overwrite, duplicate or skip the existing material.
If the *Apply to all materials in this operation* option is checked, the option is applied to all existing materials.
Duplicated materials have an index number added to their name:



Deleting Physical Materials

On the *Physical Materials* dialog box, select the material(s) in the *Project* or *Library* database.
Click the *Delete* button.

NOTE Materials that are used in a composition or a mechanical component cannot be deleted.

Compositions

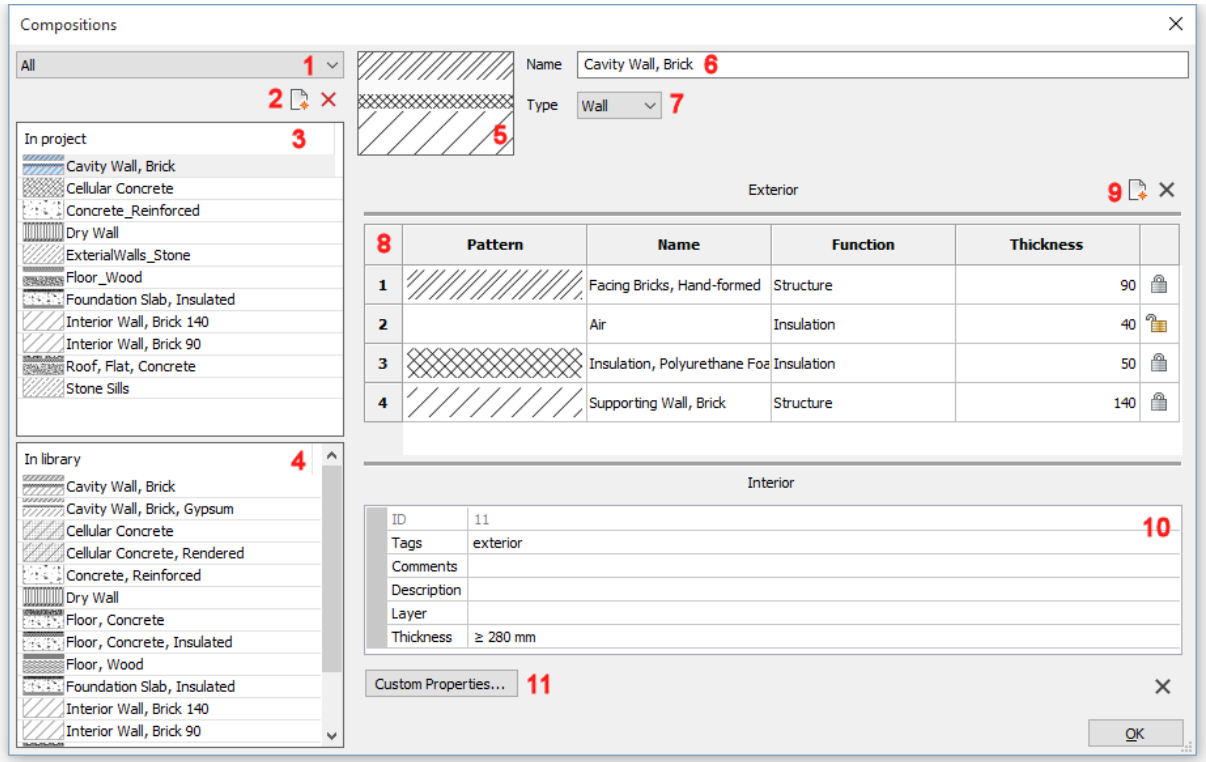
To Open the Composition Editor

Do one of the following:

Type *blcompositions* in the command bar.

Click the *Compositions* tool button ( ) on the BIM Libraries panel.

Double click a composition in either the *In Project* or the *In Library* list on the BIM Libraries panel.



1. Composition category selection button. Click the button, then select a category.

The options are *Wall*, *Slab*, *Roof* or *Generic.*

2. Compositions toolbar.

3. Compositions of the selected category (*Wall*, *Slab*, *Roof* or *Generic*) in the *Project* database.

If the Filter button ( ) is pressed, the current filter is applied to the compositions list.

4. Compositions of the selected category (*Wall*, *Slab*, *Roof* or *Generic*) in the *Library* database.

If the Filter button ( ) is pressed, the current filter is applied to the compositions list.

5. Preview of the currently selected composition.

6. Name Field.

7. Composition category selection button.

8. Ply grid.

9. Ply toolbar.

10. Property grid.

11. Opens the *Custom Properties* dialog.

The Composition Thickness

The overall thickness of a composition equals the sum of the thicknesses of the composing plies.

The thickness of a ply is either locked or unlocked:

Locked thickness ( ): the thickness of the ply is fixed.

Unlocked thickness ( ): the thickness of the ply is variable.

Only one ply in a composition can have an unlocked thickness.
As a result the total thickness of a composition can be:

> **fixed**: all its plies have a locked thickness,
>
> **minimal**: the composition contains at least two plies and one ply has an unlocked thickness. The minimum thickness equals the sum of the thicknesses of the locked plies.
>
> **free**: a single ply composition with an unlocked thickness.
>
> The thickness of a ply can be unlocked only if the *Allow Custom Thickness* property of the material is *Yes*.

When a Composition is attached to a 3D Solid, the thickness of the Composition is applied as follows:

Detect the side faces of the solids.

The side face at the outside of the building is marked as the reference face.

The opposite face (inside face) is moved until the distance between the two side faces is equals the thickness of the Composition.

The BIMFLIP command allows to flip the reference face.

The BIMUPDATETHICKNESS allows to reapply the Composition thickness to the 3D solid.

Sections created with VIEWSECTION will show the plies from the Composition. The reference face is used to indicate the top ply for Slab Compositions, and the exterior ply for other Compositions. Starting from the reference face, the 3D Solid is sliced by planes parallel to the reference face, at an offset equal to the ply thickness.

Adding Compositions

Compositions can be created in either the Project or the Library database.

A new composition can be created from scratch or as a copy of an existing composition.

Single-ply compositions can be created automatically from a material definition.

Adding a composition from scratch

Do one of the following:

> Place the cursor in either the *In Project* or the *In Library* list on the Compositions dialog then right click choose *Add Composition* in the context menu.
>
> Click the *Add Composition* tool button ( ).

A composition named *New* is added in the selected database.

Edit the composition.

Adding a composition as a copy of an existing composition

Select a composition in either the *In Project* or the *In Library* list on the Compositions dialog.
Right click and choose *Duplicate* in the context menu.
A composition named *<existing_composition>-2* is added in the selected database.

Edit the composition.

Creating single-ply compositions

Click the *Materials* button ( ) on the BIM Libraries panel.

Select one or more materials in either the *In Project* or the *In Library* list.

Right click and choose *Make Composition* in the context menu.

Editing a Composition

Select the composition in either the *In Project* or the *In Library* list on the Compositions dialog.
The current settings of the selected composition display.

Type a name in the *Name* field.

Click the *Category* button, then select a composition category: *Wall*, *Slab*, *Roof* or *Generic*.
Composition categories are used to filter the compositions list on the BIM Libraries panel and the Compositions dialog.

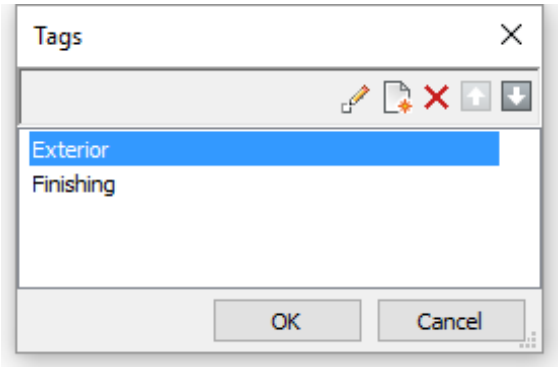
Edit the composition plies in the *Ply* grid.
Here you can:

- Modify the thickness of a ply.
- Change the material of a ply.
- Add a ply.
- Modify the order of the plies.
- Delete a ply.

Edit the properties of the composition in the *Properties* grid.

Edit the *Tags* field: tags are separated by commas.
Tags are case sensitive.

Click the *Browse* button (...) to open the *Tags* dialog.



Click the *New Item* button ( ) to add a new tag at the bottom of the list.

Select a tag in the list then click a tool button:

- : edit the selected tag
- : delete the tag
- : move the tag up
- : move the tag down

Edit the *Comments* field.

Click the *OK* button to save the changes.

To modify the thickness of a ply.

The thickness of a ply can be modified only if the Allow Custom Thickness property of the Material is *Yes.*

Click the *Thickness* field in the *Ply* grid of the composition.

Type a value and press Enter.
The overall thickness of the composition is updated accordingly.

NOTE When the thickness of one or more plies in a composition have been modified, the solids which have this composition applied are not updated automatically.

The BimUpdateThickness command allows to re-apply the overall thickness of a composition to the solid.

To change the material of a ply.

Double click either the *Pattern* or the *Name* field of the ply.
The *Materials* dialog displays.

On the *Materials* dialog double click a material in either the *In Project* or the *In Library* list.
The selected material is applied to the ply.

(option) Edit the thickness of the ply.

To add a ply.

A new ply will be added above the selected ply. You can drag the ply to a different position afterwards.

Select a ply in the composition

Click the *Add Ply* tool button ( ).
The *Materials* dialog displays.

On the *Materials* dialog double click a material in either the *In Project* or the *In Library* list.
The selected material is applied to the new ply.

(option) Edit the thickness of the ply.

(option) Drag the ply to a different position.

To modify the order of the plies.

The plies of a composition are ordered from outside (top of the list) to inside (bottom of the list).

Drag the ply number (first column in the ply grid) to the desired position.

To delete a ply.

Select the ply in the composition.

Click the *Delete* tool button ( ).

Deleting Compositions

Select the compositions in the *In Project* or *In Library* list on the BIM Libraries panel or the Compositions editor.

Do one of the following:

Right click and choose *Delete* in the context menu.

Click the *Delete* button ( ) (*Compositions* editor only).

NOTE Compositions that are used in the project cannot be deleted in the project database.

Attaching a Composition to Solids

Compositions can be attached to solids using the BimAttachComposition command or by dragging a composition from the BIM Libraries panel onto solids.

Do one of the following:

Click the *Attach Composition* tool button ( ) on the *BIM* toolbar.

Choose *Attach Composition* in the *BIM* menu.

Type *bimattachcomposition* in the command bar, then press Enter.

You are prompted: Enter composition name or [Dialog] <Dialog>:

Do one of the following:

Type a composition name in the command bar, then press Enter.
Composition names are case sensitive.

Press Enter to display the Compositions dialog and double click a composition in the *In Project* or *In Library* list.

You are prompted: Select entities to attach composition:

Select one or more solids in the drawing, then press Enter.
You are prompted:The composition has been assigned to <xxx> element(s).

Dragging a composition onto a single solid

Select the composition in the *In Project* or *In Library* list on the *BIM Libraries panel*.

Press and hold the left mouse button and drag the composition onto the solid.

Release the left mouse button to attach the composition.
The thickness of the solid is adjusted if necessary.

Dragging a composition onto a selection of solids

Select the solids in the drawing.

Select the composition in the *In Project* or *In Library* list on the BIM Libraries panel.

Press and hold the left mouse button and drag the composition onto one of the selected solids.

Release the left mouse button to attach the composition.
The thickness of the solids is adjusted if necessary.

NOTES

Library compositions are copied to the Project database upon attachment.

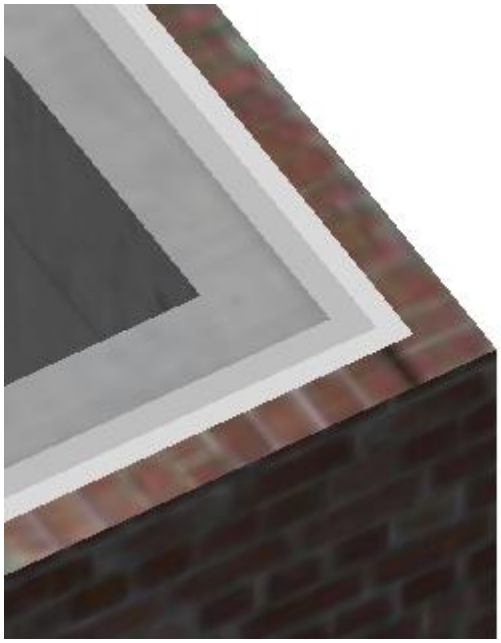
Attaching a *Wall* composition to a solid automatically classifies the solid as a wall.

Attaching a *Slab* or *Roof* composition to a solid automatically classifies the solid as a slab.

Displaying the composition plies

Each building element which has a composition attached, has a *Display Composition* property. When switched ON, the composition plies display in the model, allowing to select and edit the ply faces separately.

When *Display Composition* = *ON* the render material of each ply, which is defined in the 3D Render Material setting of the material definition, shows in the model for visual styles that have their *Face Settings / Material Display* set to *Materials and Textures* (e.g. *BIM*, *Modeling* and *Realistic*). When *Display Composition* = *OFF* the Material definition of the solid is used.



Select the solid(s) in the model (make sure the entire solid is selected, not faces).
The properties of the selection display in the Properties Bar.

Select *Display composition* in the *BIM* settings group.

Choose *On*.

BIM	
Type	Wall
Name	EW-03
Description	
Building	House
Story	Ground Floor
Composition	Cavity Wall, Brick
Display composition	On
GUID	2Xwm3WUJb2mvbcGHV6FKNn
Wall type	Exterior Wall
Room bounding	On

The plies of the selected solids display in the model.

BIM Sections

Defining BIM Sections

In BricsCAD BIM 2D documentation drawings are generated from the 3D model using BimSection definitions. The drawings are managed using an automatically generated project sheetset.

The *BimSection* command creates *BIM section* entities, which are SectionPlane entities, extended with the following properties:

Type: plan, elevation, section or detail.

Callout position: start, mid, end, start-and-end. Indicates the position where callout blocks are displayed along the section line. If desired, users can define their own callout blocks, for each section type.

Scale: sets the scale of the paper space viewport generated to display the results of the drawing generation. The default scale for new Bim Section definitions is specified by the *SectionScale* user preference.

Layerstate: Use this setting to define which layerstate shall be applied when the section drawing is (re)generated.

Apply Layerstate:

Yes: Entities on the visible layers, as defined in the assigned layerstate, will take part in the drawing generation only.

No: All currently visible entities will take part in the drawing generation.

BIM section entities are created on the current layer. It is recommended to create a dedicated layer for BIM sections, e.g. *BIM_Sections*.

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.

To launch the BIM Section command

Do one of the following:

Right click, then choose *Define Section* ( ) in the *Model* command group in the *Quad**.

Click the *Define Section* tool ( ) in the *BIM* toolbar.

Choose *Define Section* in the *BIM* menu.

Type *bimsection* in the command bar.

Prompts you: Specify section type [Other/Detail] <Other>:

Do one of the following:

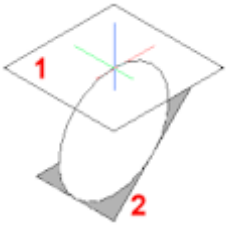
Right click to create a *Plan*, *Section* or *Elevation*.

Type *D* to create a *Detail* section.

*Right click the Quad field in the *Status Bar* and choose *Show Quad on right click* option in the context menu.

General Procedure to Create a BIM Section

When defining a BIM section, the *Section Tracker* block is attached to the cursor:



1. Section plane
2. View direction

The section plane (1) lies in the XY-plane of the current coordinate system (WCS or UCS). The view direction (2) is in the negative Z-direction of the current coordinate system.

If Dynamic UCS (DUCS) is ON, the section plane aligns to the face of the 3D solid under the cursor. Hit the TAB-key to select an obscured face.
Hit the Shift-key to lock the dynamic UCS, which allows to specify the definition point(s) for the BIM Section entity outside the face of the solid.

The section line is parallel to the X-axis of the coordinate system (WCS, UCS or DUCS). In case of a DUCS the X-axis lies on the edge by which the cursor enters the 3D solid face. If this edge is vertical, the Y-axis lies on the edge and the X-axis is horizontal.

To define a BIM section:

Launch the BimSection command.

You are prompted: Select a point to place section or [Detail/Scale]:

Do one of the following:

To create a plan section (or a section parallel to the XY-plane of the current UCS): click a point without using DUCS.

Hover the cursor over the face of a 3D solid which is parallel to the section plane you want to create.
The Section Tracker aligns to the face under the cursor, on condition DUCS is active.
Optionally, hit the Shift-key to lock the DUCS, which allows to snap a point outside the selected 3D solid face.
Click a point.

The initial section plane displays dynamically and the 3D model is clipped accordingly.
You are prompted: Specify distance:

Do one of the following:

Type the offset from the initial position.

Specify a point.

The BIM section entity is defined.

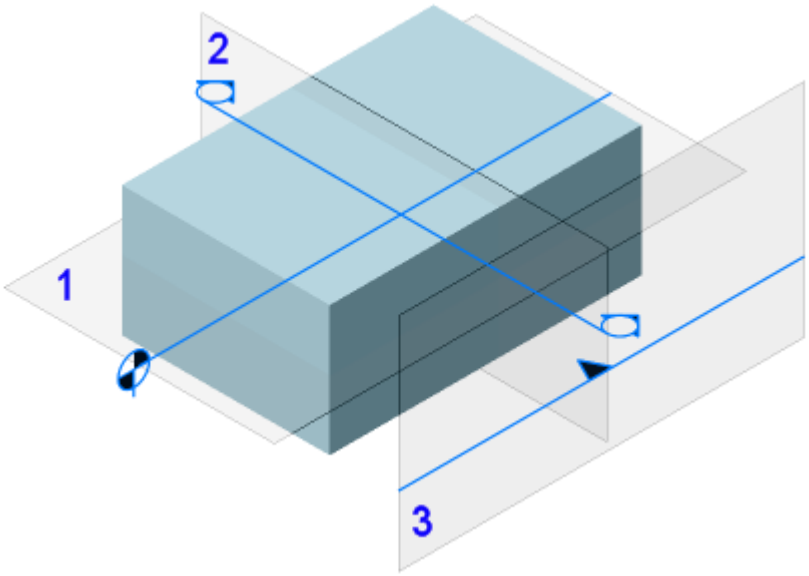
The *Section Type* property is set to:

Plan (1): for horizontal section planes.

Section (2): for vertical sections cutting through the model.

Elevation (3): for vertical sections outside the model.

The *Clip Display* property is enabled by default, except for elevations. As a result the display of the 3D geometry is clipped by the newly defined section.



Edit the BIM section entity properties.

Defining the BIM Section Entity Properties

The default properties of a BIM section entity are copied from the corresponding section type in the *_SectionSettings.dwg* drawing in the *Bim/Sections* subfolder of the *Support* folder. Edit the properties of these source BIM section planes to customize the default properties.

To open the *Support* folder: type SUPPORTFOLDER in the command bar.

Defining the BIM Properties

Select the BIM section entity (or entities) in the drawing.
If multiple BIM Section entities are selected you can specify settings for the selection set simultaneously.
The current properties display in the Properties Bar.

Section Entity	
Name	Plan 0
State	Plane
Live section	No
Clip display	Yes
Use lids	Yes
Plane transparen	70
Plane color	9
Destination file	D:\BricsCAD Training\Bricscad\
Geometry	
Elevation	1792.77 mm
Top plane	15101.59 mm
Bottom plane	19757.77 mm
Vertex	1
Position	1173.67, 7344.13, 1792.77
X	1173.67
Y	7344.13
Z	1792.77
BIM	
Section type	Plan
Callout position	Start
Scale	0.02
Layer state	
Apply layer state	Off

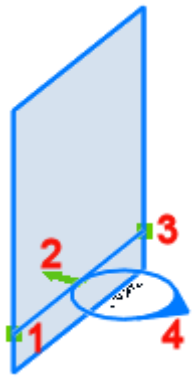
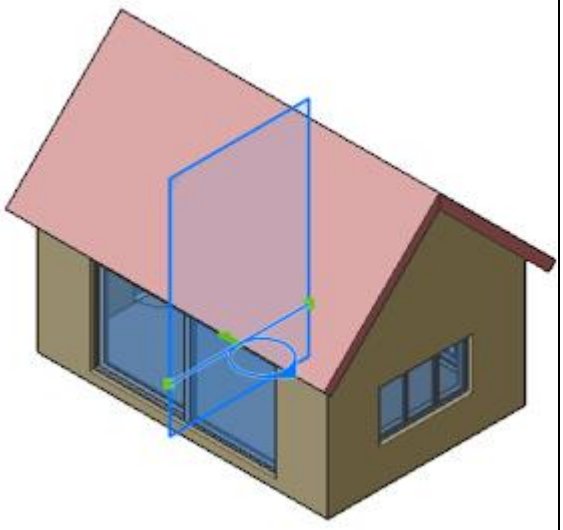
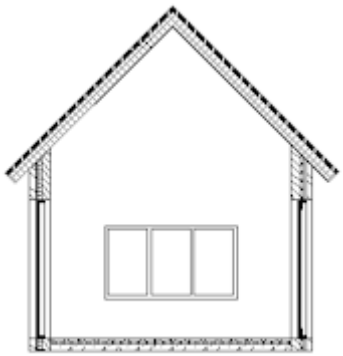
Define the properties of the BIM section entity:

Section Entity:

Name: type a name.
By default, the section name defines the name of the generated drawing.
The *Destination file* property of the *2D Section* settings of the BIM Section entity allows to set the location and file name of the calculated drawing otherwise.

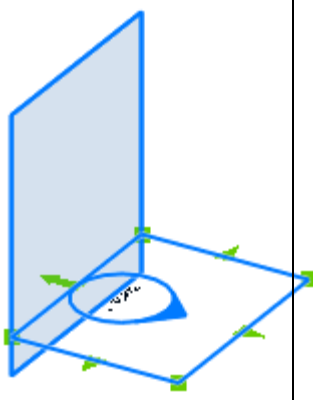
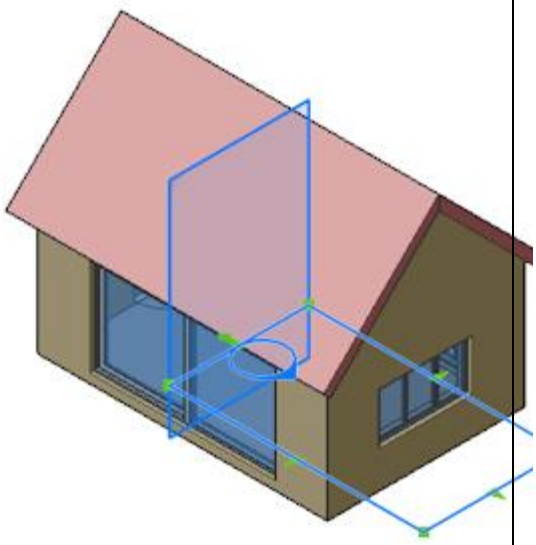
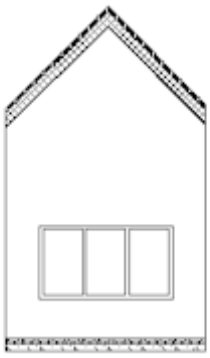
State: Click the field and select a state. The selected state determines how the section result will be clipped.

Plane: The section is defined as an infinite plane.
The section result is not clipped.

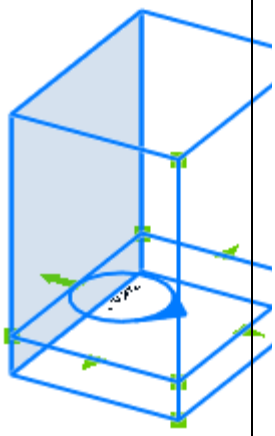
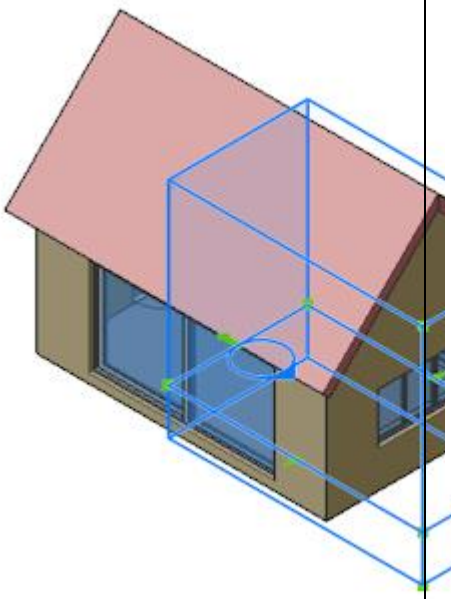
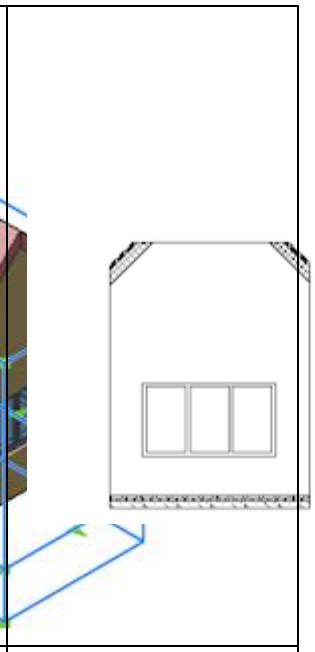
		
BIM Section entity	*3D Model*	*Generated Drawing*

1. Start grip
2. Flip view direction grip
3. End grip
4. Section call out

Boundary: The section is defined as a boundary.
The section result is clipped by an infinite tube through the section boundary.
Drag the green triangular grips to modify the section boundary.

		
BIM Section entity	*3D Model*	*Generated Drawing*

Volume: The section is defined as a volume.
The section result is clipped by the section volume.
Drag the green triangular grips to modify the vertical clipping planes.
Drag the top and bottom grips to modify the horizontal clipping planes.

		
BIM Section entity	*3D Model*	*Generated Drawing*

Live Section: sets the Live Section property of the section plane.
Use this mode for visual checking only, not for 3D modeling.

Clip Display: clips the 3D model. Editing solids is possible in Clip Display mode. As different from Live Section, multiple sections can have this mode set simultaneously.

Use Lids: closes the top faces of the clipped geometry. Solids that have a composition applied are hatched according to the materials in the composition.

Plane Transparency: sets the transparency of the section plane in rendered Visual Styles.

Plane Color: sets the fill color of the section plane in rendered Visual Styles.

Geometry:
Elevation: defines the Z-value of the start point of the section plane.

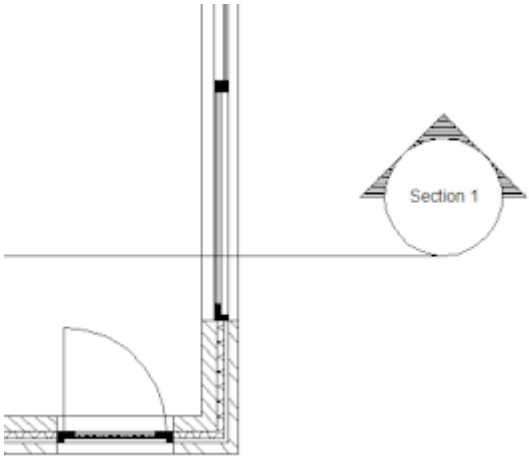
Top Plane: defines the top plane of the section volume.

Bottom Plane: defines the bottom plane of the section volume.

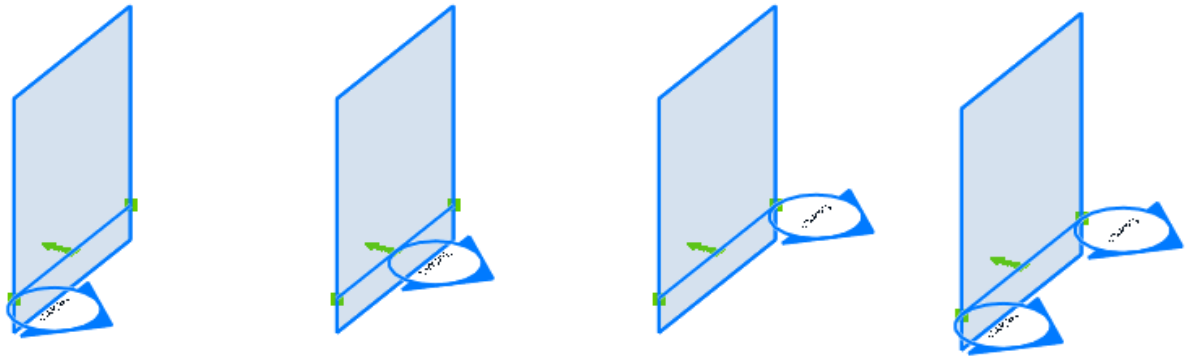
Vertex: defines to coordinates of the start point of the section plane.

BIM:

Section Type: Sets the BIM section type.
Click the field, then select Plan, Section, Elevation, Detail or Interior Elevation.
The section type defines the settings used to create the section drawing.
Sections of the Section type generate a section line and callouts in Plan sections.



Callout Position: Defines the position of the callout(s). Select an option: *Start*, *Mid*, *End* or *StartEnd*.



Start Mid End StartEnd

Scale: Sets the scale of the viewport in the generated drawing.
The default scale is defined by *SectionScale* user preference.

Layer State: Indicates the layer state to be used to calculate the section.

Apply Layer State: Determines whether the layer state will be used to calculate the section or not.
Select an option: *Off* or *On*.

Click the green arrow (⬅) to flip the view direction of the section.

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.

Defining the section entity properties

The properties of a section plane entity apply to BIM Section entities as well. However, only the *2D Section /elevation block creation settings* are used when generating a BIM section.

Choose *Drawing Explorer > Section Planes...* in the *Tools* menu.

Select the BIM Section(s) in the section plane list..
If multiple BIM Section entities are selected you can specify settings for the selection set simultaneously.

Destination

The section result is placed as a block in a new drawing. If no specific name was set for the target drawing, the name of the BIM Section entity will be used and the drawing is saved in the same folder as the model drawing.

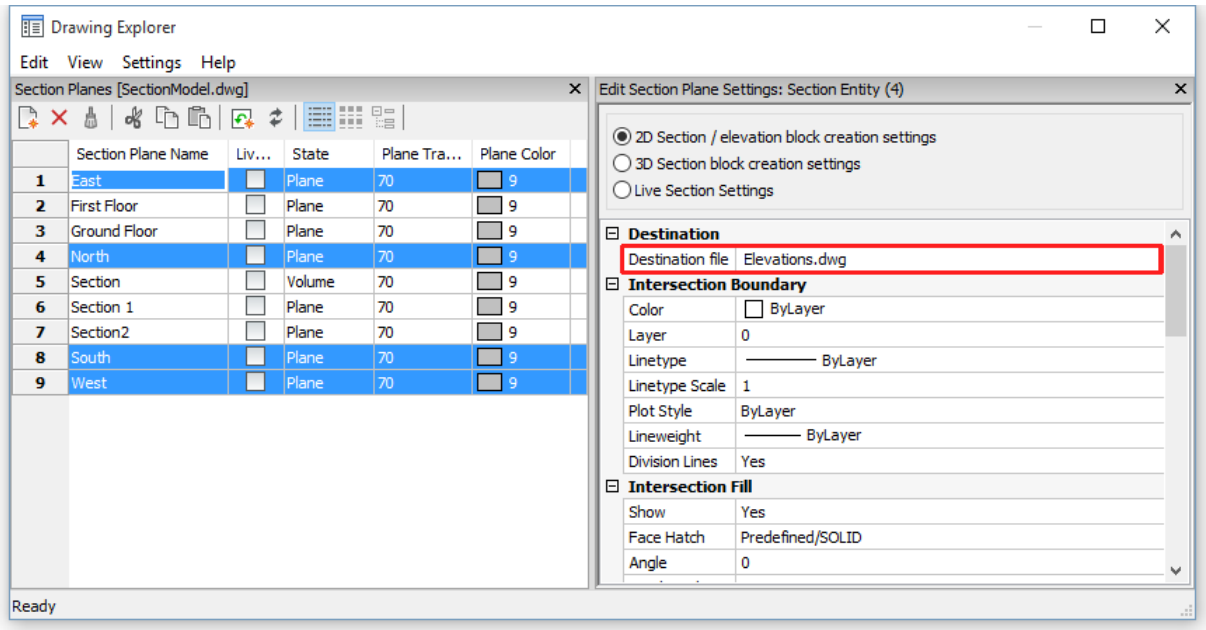
Alternatively you can specify the file name using the *Destination File* property of the *2D Section* settings of the BIM Section entity, which allows to send the section results of different BIM Section entities to the same destination file.

Under *2D Section /elevation block creation settings* > *Destination*, select the *Destination file* setting.

Do one of the following:

Type a name in the *Destination file* field.

Click the *Browse* button (...), then select the destination folder in the *File* dialog.
Type a name in the *File Name* field and click the *Open* button.
The specified folder and file name display in the *Destination file* field. The **.dwg* extension is added automatically.



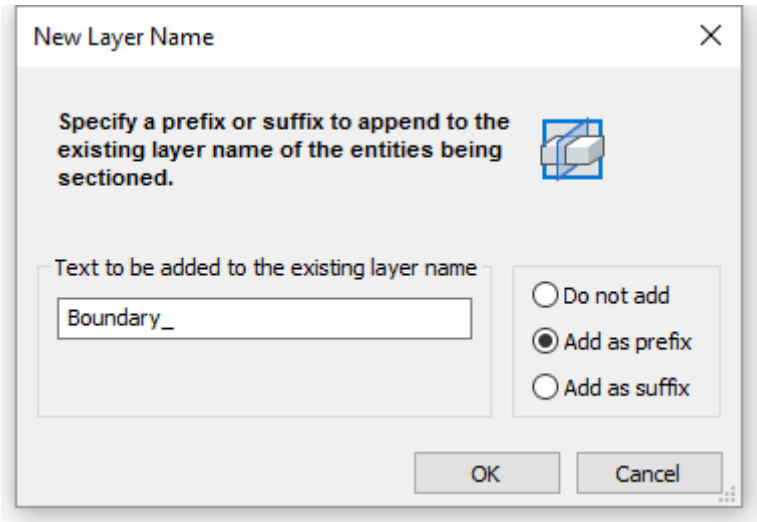
Intersection Boundary

The *Intersection Boundary* settings control the properties of the intersection lines.

It is recommended to set the *Color*, *Linetype*, *PlotStyle* and *Lineweight* settings *ByLayer*.

The Layer field is set to *Boundary_*EntityLayer** by default. The **EntityLayer** part will be replaced by the Material name of the Composition of the sectioned solid. Multiply compositions generate a different layer for each material.

(option) Click the *Layer* field and choose *New layer name settings ...* in the list.



Do the following:

Type a text in the text field

Choose to add the text to the Material name as a prefix or a suffix.

Click the OK button.
Click the *Layer* field, then choose **EntityLayer*_IntersectionBoundary*.
The format of the generated layer names is: <MaterialName>_IntersectionBoundary, e.g. Facing Bricks_ Hand-formed_IntersectionBoundary

Intersection Fill
The *Intersection Fill* settings control the properties of the hatching of the intersection boundaries.

The intersection fill hatching is set to *Fill_*EntityLayer** by default (see also the *Layer* setting of the Intersection Boundary).

It is recommended to set the *Color*, *Linetype*, *PlotStyle* and *Lineweight* settings *ByLayer*.

The *Face Hatch*, *Angle* and *Hatch spacing* properties apply only to boundaries generated from solids that have no BIM Composition attached.

Hatch scale: this setting applies to the hatch pattern of physical materials also.
The final pattern scale in a section is the product of the scale setting of the physical material and the value of the *Hatch Scale* property of the section plane entity.
Since the dimensions of predefined patterns are expressed in mm, the value of the *Hatch Scale* property is must be set with respect to the *Insertion Units* (INSUNITS) system variable of the model file.

INSUNITS	Section Scale
mm	10
cm	1
m	0.01

Background Lines
The *Background Lines* settings control the properties visible and hidden lines in the view direction of the BIM Section entity.

Show: Choose *Yes* to generated the background lines.

Hidden Line: Choose *No* to not show invisible lines.

The background lines are placed on the *Background_*EntityLayer** by default (see also the *Layer* setting of the Intersection Boundary).

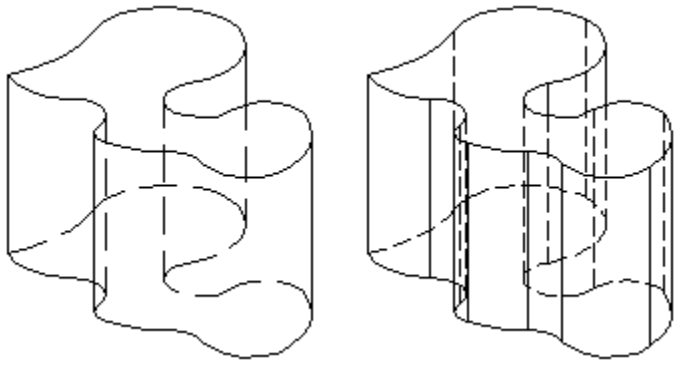
It is recommended to set the *Color*, *Linetype*, *PlotStyle* and *Lineweight* settings *ByLayer*.

Cut-away Geometry

Choose *No* to not show the cut-away geometry.

Curve Tangency Lines

The *Curve Tangency Lines* settings control the display of the transition line between two tangent faces. It is an imaginary line between the two faces. In most cases tangential edges need to be removed.



Tangency lines removed (left) or shown (right).

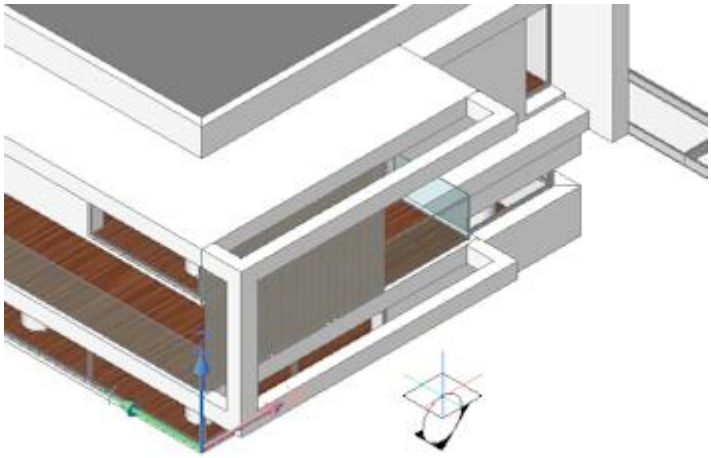
If necessary, set the properties for the *Curve Tangency Lines*.

To Define a Plan Section

A plan section is best defined in the WCS (World Coordinate System).

Launch the BimSection command.

You are prompted: Select a point to place section or [Detail/Scale]:



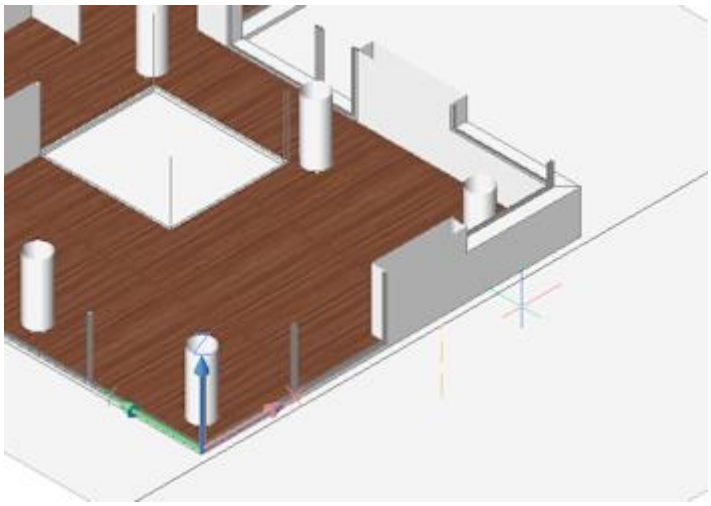
Place the *Section Tracker* near the 3D model and click.

The initial section plane lies in the XY-plane (Z=0) of the current coordinate system (e.g. the WCS).

The section plane displays dynamically and the 3D model is clipped accordingly.

You are prompted: Specify distance:

If Dynamic Dimensions is *ON*, the current distance displays in a dynamic dimension field.



Do one of the following:

Move the cursor and click when the section plane is at the desired height.

Use an Entity Snap to specify a point.

Type a value to define the offset from the initial position of the section plane.

The plan section is defined.

(Option) Select the plan section entity and edit the *Elevation* property of the BIM section entity to adjust the vertical position of the section plane.

Geometry	
Elevation	1518.20 mm
Top plane	23936.02 mm
Bottom plane	7783.98 mm

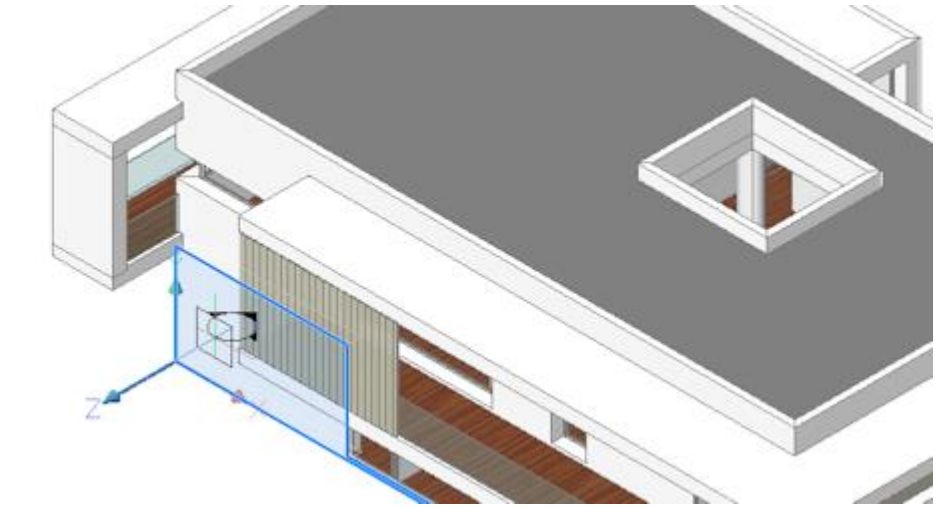
Edit the BIM section entity properties.
Horizontal sections get the *Plan* section type by default.

To Define a Cross Section

Make sure Dynamic UCS (DUCS) is enabled when defining a cross section.

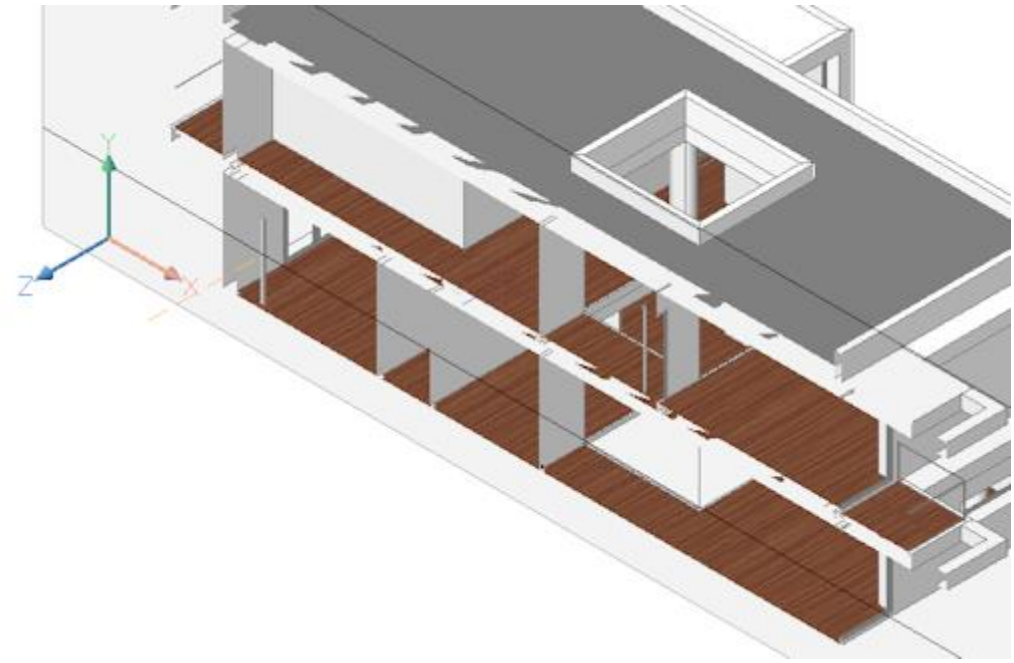
Launch the BimSection command.
You are prompted: Select a point to place section or [Detail/Scale]:

Place the section tracker over a 3D solid face which is parallel to the cross section to be defined.



Click when the face highlights and move the section tracker inside the 3D model.
The section plane displays dynamically and the 3D model is clipped accordingly.

You are prompted: Specify distance:
If Dynamic Dimensions is *ON*, the current distance displays in a dynamic dimension field.



Click when the section plane is at the desired location or specify a distance.
The cross section is defined.
Vertical sections get the *Section s*ection type automatically.

(option) Drag the Start grip to move the section plane.

(option) Drag the End grip to rotate the section.

(option) Click the Arrow grip to flip the view direction.

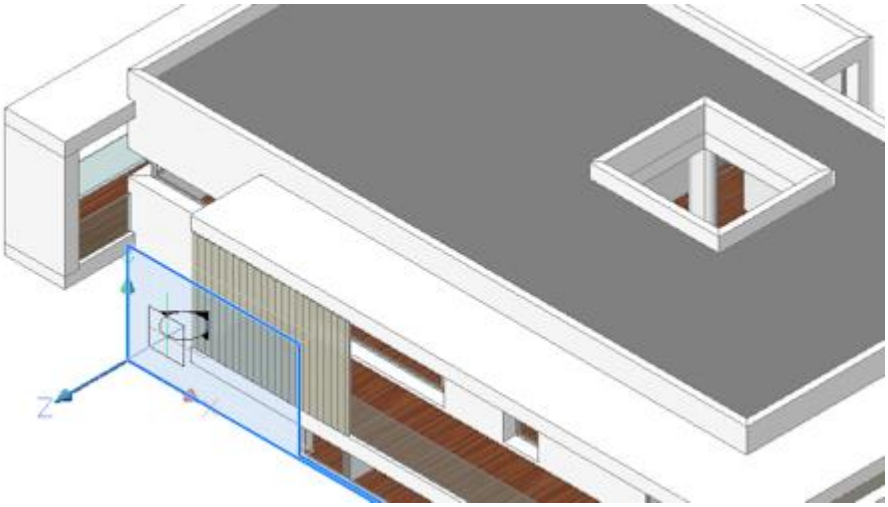
Edit the BIM section entity properties.

To Define an Elevation

Make sure Dynamic UCS (DUCS) is enabled when defining an elevation.

Launch the BimSection command.
You are prompted: Select a point to place section or [Detail/Scale]:

Place the Section Tracker over a 3D solid face which is parallel to the elevation to be defined and click when the face hightlights.
You are prompted: Specify distance:



Move the section tracker outside the 3D model and click to define the elevation.
The elevation is defined.
Vertical sections outside the model (no geometry is sectioned) get the *Elevation s*ection type automatically.

Edit the BIM section entity properties.

To Define a Detail Section

Detail sections can be defined from scratch or based on an existing section.

Detail sections are section planes which have the Volume state applied automatically.

To define a detail section from scratch

Launch the BimSection command with the *Detail* option.
You are prompted: Select first corner of the volume box base rectangle or [Based on an existing section/Scale]:

Place the section tracker over a 3D solid face which is parallel to the cross section to be defined.
Specify a point when the face highlights.
The volume box base rectangle displays dynamically.
Your are prompted: Select the opposite corner of section box:

Do one of the following:

Specify a point.

Type a value in the dynamic dimension fields.
Hit the TAB key to go back and forth between the dynamic dimension fields.
Type Enter to confirm the current values.

The volume box displays dynamically.
You are prompted: Select the height of volume box <xx.xx>:

Do one of the following:

Press Enter to accept the default height (= previously entered value).

Specify a point.

Type a value in the dynamic dimension field.

Edit the BIM section entity properties.

To define a detail section based on an existing section

Launch the BimSection command with the *Detail* option.
You are prompted: Select first corner of the volume box base rectangle or [Based on an existing section(Based)/Scale]:

Type *B*.
You are prompted: Select an existing section plane as base:

Select the BIM section entity to base the detail section on.
You are prompted: Select first corner of volume box base rectangle:

Specify a point.
The volume box base rectangle displays dynamically.
Your are prompted: Select the opposite corner of section box:

Do one of the following:

Specify a point.

Type a value in the dynamic dimension fields.
Hit the TAB key to go back and forth between the dynamic dimension fields.
Type Enter to confirm the current values.

The volume box displays dynamically.
You are prompted: Select the height of volume box:

Do one of the following:

Press Enter to accept the default height (= previously entered value).

Specify a point.

Type a value in the dynamic dimension field.

Edit the BIM section entity properties.

To Create a 2D Isometric Drawing

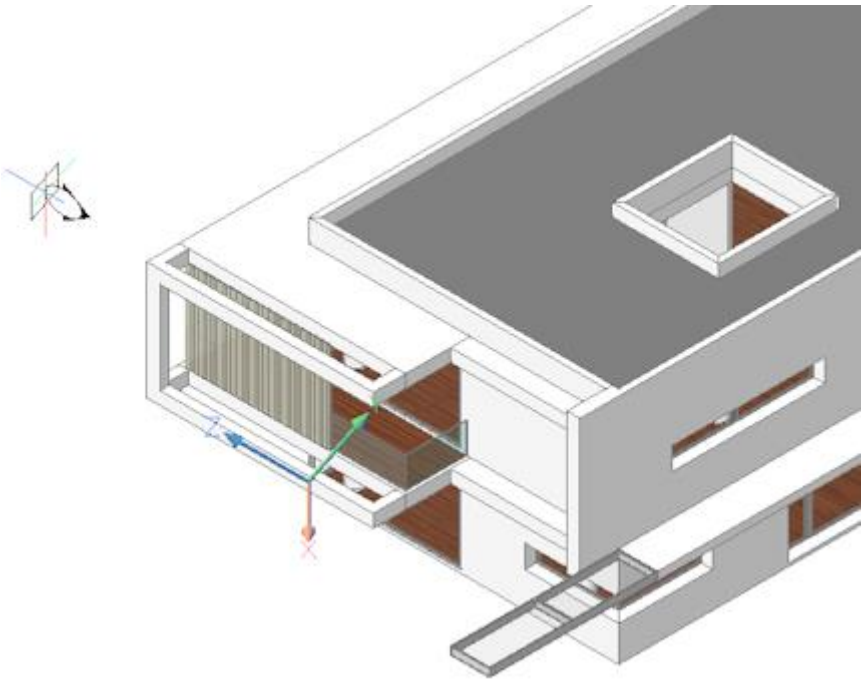
Choose an isometric view from the Look From widget.

Launch the UCS command and select the *View* option.
The UCS is now aligned with the current view.

Rotate the view about 90°.

Launch the BimSection command.
You are prompted: Select a point to place section or [Detail/Scale]:

Move the section tracker outside the 3D model and above the XY-plane of the UCS and create an Elevation type section.



Edit the BIM section entity properties.

Turning a Section Plane into a BIM Section

Section planes, created by the SectionPlane command can be turned into a BIM section entity using the BimClassify command.

Do one of the following:

Click the *BimClassify* tool button ( ) in the *BIM* toolbar.

Choose *Classify* in the *BIM* menu.

Type *bimclassify* in the command bar.

You are prompted: Classify entities as:
Wall/Column/Slab/Beam/wIndow/Door/buildingElement/Auto/Unclassify:

Select the *Auto* option.
You are prompted: Select entities to classify:

Click the section plane, then press Enter.
BricsCAD responds:
BIM data assigned to 1 of 1 object(s)
XXXX classified as BimSection

Edit the BIM section entity properties.

Creating a Jogged BIM Section

Create a jogged section plane.

To create a vertical jogged section plane.

To create a horizontal jogged section plane.

Turn the section plane into a BIM section.

Define the properties of the BIM section.

Using Clip Display

The *Clip Display* property of the BIM section allows to temporarily clip the 3D geometry.

Different from the *Live* status, the *Clip Display* status allows to continue using all modeling and editing operations on all drawing content.

3D model of the villa with a plan section for the ground floor plan.

Clip Display = No

To toggle Clip Display

Do one of the following:

Double click the section entity.

Execute the CLIPDISPLAY command.

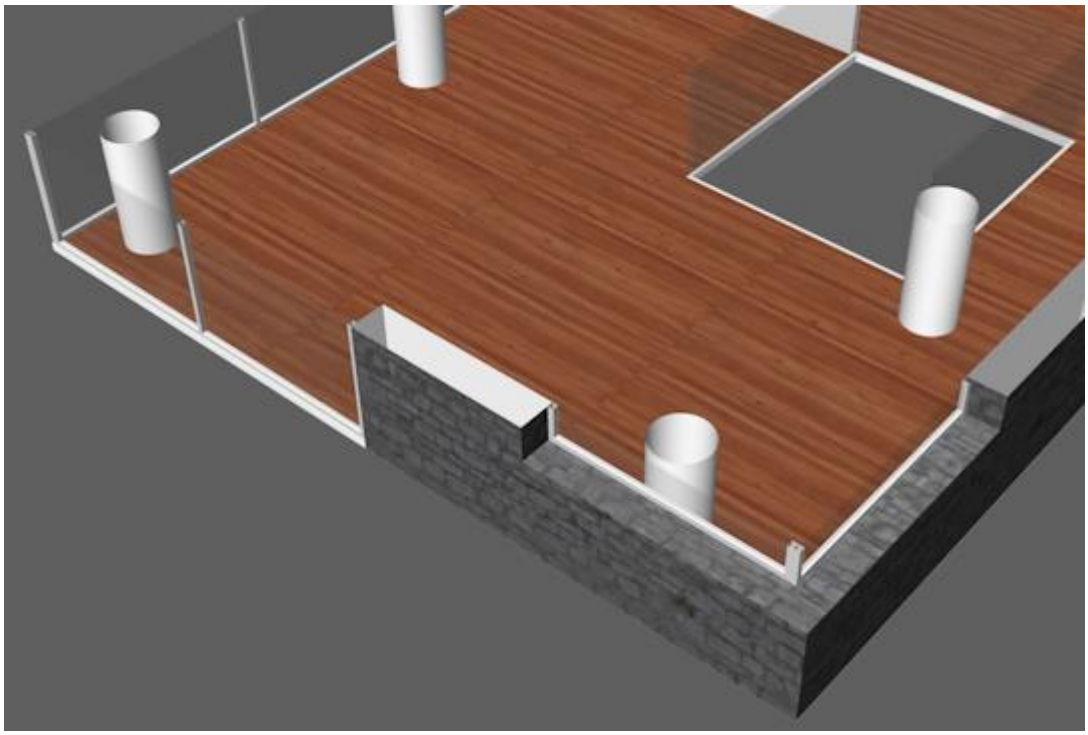
Edit the *Clip display* property of a section in the Properties Bar.

To edit the display properties of a section entity in the Properties Bar
Select the section entity in the drawing
The properties of the section entity display in the Properties Bar.

Section Entity	
Name	Plan
State	Plane
Live section	No
Clip display	No
Use lids	No
Plane transparency	70
Plane color	9

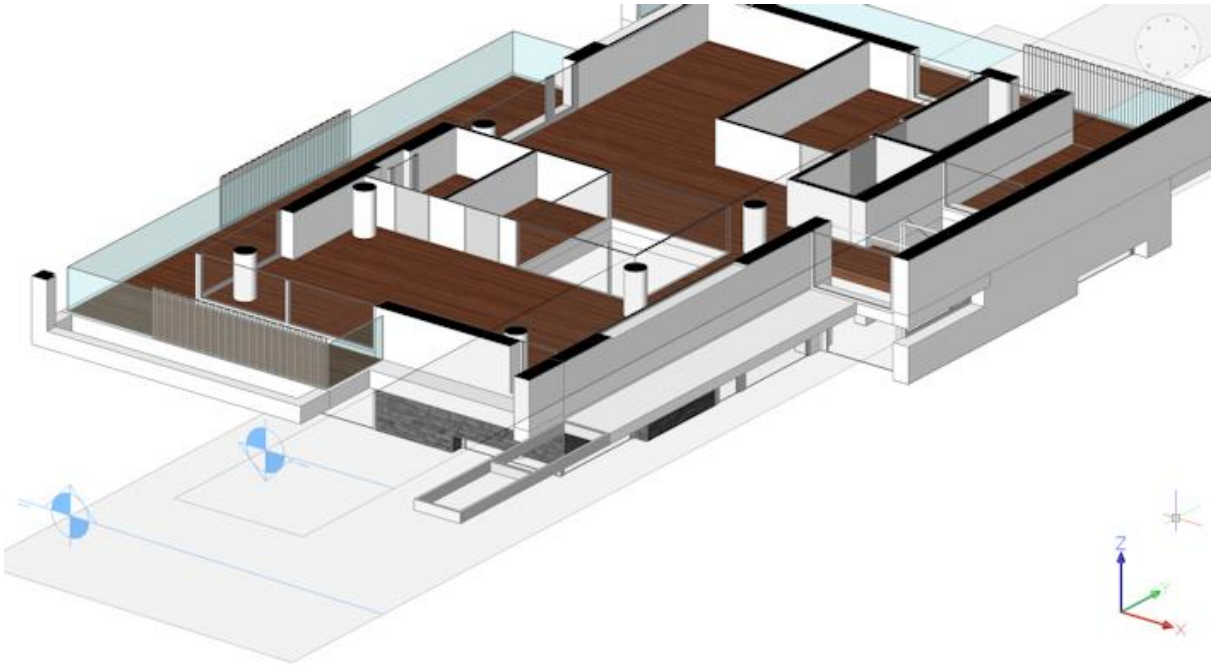
Select the *Clip Display* property and choose *Yes*.

Use Lids = Yes: The top faces of the clipped geometry are closed.
Solids that have a composition applied are hatched according to the materials in the composition.
(option) Select the *Use Lids* property and choose *NO*.



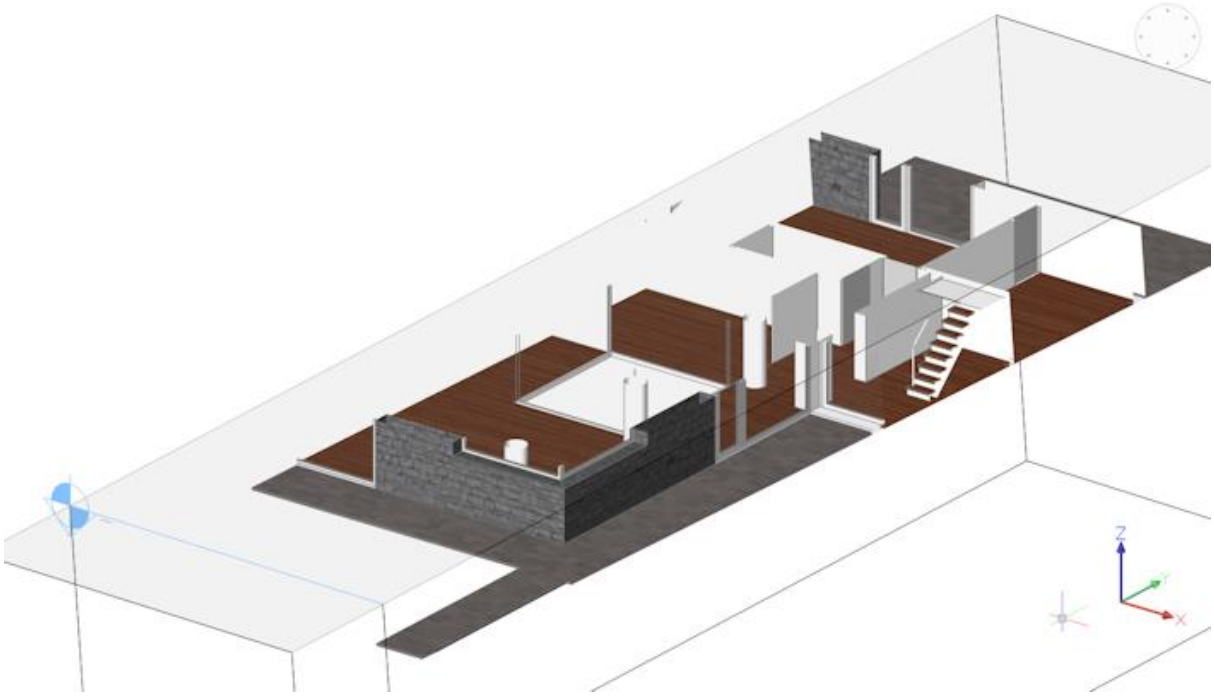
Use Lids = No: The top faces of the clipped geometry are open.

(option) Different from the Live Section property, *Clip Display* can be ON for multiple section entities simultaneously.



Clip Display = ON for two BIM section entities with opposite view directions.

(option) Set the *State* property = *Volume*.



Generating Drawings

BimSectionUpdate generates section drawings from BIM Section entities. The command can be used either by selecting a BIM Section entity in the 3D model, or by selecting an existing section result in a generated drawing. The result is placed in a new drawing, saved in the same folder as the model drawing, unless a different folder is specified in the Destination File property of the BIM Section entity.

BimSectionOpen allows to navigate between the model drawing containing a BIM Section entity and the corresponding generated drawing. The command works bi-directional: either from the section result back to the model or from the model to the section result, depending on what was selected: a BIM Section entity in the 3D model or a section block or a viewport in a generated drawing.

To Generate or Update Sections

Do one of the following:

Place the cursor over a BIM Section callout or section line, then choose *Update Section* ( ) in the *Model* command group in the Quad.

Select one or more BIM Section entities, then choose *Update Section* ( ) in the *Model* command group in the Quad.

Click the *Update Section* tool button in the *BIM* toolbar, then select a BIM section entity or entities.

Choose *Update Section* in the BIM menu, then select a BIM section entity or entities.

BIM Section entities can be selected by its Callout(s) and section line only.

BricsCAD reports the number of generated sections and their name(s) in the Command Window, e.g.:

8 section(s) were generated
D:\Bricsys Reports\V16\Villa\Elevations.dwg
D:\Bricsys Reports\V16\Villa\Plans.dwg
D:\Bricsys Reports\V16\Villa\Sections.dwg

In the example above 2 plans, 4 elevations and 2 cross sections were generated and exported to 3 drawings.

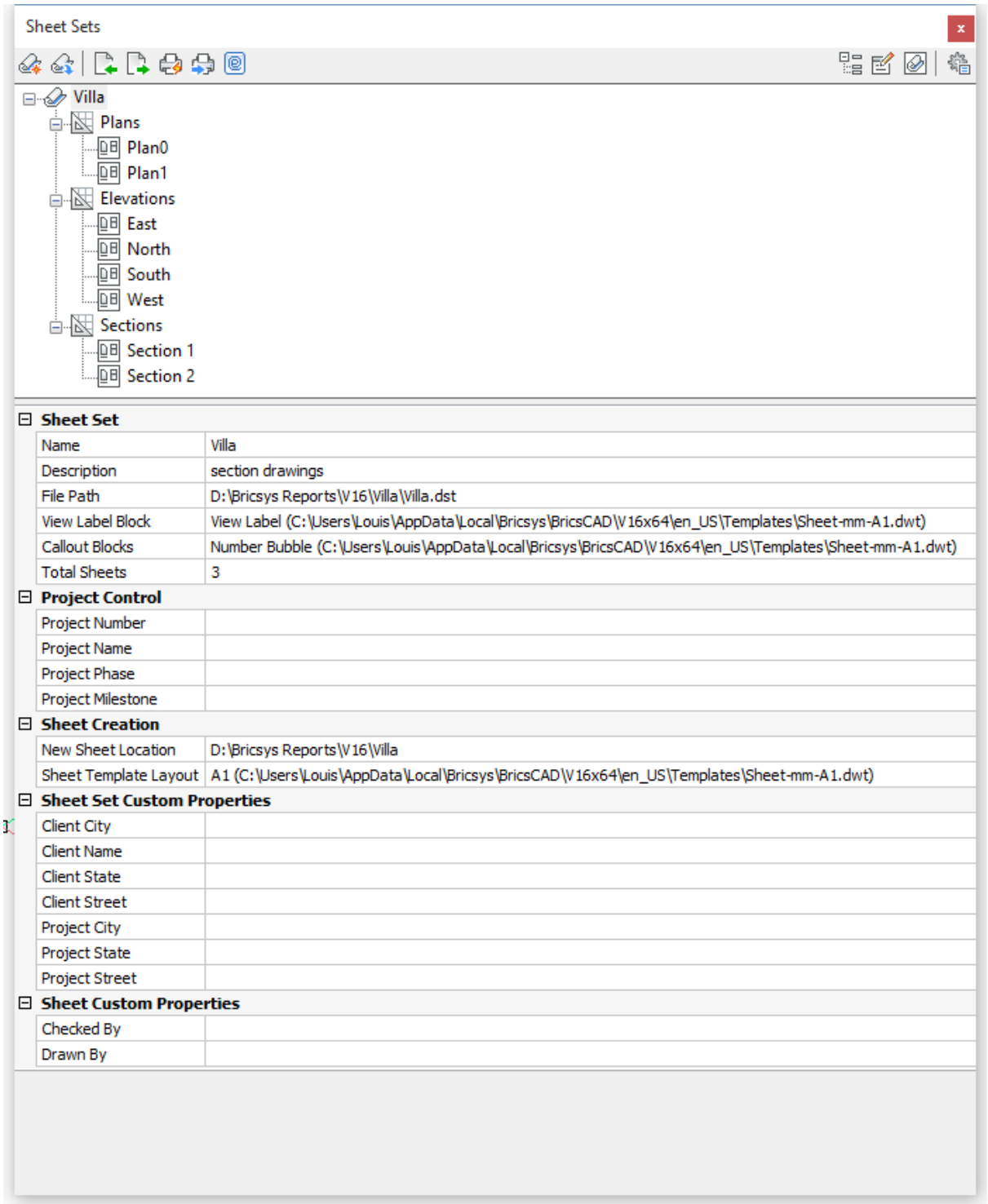
Upon the calculation of the first BIM Section in a project, a new project sheetset is created, using the sheetset template pointed to by the *SectionSheetSetTemplateMetric* (default = BIM-section-

metric.dst) or *SectionSheetSetTemplateImperial* (default = BIM-section-imperial.dst) user preference. The sheetset is stored in the same folder as the drawing.

If the PROJECTNAME system variable in the model drawing is not empty, it is used as the name for the generated SheetSet, else the name of the model drawing is used.

The template drawing for the drawing generation process is defined by the *Sheet Template Layout* setting of the project sheetset..

Paper space view definitions that correspond with the location of the viewport are created and used to set up sheet views. The generated sheetset offers view categories per section type: *Plan*, *Section*, *Elevation* or *Detail*.



NOTES

BIM_SECTIONS_PREVIOUS layer: the result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer, which is frozen by default. Thawing the layer allows for a pragmatic and fast visual check of modifications

BIM_VIEWPORTS layer: generated viewports are placed on the dedicated BIM_VIEWPORTS layer. If desired, this layer e.g. can be set as 'non-printing' in the template drawing.

Using the same folder for the sheetset definition, models and generated drawings enables painless migration from one machine to the next: the SheetSet manager will automatically and silently adjust the file paths after the project folder has been relocated.

To Open a Generated Drawing

Do one of the following

Place the cursor over a BIM Section callout or section line, then choose *Open Section Model* ( ) in the *Model* command group in the Quad.

Double click the drawing in the sheet list in the SheetSet panel.

To Update a BIM Section

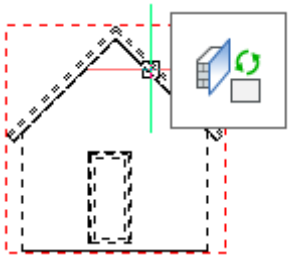
A red frame around the section result indicates that the BIM Section generated block is no longer in sync with the 3D model. Such section results can be updated without opening the 3D model. This can be done both in Model Space and in a Layout.

Update a BIM section in Model Space

Click the *Model* tab.

Place the cursor over the section block that needs to be updated.

Choose *Update Section* ( ) in the *Model* command group in the Quad.

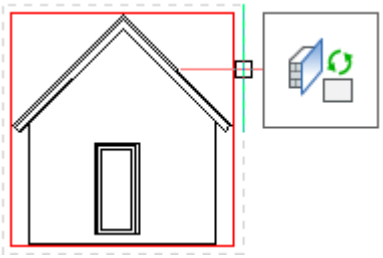


Update a BIM Section in a layout

Click the *Layout* tab.

Place the cursor over the viewport of the BIM Section that needs to be updated.

Choose *Update Section* ( ) in the *Model* command group in the Quad.



The result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer. Thaw the layer to check the modifications.

The content of the BIM_SECTIONS_PREVIOUS layer displays in magenta.

Editing Drawings

When a BIM Section is generated, a block containing the drawing generation result is added to the model space of the target drawing, as well as a paper space viewport looking onto this block. The block and the viewport initially are placed so they do not overlap with existing geometry contained in the drawing. You can move this block or viewport to a new location, which will be preserved on subsequent updates.

In the default template drawing which is used to create the generated drawings a *Viewlabel* and *NumberBubble* blocks exist. You can place them associated to the selected sheet view. The *Title* block in this drawing contains attributes that will be filled in automatically when the associated Sheet Set properties are defined.

A sheet list table can be placed on any sheet of the sheetset. In the current implementation the table content is not associative yet.

Inserting the Title Block

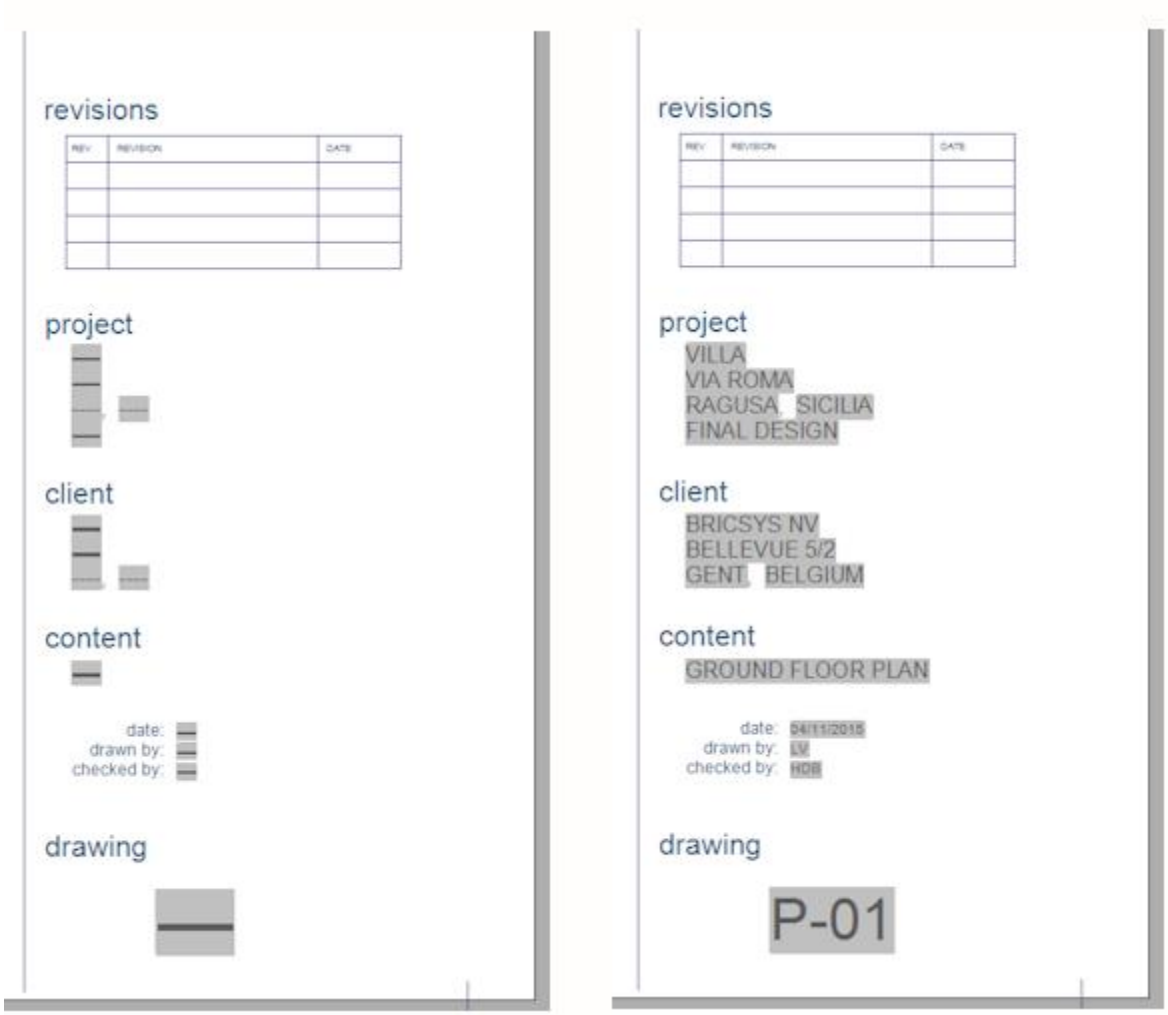
Open the Drawing Explorer - Blocks dialog.

Double click the *Title* block.
The Drawing Explorer dialog closes and the block is attached to the cursor.
In a metric units project the size of the *Title* block fits the A1 paper size (594 x 841 mm)
In an imperial units project the size of the *Title* block fits the Arch D paper size (24 x 36 inches)

Click at the lower left corner of the paper sheet in the layout.

Define the Project and custom properties in the Sheet Sets panel.

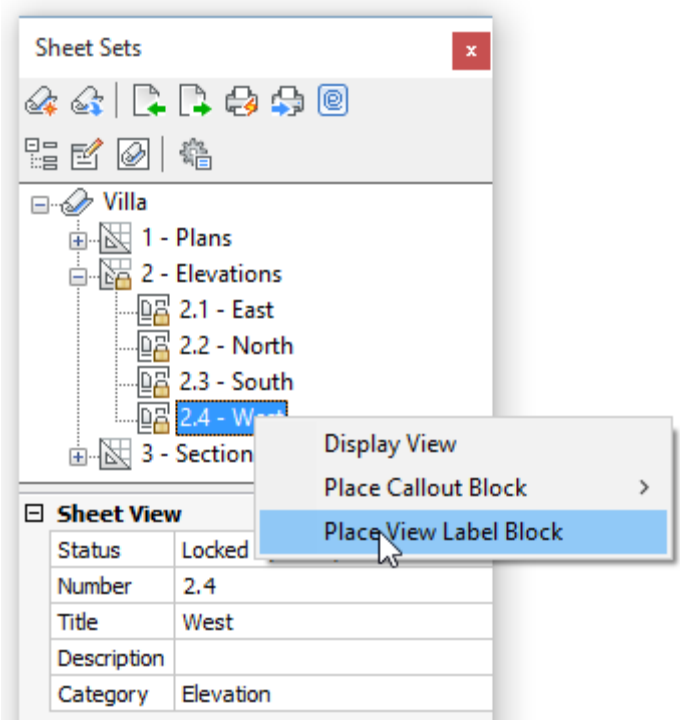
Select the *Title* block, then click the *Update Fields* tool button ( ) in the *Redraw/Regen* toolbar.



Inserting a View Label for a Sheet View

Select the sheet view In the Sheet Sets panel.

Right click the sheet view you want the view label block to insert for.

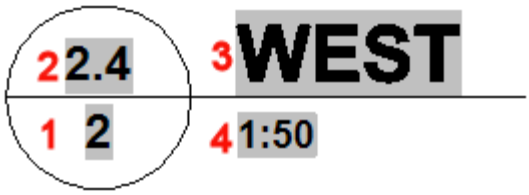


Choose *Place View Label Block* in the context menu.
Your are prompted:
: VIEWLABEL
Units: Millimeters Conversion: 1
Insertion point for block:

Specify the insertion point.
You are prompted:
Scale factor for block <1.0>:

Specify the scale factor or right click to accept the current scale.
Your are prompted:
Rotation angle for block <0>:

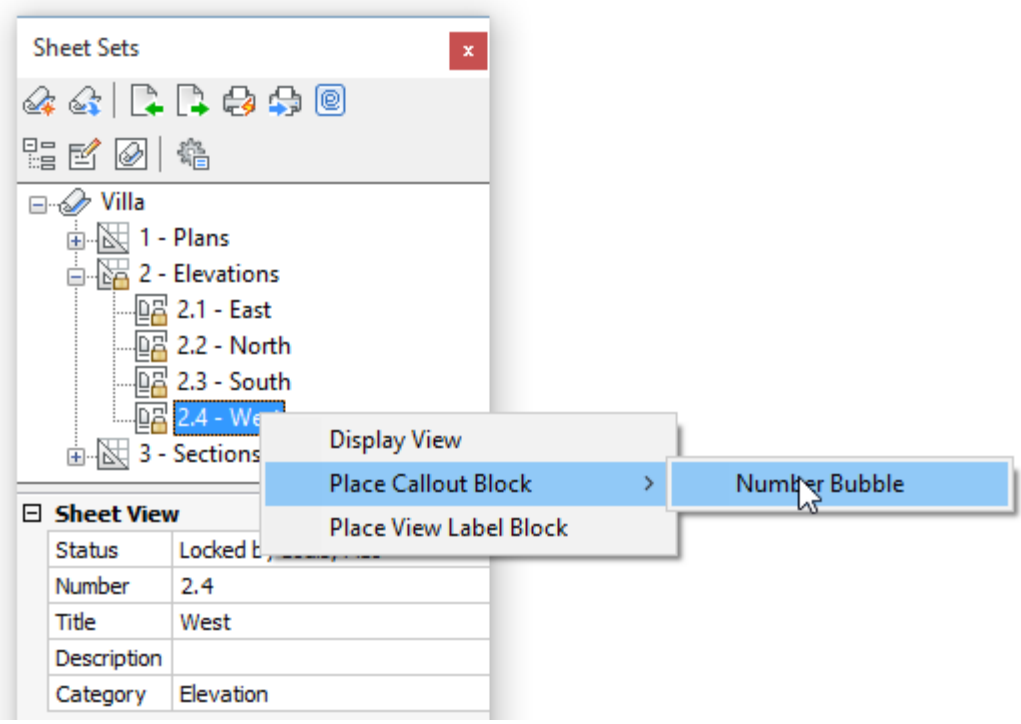
Specify the rotation angle or right click to accept the current angle.
The *View Label* block is inserted.
The Sheet Number (1), Sheet View Number (2), Sheet View Title (3) and Viewport Scale (4) of the selected sheet view are filled in automatically.



Inserting a Number Bubble for a Sheet View

Select the sheet view In the Sheet Sets panel.

Right click the sheet view you want the number bubble block to insert for.



Choose *Place Callout Block > Number Bubble* in the context menu.
You are prompted:
: ._CALLOUT
Units: Millimeters Conversion: 1
Insertion point for block:

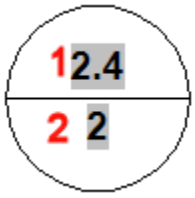
Scale factor for block <1.0>:

Rotation angle for block <0>:

Specify the insertion point.
You are prompted:
Scale factor for block <1.0>:

Specify the scale factor or right click to accept the current scale.
Your are prompted:
Rotation angle for block <0>:

Specify the rotation angle or right click to accept the current angle.
The *Number Bubble* block is inserted.
The Sheet Number (1) and Sheet View Number (2) of the selected sheet view are filled in automatically.



Inserting a Sheet List Table

A *Sheet List* table can be inserted on any sheet of the sheetset. In the current implementation the table content is not yet associative and therefore will not be updated automatically when sheets are added, removed or renamed.

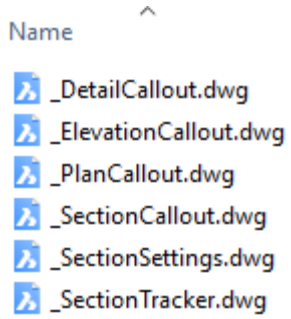
Right click the sheetset name in the *Sheet Sets* panel and choose *Insert Sheet List Table* in the context menu.
You are prompted: Select insertion point.

Specify a point in the drawing.

Customizing Callout Blocks

Source files of the various callout blocks and the section tracker exist in the *..\Support\Bim\Sections* subfolder of the *Roamable Root* folder.

By default this is:
C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\Support\Bim\Sections.



You can edit these files to customize the layout of the callouts and the section tracker blocks.

When a drawing is generated, the callout blocks needed are created from the callout blocks source file(s). When a BIM section is updated, the existing callout blocks in the target drawing are used.

Spatial Locations

Assigning a Building and Story to a Building Element

Command: BIMSPATIALLOCATIONS, BIMCLASSIFY

The *BimSpatialLocations* command opens the *Buildings & Stories* manager, which allows to edit site properties, and edit and create buildings and stories.

The *BimClassify* command classifies an entity as a *Building Element* and get a name and an internal GUID (= globally unique identifier).

A BIM model file can contain one site and multiple buildings. A building contains multiple stories.

Entities that are classified as a Building element have a *Building* and a *Story* property.

Assigning a building and story to a building element

Compose a selection set of building elements.
The shared properties of the selected entities display in the Properties Bar.

Expand the BIM node in the Properties Bar.

BIM	
Type	*Varies*
Name	
Description	
Building	
Story	
Composition	*Varies*
Display composition	Off
GUID	*Varies*

Click the *Building* Field, then click again and select a building.

Click the *Story* field, then click again and select a story.

To examine the model in the Structure panel

The Structure panel allows to examine the BIM model. By default, the structure of building elements is organized by spatial containment. Building elements are grouped first by Building, then by Story, then by BIM type, then by composition. This organization can be fully configured: any property, including all ifc properties, can be used as a grouping or sorting rule.

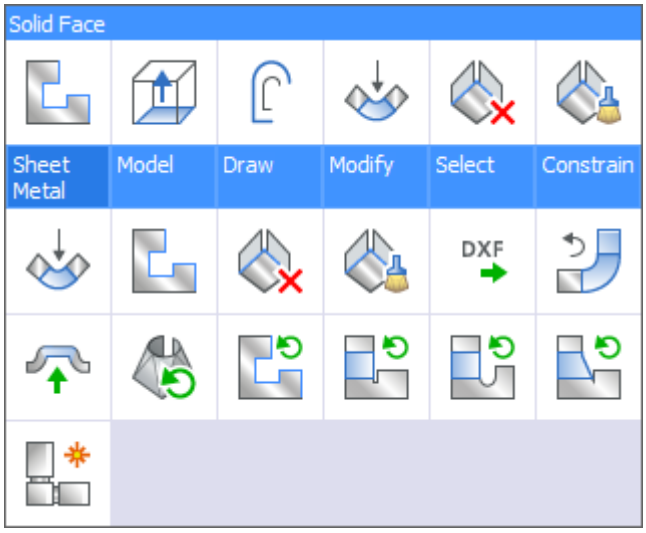
Sheet Metal

Commands and Toolbars

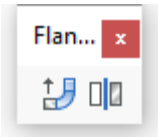
Sheet Metal Toolbar Icon	Flyout Toolbar Icon	Command	Description
		SmFlangeBase	Creates a base (initial) flange of a sheet metal part from a closed planar profile.
		SmExtrude	Creates a sheet metal part by extruding a polyline in the direction of the normal of the curve plane. Thickness is assigned with respect to the external dimensions of the polyline.
		SmLoft	Creates sheet metal part with lofted bends and flanges from two non-coplanar curves.
		SmConvert	Automatically recognizes flanges, bends and lofted bends in a 3D solid.
		SmFlangeEdge	Creates one or more flanges on a sheet metal part by pulling one or more edges of an existing flange.
		SmFlip	Changes the reference side for selected flanges with an optional shift of the flange geometry by thickness.
		BmInsert	Inserts form features.
		SmRibCreate	Creates an associative rib form feature on a flange face from a planar 2D curve.
		SmReplace	Replaces form features (including recognized ones) in sheet metal parts with form features from a built-in or user library.
		SmBendCreate	Converts hard edges (sharp edges between flange faces) and junctions into bends.
		SmBendSwitch	Converts bends to lofted bends
		SmflangeBend	Bends an existing flange along a line, obeying k-factor for given bend radius.
		SmJunctionCreate	Converts hard edges into junctions.
		SmJunctionSwitch	Changes a symmetrical junction feature to one with overlapping faces.
		SmFlangeConnect	Closes gaps between two arbitrarily oriented flanges.
		SmReliefCreate	Makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge .
		SmReliefSwitch	Converts a corner relief to a circular relief.
		SmReliefSwitch	Converts corner reliefs to a rectangular relief.
		SmReliefSwitch	Converts corner reliefs to a VType relief.
		SmReliefSwitch	Converts corner reliefs to a Smooth relief.
		SmReliefSwitch	Converts corner reliefs to a Round relief.

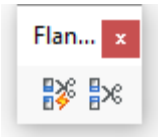
		SmReliefSwitch	Converts corner reliefs to a Rip relief.
		SmFlangeSplit	Splits a flange along a line drawn on its face.
		SmFlangeSplit	Splits a flange by a single vertex in the corner.
		SmImprint	Splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.
		SmDissolve	Removes sheet metal data from the selected faces or entities.
		SmDelete	Removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
		SmFlangeRotate	Rotates a flange.
		SmSelect Same Form Features	Selects all identical form features in the sheet metal model
		SmSelect Hard Edges	Selects all hard edges on sheet metal parts.
		SmRepair	Fixes problems specific to parts with lofted bend features: merges adjacent bends and provides tangential connection with flanges.
		SmParametrize	Creates a consist set of 3D constraints for the selected sheet metal part.
		SmUnfold	Unfolds the sheet metal body.
	OSM	SmExportOSM	Exports a sheet metal solid to the .osm (Open Sheet Metal) file format.
	DXF	SmExport2d	Exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.
	DXF	SmAssemblyExport	Performs a batch export of assembly components to unfolding, creating a .dxf per each 3D solid recognized as sheet metal part.
		FEATURECOLORS	Toggles the *FEATURECOLORS* system variable.

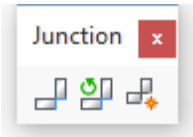
The Sheet Metal tools are available:
In the Quad cursor menu (depending on the feature under the cursor):

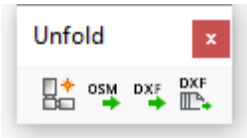


On the *Sheet Metal* toolbars:

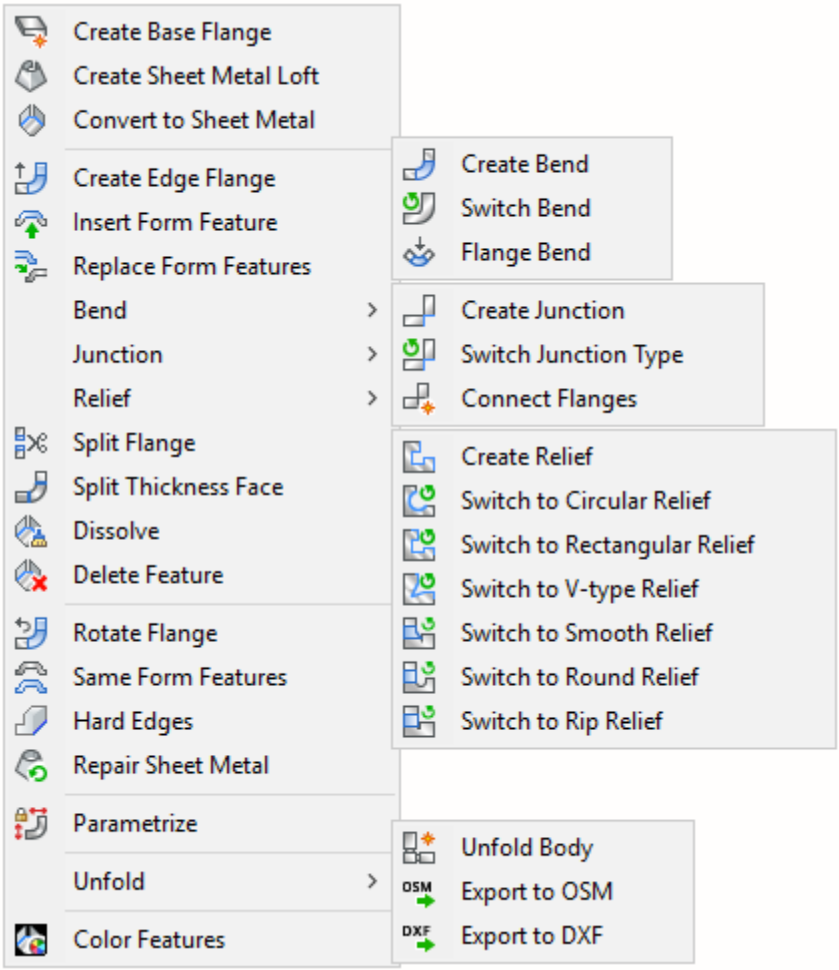




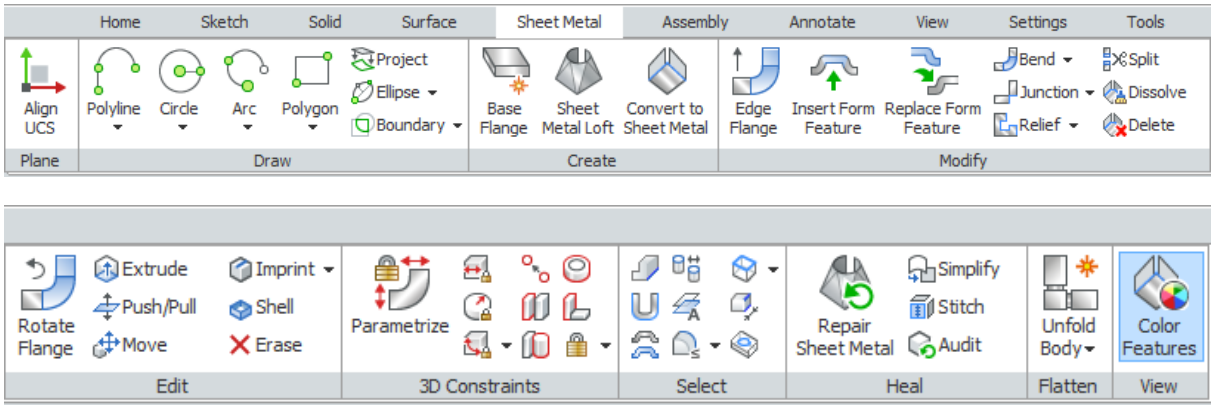




In the *Sheet Metal* menu (Sheet Metal Workspace only):



In the *Sheet Metal* tab of the *Sheet Metal* workspace Ribbon.

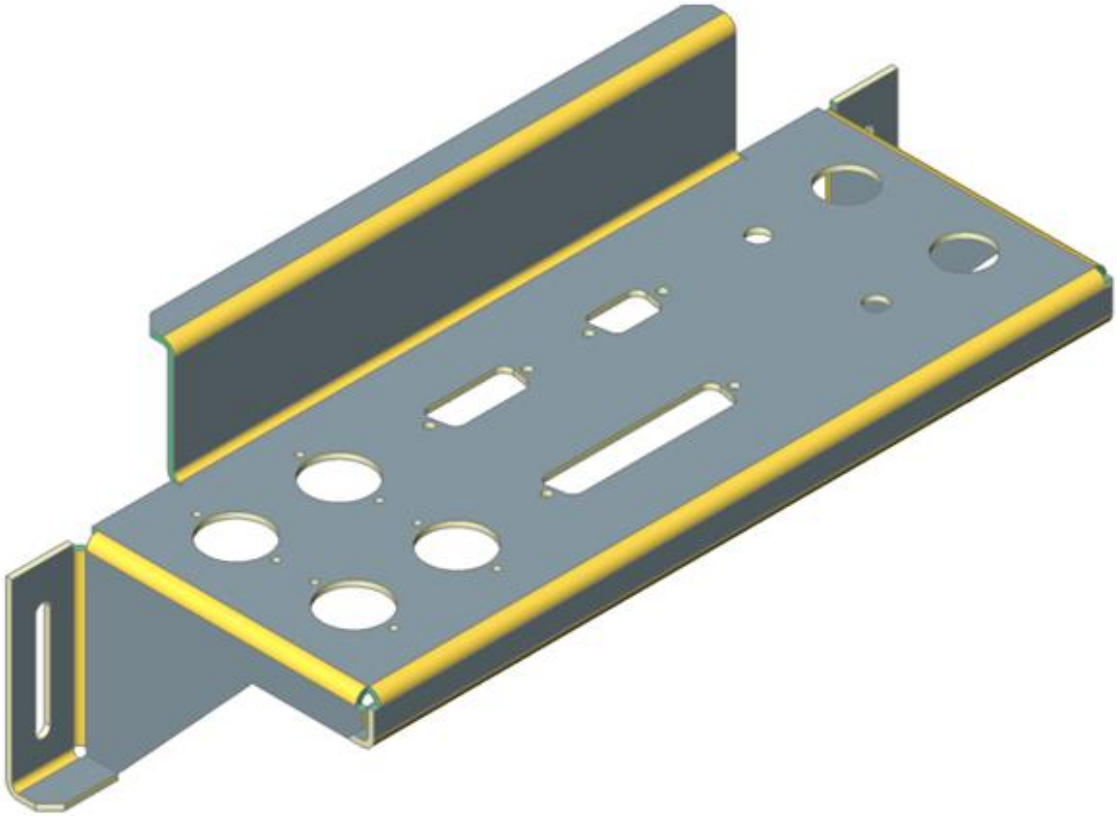


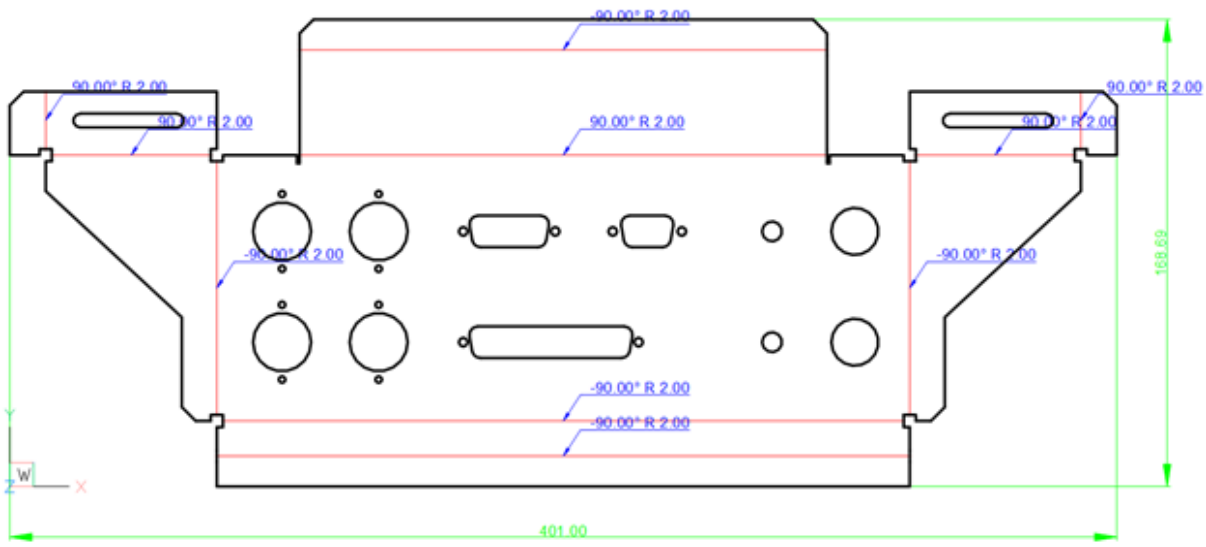
See also Direct Modeling tools.

SheetMetal Tutorials

Batch Processing in BricsCAD Sheet Metal
New Sheet Metal functions in BricsCAD V16
Feature coloring in BricsCAD Sheet Metal
Designing from scratch in BricsCAD Sheet Metal
Lofted bends in BricsCAD Sheet Metal

Sheet metal design allows you to model sheet metal parts and generate their unfolded representations with manufacturing information.





You can create complex sheet metal parts with BricsCAD easily and rapidly, because the design process is different from manufacturing process. Do not think in terms of a planar sheet that should be cut and bent, but model your part directly as you create solid bodies with Direct Modeling tools.

You can create sheet metal parts in BricsCAD in four different ways (or in any combination of them):

Create a base flange from a closed planar profile, then add flanges:

Pull its edges to create additional edge flanges with bends.

Create partial flanges with automatic creation of the corresponding bend reliefs.

Pull several adjacent edges to create several flanges at once with automatic creation of corner reliefs and junctions between them.

Bend a flange to create a new flange with a bend from its portion.

Create a combination of flanges and lofted bends connecting two 3D profiles using Sheet Metal Loft tool.

Create a combination of flanges and bends from planar profile using Sheet Metal Extrude tool.

Create a shelled body from a 3D solid, using Shell option of the SolidEdit command with a desired thickness, then convert it to a sheet metal part, and create bends and junctions from its hard edges.

Make holes by drawing closed profiles on faces of the flanges and pushing them through the flanges.

Adjust material thickness and bend radius using pre-defined parameters.

Apply Direct Modeling operations and 3D constraints to further adjust your design while always maintaining the design intent in terms of sheet metal features.

Automatically generate unfolded representation of your part and send it to a CAM system by exporting the corresponding 2D drawing with bend annotations in a .dwg or .dxf file.

You can further detail your sheet metal part using the following operations:

Make holes by drawing closed profiles on faces of the flanges and pushing them through the flanges.

Insert form features from a library. A library of parametric form features is installed with BricsCAD Sheet Metal and you can create your own form features.

Create rib form features from a 2D curve lying on the flange by the SmRibCreate command.

Adjust material thickness and bend radius using pre-defined parameters.

Apply Direct Modeling operations and 3D constraints to further adjust your design while always maintaining the design intent in terms of sheet metal features.

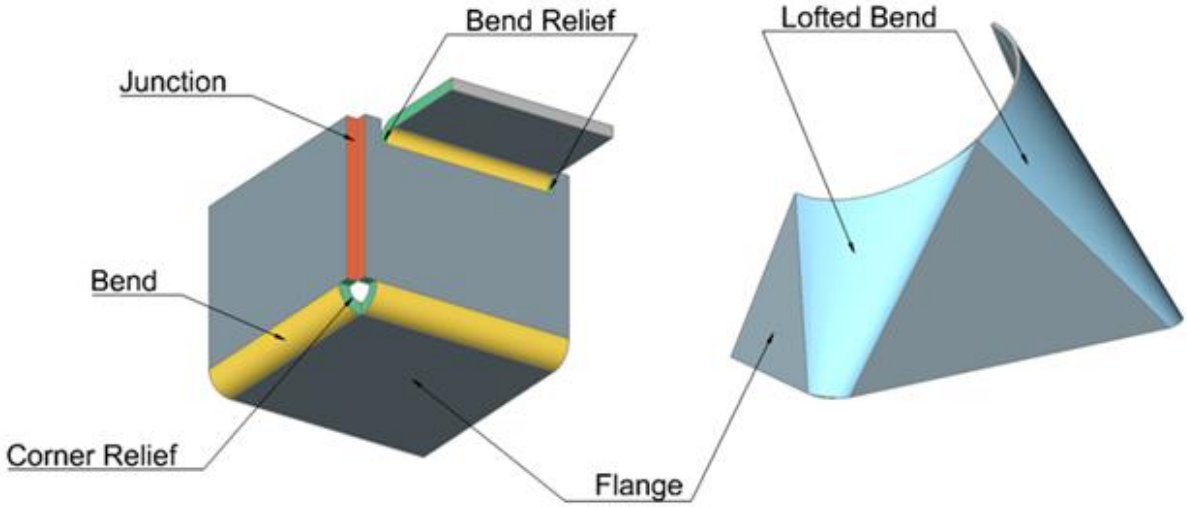
When your sheet metal part is ready, you can automatically unfold it and send the result to a manufacturing engineer by exporting the corresponding 2D drawing with bend annotations in a .dwg or .dxf file.

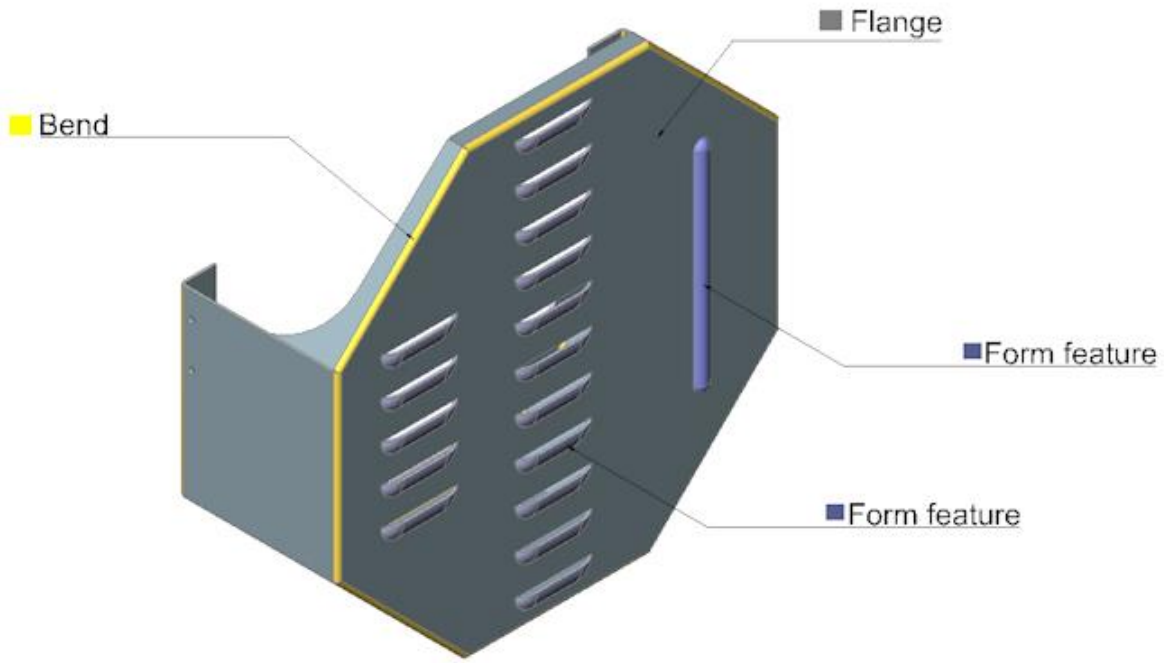
Definitions and Settings

Batch Processing in BricsCAD Sheet Metal
New Sheet Metal functions in BricsCAD V16
Feature coloring in BricsCAD Sheet Metal
Designing from scratch in BricsCAD Sheet Metal
Lofted bends in BricsCAD Sheet Metal

Sheet Metal Features

Form features are smart regions (groups of faces) of your 3D part. Each feature maintains specific spatial and parametric relationship between its faces and some adjacent faces. Form features allow you to embed design intent into your model. You do not create features by hand, they are created automatically depending on a particular geometric operation you apply.





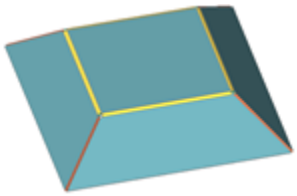
Flange

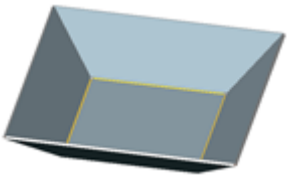
The main feature of any sheet metal part is a flange, which consists of two parallel planar faces. This distance between them is equal to the material thickness. When you modify the model, this distance relation is always maintained automatically. Other faces, which are adjacent to flange faces and do not belong to bends are called thickness faces. They are always perpendicular to flange faces.

Flange features are created by SmFlangeBase, SmExtrude, SmLoft, SmFlangeEdge, SmFlangeBend and SmConvert commands.

If the FEATURECOLORS system variable is ON, the display color distinguishes between the reference face of a flange and its opposite side. Parametric modifications of a sheet metal part, such as thickness change, do not move reference faces, if possible. The SmFlip command swaps sides of a selected flange so that reference faces are on the other geometric side of the flange; optionally the command shifts the flange over the thickness of the sheet metal part.

The images below illustrate the color legend for the flange faces of the same part.





Reference Faces *Other Faces*

Bend

Two flanges are connected by a bend. A bend consists of two coaxial cylindrical faces, which are always tangent to the adjacent planar faces of the flanges.

Bend features are created by SmFlangeBase, SmExtrude, SmFlangeEdge, SmFlangeBend and SmConvert commands.

Wrong Bend

Wrong bend features are recognized by the SmConvert command and reflects assumptions on the user's design intent. Like regular bends, wrong bends connect flanges, but the have geometrical failures, such as radii or angle mismatches which prevent using the geometry as regular bend features. Before unfolding the part all wrong bends must either be repaired by SmRepair or removed and recreated by SmBendCreate.

Lofted Bend

Lofted bend represents an abstraction of the lofted surface, which for any point on the surface has a line which completely belongs to surface. The result of thickening of lofted surface is called lofted bend. Lofted bends can connect one or two flanges or be stand-alone, without flanges at all. Sheet metal part main contain flanges, bends, reliefs, junctions and lofted bends all together.

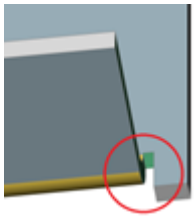
Differences between bends and lofted bends:

	Bends	**Lofted Bends**
Surface	Cylindrical	Cylindrical, conical or spline
Unfolding	Exactly	The approximate unfolding of lofted bends is controlled by an adjustable number of bend operations, which are called the Number of Subdivisions. The more operations are performed, the smoother the real part will be produced and the better it approximates the initial design.

Lofted bend features are created by the SmLoft and SmConvert commands.

Bend Relief

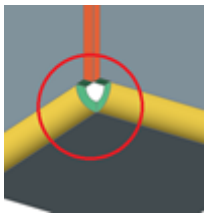
Group of faces representing a technological cut between two flanges of different width connected with a bend. The bend relief feature maintains the distance between two opposite faces of the cut.



Bend relief features are created by SmFlangeEdge, SmBendCreate and SmReliefCreate commands.

Corner Relief

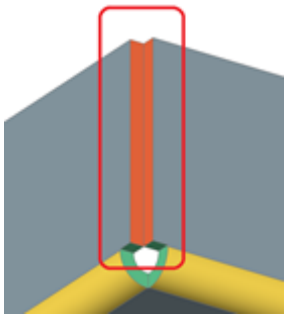
Group of faces representing a cut in the corner where three flanges meet together. Corner relief feature maintains the form and size of this cut.



Corner relief features are created by SmFlangeEdge, SmBendCreate and SmReliefCreate commands.

Junction

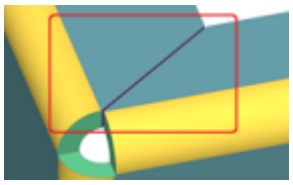
Junction feature consists of two thickness faces of adjacent flanges, which are not connected via bend.



Junction features are created by SmFlangeEdge, SmJunctionCreate and SmFlangeConnect commands.

Miter Feature

Miter features represent a split between coplanar flanges which is created by SmFlangeEdge and SmFlangeSplit commands.



Form Feature

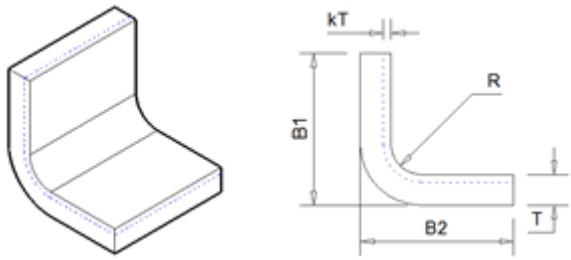
Form and rib features represent a rich variety of shapes which are produced by punching and wheeling.

Two representations of the form features are important: one in 3D and a 2D symbol in the unfolded geometry. In case of imported geometry the symbol is obtained by the projection of the 3D faces of the form feature In case the form feature was inserted/replaced from Form Feature Library, the unfolding symbol is taken from the library file.

Form and rib features are created by BmInsert, SmRibCreate and SmConvert commands.

K-Factor

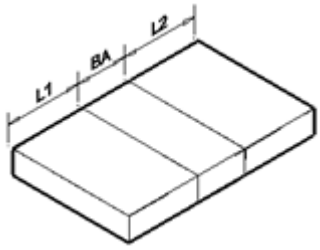
The material deformation properties for bending is based on the assumption that there is internal surface for the sheet metal part which is not deformed when the sheet is bent. From the variety of internal surfaces the one is selected, which is equidistant from one side of the bend. The surface is local for each bend, and for simple cases it can be propagated on entire part, like the one on the picture. We assume that this surface is not stretched stretch during bending.



T: thickness of the sheet metal part

R: internal radius of the bend

The K-factor (k) is the ratio of the location of the neutral surface to the material thickness. As a result the neutral surface lies at a distance kT from the internal surface of the bend. The bend radius of the neutral surface equals R + kT. The K-Factor is a simple geometric calculation of the location of the neutral surface. Forming stresses and other unknown (error) factors are not taken into account. The K-Factor depends on many factors, including the kind of material, bending type, tools, etc ... The K-Factor typically lies between 0.3 and 0.5. The default K-Factor for a bend radius equal to the material thickness (T) is 0.27324.



In the unfolded state of a sheet metal part, the flanges (L1 and L2) are not stretched. The bent part is indicated as BA: bend allowance. The BA equals the length of the unfolded neutral surface:

BA = BendAngle * (R + kT)

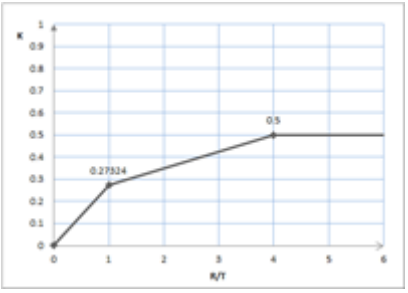
In real world practice it is hard to measure the K-Factor or the bend allowance. The following formula allows to calculate the tangential Bend Deduction (BD):

BD = B1 + B2 - Unfolded Length = B1 + B2 - (L1 + L2 + BA)

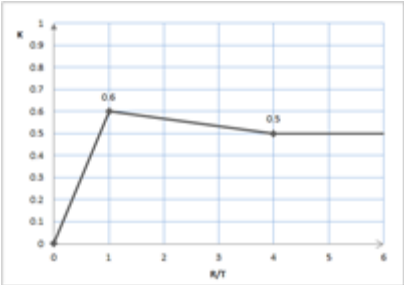
For simple cases the K-Factor value can be overridden in the browser, for maximal precision a bend table containing tangential bend deductions has to be provided. For each sheet metal part, you can either specify a K-factor or use the default one.

To change the K-Factor for a sheet metal part:

To change the K-Factor for a sheet metal part, select the root node in the Mechanical Browser and type a value in the K-Factor field. The value must lie in the range [0, 1], since the neutral surface is located inside the sheet metal part. In BricsCAD the K-Factor is defined for bend radius equal to the material thickness (R/T = 1); to compute the value of K-Factor for an arbitrary bend radius, BricsCAD uses a special interpolation technique which is proved to be valid for industrial sheet metal applications. The image below shows the interpolation strategy: for R/T < 1 the linear interpolation is done between 0 and 0.27324, for 1 < R/T < 4 the K-Factor is linearly interpolated between 0.27324 and 0.5 and if R/T is greater than 4, the K-Factor is constant and equals 0.5 . Physically it means that if the bend radius is much larger than the thickness (at least 4 times), the material stretch is can be neglected.

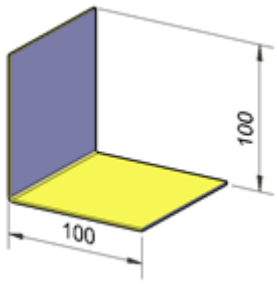


The K-Factor parameter one can control in the browser is for R/T. I.e. if we set the value to 0.6, the interpolation law leads to the following result:

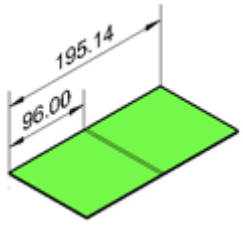


The varying K-Factor in the browser.

T = 2, R = 2, B1 = 100, B2 = 100

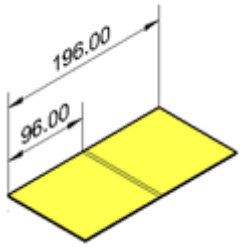


Setting K=0 gives L1=L2=96, Unfolded Length = 96, thus BA is 3.14.

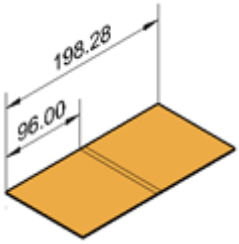


This value fits the formula exactly: a straight angle in radians roughly equals 1.57 (PI/2). The default K-Factor 0.27324 gives an Unfolded Length = 196.

Or: L1=L2=96 (flanges are not deformed), the BA = 4 as we can compute from the dimensions (196 - 2*96) or from the Bend Allowance formula: BA = BendAngle * (R + kT) = 1.57 * (2 + 2*0.27324) = 4



Finally, maximizing K-Factor results in an Unfolded Length = 198.28. This is the maximal value one can achieve in this example, since the neutral surface is taken from external side of the bend.



The Bend Deduction (BD) for k = 0.27324:

BD = B1 + B2 - Unfolded Length = 100 + 100 - 196 = 4

Bend Table

Bend tables are a more reliable way to express material deformation properties. When you bend a flat sheet of a particular material, you can measure its length before and after this process. You repeat this procedure for different bend angles, bend radius, and sheet thickness and save the measurements in a bend table. Then these measurements can be taken into account to compute the correct unfolded length for all parts made of the same material.

In a bend table, you can add the results for as many different bend angles as you want. However, in most cases it suffices to measure the length for a 90 degrees angle. BricsCAD will automatically compute the length for other bend angles using a reliable interpolation technique.

Bend tables are stored in a *.csv (comma separated values) file format. You can create such a table in an ASCII text editor, such as Notepad, but a more reliable and easier way is to create a table in a spreadsheet, such as Microsoft Excel or LibreOffice and then export it to .csv file format.

Bend tables in BricsCAD have the following structure:

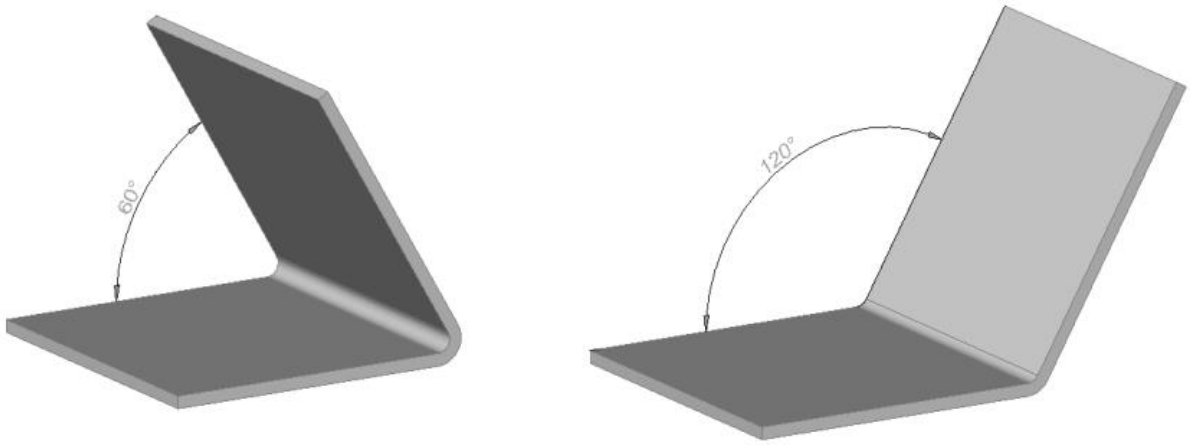
	A	B	C	D	E	F	G
1	BricsCAD						
2	Version	1					
3	AngleType	Internal					
4	LengthType	BendDeductionTangent					
5							
6	Thickness	1					
7	AngleValues		45	60	90	120	135
8	Radius	1	1	1.333	2	0.976	0.657
9	Radius	2	0.466	1.081	2.31	1.004	0.641
10	Radius	3	-0.069	0.828	2.621	1.033	0.624
11							
12	Thickness	1					
13	DieWidth	10					
14	AngleValues		45	60	90	120	135
15	Radius	1	1.225	1.533	2.15	1.076	0.732
16	Radius	2	0.616	1.214	2.41	1.071	0.691
17	Radius	3	0.006	0.895	2.671	1.066	0.649

The following rules apply:

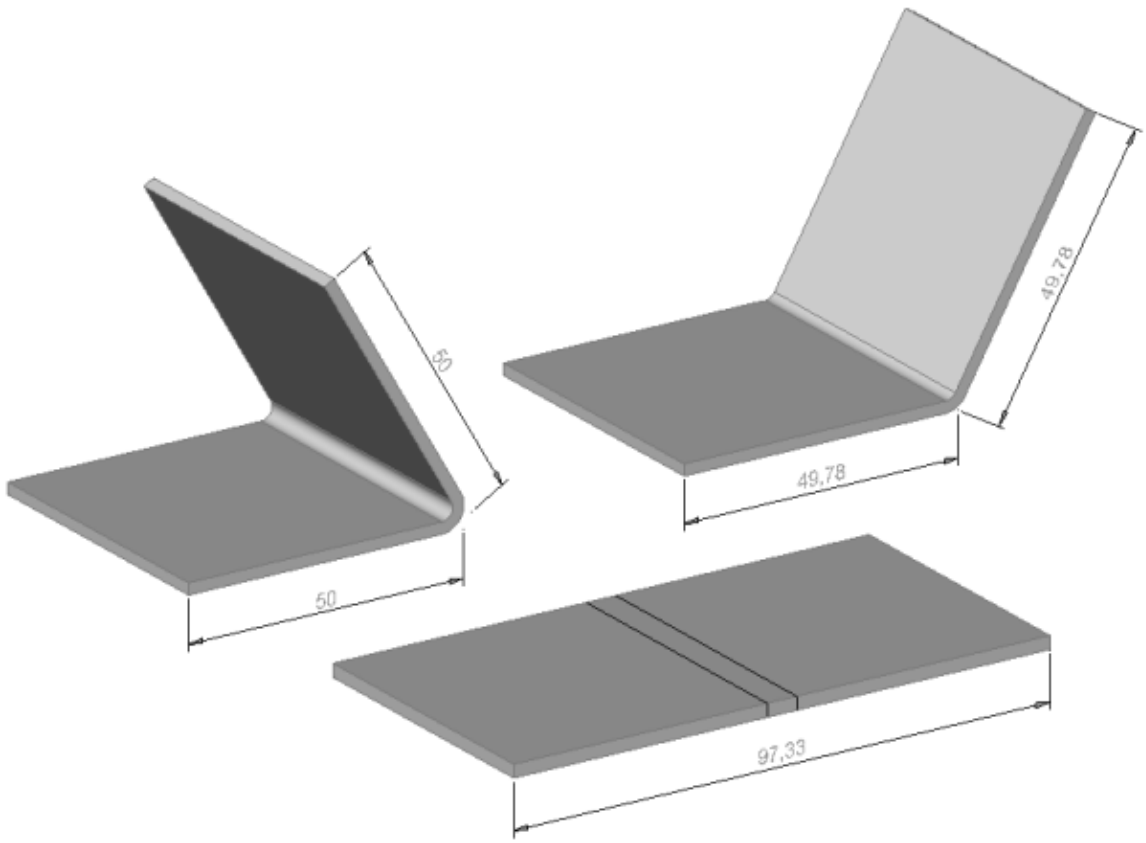
The content of the first cell reads "BricsCAD”.

Version: currently supported version is 1.

AngleType: Currently only internal bend angles are supported:



LengthType: the semantics of the value in the cells of the bend table corresponding to a particular bend angle and bend radius. Now only bend deduction measured from the tangent point is supported (”BendDeductionTangent”). Bend deduction (BD) is the difference between the sum of the lengths of two flanges measured to the tangent point on a 3D model and the length of the same fragment in the unfolded state:



Within the defined header, several tables can exist. For each table, the first key is the of the sheet metal part, the second key, *DieWidth*, is optional. The *Thickness / DieWidth* pairs must be unique. If the *DieWidth* is not defined, only one unique *Thickness* value key is allowed, but multiple thickness values can be added. When a bend table is used to control the unfolding process, the *Thickness* and *DieWidth*, defined in the sheet metal part, must exist in the bend table. If the table is not found the default K-Factor policy is used.

Once the bend table is found in the .csv, it will be used to unfold the sheet metal part. Using the sample bend table above, let’s have a look a few cases:
T = Thickness, R = Internal Bend Radius, A = Bend Angle

1. T = 1, R = 2, A = 45. Then after unfolding it will obtain the BD = 0.466, because both R and A are found in the bend table for the given thickness.

2. T = 1, R = 2.5, A = 45. A = 45 is found, but R = 2.5 is missing: the interpolation of BD between R=2 and R=3 is used.

3. T = 1, R = 2.5, A = 75. Neither R nor A is found in the table: the following cells are used to calculate the value of BD: (R=2, A=60), (R=3, A=60), (R=2, A=90) and (R=3, A=90).

The top-of-the-edge algorithm is used to calculate the interpolation. For example, linear interpolation on adjacent BD values render unnatural results, which will not match the real bending results. Also keep in mind that for a given T, R and A not all BD values make sense: a random one might place the neutral surface outside the sheet metal part, or expressed in K-Factor (which can be deduced from BD), the K-Factor can be out of range [0, 1]. In this case, the BD for the default K-Factor will be used.

If you produce your sheet metal parts with different tools, you can describe them in one bend table using *DieWidth* parameter.
A sample bend table is provided for your convenience in the *..\Samples\Mechanical\bend_tables* subfolder of the BricsCAD program folder (e.g. *C:\Program Files\Bricsys\BricsCAD V14 en_US\Samples\Mechanical\bend_tables*). Please notice that writing rights to this folder might be limited. In such case you need to copy the bend tables to a different folder in which you have full access rights.

To set the bend table for your sheet metal part:

Select the root node in the *Mechanical Browse*r.

Select the *Bend table* field.

Click the Browse button ( ).

Select the *.csv file then double click or press the *Open* button.

To update the bend table set previously:

Select the root node in the *Mechanical Browser*.

Select the *Bend table* field.

Press the *Update* button ( ).

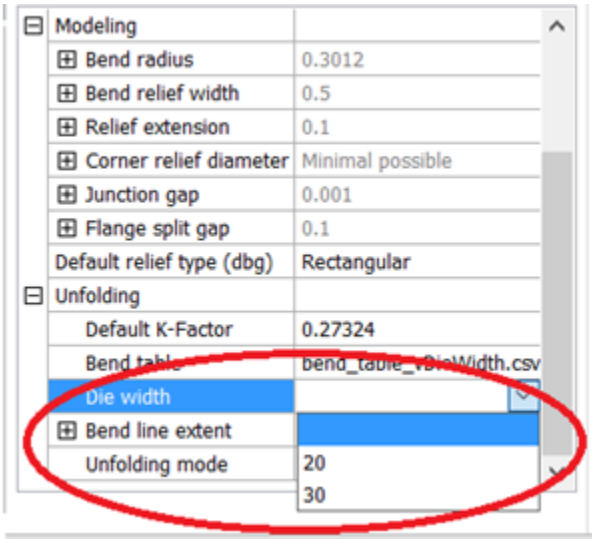
If the bend table is not assigned to the document or has no *DieWidth* secondary key, the *DieWidth* parameter can not to assigned to single bend or all the bends. In order to make the assignment, do the following steps:

Select .csv file *DieWidth* secondary keys and thickness matching the current part. On given example we assume that part has thickness of 1 mm and bend radius also equals to 1. We will vary bend deduction to different tools: first subtable is when tool is not specified, second subtable is for case when tool *DieWidth* is equal to 20, and last one is for tool with *DieWidth* equal to 30.

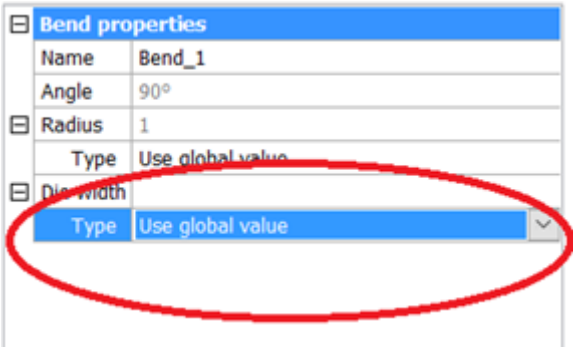
	A	B	C	D
1	BricsCAD			
2	Version	1		
3	AngleType	Internal		
4	LengthType	BendDeductionTangent		
5				
6	Thickness	1		
7	AngleValues		90	
8	Radius	1	2.1	
9				
10	Thickness	1		
11	DieWidth	20		
12	AngleValues		90	
13	Radius	1	1.7	
14				
15	Thickness	1		
16	DieWidth	30		
17	AngleValues		90	
18	Radius	1	1.8	

Assign the bend table to the sheet metal part.

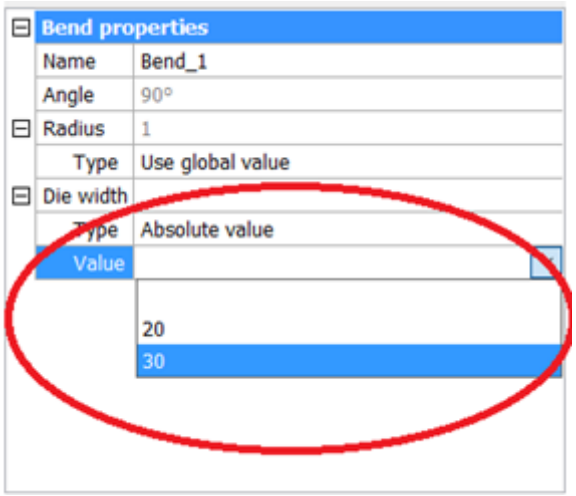
Check that in the *Mechanical Browser* new fields appeared:



In this global setting we can select any default *DieWidth* value suitable for the part thickness. The value will be assigned by default to the bends in the part, which can be checked by the property of the bend (click on the bend in the *Mechanical Browser*):



However, we can select a specific tool which will be used for the given bend by expanding the drop-down list:

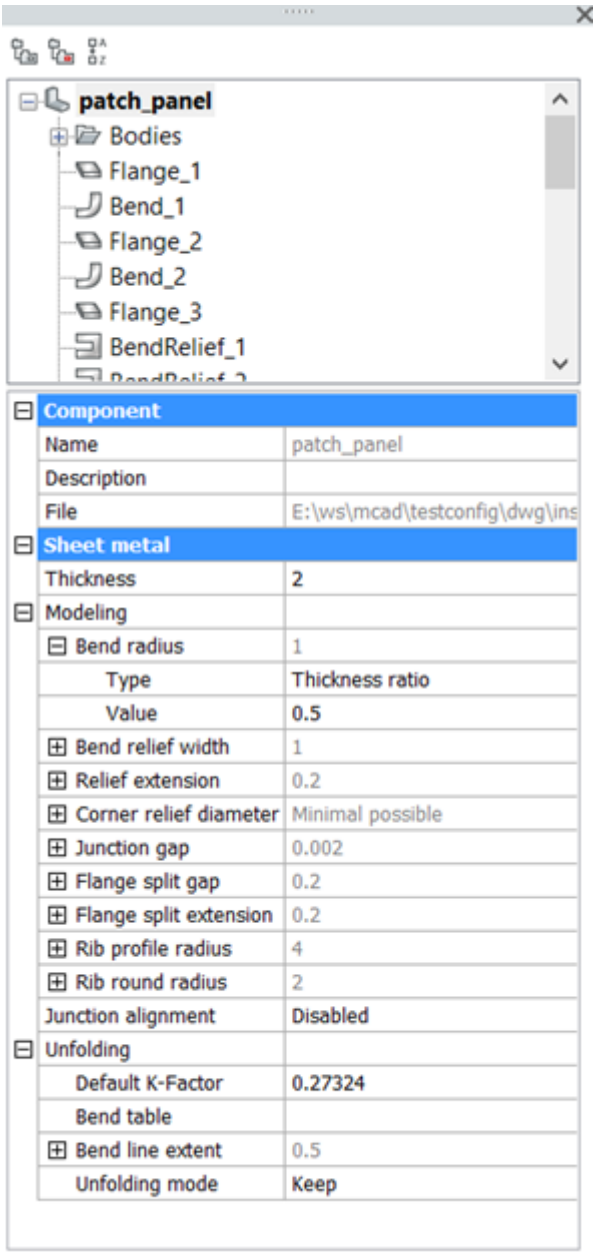


With the given selection, unfolding of Bend_1 will use a third subtable from our bend table for bend deduction and uses the value 1.8. For the rest of the bends, unless *DieWidth* is set for them, the default no-die-width value will be used, which will trigger first the bend table and uses value 2.1 for 90-degrees with a bend radius equal to 1.

NOTE After assigning the *DieWidth* to the bends and saving the document, the value is saved to the bends. However, next time the drawing is opened, in order to assign the *DieWidth* value to another bend, you have to reload the bend table first (press the green arrow button), see To update the bend table set previously. After the reload the values from the bend table will be available.

The Mechanical Browser for Sheet Metal

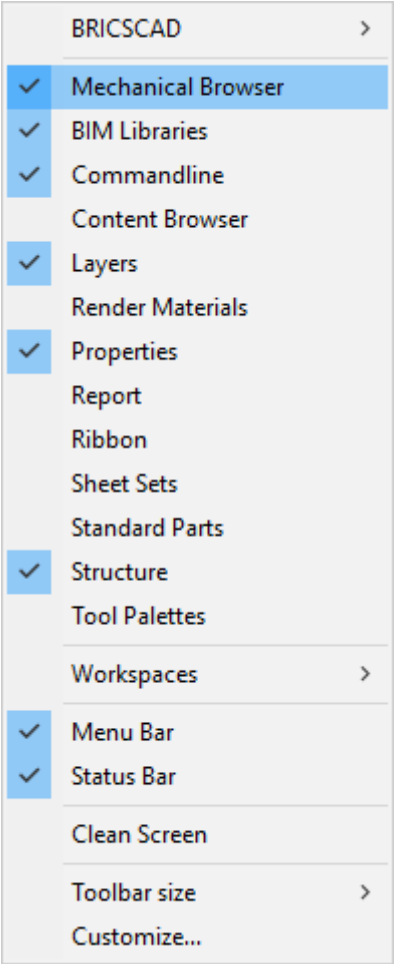
All features of a sheet metal part are listed in the *Mechanical Browser.*
*W*hen you select a feature in the *Mechanical Browser*, its faces are highlighted in the model area.



The *Mechanical Browser* opens automatically when switching to the *Sheet Metal* or *Mechanical* workspace.

To open the *Mechanical Browser* manually do the following:

Move the cursor over a toolbar, then right click.
A context menu displays:



Choose *Mechanical Browser* in the context menu.

NOTE When you right click a feature in the browser, then select *Dissolve* from the context menu. the selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

Feature Colors

For better experience and easy distinguishing between the features different colors are applied to the various features in the geometry. These colors are for visualization reasons only and will never show in generated documents, paper space views, block inserts, etc.

The display of feature colors is controlled through the FEATURECOLORS system variable and a series of system variables, one for each feature color. Although the default feature colors are selected carefully to combine with each other, you can set them differently in the *Sheet Metal* settings group on the Settings dialog.

Sheet Metal	
Feature colors	
Feature colors	☑ Toggle feature colors
Flange feature color	(144,164,174)
Flange feature reference side color	(104,164,174)
Form feature color	(135,145,225)
Bend feature color	(255,220,80)
Lofted bend feature color	(160,220,250)
Wrong bend feature color	(255,51,0)
Bend relief feature color	(100,210,150)
Corner relief feature color	(100,210,150)
Junction feature color	(255,110,64)
Miter feature color	(175,70,216)

To toggle the display of feature colors:

Click the *Color Features* tool button ( ) on the *Sheet Metal* toolbar.

Click the *Color Features* tool button ( ) on the *Sheet Metal* ribbon bar.

Choose *Color Features* in the *Sheet Metal* menu.

Layer Properties of 2D Geometry

The 2D geometry, generated by SMUNFOLD and SMEXPORT2D, has different colors for contours, bend lines up, bend lines down, overall dimensions, form features up, form features down, bend annotations and attributes.

Each entity type is placed on a dedicated layer of which the color is controlled trough a series of system variables:

Sheet Metal	
Feature colors	
Initial parameters	
Layers for unfolding	
Target CAM	
Contour Layer	
Color of the contour layer	White
Linetype of the contour layer	Continuous
Lineweight of the contour layer	0.30 mm
Bend Lines Up Layer	
Bend Lines Down Layer	
Overall Dimensions Text Layer	
Form Features Up Layer	
Form Features Down Layer	
Bend Annotations Text Layer	
Attributes Layer	

you can set the colors differently in the *Sheet Metal/Layers for unfolding* settings group on the Settings dialog.

If the 2D geometry of a form feature was taken from library component, its properties are controlled by the properties of this component at the moment of insertion. If the 2D geometry was generated automatically, which is a case of unfolding of imported form features or library features when *Unfolding Mode* is set to *Project* or *Contour*.

Managing Initial Parameters

A set of initial parameters controls the default settings used by commands such as SmFlangeBase, SmExtrude, SmFlangeEdge, SmFlangeBend, SmLoft and SmConvert in a newly created model. The initial parameters are saved in the drawing, which allows to create a set of .dwt template drawings for each type of sheet metal models created in a company.

Sheet Metal	
Feature colors	
Initial parameters	
Modeling	
Thickness value	2
Bend radius value	1
Bend radius type	0x0002 (2)
1	☐ Absolute value
2	☑ Override bend radius in SMCONVERT
Bend relief width value	0.5
Bend relief type	[0] Thickness ratio
Relief extension value	0.1
Relief extension type	[0] Thickness ratio
Corner relief diameter value	-1
Junction gap value	0.001
Junction gap type	[0] Thickness ratio
Junction alignment to relief	[0] Disabled
Flange split gap value	0.1
Flange split gap type	[0] Thickness ratio
Flange split extension value	0.1
Flange split extension type	[0] Thickness ratio
Rib profile radius value	2
Rib profile radius type	[0] Thickness ratio
Rib round radius value	1
Rib round radius type	[0] Thickness ratio
Unfolding	
K-Factor value	0.27324
Form feature unfolding mode	[4] Symbol
Bend line extent type	[0] Thickness ratio
Bend line extent value	0.25

Sheet Metal Operations

Creating the Base Flange

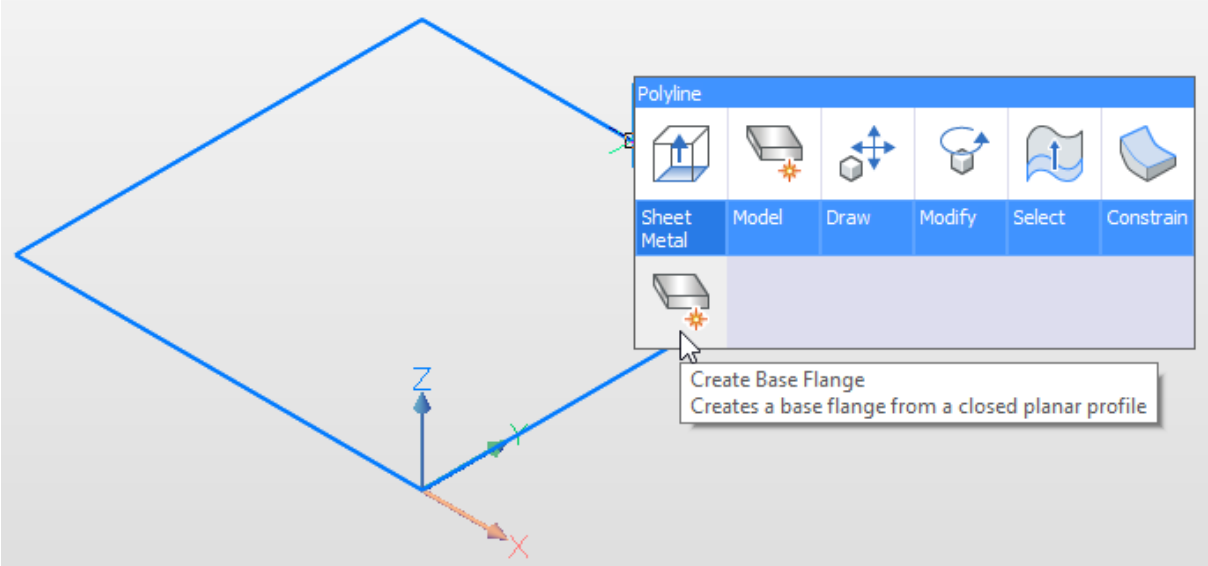
Start to design your sheet metal part with creating its base flange. To create a base flange, select a closed planar profile and call the SmFlangeBase command.

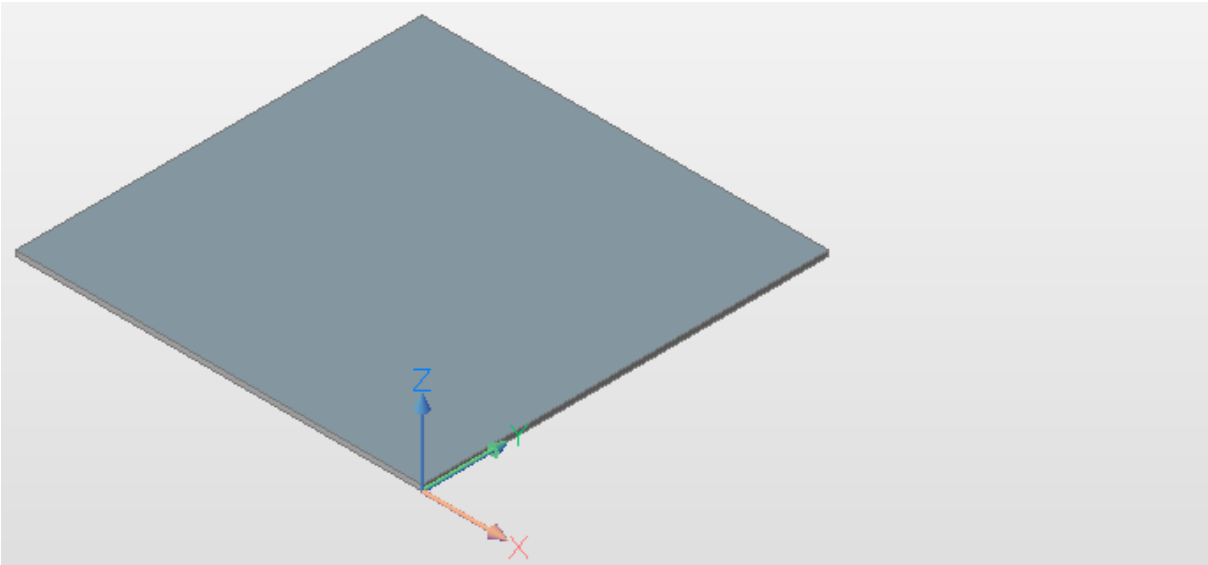
Do one of the following:

Click the *Create Base Flange* tool button ( ) on the *Sheet Metal* toolbar.

Choose *Create Base Flange* in the *Sheet Metal* menu.

Choose *Create Base Flange* in the *Sheet Metal* group of the Quad cursor menu:





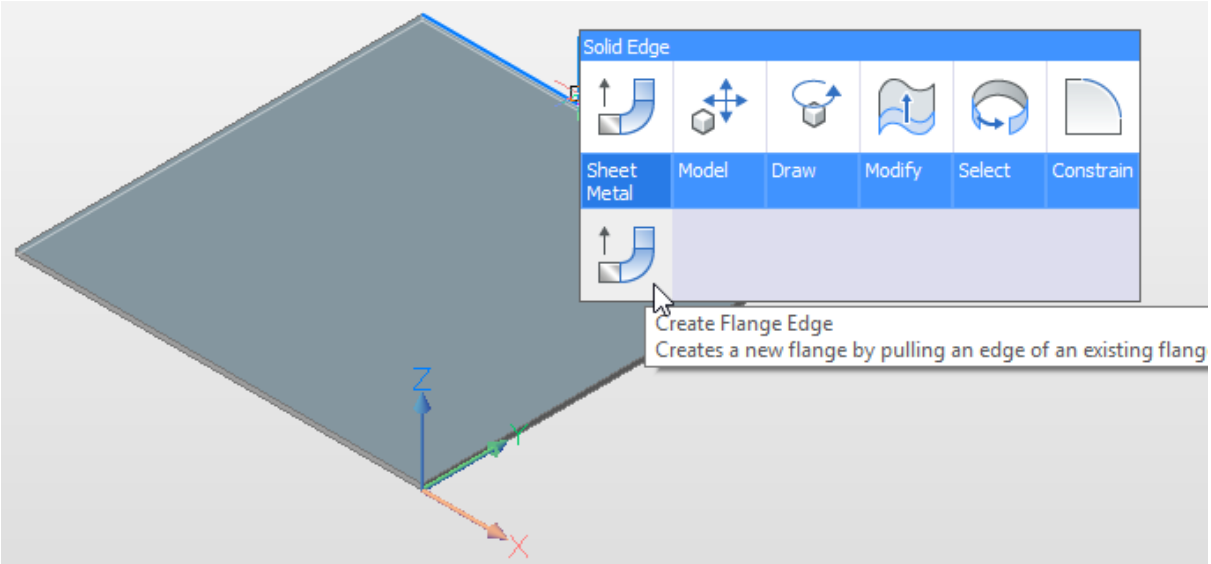
A base flange is a body created by extruding the selected profile to a height equal to the default value of the *Thickness* property of the sheet metal part

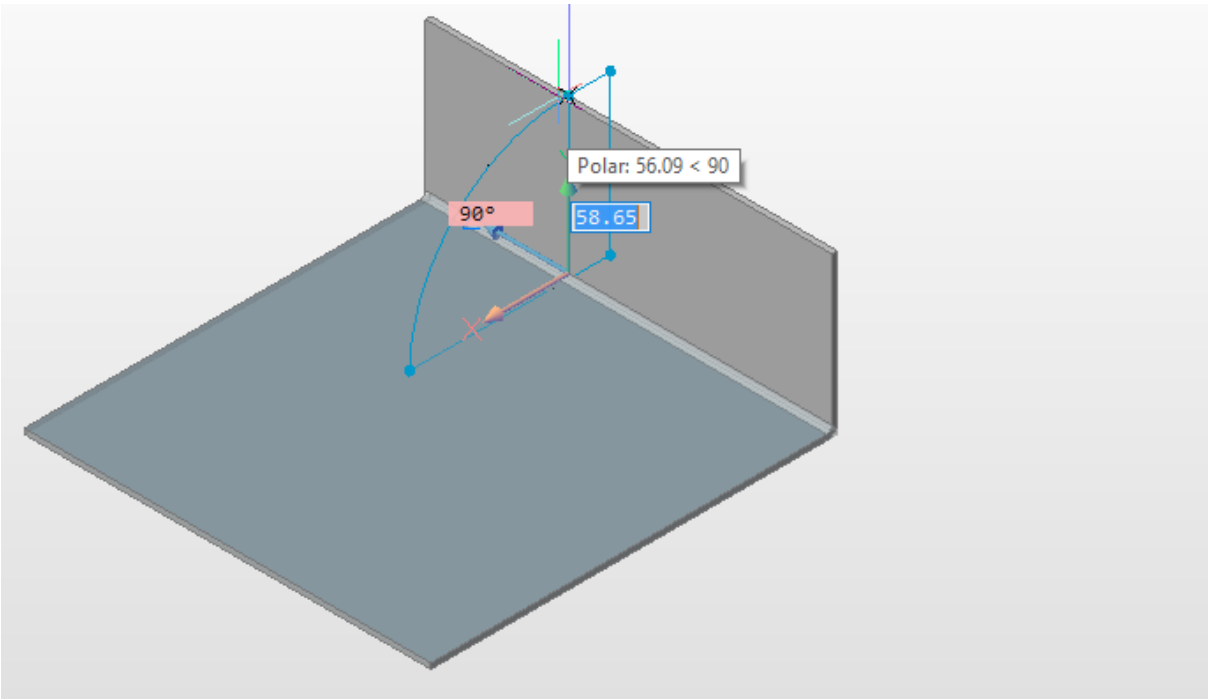
To change the thickness of your sheet metal part, type the appropriate value in the *Thickness* field in the *Mechanical Browser*.

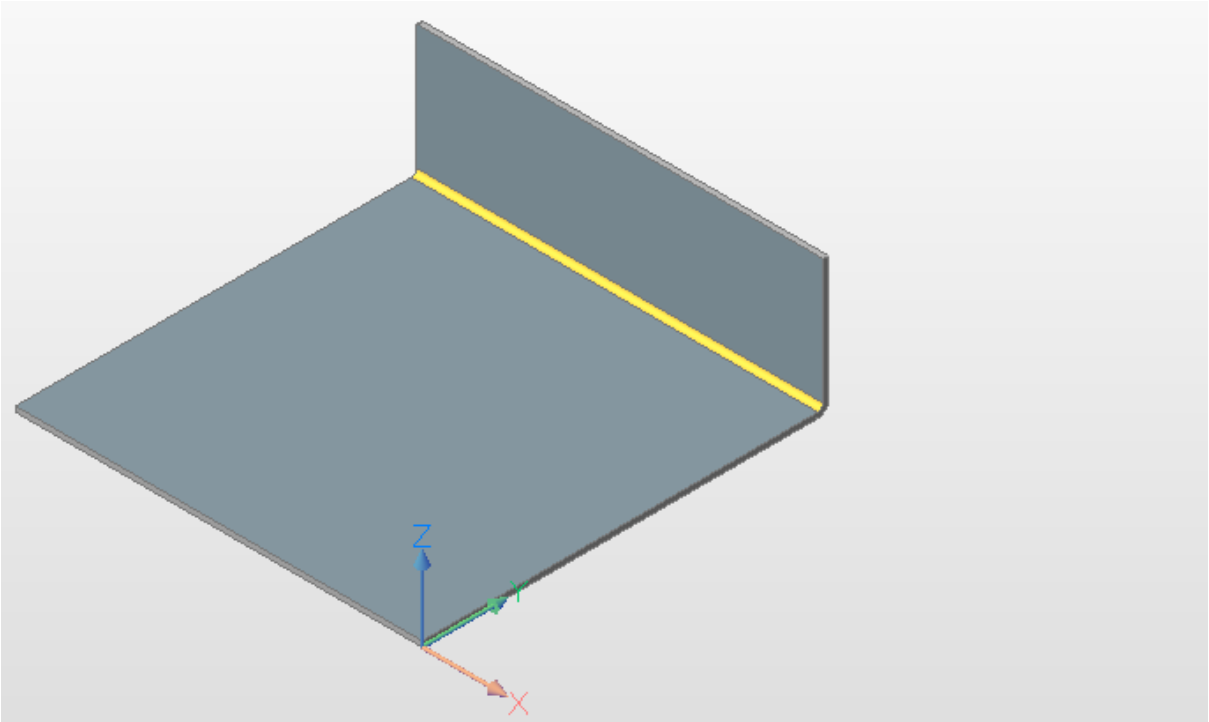
Creating Edge Flanges

You can add an additional flange to your sheet metal part by pulling a linear edge of an existing flange. Select an edge and select the SmFlangeEdge command in the *Sheet Metal* section of the Quad cursor menu.

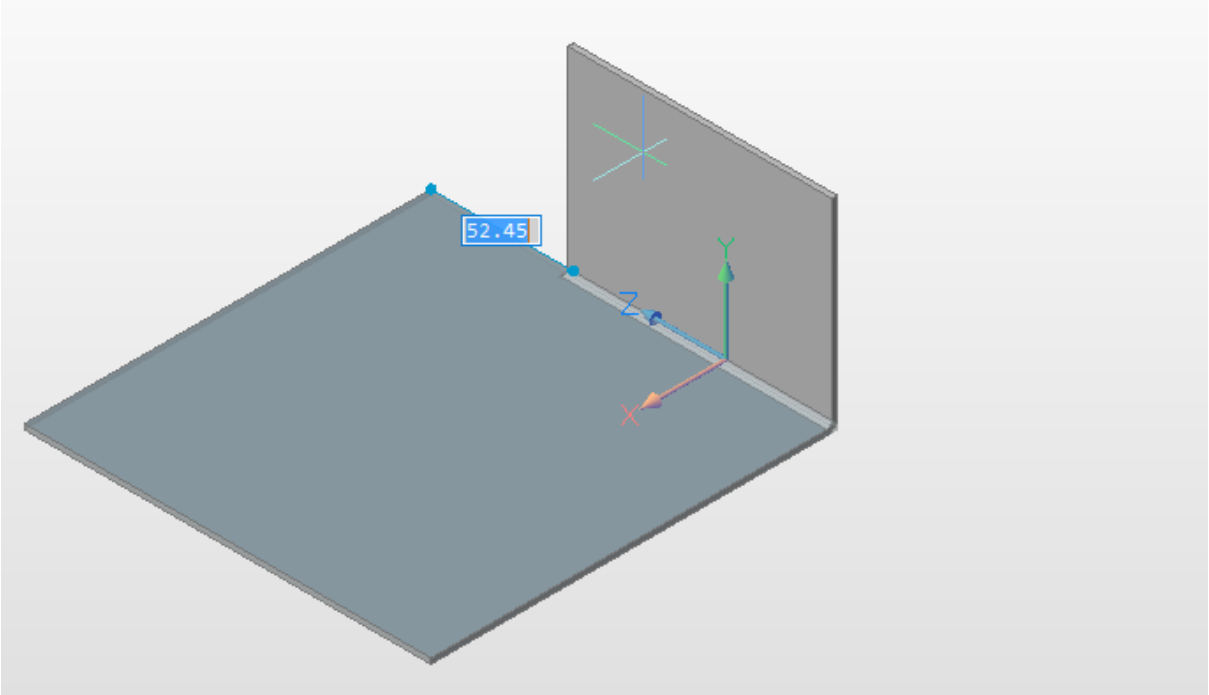
Move your mouse pointer to define the desired length of the edge flange and the angle between two flanges. You can also use the corresponding dynamic dimensions to type the desired values.

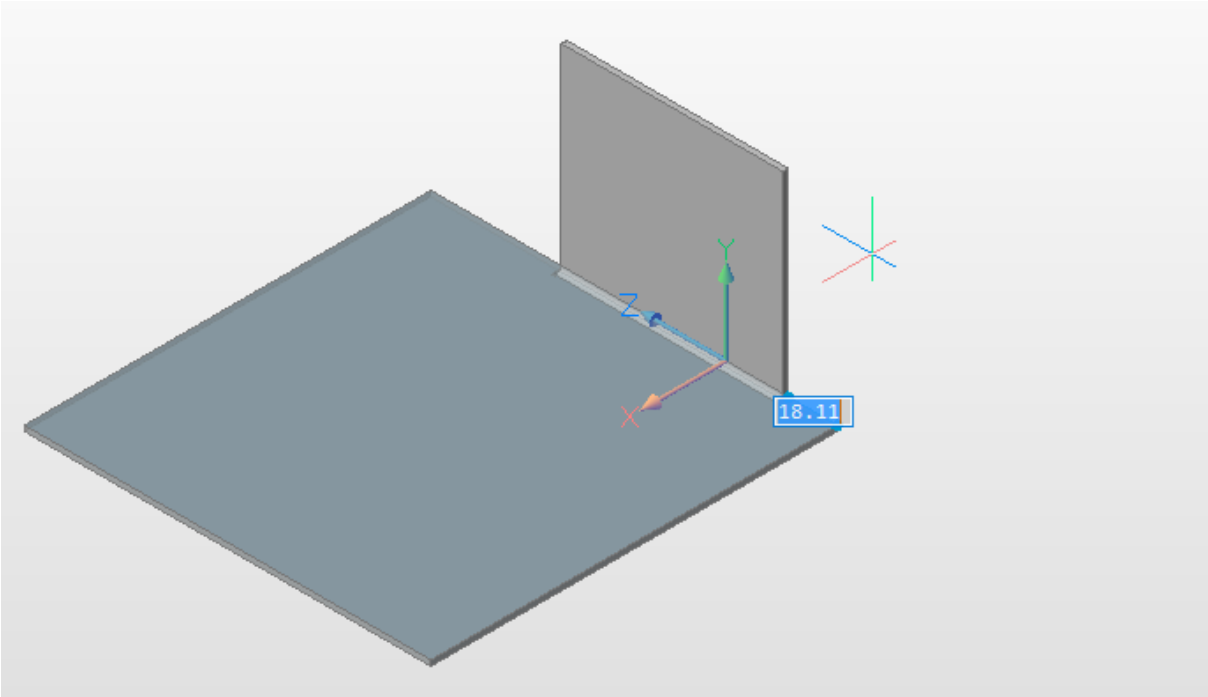


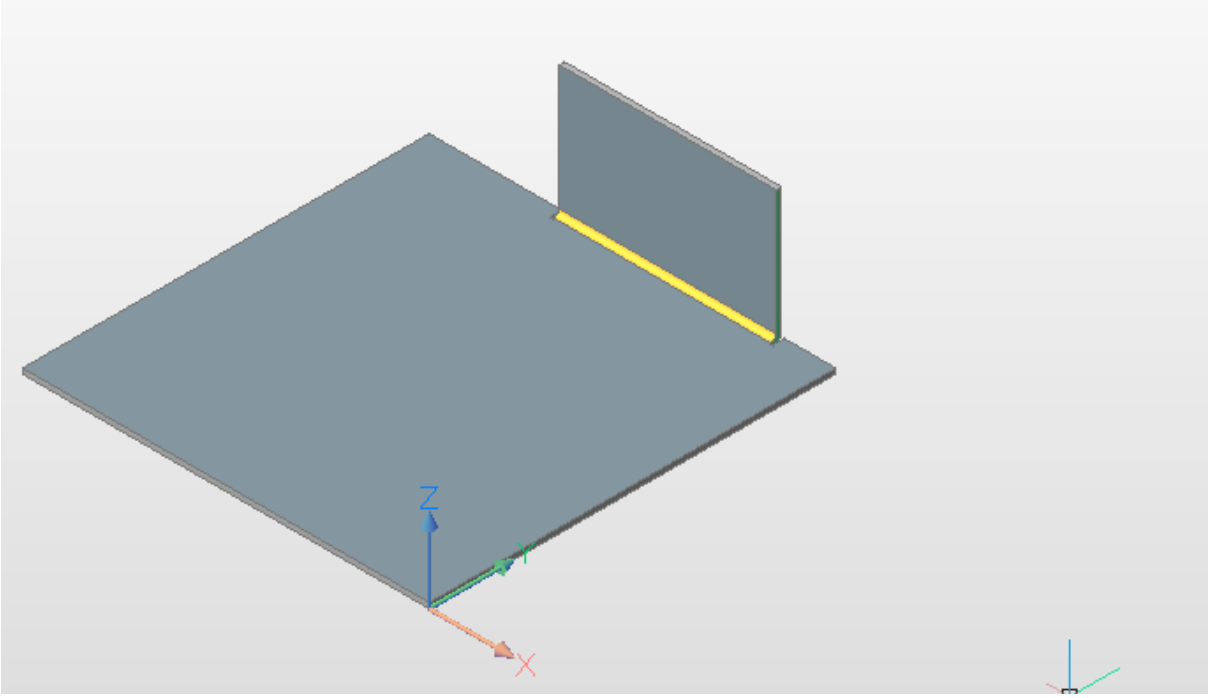




To create an edge flange of which the width is different from the width of the edge, select the *Width* option of SmFlangeEdge command and define offsets from both sides of the edge.



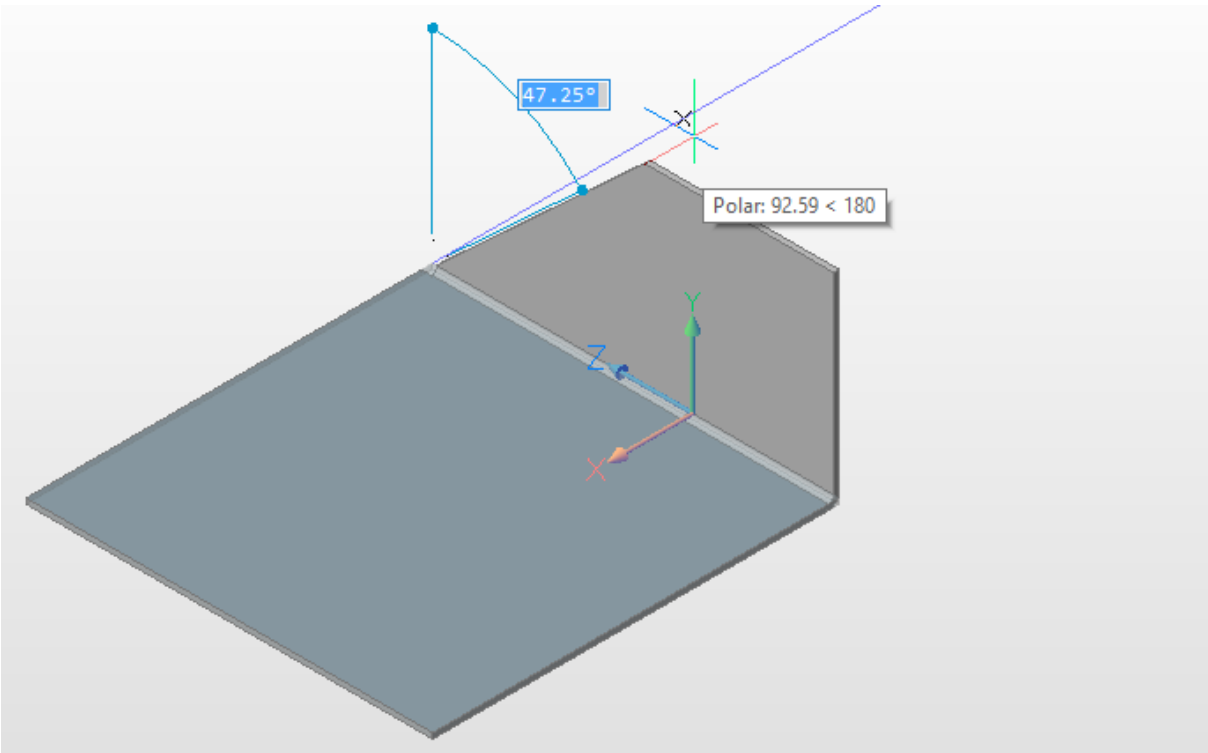


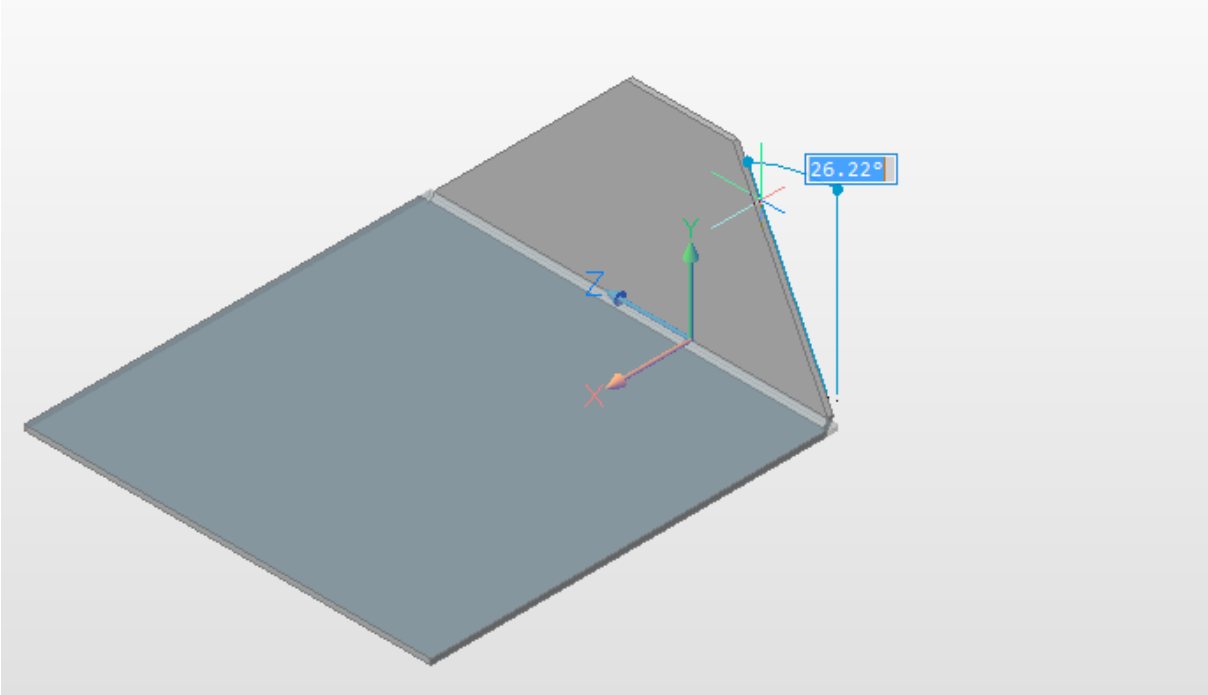


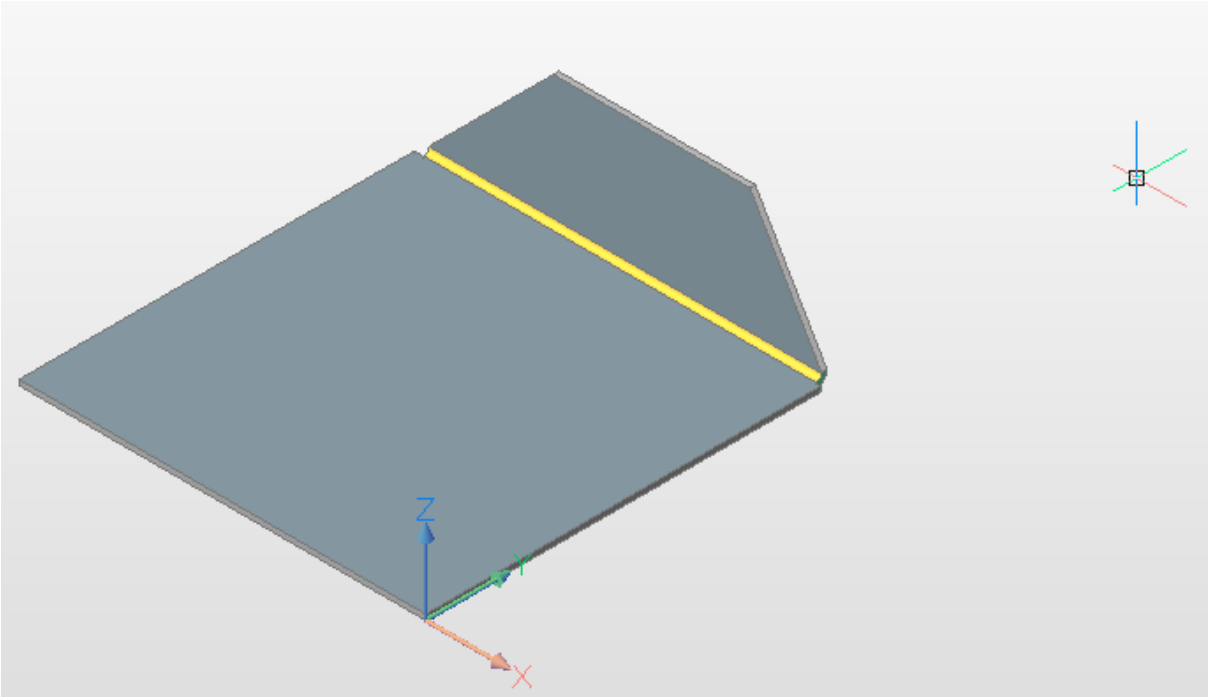
Note that when you create an edge flange of different width, the corresponding bend reliefs are created automatically.

Creating Tapered Edge Flanges

You can create trapezoidal edge flanges using *Taper angle* option of SmFlangeEdge command:







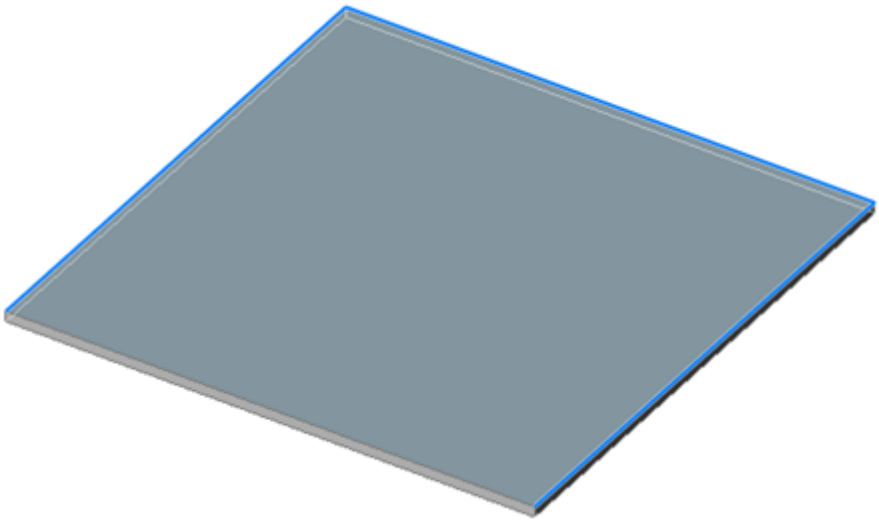
Creating Multiple Flanges

You can create several flanges simultaneously.

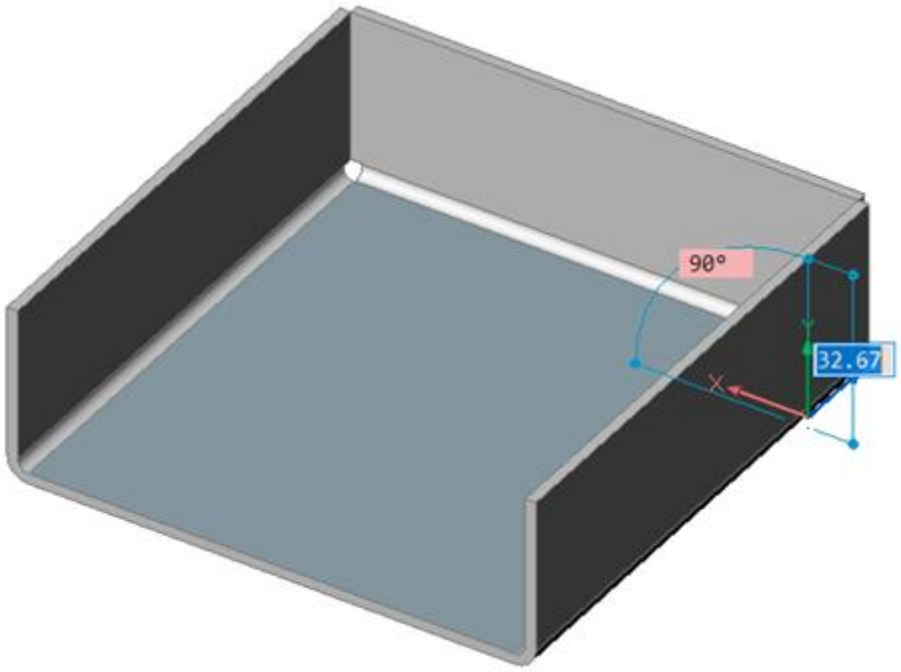
Do one of the following:

Select multiple edges, then choose *Create Flange Edge* ( ) in the *Sheet Metal* command group in the Quad.

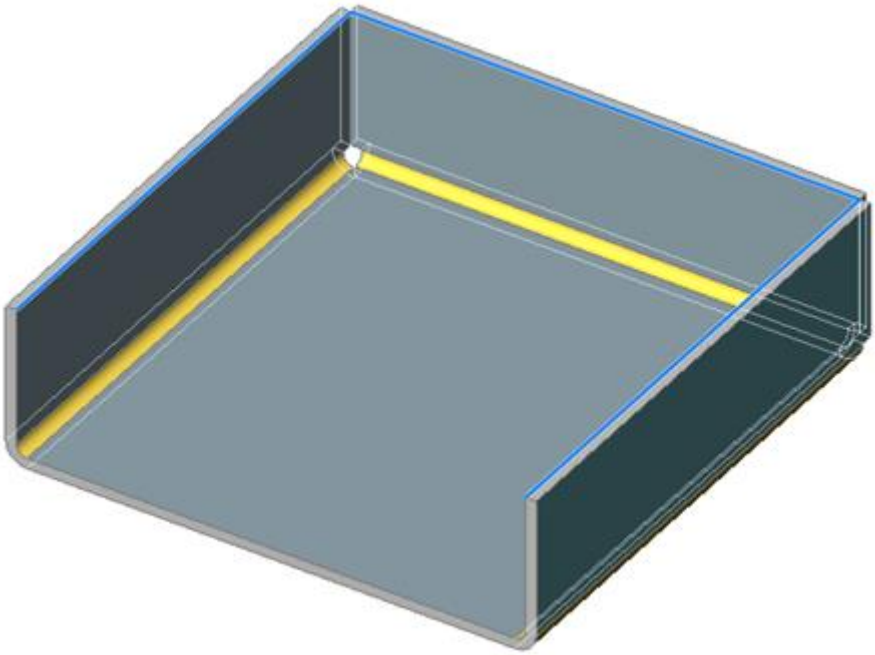
Launch the SmFlangeEdge command and select multiple edges.



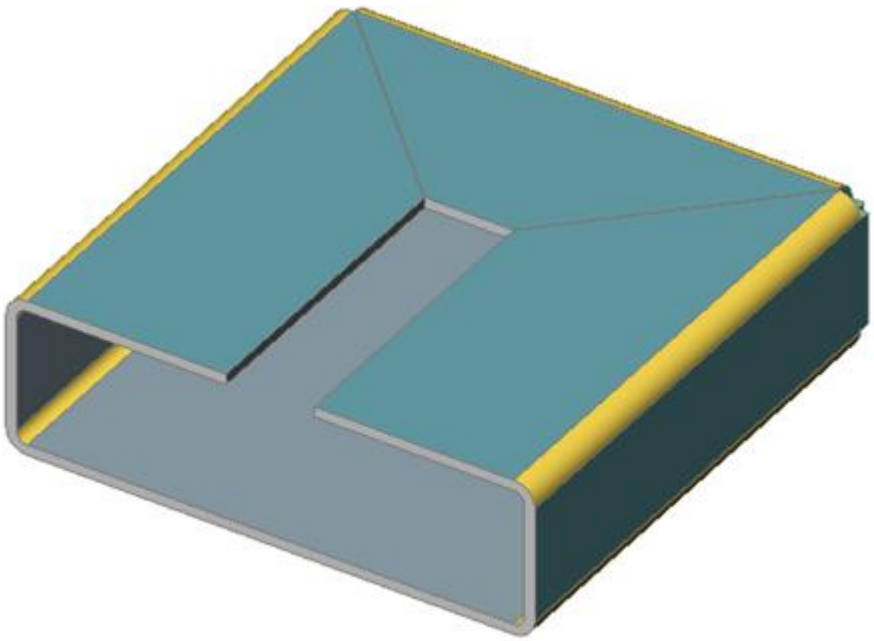
Create the edges dynamically.
Define the angle and the flange height using the dynamic entry fields. Hit the TAB-key to switch between the *Angle* and the *Distance* field.



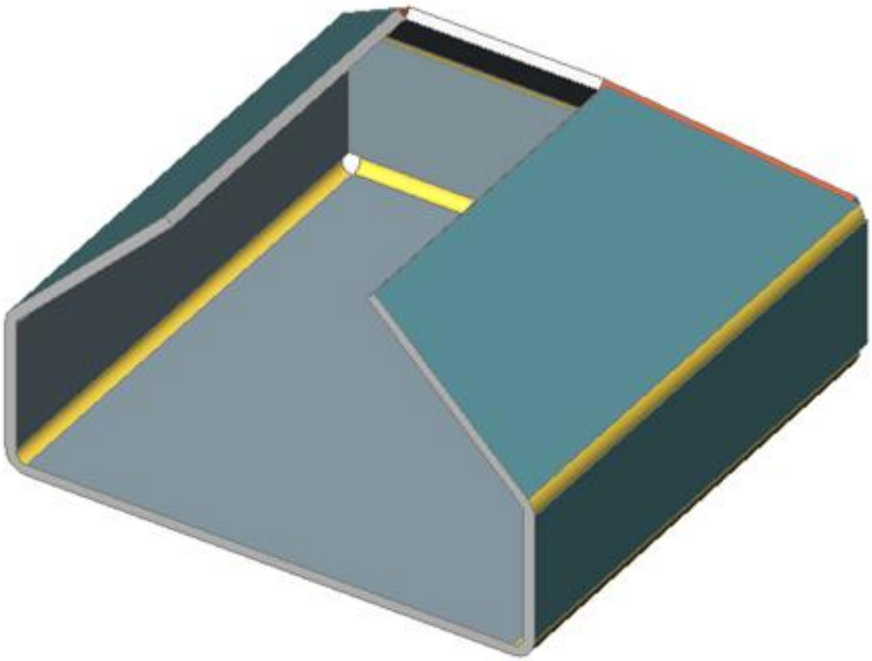
Select the inside edges of the newly created flanges:



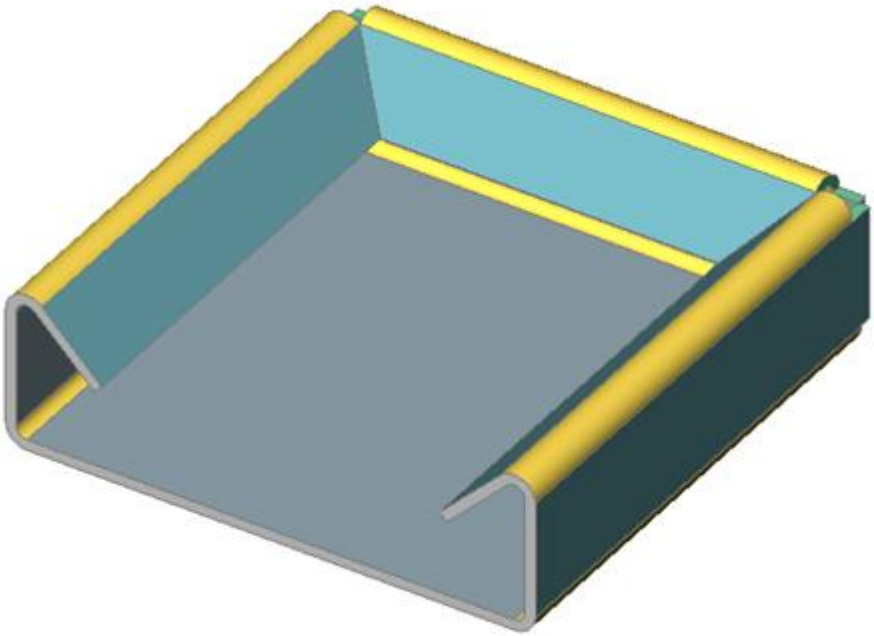
Create the flanges. The command creates self-intersecting of the flanges and depending on the bend angle creates junction or miter features.
At 90 degrees miters are created:



For other angles junctions are created:



In all cases material conflicts are avoided:



To Bend a Sheet Metal Body

The SmFlangeBend command allows to bend a sheet metal body using a line. You can either draw the line first, then start the command or us the *New Line* option of the command.

To bend multiple flanges simultaneously

Do one of the following:

Click the *Bend* tool button ( ) on the *Sheet Metal* toolbar.

Choose *Bend > Flange Bend* in the *Sheet Metal* menu.

Choose *Create Bend* ( ) in the *Modify* panel of the *Sheet Metal* tab in the ribbon.

Hover over the flange you want to bend and choose *Bend Flange* in the *Sheet Metal* command group in the Quad.

You are prompted: Select a flange face:

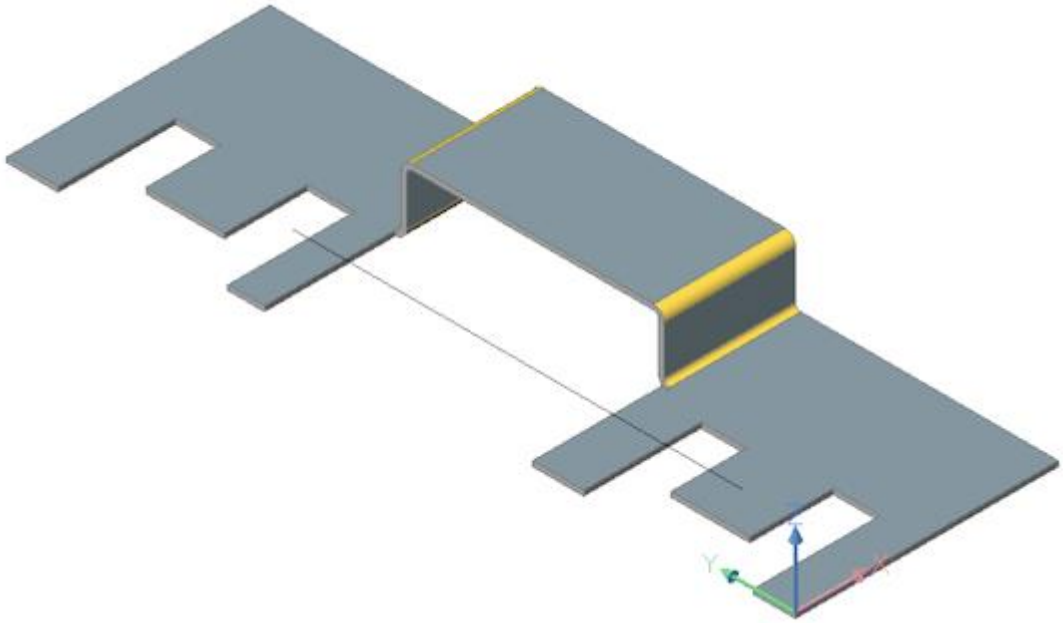
Select the flange you wan to bend.
You are prompted: Select line or edge to bend the solid or draw a [New line] <New line>:

Do one of the following:

Select a line that overlaps the flanges to be included in the bend.

Choose the *New Line* option.
You are prompted to pick two points to define the line.
Draw a line on top of the flanges to be included in the bend.

You are prompted: Pick a point on the side that will be moved:



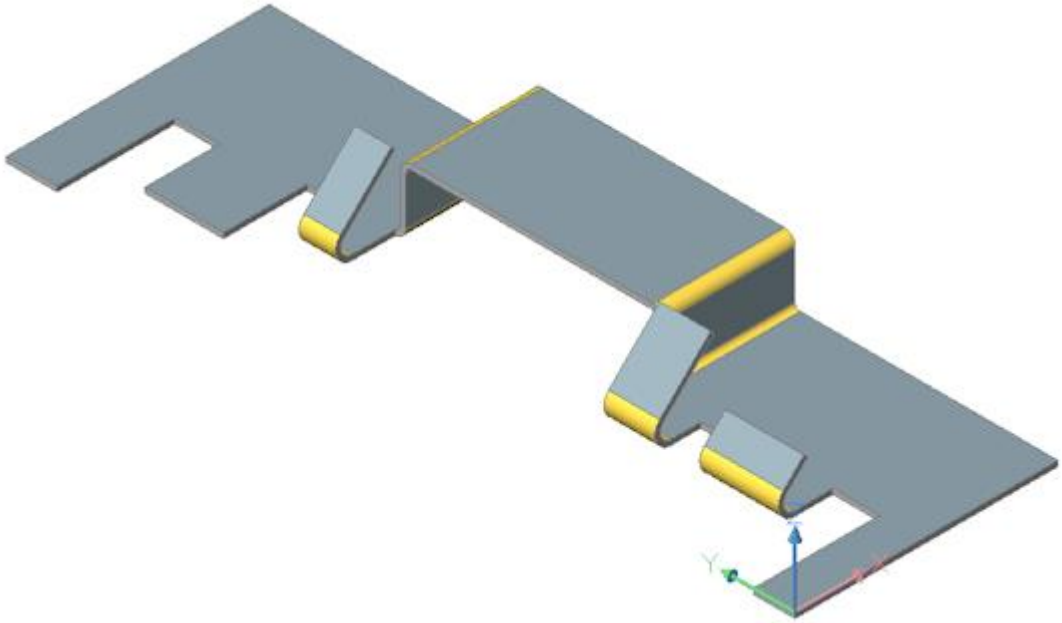
Pick a point at the side of the line you want to move.
The selected part is bent dynamically.
You are prompted: Enter bend angle or set [Radius/Switch side]:

Do one of the following:

Type a value to define the bend angle, using the default bend radius..

Select the Radius option.

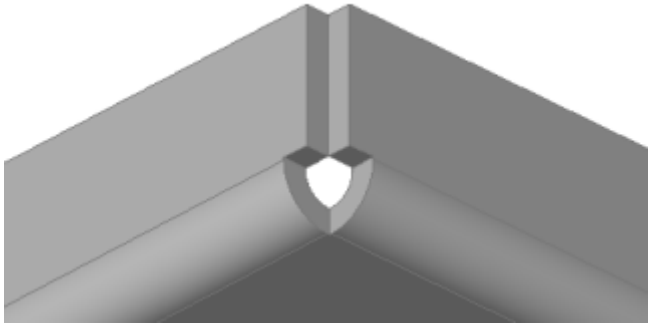
You are prompted: Enter bend radius [Back] <Back>:
Type a value to override the default bend radius and press Enter.
You are promted: Enter bend angle or set [Radius/Switch side]:
Type a value to define the bend angle.



Corner Relief and Junctions

When you pull an edge of a flange that is adjacent to a bend edge to create a new flange with SmFlangeEdge command, a corner relief is created automatically in the corner where three flanges meet together:

This operation also creates a junction between two flanges, which are not connected with a bend.



To Rotate a Flange

You can rotate a flange with DmRotate command ( ) , however, you will usually obtain a better result with SmFlangeRotate ( ). This command automatically selects the rotation axis to respect the design intent of your sheet metal part.

Do one of the following:

Click the *Rotate Flange* tool button ( ) on the *Sheet Metal* toolbar.

Choose *Rotate Flange* in the *Sheet Metal* menu.

Type *smflangerotate* in the command bar, then press Enter.

You are prompted: Select a flange face to rotate:

Flange faces highlight under the cursor.

Click to select a flange face.
The flange rotates dynamically.
A dynamic dimension shows the current angle with respect to the base flange.

Do one of the following:

Select a point.

Type a value in the dynamic dimension field.

Press the TAB key, then specify the absolute rotation angle in the dynamic dimension field.

NOTE If you apply DmRotate command to a flange, which is connected with other flanges via junction features, these features are automatically dissolved before the rotation.

Creating Lofted Bends

Launch the SmLoft command.
Do one of the following:

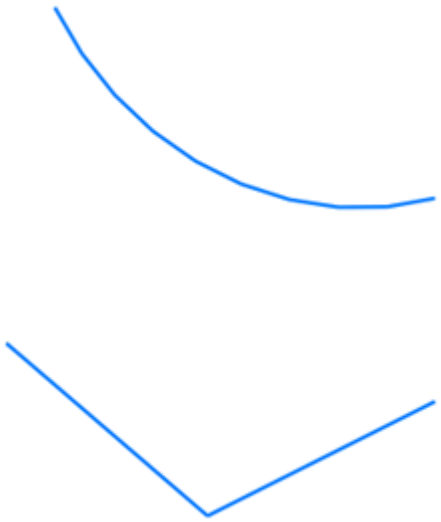
Click the *Create Lofted Bend* tool button ( ) on the *Sheet Metal* toolbar.

Click the *Create Lofted Bend* tool button ( ) on the *Sheet Metal* ribbon bar.

Choose *Create Sheet Metal Loft* in the *Sheet Metal* menu.

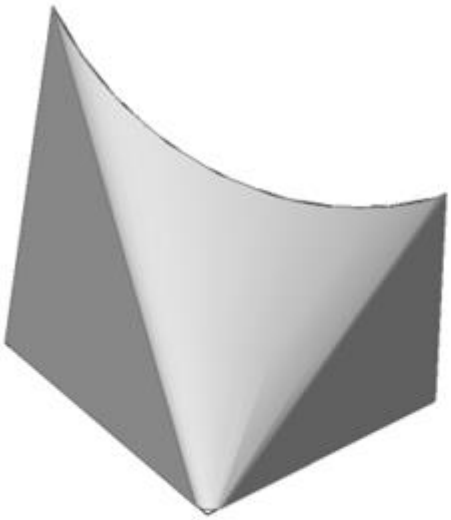
You are prompted: Select two cross sections:

Select two profile entities.

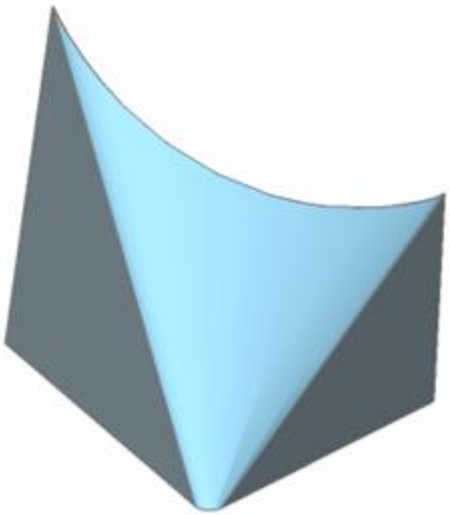


You are prompted: Set fillets Radius/Thickness/Single bend/<Flanges with bends>

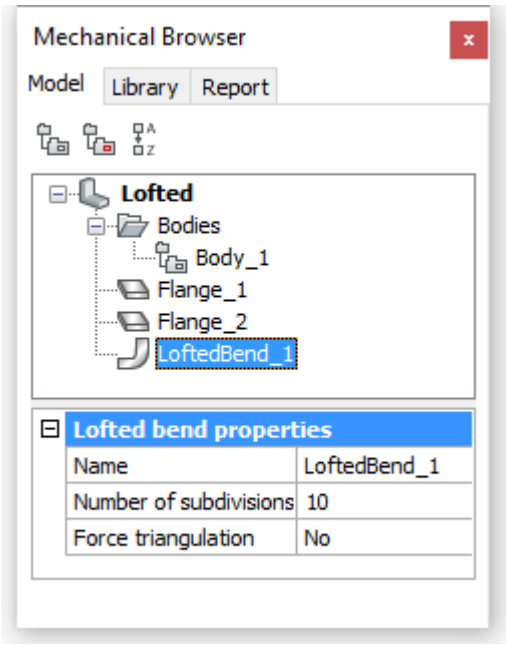
Press Enter to accept the default settings.
A preview of the sheet metal part is created.
You are prompted: Set fillets Radius/Thickness/Single bend/Flanges with bends or make thicken Both sides/Inside/<Outside>:



Hit the Ctrl-key to choose the position of the sheet metal part thickness with respect to the source profiles: *Left*, *Center* or *Right*.
If TIPS = ON the tips widget displays.
Press Enter to create the sheet metal part.



If FEATURECOLORS = ON, you can distinguish between the two flanges ( ) and the lofted bend ( ).
The properties of the lofted bend display in the Mechanical Browser:



Optionally select the lofted bend feature to edit its properties:

Number of subdivisions: the number of flat regions in the unfolded representation of a lofted bend feature (the greater the number, the more bend lines will be added to the unfolded view for the lofted bend).

Force triangulation: lots of lofted bends can be unfolded using an intelligent algorithm of quad tessellation; however sometimes it might produce non-optimal results. In such cases set this property to *Yes*.

Converting Geometry to Sheet Metal

You can create a sheet metal part from any existing 3D solid geometry, created in BricsCAD or imported from another CAD system. However, the geometry might be incorrect as a sheet metal model:missing bends and reliefs or thickness faces which are not orthogonal to the flange faces. BricsCAD Sheet Metal contains a dedicated toolset to convert such geometry to a valid sheet metal part and add all missing sheet metal features.

Launch the SmConvert command.
Do one of the following:

Click the *Convert to Sheet Metal* tool button ( ) on the *Sheet Metal* toolbar.

Click the *Convert to Sheet Metal* tool button ( ) on the *Sheet Metal* ribbon bar.

Choose *Convert to Sheet Metal* in the *Sheet Metal* menu.

You are prompted: Select 3D solids/<Entire model>:

Select the 3D solid to convert.
You are prompted: Select 3D solids/<Entire model>:

Press Enter to convert the 3D solid to sheet metal part.
The properties of the sheet metal part display in the Mechanical Browser.

NOTE Converted bodies can be unfolded or exported to .dxf or .osm file formats for processing by a CAM system.

In some cases it might be necessary to fix converted bodies before unfolding/exporting:

Create missing reliefs: SmReliefCreate command.

Convert hard edges to bends or junctions: SmBendCreate and SmJunctionCreate commands.

Split flanges: SmFlangeSplit command.

Repair sheet metal bodies: SmRepair command.

To Connect Flanges

The SmFlangeConnect command closes gaps between two arbitrarily oriented flanges.

Do one of the following:

Click the *Connect Flanges* tool button ( ) on the *Sheet Metal* toolbar.

Choose *Connect Flanges* in the *Sheet Metal* menu.

Type smflangeconnect in the command bar, then press Enter.

You are prompted: Select planar thickness faces of two flanges:

Flange faces highlight under the cursor.

Select the thickness face of the first flange.
You are prompted:
Entities/subentities in set: 1
Select planar thickness faces of two flanges:

Select the thickness face of the second flange.
The selected thickness faces are connected.

To Create Missing Reliefs

Do one of the following:

Click the *Create Relief* tool button ( ) on the *Sheet Metal* ribbon tab

Choose *Create Relief* in the *Sheet Metal* menu

Type *smreliefcreate* in the command bar

You are prompted: Select a hard edge or bend face, flange face, 3D solid / <Entire model>:

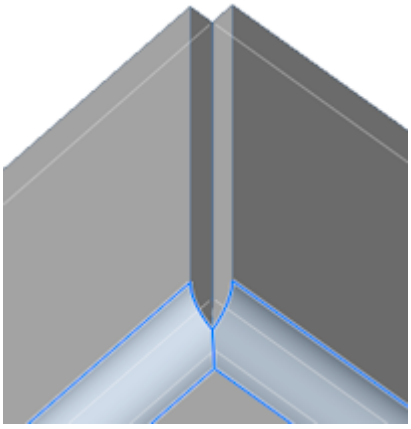
Select an item (see the note below)
Enter relief size by bend radius ratio <1.0>:

Do one of the following:

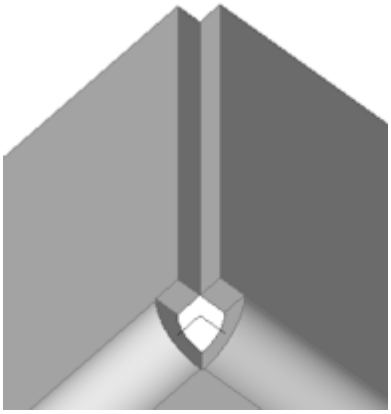
Press Enter or right click to accept the 1.0 bend radius ratio.

Type a value at the command prompt, then press Enter.

The relief is created.



Selected bend faces



Created reliefs

NOTE The selected items can be a 3D solid with sheet metal features, a face or a hard edge of the solid.

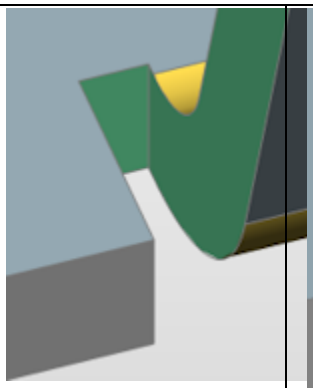
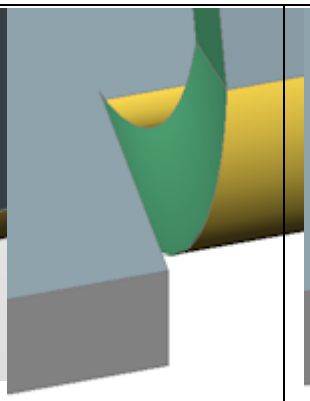
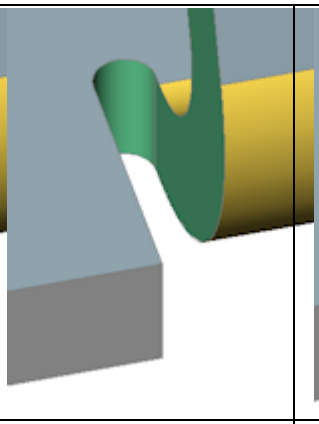
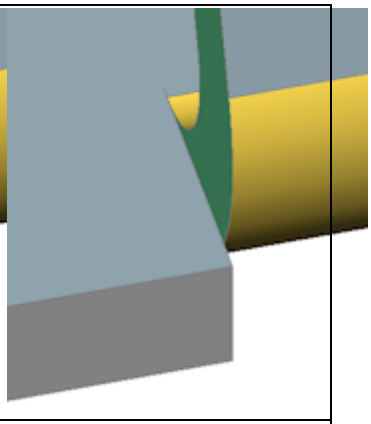
To create a relief locally, you can select one or two hard edges, one or two bends or a bend and a hard edge.

If a 3D solid is selected the command recognizes the places where corner or bend reliefs are needed and creates them automatically.

If two hard edges or two bends or a bend and a hard edge at a corner are selected, the command tries to create a corner relief.

Switching the Bend Relief Type

The bend relief type can be switched between four types:

			
Rectangular	**Smooth**	**Round**	**Rip**

NOTES

Rip relief has a very small width set to 0.001 * thickness. If you need to remove the relief completely, change its width to 0. This operation removes both the bend relief geometry and the bend relief feature.

Smooth, *Round* and *Rip* bend relief types can be applied only if the current bend relief type is *Rectangular*.

Using the ribbon, menu or toolbar

To choose the desired relief type, do one of the following:

Click the *Relief* drop down button in *Sheet Metal / Modify* panel on the ribbon and select *Switch to Rectangular*, *Switch to Smooth, Switch to Round* or *Switch to Rip*.

Click a tool button on the *Relief* toolbar: *Switch to Rectangular Relief* ( ), *Switch to Smoot Relief* ( ), *Switch to Round Relief* ( ) or *Switch to Rip Relief* ( ).

Choose the relief type in the *Sheet Metal / Relief* menu.

You are prompted: Select face(s), 3d solid(s) to switch relief(s) to Rectangular [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/Entire model] <Entire model>:

Do one of the following:

Press Enter to apply the selected relief type to the entire model.

Click a neighboring flange or bend or a face of the bend relief.
The bend relief is converted.

You are prompted: Select face(s), 3d solid(s) to switch relief(s) to Rectangular [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/Entire model] <Entire model>:

Do one of the following:

Click a neighboring flange or a face of another bend relief.

Press Enter to apply the selected bend relief to the entire model.

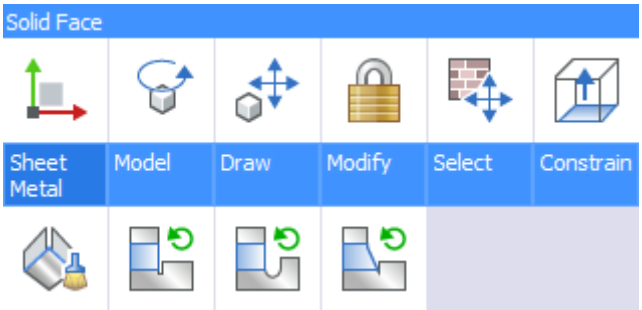
Press the Esc-key to stop.

Using the Quad

Hover over neighboring flange or bend or a face of the bend relief.

If the *Select Faces* option of the SELECTIONMODES system variable is selected, the flange or face highlights; otherwise hold the Ctrl-key down when hovering until the face highlights.

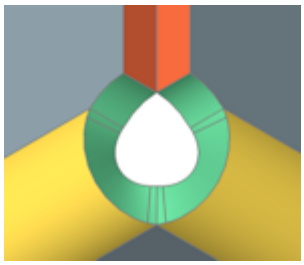
Select the desired relief type in *Sheet Metal* command group in the Quad.



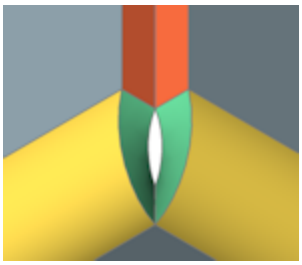
Switching the Corner Relief Type

The corner relief type can be switched between three types:

Rectangular



Circular



V-type

Using the ribbon, menu or toolbar:

To choose the desired relief type, do one of the following:

Click the *Relief* drop down button in *Sheet Metal / Modify* panel on the ribbon and select *Switch to Circular*, *Switch to Rectangular* or *Switch to V-Type*.

Click a tool button on the *Relief* toolbar: *Switch to Circular Relief* ( ), *Switch to Rectangular Relief* ( ) or *Switch to V-Type Relief* ( ).

Choose the relief type in the *Sheet Metal / Relief* menu.

You are prompted: Select face(s), 3d solid(s) to switch relief(s) to circular [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/SEttings/Entire model] <Entire model>:

Do one of the following:

Press Enter to apply the selected relief type to the entire model.

Click a neighboring flange or bend or a face of the corner relief.
The corner relief is converted.

You are prompted: Select face(s), 3d solid(s) to switch relief(s) to circular [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/SEttings/Entire model] <Entire model>:

Do one of the following:

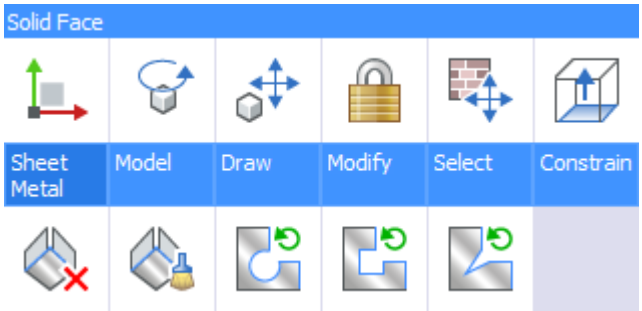
Click a neighboring flange or a face of another corner relief.

Press Enter to apply the selected corner relief to the entire model.

Press the Esc-key to stop.

Using the Quad:

Make sure the *Select Faces* option of the SELECTIONMODES system variable is selected.
Hover over neighboring flange or bend or a face of the corner relief.
Select the desired relief type in *Sheet Metal* command group in the Quad.



Converting hard edges to bends or junctions

If adjacent flanges of a sheet metal part are not connected by a bend or junction, the part cannot be unfolded or exported to a CAM system. To fix this, each hard edge between adjacent flanges must be converted to a bend or a junction.

Converting Hard Edges to Bends

Do one of the following:

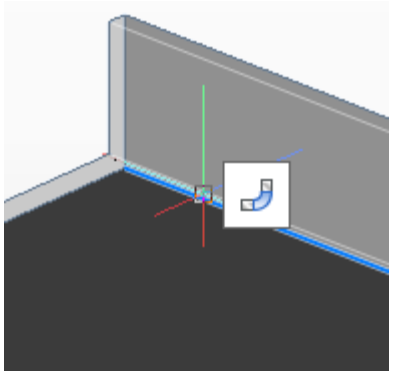
- Hover the mouse cursor over a hard edge and click the *Create Bend* ( ) tool in the Quad menu
- Click the *Create Bend* ( ) tool buttons in the *Sheet Metal* ribbon tab
- Choose *Create Bend* in the *Sheet Metal* menu
- Type *smbendcreate* or the command bar.

You are prompted: Select flange faces or hard edges:

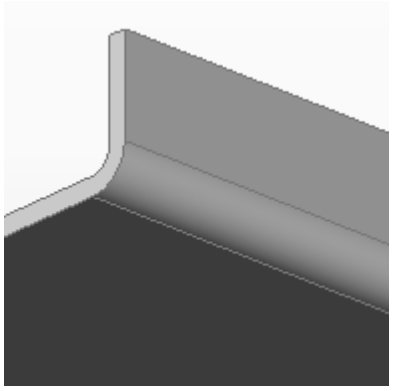
Select a flange face or a hard edge.
You are prompted:
Entities in set: 1
Select flange faces or hard edges:

Do one of the following:

- Select more faces or hard edges.
- Right click or press Enter to create the bends or junctions on the selected items.



Selected hard edge



Created bend

Converting Hard Edges to Junctions

Do one of the following:

Hover the mouse cursor over a hard edge and click the *Create Junction* ( ) tool in the Quad menu

Click the *Create Junction* ( ) tool buttons in the *Sheet Metal* ribbon tab

Choose *Create Junction* in the *Sheet Metal* menu

Type *smjunctioncreate* in the command bar.

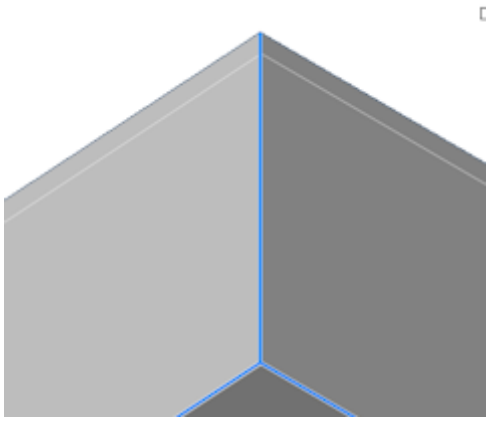
You are prompted: Select flange faces or hard edges:

Select a flange face or a hard edge.
You are prompted:
Entities in set: 1
Select flange faces or hard edges:

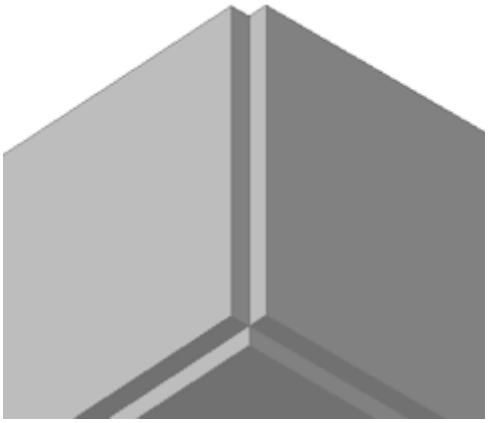
Do one of the following:

Select more faces or hard edges.

Right click or press Enter to create the bends or junctions on the selected items.



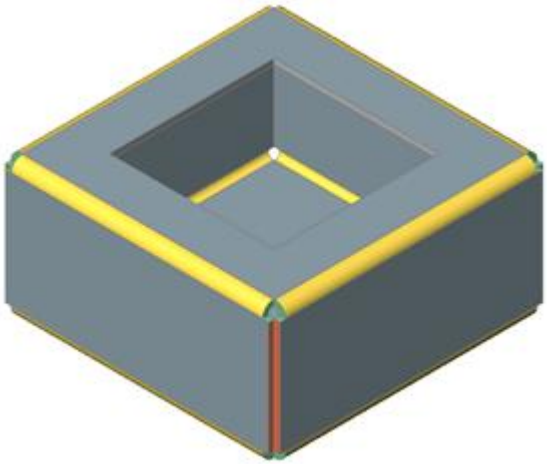
Selected hard edges



Created junctions

To Split a Flange

In some cases a flange is better split to minimize the material consumption, e.g. when a flange has a large hole like on the image below:



Do one of the following:

Click the *Split Flange* tool ( ) on the *Sheet Metal* ribbon tab

Choose *Split Flange* in the *Sheet Metal* menu

Type *smflangesplit* in the command bar

You are prompted: Select a flange face:

Select a flange face.
You are prompted: Select lines, edges to split the flange or [use SMart split/draw a New line/Settings] <draw a New line>:

If no line exists yet on the flange face, right click.
You are prompted: Start point of the line:

Specify the start point of the line.
You are prompted: End point of the line:

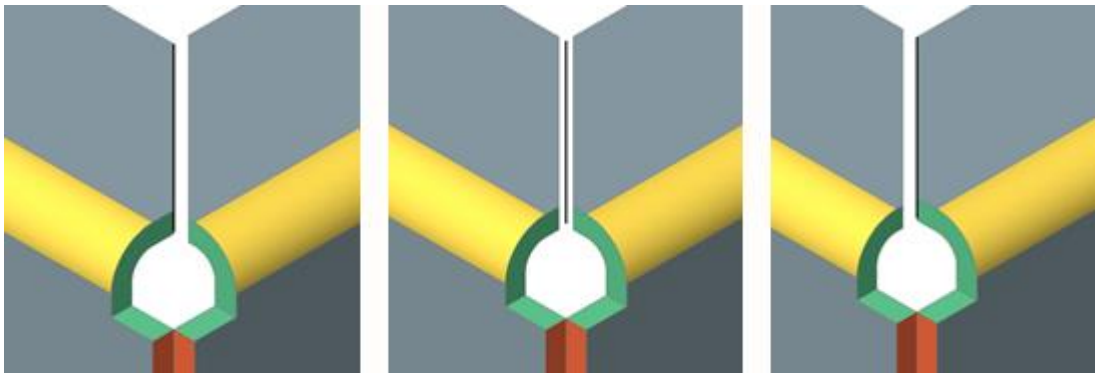
Specify the endpoint of the line:
You are prompted: Make split Center/Left/Right/<Accept model>:

To adjust the position of cut in relation to the line, do one of the following:

Hit the Ctrl-key to toggle the Center/Left/Right options.
If TIPS is ON the currently selected option is indicated in the Tips Widget.

Select *Center*, *Left* or *Right* in the Prompt Menu.

Type an option in the command bar, then press Enter.



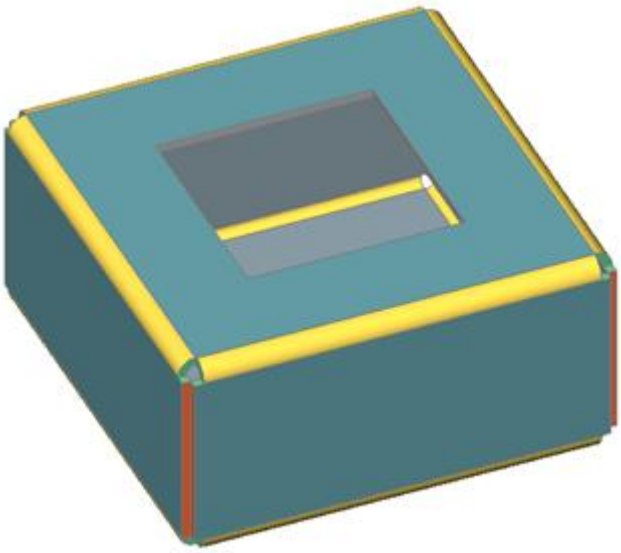
The flange is cut by the line.

Smart Tools to Split a Flange

The scenario below illustrates the other capabilities of the SmFlangeSplit command:

Smart split by selecting a single vertex in the corner.

Split of the bend and conversion of its part adjacent to the corner into the junction.



Do one of the following:

Click the *Use Smart Split Flange* tool ( ) on the *Sheet Metal* ribbon tab

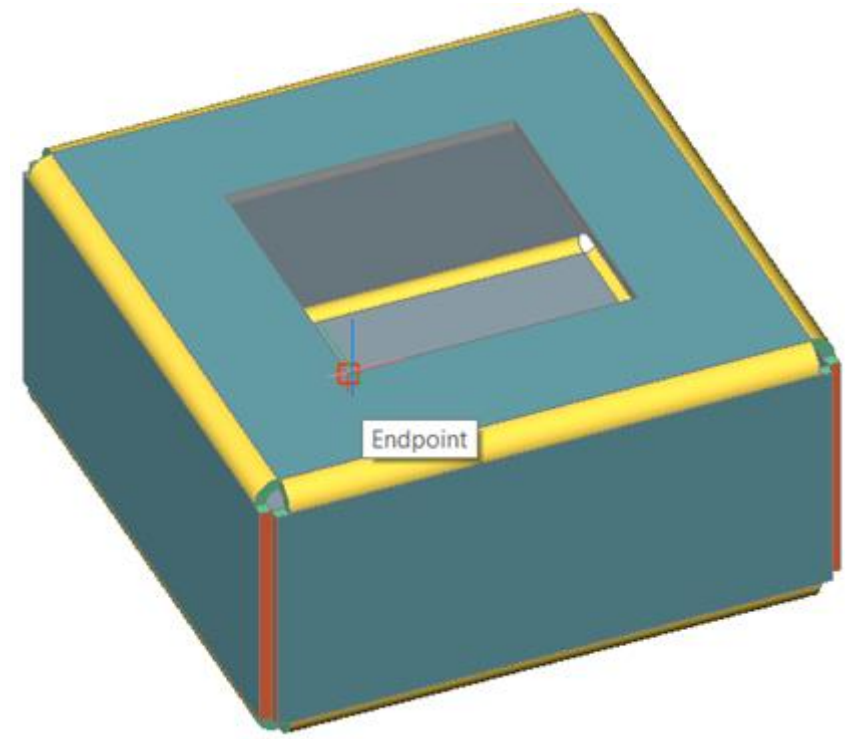
Choose *Use Smart Split Flange* in the *Sheet Metal* menu

Type *smflangesplit* in the command bar

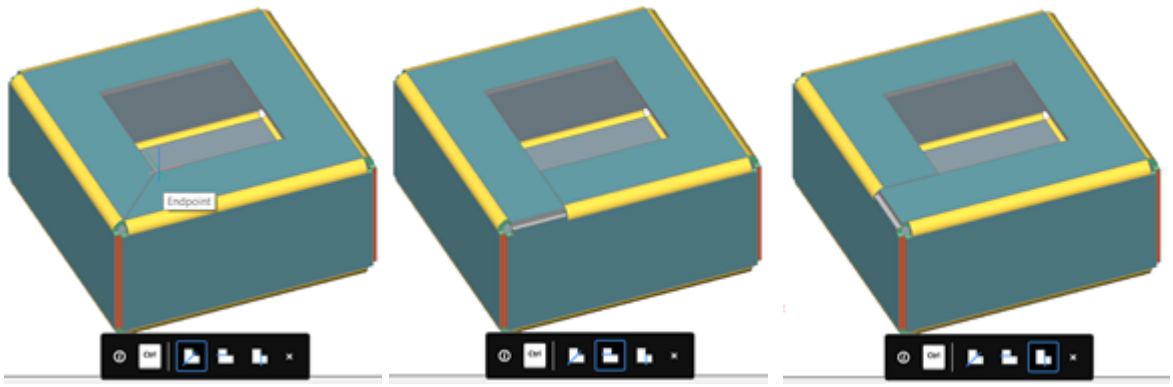
You are prompted: Select a flange face:

Select a flange face.
You are prompted: Select lines, edges to split the flange or [use SMart split/draw a New line/Settings] <draw a New line>:

Select use the *Use Smart split* option and select a corner vertex:



You are prompted: Make split [Center/Left/Right/Accept] <Accept>:
Possible variants for selected vertex are:



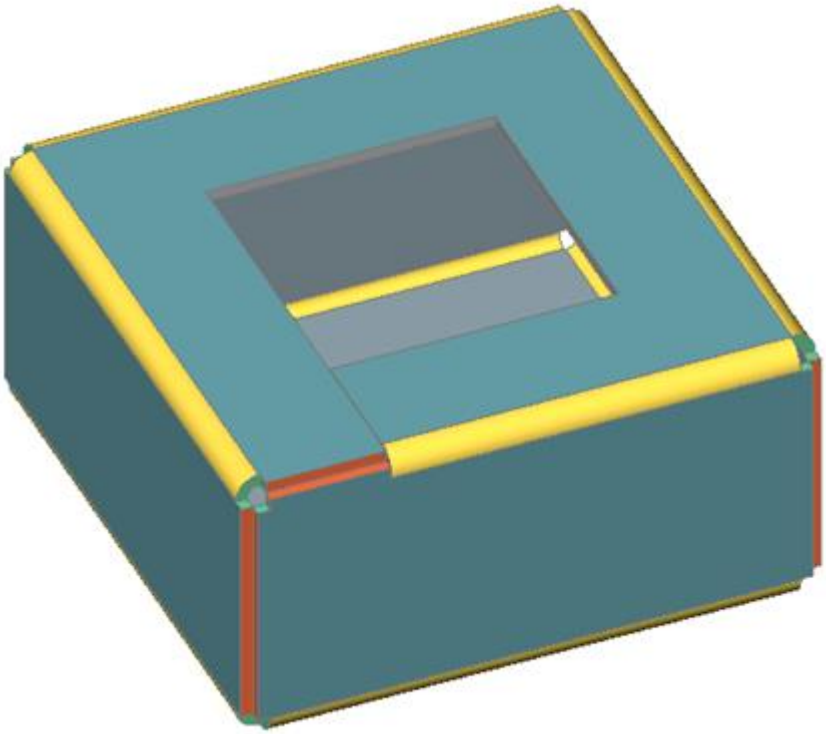
Note: The conversion of part of the bend into the junction is unrelated to the *Smart Split* mode.

To adjust the position of cut in relation to the line, do one of the following:

Hit the Ctrl-key to toggle the Center/Left/Right options.
If TIPS is ON the currently selected option is indicated in the Tips Widget.

Select *Center*, *Left* or *Right* in the Prompt Menu.

Type an option in the command bar, then press Enter.
The flange is cut by the line, a bend is partially converted into the junction:

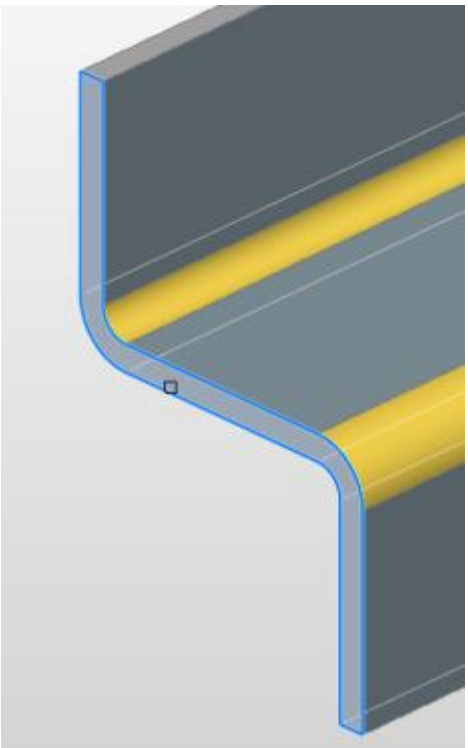


To Split a Thickness Face

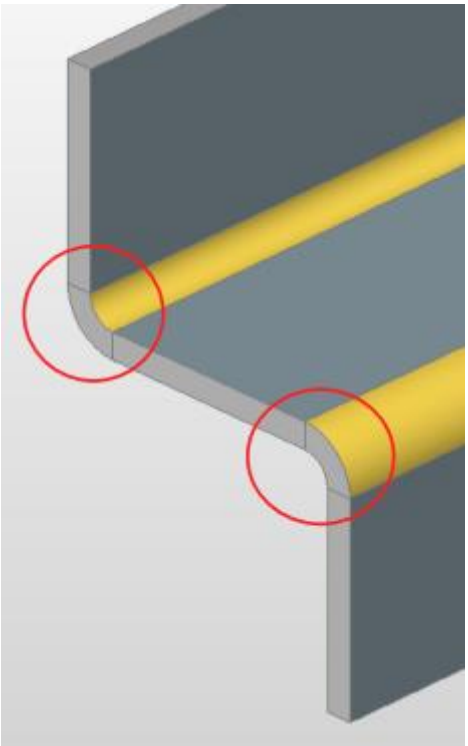
In order to extend a flange which shares the same thickness face with other flanges, the thickness face needs to be split.

Launch the SmImprint command ( ).
Prompts you: Select a thickness face:

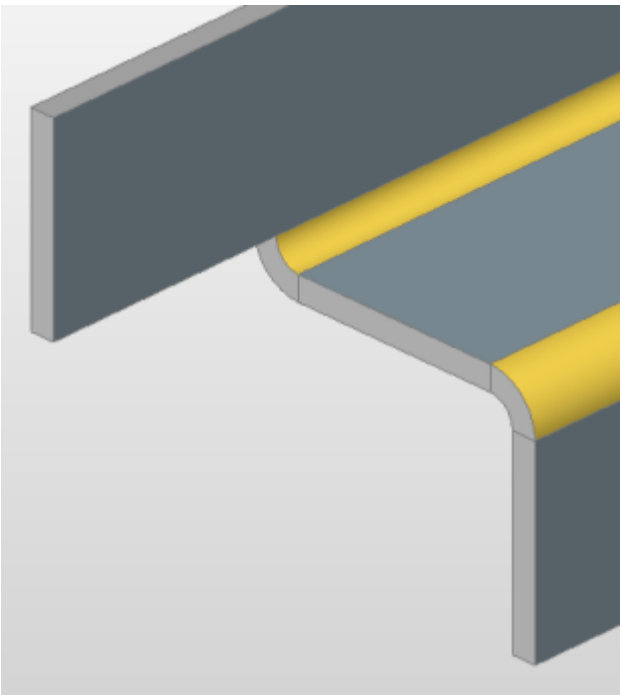
Select the shared thickness face.



Right click or press Enter to split the face.



The flanges can now be extended separately:



Execute the DmSimplify command to remove the unneeded splits, thus unifying the coplanar thickness faces.

To Repair a Sheet Metal Part

A Sheet metal part may have inaccuracies in geometry and topology or sheet metal design issues, which is often the case for imported parts designed in geometrical kernels different from ACIS.

Examples of such problems are:

thickness faces, which are not perpendicular to the flange/bend faces;

lofted bends are not-tangential to the adjacent flanges/bends;

wrong bends;

general ACIS issues.

Such problems can prevent editing operations from being executed successfully.

The recommended workflow to fix such problems is:

Execute the DMAUDIT command to fix general modeling issues.

Execute the SMREPAIR command.

To Repair a Sheet Metal Body

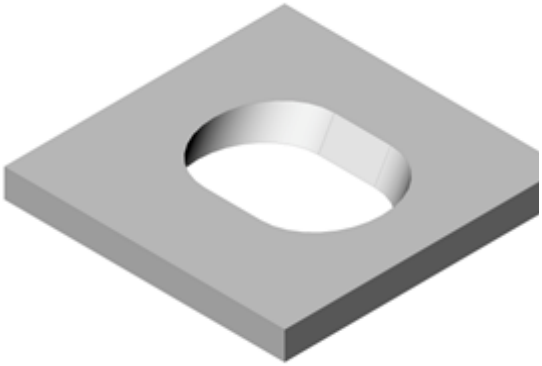
If a sheet metal body has thickness faces, which are not perpendicular to the flange/bend faces, such a part cannot be correctly manufactured from a sheet of metal using traditional tools.

Do one of the following:

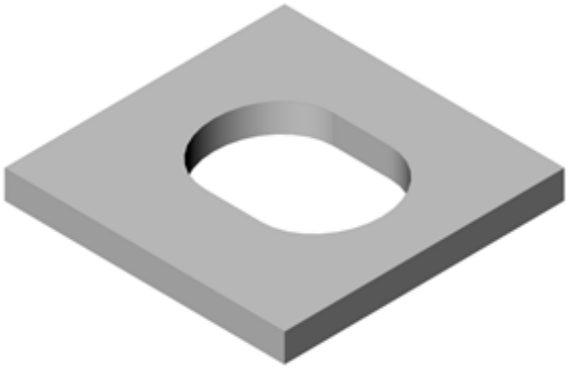
- Click the *Repair Sheet Metal* tool ( ) on the *Heal* panel of the *Sheet Metal* ribbon tab
- Click the *Repair Sheet Metal* tool ( ) on the *Sheet Metal* toolbar.
- Choose *Repair Sheet Metal* in the *Sheet Metal* menu.
- Type *smrepair* in the command bar.

You are prompted: Select a face to repair:

Select the face to be repaired.

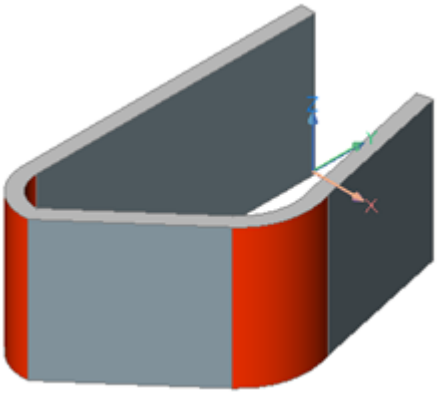


The same model before ...

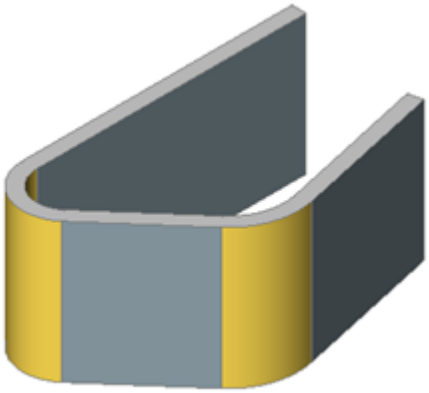


... and after applying a repair operation:

If model has wrong bends SmRepair transforms them to regular bends:

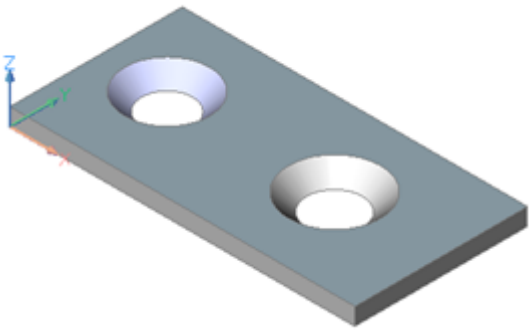


Wrong bends, which have a non-tangential connection with the flanges.

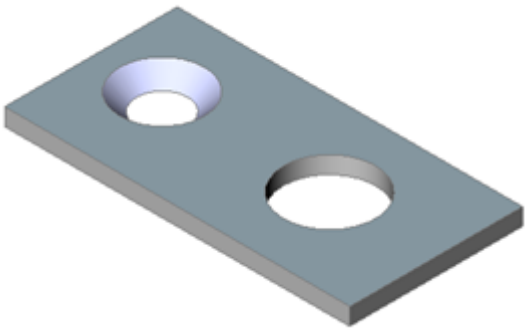


Correct bends after repairing.

Form features are translated to the new repaired model:



model before repairing



form feature has not changed while hole, which is not a form feature, is orthogonalized

To Repair a Sheet Metal Body with Lofted Bends

Do one of the following:

Click the *Repair Sheet Metal* tool ( ) on the *Heal* panel of the *Sheet Metal* ribbon tab

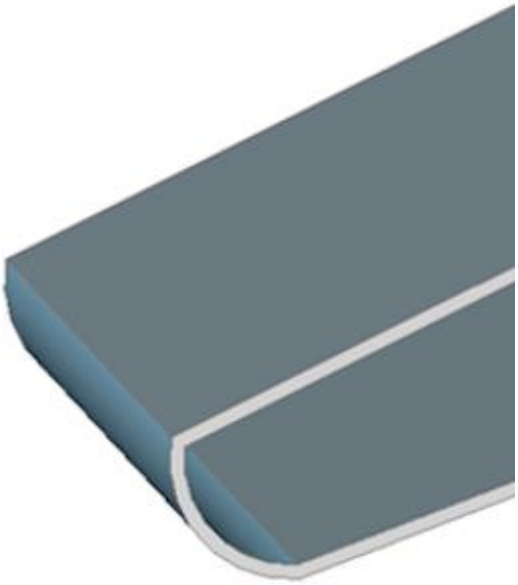
Click the *Repair Sheet Metal* tool ( ) on the *Sheet Metal* toolbar.

Choose *Repair Sheet Metal* in the *Sheet Metal* menu.

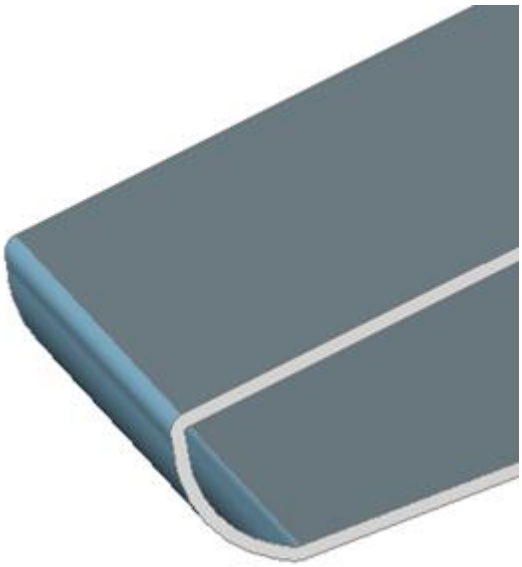
Type *smrepair* in the command bar.

The command bar reads: Select a face to repair.

Select a face of a flange, bend or lofted bend. The selected face generates the surface to be thickened.



Before: the lofted bend is not tangent to the flange. Unfolding is impossible.



After: notice the difference in the connection of the blue lofted bend with the top flange, which is smooth now. Unfolding is possible.

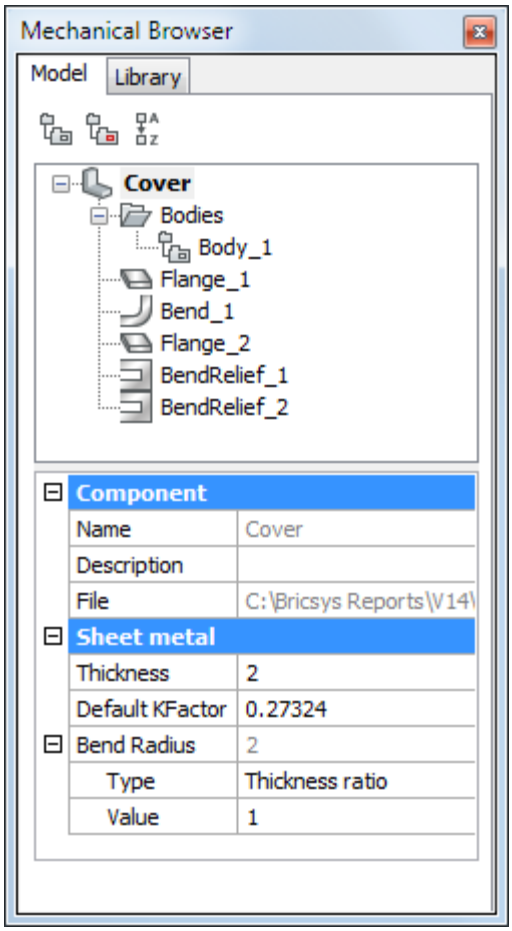
Option: The *Merge lofted bends* option in the SmRepair command allows to merge adjacent lofted bend into one.

Changing the Thickness

To change the thickness of a sheet metal part:

Select the root node in the *Mechanical Browser.*

*T*ype a value in the *Thickness* field.



To change the overall bend radius:

Select the root node in the *Mechanical Browser.*

Find the *Bend radius* group in the bottom part of the *Mechanical Browser*.

Do one of the following:

Set *Type* to *Absolute Value*, then type the radius value in the *Value* field.
The *Bend Radius* field changes accordingly.

Set *Type* to *Thickness ratio*, then type a value in the *Value* field.
The *Bend Radius* field is calculated as the product of the *Thickness* and the *Thickness ratio*.

To change the bend radius for a particular bend:

Select the bend node in the *Mechanical Browser.*
The selected bend highlights in the model.
The Bend Properties display in the bottom part of the *Mechanical Browser*.

Do one of the following:

Set the *Type* to *Global value.*
The current global bend radius is applied to the selected bend.

Set the *Type* to *Absolute value,* then type the radius value in the *Value* field.
The *Bend Radius* field changes accordingly.

Set the *Type* to *Thickness Ratio* and type a value in the value field.
The *Bend Radius* field is calculated as the product of the *Thickness* and the *Thickness ratio*.

Changing the Junction Gap

To change the overall junction gap:

Select the root node in the *Mechanical Browser.*

Find the *junction gap* group in the bottom part of the *Mechanical Browser*.

Do one of the following:

Set *Type* to *Absolute Value*, then type the gap value in the *Value* field.
The *junction gap* field changes accordingly.

Set *Type* to *Thickness ratio*, then type a value in the *Value* field.
The *junction gap* field is calculated as the product of the *Thickness* and the *Thickness ratio*.

To change the junction gap for a particular bend:

Select the bend node in the *Mechanical Browser.*
The selected bend highlights in the model.
The Bend Properties display in the bottom part of the *Mechanical Browser*.

Do one of the following:

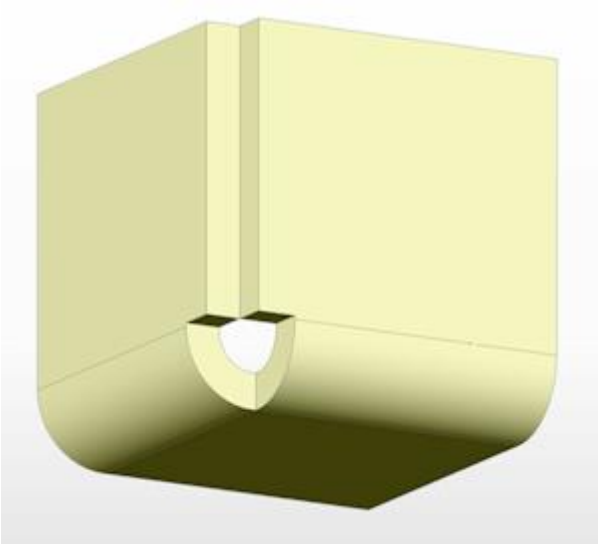
Set the *Type* to *Global value.*
The current global junction gap is applied to the selected bend.

Set the *Type* to *Absolute value,* then *t*ype the gap value in the *Value* field.
The *junction gap* field changes accordingly.

Set the *Type* to *Thickness Ratio* and type a value in the value field.
The *junction gap* field is calculated as the product of the *Thickness* and the *Thickness ratio*.

Changing the Junction Type

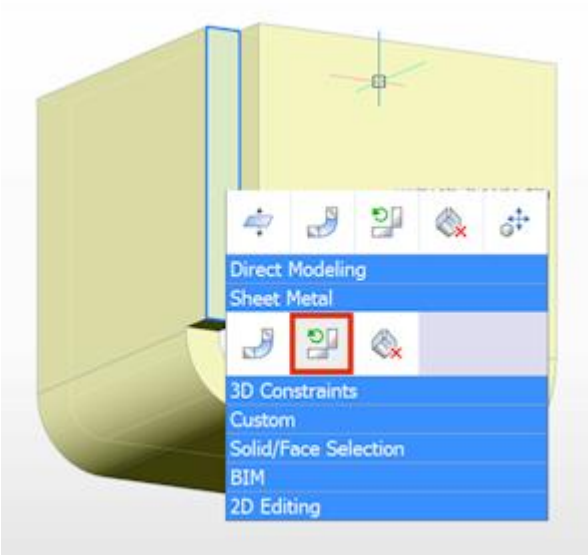
By default, BricsCAD creates symmetric junctions, as shown in the image below:



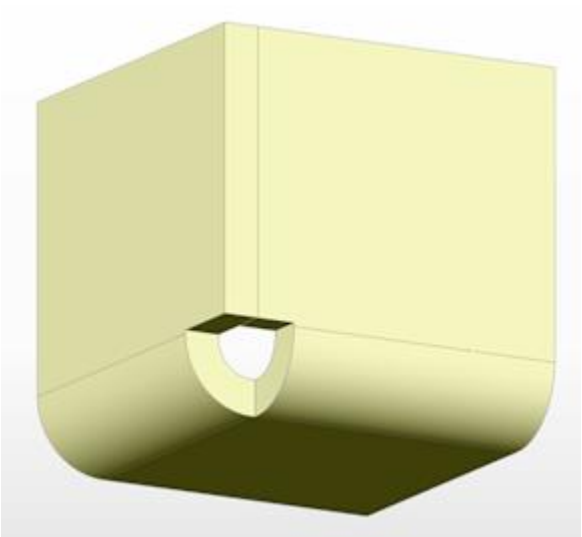
The SmJunctionSwitch command allows changing a symmetrical junction feature to one with overlapping faces.

To change the junction type using the Quad:

Hover the cursor over one the thickness faces of this junction,
Select *Switch Junction Type* in the Sheet Metal command group in the Quad.



The selected thickness face is aligned with the flange face.



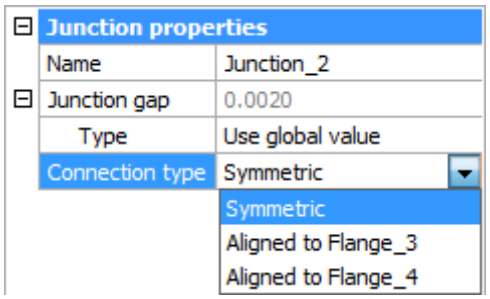
To change the junction type in the Mechanical Browser

Method 1:

Select the corresponding junction in the tree.
The junction properties display in the bottom part of the Mechanical Browser.

Select the *Connection type* property.

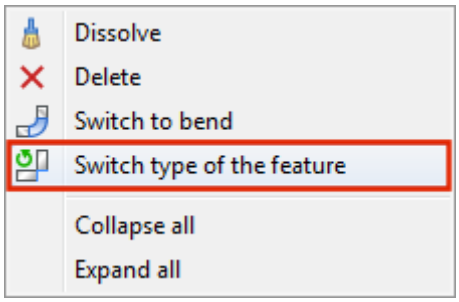
Choose the desired junction type.
The options are: *Symmetric*, *Aligned to Flange_X* or *Aligned to Flange_Y*.



The junction is updated accordingly.

Method 2:

Select the corresponding junction in the tree.
A context menu displays:



Right click and choose *Switch type of featur*e in the context menu.

NOTE Both symmetric and aligned junctions maintain the same gap value between connected flanges.

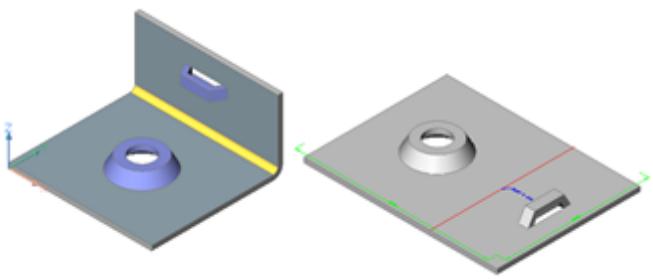
To Unfold the Sheet Metal Body

To unfold your sheet metal body, select a start flange face and run the SmUnfold command. BricsCAD will automatically create a solid body corresponding to a flat sheet of metal needed to manufacture your sheet metal part using bending techniques. This sheet is placed on XY-plane and is oriented along the coordinate axes similarly to the orientation of the initial body in 3D space. To change the orientation of the unfolding in OZ axis, run SmUnfold command again and select the opposite face of the initial body.

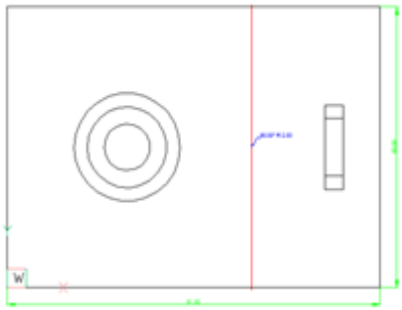
By default, form features are unfolded according to the value of the SMFORMFEATUREUNFOLDMODE system variable.

The options are:

Keep (0): allows to keep form features geometry on the unfolded representation

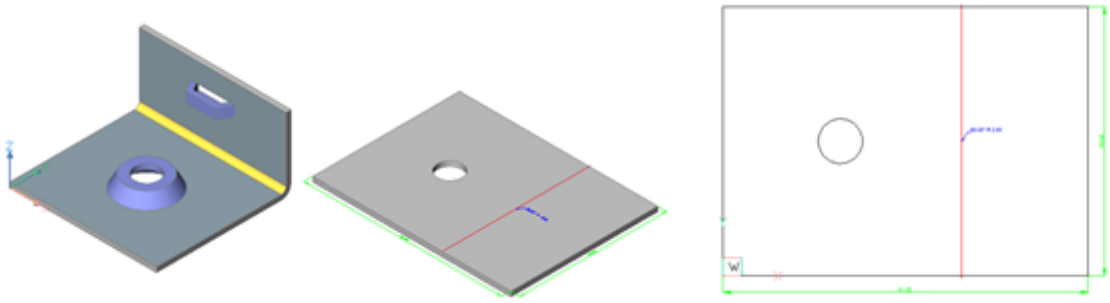


3D model and unfolded body



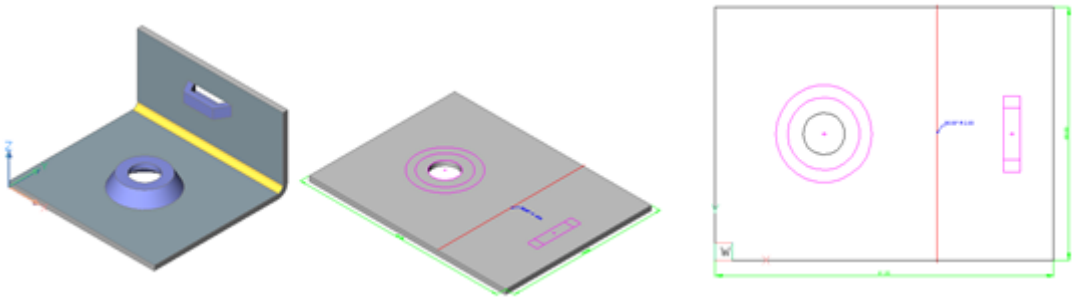
2D

Remove (1): allows to remove form features geometry from the unfolded representation



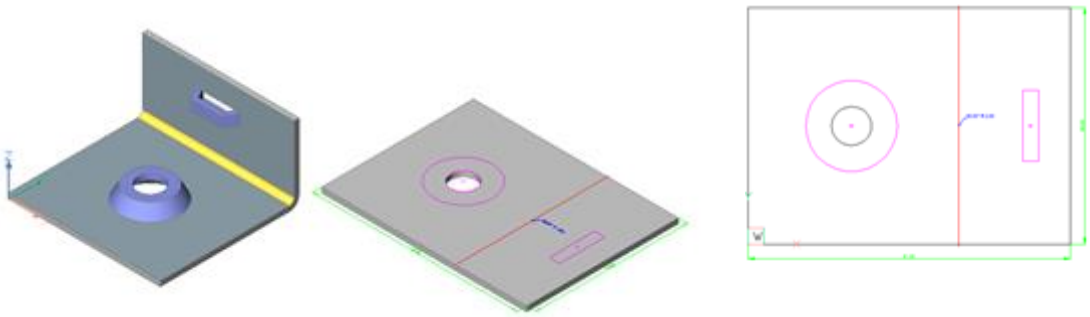
3D model and unfolded body 2D

Project (2): generates a form feature symbol by projecting the form feature edges on the flange plane



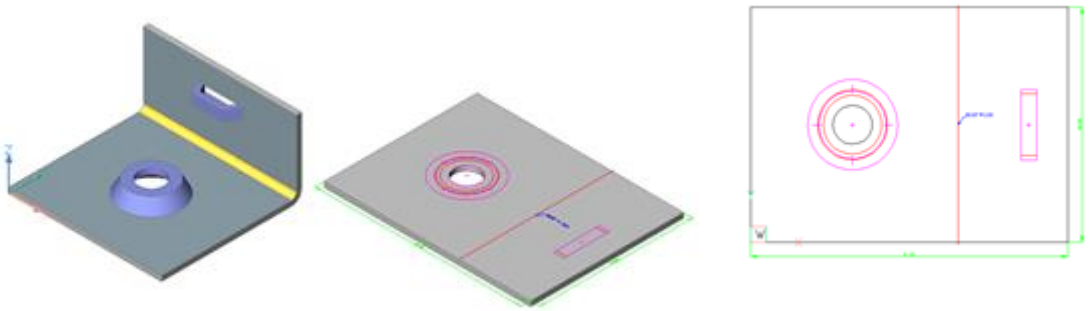
3D model and unfolded body 2D

Contour (3): generates a form feature symbol by projecting an external contour and holes of the form feature on the flange plane



3D model and unfolded body 2D

Symbol (4): copies the unfold symbol on layer UNFOLD_SYMBOL_UP or layer UNFOLD_SYMBOL_DOWN depending on the form feature direction) to the unfolded representation



3D model and unfolded body 2D

The unfold mode for current document can be set in Mechanical Browser or by selecting the appropriate option in the SmUnfold command:

You are prompted:

Select a flange or lofted bend face to start unfolding [Settings]: (type Settings)

Select [lofted bend Tolerance/Form feature behaviour] <Form feature behaviour>: (type Form or press Enter)

Select form feature behaviour [Keep/Remove/Project/Contour/Symbol] <Symbol>: (select required mode of form feature unfolding)

The *SmUnfold* command prompts you to either:

Keep the 3D solid in the model.

Save the 3D solid in a separate drawing file.

Convert the solid to a 2D drawing file (*.dwg or *.dxf)

Overall dimension and bending annotations for manufacturing are added to the exported drawing automatically.

The SmUnfold command takes the deformation of the sheet metal material during bending into account. When a flat sheet of metal is bent into a 3D part with a bending tool (like a press break), the material is plastically deformed, it is compressed inside the bend and stretched outside of it. So the length of the part measured along its surface is different in flat and bend states. BricsCAD is able to automatically compute the proper unfolded length of your part based on the material deformation properties. These properties can be defined by setting the value of K-Factor parameter or by attaching a bend table.

To Export the Sheet Metal Body

Sheet metal parts created or edited in BricsCAD can be then processed by different CAM systems (such as JETCAM or CADMAN-B) to generate NC code for cutting and bending machines.

The SMTARGETCAM system variable controls options in the SmUnfold command which customizes the .dxf file to be compatible with list of popular CAM systems.

The SmExport2d command exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.

The SmExportOSM command exports a sheet metal solid to the .osm (Open Sheet Metal) file format (native for CADMAN-B CAM system).

To Export an Sheet Metal Assembly to DXF

The SmAssemblyExport command automatically traverses the assembly tree and for each 3D solid tries to recognize sheet metal design and export it to .dxf. As a result a set of .dxf files sorted by thickness in different subfolders is created in the output folder. A HTML report generated by the command allows to control the results in clear way. Using this command most of the sheet metal parts are processed automatically, a minor number of poor sheet metal parts are detected and indicated by the command for manual rework.

Working with Form Features

Commands: SMFORM, SMCONVERT, SMREPLACE, SMRIBCREATE, BMINSERT

SmForm: converts a selected set of faces to a form feature. A drawing file with a user-defined form feature can be saved and then inserted using the BMINSERT command.

SmConvert: recognizes form features in imported geometry.

SmReplace: allows replacing form features (including recognized ones) in sheet metal parts with form features from the built-in or user library.

SmRibcreate: creates associative Rib form features on sheet metal parts from 2D profiles. The feature is updated automatically when its defining profile is changed.

BmInsert: inserts an existing mechanical component or sheet metal form feature into the current drawing.

Form features a kind of sheet metal features, corresponding to the result of application of a forming tool to a sheet. Form features can be recognized in imported geometry or inserted from built-in or user-defined libraries. Form features can be edited directly and parametrically. Form features are listed in the Mechanical browser with their parameters. Parameters of form features are also displayed and can be edited in the Properties bar upon selection of one or several faces of one or several form features.

A library of parametrized sheet metal form features has been added. It is available from the Form Features tab of the Tool Pallettes panel. The library contains most commonly used form feature types. To insert a form feature to a sheet metal 3D solid drag the form feature icon from the Tool Palettes panel to a flange of the solid.

A library of parametrized sheet metal form features has been added. It is available from the Form Features tab on the Tool Palettes panel. The library contains most commonly used form feature types. To insert a form feature to a sheet metal 3D solid drag the form feature icon from the Tool Palettes panel to a flange of the solid.

SMFORMFEATUREUNFOLDMODE: system variable controls the appearance of the form features in 2D and 3D unfolded model representations.

To Insert a Form Feature

Form features are added to sheet metal part by insertion on the existing flanges. The form features can be inserted by means of BmInsert command or by drag-and-drop process of required form feature from Form Features tab of Tool Palettes.

Make sure Dynamic Dimensions (DYN) and Dynamic UCS (DUCS) are active.

Do one of the following:

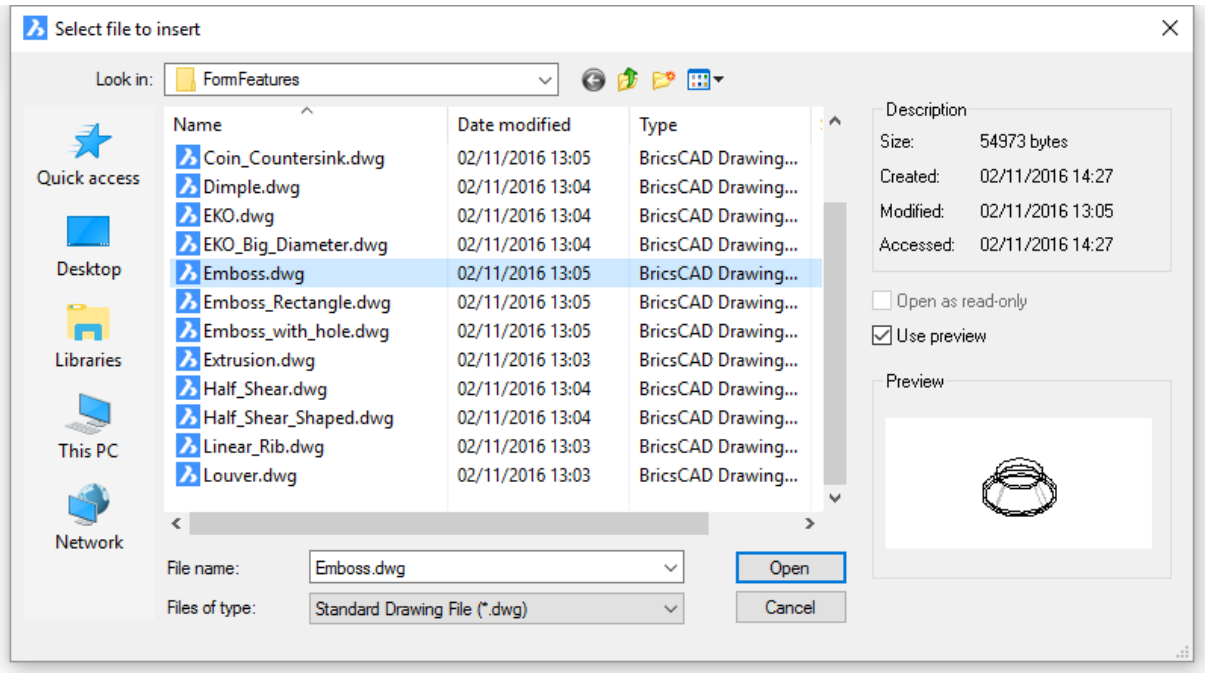
Click the *Form Feature* tool button ( ) on the *Sheet Metal* toolbar.

Click the *Form Feature* tool button ( ) on the *Sheet Metal* ribbon bar.

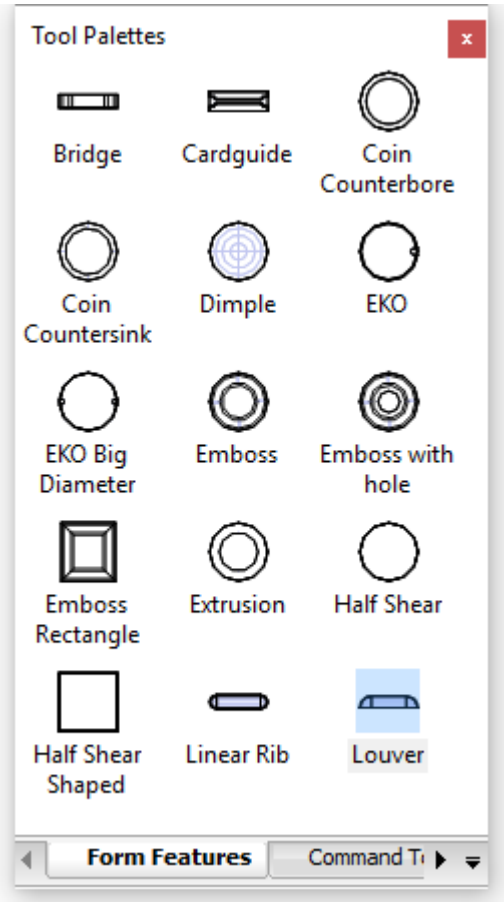
Choose *Insert Form Feature* in the *Sheet Metal* menu.

Type *bminsert* in the command bar, then press Enter.

The *Select file to insert* dialog displays the *FormFeatures* folder *C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V17x64\en_US\Support\DesignLibrary\SheetMetal\FormFeatures:*



Select the desired form feature element from file list, then double click or press the *Open* button.
Select the form feature on the *Form Features* tab of the Tool Palettes panel:



Move the cursor over the flange face you want to insert the feature in.
The dynamic UCS aligns to the selected face.
The orientation of the UCS depends on the edge you enter the face.
Dynamic dimensions display from the origin of the UCS to the insertion point of the form feature.

The command bar reads: Select insertion point or [Edit inserted entity/Rotate component/set Base point/Name/insertion Type/Flip/mUltiple] <0, 0, 0>:

(option) Choose an option (see the BmInsert command for more details):

Edit: Allows to edit the parameters of the selected insert.

Rotate: Allows to define the rotation angle of the component.

Base Point: Allows to specify a different base point to insert the component.

Name: Allows to define the insert name of the component.

Insertion Type: Allows to specify the insertion type: Local or External

Flip: Inserts the form feature in the opposite face of the flange.

Multiple: Allows to insert multiple inserts of the same component or create an array.

To insert a the form feature do one of the following:

Specify a point.

Type a distance in the dynamic dimension fields. Hit the TAB key to go back and forth between the entry fields.

Repeat step 5 to insert another copy or press Enter to stop.

Editing the Parameters of a Library Form Feature

Form features in the form feature library are parametric components. Several parameters control their dimensions. Parameters to control the Length, Width, Height and Radius of the form feature are correspondingly named L, W, H and R. The thickness of the form feature is set equal to the thickness of sheet metal part automatically.

The parameters of a single form feature can be edited either in the Properties Bar or in the Mechanical Browser. Parameters of multiple form features can be edited in the Properties Bar only.

Methods to select form features

Check the *Select Faces* option of the SELECTIONMODES system variable, then click the feature(s) in the model.

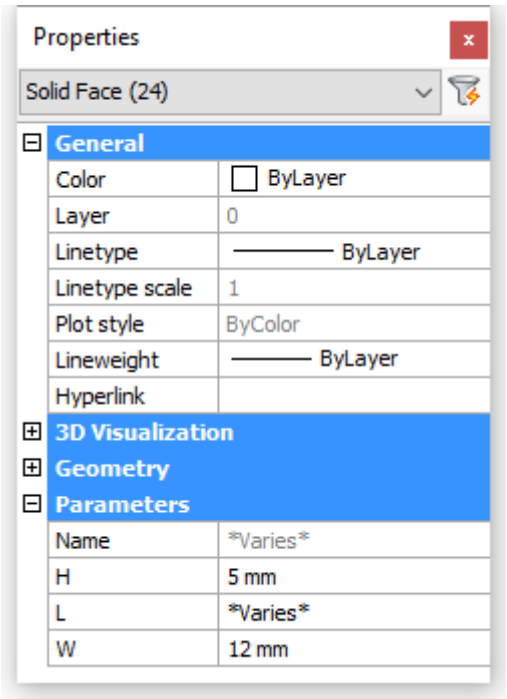
Select the feature(s) in the Mechanical Browser, then press the space bar or right click and choose *Select* in the context menu. Press and hold the Ctrl-key to select multiple form features.

A sheet metal part can have lots of different form features, which might complicate operations such as deletion or replacement, especially if only a selection of features, that meet certain conditions, need to be processed. In order to help you to select form features more easily, the SmSelect command allows to select form features that are either identical to a selected feature or similar.

Editing parameters in the Properties Bar

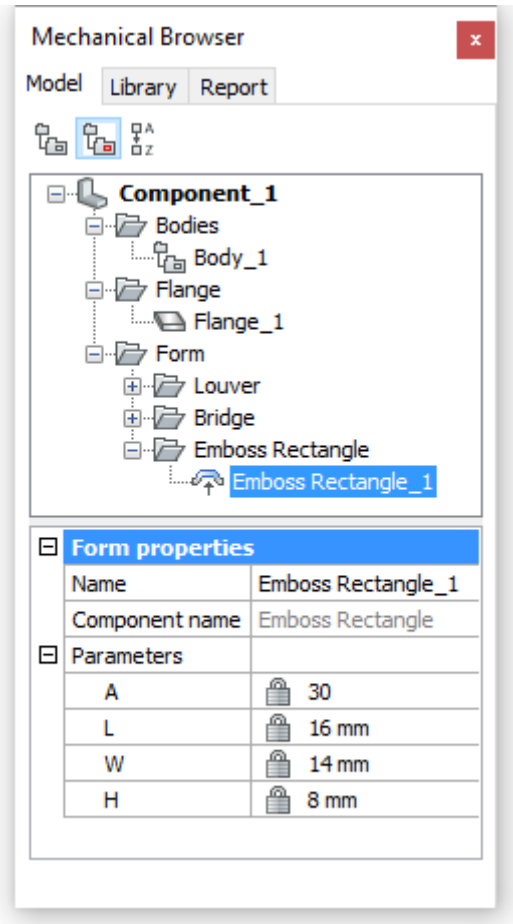
Select the form feature(s).

Select a parameter in the Parameters node in the Properties Bar.



Editing parameters in the Mechanical Browser

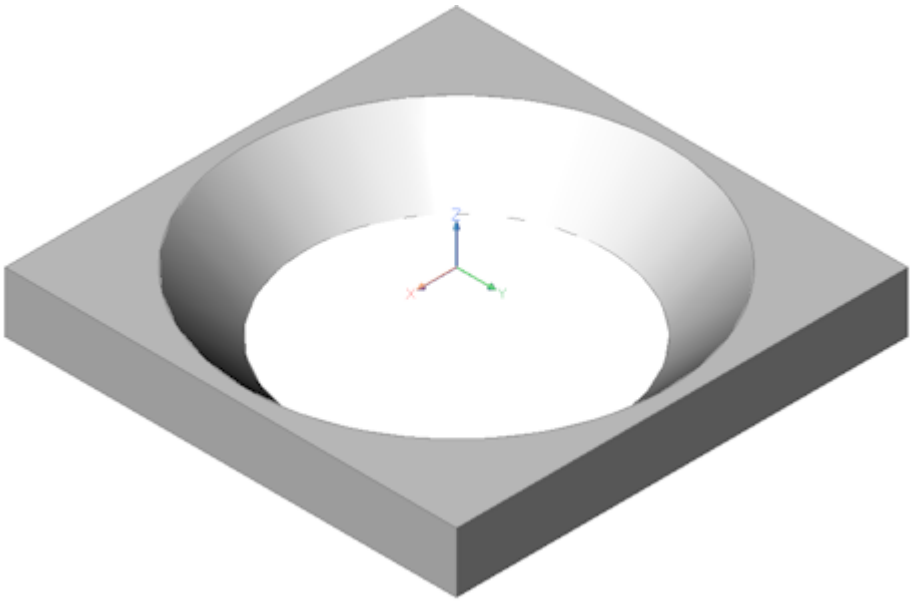
Select the form feature in *Form* node of the component structure tree.



Select a parameters in the lower panel of the Mechancial Browser and type a new value.

Creating a Custom Form Feature

Open a new drawing using *Mechanical-metric.dwt* as the template.
Create the 3D geometry of the form feature.



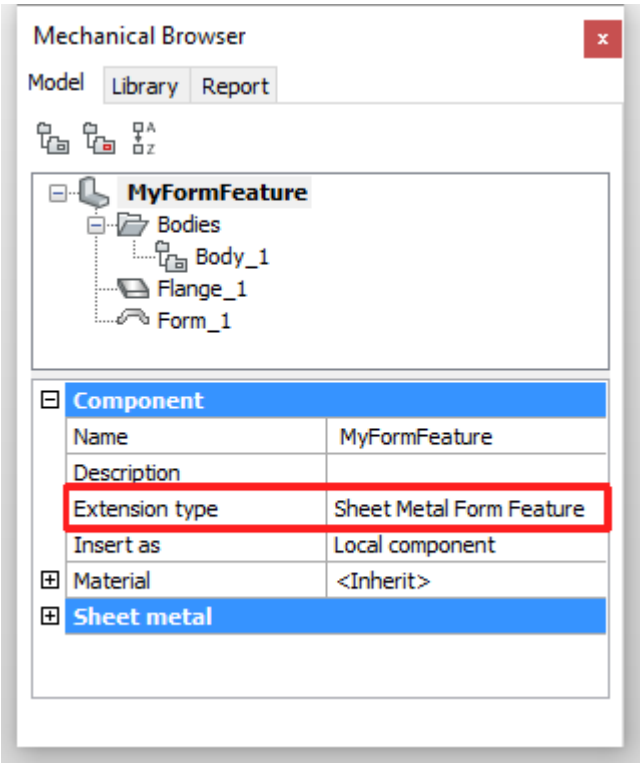
Execute the SmConvert ( ) command.

Exectute the BmMech ( ) command.
You are prompted: Name for mechanical component <Component_2>:

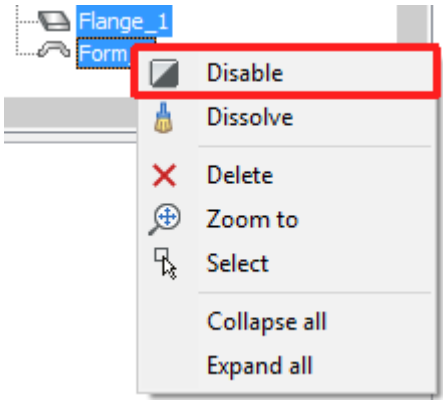
Type a name for the form feature.
You are promted: Convert blocks and external references to mechanical components? [Yes/No] <Yes>:

Select the *No* option.

In the *Extension type* field in Mechanical Browser select *Sheet Metal Form Feature*.



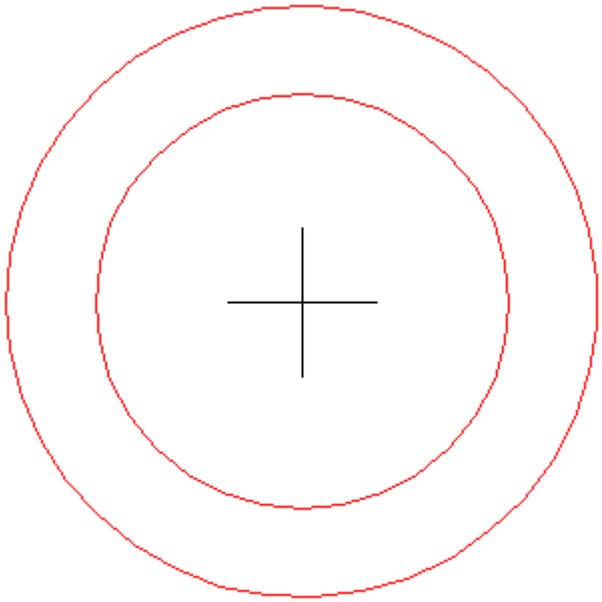
Select the Flange_1 and Form_1 features in the *Bodies* node, then right click and select *Disable* in the context menu.



Create 2 new layers: UNFOLD_SYMBOL_UP and UNFOLD_SYMBOL_DOWN.

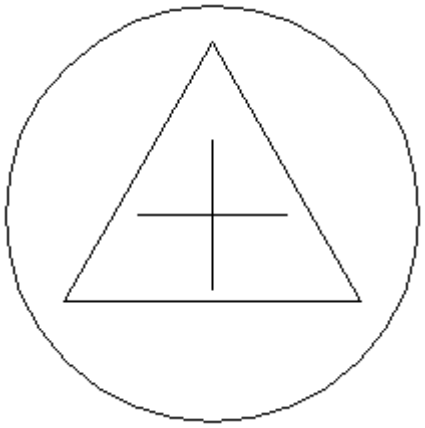
Turn layers 0 and UNFOLD_SYMBOL_DOWN *Off* and set layer UNFOLD_SYMBOL_UP *Current*.

Create the unfold symbol, which will be placed on the face the form feature is placed in.



Turn off the UNFOLD_SYMBOL_UP layer and turn on the UNFOLD_SYMBOL_DOWN layer. Set the UNFOLD_SYMBOL_DOWN layer current.

Create the unfold symbol, which will be placed on the opposite face the form feature is placed in.



Turn on layer 0 and turn the UNFOLD_SYMBOL_UP and UNFOLD_SYMBOL_DOWN layers off. Set layer 0 current.

Save the drawing.

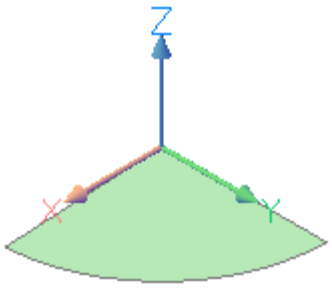
Creating a Custom Parametric form Feature

We will now parametrize the custom form feature we created in the previous procedure.

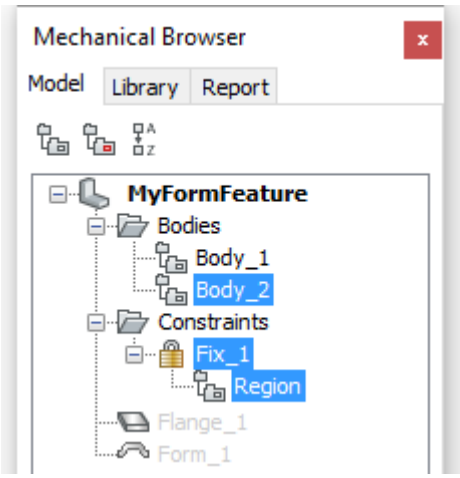
Two parameters are needed here: D for diameter of the bigger edge of the conical hole and A for angle of the face of the conical hole. In Sheet Metal we have a parameter _SM_THICKNESS which is equal to the thickness of the sheet metal part.

We will first create an auxiliary body for constraining on a new CONSTRUCTION layer. Make this layer current and switch off all other layers.

Create a Region on the CONSTRUCTION layer in the XY plane of the WCS.

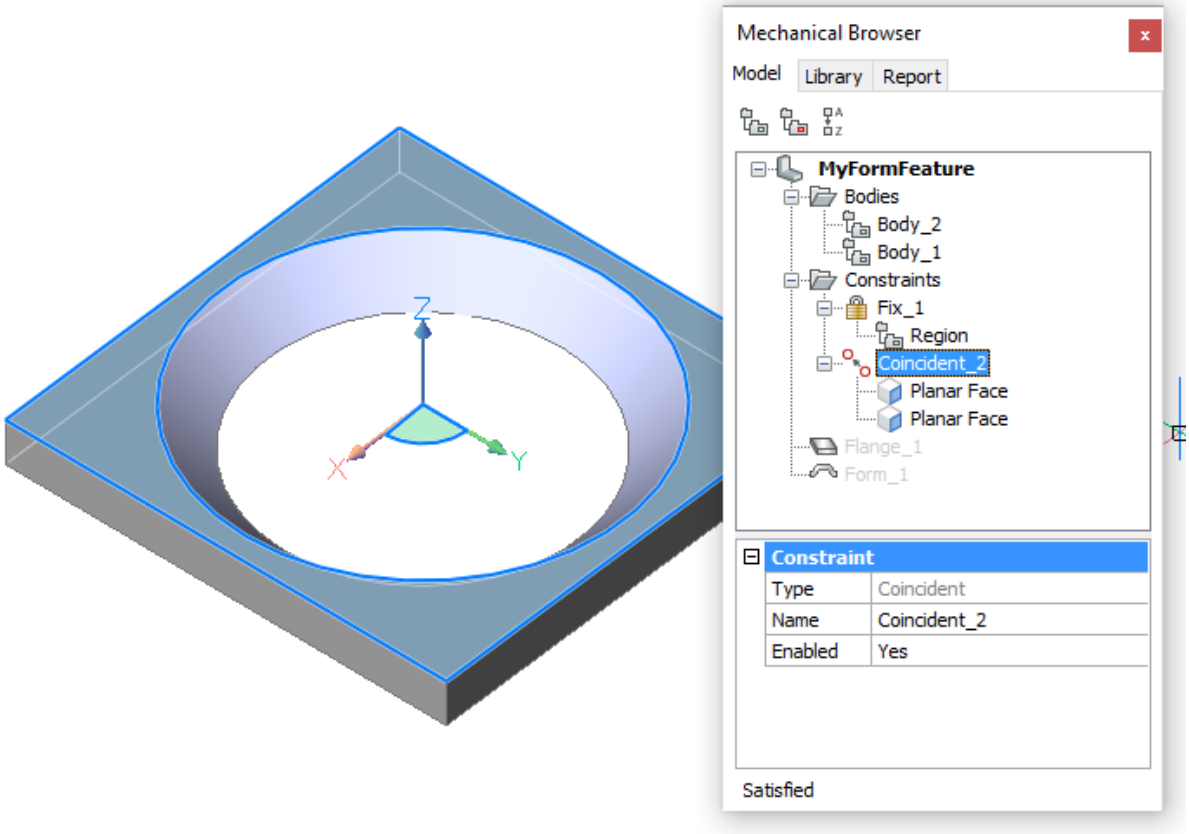


Apply a DmFix3D constraint ( ) to the region.



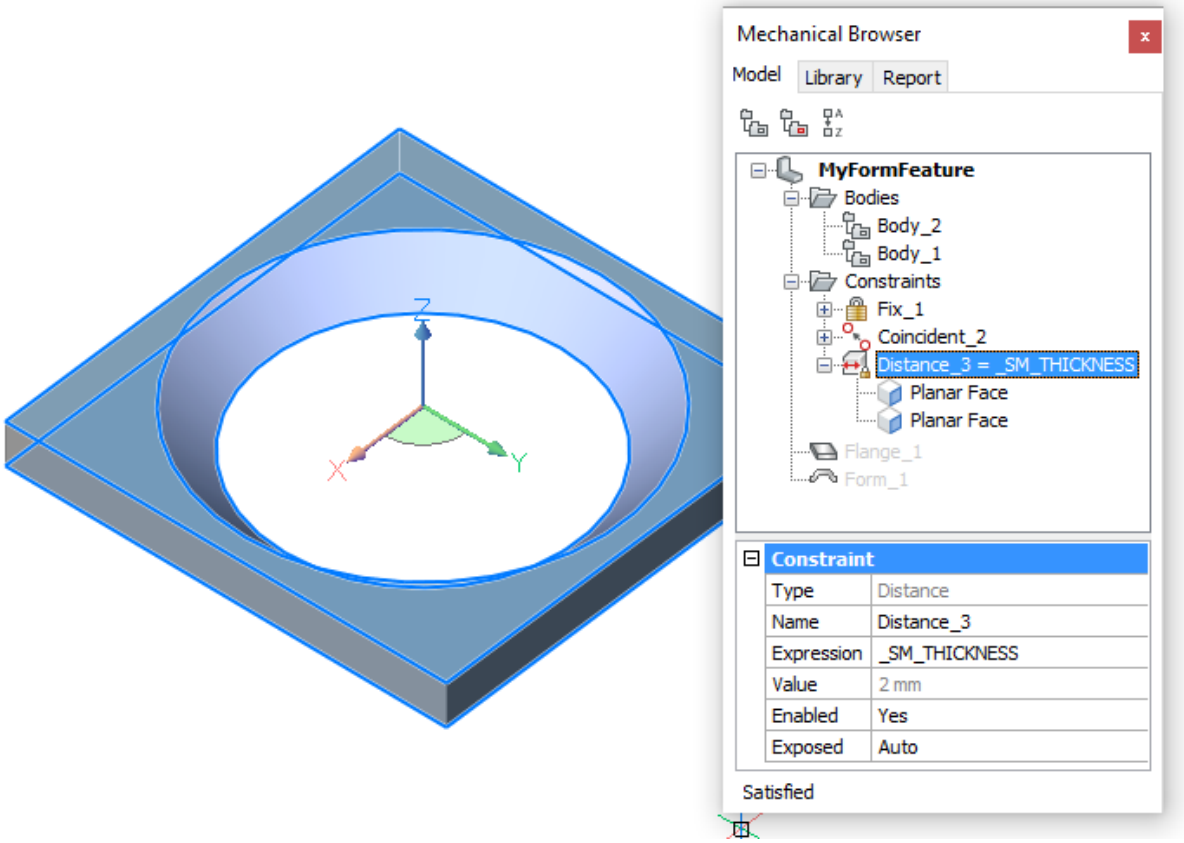
Switch on layer 0.

Apply a DmCoincident3D constraint ( ) between the region face and the flange top face.

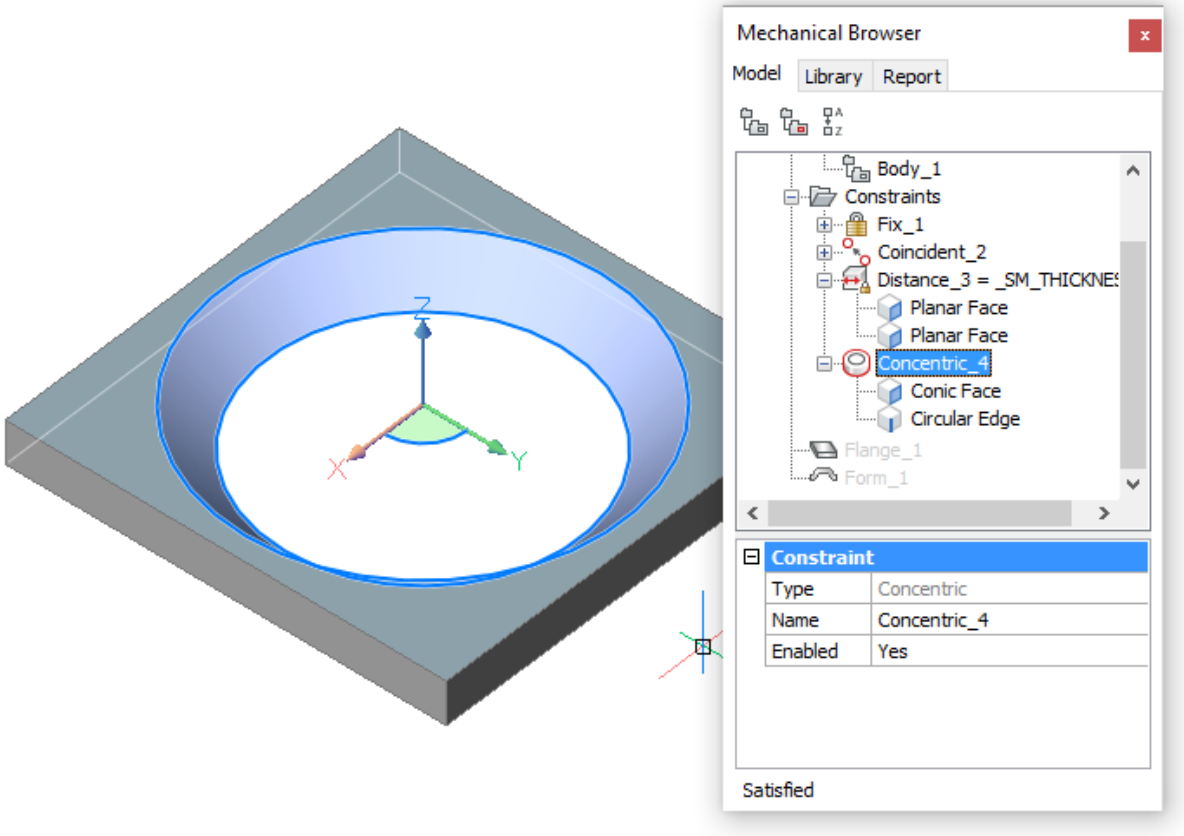


Apply a DmDistance3D constraint ( ) between the flange top and bottom faces.
First select the top face, then hit the TAB key to select the bottom face.
Accept the default value in the dynamic dimension.

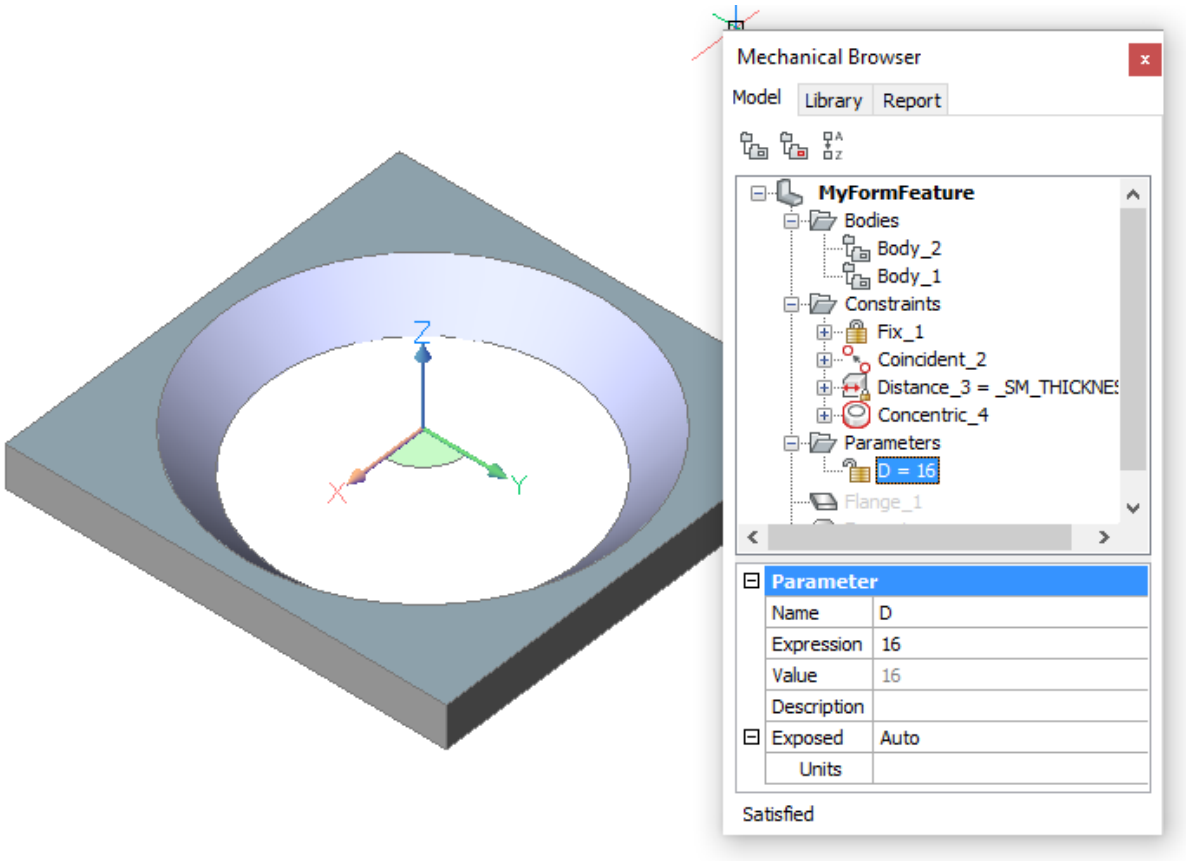
Type _SM_THICKNESS in the *Expression* field of the Distance_3 constraint.



Since our Form Feature is conical we can modify the flange from parallelepiped to cylinder

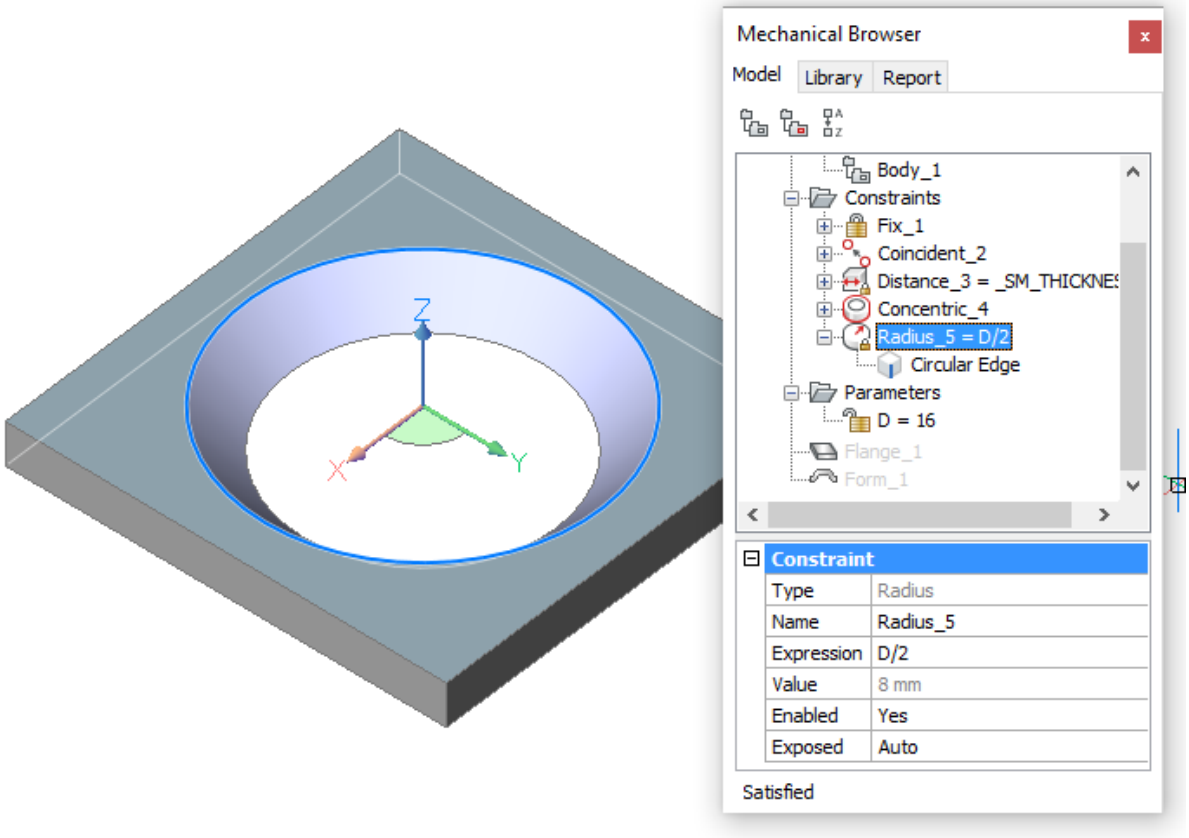


In the Mechanical Browser right click the feature name and select *Add new parameter* in the context menu.
Rename new parameter to *D* .
Select mm for the *Units* field.
Type 16 in the *Expression* field.



Apply a DmRadius3D constraint ( ) to the top edge of the conical face. Accept the default value in the dynamic dimension.

Type D/2 in the *Expression* field of the Radius_5 constraint.



In the Mechanical Browser right click the feature name and select *Add new parameter* in the context menu.
Rename new parameter to *A* .
Leave the *Units* field empty.
Type 45 in the *Expression* field.

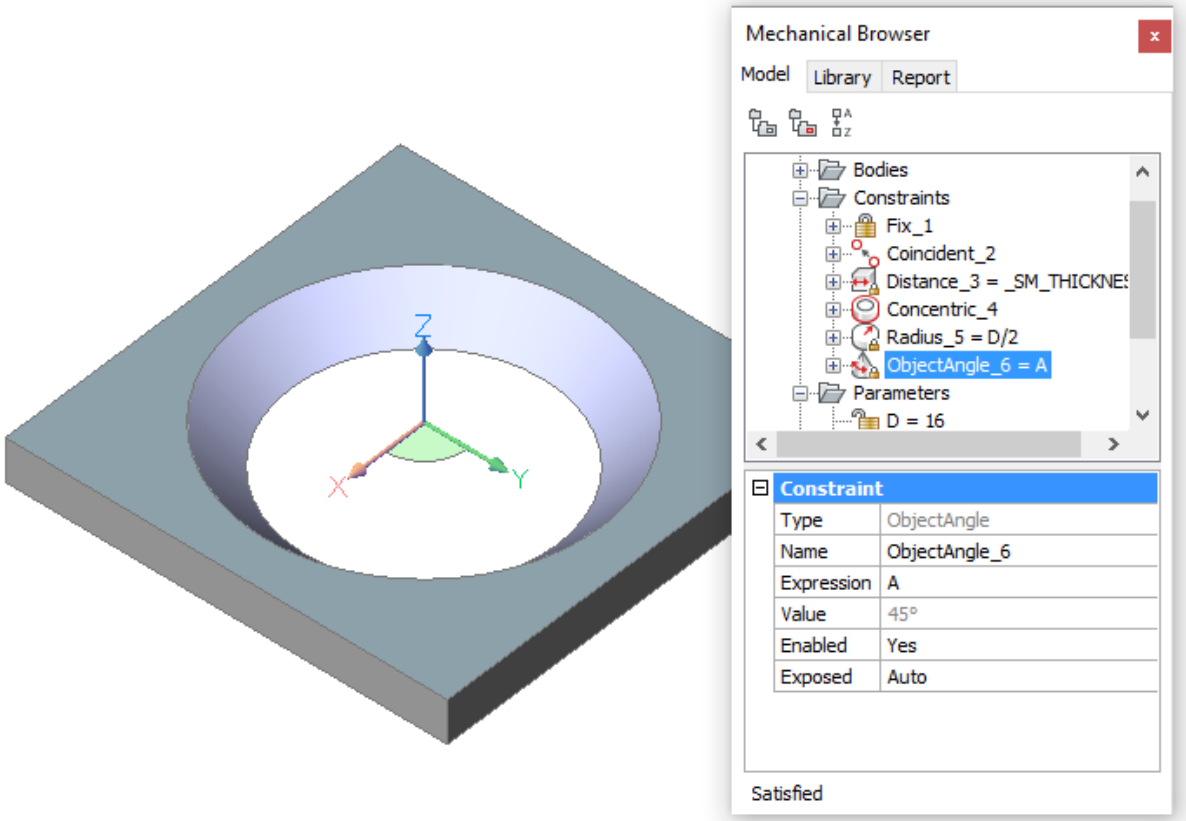
Do one of the following to create a *Cone Angle constraint* to the conical face:

Launch the DmAngle3d command (), then choose the *Cone Angle constraint* option, then select the conical face.

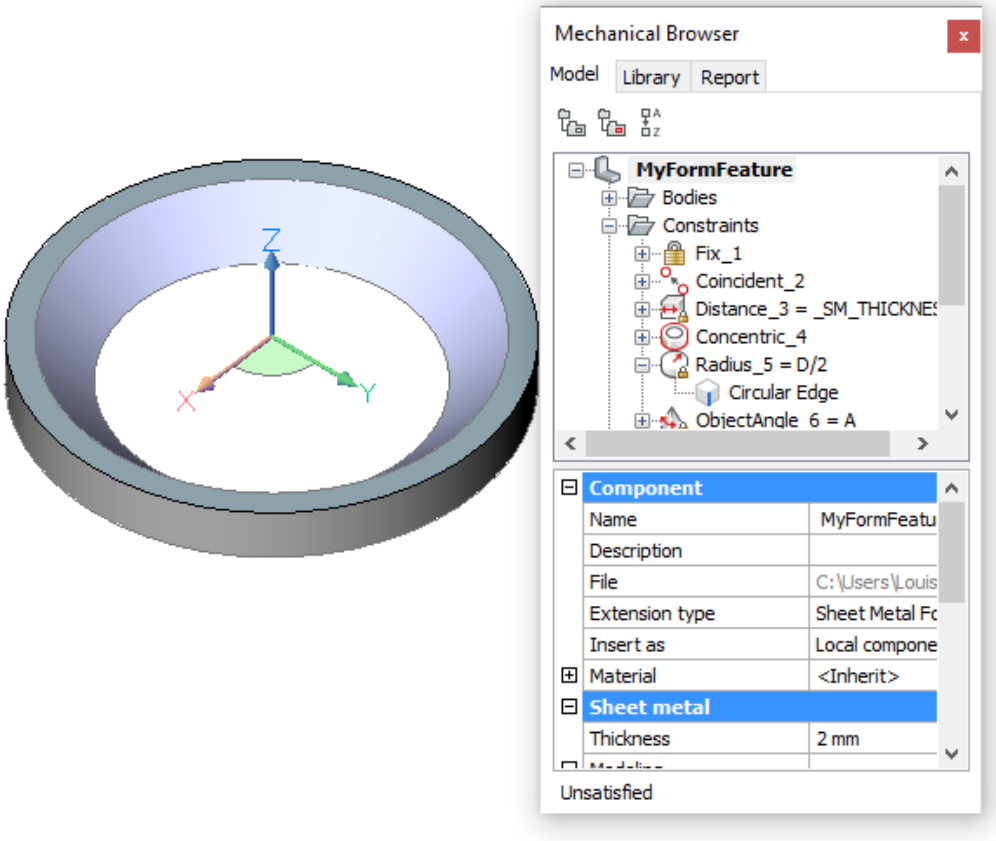
Hover over the conical face, then choose *Cone Angle constraint* () in the *Constraints* command group in the Quad.

Accept the default value.

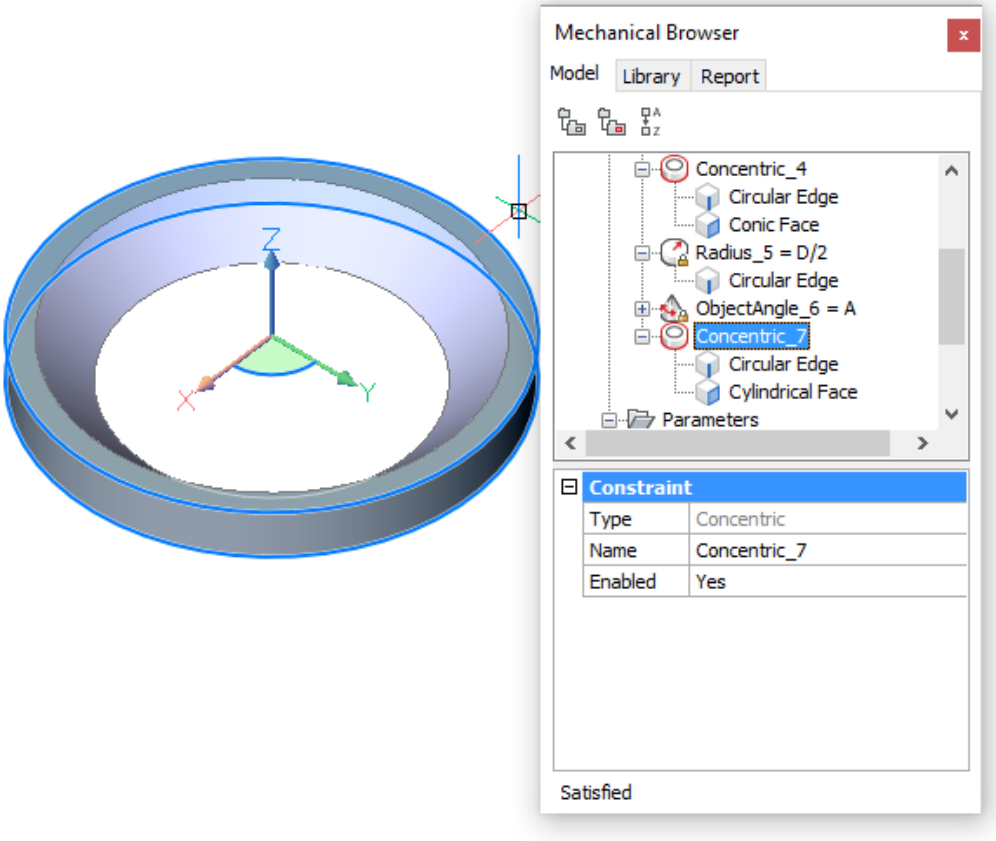
Type A in the *Expression* field of the ObjectAngle_6 constraint.



Since our Form Feature is conical we will modify the flange from parallelepiped to cylinder:
Draw a circle on top or the flange. The radius of the circle is 9 mm.
Select *Enable Boundary Detection* in the *Settings* panel on the *Home* tab in the Ribbon.
Place the cursor over the area between the circle and the square edge of the flange, then choose *Extrude* ( ) in the Quad and move the cursor downwards.

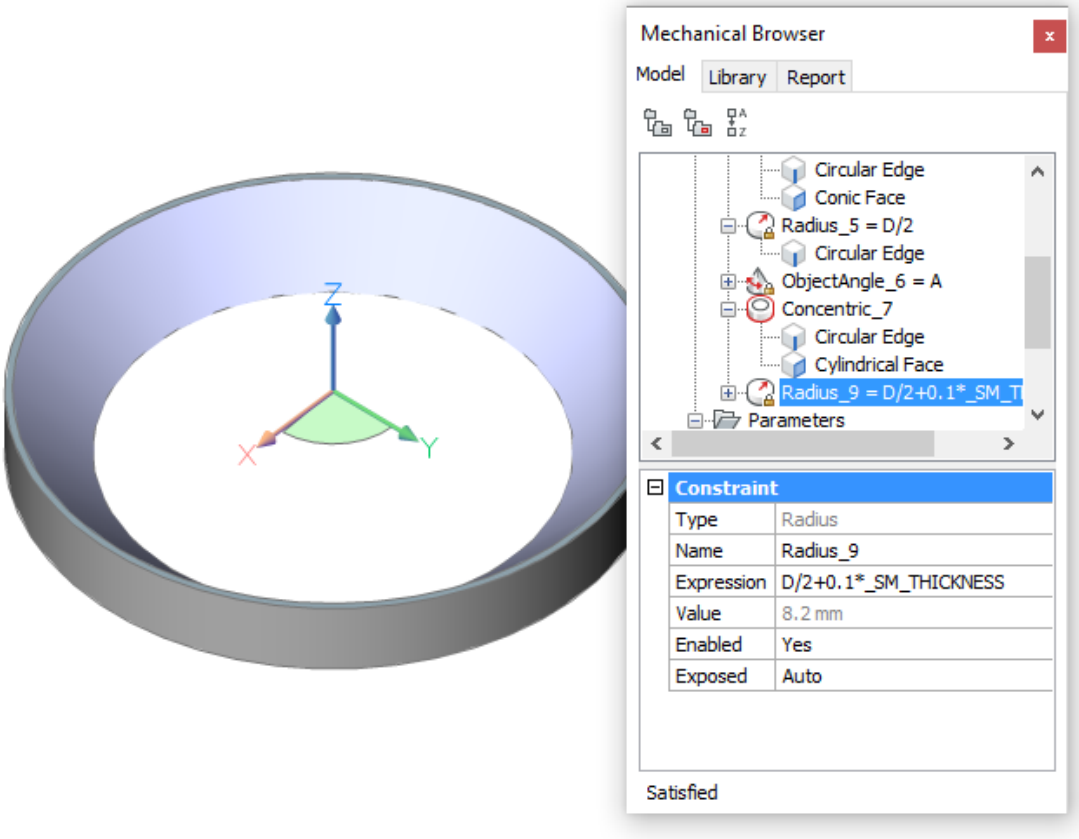


Apply a DmConcentric3D constraint ( ) between the circular edge of the region and the cylindrical face.

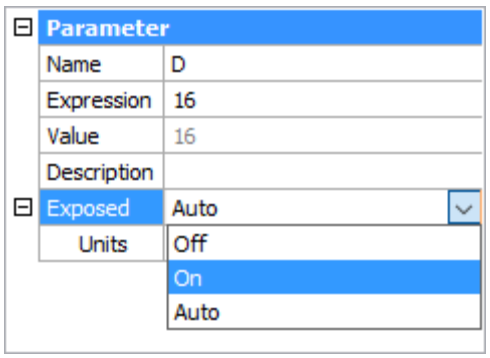


Apply a DmRadius3D constraint ( ) to the cylindrical face. Accept the default value in the dynamic dimension.

Type D/2+0.1*_SM_THICKNESS in the *Expression* field of the Radius_8 constraint.



Set the *Exposed* field of both parameter A and D *On*.



We will now create 2D constraints for the unfold symbols.

Make layer UNFOLD_SYMBOL_UP current and switch off all other layers.

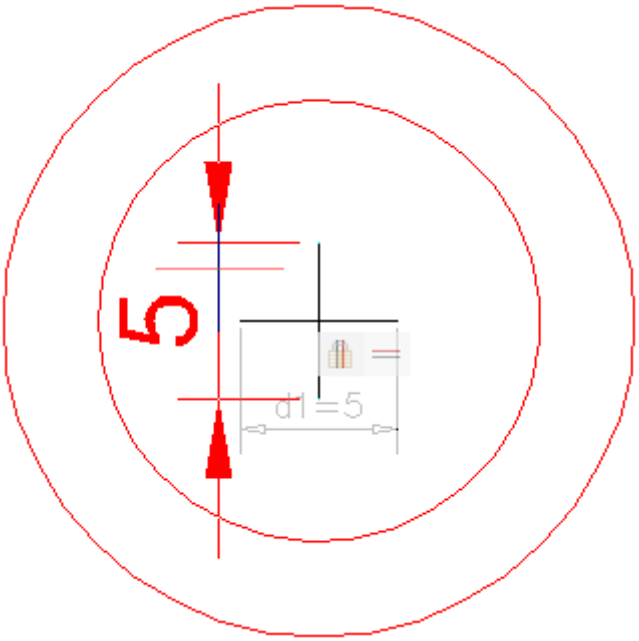
Apply a GcFix constraint ( ) to both crossing lines indicating the center of the circles.

Apply a GcHorizontal constraint ( ) to the horizontal center line.

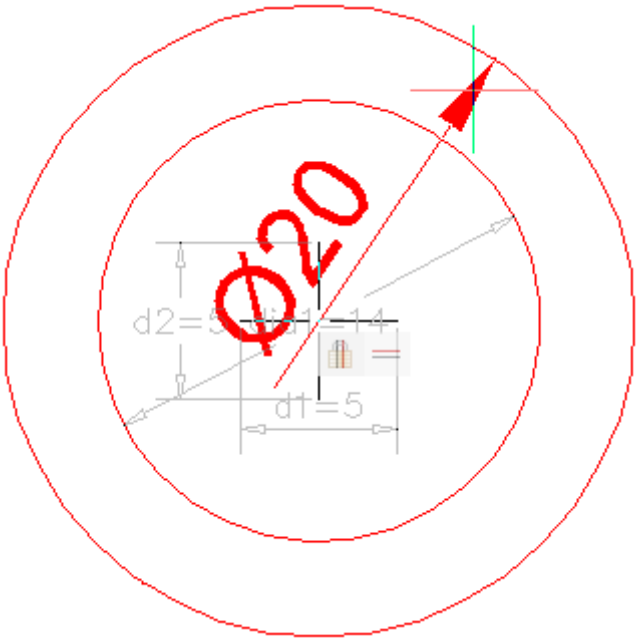
Apply a GcVertical constraint ( ) to the vertical center line.

Apply a GcConcentric constraint ( ) between the two circles.

Apply a DcLinear constraint ( ) between the endpoints of the center lines. Accept the default value.



Apply a DcDiameter constraint ( ) to both circles. Accept the default value for dia1 (= inner circle) and dia2 (= outer circle) .



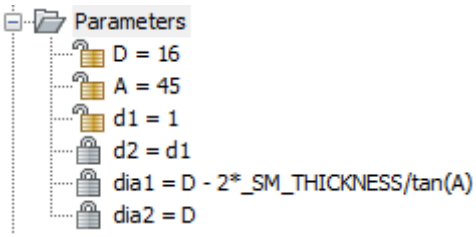
Adjust the *Expression* fields of the 2D constraints as follows:

d1 = 1

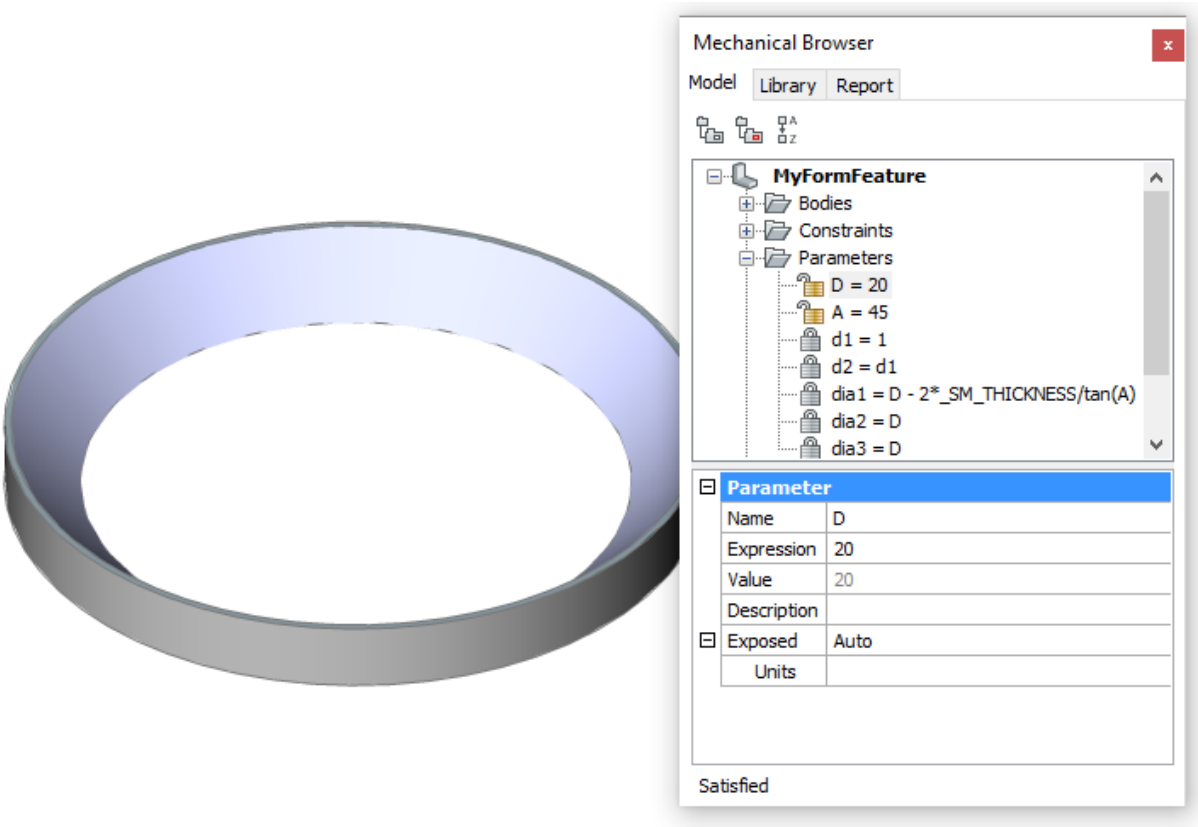
d2 = d1

dia 1 = D - 2*_SM_THICKNESS/tan(A)

dia 2 = D

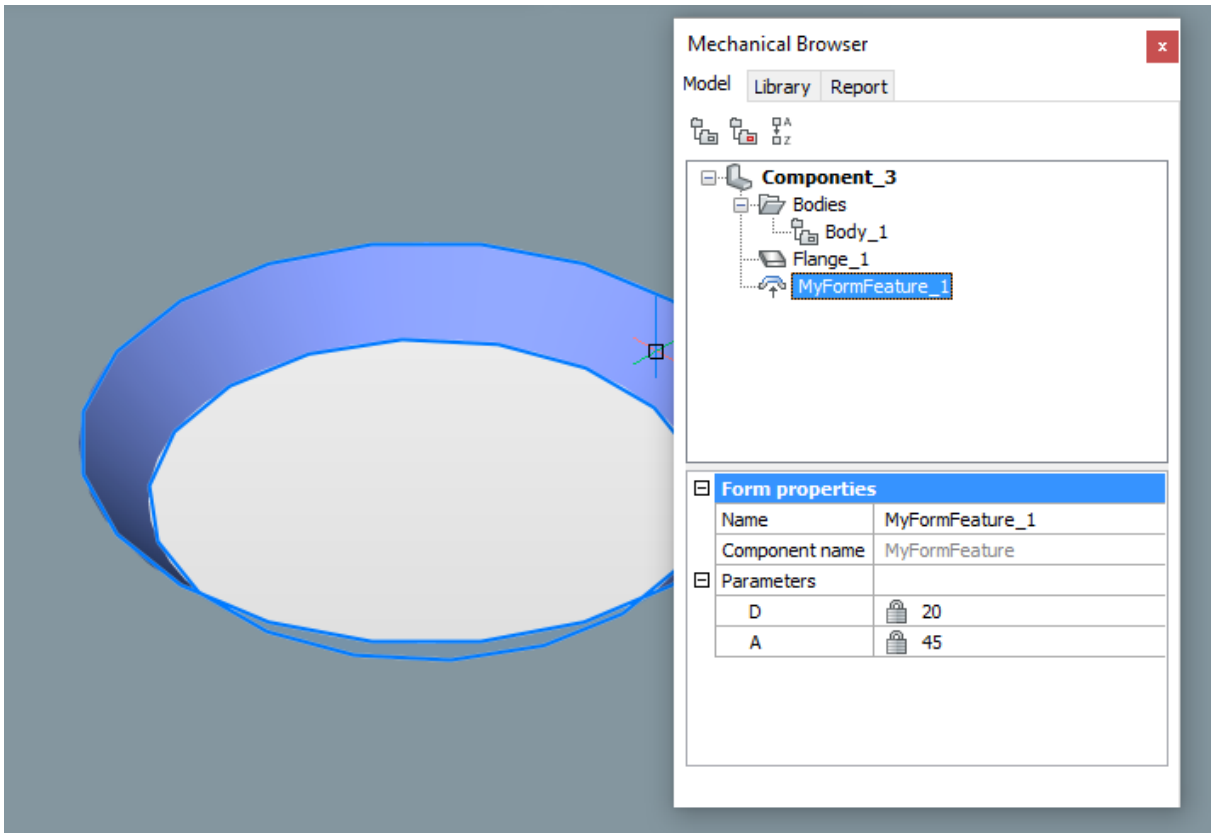


Switch off the display of all layers, but layer 0, then save the file.



The form feature can now be used to make conical parametric openings in a sheet metal flange:

Use the *Insert Form Feature* tool ( ), which launches the BmInsert command.



Optionally: Add the form feature as a component to a tool palette.

Creating a Rib Feature

Rib features are created by SmRibCreate command from a 2D curve on flange face.

Do one of the following:

Click the *Create Rib Feature* tool button ( ) on the *Sheet Metal* toolbar.

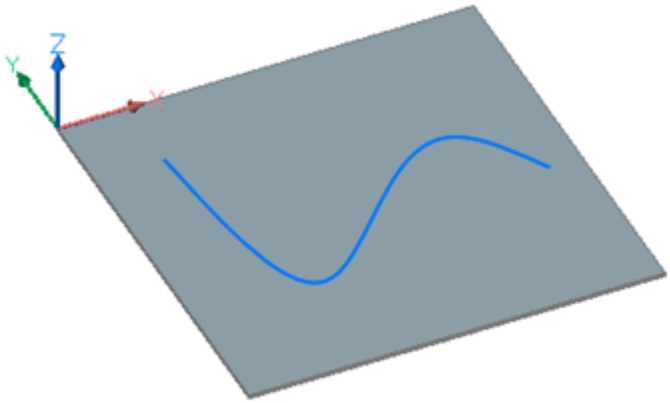
Click the *Create Rib Feature* tool button ( ) on the *Modify* panel of the *Sheet Metal* tab on the ribbon.

Type *smribcreate* in the command bar, then press Enter.

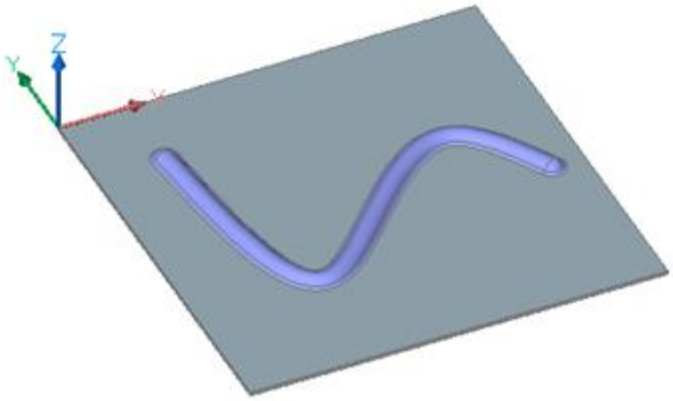
Prompts you: Select flange face:

Select the flange face.
Prompts you: Select curve or [Profile radius/Round radius]:

Select a curve on the selected flange face.
Lines, polylines, circles, arcs, ellipses, elliptical arcs and splines are accepted. The 2D curves can be open or closed but cannot intersect itself or other form features.



After selection of the curve rib feature is created:



Modifying a Rib Feature

To modify a rib feature:

Modify the generatrix curve using its grip points.

Edit the *Profile* and *Round* radius in the Mechanical Browser.

Modify the generatrix curve

Select the curve.
Because the curve is hidden by the rib feature, press the TAB key to select obscured entities, or select the curve in the Structure panel.

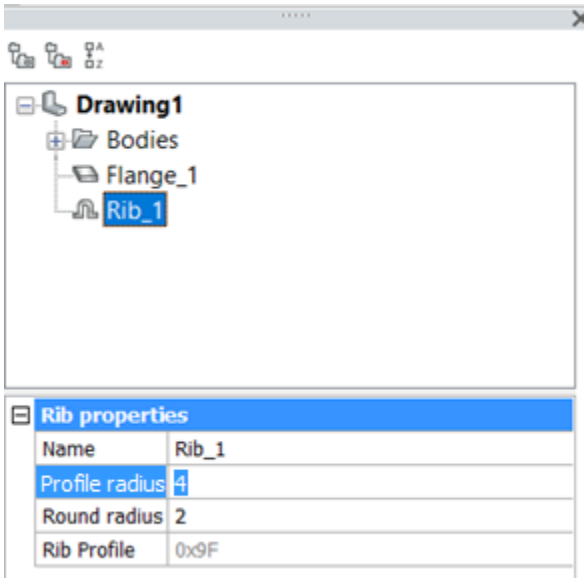
Grip edit the curve.
The Rib feature updates according to the changes of the curve.

After selecting of the flange face command prompts: Select curve or [Profile radius/Round radius]:.
Select curve on selected flange side.

Edit the properties in the Mechanical Browser

Select the rib feature in the Mechanical Browser
The rib feature highlights in the model.

Select the property in the *Rib Properties* grid.



Type a value.
The selected rib feature is updated.