

What's New? BricsCAD V19

by Ralph Grabowski

What's New in BricsCAD V19

This list of BricsCAD's new and changed functions was compiled from version 19.1.06-2. Changes are highlighted throughout this book, but be aware that information on these pages is not comprehensive, as Bricsys continually updates this software. For information on functions added since this book was published, please see <https://www.bricsys.com/common/releasenotes.jsp>.

New command and variable names are shown here in boldface **blue**, and updated ones in boldface **black**. They are listed in alphabetical order in the following sections:

- User interface
- General commands & variables
- Assemblies
- BIM module
- Communicator module
- Import & export commands
- Generated views
- Sheet metal module
- APIs
- Licenses

BricsCAD V19 installs and runs independently from previous BricsCAD versions.

Visit the *BricsCAD for AutoCAD Users* Web site at <http://www.worldcadaccess.com/ebooksonline/>. At this Web page, editions of this book are available for BricsCAD V8 through V18.

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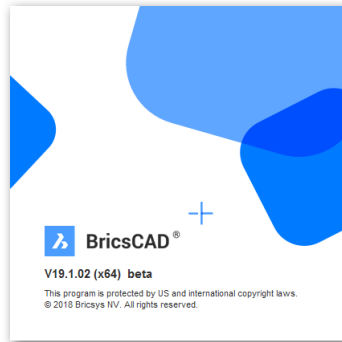
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This work is based upon knowledge available at time of writing, and does not claim to be a full representation of what is new in BricsCAD V19, nor does it claim to be error-free. Do not make any decisions based upon the content of this book.

WHAT'S NEW IN THE USER INTERFACE

BricsCAD V19 displays a new splash screen when it starts up:

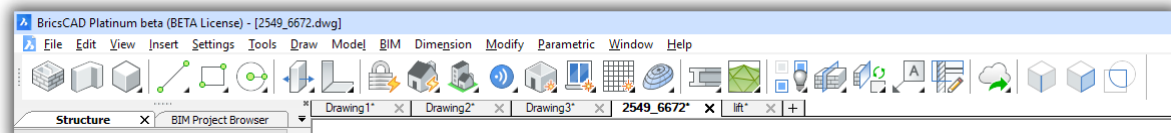


BricsCAD showing new splash screen for V19

Workspace command add a new workspace, "Drafting (Toolbars)," which replaces the ribbon with toolbars and the menu bar.

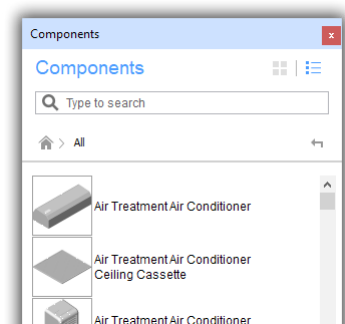
ProfileManager command now can change profiles without requiring Bricscad to restart.

The BIM workspace is changed to look like that of Shape, with a toolbar-like ribbon.



BIM workspace sporting a toolbar-like ribbon

Some panels (palettes) have taken on the look from Shape, such as the panel name in a large blue font.



Panel showing Shape-like look

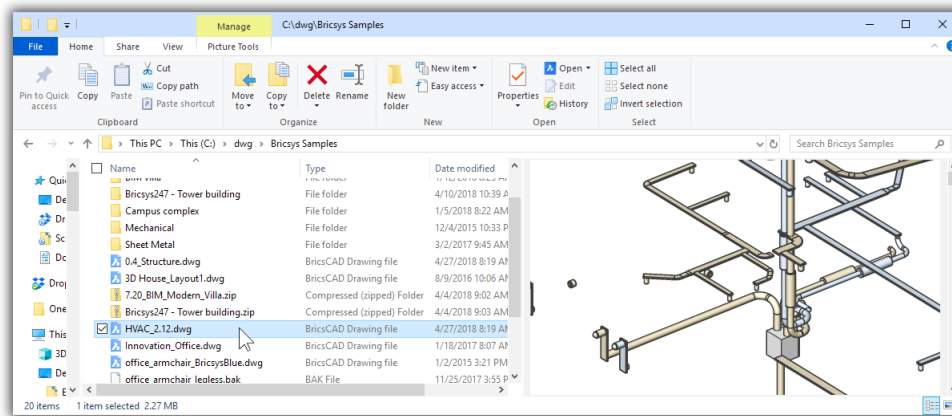
The ribbon now supports row breaks.

V19 improves display performance of BricsCAD:

- Zooms and pans are 2x faster for drawings containing a lot of tiny geometry; when using anti-aliased mode (AntiAliasScreen > 1) the performance improvement is 5x.
- Hatching is 100x faster for hatches with boundaries containing thousands of segments.

Dragged entities now remain visible during view manipulations like zoom, pan and view rotation.

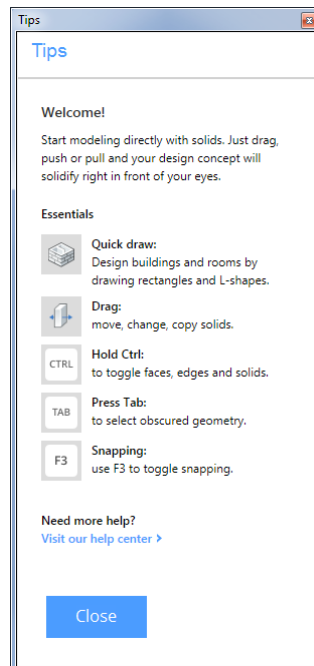
V19 registers with Windows to provide preview images for File Explorer. This lets you quickly view the contents of files in folders of DWG files before opening them in BricsCAD.



Previewing a DWG file in Windows Explorer

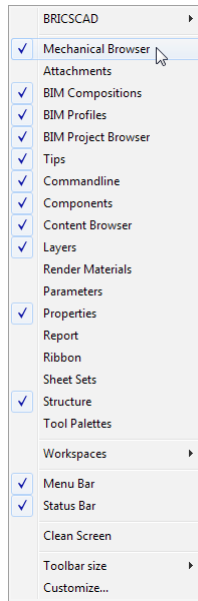
Help command provides help and tutorials online at <https://help.bricsys.com/hc/en-us>.

The new Tips panel animates command tutorials. To access it, right-click the ribbon or a toolbar, and then choose **Tips**.



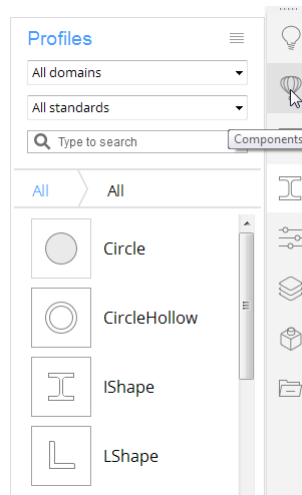
Animated help in the Tips panel

BricsCAD now has 19 panels; right-click a user interface element, such as the ribbon or a toolbar to see the complete list.



Panels included in V19

Flyouts minimize the space panels take up, replacing the tabs from earlier releases of BricsCAD. Flyouts can appear along the left or right edge of the drawing screen, and can be dragged from one edge to the other.

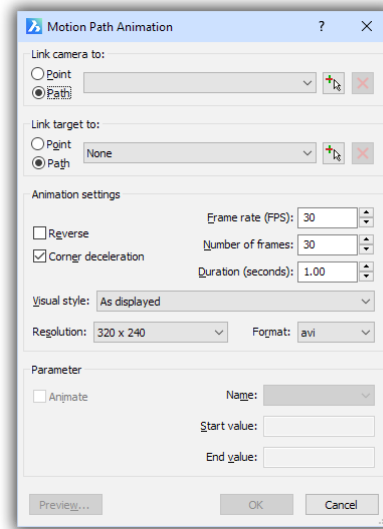


Flyouts appearing from the right edge of the V19 screen.

“BricsCAD V19 Mechanical” is the name of a new vertical package. It replaces the previous Sheet Metal add-on, and combines mechanical design and sheet metal design with BricsCAD Platinum.

WHAT'S NEW IN GENERAL COMMANDS & VARIABLES

Anipath command's dialog box now supports variable parameter values during movies.



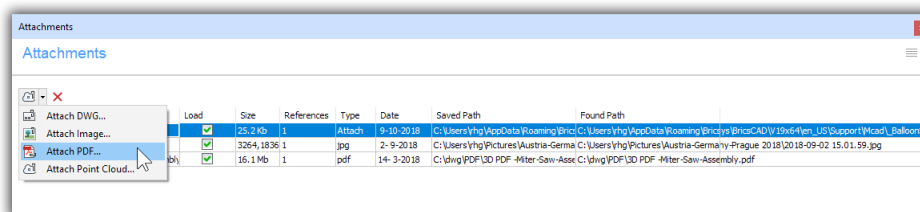
Parameter settings are found in the lower portion of the Motion Path Animation dialog box

Array command now works with parametric components.

TIP Component-based features (such as windows and doors) created with the new BC_UNITE and BC_SUBTRACT layers can be multiplied using associative arrays. This makes it possible to create solids, such as walls, with repetitive openings like windows.

AttDef command accepts **Ctrl+Enter** as the shortcut for clicking the **OK** button.

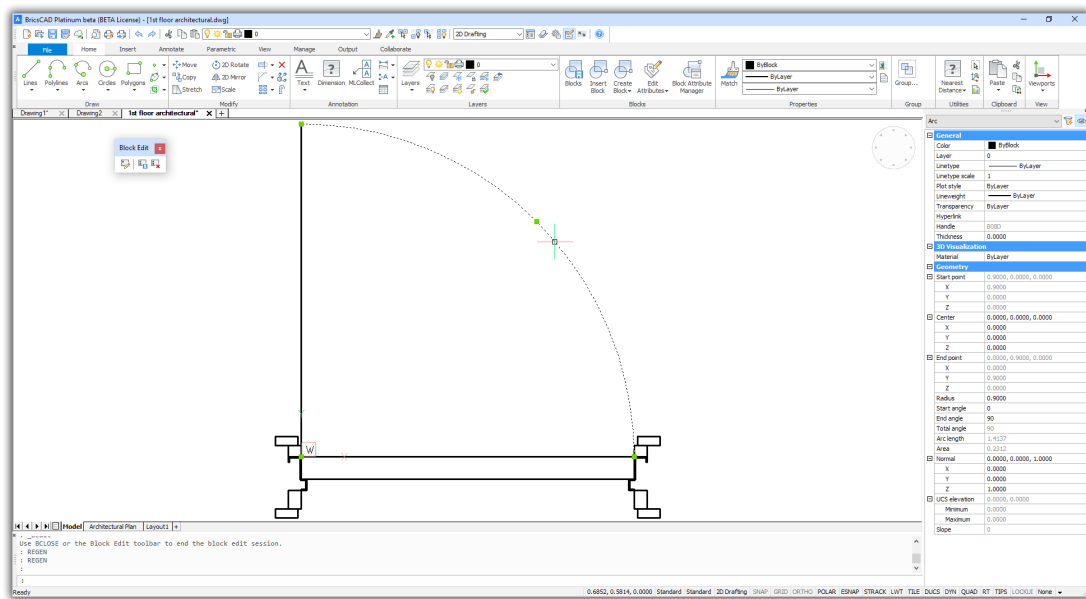
AttachmentsPanelOpen opens the Attachments panel for loading and managing xrefs, raster images, PDF files, and pointcloud attachments.



Attachments panel controlling DWG, image, PDF, and point cloud attachments

AttachmentsPanelClose closes the Attachments panel.

BEdit displays block entities in a temporary drawing session to create and edit block definitions; the block is opened in using the its local coordinate system. (It does not support AutoCAD's dynamic blocks.)



Block editing environment with Block tab in the ribbon

TIP Double-click a block to open it in the new block editor environment.

BClose exits the block editor, saving or not saving changes to the block.

Blockify command detects equally-shaped entities, and then replaces them with an equivalent block. The entities can be manually selected 2D entities (lines, polylines, arcs, circles, ellipses, splines, points) or 3D solids, or 3D solids automatically selected by BricsCAD. In this release, a mixed selection of 2D and 3D entities is not supported.

For example, select a line segment. BricsCAD searches the drawing for all other lines of the same length, creates a block that mimics them, and then replaces the lines with the block:

```
: BLOCKIFY
Select input entities or [Find all groups] <Find all groups>: (Select the Line segment)
Entities in set: 1
Select input entities or [Find all groups] <Find all groups>: (Press Enter to continue)
Select search space or [use entire Model space] <use entire Model space>: (Press Enter to search the entire drawing)
Duration of finding similar groups: 0.000164 sec
Number of block inserts: 7
Select block insertion point or [use Default point] <use Default point>: (Press Enter)
```

The drawing looks no different, as the created block looks identical to the replaced entities. BricsCAD gives the block a generic name, such as 'block1'.

Here is the meaning of the Blockify command's options:

Select input entities	Select one or more representative entities to be replaced by block
Find all groups	Have BricsCAD find multiple instances of entities automatically
Select search space	Select the area of the drawing to search
use entire Model space	Have BricsCAD search the entire drawing
Select block insertion point	Pick a point, should you wish the block offset from the entities
use Default point	Have BricsCAD use the natural insertion point

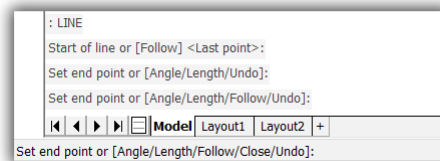
TIPS Replacing common elements with block references reduces the drawing size in memory and when saved to disk, as well as improving opening, drawing, zooming, and saving performance.

Use Blockify to convert general polylines in imported PDF files to blocks.

Centerline and **CenterMark** commands can now be applied to geometry in blocks and in drawing view viewports. The centerline entities can be copied and pasted, and exploded. **CL** and **CM** aliases are added.

TIP V19 no longer allows the creation of Center entities referring to geometry contained in non-uniformly scaled blocks

ClipPromptLines variable specifies the number of lines of command history to appear in the drawing area. (CLI is short for “command line interface.”) When the command bar is closed (with Ctrl+9), the text of the command history is displayed in the drawing area. The text fades away after the number specified by this variable.



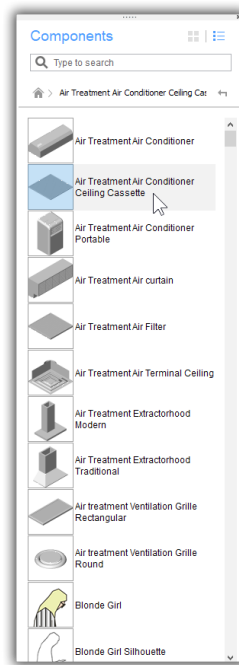
Prompt lines in the drawing area

0 Turns off the display of the command history in the drawing area

4 Default

64 Maximum number of lines of command history displayed

ComponentsPanelOpen command displays the Components panel for accessing standard parts, holes and sheet metal form features. It also acts as a placeholder for user-defined parametric components.



Components panel handles parametric blocks

TIP The Standard Parts library was reworked in V19 to become parametric components inserted with the **bmlInsert** command; parameters can be changed after insertion. The parts are now held by the new Components panel.

ComponentsPanelClose closes the Components panel.

ConvToMesh command converts the following entities mesh objects (mesh smoothing is not yet supported):

- > 3D faces
- > 3D solids
- > 3D surfaces
- > Closed polylines
- > Polyface meshes
- > Polygon meshes
- > Regions

ConvToSolid command converts the following entities to 3D solids:

- > Watertight 3D meshes
- > Watertight surfaces
- > Polyface meshes
- > Closed polylines with thickness and uniform width
- > Circles with thickness

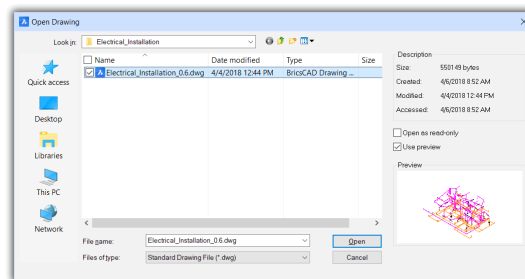
TIP A watertight mesh is one that completely encloses a volume with no gaps or openings – so that no “water” can leak out.

ConvToSurface command converts the following entities 3D surfaces:

- 2D solids
- 3D solids
- Arcs with thickness
- Lines with thickness
- Meshes
- Open polylines with thickness but of zero width
- Planar 3D faces
- Regions

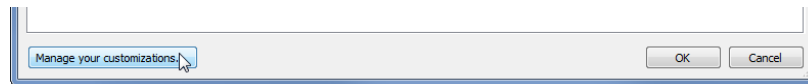
TIP The **DelObj** variable controls whether entities converted to 3D surfaces are deleted.

CreateThumbnailOnTheFly variable toggles whether thumbnail preview images are generated for previews, such as in the Open dialog box (see figure below) and File Manager, when drawings lacks them.



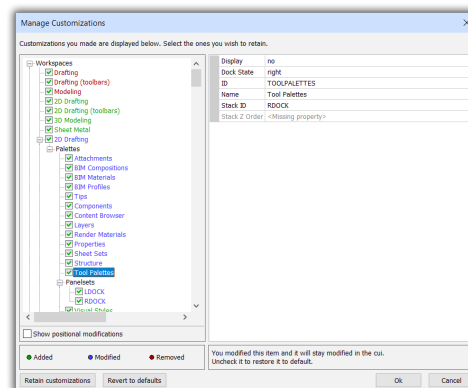
Preview image showing the content of the selected drawing

Customize command now reports on the status of user interface customizations. To access this function, in the Customize dialog box, click the new **Manage your customizations** button. The Manage Customizations dialog box lists the contents sorted by workspace.



New button in Customization dialog box

None of the fields in the right pane can be modified. In the left pane, the colors of the items have the following meaning:



Managing customizations

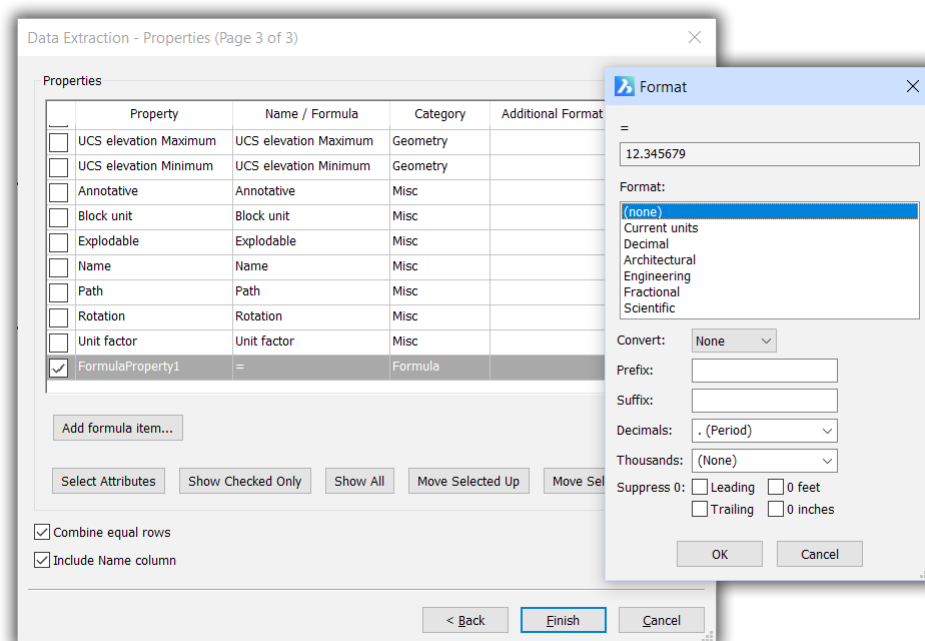
Check mark	Keep modification
No check mark	Revert modifications to the default
Green text	Content added to the customization
Blue text	Content that was changed
Red text	Content that was removed

TIP This command also reverts changes made to IU customizations. This is handy when merging your personal CUI settings with an updated CUI file installed with a BricsCAD update.

VersionCustomizableFiles variable reports the current version of the CUI and PGP files, such as 317 (read-only).

DataExtraction command has a new Formula column to show results from combining values from regular columns, and defines filters in the DXD (data extraction definition) file:

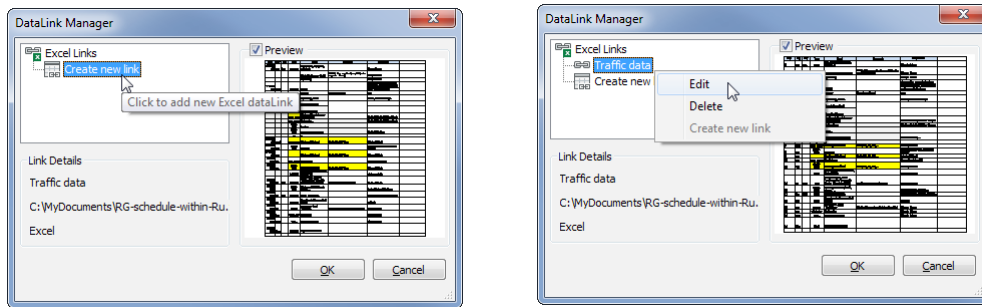
- Drawing properties
- Handle and entity type properties
- Coordinate properties
- Fixed symbol table record properties
- Vertex properties
- BIM ply properties
- Dynamic block properties



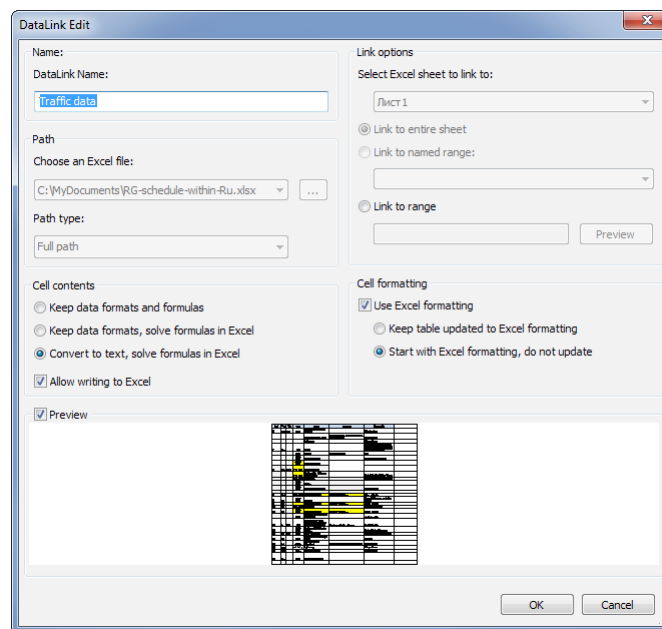
Adding formulae to the data extraction template

Datalink command imports Excel spreadsheet ranges into tables; preserves the link to update the table when the spreadsheet changes.

Warning This command works only when Excel is installed on the computer; it does not work with the view-only version of Excel, nor do Excel substitutes, such as Libre Office.

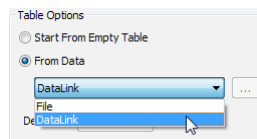


Left: Creating new data links; right: Managing links



Editing the data link

TIPS To link a table with a spreadsheet, start the **Table** command and then choose “Datalink” from the **From Data droplist** in the Table Options section of the Insert Table dialog box.



You can copy'n paste a range of cells from a table to Excel, and vice versa.

DataLinkNotify variable controls data link notifications:

- 0 Disabled; changes to the data link are not reported
- 1 Enabled; changes to the data link are reported
- 2 Enabled; changes to the data link are reported with a balloon (default)

DatalinkUpdate command updates links in drawings with Excel spreadsheets:

: DATALINKUPDATE

Select an option[Update data link / Write data link] <Update data link>: (Press Enter)

Select objects or [Update all]: (Press Enter)

Dim command is rewritten to apply dimensions based on the entity selected, such as radial dimensioning of circles. Select a dimension to continue dimensioning in continuous or baseline mode;. The updated command also works with sub-entities and dimensions through layout viewports.

Here is the full prompt line, which includes the new **DISTribute** and **LAYER** options:

: DIM

Dimensioning command [HORizontal/VERTical/Aligned/ANGular/Leader/OBlique/ROTated/CENter/
Diameter/RAdius/Baseline/CContinue/ORdinate/Position/DISTribute/UPdate dimensions/variable
SStatus/OVerride/SEttings.../LAYER]:

When the cursor passes over a circle, for instance, the prompt changes to the following:

Select arc or circle to specify diameter or [Radial/Angular]: (Pick a circle)

Location of dimension line [Angle/Text]: (Pick a point)

Dimension text: = 5.0474

Most of the terse option names are self-explanatory; here is the meaning of the option names I find vague:

Position repositions the dimension text (and associated leader, if any)

DISTribute spaces dimensions evenly or by an offset distance

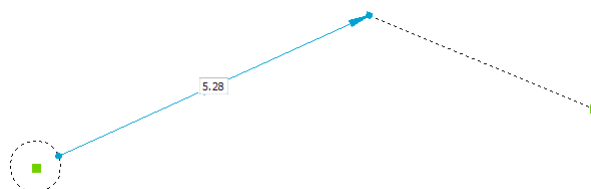
SEttings displays the Drawing Explorer window with dimension styles

LAYER specifies the name of the layer on which to place the dimension

DimLayer variable specifies the existing layer on which to place dimensions; creates the 'dimlayer' (for dimensions), 'centerlayer' (for center marks), and 'hplayer' (for hatch patterns) layers when you type in the names, but they do not already exist in the drawing. Default is the current layer (.).

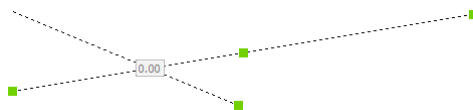
Dimension style and text style controls now change the style of dimension and text entities in the current selection set.

NearestDistance variable toggles a dynamic dimension that reports the nearest distance between two selected entities. The distance reported is the shortest distance. (This function does not work when three or more entities are selected.)



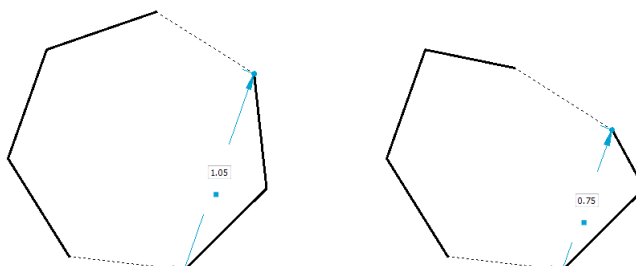
A dynamic dimension showing the distance between two selected entities

When two entities intersect or connect, the distance is zero; the distance cannot be changed.



*Zero distance between intersecting lines;
gray field cannot be edited*

Hold down the **Ctrl** to select sub-entities, such as two of the polyline segments that make up a rectangle.



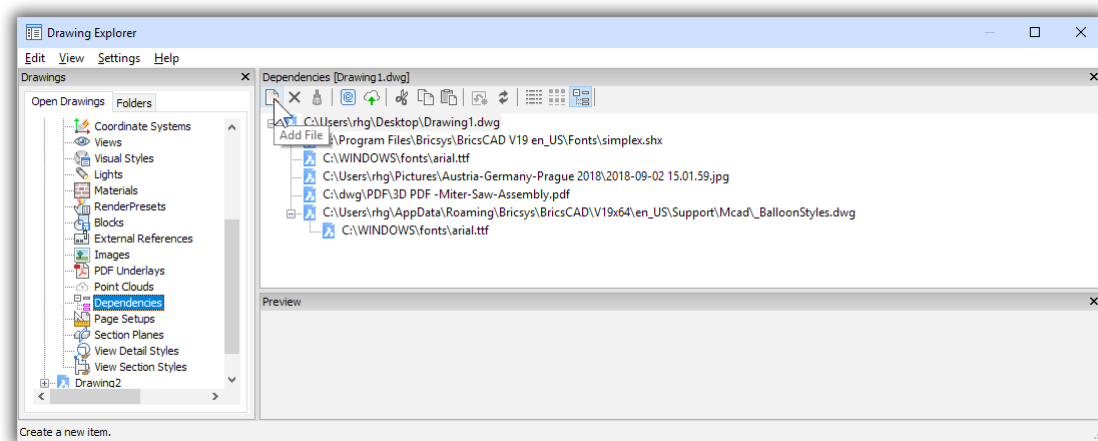
*Left: Shortest distance between two segments of a polygon;
right: editing the distance between them*

TIP You can edit the distance to move the last selected entity; the arrowhead points at it. When an entity cannot be moved, the dimension field is read-only (not editable).

Entity snaps now snap to the geometry of **AcDbSubDMesh** mesh entities and the content of mleader blocks. **Tangent** and **Perpendicular** entity snaps work with grips to edit coincident entities.

TIP With **Tangent** and **Perpendicular** entity snaps, you can select the opposite end of a line that's coincident with an arc. As you move the entity close to being perpendicular or tangent, it snaps into place.

eTransmit command now adds files to transmittal packages through the **Add File** button. (In previous versions, the Add File button was grayed out and unavailable.)



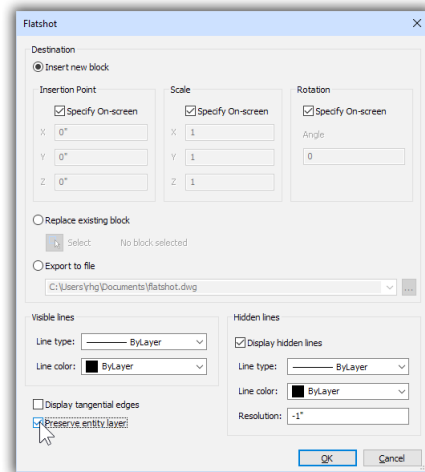
The Add File button works for transmittals in V19

ExportLayout command no longer explodes hatches laying on the view boundary.

TIP Unlike other dwg editors, V19 preserves the hatch and trims its boundary to match the viewport boundary. Preserving the hatch avoids creation of potentially thousands of lines for the dashes.

Field command supports multi-part formulas, such as (Table(261).Sum(A2:A3)+I+Table(261).A2).

Flatshot command adds a **Preserve entity layer** check box in its dialog box to preserve the layer settings of entities in hidden-line perspective drawings. The command now accepts PolyfaceMeshes and 3dFaces as 3D entities that can be flattened.



Preserve entity layer option added to Flatshot dialog box

Gradient command gets variables to control the look of gradient hatches; see list below.

GfAng variable specifies the angle of a gradient; range is 0 to 360 degrees.

GfClr1 variable specifies the first color gradients; any color designation can be entered.

GfClr2 variable specifies the second color gradients; any color designation can be entered.

GfClrLum variable specifies the level of shade level in one-color gradients; range is between 0 (black) to 1 (white).

GfClrState variable determines whether the gradient uses one or two colors:

- 0 Two-color gradient (default)
- 1 One-color gradient

GfName variable determines the look of the gradient:

- 1 Linear (default)
- 2 Cylinder
- 3 Inverted cylinder
- 4 Sphere
- 5 Hemisphere
- 6 Curve
- 7 Reverse sphere
- 8 Reverse hemisphere
- 9 Reverse curve

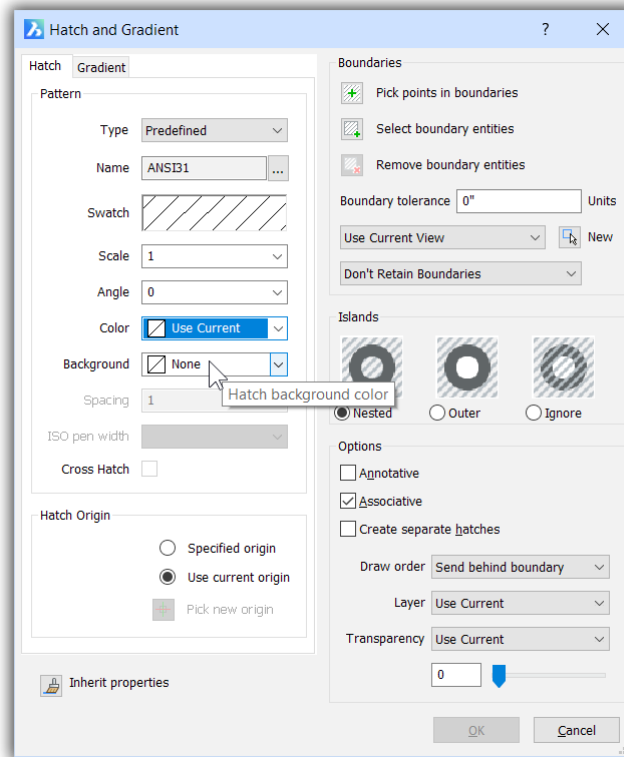
GfShift variable toggles the center of the gradient:

- 0 Center
- 1 Upper left corner

Grading command shapes terrains for building sites, such as building pads, retaining walls, parking lots, and streets.

Grid command now create rectangular and radial grids with automatic labels.

Hatch command operates as much as 100x faster for hatches with boundaries containing thousands of segments. Also, it gets new color and background colors in the Pattern section of the dialog box, along with matching **HpColor** and **HpBackgroundColor** variables.



TIP **HpStyle** variable is renamed **HpIslandDetection**.

HpBackgroundColor variable specifies the background color for new hatch patterns; any color designation can be entered.

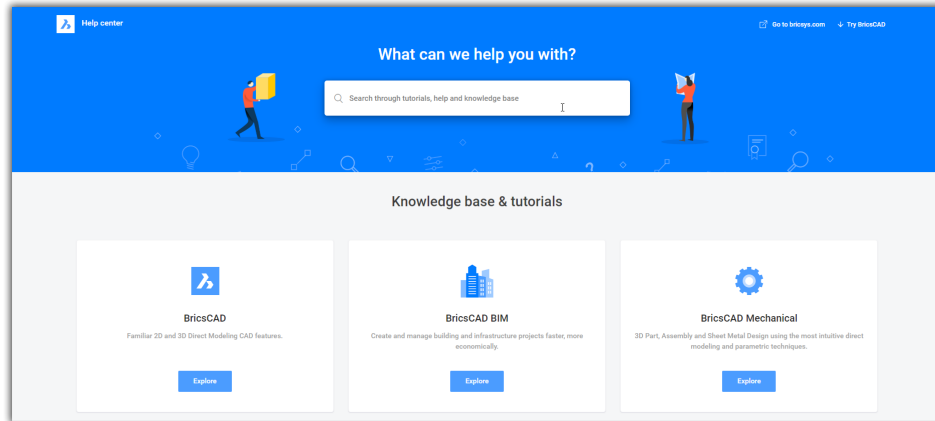
HpColor variable specifies the color of new hatch patterns; any color designation can be entered.

HpIslandDetection variable determines how islands are handled as hatch boundaries:

- 0 Normal -- island areas are hatched and not-hatched in alternating order (default)
- 1 Outer -- only the outermost area is hatched
- 2 Ignore -- everything is hatched within boundaries

HpStyle variable is removed from V19 and replaced by **HpIslandDetection**.

Help command offers BricsCAD documentation online at <https://help.bricsys.com/hc/en-us>.



New look of the online help for Bricsys products

InsUnitsScaling variable controls the INSUNITS variable:

- 1 Enable InsUnits-based scaling flag
- 2 Disable InsUnits, and instead use paper size unit for paper space insertions

LConnect command creates a connection between two faces of 3D solids:

: **LCONNECT**

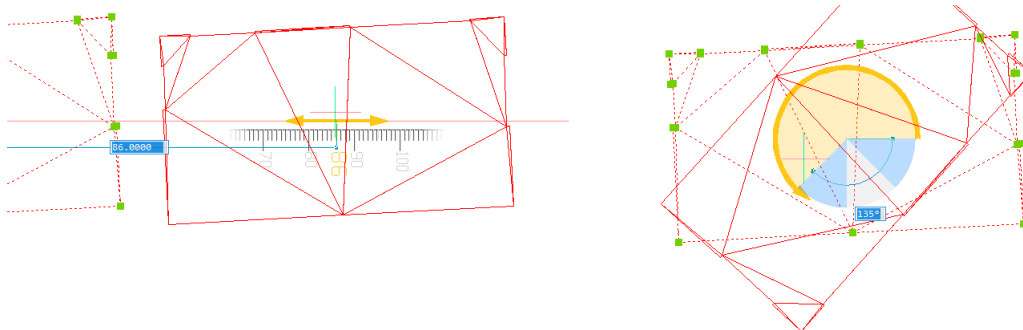
Select solids or faces to connect:

Select solids or faces to connect:

MaterialAssign command applies the material specified by the **CMaterial** variable to the selected 3D entities. Alternatively, you can drag a material from the Material Browser panel onto the entities.

TIP To change the material during the **MaterialAssign** command, hold down the **Alt** key while selecting the entity.

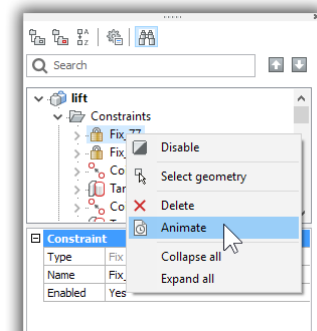
Manipulator command adds a ruler for distances, and a protractor for angles. The size of the markings change dynamically according to zoom level. After making a copy, the copied entities now become the selected ones.



Left: Manipulator with ruler for distances; right: with protractor for angles

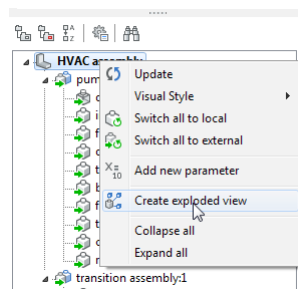
Also, the manipulator is added to many more commands such as Drag and PushPull. The manipulator also appears when you press **Ctrl+A** to select all objects in the drawing.

MechanicalBrowserOpen command can search for nodes by name through the new **Show Search** button.



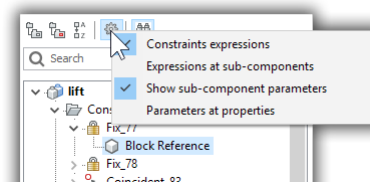
New Animate option in the context menu, and Search button on the toolbar

When right-clicking parameters and 3D constraints, the context menu lists the new **Animate** option (see above). to run a short animation that shows the geometry controlled by the selected item The topmost node gets a new **Create Exploded View** option (see below); it runs the bm-Explode command.



Newly added Create Exploded View option

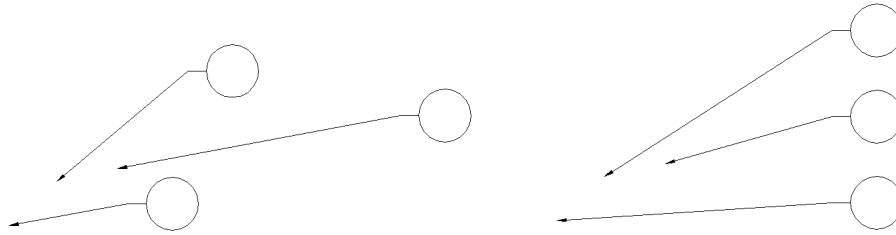
The new **Settings** menu (see figure below) controls the visibility of parameters of sub-components.



Settings button displaying options in Mechanical browser

Searches in the Mechanical Browser are activated by pressing the **Show Search** button on toolbar panel, or by pressing **Ctrl+F**, or by just typing some text while the focus is inside the browser.

MLeaderAlign command aligns mleaders:



Left: Mleaders before... ; **right:** ...and after being aligned

```
: MLEADERALIGN
Select multileaders: all
Entities in set: 3
Select multileaders: (Press Enter to continue)
Specify first point or [Options]: (Pick a point)
Specify second point: (Pick a second point)
```

Here is the meaning of the MLeaderAlign command's options:

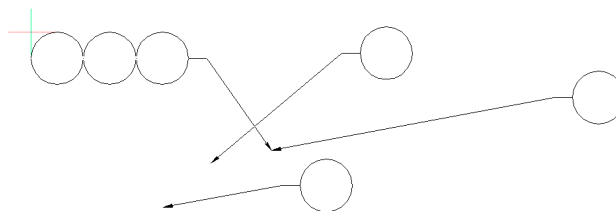
byMleader	Aligns other mleaders to the selected mleader
onpolyLine	Arranges mleaders' contents along a path whose points you pick
Parallel	Arranges mleader lines parallel to the selected one
Spacing	Spaces mleaders the specified distance from the selected one
Circle	Arranges mleaders in a circle around a specified center point

TIP This command can be used repeatedly to keep changing the alignment of the group of mleaders.

MLeaderCollect command collects two or more multi-leaders *with blocks* and then combines all leader lines into a single one, with the blocks aligned vertically or horizontally, or wrapped to a specified width:

```
: MLEADERCOLLECT
Select entities: all
Entities in set: 3
Select entities: (Press Enter to continue)
Specify collected multileader location or [Vertical/Horizontal/Wrap/Sorting/Collapsing]:
(Pick a point)
```

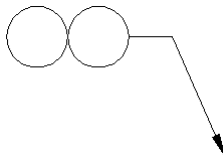
TIP This command works only with mleaders that have blocks as their annotation. Blocks can be sorted in ascending and descending order. The command cannot be reused once applied to a group of mleaders.



Three mleaders (at right) being collected into a single one (at left)

Here is the meaning of the MLeaderCollect command's options:

Vertical	Stacks blocks vertically
Horizontal	Arranges blocks horizontally
Wrap	Limits the maximum number of blocks in a user-specified width
Sorting	Sorts the blocks in ascending or descending order
Collapsing	Joins selected leaders into a single leader, preserving blocks



Two mleaders collapsed into one

MText command supports the NODe entity snap.

NODe command now supports mtext entities

Nudge moves selected entities by small increments:

X direction	Hold down Ctrl key and press Left and Right arrow keys
Y direction	Hold down Ctrl key and press Up and Down arrow keys
Z direction	Hold down Shift and press Up and Down arrow keys

NUDGE: offset along the X axis of the UCS: 5/128"

NUDGE: offset along the Y axis of the UCS: 5/128"

NUDGE: offset along the Z axis of the UCS: 5/128"

TIP The step size is specified by the adaptive grid-snap resolution: the further out the drawing is zoomed, the bigger the nudge distance.

Number command draws incremented number tags.

: NUMBER

Select entities to number and press Enter **all**

Entities in set: 6

Select entities to number and press Enter (*Press Enter to continue*)

Current settings: First index = 1, Increment = 1, Prefix = "", Suffix = "", Number Style = Arabic, Entities Sorting = None, Existing Numbers are <overwritten>

Provide First index or [Increment/Prefix/Suffix/Number style/Entities sorting/Overwrite Numbers] <1>:

Here is the meaning of the Number command's options:

First index	Value with which numbering should start; default = 1
Increment	Value by which numbers increment; default = 1
Prefix	Prefix for numbers; default = "" (none)
Suffix	Suffix for numbers; default = "" (none)

Number style	Specifies the type of numbering:
0	Arabic, such as 1, 2, 3
1	Roman uppercase, such as I, II, III
2	Roman lowercase, such as i, ii, iii
3	Letters uppercase, such as A, B, C
4	Letters lowercase, such as a, b, c
Entities sorting	Sorts the entities by z, y, z, coordinates
Overwrite numbers	Determines if numbers are kept or overwritten

OrbitAutoTarget variable controls the target point for real-time view rotations:

- 0 Target point is where you click to start orbiting
- 1 Target point is at the center of all entities seen on the screen, or of selected entities (default)

Pan command is 2x faster for drawings containing a lot of tiny geometry; when using anti-aliased mode (**AntiAliasScreen** > 1) the performance improvement is 5x.

Panelize command draws free form surfaces as subdivision meshes, collected into a block.

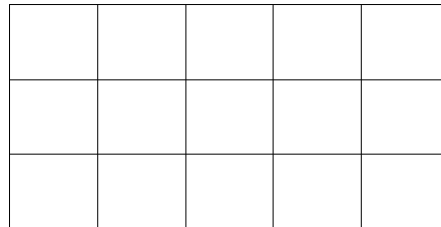
: **PANELIZE**

Select face or set [Result]:

Create grid or [Length U panels/Length V panels/Number U panels/Number V panels]:

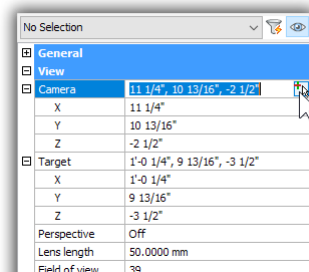
Max deviation from planarity: 0.000000

The **Result** option determines if the grid is made of polylines or meshes.



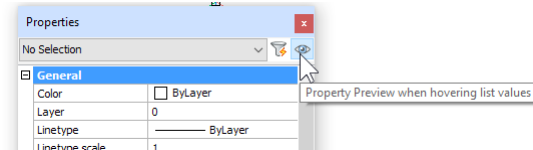
Face panelized by polylines

Properties command's panel changes the **Camera** and **Target** properties from read-only to editable.



Editable Camera and Target fields in the Properties panel

The new Eye icon toggles whether previews of selected entities are displayed.



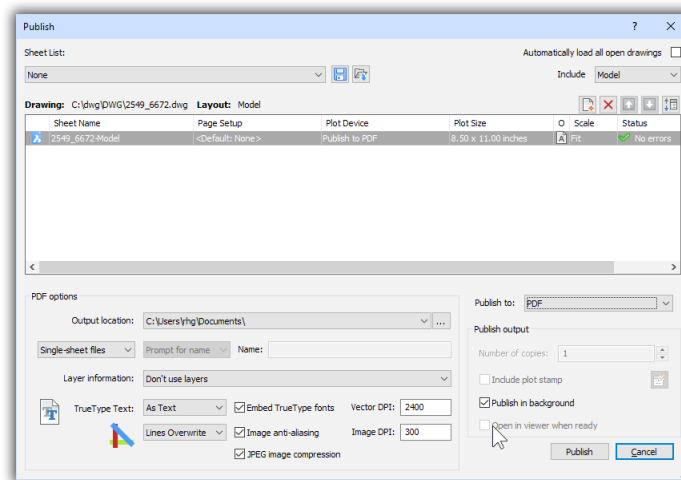
The Eye icon

PropertyPreview variable toggles the display of previews of selected entities. This variable can be toggled by the new eye button in the Properties panel, as shown above.

PropertyPreviewDelay variable specifies a delay before PropertyPreview starts; range is 100 to 10000 milliseconds; default is 500..

PropertyPreviewObjLimit variable specifies the maximum number of entities used by PropertyPreview; range is 1 to 30,000; default is 500.

Publish command gains a **Open in viewer when ready** check box for PDF files; PDF files are opened after publishing when 'Publish to' is set to PFD, and **Publish in background** is off.



New 'Open in viewer when ready' option in Publish dialog box

Purge command now purges the following unused tables from drawings, but still no dialog box!

- Detail view styles
- Groups
- Multileader styles
- Section view styles
- Shapes
- Zero-length geometry

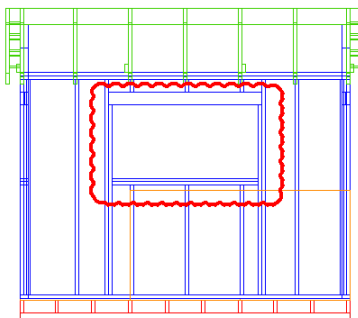
: PURGE

Purge [Batch all/purge All/Blocks/**DEtail view styles**/Dimension styles/**Groups**/LAYers/Line-Types/Materials/MLine styles/**MUltileader styles**/Plot styles/Regapps/**SEction view styles**/**SHapes**/Table styles/text STyles/Visual styles/**Zero-length geometry**/Empty text entities/Orphaned data]: (Enter an option)

RevCloud command adds **Rectangular** and **Polygonal** options for shapes of revision clouds:

: REVCLLOUD

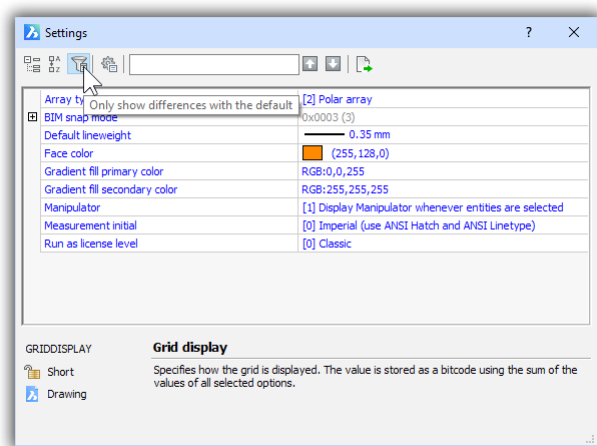
Specify first corner point or [Arc length/Entity/**Rectangular**/**Polygonal**/Freehand/Style] <Entity>:



Rectangular revision cloud

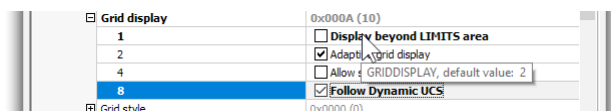
SectionPlaneToBlock command now accepts PolyfaceMeshes and 3dFaces as valid types.

Settings command's dialog box now shows non-default values in a user-definable color; click the Filter button:



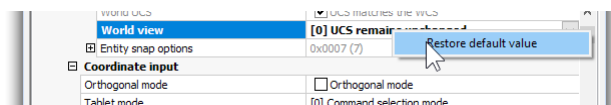
Settings dialog box showing only those values that differ from the defaults

A tooltip reports the default value when hovering over a setting:



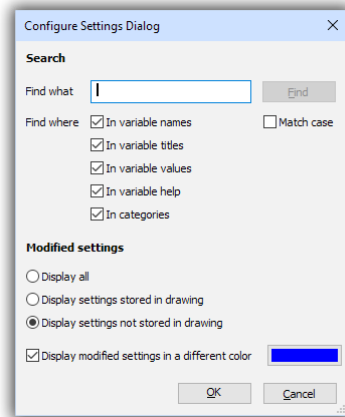
Tooltip reporting the default value of a setting

An option in the right-click menu resets the value to its default value:



Right-clicking a changed value to reach the restore option

The new **Configuration** button determines how settings are displayed:



Configuring the Settings dialog box

The buttons for jumping directly to Drawings, Dimensions, and Program sections were removed from V19

Site command imports points files in CSV (comma-separated values) format, drawing entities or Civil 3D surfaces to create terrain models.

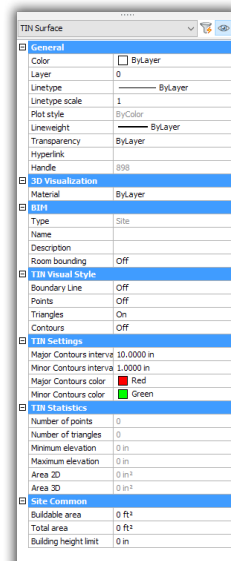
: SITE

Select entities to create site or [Import from file/Place points/create from civil 3d surface]: *(Enter an option)*

Here is the meaning of the Site command's options:

Entities	Use entities already in the drawing
Import from file	Import data from a points file, saved in CSV format
Place points	Pick points in the drawing
create from civil 3d surface	Use an existing Civil 3D surface already in the drawing

V19 works with a new entity called "TIN Surface"; TIN is short for triangular irregular network.



Properties for the new TIN entity

SiteEdit command modifies terrains by adding/removing boundaries, breaklines, and points.

SheetSet command switched creating sheet list tables from plain text to fields and hyperlinks, which refer to sheet properties..

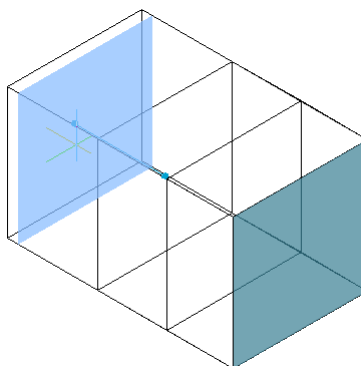
TIP Because it uses fields and hyperlinks, the sheet list now automatically updates when the properties of sheets are changed.

SnapType variable adds adaptive snap step size option (2) to grids.

TIP The grid display and the snap distance change according to the current zoom factor. This adaptive snap step is also used by the new Manipulator ruler and new **Nudge** command

Slice command gains the **Multislice** option.

```
: SLICE
Select entities: (Select one or more 3D solid entities)
Entities in set: 1
Select entities: (Press Enter to continue)
Specify first point on slicing plane or [Object/Surface/Zaxis/View/XY/YZ/ZX/Line-
point/3points/Multislice] <3points>: m
Select a planar Face or a planar Surface or <XY>: (Pick a face)
Specify distance to create slice: (Pick one or more slice locations)
Specify distance to create slice or Repeat [Repeat]: (Press Enter to exit)
```



Making multiple slices

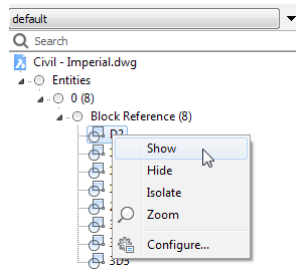
Spell command now works with fields in texts, mtexts, leaders, mleaders, tables, and block attributes.

DctCust variable now accepts just a name, or a name and path for the custom dictionary; if the dictionary can't be found, then a new one is created.

Spline command gains the **Undo** option to back up through pick points:

```
: SPLINE
First point for spline: (Pick a point)
Second point: (Pick another point)
Set next point or [Close/Fit tolerance/Undo]: (Pick another point)
```

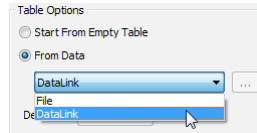
StructurePanel command gets **Show**, **Hide**, and **Isolate** options in its right-click-menu; also get a new text search field.



New options in the context menu

TIP The Structure panel allows you to select specific entities to change visibility, unlike the **Unhide** and **Unisolate** commands, which make all hidden entities visible. Hidden entities are shown in gray text.

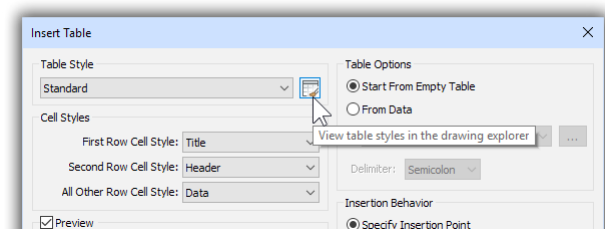
Table command now links with Excel spreadsheet files; see the **DataLink** command.



Starting the link between table and spreadsheet

TIP Changing the spreadsheet changes the content of the table, and vice versa.

The dialog box's user interface gains tooltips.



Tooltips explaining the meaning of elements in the Insert Table dialog box

TConnect command connects planar faces and linear solids to other solids and faces. (See also **LConnect** command.)

: **TCONNECT**

Select face to connect:

Select face to connect:

Select entities to connect to or [Connect to nearest] <Connect to nearest>:

TIP Press the **Tab** key to select the face you want.

TextEd variable's default value is changed to 2 so that the in-place editor is used for single line text.

TK (tracking) now allows any number of tracking segments, and is no longer limited to 7.

Tolerance command-created entities now launch the **DdEdit** command — instead of the **Properties** command — when a tolerance is double-clicked.

ToolPalettes command can now configure dynamic block properties inserted from the Tool palette panel.

VisualStyles command now displays a warning when a visual style is edited in a way that is incompatible with AutoCAD.

Zoom command is 2x faster for drawings containing a lot of tiny geometry; when using anti-aliased mode (**AntiAliasScreen** > 1) the performance improvement is 5x.

WHAT'S NEW IN ASSEMBLIES

(dm- commands; requires a Platinum license)

dmAngle3d command now creates Planar Angle constraint by default, using coordinate planes of the WCS as the third reference entity when possible.

dmAudit command now checks and heals blocks, sliver faces, and coincident faces.

dmAuditAll command extends the **dmAudit** command to check for, and heal, flaws in drawings inserted as external references.

```
: DMAUDITALL
Select entities to audit [Entire model] <Entire model>: (Press Enter)
Choose action [Check/Fix/Settings] <Fix>: (Enter an option)
Selected count: 4
===== Block "Model space" =====
----- Solid -----
      Handle: 79
      Name in Mechanical Browser: <unknown>
      Flaws: None
----- Skipped: -----
      1 Spline
No flaws were found.
===== Before fix =====
Flaws found in this drawing: 0
===== After fix =====
Flaws found in this drawing: 0
```

The meaning of the dmAuditAll command's options are the same as for the dmAudit command:

dmConstraint3d command now manages 3D constraints and edits their properties, as shown by the new prompt line below.

```
: DMCONSTRAINT3D
Enter option [New/Edit/Rename/Delete/?] <?>: new
Specify 3D constraint type [Fix/Coincident/CONcentric/Parallel/PErpendicular/Tangent/RIgid-
set/Distance/Radius/Angle]:
```

Here is the meaning of the dmConstraint3d command's new options:

New	Displays the list of 3D constraints that can be applied
Edit	Prompts to enter the constraint's name to be edited
Rename	Prompts to enter the constraint's name to be renamed
Delete	Prompts to enter the constraint's name to be deleted
?	Displays help for this command

The **3Ddistance** and **3Dcoincident** constraints now support point-cylinder and point-sphere combinations.

3D constraints now display widgets when a constrained 3D solid face is selected, and receive “Directions” and “Placement” properties. The widget for the **3Ddistance** constraint displays a dimension.

3Dnearest now snaps to edges of ACIS entities.

TIP Mechanical Browser can be used to edit the new properties of 3D constraints.

dmExtrudeMode variable gets new flag, **4**, to prevent intersecting 3D solids from being modified (as created by the dmExtrude command):

4 Set on to not modify solids which intersect with created volume (off, by default)

dmPushPull command makes it easier to choose a reference face: hover the cursor over a reference face, and then press **Tab** — or **Shift+Tab** — to select the opposite parallel face.

dmSimplifyAll command extends the **dmSimplify** command to simplify geometry in drawings inserted as external references.

WHAT'S NEW IN BUILDING INFORMATION MODELING

(bim- commands require an extra-cost module)

The BIM workspace has a new user interface that mimics the Shape program, with a toolbar made of large icons and tabbed dock panels. BIM-related panels are re-designed to match the look of Shape.

V19 now classifies elements according to any national or company standard classification system.

Linear building elements now support the same grip-stretch operations as with lines.

Parametrize automatically adds constraints and parameters to pure geometric models of BIM components.

Reflected ceiling plans are 2D sections showing elements on the ceilings of rooms and other spaces.

When a connected structural element is rotated by 90°, the connection is (optionally) restored.

bimApplyProfile gains the [convert solids to Line](#) option to convert linear solids (straight solids) to lines (along their axes:

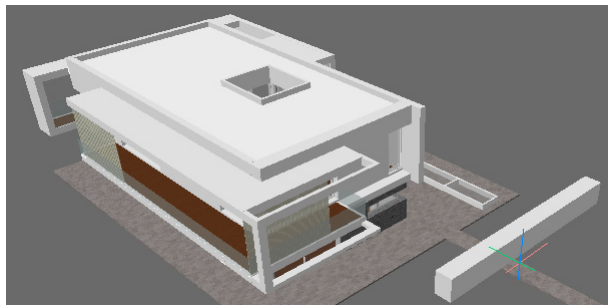
```
: BIMAPPLYPROFILE
Select path(s): (Choose a Linear solid)
Entities in set: 1
Select path(s):
Select profile [in Dialog/convert solids to Line] <in Dialog>: 1
```

bimAttachComposition command now allows you to choose reference and opposite surfaces manually with the new [Entity](#) option.

```
: BIMATTACHCOMPOSITION
Select entities to attach composition: (Pick an element)
Entities in set: 1
Select entities to attach composition: (Press Enter to continue)
Enter composition name or [Dialog/Entity] <Dialog>: e
Select other entity to get composition: (Select an entity)
```

[bimCopy](#) command works like the **bimDrag** command for moving entities normal (at 90 degrees) to the selected face, but this command makes a copy.

```
: BIMCOPY
Select several entities/subentities: (Select one or more entities)
Entities in set: 1
Select several entities/subentities: (Press Enter to continue)
Specify distance to create copy: (Move the cursor or enter a value)
Specify distance to create copy or switch to [Copy/Repeat/Accept] <Accept>: (Press Enter)
```



White beam being copied by the bimCopy command

bimCreateComponent command creates components from selection sets or the drawing, and then adds them to the library.

bimCurtainWall command creates curtain walls made of planar quadrilateral panels from free form surfaces.

: BIMCURTAINWALL

Select a face: *(Select a face)*

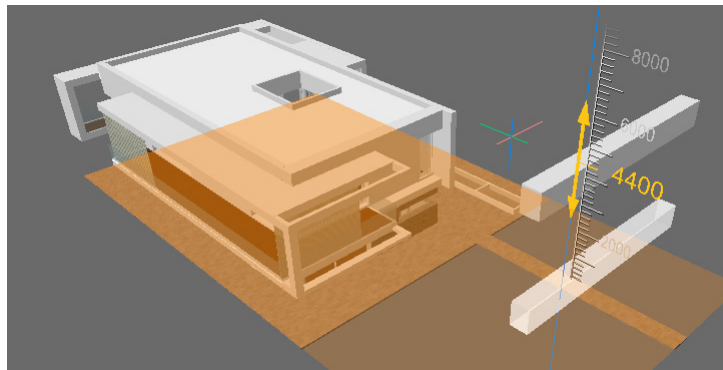
bimDisplayComposition variable toggles the display of compositions on and off.

: BIMDISPLAYCOMPOSITION

Display composition: [Toggle/On/off]: *(Enter an option)*

Select entities to display the composition <Drawing>: *(Select one or more entities, or press Enter)*

bimDrag command now moves the entire 3D solid when dragging a face; it maintains connectivity between non-orthogonal walls; and works with the Manipulator.



bimDrag displaying the ruler from the Manipulator

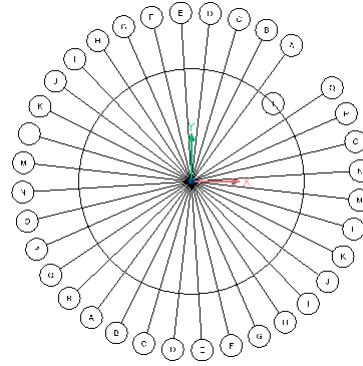
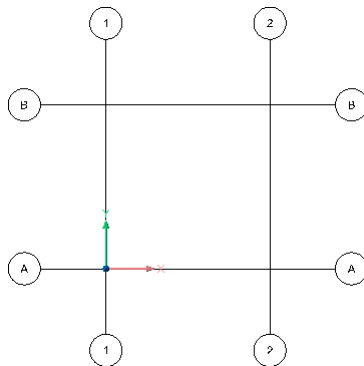
TIP Hold down the **Ctrl** key to turn off connectivity. Use **dmPushPull** to move the face of a 3D solid.

bimGrid command creates rectangular and radial grids with automatic labels.

: BIMGRID

Enter first point [offsetU/offsetV/offset Labels/Radial]: **0,0**

Enter second point [offsetU/offsetV/offset Labels]: *(Pick a point)*

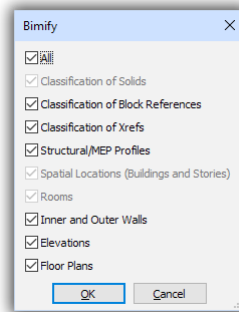


Left: Rectangular grid... ; right: ...and radial grid

Here is the meaning of the bimGrid command's options:

First point	Specifies the starting point of the grid
Second point	Specifies the opposite corner for a rectangular grid
offsetU, offsetV	Specifies the distance between u and v grid axes
offset Labels	Specifies the offset distance for labels
Radial	Switches to the radial grid

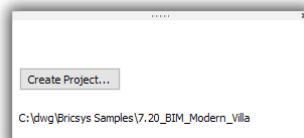
bimIfy command now detects flow segments and fittings, depending on the drawing type (such as architectural, structural, or MEP); classifies block references automatically; detects rooms and outer walls automatically; and adds a dialog box to select entities for a partial bim-ification.



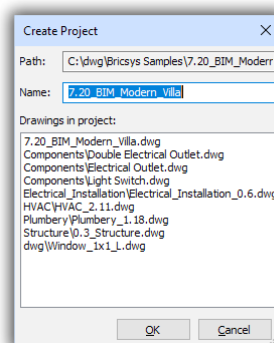
New dialog box added to the bimIfy command

BIM Project Browser panel generates and navigates between models, sheets, and schedules in the BIM project. When no project has been created for the current BIM drawing, follow these steps:

1. Click the **Create Project** button.

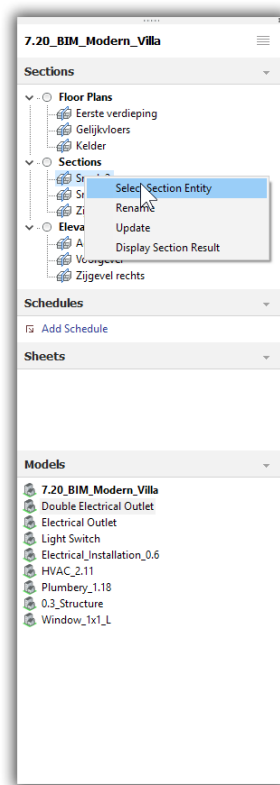


2. Fill in information about the project. If you are in a hurry, just click **OK**.



Creating a new project;

TIP There is no command to open this panel. Instead, right-click the ribbon, and then choose **BIM Project Browser** from the context menu.



BIM Project Browser panel

bimProperties command now imports custom properties from XML files.

bimPropagate command (replaces **bimSuggest**) maps any detail (such as solids, holes, and finishing geometry) from certain base solids to all similar base solids, as well as on a grid.

: **BIMPROPAGATE**

Select base solids. *(Select one or more solids)*

Entities in set: 1

Select base solids. *(Press Enter)*

Select detail objects or detail sub entities (optional).

Four geometry-specific versions of this command are available:

bimPropagateEdges command propagates along the edges of planar solids, such as with railings, gutters, borders, and wall caps.

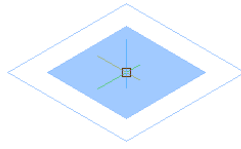
bimPropagateLinear command propagates connections to linear elements, such as beams, columns, pipes, ducts, and connections to walls and slabs.

bimPropagatePattern command propagates a single element on flat surfaces to multiple locations and grids, such as with lights, light switches, windows, and air diffusers.

bimPropagatePlanar command propagates connections to planar elements, such as walls, slabs, and roofs.

bimQuickDraw command draws rectangles and L-shapes with height for conceptually designing buildings and room layouts.

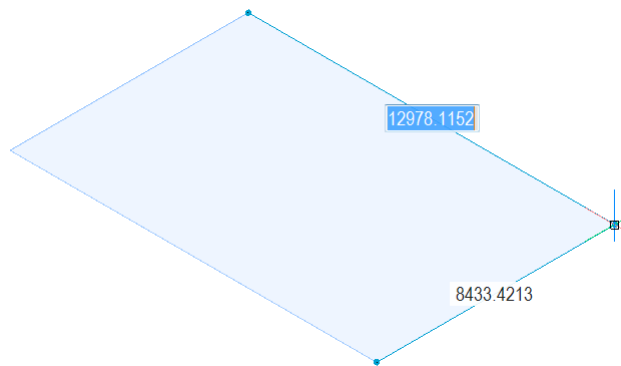
1. When you start the command, you see this initial square:



Starting the bimQuickDraw command

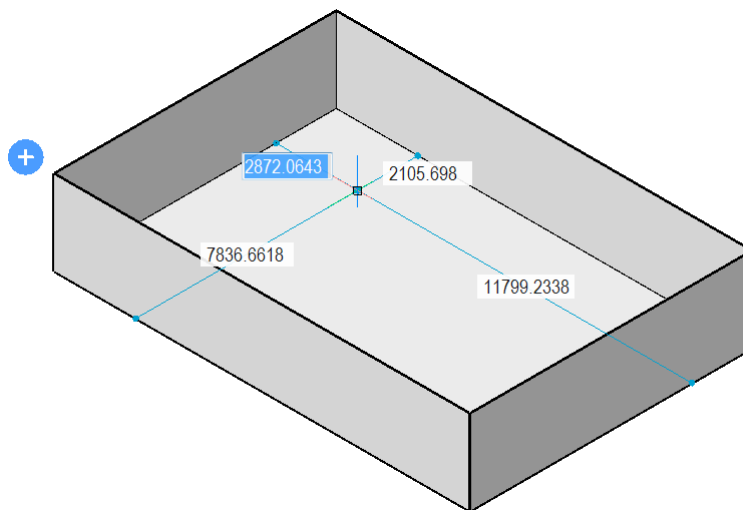
The blue square represents the floor area, the white outline are the walls.

2. As you move the cursor, the square elongates:



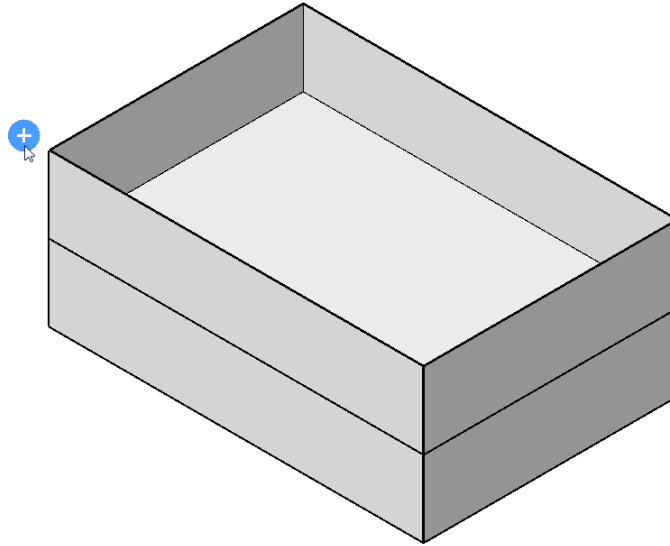
Moving the cursor to set the size of the room

3. When you click a point to indicate the opposite corner (and the size of the floor), walls appear. The thickness and height of the walls are fixed at 1/4" (5mm) wide and 10' (3m) tall.



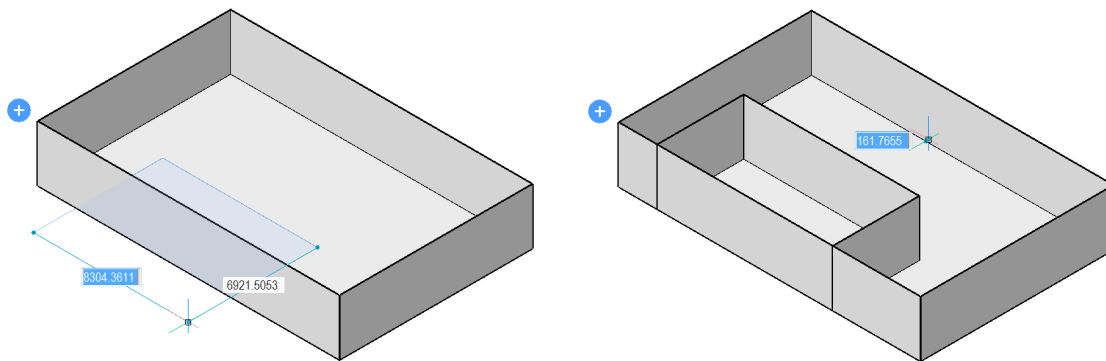
Defining a room with two points

4. Click the blue + to add stories to the floor. You cannot subtract stories while this command is active.



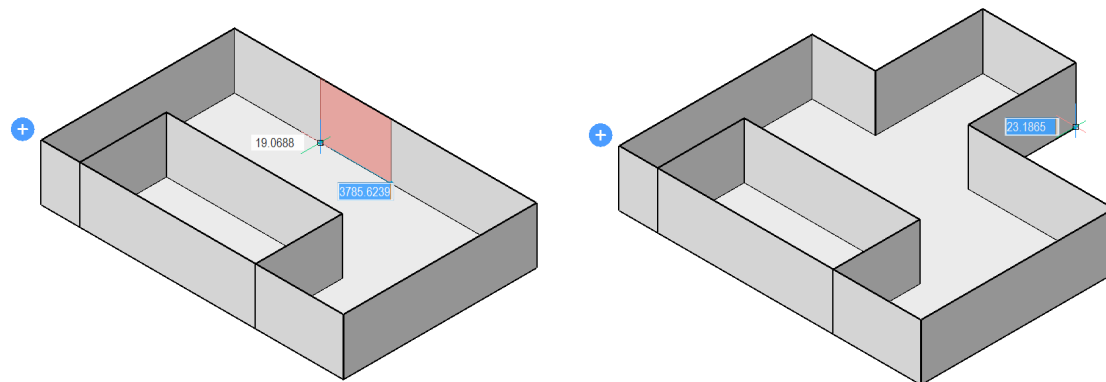
Adding stories by clicking the + button

5. Draw additional attached rooms by starting next to the existing ones.



Left: Drawing more rooms; **right:** ...attached to the first one

6. Cut out a wall by clicking at the base of a wall. The red section hints at the area that will be cut out.

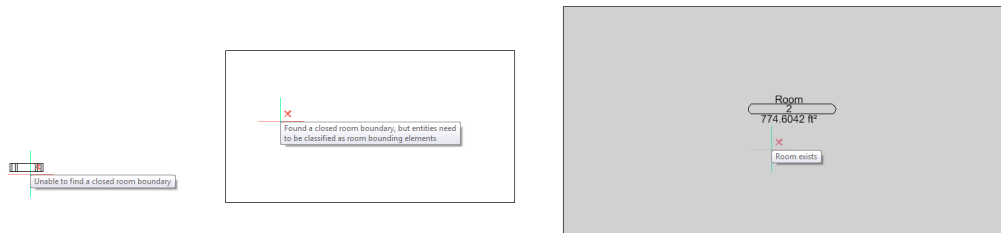


Left: Starting to cut a wall; **right:** ...and adding an L

TIP During the **bimQuickDraw** command, hold down the **Alt** key to move around the model.

bimRepositionWindow command removed from V19 and replaced by editable dimensions to neighbouring edges.

bimRoom command now displays an icon at the cursor to indicate whether a room can be created at the cursor position; if not, the tooltip explains why.



Left to right: No room detected; rectangle not yet a room element; room detected

Rooms can now be calculated on the center lines of walls, curtain walls and columns, and configured for each of bounding element individually.

bimSchedule command generates schedule tables after analyze building elements in BIM models; schedules update when the 3D model changes.

bimSection command now creates reflected ceilings with the new **Reflected ceiling** option:

```
: BIMSECTION
Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]: r
Select a point to place section: (Pick a point)
Specify distance: (Pick a point)
```

bimSectionUpdate command now generates grid curves and labels on 2D drawings; boundary lines of envelope solids; and story indicator lines and symbols showing story names and elevations.

bimSetReferenceFace command controls the layout of plies by selecting a reference and an opposing face to control ply layout; the reference and opposing face can be non-parallel when the composition has variable thickness.

```
: BIMSETREFERENCEFACE
Select reference face: (Pick a face)
Opposing face(s) are detected [Select manually] <Accept>: (Press Enter)
```

TIP For non-parallel faces, first the fixed-thickness plies are set out starting from each reference face, and the remainder of the solid, which does not have a variable thickness, is filled by the variable ply.

bimSuggest is removed from V19.

bimTag command reads mappings between BIM types and tag styles from the `_TagTypeToStyle.csv` file; improves automatic placement of tags; supports more properties, including native properties and quantities; and uses a new syntax for tag attribute names 'property category'/'property name'

bimWindowCreate command now works with a grid that defines subdivisions of the window.

WHAT'S NEW IN COMMUNICATOR

(Requires a separate license and is updated independently of BricsCAD.)

BricsCAD V19 is not compatible with Communicator V18, an upgrade to Communicator V19 is required. Communicator launches as a separate process to increase stability of it and BricsCAD.

InsUnitsScaling variable is supported on import and export.

Changes the names of all unacceptable incoming symbols to those that can be stored in the .dwg file format; similarly, changes the names of hidden blocks and complex mechanical structures upon exporting.

ImportProductStructure variable's default value is now set to

2 (mechanical components)	for Platinum licenses
1 (blocks)	for non-Platinum licenses

ExportStructure variable's default value is set to 1 (mechanical structure) by default, but is considered as blocks for non-Platinum licenses.

WHAT'S NEW IN IMPORT & EXPORT COMMANDS

DgnImport command imports DGN (design) files created by Microstation from Bentley Systems into the current drawing.

TIP Microstation uses the word “elements” for entities or objects.

The following system variables determine how the design files are imported:

DgnImp2dClosedBSplineCurveImportMode variable determines how to convert closed 2D b-splines:

0	Convert to spline (default)
1	Convert to region

DgnImp2dEllipseImportMode2D variable determines how to convert ellipses:

0	Convert to ellipse (default)
1	Convert to region

DgnImp2dShapeImportMode variable determines how to convert 2D shapes and 2D complex shapes:

0	Convert to polyline (default)
1	Convert to region
2	Convert to polyface mesh

TIP Microstation uses the words “shape” and “complex shape” for polyline elements. If an element (entity) is filled, then a hatch is created.

DgnImp3dClosedBSplineCurveImportMode variable determines how to convert closed 3D b-spline curves.

0	Convert to spline
1	Convert to region (default)

DgnImp3dEllipseImportMode variable determines how to convert 3D ellipses:

- 0 Convert to ellipse (default)
- 1 Convert to region

DgnImp3dObjectImportMode variable determines how to convert 3D elements:

- 0 Convert to polyface mesh (default)
- 1 Convert to a 3D solid or body

DgnImp3dShapeImportMode variable determines how to convert 3D shapes and 3D complex shapes:

- 0 Convert to polyline
- 1 Convert to region (default)
- 2 Convert to polyface mesh

DgnImpBreakDimensionAssociation variable determines if dimensions lose their associativity upon import:

- 0 Associativity is maintained (default)
- 1 Dimension associations are broken

DgnImpConvertDgnColorIndicesToTrueColors variable determines how Microstation colors are mapped to BricsCAD colors:

- 0 Converts DGN color indices to DWG color indices
- 1 Converts DGN color indices to RGB true colors (default)

TIP Microstation assigns color numbers to different colors from BricsCAD, and so BricsCAD gets the color from the DGN color table and then attempts to match it with a color in the DWG color table. If no match is found, the DGN color is saved as an RGB (true color index) value.

DgnImpConvertEmptyDataFieldsToSpaces variable determines how empty field values are handled:

- 0 Replaced by underscore symbols (_)
- 1 Replaced by space symbols () (default)

DgnImpEraseUnusedResources variable determines how to import unreferenced elements, such as text styles and linetypes:

- 0 Import unreferenced elements (default)
- 1 Erase unreferenced imported items

DgnImpExplodeTextNodes variable determines how to handle text nodes (empty text fields):

- 0 Convert them to multiline text (default)
- 1 Convert them to simple entities, such as text and lines

TIP Microstation uses “text nodes” as empty fields used to reserve space for text that will be added later, and so the length of the text is not yet known.

DgnImpImportActiveModelToModelSpace variable determines how active spaces are handled:

- 0 Import active models to model space first, and then attach the design model from model table
- 1 Import the active model to model space (default)

TIP Microstation uses the phrase “design model” for model space, and “active model” for the current view of a model.

DgnImpImportInvisibleElements variable determines how invisible elements (entities) are handled:

- 0 Skip invisible DGN elements
- 1 Import invisible them as invisible entities (default)

DgnImpImportPaperSpaceModels variable determines how to import sheet models (paper space):

- 0 Import one DGN model to model space only
- 1 Import all DGN sheet models to paper space layouts (default)

TIP Microstation uses the phrase “sheet model” for paper space.

DgnImpImportViewIndex variable determines DGN view settings:

- 0 - 7 Specifies level mask
- 1 View is not defined

TIP Microstation uses the word “level” for layers; a mask hides content in areas or levels.

DgnImpRecomputeDimensionsAfterImport variable determines how to handle dimensions:

- 0 Create DGN-style dimension geometry blocks (default)
- 1 Re-compute all dimensions to create DWG dimension geometry blocks

DgnImpSymbolResourceFiles variable specifies the paths to folders holding DGN and RSC files.

TIP Microstation uses RSC resource files to store fonts, line styles, and so on.

DgnImpXRefImportMode variable determines how to import reference attachments:

- 0 Don't import DGN reference attachments
- 1 Convert attached DGN files to DWG-style xref files
- 2 Create a block definition of the attached DGN file, then create a block reference (default)
- 3 Attach the DGN references as an underlay

TIP Microstation uses the word “cell” for blocks.

Export command now creates 24-bit-per-pixel BMP files instead of 8-bit-per-pixel ones; as well, it records log files in the folder of exported IFC files; beams and columns with library profiles are exported with the profile name.

FbxExport command exports drawings as FBX files (short for “filmbox”):

```
: FBXEXPORT
What entities to export? [Selected/Visible] <Visible>:
Export options. Entities: yes, lights: yes, cameras: yes, materials: yes
What types to export? [Select/All] <Select>:
Export entities? [Yes/No] <Yes>:
Export lights? [Yes/No] <Yes>:
Export cameras? [Yes/No] <Yes>:
Export materials? [Yes/No] <Yes>:
```

Export options. Entities: yes, lights: yes, cameras: yes, materials: yes

How to export textures? [Embedded/Reference to file/Copy of file] <Embedded>:

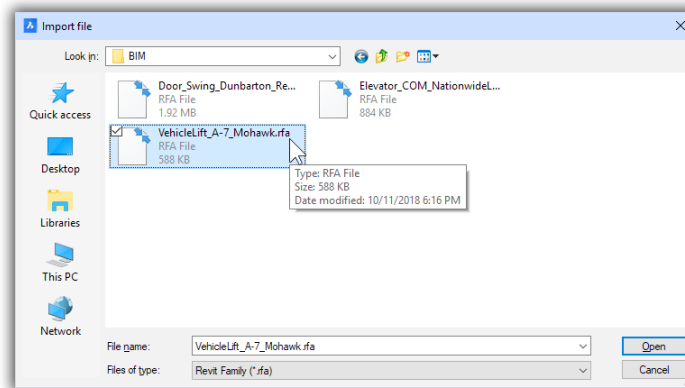
Enter path to export fbx file: [C:\Program Files\Bricsys\BricsCAD V19 en_US\Drawing2.fbx]:

Export of 'C:\Program Files\Bricsys\BricsCAD V19 en_US\Drawing2.fbx' succeeded: 10 entities, 1 lights, 2 cameras, 6 materials

TIP The FBX format was invented by Kaydara and is now maintained by Autodesk. It is used to transport 3D models with light, camera, and material data to rendering software like 3ds Max and Blender.

-FbxExport command does the same thing.

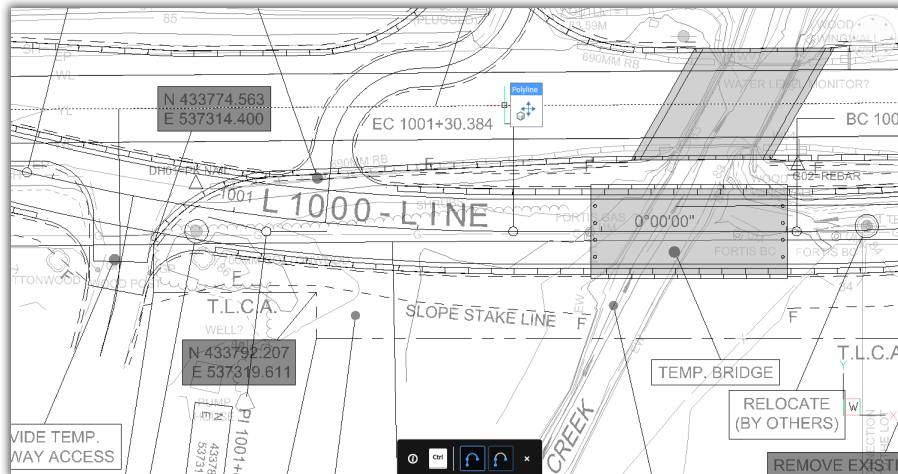
Import command now imports Microstation DGN and Revit Family RFA files into new drawings; IFC profile definitions are now imported to the profile library. (See DgnImport command.)



Import dialog box selecting an RFA file to import

TIP An RFA file holds Revit families, which are like parametric parts but in a Revit format. This command opens RFA files in a new drawing. To insert an RFA file as a component (block) into an existing drawing, use the **bminsert** command.

PDFimport command converts vector data in PDF files to DWG entities, primarily polylines.



PDF imported as polyline entities

: PDFIMPORT

Select pdf underlay or [File] <File>: (Press Enter to access the Select PDF File dialog box)

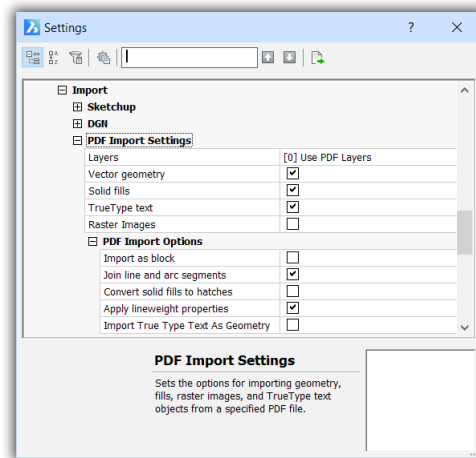
Specify page number to import or list pages [?] to list the pages / Settings] <1>: (Press S for settings, or else enter a page number)

Specify insertion point <0, 0>: (Press Enter)

Specify scale <1.>: (Press Enter)

Specify rotation <0.>: (Press Enter)

TIP You may need to use the **Zoom Extents** command to see the entire imported PDF file.



Settings dialog box for changing PDF import options

PdfImportApplyLineweight variable toggles whether BricsCAD honors the lineweight values in the PDF file:

- 0 Ignore lineweights in the PDF file
- 1 Honor the lineweight values (default)

PdfImportAsBlock variable toggles whether the import PDF is stored as a block:

- 0 Insert PDF file as individual entities (default)
- 1 Insert PDF file as a block

PdfImportConvertSolidsToHatches variable toggles whether to convert areas filled with solid colors in the PDF file to solid-filled hatches:

- 0 Leave solid-filled areas as-is (default)
- 1 Convert solid-filled areas into solid-color hatches

PdfImportJoinLineAndArcSegments variable toggles whether to join individual lines and arcs into polylines:

- 0 Leave lines and arcs as individual segments; entities in PDFs file already joined imported as polylines
- 1 Join lines and arcs into polylines (default)

PdfImportLayersUseType variable determines how layers in PDF files are handled:

- 0 Import layer names from PDF file and create matching layer names in the drawing
- 1 Create layers in the drawing for each entity type, such as (default)
- 2 Place all PDF entities on the current layer in the drawing

TIP BricsCAD creates the following layers to segregate PDF content:

- PDF_Geometry for vector content
- PDF_Images for raster content
- PDF_SolidFills for solid-filled areas
- PDF_Text for TrueType text

PdfImportRasterImages variable toggles whether to import raster images from the PDF file:

- 0 Don't import raster content
- 1 Import raster content and place as a raster image in the drawing

PdfImportSolidFills variable toggles how to handle solid filled areas:

- 0 Ignore solid-filled areas in the PDF file
- 1 Import solid-filled areas and convert them to solid-color hatches (default)

TIP Solid filled areas are automatically given a 50% transparency level by BricsCAD so that underlaying content can be seen.

If the PDF file was generated by an AutoCAD workalike, then solid filled areas include the following:

- 2D solids
 - Arrowheads with width (such as from dimensions and leaders)
 - Hatched areas filled with solid colors
 - Polylines with width
 - Wipeouts
-

PdfImportTrueTypeText variable toggles whether text made from TrueType fonts are imported:

- 0 Does not import text
- 1 Import TrueType text as mtext, and create a text style named after the font (default)

TIP The text style created for imported TrueType fonts has a "PDF" prefix to the name, such as "PDF Arial."

PdfImportTrueTypeTextAsGeometry variable determines how text made of TrueType fonts is imported:

- 0 Convert text to TrueType text in the drawing (default)
- 1 Convert text to equivalent entities in the drawing

PdfImportVectorGeometry variable toggles whether vector geometry is imported:

- 0 Don't import vector geometry
- 1 Import vector geometry from the PDF file and convert them to the nearest DWG entities (default)

TIP Most vector geometry is converted to polylines; geometry that is similar enough to arcs, circles, and ellipses are converted to such. Solid filled areas are converted to solid-filled hatches.

WHAT'S NEW IN GENERATED VIEWS

GenerateAssocViews variable (when on) no longer generates views from entities on layers that are off, hidden, or frozen to improve the performance of **ViewBase** and **bimSectionUpdate** commands; associative data are not set on switched off, hidden or frozen layers.

ResetAssocViews command now removes associative data from nested blocks.

ViewBase command now processes 3dFace and PolyFaceMesh entities, and creates exploded 2D representations of 3D assemblies through the new **Special views** option:

```
: VIEWBASE
Preset: "None", View scale: "Adapt to paper size"
Select objects or [Entire model/preseTs/Special views] <Entire model>: s
Select view [Exploded view/Back] <Back>: e
```

A warning is displayed to emphasize that this command meant for mechanical drawings, not BIM ones; the warning can be disabled.



Warning dialog box from ViewBase

ViewExport command now respects hidden and tangent lines.

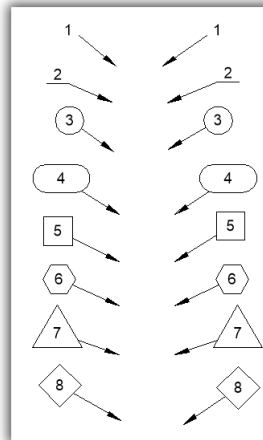
WHAT'S NEW IN MODELING

(bm- commands are available in the Platinum edition of BricsCAD only)

bmBalloon command adds the **Auto mode** option to place balloons on all components automatically in a specific drawing view; uses user-defined styles and predefined balloon styles from *_BalloonStyles.dwg*.

: **BMBALLOON**

Select a component insert [select Table/select Style/**Auto mode**]:



Default ballon styles included with BricsCAD

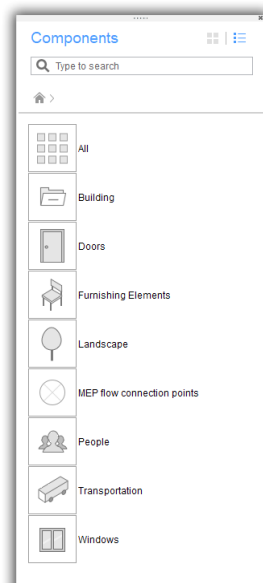
bmBOM command adds a BOM (bills of material) status parameter to determine whether components and their subcomponents are included in BOM tables.

-bmCreateComponent command adds entities in the drawing to the Components panel/library:

: **-BMCREATECOMPONENT**

Select entities to create component from or use <Drawing>: (Select one or more entities, or press Enter to select the entire drawing)

Specify component category: [1 - All/ 2 - Building/ 3 - Doors/ 4 - Furnishing elements/ 5 - Landscape/ 6 - Mep flow connection points/ 7 - People/ 8 - Transportation/ 9 - Windows/ 0 - Add new category/] <Current>: (Choose a category to which to add the component)



The default component categories

bmExplode command now edits exploded representation and generates 2D drafting views; exploded representations can be edited in the Mechanical Browser by adding, deleting, and reordering components. It gains a [Linear](#) option to explode assemblies linearly in a given direction, taking into account collisions between components. It animates either selected steps or the entire sequence.

: BMEXPLODE

Select explosion algorithm [Table by types/[Linear](#)/Manual/Settings] <Manual>: *(Enter an option)*

bmExplodeMove command creates user-defined exploded representations.

: BMEXPLODEMOVE

Select entities to explode [Entire model] <Entire model>: *(Press Enter)*

Select base part <None>: *(Select an entity)*

Entities in set: 1

Select base part <None>: *(Press Enter to continue)*

Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>: **2p**

Specify start point of axis <0,0,0>: *(Press Enter)*

Specify end point of axis or <use axis parallel to view direction>: *(Pick a point)*

bmInsert command now inserts Revit Family RFA files as components; it also creates parametric components, and supports window insertions similar to the **bimInsert** command.

IP To open RFA files in a new drawing, use the **Import** command.

-bmParameters command now edits associative arrays of parametric components and can be assigned to an expression using this command, or Mechanical Browser or Properties panel.

bmReplace command now prompts with [from File](#) to select a replacement component, instead of opening the File dialog box; adds the [Parameters changes](#) option to control which parameter values are to be used after replacement — reapply changes of parameters for the component being replaced or use replacement as-is.

: BMREPLACE

Select component inserts to be replaced: *(Select one or more components)*

Select component inserts to be replaced: *(Press Enter to continue)*

Select component insert to use as a replacement [SIMilar inserts/component Type/ [Parameter changes](#)/[from File](#)] <from File>: *(Enter an option)*

WHAT'S NEW IN POINT CLOUDS

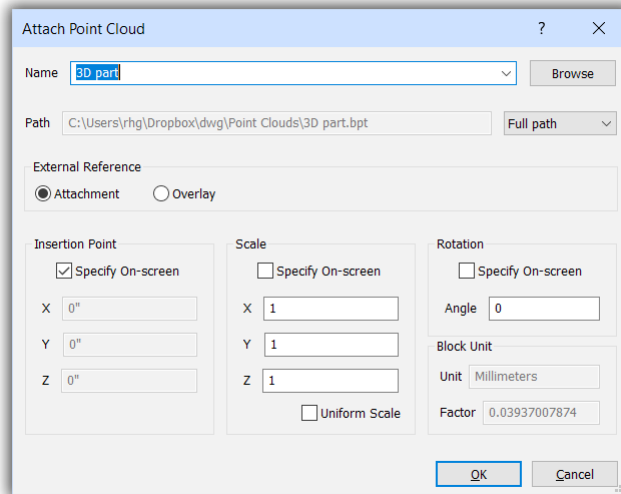
Point clouds are a new entity type in BricsCAD V19: AcDbPointCloudEx.

Attaching a point cloud file takes two steps:

1. Run the **PointcloudPreprocess** command to convert PTS, PTX, and LAS files to Bricsys' own BPT format.
2. Run the **PointCloudAttach** command to attach the BPC file to the current drawing.

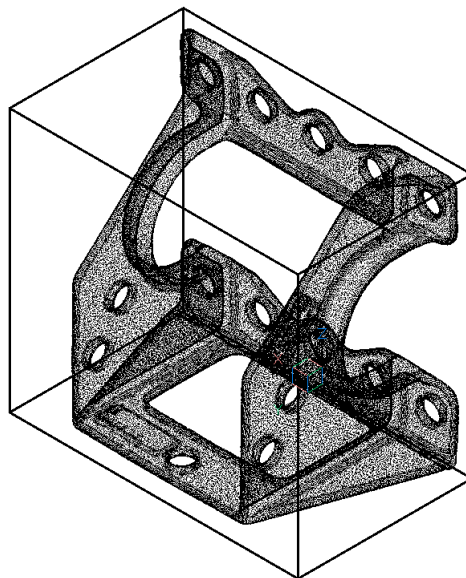
PointcloudAttach command opens a BPT point cloud file, and then attaches it to the current drawing.

: POINTCLOUDATTACH



Specifying the attachment settings

Set point cloud UNITS (0..20), default is to use current INSUNITS value <1>: (Press Enter)
SUCCEEDED to attach Bricsys point tree C:\Users\rhg\Dropbox\dwg\Point Clouds\3D part.bpt!!

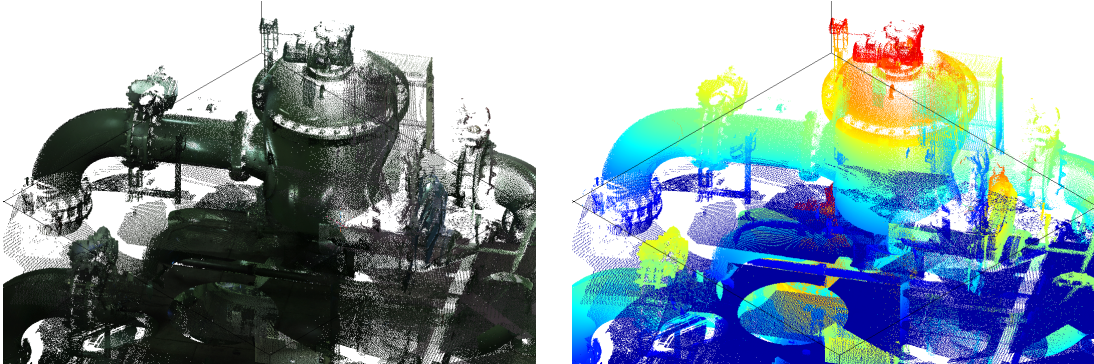


Point cloud displayed inside its bounding box

PointcloudColorMap command displays the Point Cloud Color Map dialog box to set colors based on Intensity, Elevation, and Classification of the point clouds.

: POINTCLOUDCOLORMAP

Color map [Scan/Jet/Earth/Hydro/Gray] <Scan>: (Enter an option)



Left: Color of points set to “scan”; right: color set to color “jet”

The options of the PointcloudColorMap command have the following meaning:

Scan	Display points by the colors specifies in the source laser capture file
Jet	Display points in a range of colors from red through green to blue
Earth	Display points in colors ranging from orange to brown
Hydro	Display points in colors ranging from light to dark blue
Gray	Display points in colors ranging from black to light gray

PointcloudPointSize variable sets the size of new point cloud objects; range is from 1 to 10 pixels; default is 1 pixel.

PointcloudPointSize_Minus command decreases the size of points.

: POINTCLOUDPOINTSIZEMINUS

Decreased point size to 0.500000

PointcloudPointSize_Plus command increases the size of points.

: POINTCLOUDPOINTSIZEPPLUS

Increased point size to 1.000000

PointcloudPreprocess command converts raw point cloud data from PTS, PTX, and LAS files to the optimized BPT format, which can be attached to a drawing.

: POINTCLOUDPREPROCESS

The command opens the Point Cloud Data File dialog box; choose one of the recognized formats, and then save it in BPT format in the Give Name for Preprocessing Output File dialog box.

BPT	BricsCAD point cloud format (compressed)
LAS	LASer (lidar) file format
PTS	Leica PoinTS cloud format (x,y,z points only)
PTX	Enhanced Leica point cloud format (with luminosity)

TIP Point cloud data always consists of an x,y,z-coordinates of each scanned point, and sometimes the color and light intensity — usually in simple ASCII format.

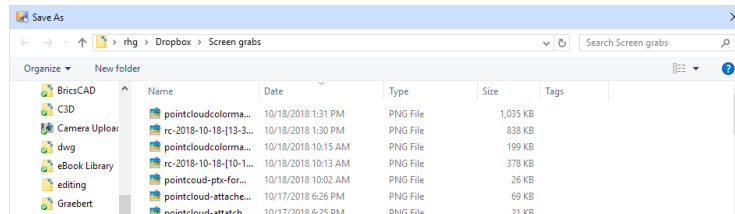
Next, the command prompts you as follows:

Start preprocessing of raw point cloud data?

This is a necessary step, but will take ~1 minute per million points. [Yes/No] <Yes>: (Press Enter)

Preprocessing data: C:\Users\rhg\Dropbox\dwg\Point Clouds\3D part.bpt -> This will take approx 1 minute per million points source data.

The processing is done in the background, and may take several minutes or hours, depending on the number of points in the file. In the meantime, you can do other things in BricsCAD. When the processing is done, BricsCAD displays the following dialog box:



Being notified that the processing is complete

WHAT'S NEW IN SHEET METAL

(sm- commands require a separate license)

Smart feature technology rebuilds relief features and geometry upon local changes, when cut is no longer needed for the new design. Also it makes edges, imprinted on thickness faces, resistant to local simplifying operations.

The **Settings** option is removed from sm- commands and relocated to the Sheet Metal group in the Settings dialog box.

Quad menu updates the “Sheet Metal” section to provide the complete set of sheet metal tools.

smAssemblyExport command now recognizes single parts, and skips entities on layers ‘bim_subtract’, ‘bim_unite’, ‘bc_subtract’, and ‘bc_unite’.

smConvert command now recognizes straight holes as a special type of form feature, as well as form features not smoothly connected with flanges. It uses the Sharp Bend Radius setting to distinguish regular bends from lofted bends, and defillets recognized control curves for rib features.

smExportOsm command now exports straight holes and non-sheet metal solids as inserts within selected sheet metal parts.

smFlangeConnect command in some cases no longer needs to create corner reliefs

smFlangeEdge command now creates miters and junctions automatically for a number of configurations, and eliminates self-intersections.

smFlangeSplit command is removed from V19; replaced by the **smSplit** command.

smFlip command removes some options from the prompt and adds the [Flip reference side only](#) option:

: SMFLIP

Old prompt:

Select solids, flange faces ~~and allow rebuild geometry~~ [Disallow rebuild geometry/Entire model] <Entire model>:

New prompt:

Select solids or flange faces to flip flange(s) [[Flip reference side only](#)/Entire model] <Entire model>:

smJunctionCreate command now produces junction cuts on curved hard edges, including elliptical and spline curves, and assigns junction features to cutouts solely created on linear hard edges.

smLoft command now converts lofted bends to ordinary bends, if possible.

smReliefCreate command is more accurate near non-sharp, large-radius bends by minimizing the cut to link better with junctions on curved edges.

: SMRELIEFCREATE

Select a hard edge, bend face, flange face or 3D solid [Entire model]:

Enter relief size by bend radius ratio or [[force Bend reliefs](#)/Auto] <Auto>:

smReliefSwitch command now manually selects the faces to be converted to smooth bend reliefs.

: SMRELIEFSWITCH

Select faces, 3d solids to switch reliefs to V-type [RECTangular/Circular/V-type/SMooth/RIP/ROUND/[relief EXTension](#)/Entire model] <Entire model>:

smRibCreate command now uses a fillet radius as a parameter; ribs can be exploded into flanges, bends, and other features.

smRepair command now repairs flange faces locally.

smParametrize command now recognizes straight hole rectangular arrays.

smSelect command get a new option **Select non-orthogonal geometry** option select faces of the same thickness but are not orthogonal.

: **SMSSELECT**

Select the option [Hard edges/Same form features/Similar form features/**Non-orthogonal thickness faces**]:

smSplit command combines the smFlangeSplit command and the new Lofted Bend Face functions to split lofted bends.

: **SMSPLIT**

Select flange **or lofted bend face**: (*Pick a flange or Lofted blend feature*)

Select lines or edges to split the flange or [use SMarT split/draw a New line] <draw a New line>: (*Enter an option*)

smTargetCAM variable specifies the target cams to be used by smUnfold.

smUnfold command now has associativity between the 3D model and its 2D unfolded model; also, it does not change the properties of layers that exist.

TIP When the layer has to be created, the settings of the various unfolding variables are used.

WHAT'S NEW IN APIs

BricsCAD V19 is compiled with Visual Studio 2017. To be compatible, C++ extension DLLs must be compiled with the same platform toolset.

PDF-related commands now use the Google's free PDFium engine.

BLADE for interactive LISP development has been updated.

New entity types in V19:

- ▶ Point clouds are entity type 'AcDbPointCloudEx'
- ▶ Terrain surfaces are entity type 'TIN Surface' (TIN is short for triangular irregular network)

WHAT'S NEW IN LICENSES

BricsCAD V19 Pro license is required for

- ▶ Rendering
- ▶ ACIS-based 3D modeling
- ▶ Drawing view generation

BricsCAD V19 Platinum license is required for

- ▶ 3D constraint
- ▶ Assemblies
- ▶ Deformable modeling

BricsCAD V19 BIM license is required for

- ▶ BIM design

BricsCAD V19 **Mechanical** (formerly known as Sheet Metal) license is required for

- ▶ Mechanical design
- ▶ Sheet metal design

Communicator V19 license is required for

- ▶ Advanced file translation

The 30-day demo version of BricsCAD V19 can be downloaded from <https://www.bricsys.com/protected/download.do>

