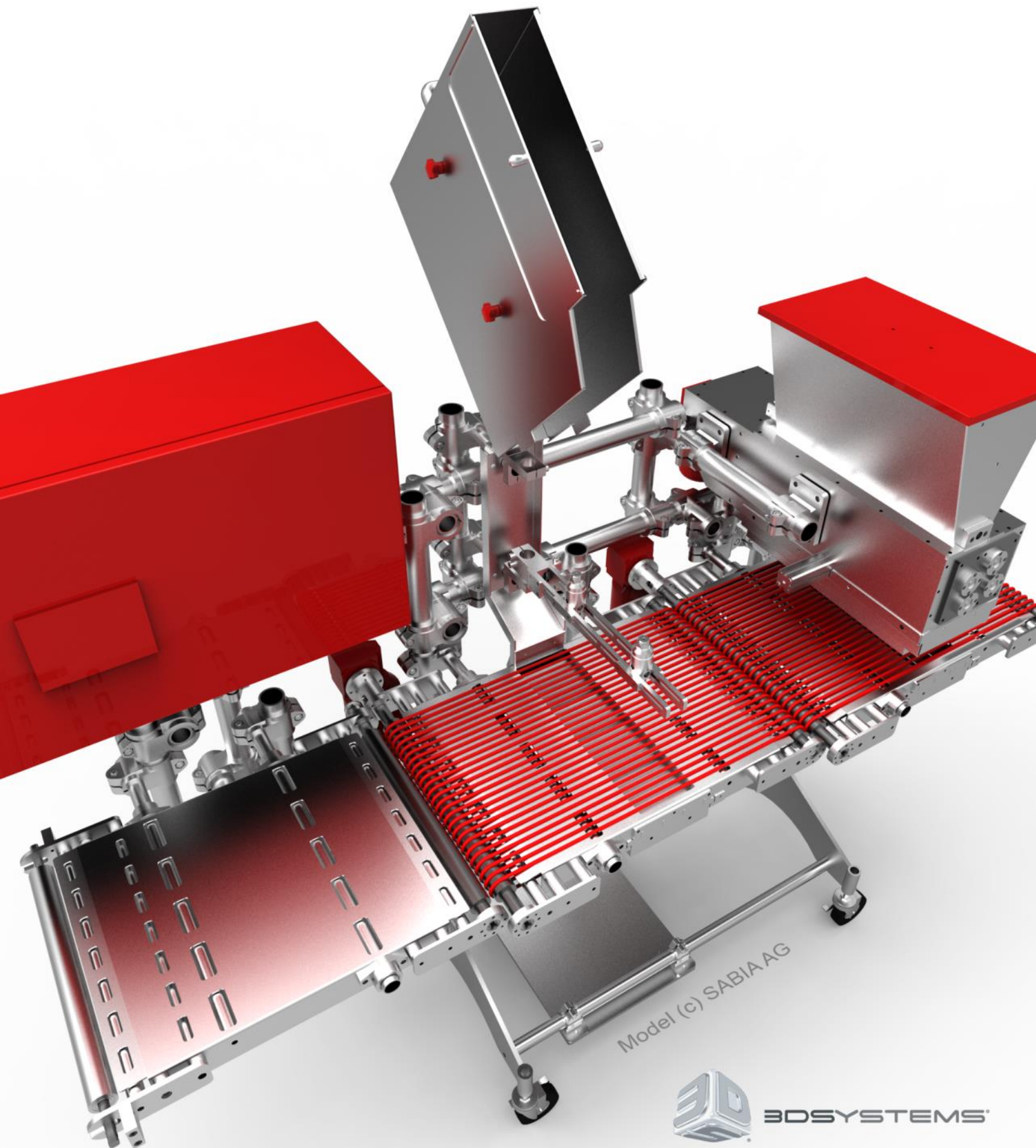


Geomagic Design™

What's New



Model (c) SABIA AG



BD SYSTEMS®

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GENERAL USER INTERFACE

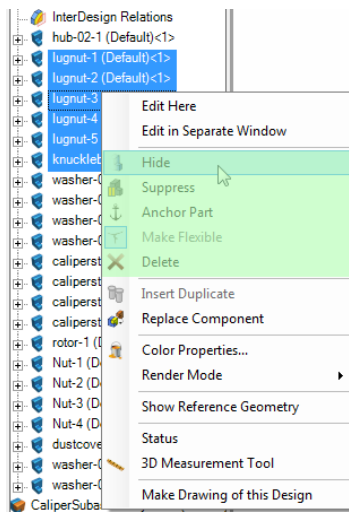
USER INTERACTION CONSISTENCY

CTRL AND SHIFT BEHAVIORS

- Multi-selection in the Design Explorer and Work Area using Ctrl and Shift has been uniformly implemented. Previously their use was treated differently in various tools or areas of the product.
- Dialog boxes that use multiple inputs (for example, a Loft dialog that can take multiple profile sketch inputs) no longer require you to use CTRL or SHIFT to select multiple objects.
- More commands can be applied to multiple objects in a single operation:
 - Anchor
 - Hide
 - Color Properties
 - Suppress
 - Show Reference Geometry
 - Move View to Sheet
 - Align View
 - Show Hidden Lines on View
 - Show Tangent Edges on View
 - Reproject Design Dimensions on View
 - Insert Centers on View
 - Insert Cosmetic Threads on View
 - Insert External Thread Call Outs on View
 - Insert Hole Call Outs on View
 - Scale View

RIGHT-CLICK BEHAVIOR

When selecting multiple objects and then right-clicking, there is now a consistent experience to show options that are available to all selected items. Previously, in some cases a right-click would deselect all selected items. This is particularly noticeable in assemblies when trying to anchor, hide, suppress, etc. multiple parts simultaneously using mouse commands.



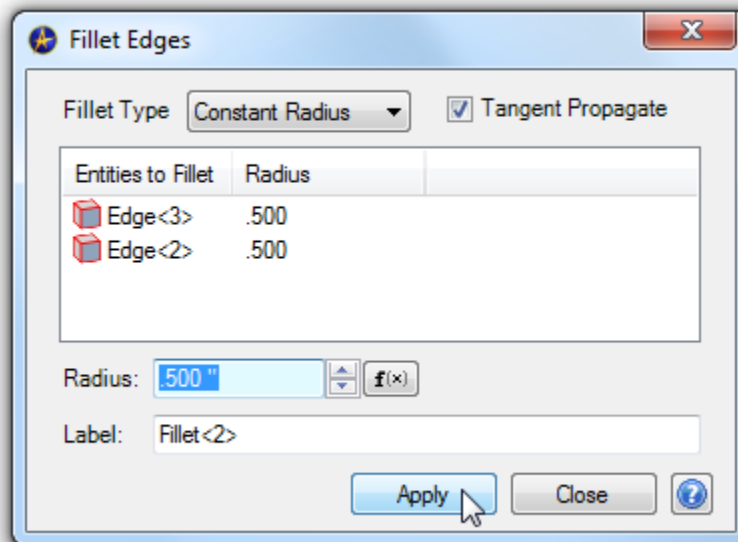
DRAGGING THE DOG BONE

Dragging the Dog Bone in a Design, Drawing, or Assembly Explorer now behaves as expected. Previously, the action would cause unexpected behavior and would make dragging the Dog Bone unpredictable when dragging involved the Explorer pane scrolling up or down.

APPLY BUTTON ADDED TO DIALOGS

The following dialogs now contain an Apply button so you can make multiple instances of the features without having to re-invoke the tools:

- Insert Point
- Insert Axis
- Insert Plane
- Sketch -> Insert Point
- Sketch -> Insert Axis
- 3D Fillet
- 3D Chamfer
- Sheet Metal Closed Corner
- Sheet Metal Corner Round
- Sheet Metal Corner Chamfer

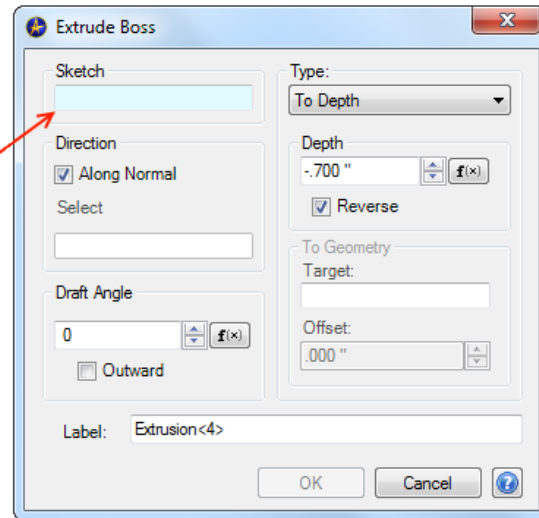
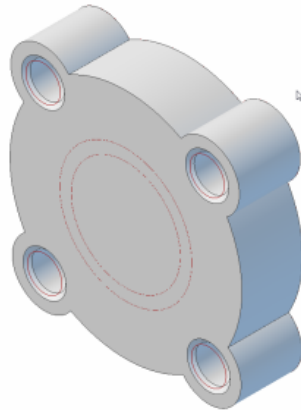


MISSING DESIGN DIALOG RESIZABLE

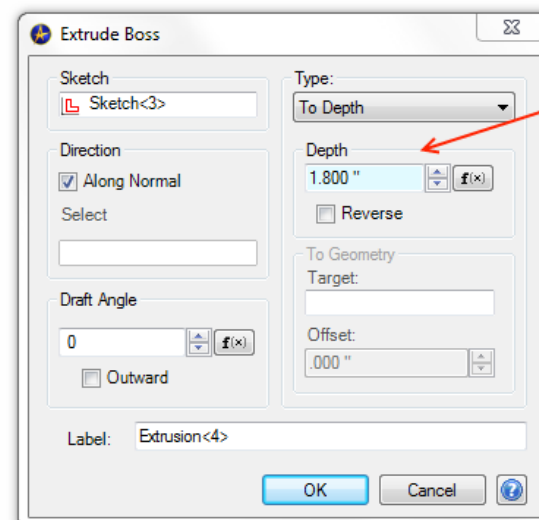
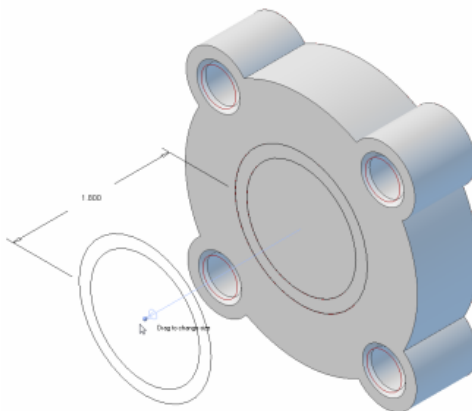
The Missing Design dialog is now expandable. This makes it easier to see what is going on in cases where the file paths are very long or the assembly is extremely nested.

FOCUS BEHAVIOR IN DIALOGS

In many dialogs that require multiple inputs, for example Revolve Boss (requires Sketch, Axis, and Angle), the focus in the dialog progresses automatically as you make inputs. You no longer need to move back and forth between your work area and the dialog to constantly switch the active input field. This reduces mouse travel and allows you to design more quickly.



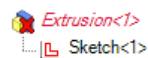
Dialog highlights Sketch field as active input.



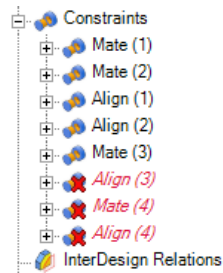
After selecting a sketch, the input switches automatically to the next input.

IMPROVED CONSTRAINT AND FEATURE CONFLICT VISIBILITY

- Failed features are now more visible with an improved failed feature icon and red text in the Design Explorer.

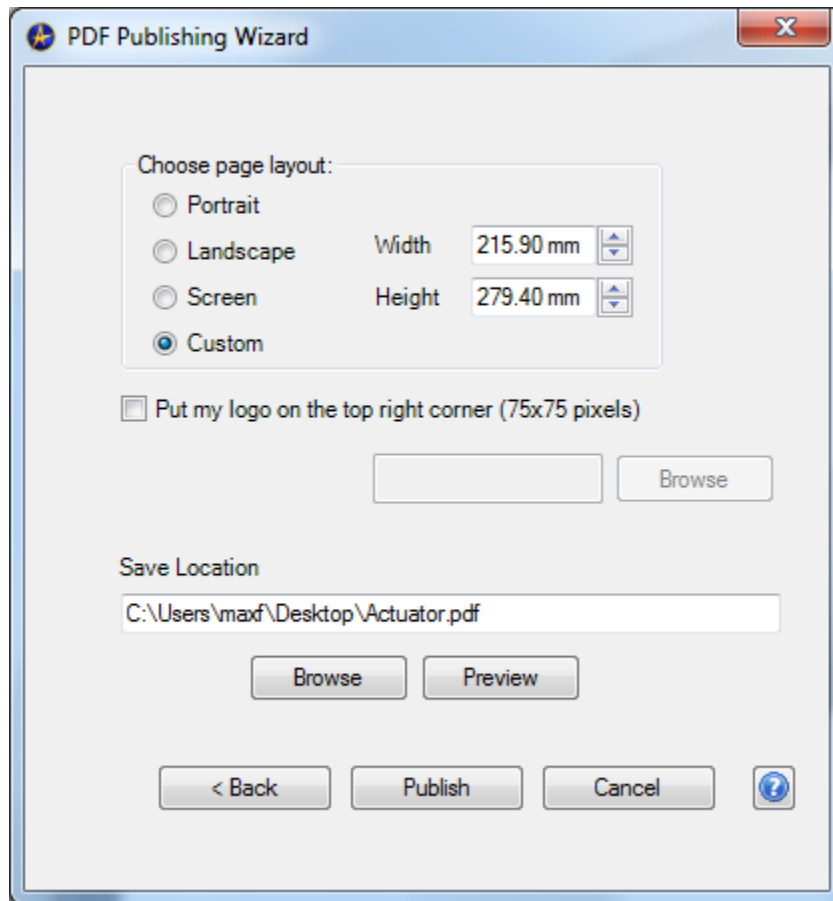


- Constraint conflicts are now shown in the same manner as failed features, improving the visibility of constraint problems.



3D PDF EXPORT – METRIC UNITS

When a workspace is set to use metric units, the Publish to PDF dialog automatically changes to metric sizing.



UNIFIED MATERIAL SELECTION

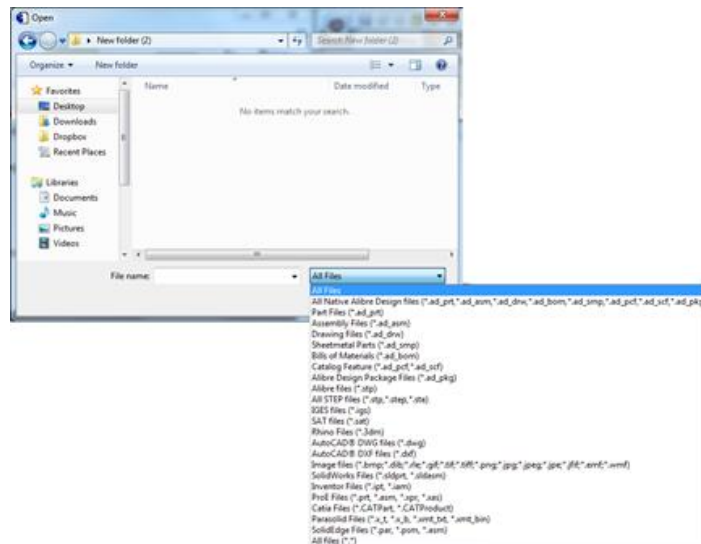
When selecting a material for a part on the Material pane of the File Properties dialog, the Material Property of the Part Data is automatically updated. The Material Property of the Part Data pane is what is used when projecting fields in 2D drawings or in BOMs. This means that you can select a material in one place instead of two.

LICENSING DIALOGS CLEANED UP

The dialogs for license updates and changes have been cleaned up.

OPEN/IMPORT DIALOGS CONSOLIDATED

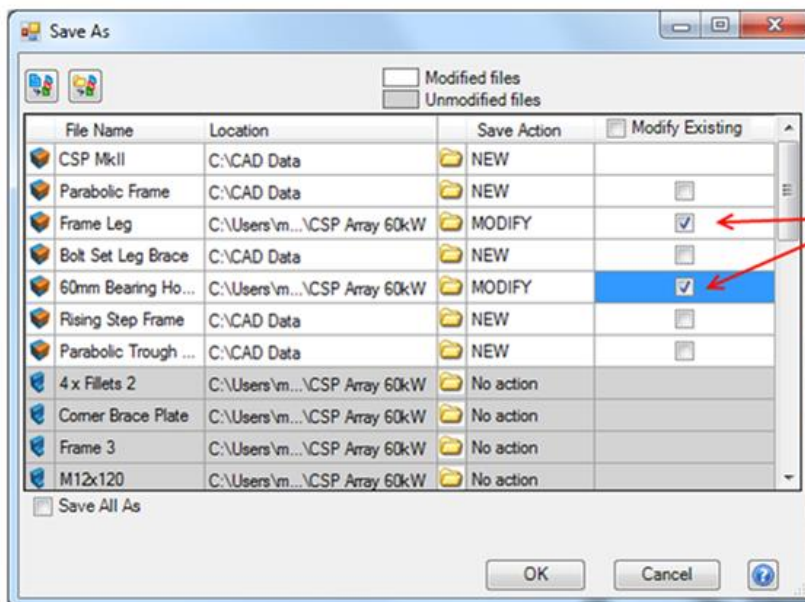
The Open and Import dialogs have been consolidated into a single Open dialog:



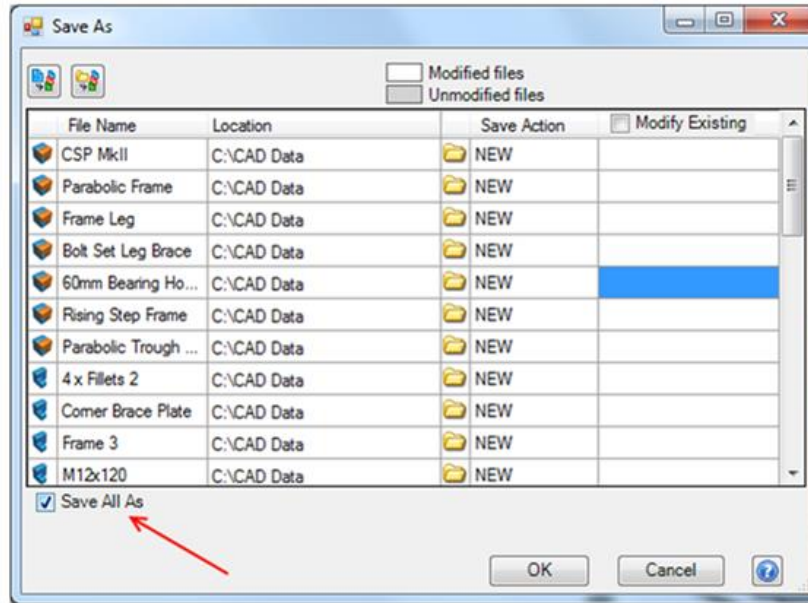
SAVE AS & SAVE ALL AS DIALOGS CONSOLIDATED

The following changes have been made to the Save As and Save All As dialogs:

- The two dialogs have been consolidated.
- You now have additional options to selectively create a new file or overwrite a previously saved file, which means you can selectively choose to leave certain files in their existing locations during a Save As operation.



The new Modify Existing checkbox allows you to overwrite certain files selectively during a Save As operation, instead of forcing you to create duplicates.



The Save All As checkbox allows you to save all constituent files as new files.

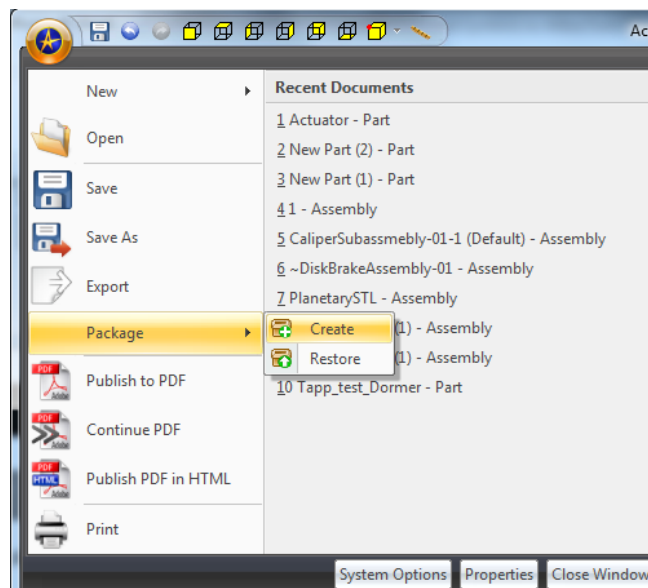
GENERAL CHANGES

ACIS VERSION

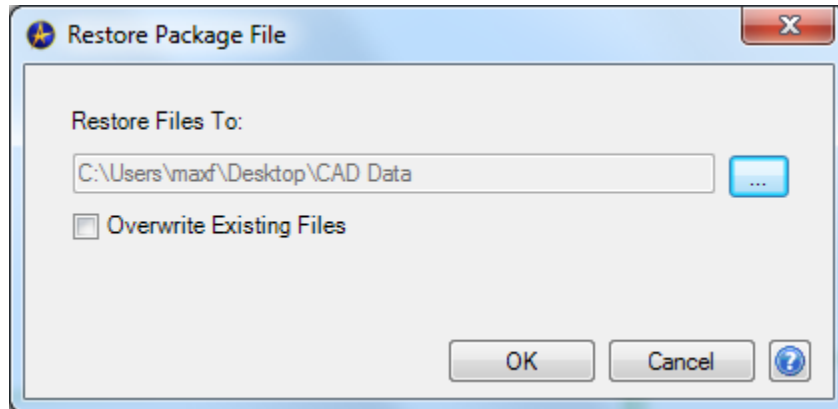
The ACIS version has been upgraded to R23.

ALIBRE STEP REPLACED WITH GEOMAGIC PACKAGE

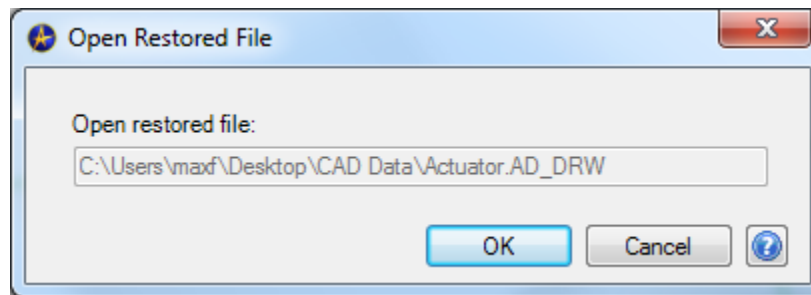
The Alibre STEP format has been replaced by the Geomagic Package. The package format acts in much the same way as the Alibre STEP format, but has improved data fidelity. You can Create and Restore a Package file from the Gem Menu:



This will create a single file that contains all the data for your design. Similar to the Alibre STEP, creating a package of an assembly or a 2D drawing will include all the constituent files, such as parts and assemblies. When restoring a package, you will select where to deposit the original design files:



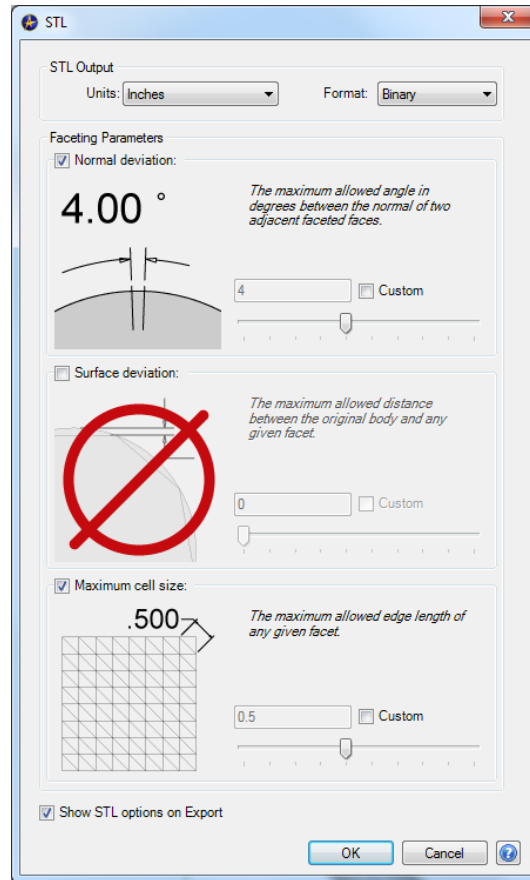
Subsequently, you will be asked if you want to open the file from which the package was originally created:



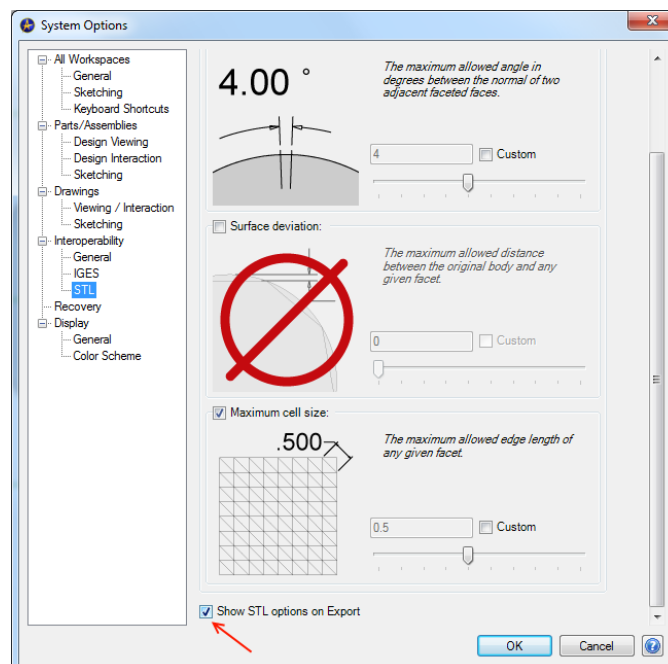
STL EXPORT OPTIONS

ON-DEMAND OPTIONS

When exporting an STL, you will be prompted with a new dialog that allows you to select options specific to the design you are exporting. This makes it easy to change your settings between files rather than having to change the global options. You can also select which units you would prefer the STL to be output in, which is useful for some downstream applications that recognize STL units.



If you prefer to set the options one time and not get prompted on every STL export, open the System Options and uncheck **Show STL options on Export** from the STL pane:



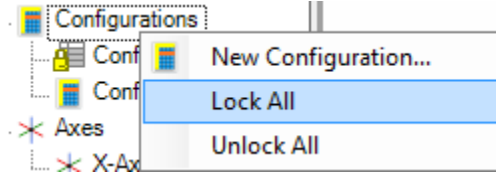
BINARY FORMAT

Geomagic Design now supports the binary STL format. A binary STL offers identical fidelity to a human-readable STL but with a significantly compressed file size. You will almost always want to choose binary STL as your export option because of the file-size advantage.

CONFIGURATIONS

LOCK & UNLOCK ALL

A new option to lock or unlock all Configurations is available via the right-click menu.

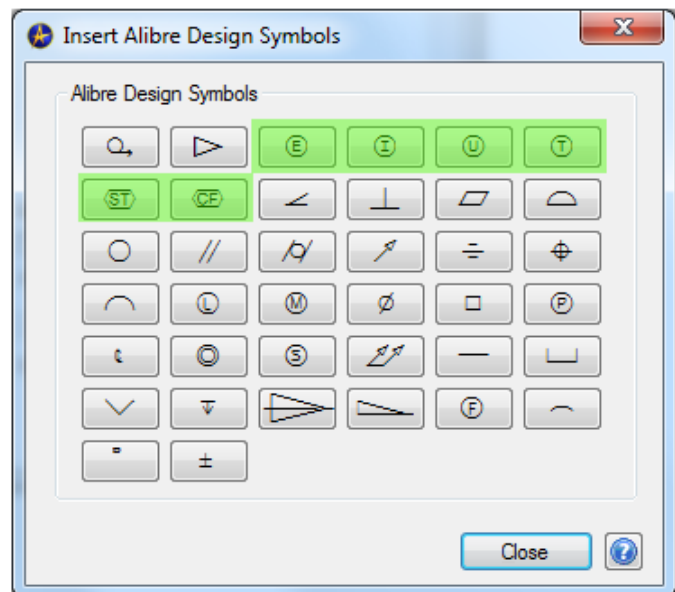
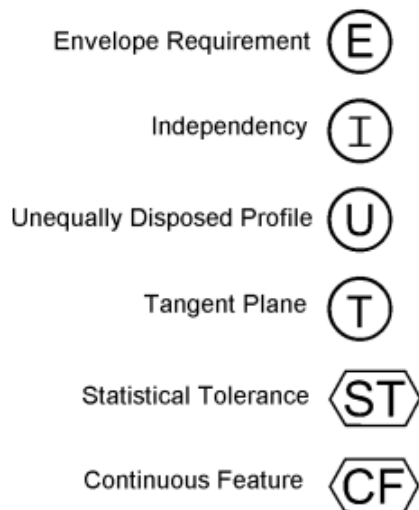


LOCK AND UNLOCK BEHAVIOR MORE INTUITIVE

Previously when locking a configuration, all lockable settings for the target configuration were automatically locked, overriding the user-selected settings. This forced you to reconfigure your settings when switching between a locked and unlocked state. This behavior has been modified to retain the user-selected lockable settings during a lock and unlock action.

SYMBOL LIBRARY

The following symbols have been added to the Symbol Library:



MEASUREMENT INFORMATION AVAILABLE IN THE STATUS BAR

Basic measurement information is now available in real-time in the Status bar at the bottom right of the screen based on your active selection(s). This allows you to avoid invoking the measurement tool for every measurement. Click on 3D geometry to check its measurement. The following measurements are supported:

Fillets, Cylindrical Faces, Spherical Faces, Circular Edges: Radius

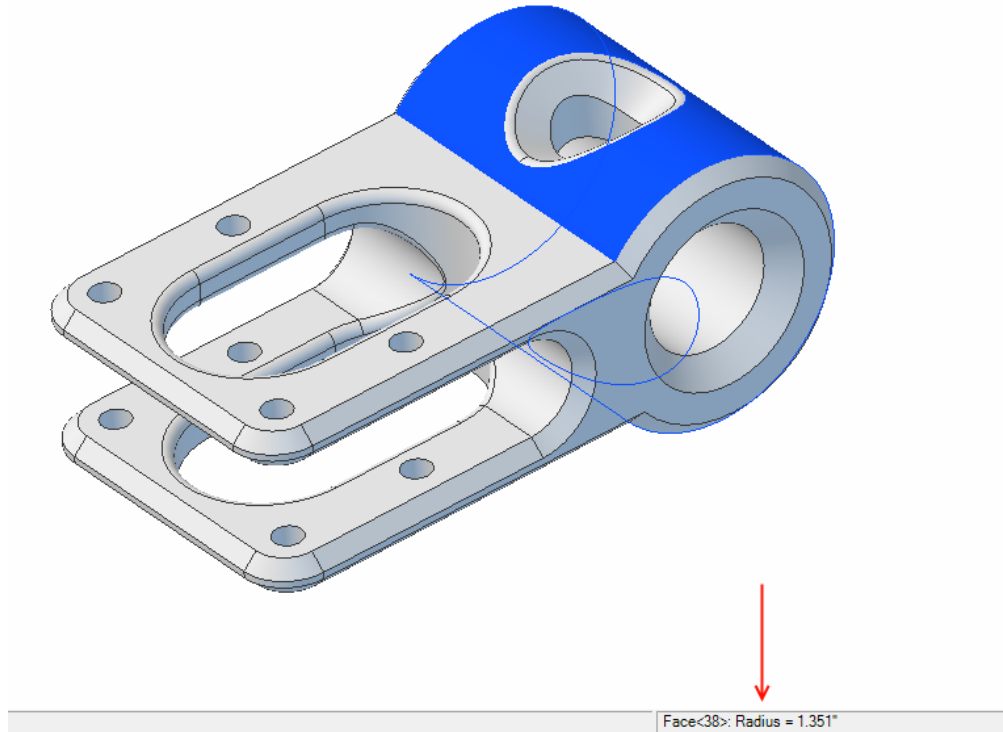
Non-circular Edges: Length

Non-cylindrical and Non-spherical Faces: Surface area

2 Parallel Faces or Edges: Distance between the selected items as measured from the nearest points

2 Cylindrical Faces or Edges: Angle between Axes

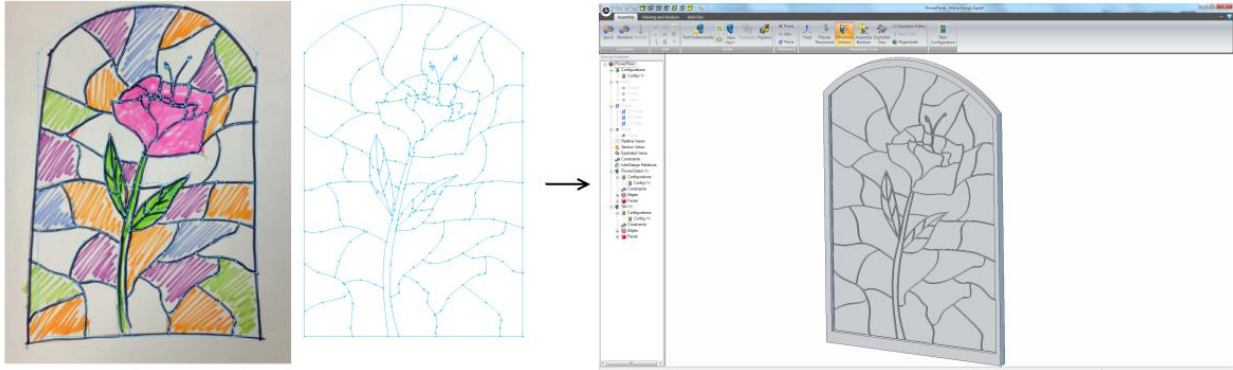
2 Planar Faces or Linear Edges at an Angle: Angle between faces



SKETCHING

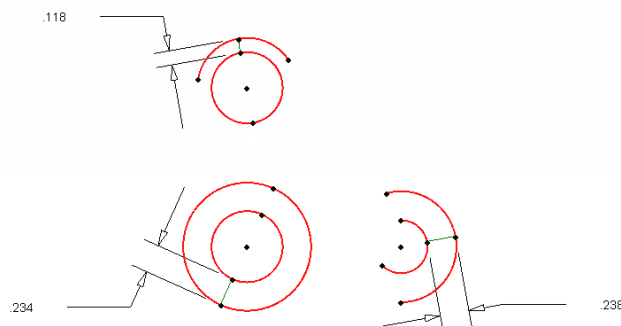
TRACING PERFORMANCE

When tracing images in a 2D workspace to generate a sketch to be used in a 3D model, the sketching performance is significantly improved and more responsive, especially when zoomed in. Previously, zooming in on an image slowed down the sketching experience roughly proportionally to the zoom level.



DIMENSIONING BETWEEN CONCENTRIC ARCS AND CIRCLES

Dimensions can now be placed between concentric arcs and circles to define the difference in their radii. Previously, you had to use the equation editor to define the offset distance from one parameter to another. This method is significantly faster and more intuitive.



DISJOINT ENDS

All Disjoint Ends are automatically healed when the active Sketch is exited. You are no longer prompted that sketch problems exist when only Disjoint Ends exist, as they are automatically healed. Disjoint Ends were typically caused during the following scenarios:

- Mirroring or patterning a figure or figures that have nodes that would “land” on top of other figures
- Copying a 2D profile that was imported into a 2D workspace, for example a DXF or DWG file, into a 3D workspace to create a feature

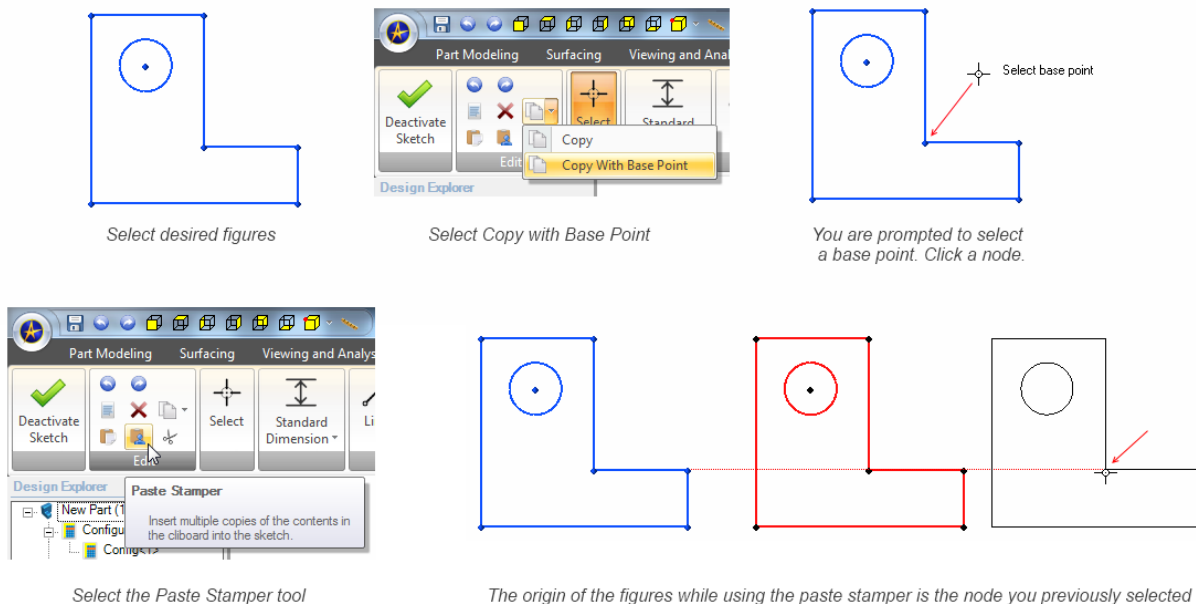
Disjoint Ends almost always resulted in unnecessary re-entry into Sketch Mode to fix them via the Sketch > Analyze tool and are now handled by the software.

PRESSING ESCAPE ACTIVATES SELECT TOOL

Pressing Escape while in 2D Sketch Mode will make the Select tool active. If a sketch figure has been partially placed, the first Escape will cancel the figure and a subsequent Escape will invoke the Select tool.

COPY WITH BASE POINT

A new Copy command, Copy with Base Point, allows more precise control when using the Paste Stamper. The tool allows you to define the point that will be coincident with your mouse while you use the Paste Stamper, which can be especially helpful when utilizing inference lines while pasting.



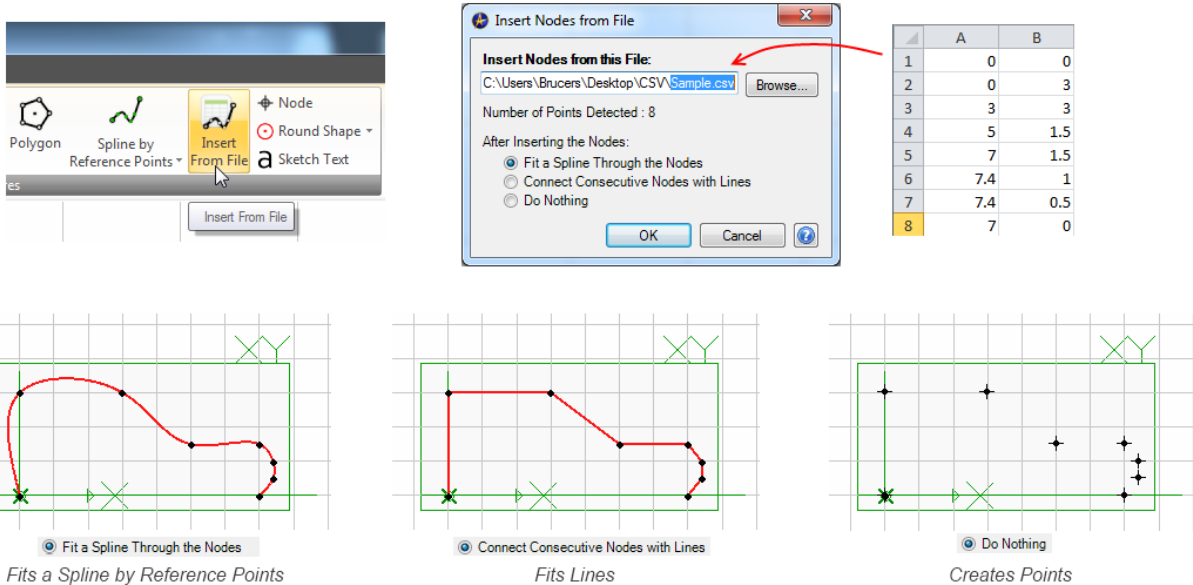
Tip:

- Copy with Base Point and Paste Stamper are particularly useful tools. The default keyboard shortcuts for these commands are Shift+C and Ctrl+T.

IMPORTING POINTS INTO A 2D SKETCH

You can now import Points from a CSV file into a 2D Sketch. Previously, this was only possible for a 3D Sketch. You have several options on how the software will treat the imported points:

- Fitting a Spline by Reference Points through the points
- Fitting a series of Lines through the points
- Fitting no figures and only creating points



Requirements for the CSV file:

- The CSV file should contain no headers
- The CSV file should be formatted in the following way:
 - The first column should contain the X values
 - The second column should contain associated Y values

Tips:

- The software will treat the points in the order they are presented in the CSV file. This is important when fitting figures through the points.
- If you are fitting a Spline or Lines through the points and desire the profile to form a closed loop, repeat the first point definition at the end of the CSV file.

SKETCH ANALYZE DETECTS SELF-INTERSECTING SPLINES

The sketch analyze tool now detects self-intersecting splines.

HORIZONTAL AND VERTICAL CONSTRAINTS

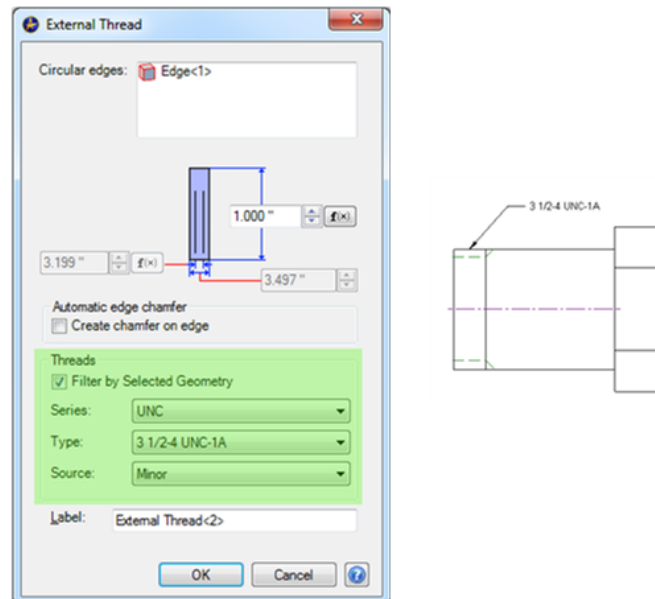
Horizontal and Vertical constraints can now be applied to multiple figures simultaneously.

PART MODELING

EXTERNAL THREADS HAVE PREDEFINED THREAD DEFINITIONS

The External Thread tool now offers drop-down menus of thread definitions. This information carries over into 2D drawings automatically when you select to include External Thread Callouts on a created view. You can optionally

filter the thread options to display only the threads that typically would make sense based on the size of your selection. Disabling the Filter Threads by Selected Geometry option allows you to select any available thread. Previously External Thread Callouts had to be entered manually in the External Thread dialog.



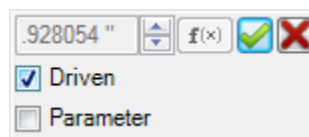
DRIVEN DIMENSIONS

Three significant changes have been made to dimensions that greatly enhance their potential:

- You can now designate any 2D dimension to be **Driven** during dimension placement or after dimension placement by modifying the dimension. “Driven” refers to the fact that the particular dimension does not explicitly drive model geometry, but rather it reflects the current measurement. Most dimensions you place in Geomagic Design are Driving dimensions – placing Driving dimensions will explicitly define the size or orientation of sketch figures.

Driven dimensions update in real-time as you move figures or further define objects in the sketch. One useful way to use this is to use the Driven dimension as a “probe” to determine the dimensions of certain critical aspects of the design while you define the design with explicit dimensions.

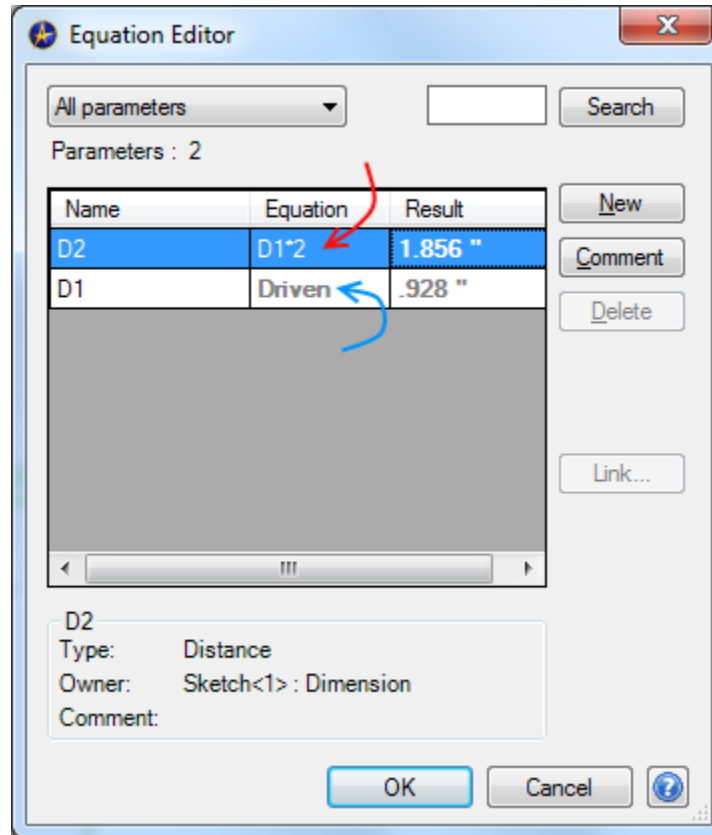
Previously, Driven dimensions could only be created automatically when you attempted to dimension a figure that was totally defined, and as such its dimension was automatically inferred based on the other dimensions or constraints in the sketch.



- You can now designate that a Driven dimension show up in the Equation Editor as a parameter. Previously Driven dimensions could not be parameterized. Allowing the parameterization of Driven dimensions in the Equation Editor provides some interesting and useful workflow improvements, noted later.



- You can now use Driven Dimensions that are parameterized to drive other parameters:



USING DIMENSIONS FROM ONE PART IN ANOTHER PART

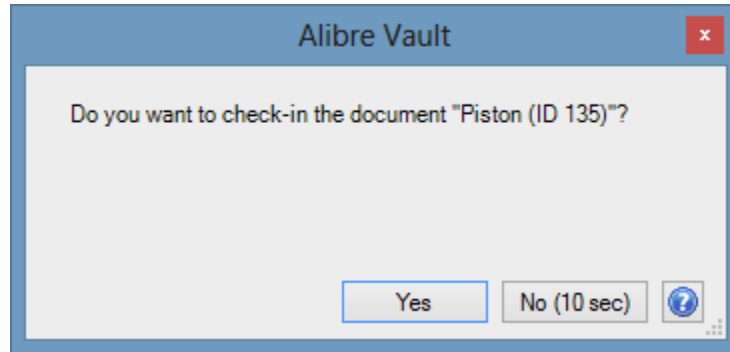
By utilizing the changes shown above, you can effectively use dimensions from one part in another part. To do so, use the following procedure:

- In an assembly, create your first part.
- Create a new part in the context of the assembly.
- Enter 2D Sketch Mode and use Project to Sketch with Maintain Association checked on to project the desired geometry that you would like to reuse.
- Dimension this geometry. These dimensions will be Driven dimensions by default. During the dimensioning process, choose the Parameter option.
- In the Equation Editor, you will now have parameters for the projected figures which represent the parameters you used during the creation of the first part.

ALIBRE VAULT INTEGRATION

AUTOMATIC CHECK-IN PROMPT

When you close a workspace that is using a file that has been checked-out from the Vault, you are now prompted on whether you want to check the file in. Closing a workspace in which you have modified several constituent files (for example, in an assembly) will prompt you to Check In all constituent files.



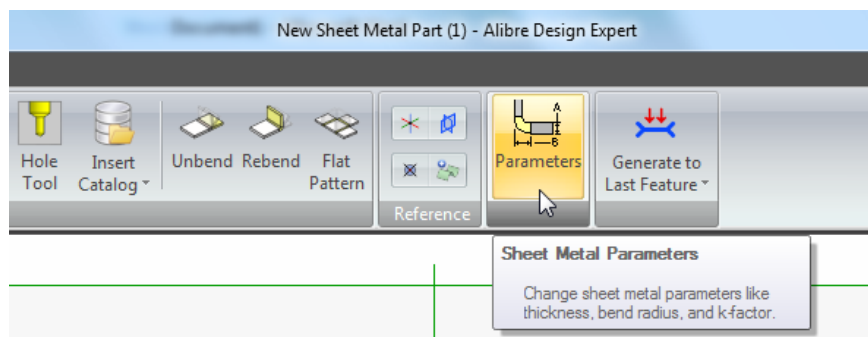
SHEET METAL

LOFTED FLANGES & ASSEMBLY CONSTRAINTS

Circle – Circle Lofted Flanges are now significantly more robust in assemblies and allow assembly constraints to be placed using the model geometry. Previously you had to use the Show Reference Geometry function on Circle-Circle Lofted Flanges in order to use the default or user-created Axes and Planes to constrain these parts to other parts in the assembly.

SHEET METAL PARAMETERS IN RIBBON

Sheet metal parameters can now be accessed directly via the Ribbon:

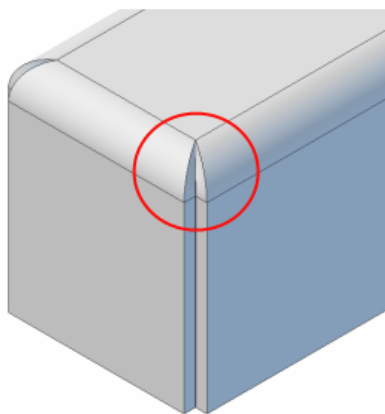


CONVERT TO SHEET METAL NAMING

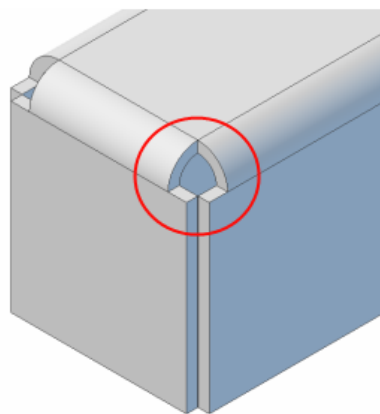
When using Convert to Sheet Metal, the new sheet metal part inherits the name of the part file it is created from.

CONVERT TO SHEET METAL – TRIM CORNER OPTION

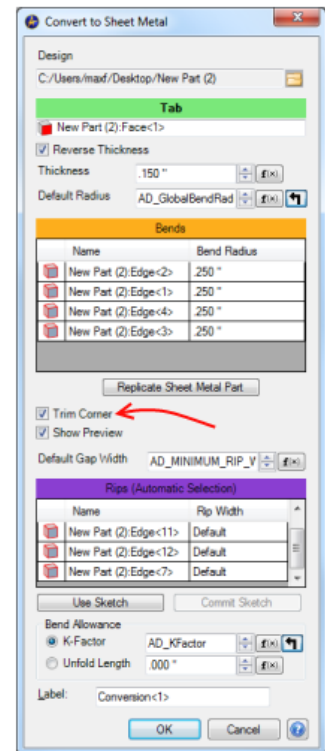
A new option in Convert to Sheet Metal allows you to automatically trim the bend areas where corners converge.



Without Trim Corner Selected



With Trim Corner Selected

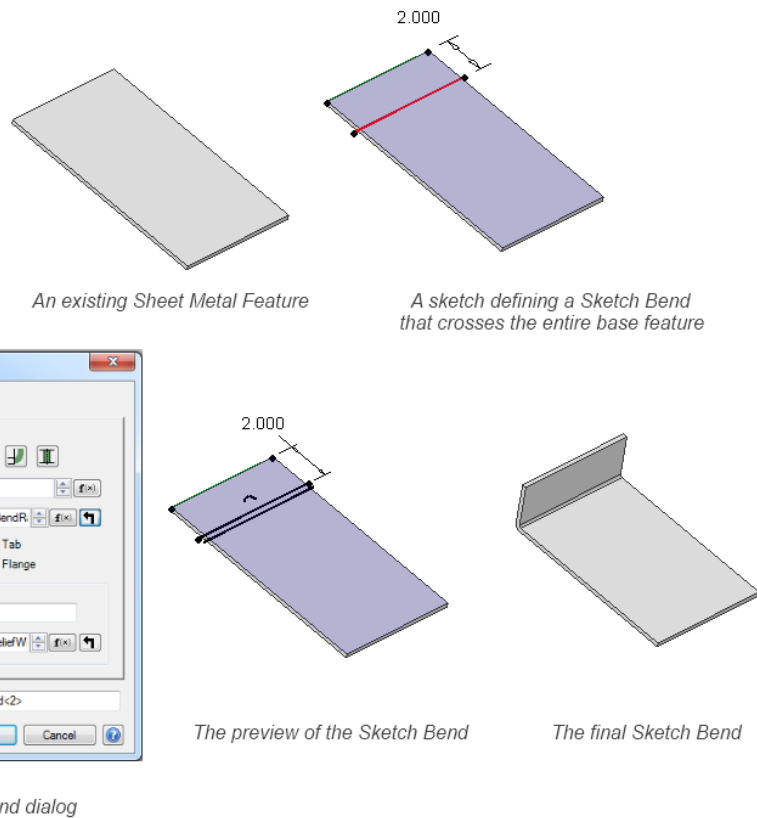


SKETCH BENDS

This new tool allows you to create bend lines directly from a sketch. These bend lines are used to fold a tab or flange. This is particularly useful if you are trying to recreate a sheet metal part from an imported 2D drawing of a flat pattern.

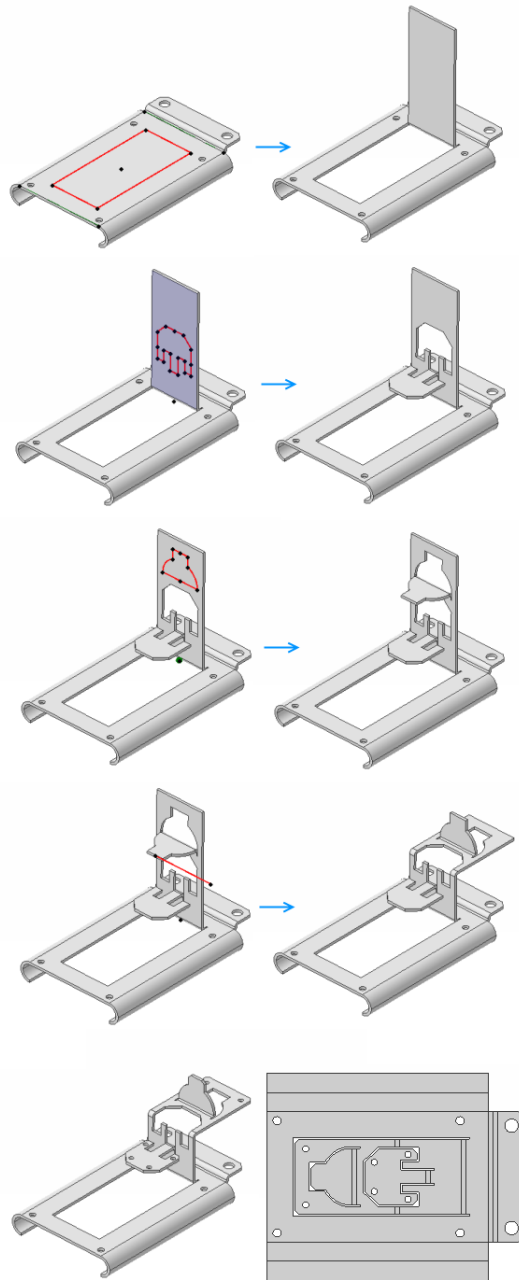
GENERAL SKETCH BENDS

General Sketch Bends involve a sketch that crosses the entire length of the item you wish to bend. In this case, a Flange will be created with no Offsets:



CLOSED-FIGURE SKETCH BENDS

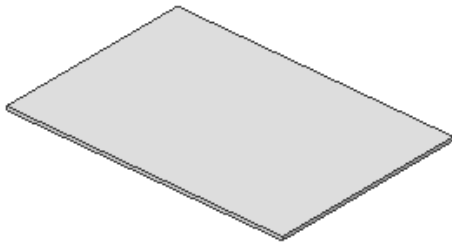
When you create a sketch that contains a single closed loop, the Sketch Bend command will create a flange from the enclosed material as in the following example:



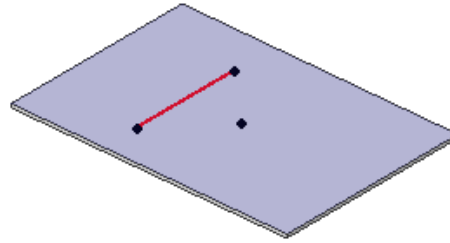
SKETCH BENDS WITH RIPS

Sometimes you will want to create a Sketch Bend that also contains rips. This is done automatically when:

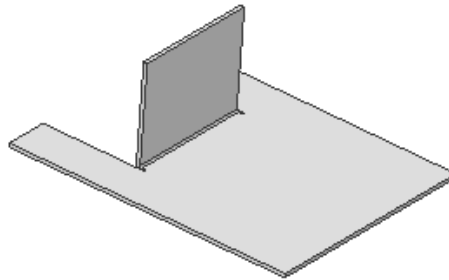
- The sketch does not fully cross the face on which it is being drawn



An existing sheet metal feature

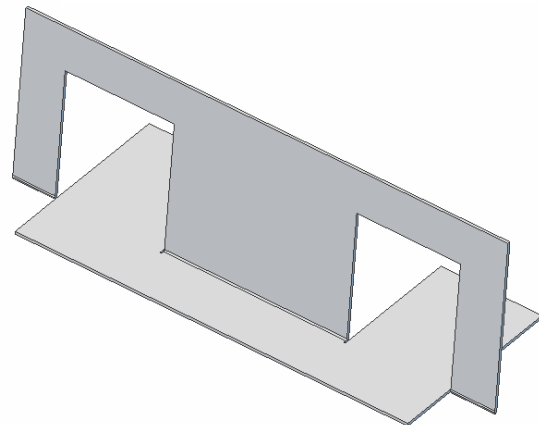
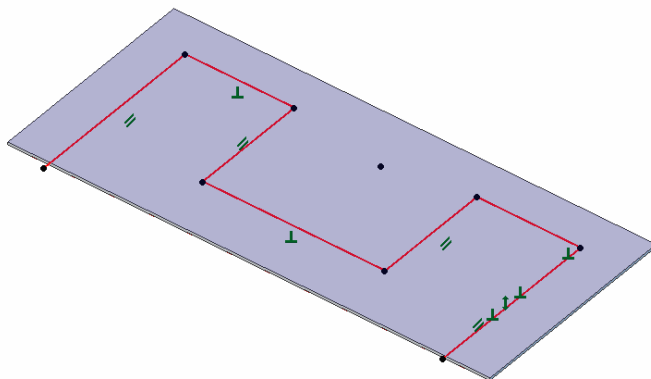


A sketch created for a Sketch Bend with a rip



Resulting Sketch Bend with Rip

- The sketch you create does fully cross the face upon which it is drawn, but contains multiple figures



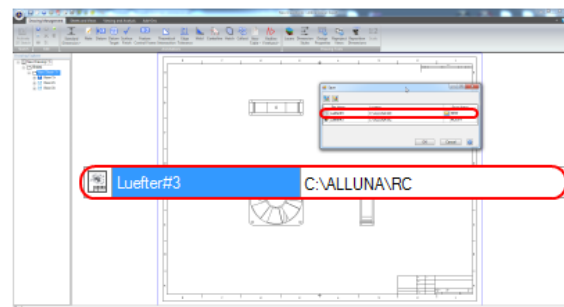
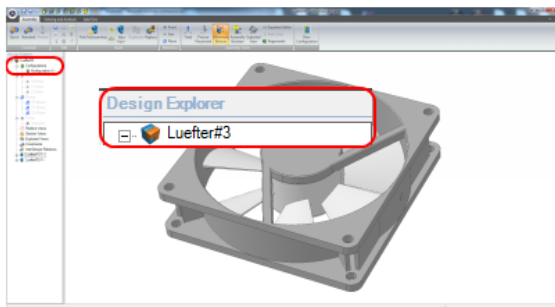
Tips:

- Reference the Help to determine how the two options **Reverse Tab** and **Reverse Flange** will change the outcomes of the Sketch Bend operation.
- Only contiguous figures are allowed; multiple sketch loops will cause the feature to fail.
- Bends can only occur on linear figures.

2D DRAWINGS – GENERAL

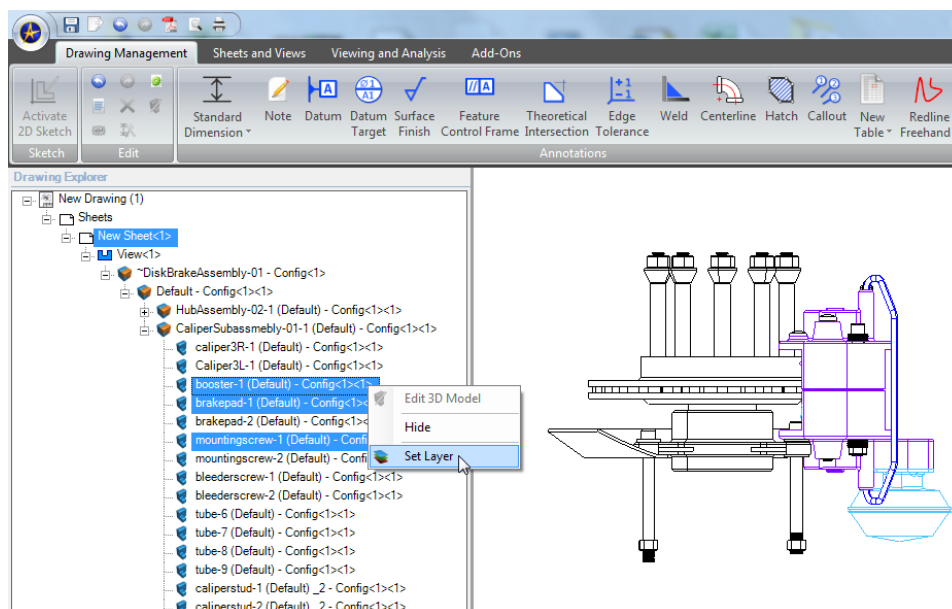
DEFAULT DRAWING NAME

The default drawing name is now taken from the top-level part or assembly on which it is based. Often this is the desired behavior and automates the naming of drawing files after their logical constituents. You may override this behavior if desired.



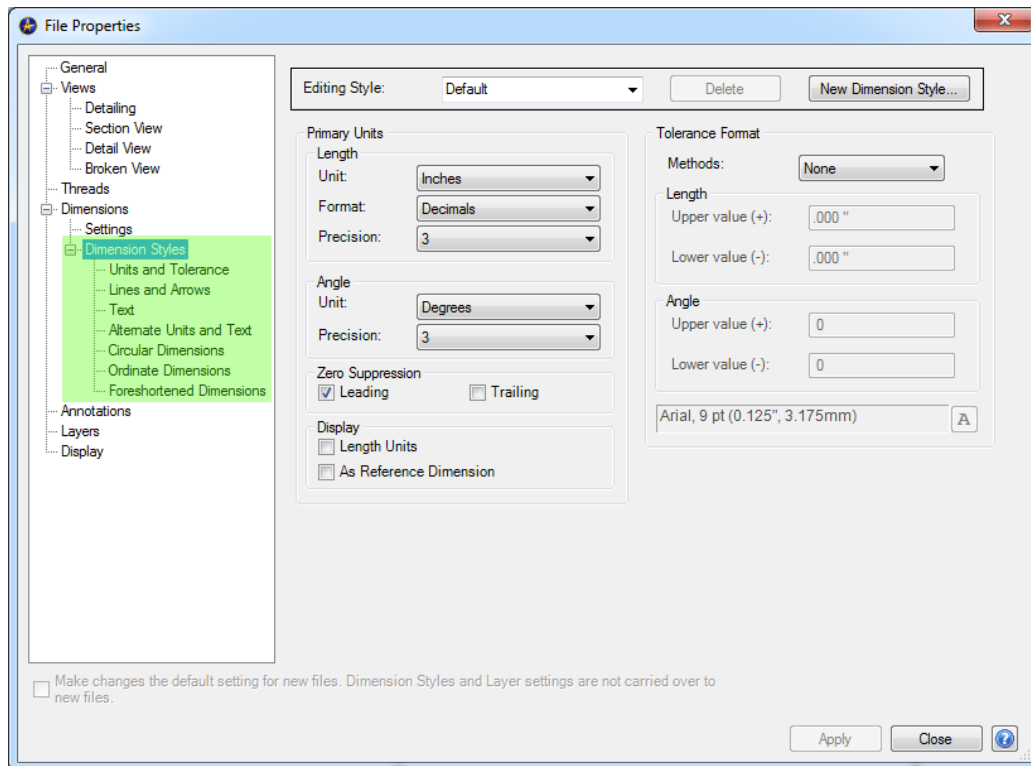
SET LAYER BY PART OR SUBASSEMBLY

You can now set the layer of parts, assemblies, or subassemblies in 2D drawings. Previously, you could only set the layer of the projected edges. This makes applying different layer styles to different parts in a drawing extremely fast. This is particularly useful when showing the progression of a sheet metal forming process and for highlighting important areas in a drawing.



DIMENSION STYLES USER INTERFACE

The Dimension Styles interface has been migrated into the File Properties dialog, accessible from the Gem Menu > Drawing Properties in a 2D drawing workspace.



GENERAL DIMENSION TOOL BEHAVIOR (2D DRAWINGS)

When using the Dimension tool in a 2D drawing workspace, the Sketching interface is no longer automatically invoked. Instead, you can use the Dimension tool while maintaining the Drawing Management tab of the Ribbon Interface. This means that you can quickly switch between applying dimensions and other annotations without having to switch tabs or exit Sketch Mode. Only applies to the Ribbon interface.

DIMENSION REPOSITION TOOL AVAILABLE IN SKETCH MODE (2D DRAWINGS)

The Dimension Reposition tool is now available in regular 2D Sketch Mode when sketching on the sheet. Previously it was only available for dimensions that had been placed on projected figures.

SPLINE & FONT USAGE

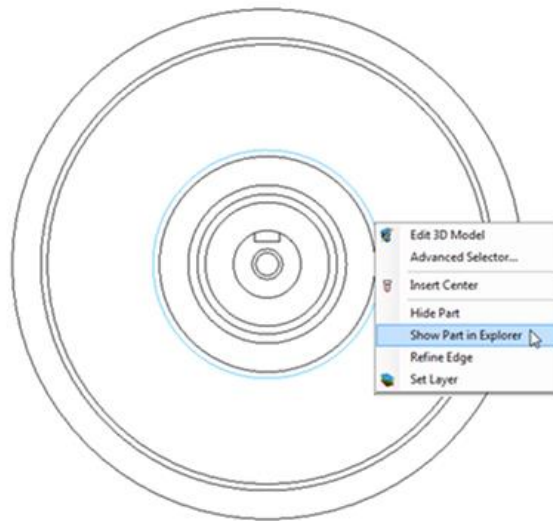
Several improvements have been made regarding imported splines from a DWG or DXF and regarding using splines related to the Text Tool. The improvements make handling particularly difficult splines or unconventional font definitions more robust and less likely to fail during 3D modeling operations.

CONSTRAINT INFERENCING WHILE SKETCHING IN 2D DRAWINGS

While sketching in 2D drawings, all constraints are now inferred as if you were drawing in a 3D workspace. Previously, certain constraints, such as Concentric, were not inferred automatically.

SHOW PART IN EXPLORER

The part associated with a projected edge can now be displayed in the Drawing Explorer by right clicking on the edge and selecting Show Part in Explorer:



DRAGGING MULTIPLE VIEWS

Multiple views can now be dragged simultaneously. Select the desired views while holding down Shift to multi-select them, and then drag to the desired location.

NEW DRAWING TEMPLATES

- ANSI Templates have been updated to add appropriate margins.
- ISO Templates have been redesigned to conform to standards. Official ISO templates and simplified ISO templates are now available.



Sample ISO Template



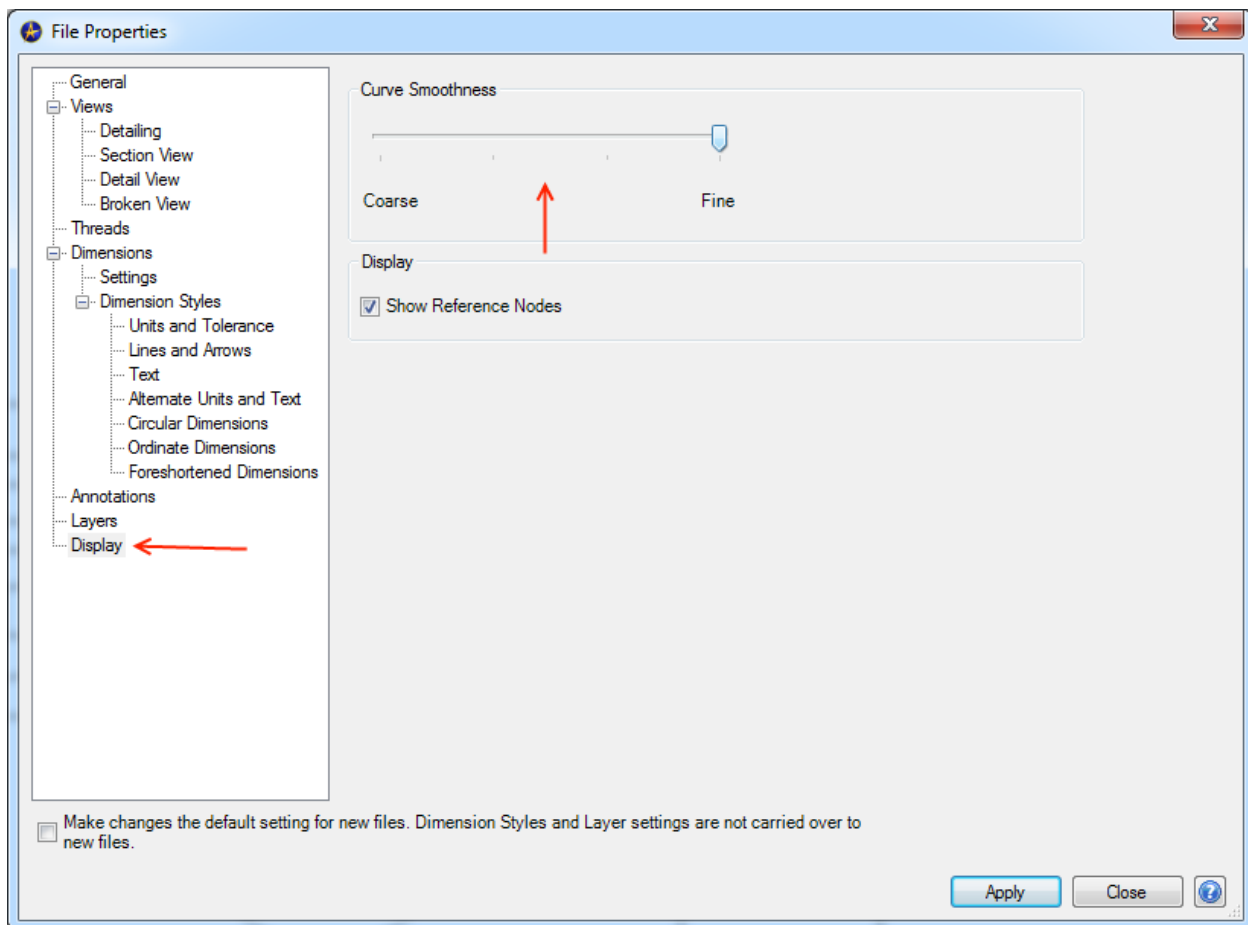
Sample Simplified ISO Template

2D DRAWINGS – VISUAL FIDELITY VIA ADAPTIVE FACETING

The visual fidelity of curved items in drawings has been significantly improved. Now several options exist in addition to fundamental algorithmic changes:

- Adaptive Faceting – automatic and always on:
 - Faceting has been significantly improved to consider the relative sizes of the 2D geometry during 2D drawing creation. The result is significantly improved visual fidelity.
- Adaptive Faceting slider option:
 - The Curve Smoothness value now allows you to select multiple settings for adaptive faceting. Setting higher values means the upper and lower bounds for curve smoothness will be set higher. Setting the value lower will lower both values.
 - It is important to note that not all drawings require the highest level of smoothing, especially if your drawing contains primarily linear figures. If you are suffering from performance problems, especially in very large or complex drawings, you may benefit from reducing the Curve Smoothness value.

To access the adaptive faceting options, select the Gem Menu > File Properties > Display tab from a 2D Drawing workspace:



2D DRAWINGS – PRINTING AND PRINT PREVIEW

PRINT PREVIEW FOR 2D DRAWINGS

Print Preview has been completely redone. It now has the following benefits:

- Significantly higher fidelity
- Opens full screen
- Interactive (easy zoom, pan, etc.)

2D PDF GENERATION

Publishing to PDF in a 2D Drawing will now automatically set the size of the PDF.

PRINTING

LINE WEIGHT AND SCALE

Improvements have been made to properly print line weight and scale.

PAGE SIZE SELECTION

When printing your paper size is automatically set to match the size of your sheet when possible.

MARGINS

Margins in templates are now upheld when printing.

PAGE DOES NOT FIT BEHAVIOR WHEN USING METRIC TEMPLATES

The Page Does Not Fit error now shows appropriately. Previously, templates defined in metric units sometimes caused the unnecessary display of this message.

2D DRAWINGS – DWG/DXF IMPORT/EXPORT

AREAS OF FOCUS

Significant improvements to DWG/DXF interoperability have been made regarding the accurate representation of various components of 2D documentation when importing or exporting. The following import and export-related areas are more robust:

- Feature Control Frames
- Word wrap
- BOM text
- Dimension lines with symmetric tolerances
- Dimension arrows, arrow heads, and section lines being correctly represented
- Various Text issues
- Various issues related to fractions
- Issues related to certain views coming in as solid when imported into Geomagic Design
- Font size or changes being incorrectly represented in external programs

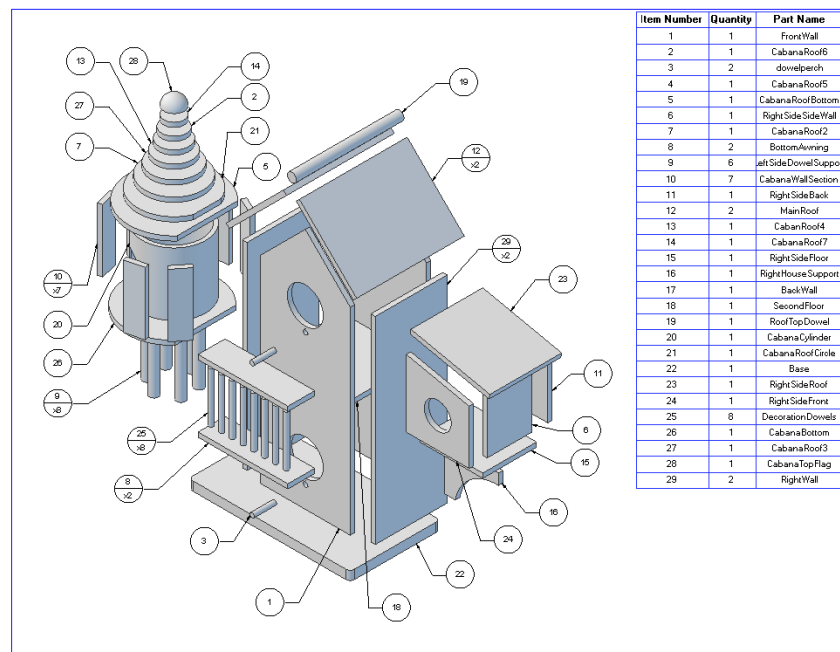
- Scaling of various font issues
- Issues related to Corel outputs
- Incorrect display of section view lines in some instances of imported DWG files
- Various issues related to French operating systems
- Issues related to imported hatches
- Issues related to scaling templates
- Layer support for tables when importing
- Layer color support

2D DRAWINGS – VIEWS

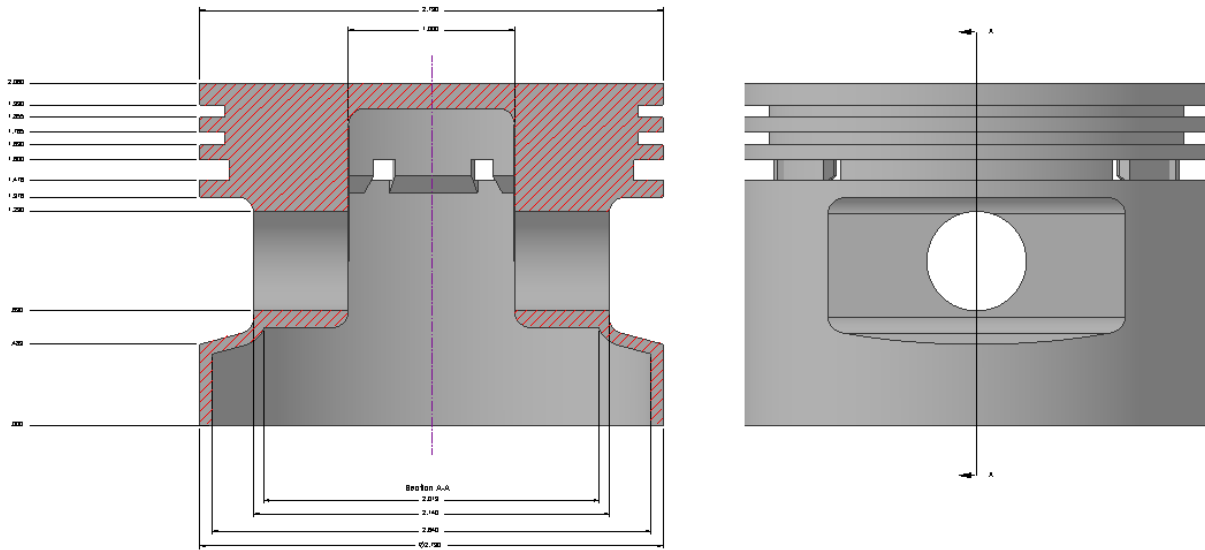
NEW VIEW TYPES

SHADED VIEWS

Shaded Views allow you to mimic the 3D visualization of components in a 2D Drawing.

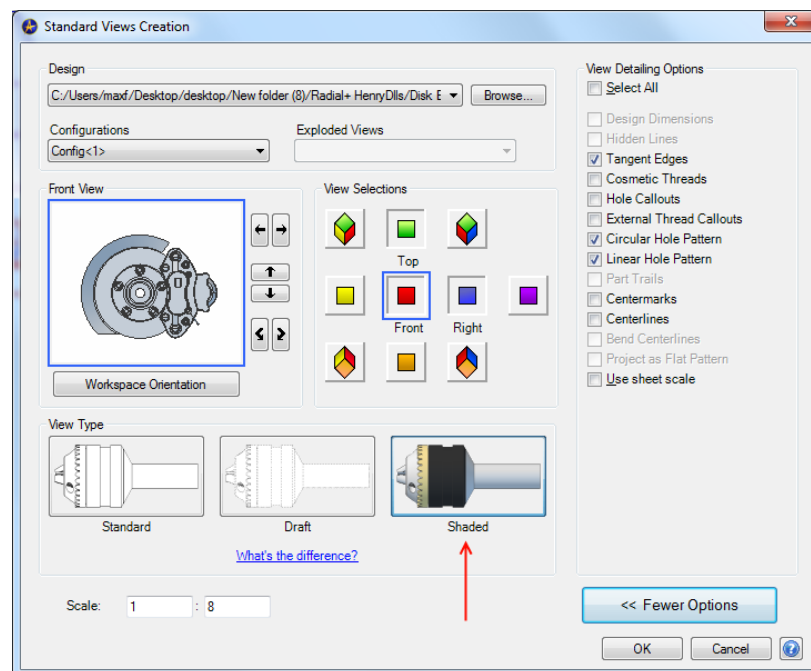


Shaded Views can significantly increase the understanding of engineering drawings and downstream applications such as marketing materials. They can also help to create buy-in from those less-familiar with traditional 2D documentation concepts as they are more readily understood by non-technical people. Shaded Views are available for all view types. For example, you can use Shaded Views to help illustrate a section drawing:

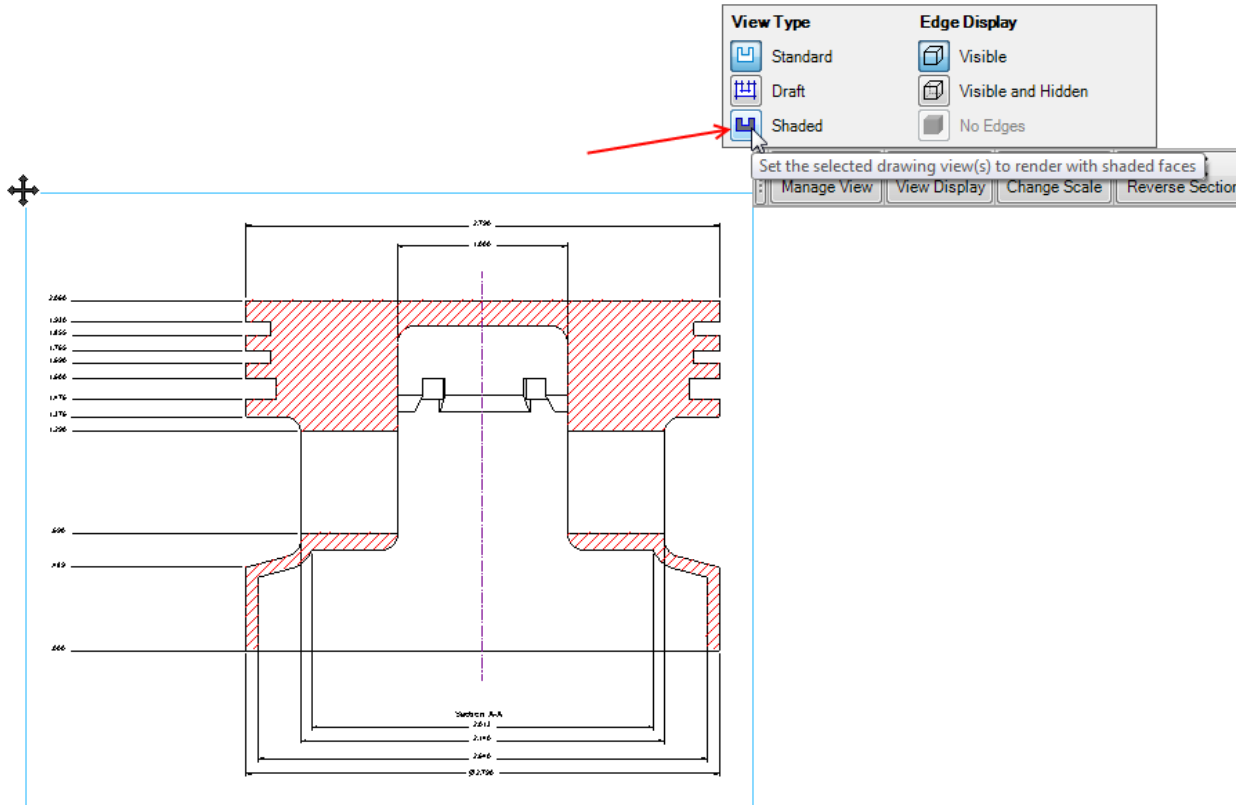


Create a Shaded View in the following ways:

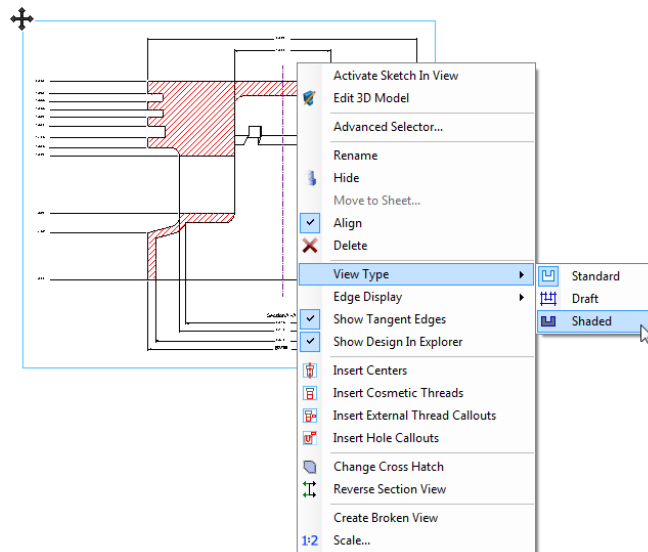
- During initial view creation in the Standard View Creation dialog, select Shaded View from the View Type area.



- Convert any existing view into a Shaded View via the popup menu by:
 - Left-clicking on the view to invoke the popup. Hover over the View Display area and select Shaded.



- Or right-clicking on any view and selecting **View Type > Shaded**:



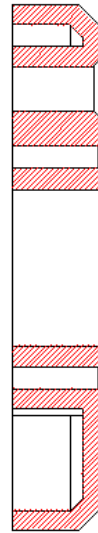
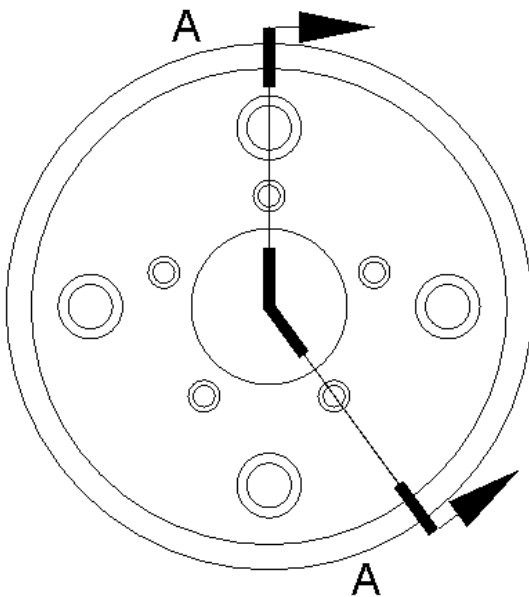
Tips:

- Shaded Views inherit their color from part and assembly colors.
- To change the color of a shaded view, edit the color of the part or assembly. Save the part or assembly and reproject the drawing.

- **Shaded Views have the same limitations as Fast Views. Check the Help for Fast View and Shaded View limitations to determine if you need to do any up-front planning based on your desired outcome.**

SECTION VIEW – ALIGNED

A new section view type, Aligned Section View, is now available from the Section View dropdown. It is used when the desired section lines are at an angle. Regular section views will cause certain features to appear foreshortened when the section definition contains figures at an angle. The Aligned Section view allows you to create a proper section without foreshortening.



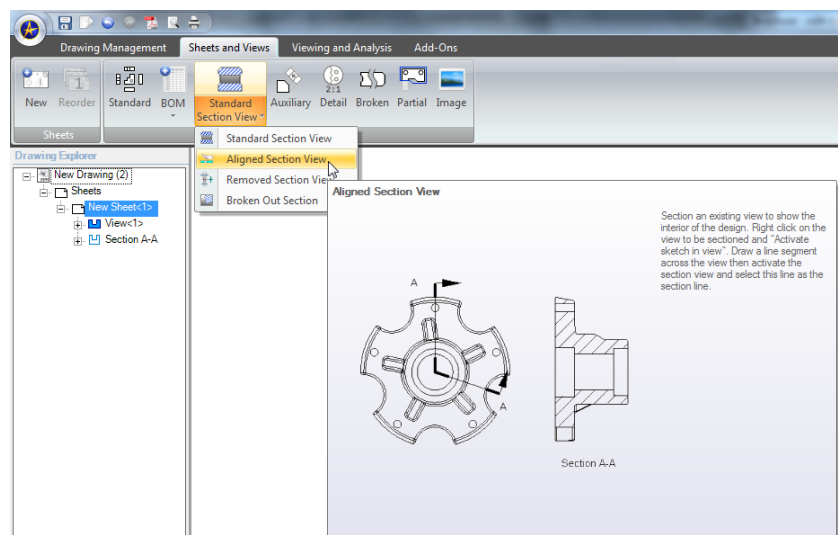
Section A-A

Aligned Section View

Section A-A

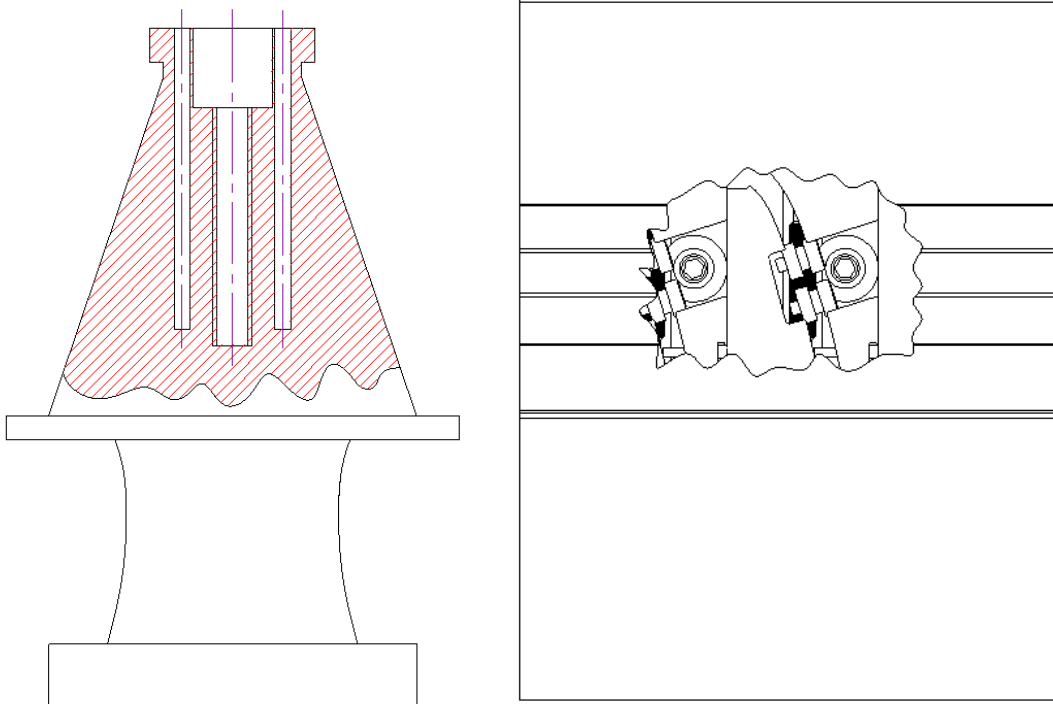
Standard Section View

To access the Aligned Section view, select the Sheets and Views tab in a 2D drawing and then select Aligned Section View from the section dropdown:

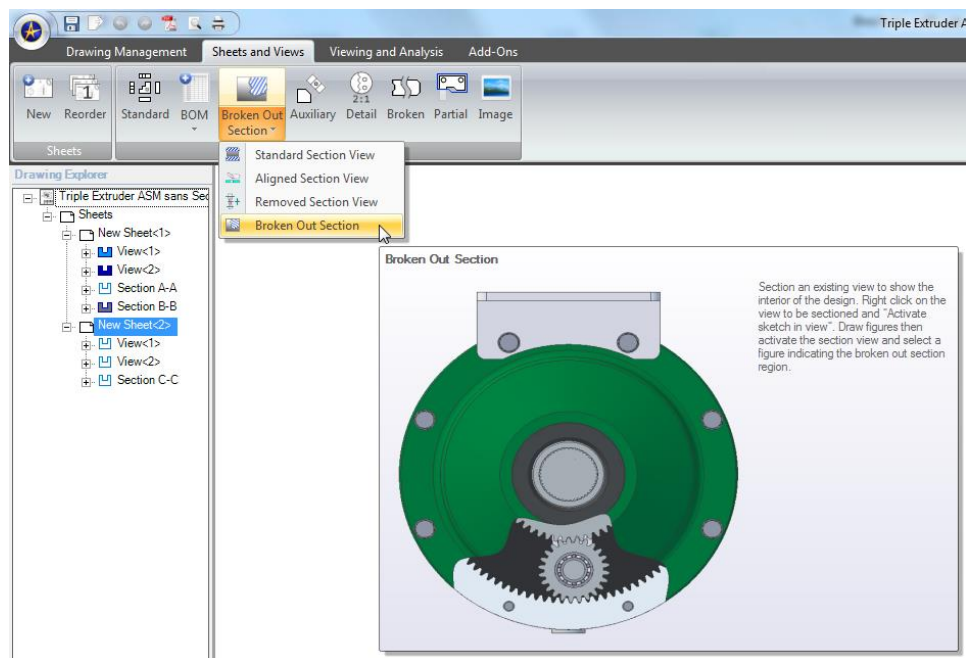


SECTION VIEW – BROKEN OUT

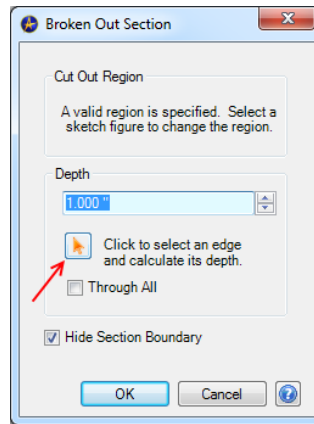
Broken Out Section Views act similarly to an Extrude Cut and are useful for showing the occluded interiors of objects.



To access the Broken Out Section View, select the Sheets and Views tab in a 2D drawing and then select Broken Out Section from the section dropdown:

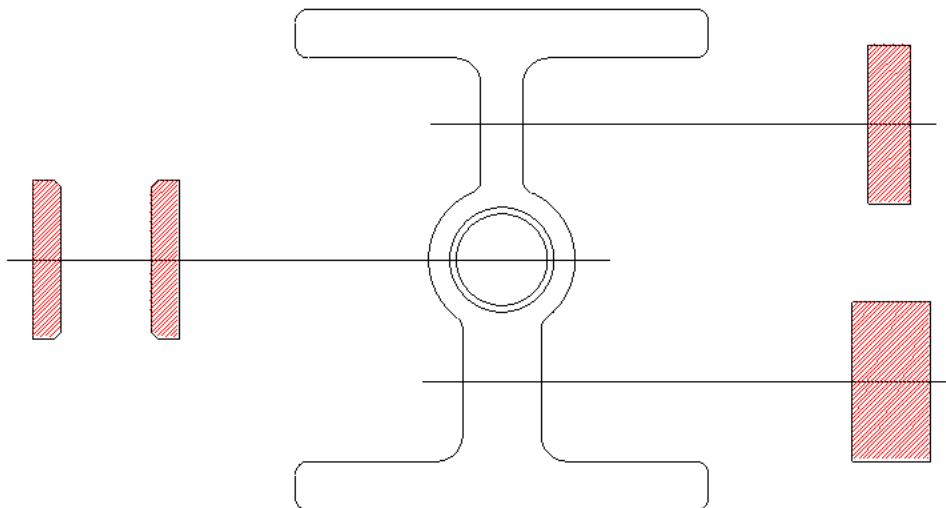


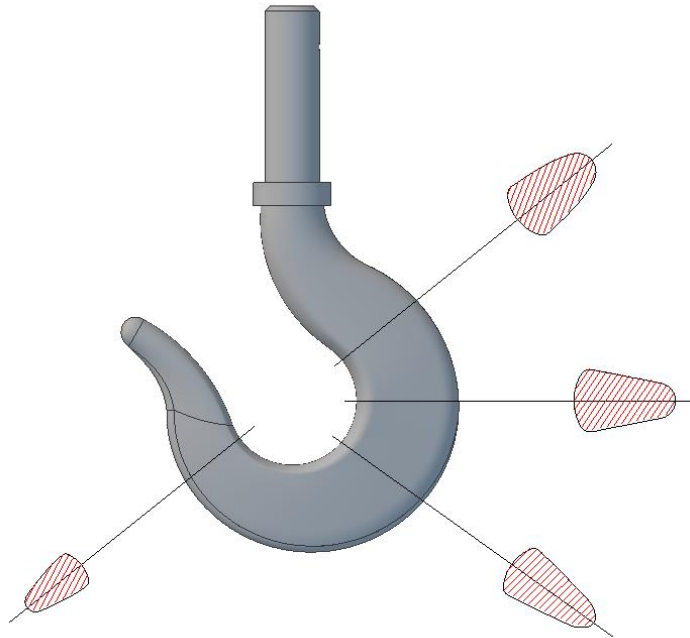
Tip: You can define the depth of the Broken Out Section by selecting a projected edge that represents the depth you desire. You can select this edge from any view. Often, you will select this edge from a view that is rotated 90 degrees.



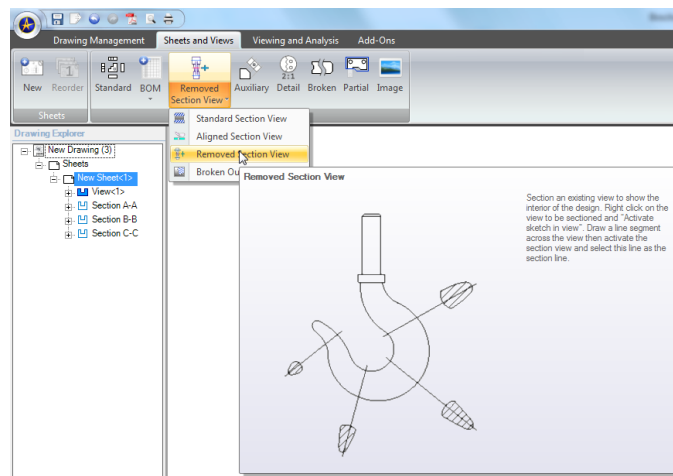
SECTION VIEW – REMOVED

Removed Section Views will create a section view that only displays geometry that lies directly on the cutting plane. This is useful for showing localized detail when a regular section view would create unnecessary geometry that complicates the desired detailing.



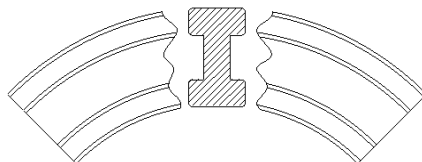


To access the Removed Section view, select the Sheets and Views tab in a 2D drawing and then select Removed Section View from the section dropdown:



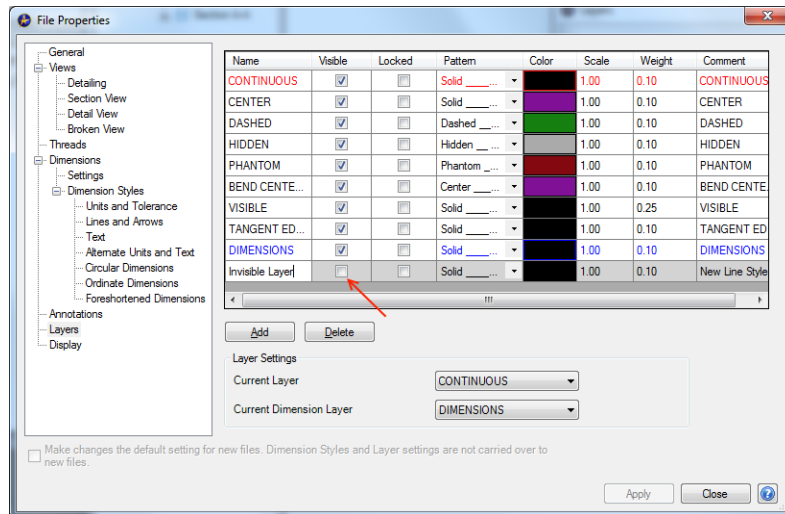
SECTION VIEW – REVOLVED

Revolved Section Views are not an explicit command but can be accomplished by combining a Broken Out Section view and a Removed Section view.

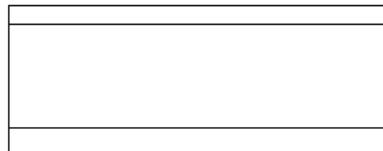


To create a Revolved Section view, follow these steps:

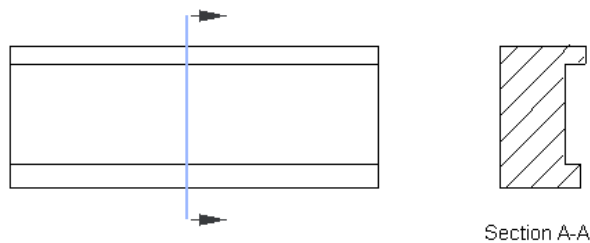
1. Create a new layer type that has the **Visible** checkbox unchecked. This will create a layer that allows you to hide things from the drawing:



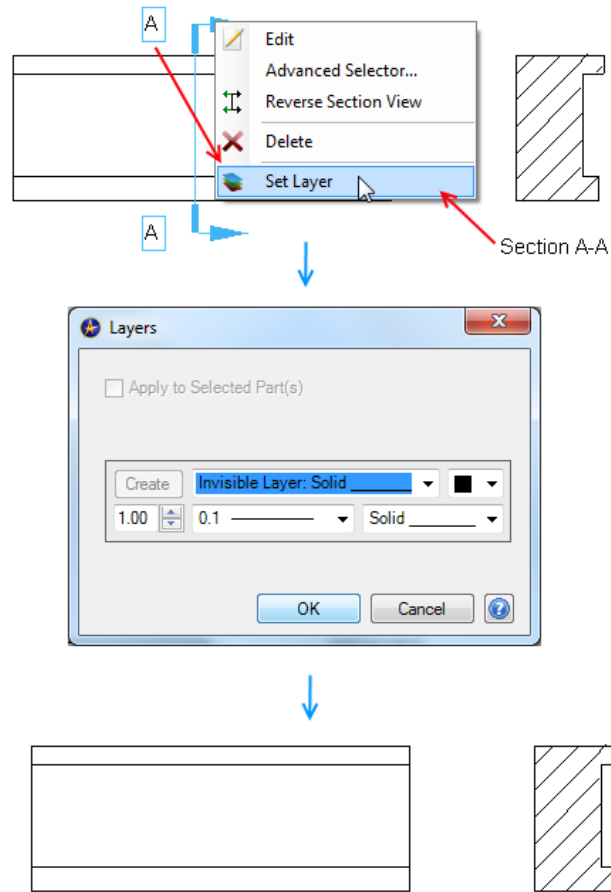
2. Project a view



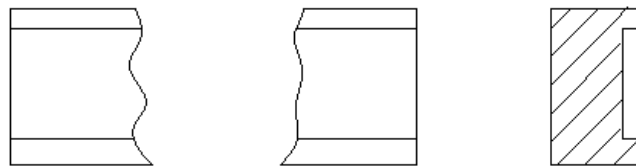
3. Create a Removed Section view



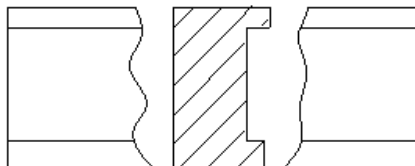
4. Set the Section Line and Section Label layers to invisible by right clicking on them, selecting Set Layer, and selecting the invisible layer you created in step 1.



- Optional: Create a Broken Out Section view that overlaps the Removed Section view with the **Through All** option and **Hide Boundary** options selected



- Move the Removed Section view you created in step 3 into the Broken Out section view you optionally created in step 5.

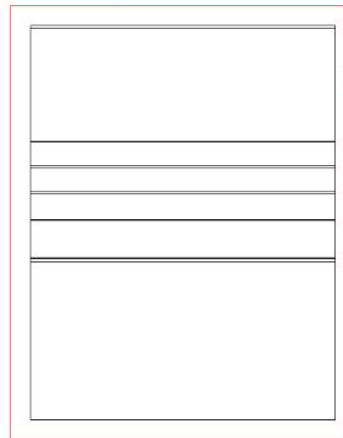


VIEW CREATION WORKFLOW

A new workflow possibility now exists when creating view types that rely on your defined geometry, such as a section view relying on a section line. Previously, you had to Activate Sketch on a view, create the geometry, and then invoke a view tool and select the geometry you made. Now you can select which type of view you want to make first, select the target view, and automatically start drawing the geometry.

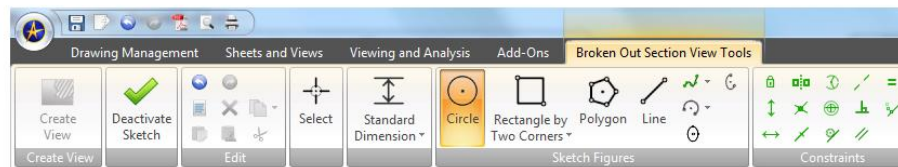


Select the desired view

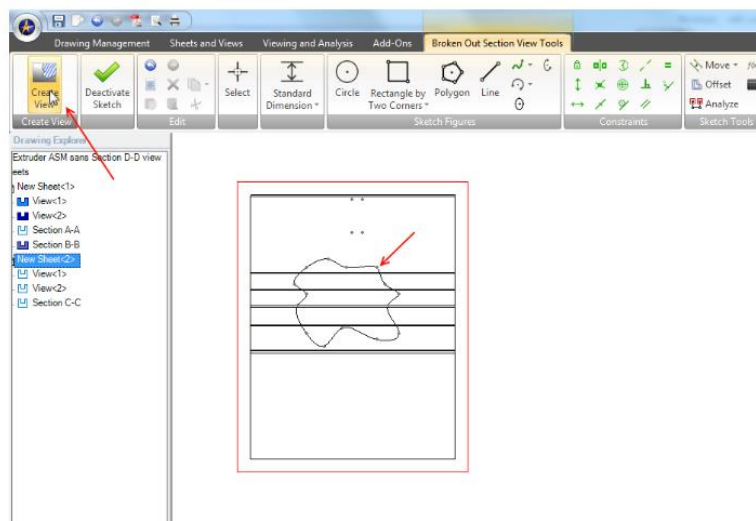


Select an existing sketch or select a view to create a new sketch.

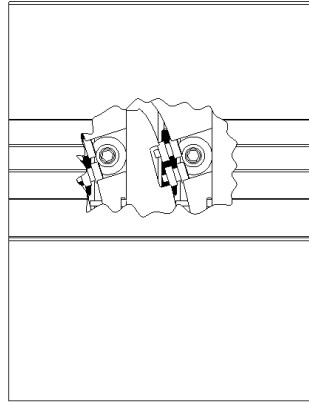
You are prompted to select the base view for the new view.



The sketching tools open up automatically.

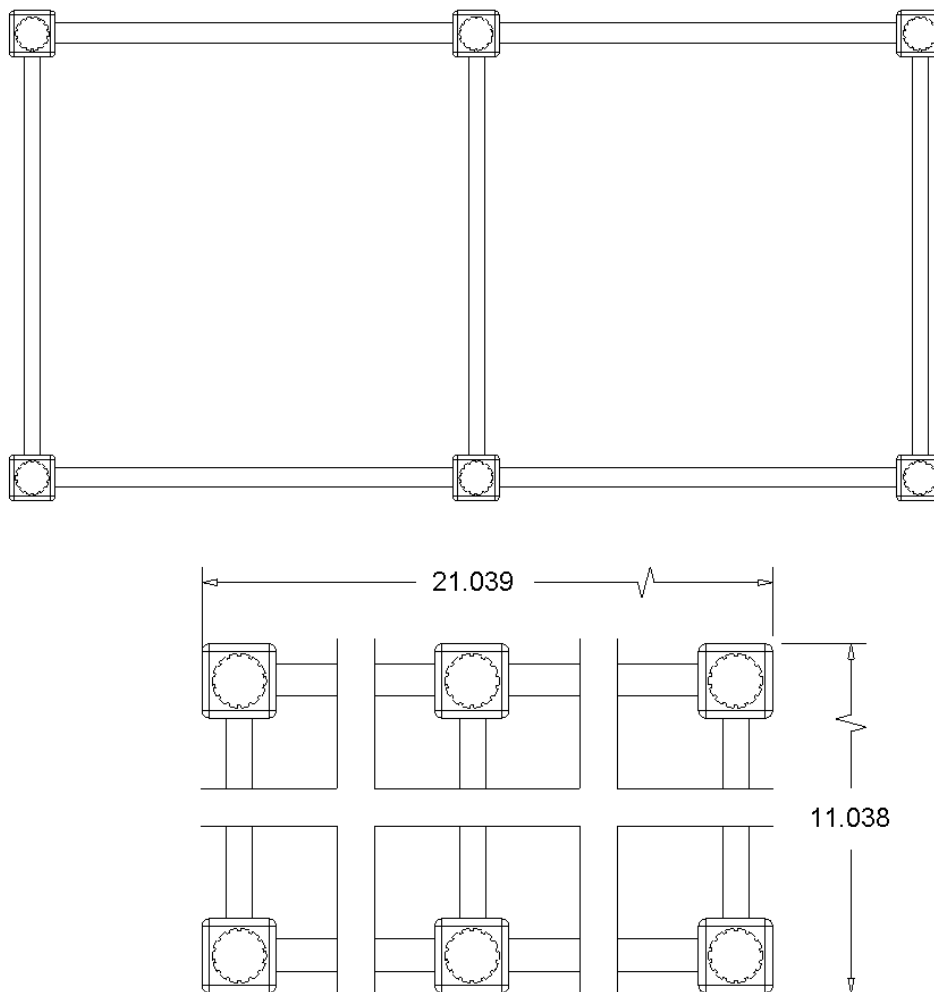


Create the desired sketch geometry.
Select Create View and the view is created.



ADDITIONS TO BROKEN VIEWS

Multiple Breaks can be applied to a single view. Overlapping breaks are supported (vertical break + horizontal break). Break line extents are now sized to the model rather than the view bounding box.

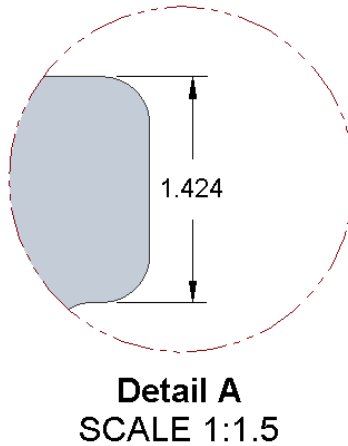


ASSOCIATIVE VIEW LABELS

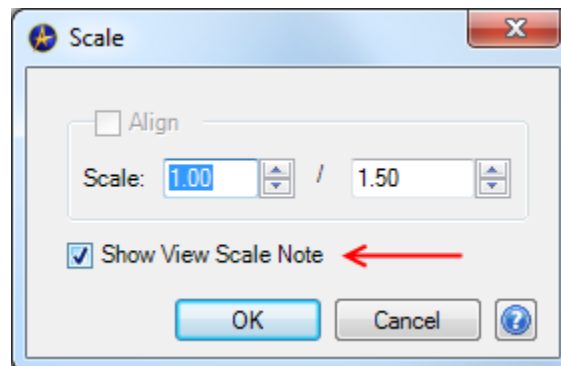
Detail and Section view labels are now associated between the view and the annotation. Changing the name for one will automatically change the name for the other.

ADDING SCALE LABELS TO VIEWS

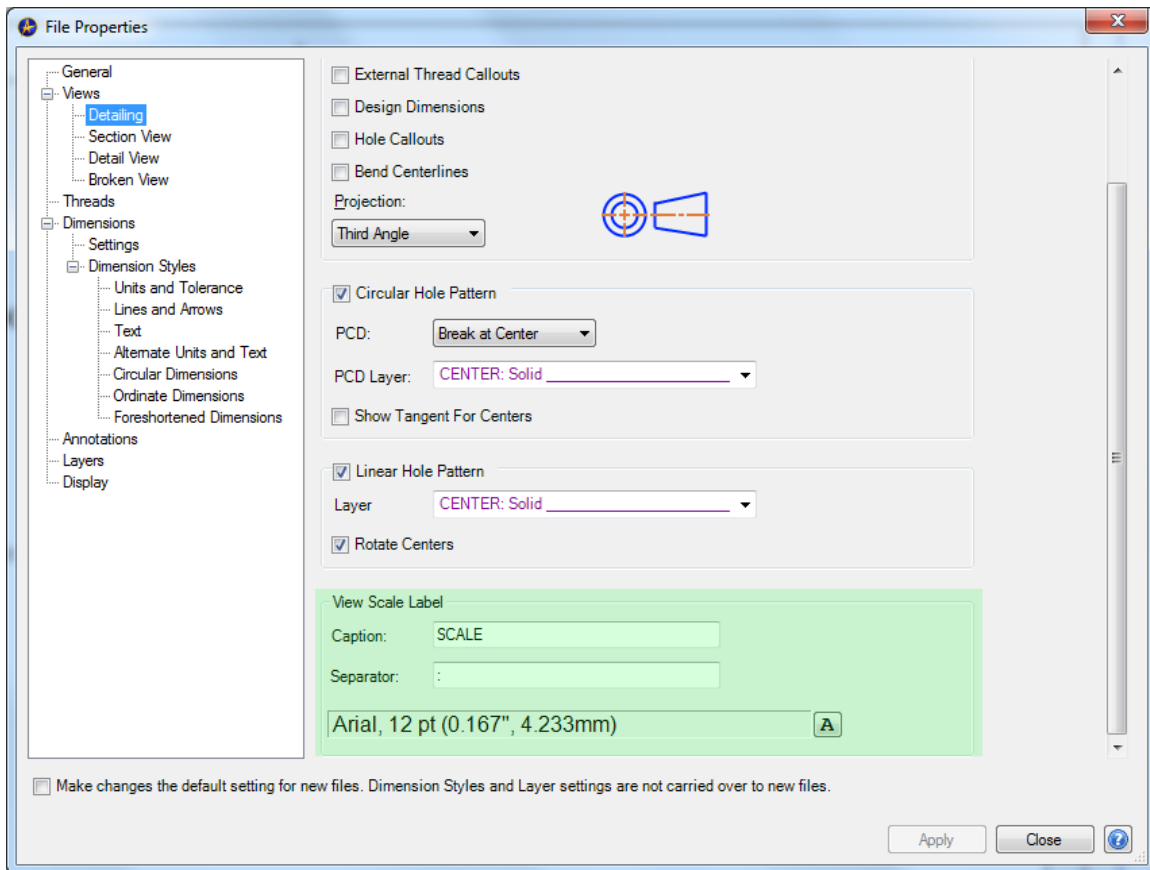
You can add a Scale value to any view that will update as the view scale changes:



To add a scale label, right click on the view and select Scale. Check the Show View Scale Note box:



You can change the way Scale information is presented the Drawing Properties dialog:



2D DRAWINGS – ANNOTATIONS

GENERAL

WEIGHTS

Annotations now support line weights.

LAYER SUPPORT FOR TEMPLATES AND CUSTOM SYMBOLS

Templates and Custom Symbols now fully support layers and line weights.

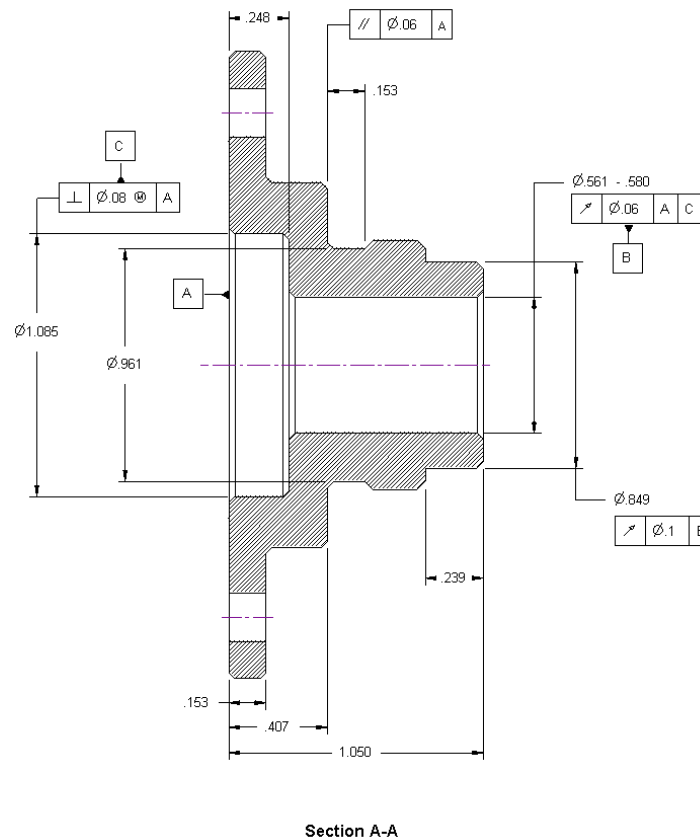
Templates and custom symbols retain their original layers when inserted into a new drawing. Previously, all templates and custom symbols would be set to the continuous layer, and changing the layer attributes of an individual component of a template or custom symbol required the Explode command. Now, you can create templates and custom symbols with any variety of existing or custom layers and reuse them exactly as they are.

LAYER SUPPORT FOR TABLES

Tables can now be set to an existing or custom layer by right clicking on the table and selecting the Set Layer command.

DATUM, DATUM TARGET, AND FEATURE CONTROL FRAME ATTACHMENTS

Now you can attach Datums, Datum Targets, and Feature Control Frames to Dimension Leaders and Feature Control Frames. The attachments are associative and will move when the original item is moved.

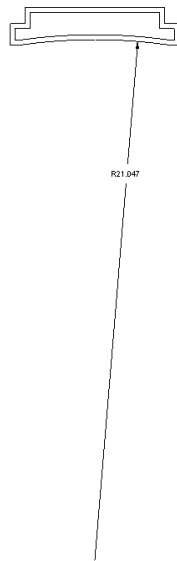


When placing a Datum, Datum Target, or FCF hover over a dimension or FCF until it highlights. Clicking in this position will attach the leader.

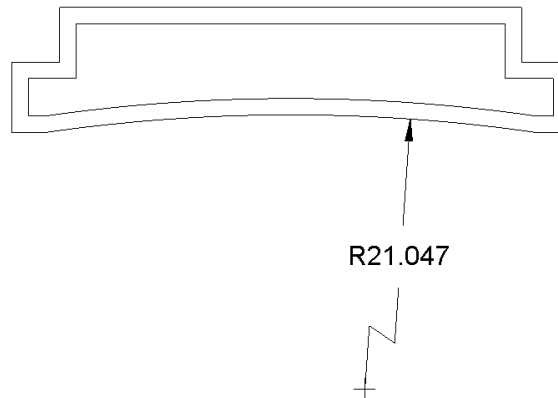
DIMENSIONS

FORESHORTENED DIMENSIONS

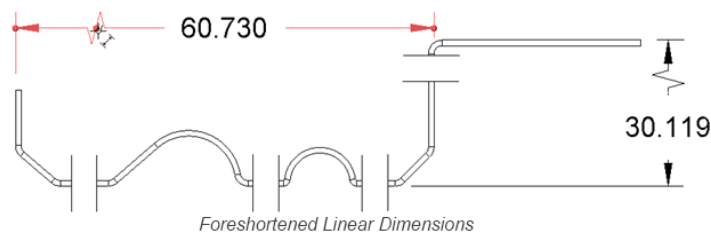
It is now possible to foreshorten a linear or circular dimension. Configuration options are available from Dimension Styles. You can change the position of the jag for linear dimensions by hovering over it and dragging the node. You can change the position of the end-point (and subsequently the jag) of a radial dimension by hovering over it and dragging it.



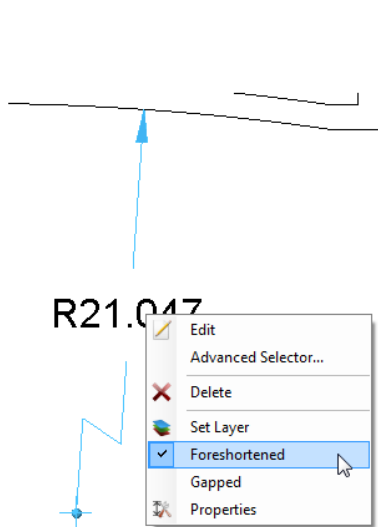
Regular Radial Dimension



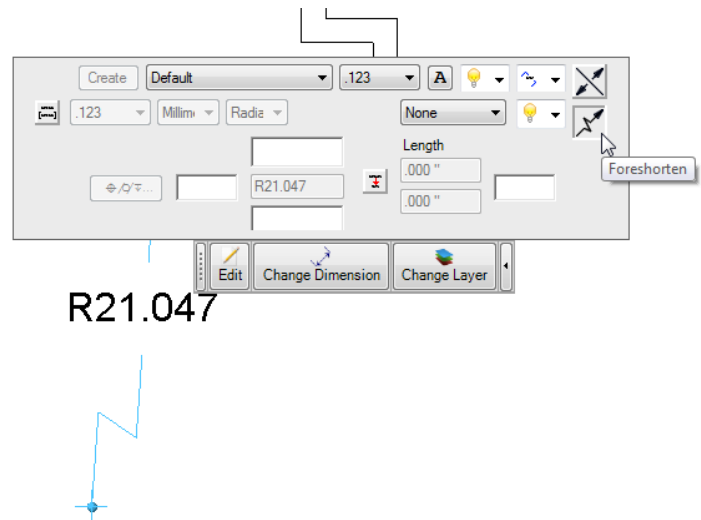
Foreshortened Radial Dimension



To foreshorten a dimension or dimensions, use the right-click menu or the in-place editing menus:

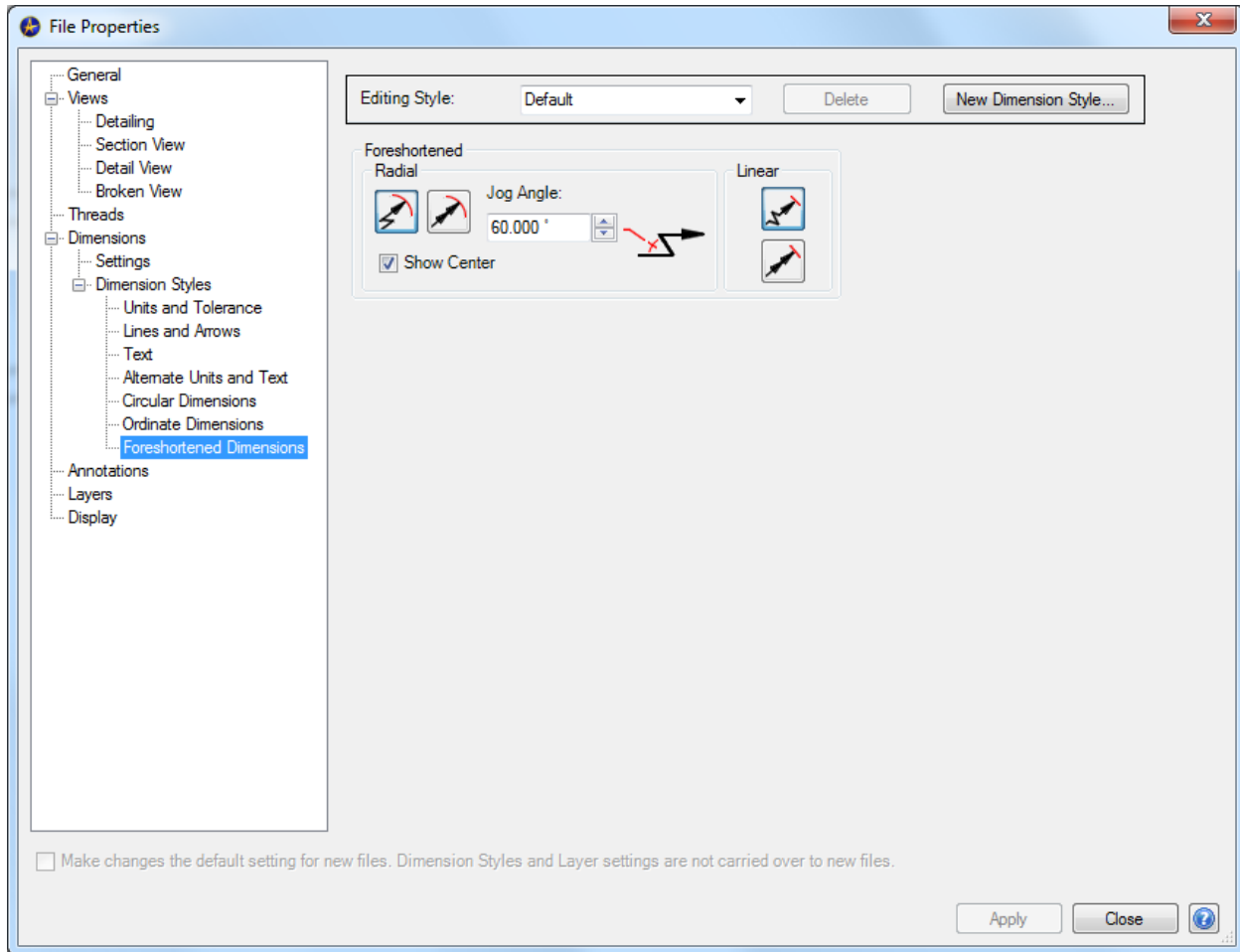


Foreshortening a dimension from the right-click menu



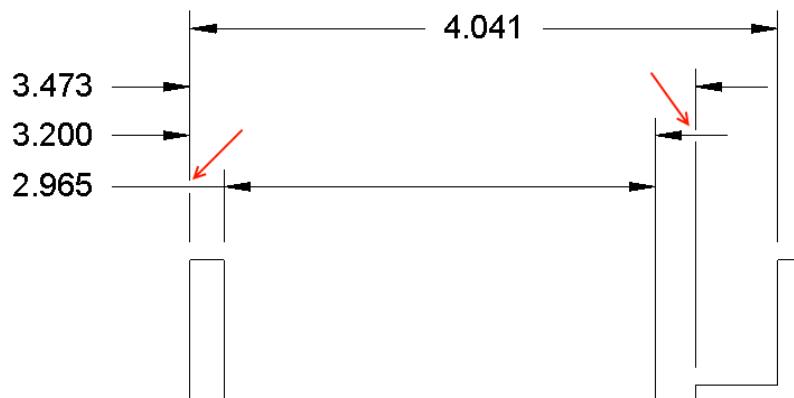
Using the in-place editing menu to select the foreshorten option

To change the appearance of foreshortened dimensions, access the Foreshortened Dimensions pane of the Drawing Properties dialog:



DIMENSION GAPS

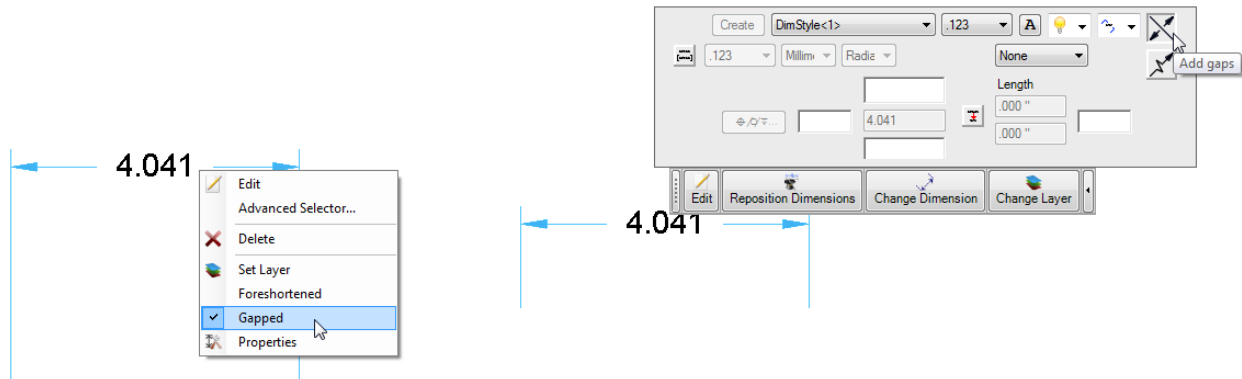
You can set dimensions to be gapped such that when they cross other dimensions a gap will automatically be placed at the intersection point.



Geomagic Design™

What's New

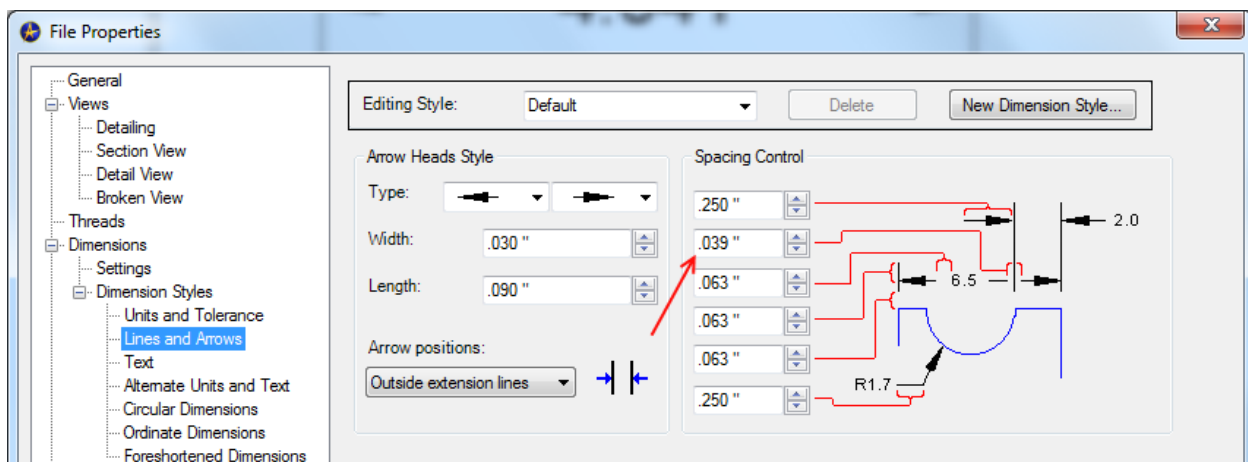
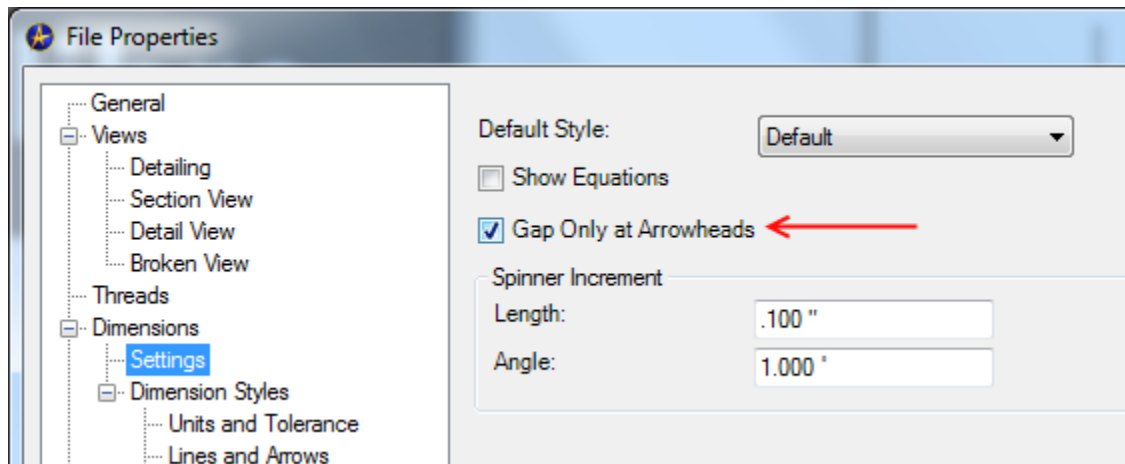
To designate a dimension as a gapped dimension, use the right-click menu or the in-place editing menu:



Adding gaps to a dimension via the right click menu

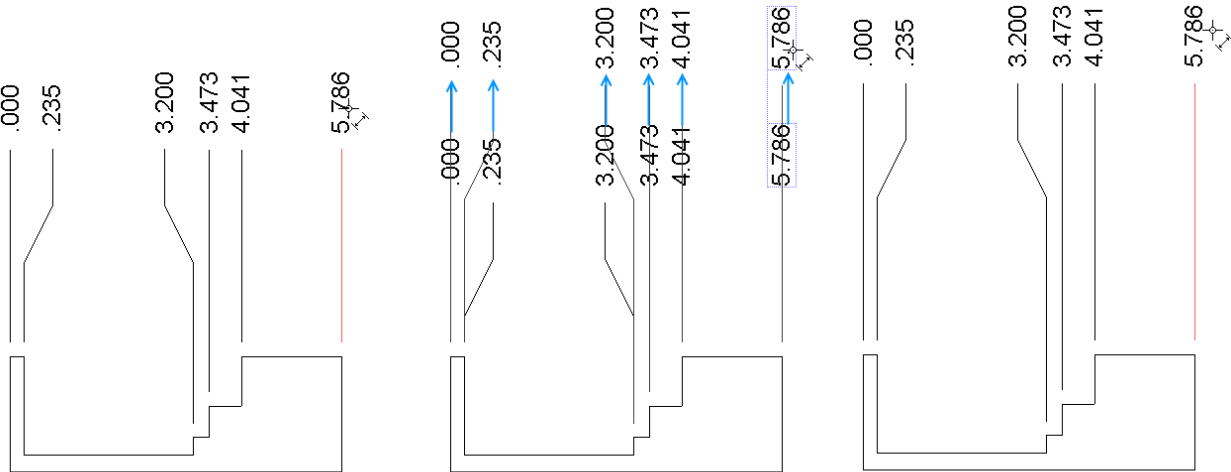
Adding gaps to a dimension via the in-place editing menu

There are two options for gapped dimensions available from the Drawing Properties dialog: **Width** and **Gap Only at Arrowheads** at Arrowheads:

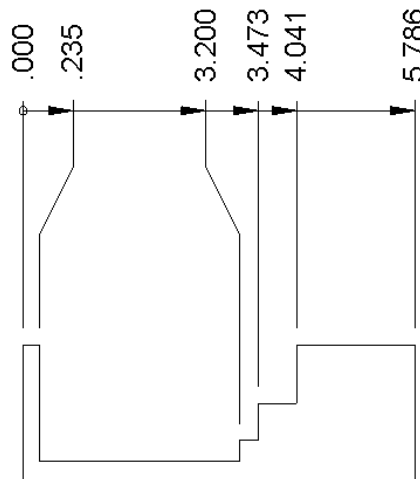


ORDINATE DIMENSIONS

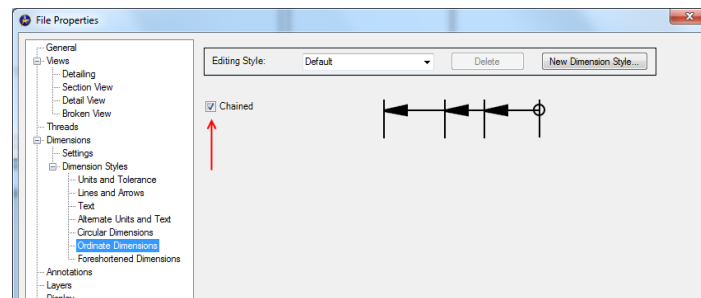
Ordinate dimensions are now easier to move. They automatically align and when you move one, the rest move in unison.



Chained Ordinate Dimensions are now supported:

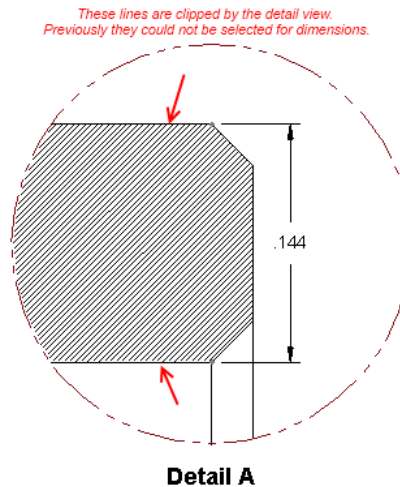


To set ordinate dimensions to be chained, select the option from the Drawing Properties dialog:



DIMENSIONING CLIPPED FIGURES

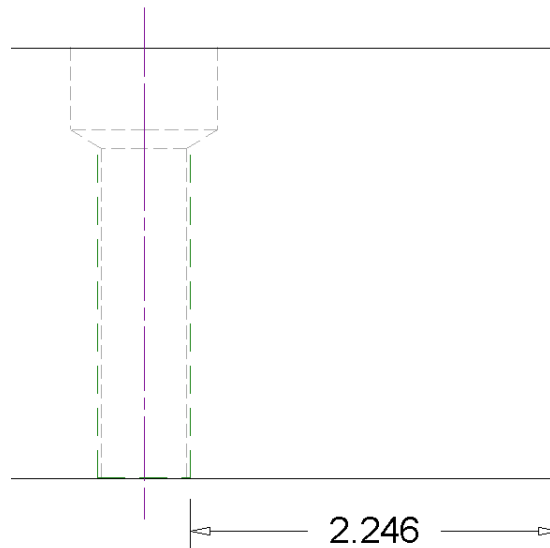
Previously when figures were clipped in certain view types, for example partial and detail views, they could not be dimensioned and you would be required to select their end nodes instead. Now all figures can be dimensioned regardless of their clipped status.



THREADS

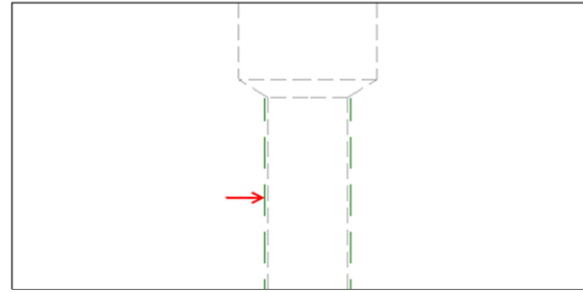
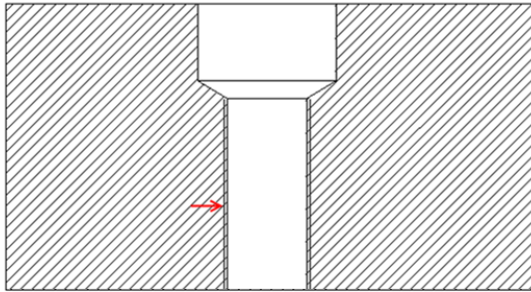
ABILITY TO USE COSMETIC THREADS FOR DIMENSIONING

You can now select the cosmetic thread lines in 2D drawings for the placement of dimensions:



VISIBLE AND OCCLUDED HOLE THREAD REPRESENTATION

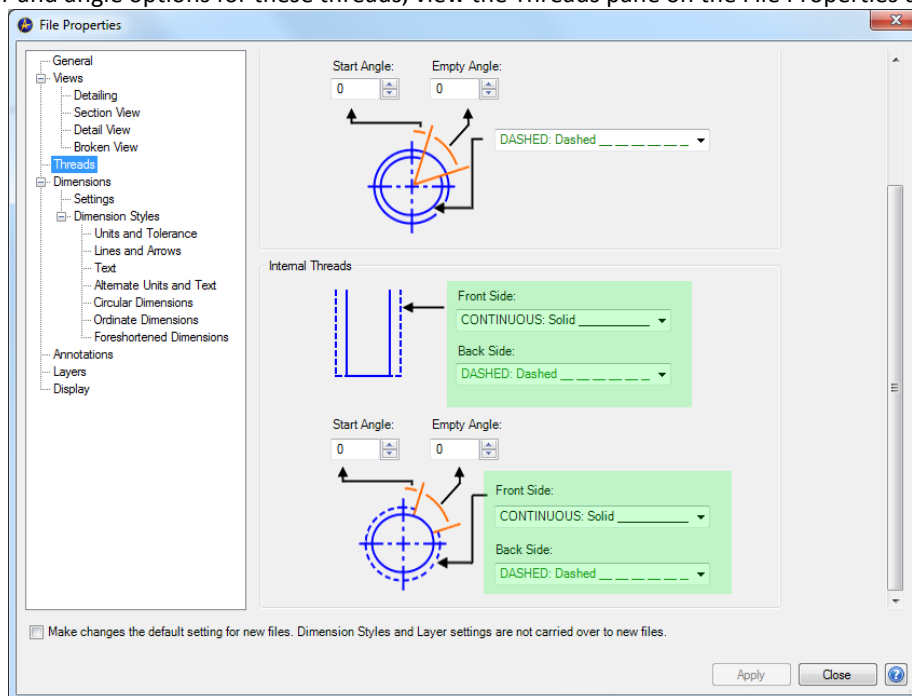
You can set the layer of the thread representation for internal cosmetic threads based on whether the threads are occluded or not (for example, in a section view):



Thread representation in section and non-section views can be set to different layers
Similar options exist for top and bottom views.

Section A-A

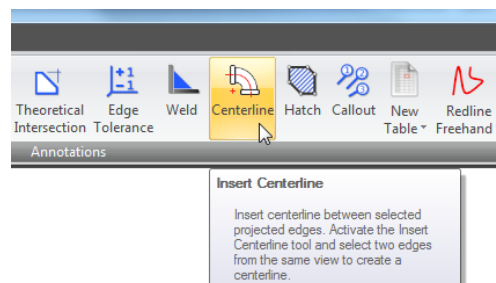
To set the layer and angle options for these threads, view the Threads pane on the File Properties dialog:



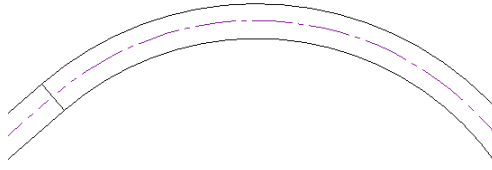
CENTERLINES

MANUAL INSERTION FOR LINEAR AND CIRCULAR GEOMETRY

The Insert Center tool is available from the Drawing Management tab of the Ribbon:



This allows you to manually insert centerlines between two linear or two circular edges regardless of the source geometry.



OVERLAPPING CENTERLINES

Centerlines that overlap linearly are now consolidated into a single centerline.

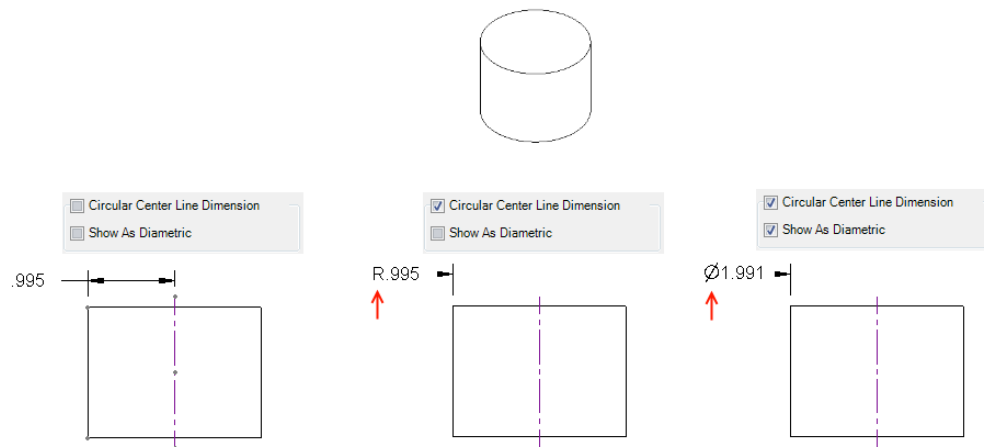
DRAGGABLE CENTERLINES

Centerlines now have draggable nodes. To resize the centerline, select a node and drag it.

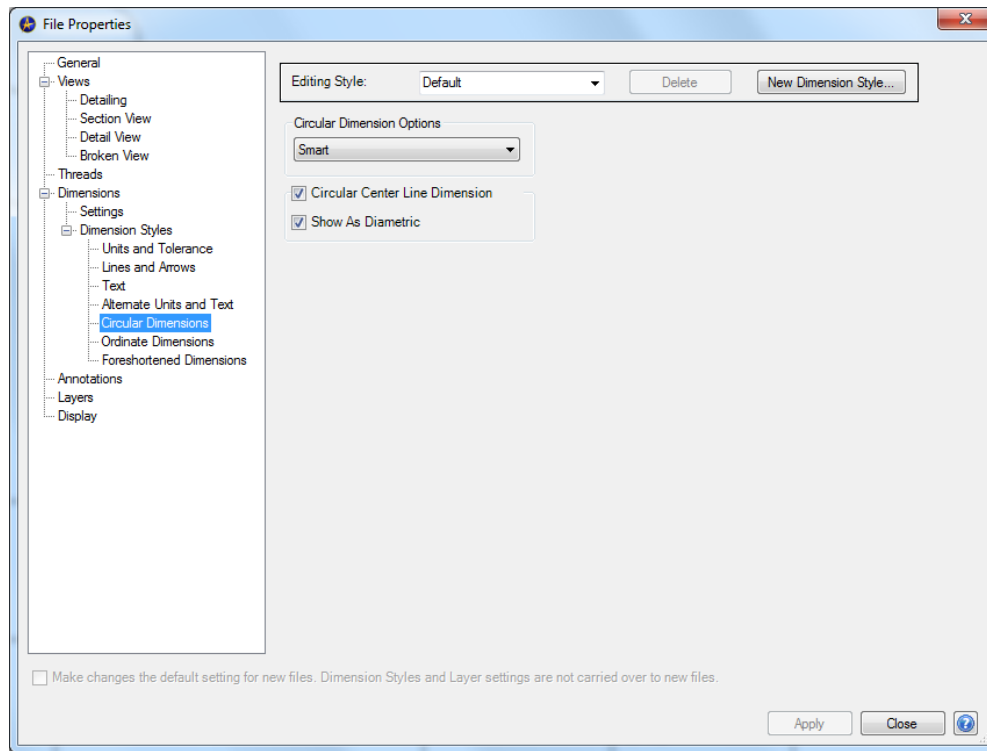


INTELLIGENT DIMENSIONING TO CENTERLINES OF CIRCULAR OBJECTS

When dimensioning to the centerline of a circular feature, several options exist on how to treat the dimension:

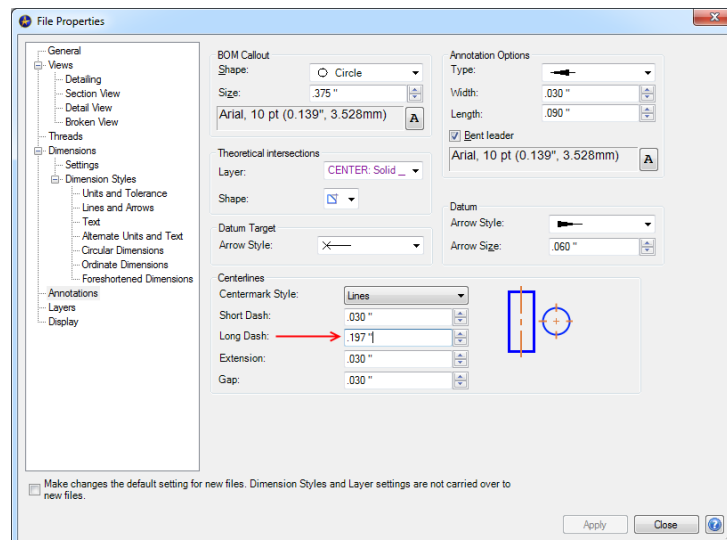


To access the options, select the Circular Dimensions pane from the Drawing Properties dialog:



ADDITIONAL SPACING OPTIONS

A new Long Dash option is available on the Drawing Properties > Annotations pane:



DIMENSIONING CENTER MARKS

Now dimensions can be placed between the central nodes of center marks. This is useful in cases where the rest of the center mark is not oriented in a horizontal/vertical position, for example when they are rotated in a PCD layout.

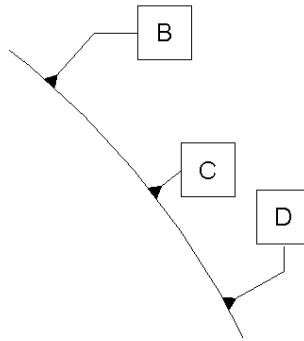
DATUMS

MULTIPLE DATUMS WITH SAME LABEL

Datums can now be created that have the same label as a previously created Datum on a single sheet or across multiple sheets.

DATUM SNAPPING

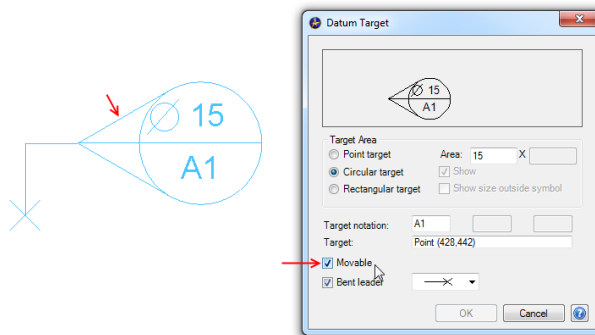
Datum triangles now snap to selected geometry at the correct angle. Smart Anchoring applies to datums so the leader behavior is correct. Snapping such that the leader is directly horizontal or vertical to the datum is automatic.



DATUM TARGETS

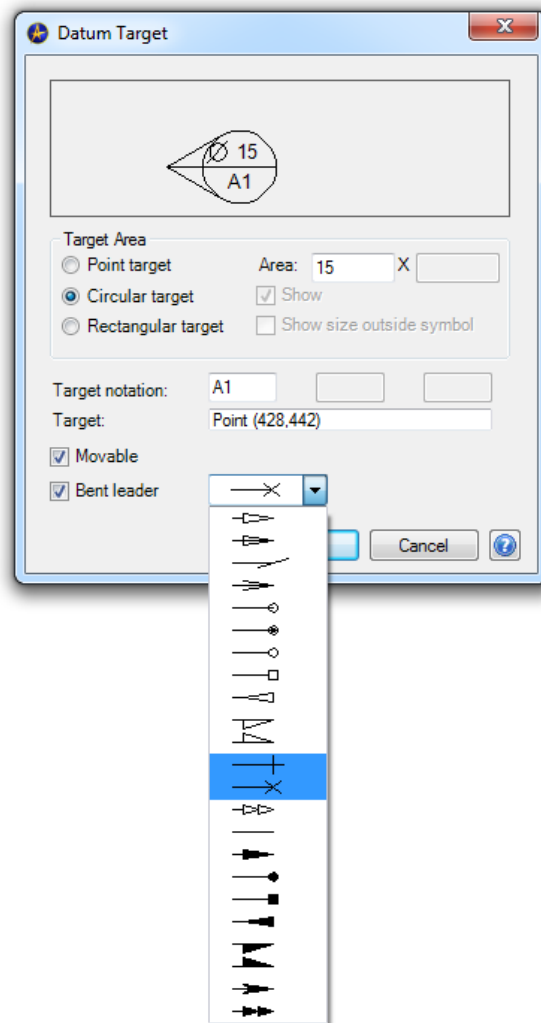
MOVABLE OPTION

A new option to create a movable datum target is available:



NEW ARROWHEAD TYPES

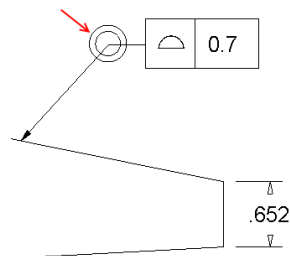
The Cross and X are now available:



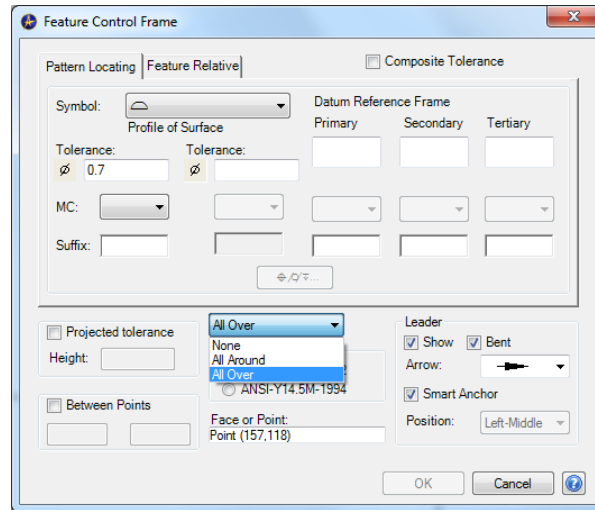
FEATURE CONTROL FRAMES

ALL OVER LEADER SYMBOL

The All Over symbol is now supported for feature control frame leader lines:

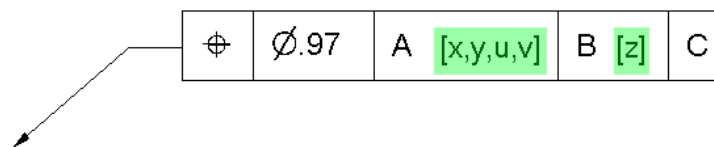
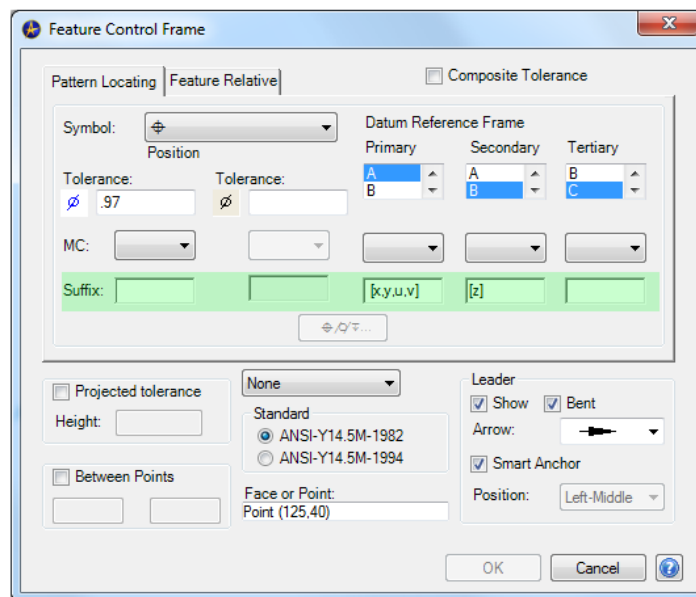


This is accessible from the Feature Control Frame dialog:



CUSTOM TEXT IN FEATURE CONTROL FRAMES

Custom text can now be entered in the Feature Control Frame dialog:



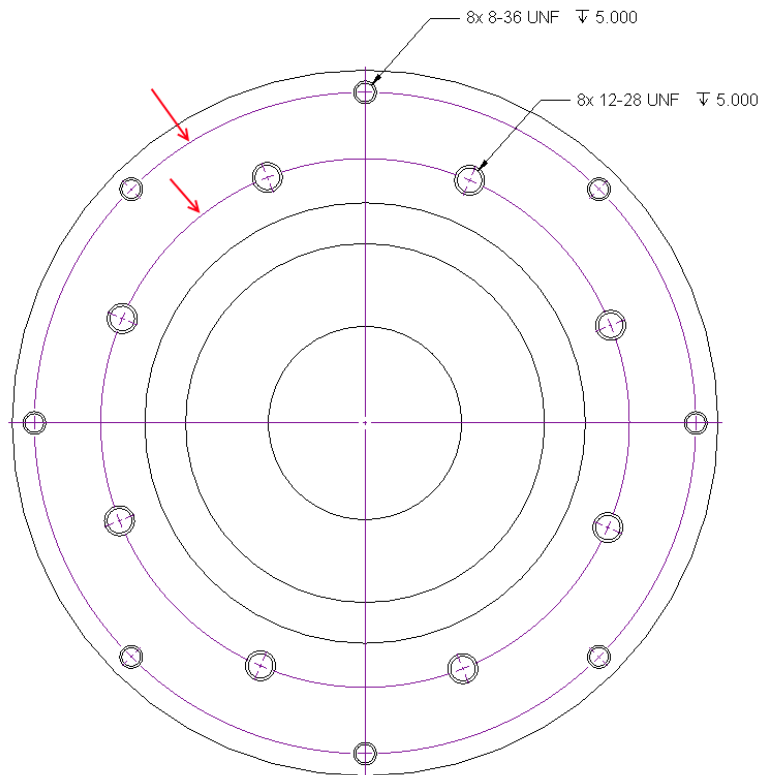
HOLE PATTERNS

CALLOUTS

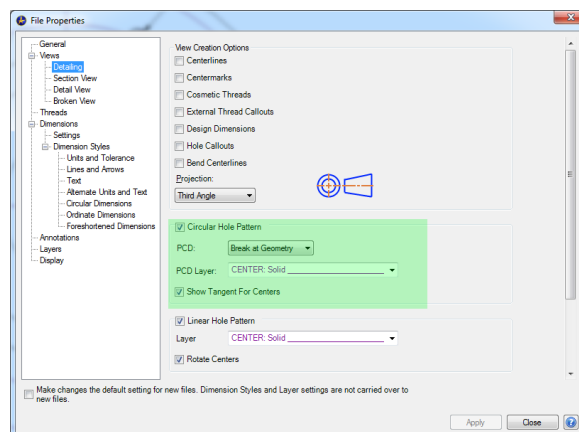
When using a Pattern or Mirror operation on a 3D Hole feature, the 2D callout now reflects the total number of holes properly.

PCD

If you use a Circular Pattern on a 3D Hole feature, you can now project Centers in a 2D drawing with PCD lines. In this case, the centers of the holes are also rotated automatically.



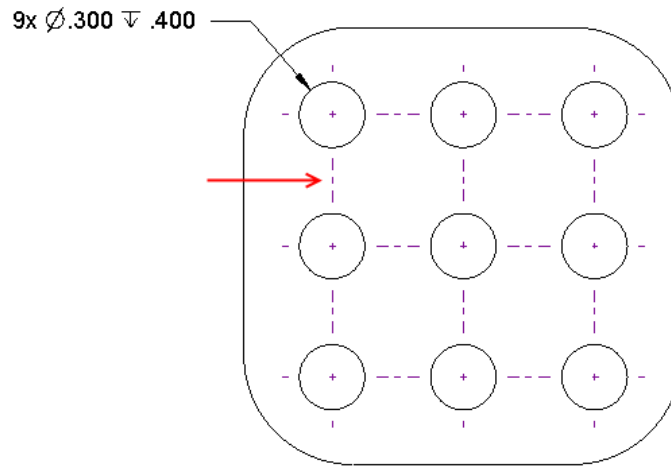
PCD options are located in the Document Properties on the Detailing pane:



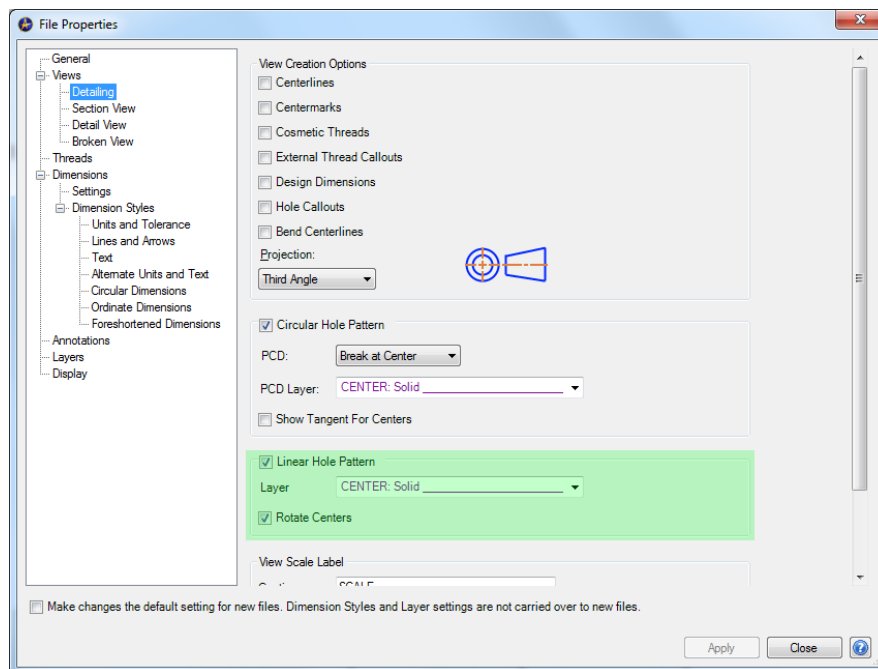
You can disable PCD lines by unchecking the Circular Hole Pattern option. You can also adjust the layer and PCD type from this dialog.

LINEAR HOLE PATTERN ANNOTATIONS

When creating an 2D drawing of a part that has a 3D Hole feature that uses a linear pattern, the pattern lines will be created when projecting Centers.

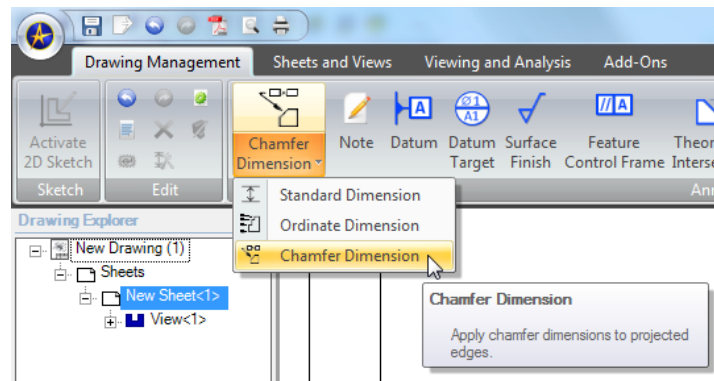


Linear hole pattern options are available from the Drawing Properties > Detailing pane:

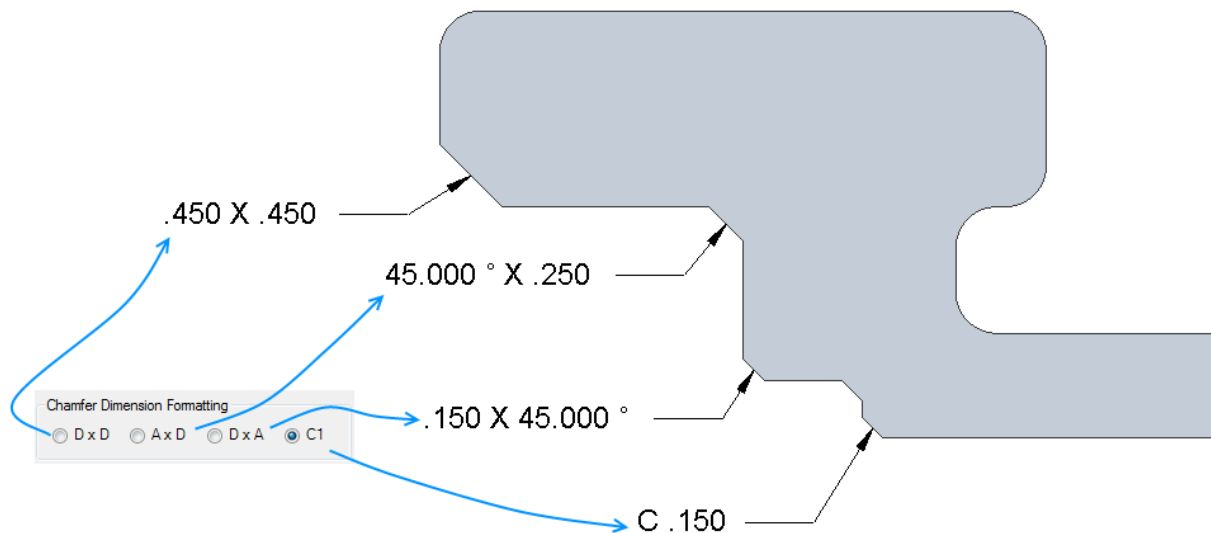


CHAMFER CALLOUTS

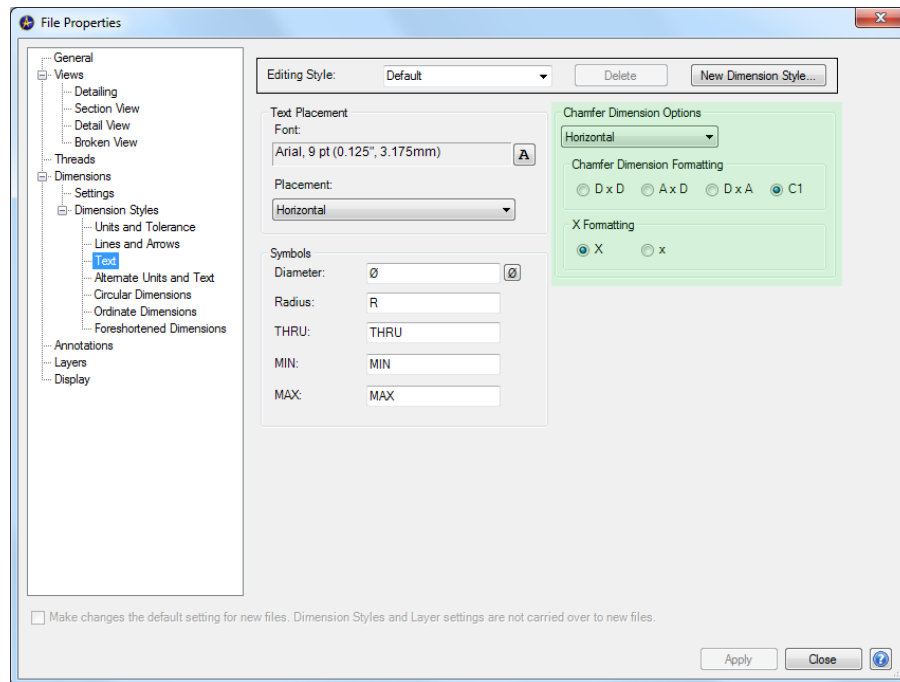
A new chamfer dimension tool is available from the Dimension dropdown in 2D drawings:



To use the tool, select the chamfered edge, then select the first and second reference edges as indicated by the status bar at the bottom of the screen. The chamfer will be ready to place. Several style options exist for the chamfer dimension:



Options can be selected from the Drawing Properties dialog on the Text pane:

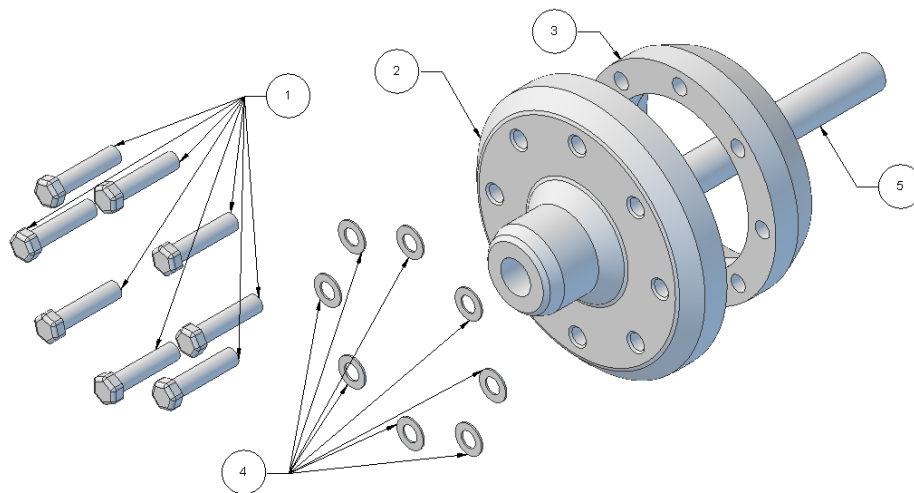


POINTS

When in 2D Sketch Mode inside a 2D drawing, using the Point tool will create points that are visible in prints and exports.

BOM CALLOUTS

Creating BOM callouts with multiple leaders is now easier.



To create multiple leaders, select the initial part and place the callout. Continue to click on additional parts to add additional leaders.

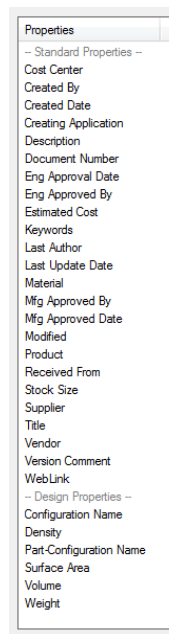
BILL OF MATERIALS

BOM PROPERTIES

- Surface Area, Volume, and Density are now available to be inserted into a BOM. The data is pulled from the model files.

Quantity	Item	Surface Area (in ²)	Volume (in ³)	Weight (kg)	Density
8	1	1.289	.075	0.003	0.00277
1	2	43.145	7.176	0.317	0.0027
1	3	24.187	2.833	0.125	0.0027
8	4	.298	.003	0.013	0.0027126
1	5	11.697	.992	0.045	0.00274

- BOM Properties are now organized in a clean, easy-to-read manner with properties separated by functionality:
 - Standard text properties
 - Design driven properties
 - Custom vault properties



BOM TEXT – AUTOWRAP AND MULTI-LINE

Text wrapping is now supported. Multiline text is also supported. To create a multi-line entry, press Shift+Enter to begin a new line.

Quantity	Item	Comment
8	1	Source from Pacific supplier.
1	2	Confirm pricing is under \$43 / unit. If not, consider sourcing.
1	3	
8	4	
1	5	Approval Chain: - Sebastian F. - Ralf S. - Lew M. - Sally Y.

Text wrapping

Multi-line entries via Shift+Enter

AUTOFIT COMMAND

Autofit now applies fitting logic to both column width and row height. Previously it only applied this logic to column width. Autofit can now be applied to the entire BOM in a single operation.

API ENHANCEMENTS

The following enhancements were made to the API:

- Modify Sketch Text
- Suppress multiple features
- Disable auto-regenerate
- Optimized applying assembly occurrence transformations with constraints
- Create assembly constraints
- New property for Session TimeStamp/VersionReference
- Set view transform
- Selecting individual sketch figures

disponibile presso:

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