SpaceClaim 2015

Release Notes

SPACECLAIM
CORPORATION

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SpaceClaim 2015 Beta Enhancements Overview

This release improves SpaceClaim in the areas of STL/faceted modeling, prep for manufacturing and simulation, and overall usability, performance, and direct editing.

These enhancements seek to address top customer requests and expand the capabilities of the software while maintaining a focus on keeping SpaceClaim easy to use, simple, and fast.

New capabilities in SpaceClaim 2015 include:

- New tools for editing stl and other faceted file formats. Specifically:
    - Smooth tool for smoothing rough edges
    - New faceted repair tools for fixing holes and sharps
    - Dedicated Separate and Join tools for separating mesh objects into connected sets
    - Merging and splitting of *open* mesh objects
    - Non-uniform scaling
    - A Thicken tool for thickening a faceted surface
    - Regularize tool for evenly spacing a selected set of facets
    - Thickness and Overhang analysis tools for analyzing characteristics of a mesh
    - Mass property and area calculations
- Manufacturing prep tools. Specifically:
    - Deburring Toolpath tool for creating an offset toolpath from a set of connected edges
    - Turn Profile tool for generating tool paths from revolute faces
    - Relief tool for automatically finding and adding corner reliefs to flat parts
    - Wrap tool for folding solid and surface geometry around other solids
    - Curve table tool for creating a table of text definitions for a selected set of curves
- Powerful modeling enhancements such as:
    - Improvements to Pull/Move of surface edges
    - Blend surface controls
    - Round re-ordering
    - Variable radius round improvements
    - Performance
    - Point projection with dynamic update
- Sheet metal improvements such as:
    - Swept walls
    - Improved bend modifications
    - Rounded louvers
    - Power selection

Junction improvements

Notes

Languages:

You should switch to the English locale if you want to test the new translators.

The translators in SpaceClaim 2015 Beta do not work correctly for non-English locales (basically, any foreign language locale that uses a comma instead of a period as a separator, which includes countries like, France, Germany, Italy, Poland, etc.).

On the Win 8.1 32bit OS in **_all_** languages, some assembly models won't open in SpaceClaim for all translator file formats (i.e. CATIA, PS, NX, Inventor, etc….).

ACIS:

ACIS R25 does not support 32bit on Windows 8.0/8.1 OS's. Only 64bit is supported on Windows 8.x.

This means there is no support for:

- Windows XP (32 & 64bit)
- Vista (32 & 64bit)
- Windows 8.0/8.1 (32bit)

The 64bit version on Win8.1 works fine.

CATIA V5:

SpaceClaim 2015 no longer supports reading of CatiaV5 files created in versions older than CatiaV5R8

TraceParts:

Inserting parts from the TraceParts web page is not working. If you want to use TraceParts files, you will need to use the TraceParts web download option.

Pull

Pivot Edge direction is now remembered in Groups. Previously, as shown in the example below, the direction could flip. The left image shows the initial ruler dimension added to the group. The right image illustrates the old behavior. After changing the group value and then clicking in white space, the direction flipped when the group was re-opened.
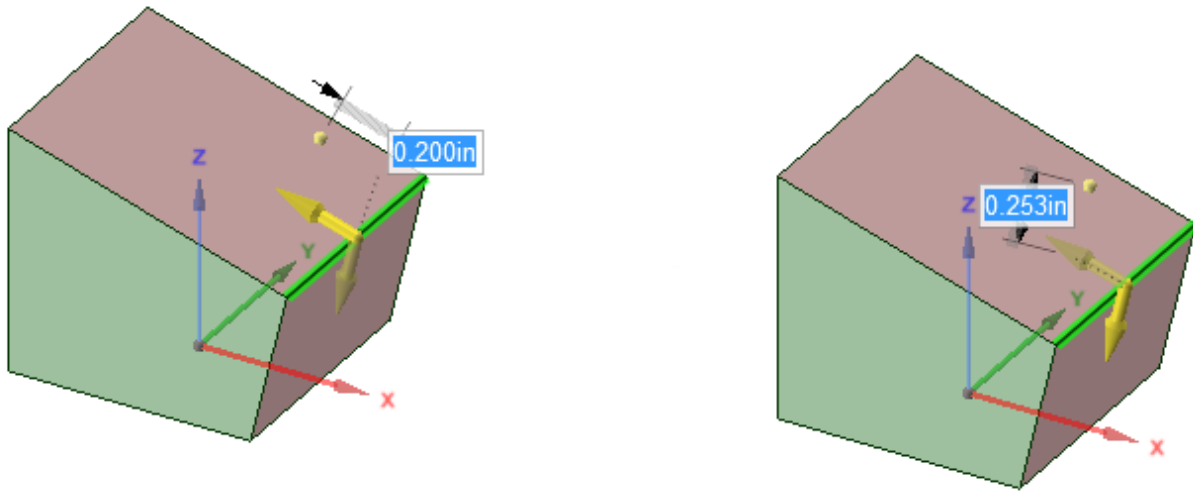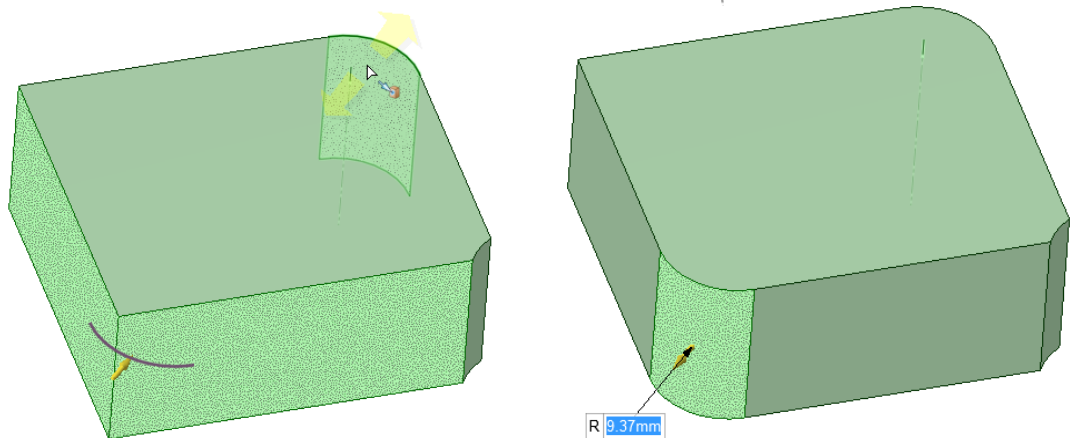

You can now Pull Face-Face rounds and match their radius to other rounds using **Up To**. Some examples are shown here.

The first example matches an interior round to the exterior round radius.
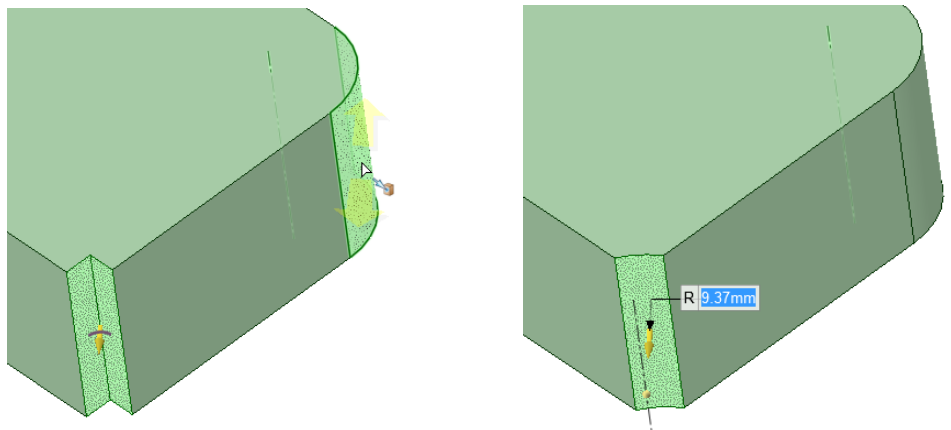
The second example shows a Face-Face exterior round.

You can now rotate

multiple coplanar faces to create a solid. Previously, you could select the faces and an axis to rotate about, but the rotation would produce no result. Now, as seen in the example below, the faces rotate into a solid.

Full Pull
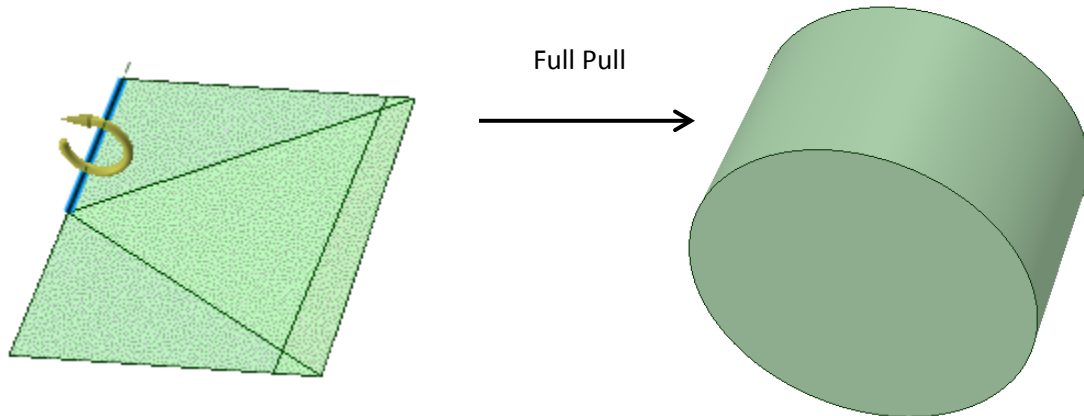
The **Pull** tool no longer defaults to Rotation after a non-translation tool guide selection. Previously, if you did the following:

1. Enter the **Pull** tool
2. Click the **Sweep** tool guide
3. Select a trajectory
4. Click the **Select** tool guide

the tool would switch to Rotation mode. Now, the tool stays in Sweep mode.

The keyboard Nudge shortcuts (using Ctrl) no longer create copied surfaces instead of Pulling the part at the large Nudge value. Previously, you could get the result shown on the right.
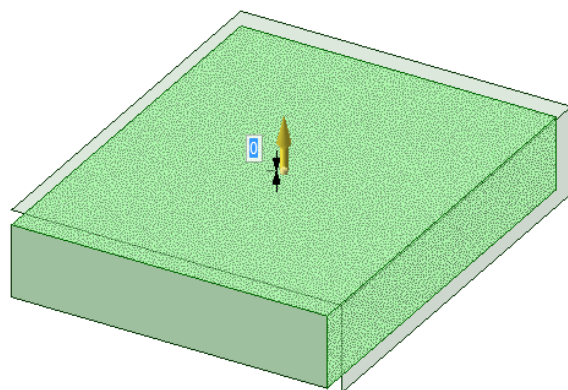
Round modification is being enhanced. Some modifications fail because the entire round chain (along with all chains required to be removed by the modification) are all included in the modification. In some

cases, parts of the chain (or others, even far away ones) are restricted by surrounding geometry and the modification fails. Now, to make modifications more robust, they are localized to the selected piece of the round.

A key to this working is that the ends of the localized section need to be "auto-capped". In other words, they need to be able to be closed off by a plane triangular face, essentially normal to the last-visited construction edge. The following example illustrates this.

The radius is going to be increased at the selected edge.
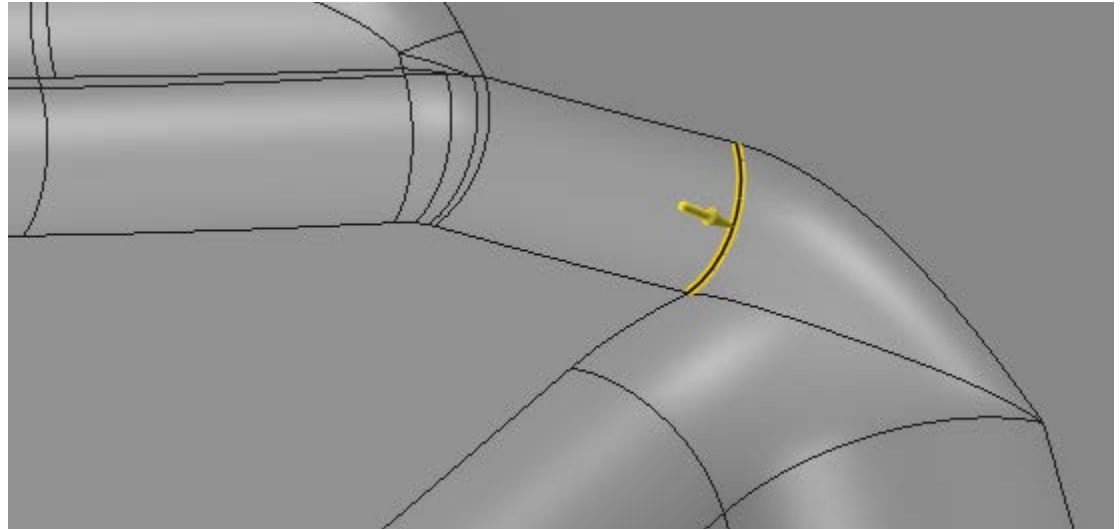
Localizing means that the round pieces on either side of the edge are removed first. For this to work, the ends of the removed pieces need to be "capped" with a triangular face or be the start or end point of the round chain. These end conditions are necessary to rebuild the new variable round between the ends.
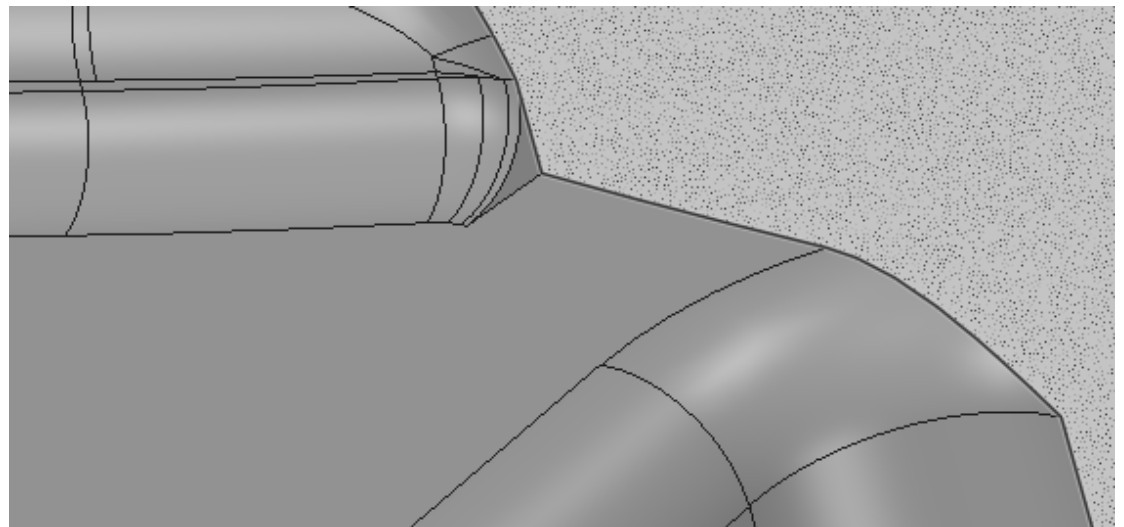
The Localized round is reconstructed with the modified radius.
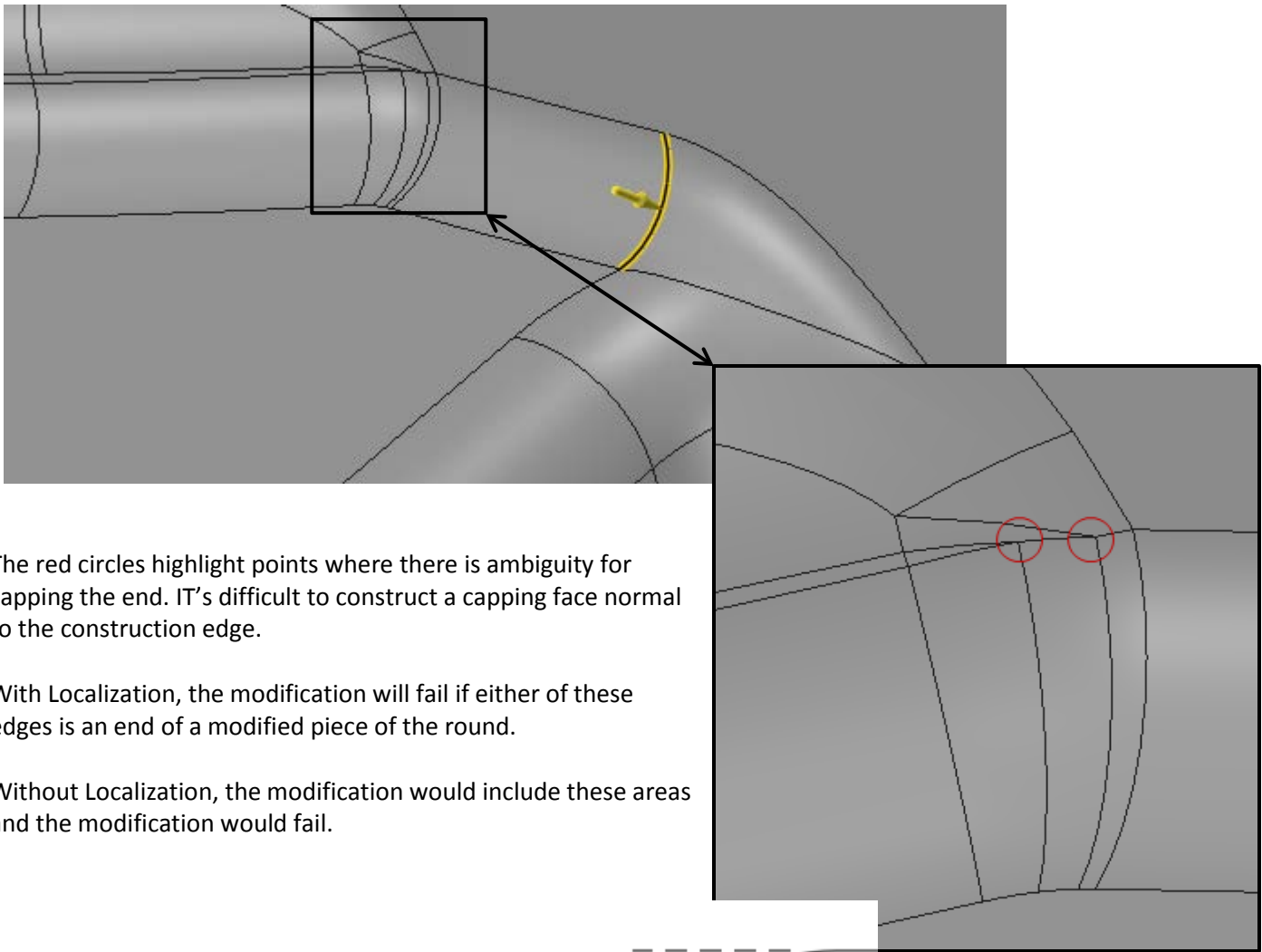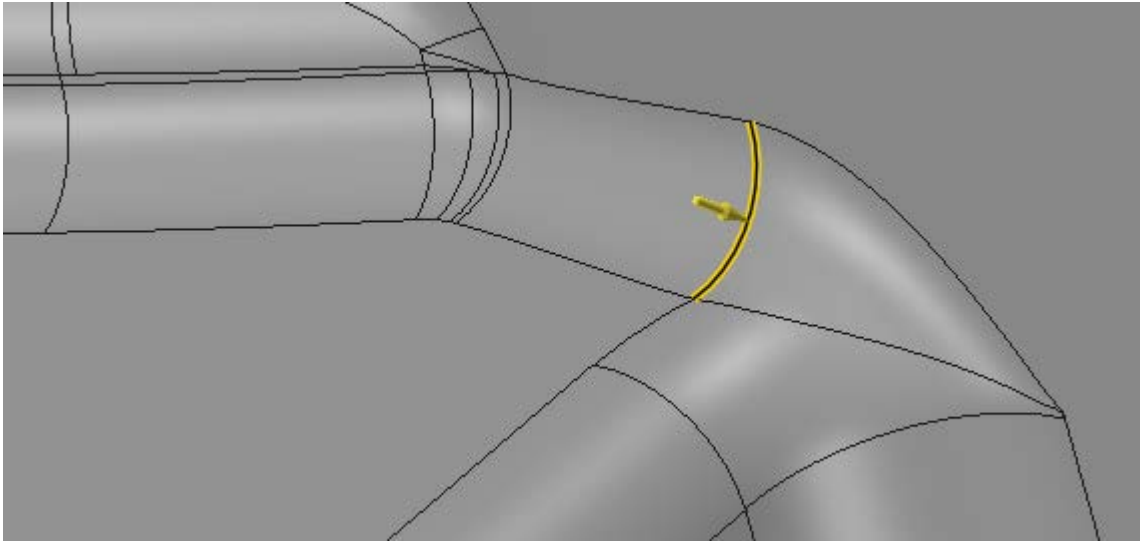
The red circles highlight points where there is ambiguity for capping the end. IT's difficult to construct a capping face normal to the construction edge.

With Localization, the modification will fail if either of these edges is an end of a modified piece of the round.

Without Localization, the modification would include these areas and the modification would fail.

In order to remove a round for modification, the side faces need to be able to Extend up and form a sharp edge.

In some cases, extra surface patches can cause ambiguous situations where the sharp edge cannot be determined and the round cannot be removed for modification. The image below is an example of a round that cannot be removed because of face extension ambiguity.
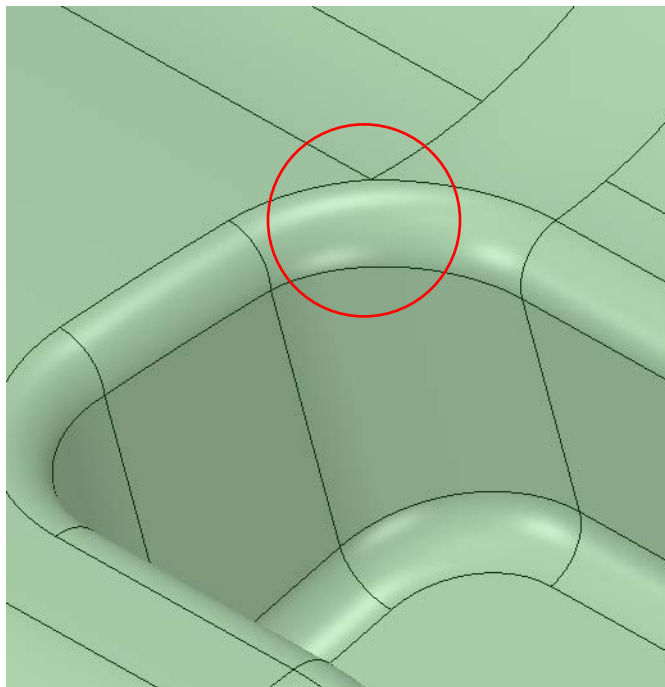
Round Options have been added to the Pull tool. **Edit as Variable Radius Round** has been moved from the RMB to Round Options.
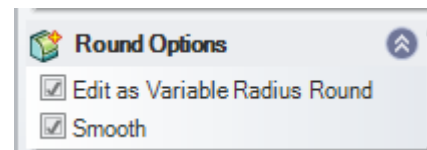

The **Smooth** option becomes enabled for constant radius rounds when the geometry allows it.

Options - Pull

⚙ General

➕ Add ➖ Cut 🚫 No merge

🔷 Round Options

☐ Edit as Variable Radius Round
☐ Smooth

Check Smooth and Pull
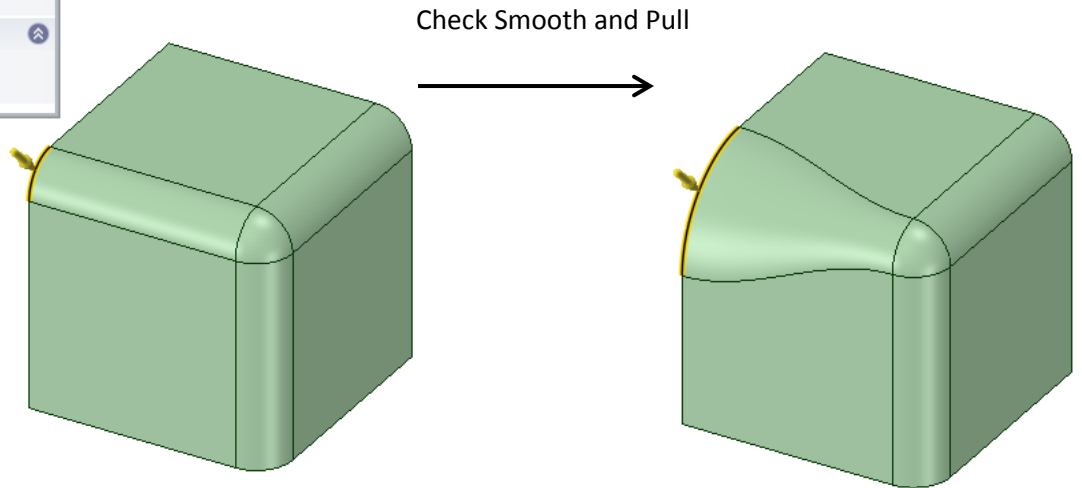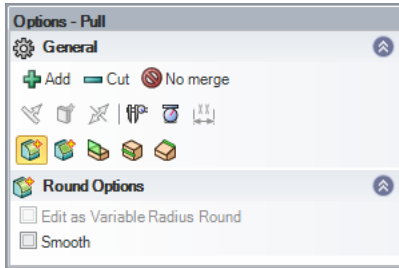
The **Smooth** option for creating Variable Rounds is 'sticky' across tool usages. Once you set it for one round, it will remain set for all subsequent rounds, until you change it. Click an edge of an existing constant round that can be made to vary smoothly, check the Smooth option and then Pull. The option will remain checked for subsequent round modifications.

Click another constant edge and Pull. The Smooth option remains active.
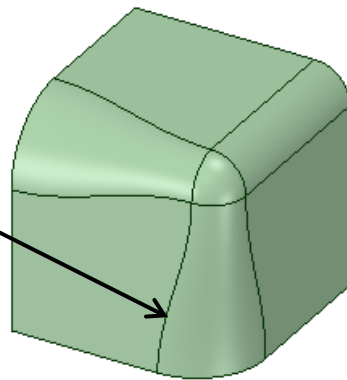
For periodic round faces (faces that go 'all-around'), the second variable radius handle is now created automatically. You no longer need to add the second handle by clicking on the tangent Pull handle and dragging or entering a value (image below right).
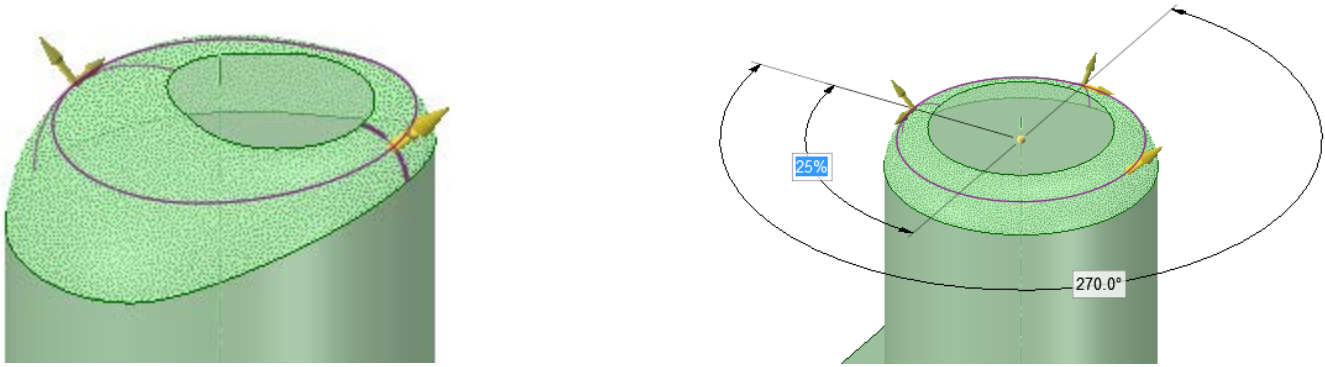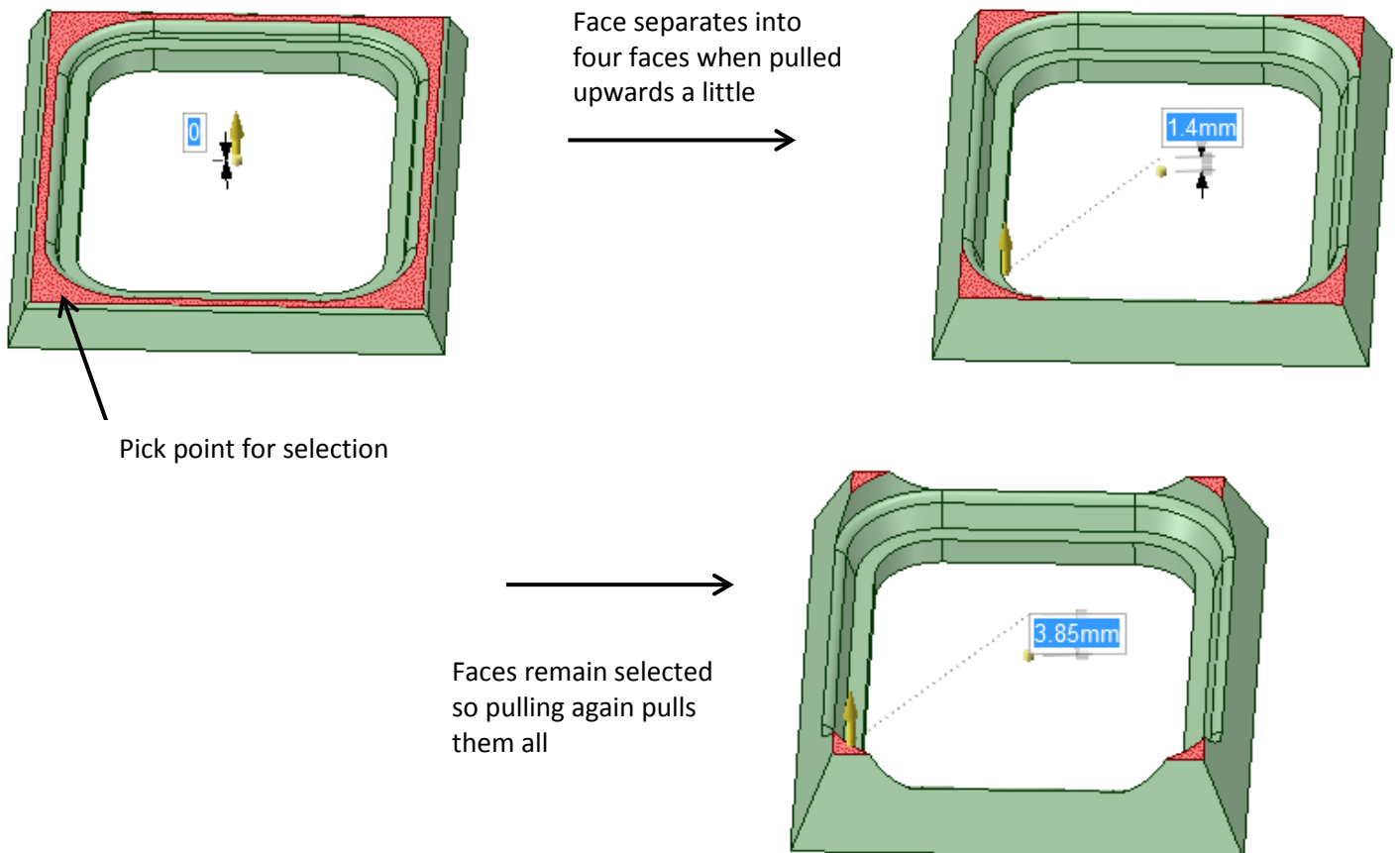
The Thicken option now works correctly when using Up To and pulling surface faces. An example is shown below. Previously, pulling the face on the left Up To the face on the right would create a solid body in between the faces.

Pull Up To this face

Bottom face is extended and no surfaces are thickened

Old behavior
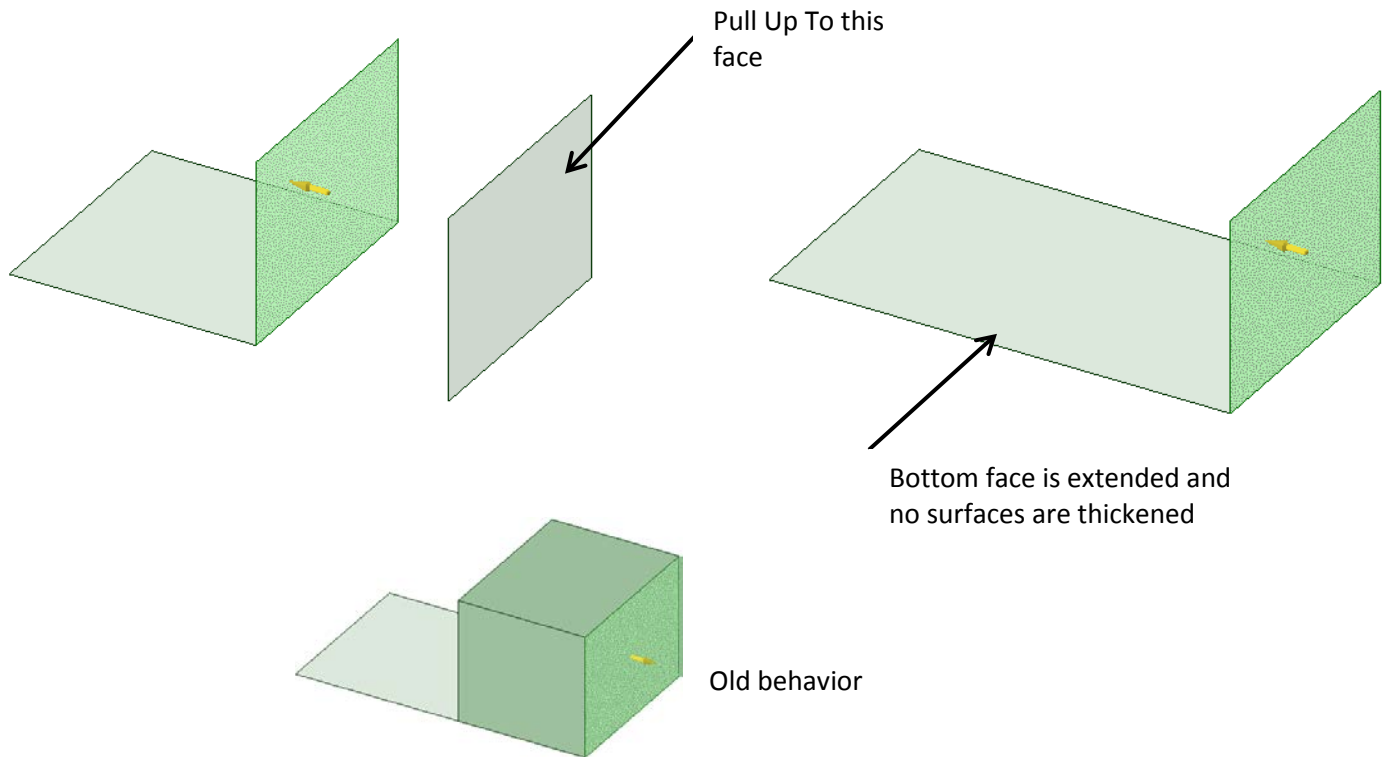
You can now drive a dimension to zero when a modification is tied to a spark dimension. An example is shown below. The red face is being pulled up which causes the spark dimension to shrink. Previously, if you entered "0" for the dimension, the modification failed.


✳0.778mm

Change 0.778 to 0 →
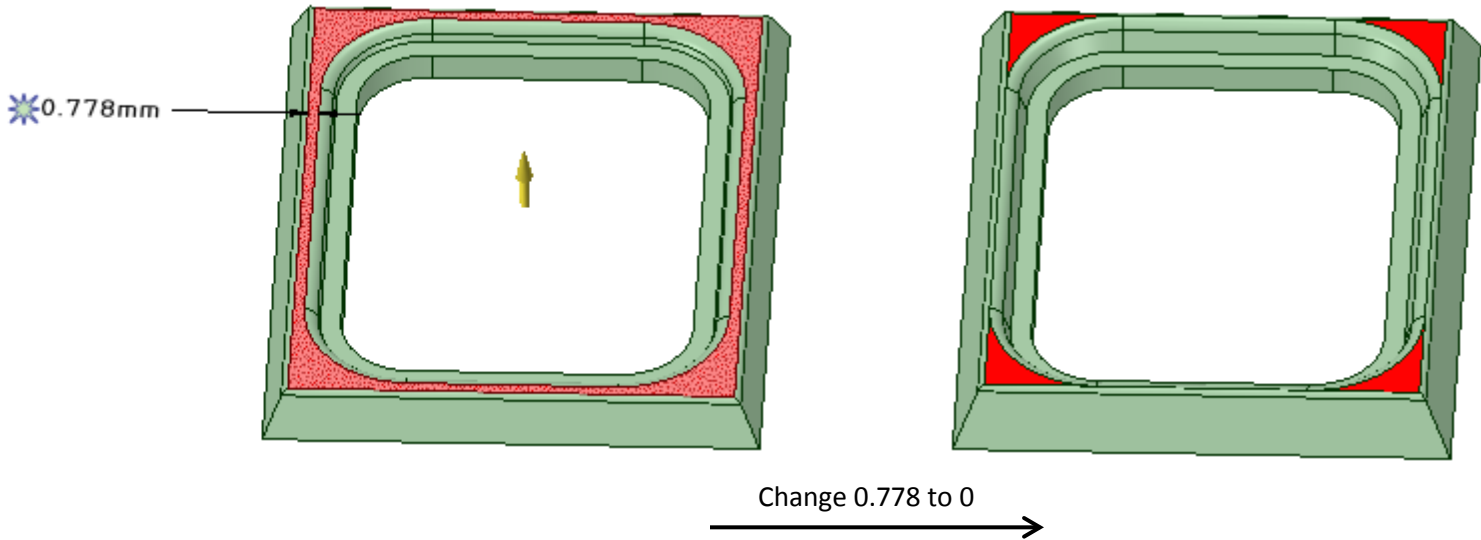
When a Pull operation results in the selected face separating into individual faces, the resulting faces remain selected. Previously, only the face containing the pick point of the original selection remained selected. The example below shows the new behavior.

Face separates into four faces when pulled upwards a little →

1.4mm

Pick point for selection

Faces remain selected so pulling again pulls them all →

3.85mm

You can now reverse the order in which a selected round is created. The order in which rounds are created influences the appearance and quality of the resultant geometry. You can change the order of round creation to clean up split rounds that produce extra edges, as shown in the example below.

Round an edge that is broken by existing rounds

Reverse the order to clean up the round
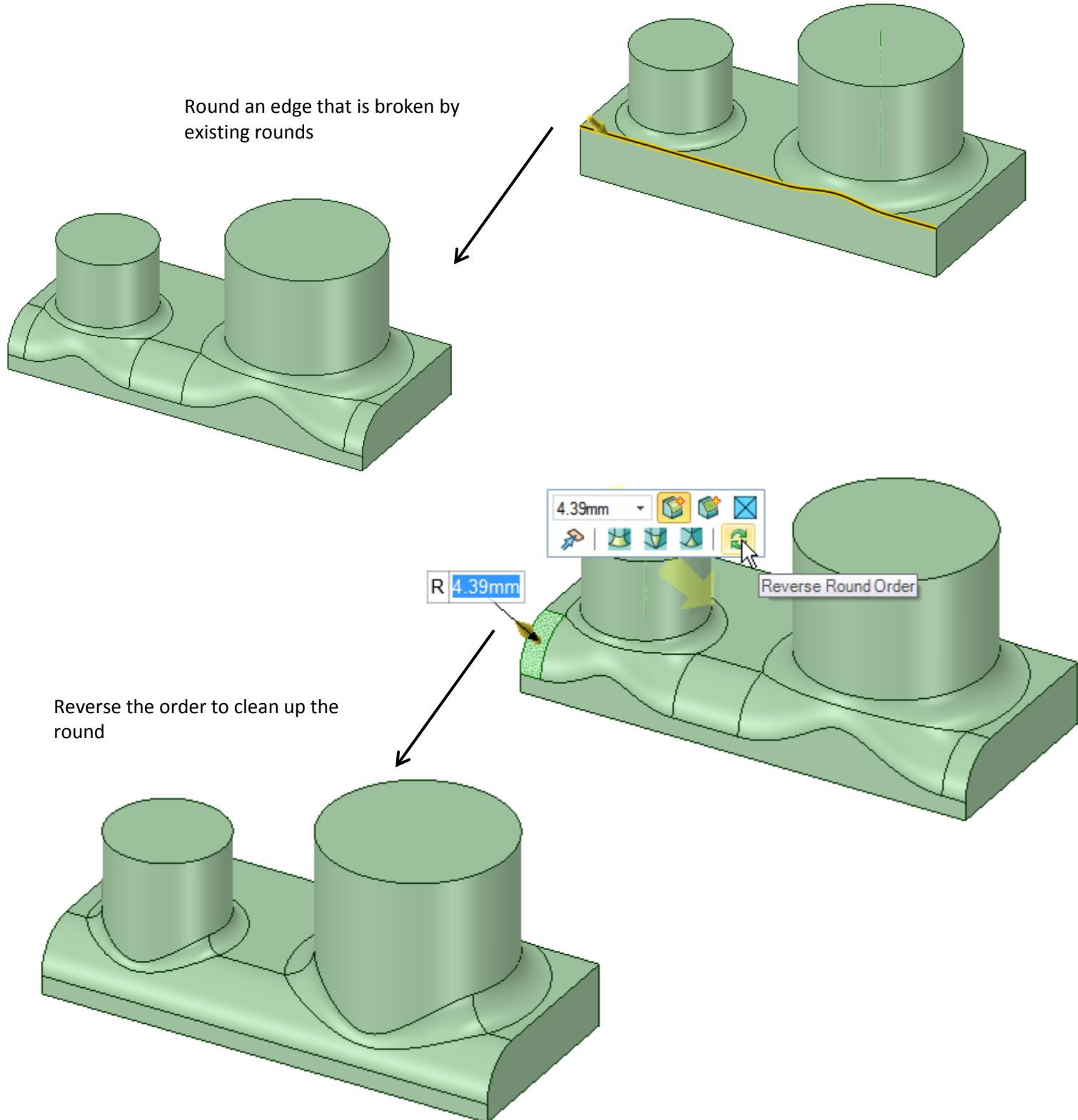
Surface edges now correctly extend **Up To** a datum plane. The images below show an example. Previously, when you clicked the Up To option and selected the datum plane, the operation failed.

Click the Up To option and select the plane
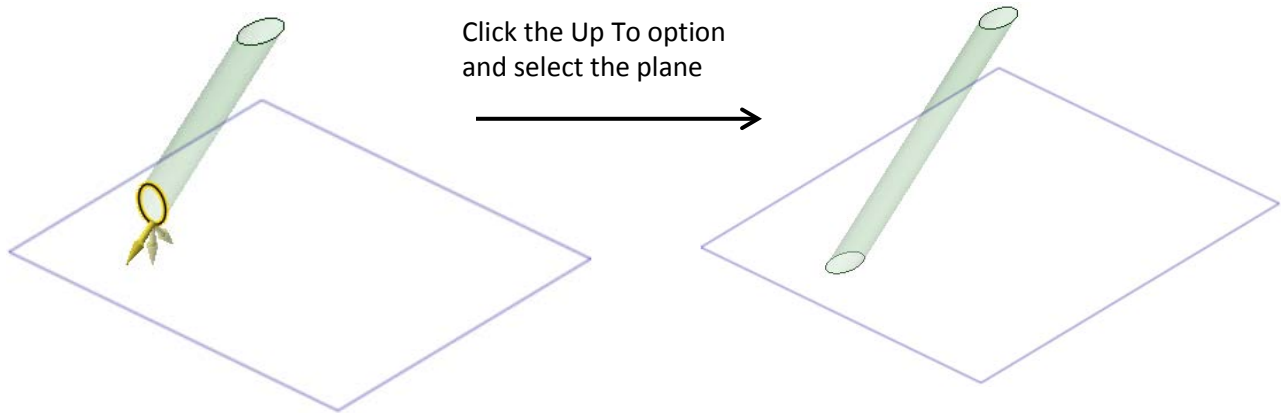
You can now toggle between **Tangent** and **Natural** (curvature continuous) extension when dragging the end of a curve. To drag the extension Tangent to the curve, press and hold Ctrl prior to dragging. You can release the Ctrl key once you start dragging. If you do *not* press Ctrl, the extension is created Curvature Continuous.

- Works for Splines (extended or trimmed-back)
- Works for Arcs
- The drag behavior has been improved in terms of speed of change versus drag cursor movement

Press Ctrl to drag Tangent to the curve

Without pressing Ctrl, the drag follows the natural curvature
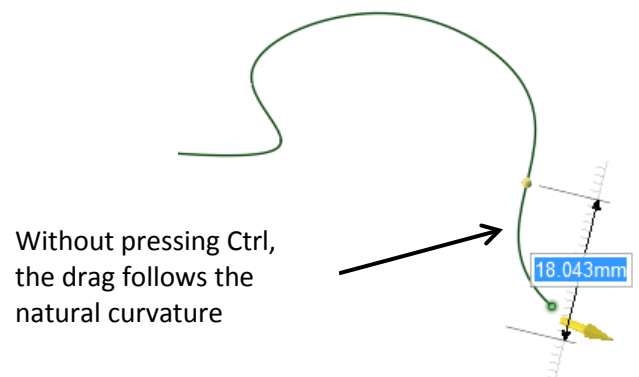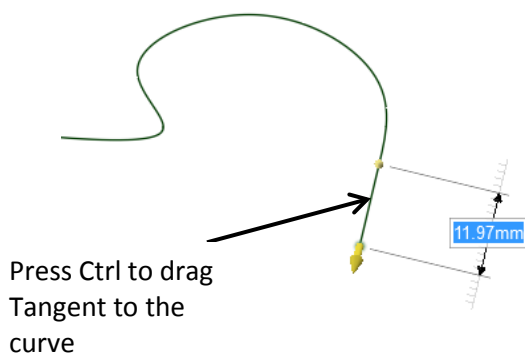
Rounds that roll over other rounds now adapt smoothly while pulling. The images below show the old behavior. Beginning with the image on the left, pulling the round smaller and then larger would cause the rounds to separate strangely.
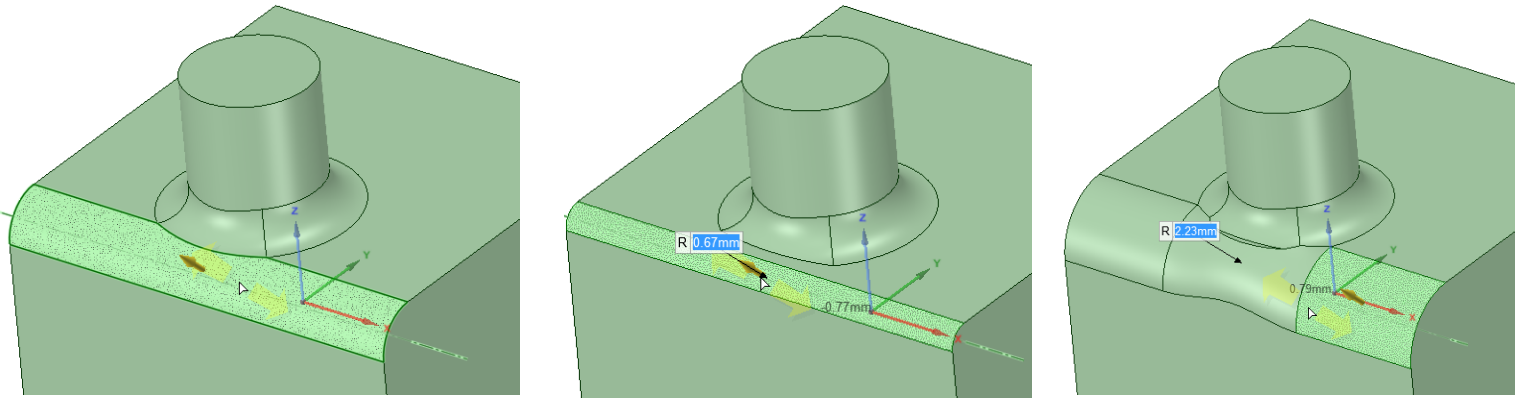
The images below show the new behavior. Beginning with the image on the left, pulling the round smaller and then larger causes both rounds to update smoothly throughout the Pull.
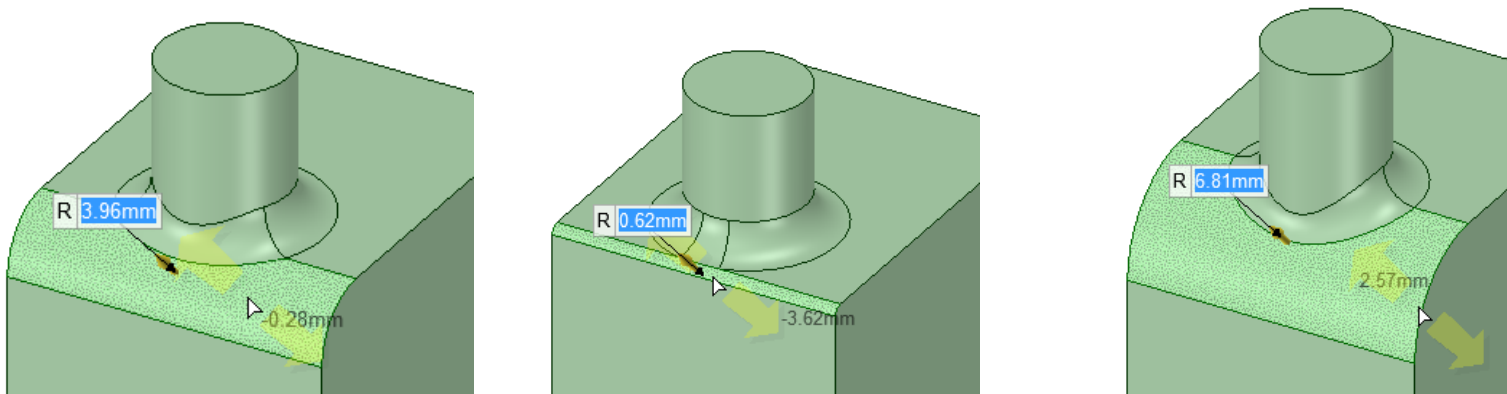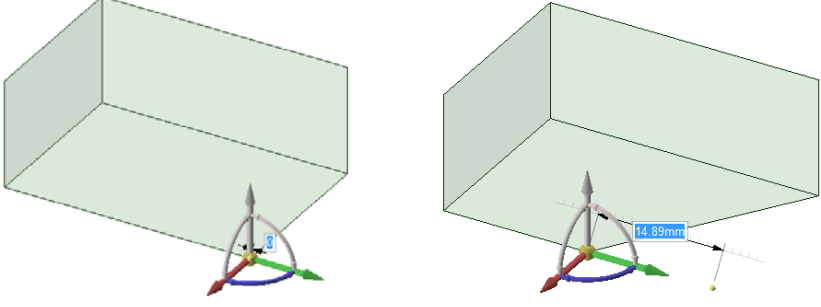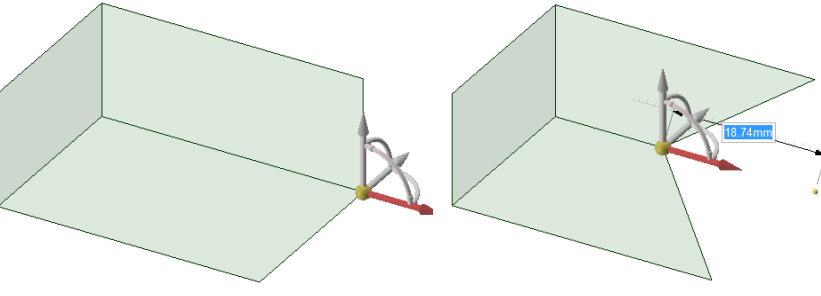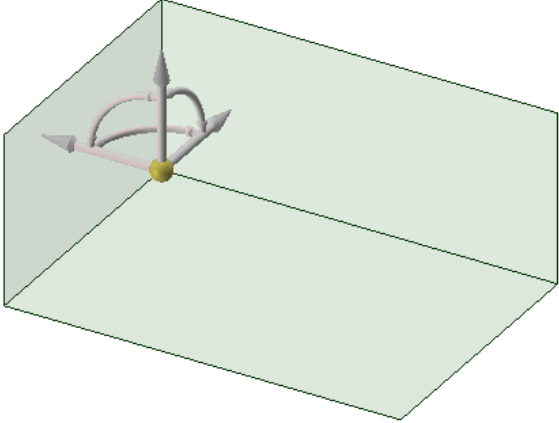

Surface edges can now be Pulled normal to their neighboring face. Press the Ctrl key when you begin the Pull. In the Sphere on the right, this results in a new cylindrical surface face that is tangent to the original sphere.
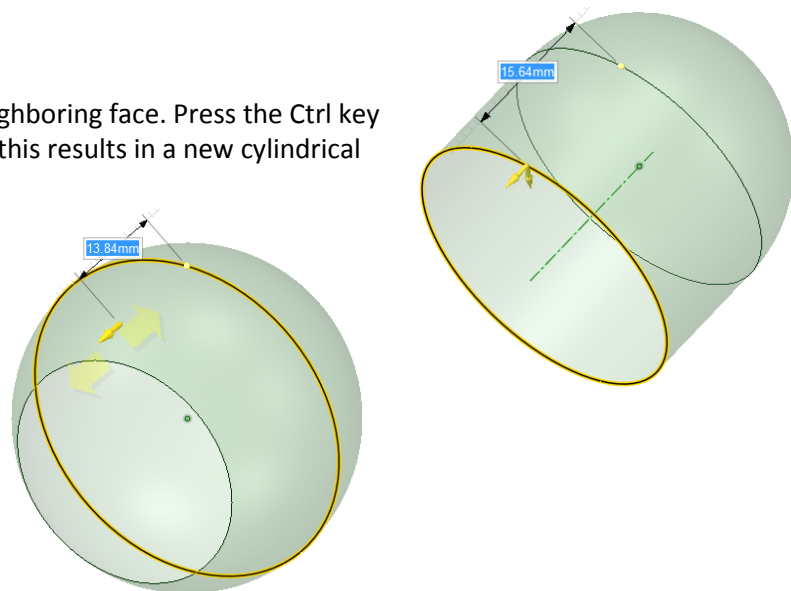
Previously, The new surface could only follow the spherical contour of the original.

Move

You can now drag a vertex of a planar surface as if it were a sketch vertex.

An open corner can be moved in two directions or moved freely within the plane.	
A vertex shared by two surfaces can be moved along the common edge.	
A vertex shared by three surfaces has no degrees of freedom.	

You can now pivot the edges of two planar surfaces about a point. Previously, you would get a message that the objects could not be moved.
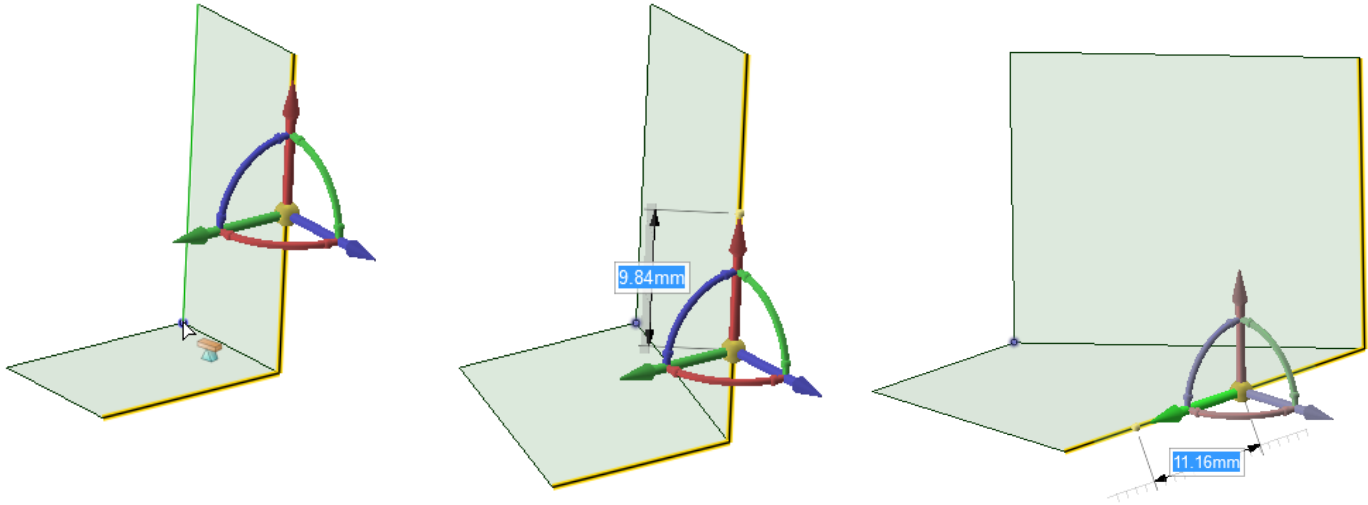
When in Section mode, you can now anchor the Move Handle to axes and temporary axes in the section. Previously, in the example shown here, if you moved the handle anywhere else in the section, you could not move it back to the center axis.
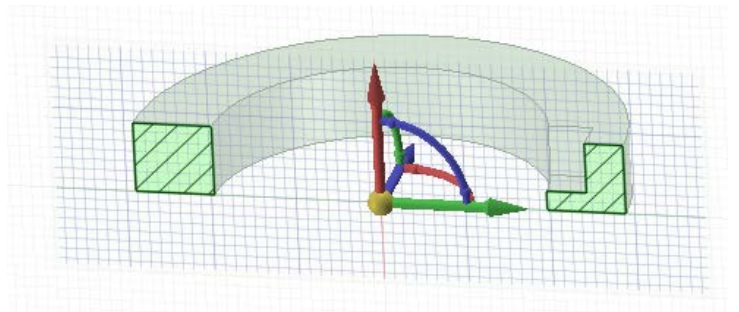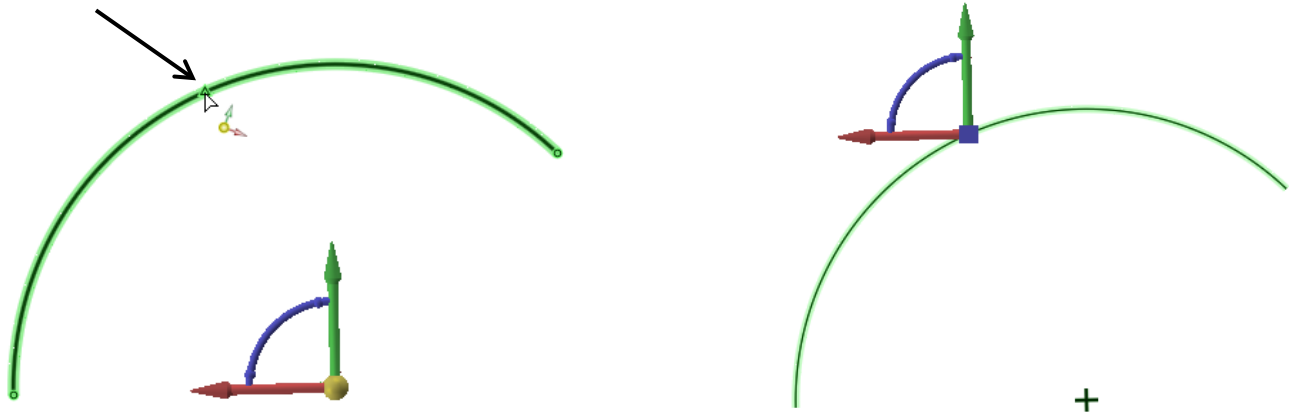
Points

You can now project sketched 3D Points onto surfaces using the **Project** tool. In the example, points were sketched to trace out the shape of a mouse button and then projected down onto the top surface of the mouse. The workflow is the same as projecting Faces, Surfaces, Curves, Edges, or Note Text.
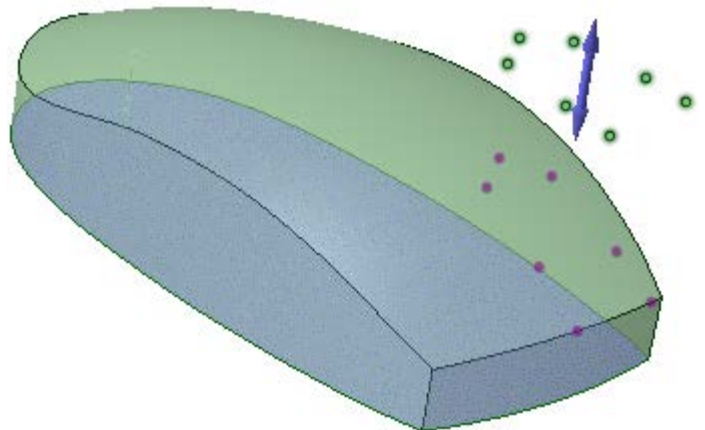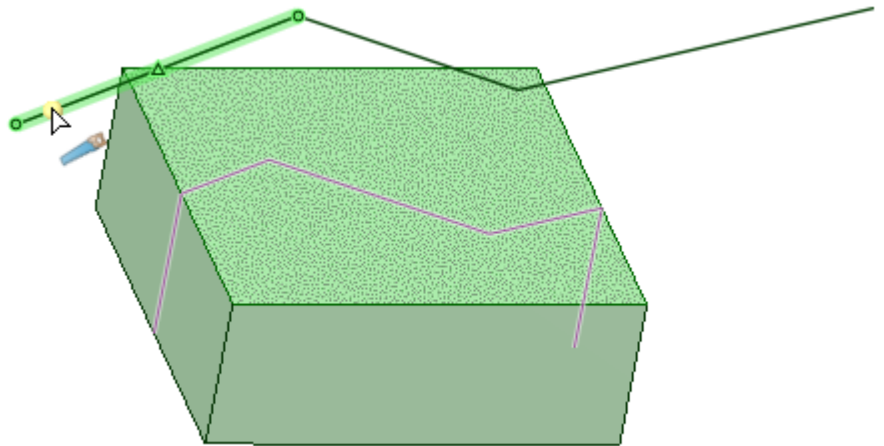
Sketch

You can now **Anchor** the Move Handle to the *midpoint* of a sketched three-point arc. Previously, selecting the midpoint on the arc would anchor the Move Handle to the arc center.

Split Face

You can now Split Faces using Curves, whether they are touching the face or not. Connected curves can be used as the cutter by holding the **Shift** key. An example is shown to the right.
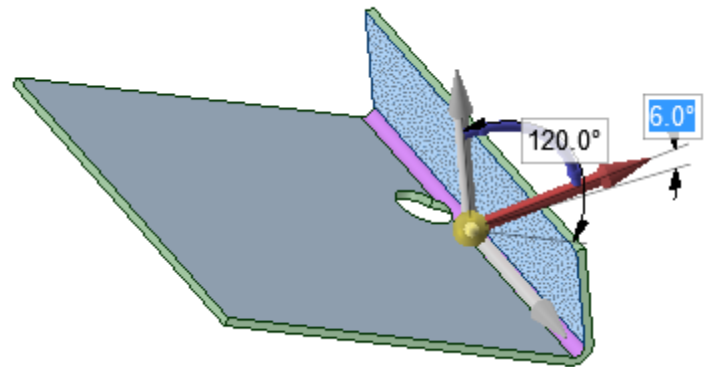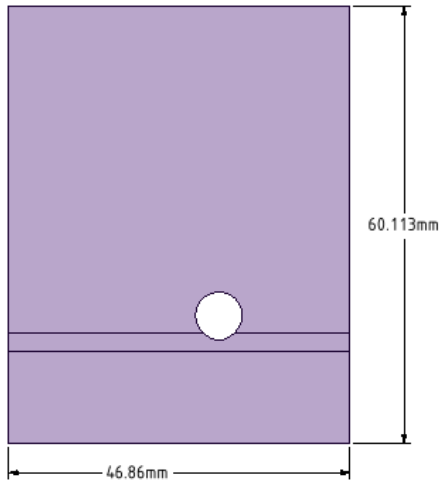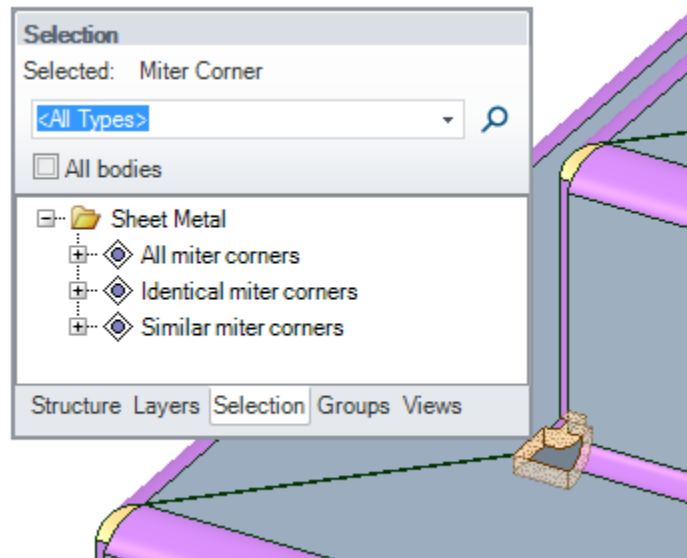
Project

When you project Points using the **Project** tool, Datum Points are created, instead of Sketched Points, so that the projected points can be Pinned.

Sheet Metal

You can now rotate walls whose bends are partially cut out. An example is shown below. A simple L-shape part is unfolded and a cut is made in the flat.

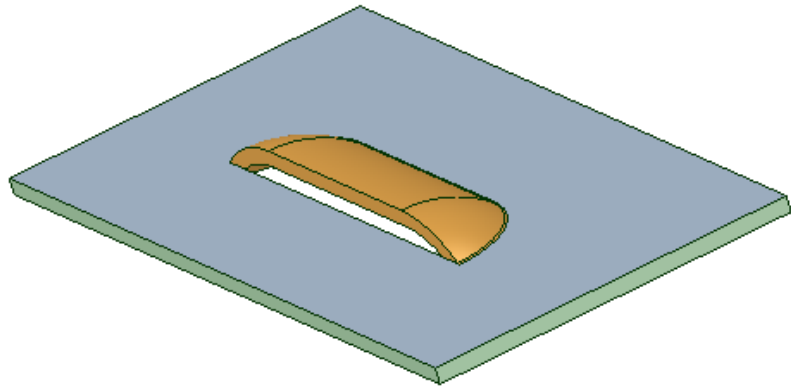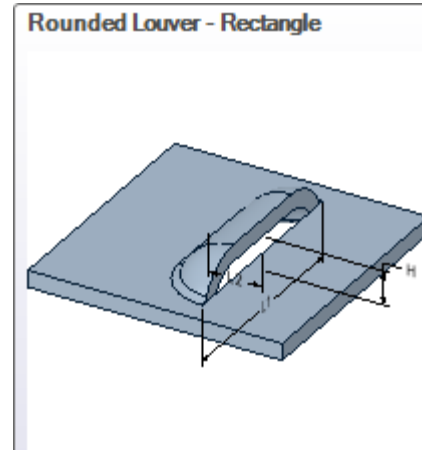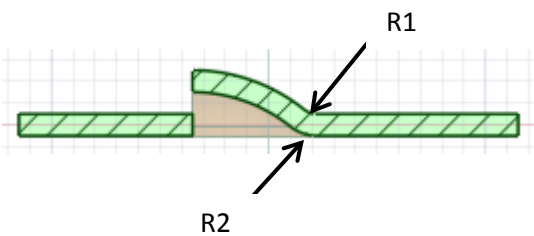You can now Power Select sheet metal Miter Corners.

Forms:

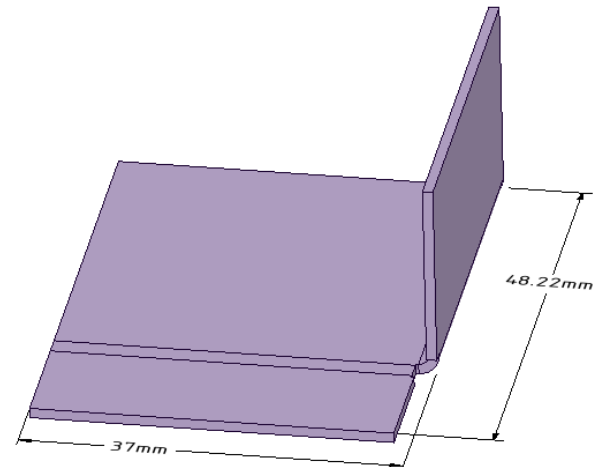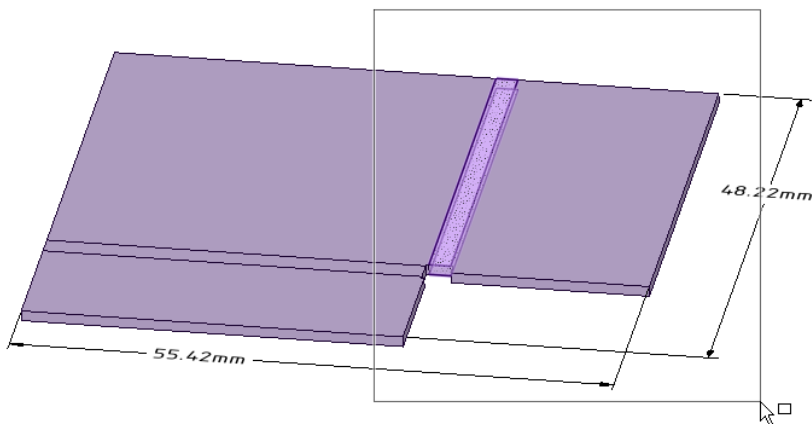There is a new Form for creating **Rounded Louvers**.

It has the following parameters:

- **H** – Height
- **L1** – Long length
- **L2** – Short length
- **R1** – Round radius underneath
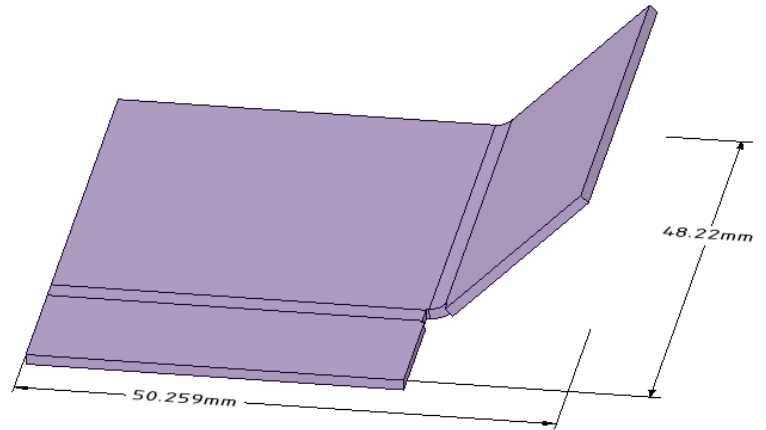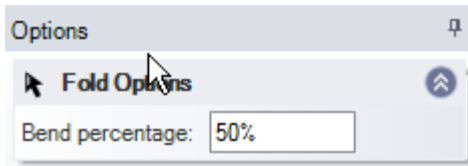- **R2** – Round radius on top

Rounded Louver - Rectangle

Fold Walls Tool:

For unfolded parts, there is now a **Fold Walls** tool. You can select Bends individually, by clicking on the Bend area – not the wall, or by using box-select.

You can also use the **Bend percentage** option to partially bend a wall.

Pull and Move:

You can now **Move** and **Pull** walls in the cases shown below to increase the distance and bend radius between the walls.

To create these cases:
1. Create 180 degree bends
2. Use a perpendicular bend line to create the parallel case, or an angled bend line to create the angled case
3. Pull or Move one of the walls to increase the gap
4. The wall will detach briefly but then the Bend will update to the new distance
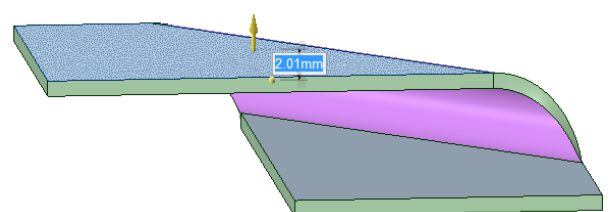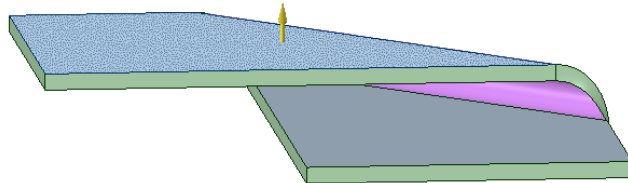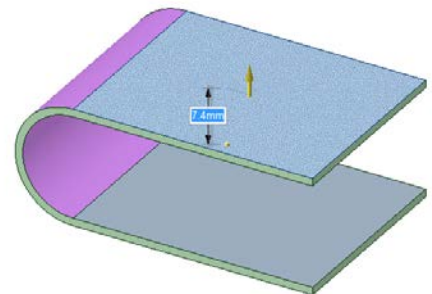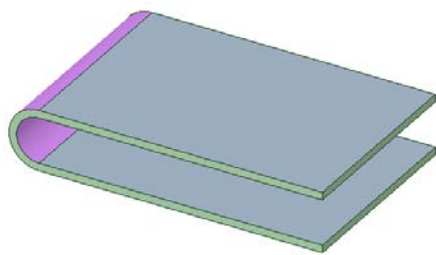
Bend Tool:

When creating a **Bend** using the **Select Two Cutter Points** tool guide, the bend line is previewed after the first point is set (by clicking or entering a value).

o   The line previews as you drag the cursor to set the second point
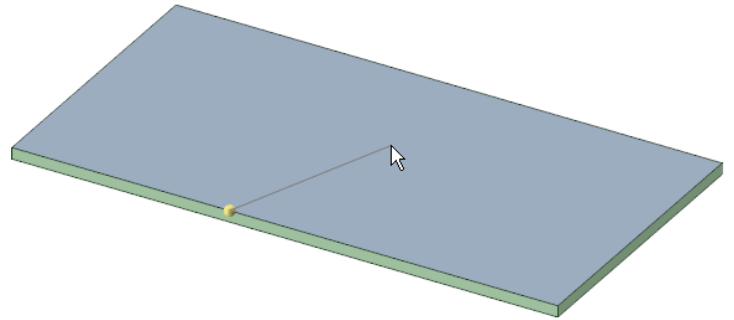
o   The bend extents are displayed when the cursor reaches another edge
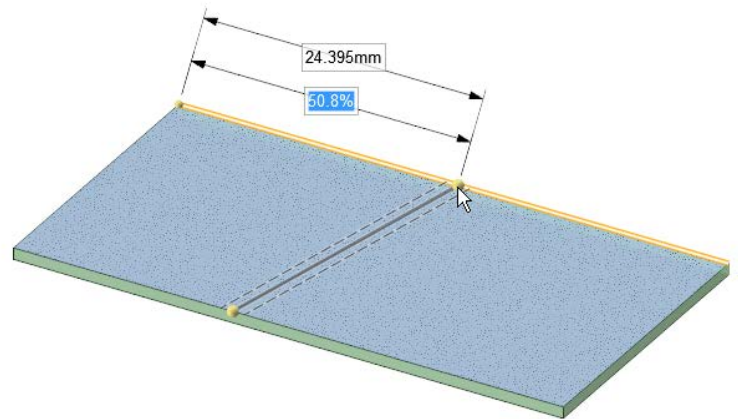
When a Bend is made and a Hinge is present, the Hinge and related walls now follow the bending up.

Partial bends now unfold properly when the part has Gussets.

Bend Notches no longer adjust when different bend allowances are applied to the bends of the Notch. Previously, if you changed the bend allowances, the Notch would appear warped in the unfolded model. You could reset the Notch but that would also reset the bend allowances to their original values. Now, the Notch settings and Bend Allowances can be changed independently. The image on the right shows an example of a Bend Notch.
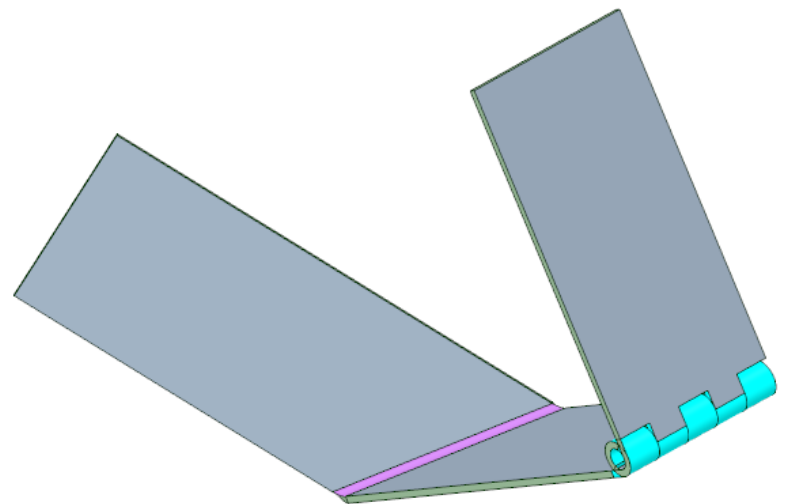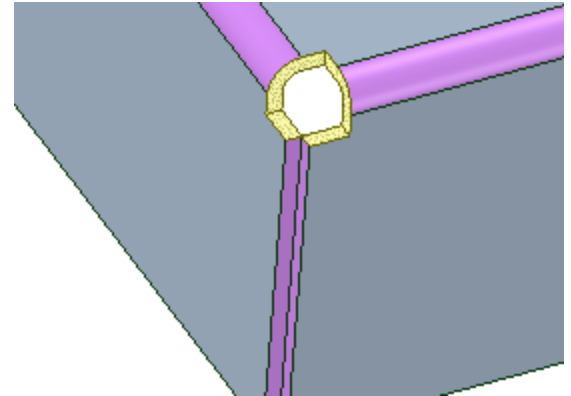
Repairing inaccurate Bends now works differently. Previously, SpaceClaim would simply identify cut bend faces that were not perfectly offset from each other or other faces not nicely paired, and recreate the Junction from scratch. Now we have the additional capability (behind the scenes) to also Replace Faces in repairing them.

You can now change a Bend radius if an edge relief is present. Previously, in the model shown below, you could select the Bend and enter a new radius value (e.g. change 0.08 to 0.0394) and it would be ignored. Now, the bend radius can be changed to 0.0394.
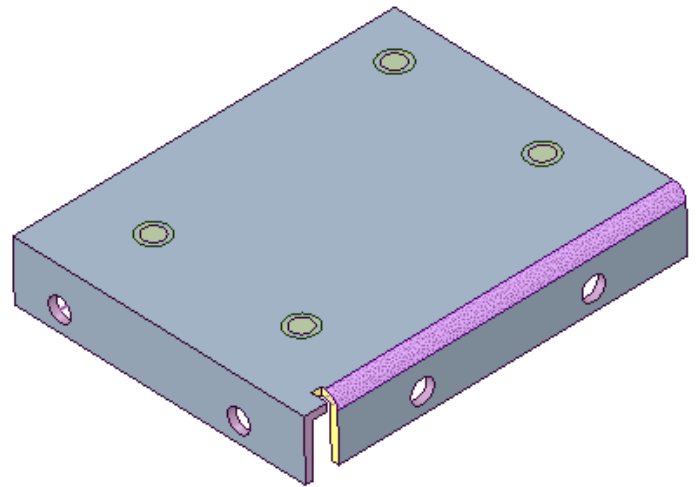
Sheet Metal Options:

A new Junction gap field has been added to the Sheet Metal page of SpaceClaim Options. This is currently UI-only with no functionality attached.

Basic defaults

Thickness:	1mm
Bend table:	None
K-Factor type:	Variable
K-Factor (for R=T):	0.27324
Bend radius:	Thickness ratio	1
Split face gap:	Value	0.1mm
Junction gap:	Value	0.1mm
	Thickness ratio	
	Value	

Relief defaults

Sheet Metal Properties:

With a Sheet Metal part selected, the RMB now has an **Export Sheet Metal Properties** command. It opens the Export Sheet Metal Properties dialog for writing a CSV file. The file has columns for the following properties.

- Thickness
- Area of the flattened part
- Perimeter of the flattened part
- Maximum X and Y extents of the flattened part
- Number of Holes
- Number of Bends

You must first **Unfold** the part to have the flat properties export.

Junctions:

You can now enter a Percentage for **Partial Overlap Junctions**. Previously, you could only create Full or Half overlap.

Overlap Proportion has been added to the Sheet Metal Properties of Partial Junctions. By default, the percentage is 50, which you can change to anything greater than 0% (0.0) and less than 100% (1.0).
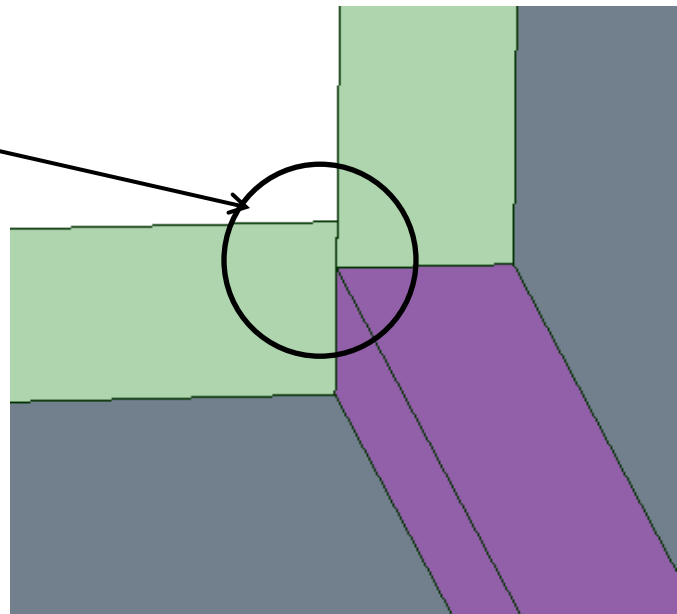
Flange junction conversion has been improved.

Converting Full Overlap junctions to Bends has also been improved

The **Junction ga**p option in the Sheet Metal page of SpaceClaim Options now works correctly. Previously, if you entered a new value, it was ignored. The only way to change the values was to select the Junction and change its properties.

Bend Tables:

There is a new option in Bend Tables. Use:

```
KFactorFallback=true/false
```

to control what happens when an invalid VeeDieWidth is encountered in a Bend Table applied to a model.

When set to True, the K-Factor is used. When set to False (the default), the bend fails.

Bend Tables containing errors are now handled differently. Previously, the load would abort and no bend tables would be loaded. Now an error message is shown and the load continues.

Sheet Metal Sketching:

A problem has been fixed with sketching sheet metal walls using the **Profile** option. Previously, if you sketched the U-shaped profile on the right using **RMB > Switch to Arc** and **RMB > Switch to Line**, the system would create three separate bodies. Now, as long as *all* segments are *tangent*, only one body is created. Non-tangent segments are created as separate bodies.
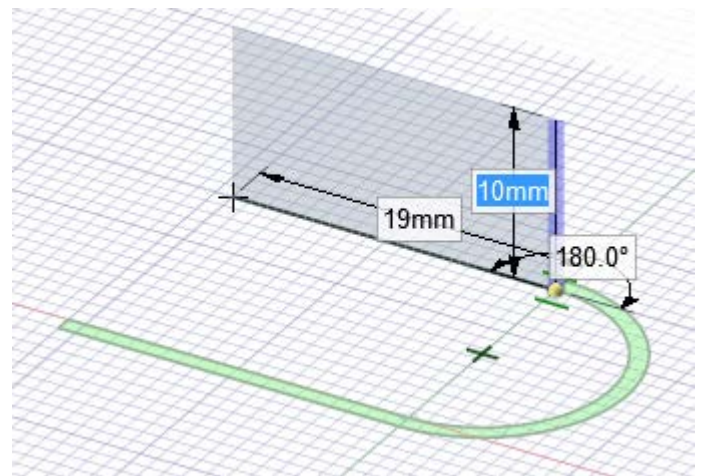
Swept Edge Forms:

To create a Swept Edge Form:
1. Sketch a profile at an endpoint of the edge
2. Enter the **Pull** tool
3. Click the **Sweep** tool guide
4. Click the **Full Pull** tool guide

Full Pull sweep of a line profile along an edge
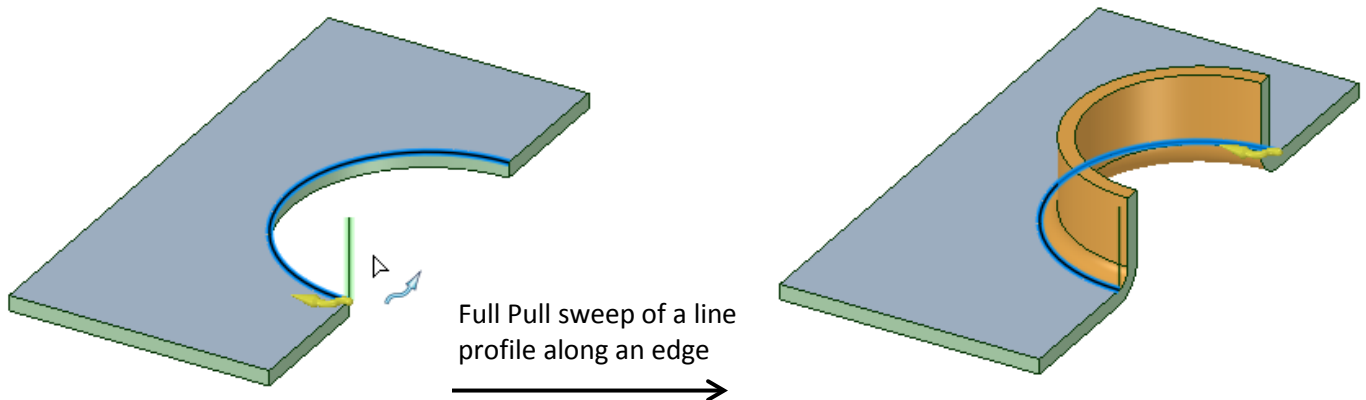
Notice that the sweep profile is a single line and an Inside Radius arc is automatically added to the profile, at the attachment point, to produce a bend. If the user draws a tangent arc along with a line, then that arc determines the bend radius and overrides any IR automation that is done in the case of only a straight line.

Swept Edge Forms can also now be unfolded.
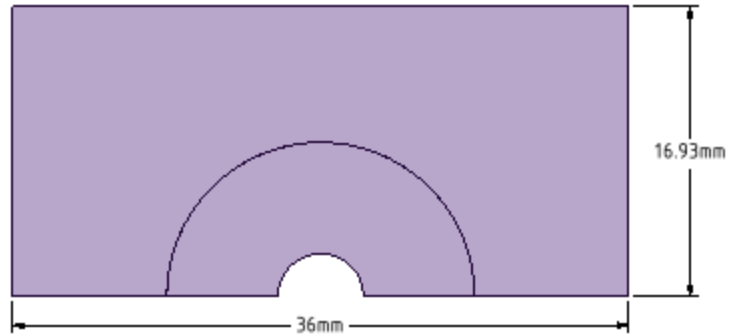
Gussets:

Gussets now center correctly on non-90° Bends.
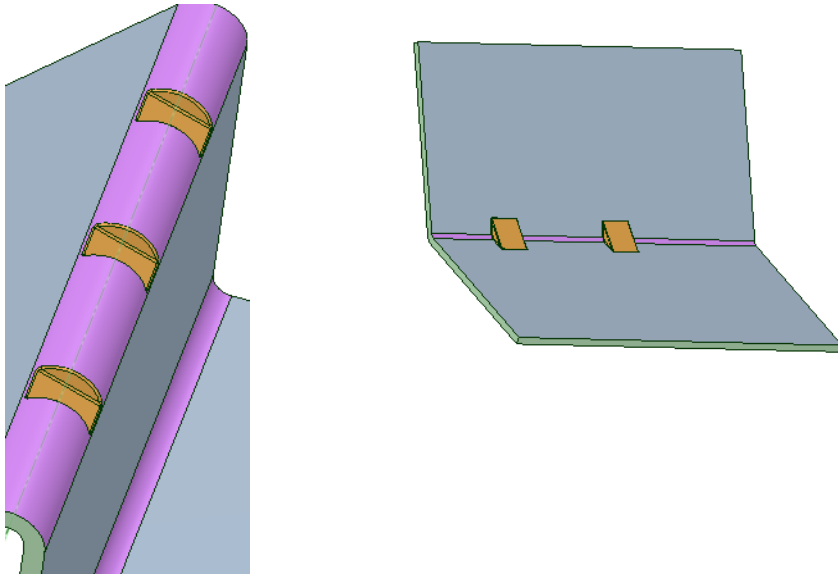
Detail

It was possible for a document to have more than one Drawing. This occurs when an external component with a Drawing is Internalized in a design that also has a Drawing. In this case, it was possible (with drag and drop) to move a view from one of the drawings to another. The functionality is not stable and has been disabled.

While <u>creating</u> a Leader, hold the Shift key to snap the Leader to the nearest 45° increment.

1. Click to set the start point of the Leader
2. Start dragging to get near the desired orientation
3. Press and hold the Shift key
4. The Leader snaps to the nearest 45° increment
5. Continue holding Shift down and dragging to set the length
6. Click to set the endpoint of the Leader

You can now toggle a Leader to underline its attached text. If the Leader is attached to the Note's underline location, you can use the RMB to extend the tail of the Leader and underline the text.
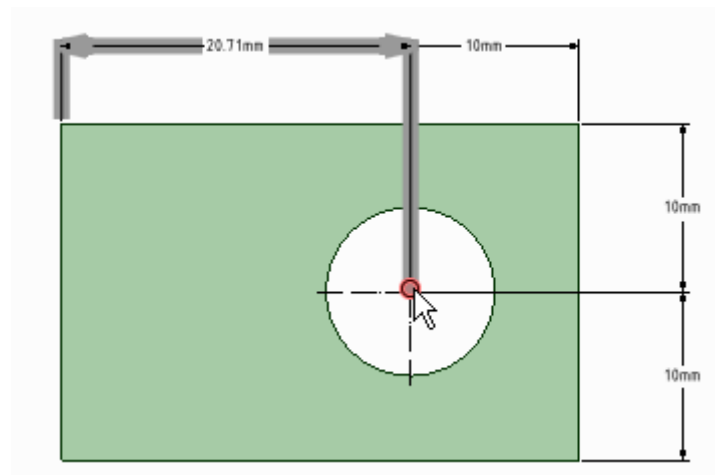
1. Create a Note
2. Attach a Leader to the underline location
3. Exit the **Leader** tool
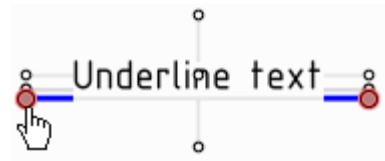
4. Select the Leader
5. RMB > Underline
6. The text is underlined
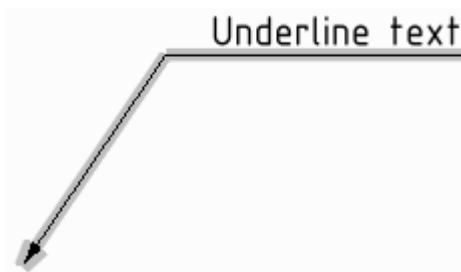
Now, when you select the endpoint of a dimension extension line, the dimension it belongs to is highlighted. Previously, there was no way to know which dimension extension you were modifying.

ISO Foreshortened dimensions now display and move correctly when switched to JIS standard. Previously, after creating and moving an ISO Foreshortened dimension, if you switched to JIS standard, you would have to reposition the dimension.

When creating Layout Curves in a drawing, the **Create curves shown only in current view** option now works correctly. Previously, if you created Layout Curves in a view and then created a new Projected View from the current view, the curves would be shown in the new view.
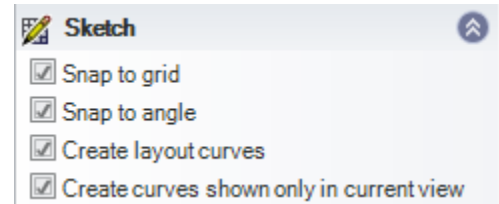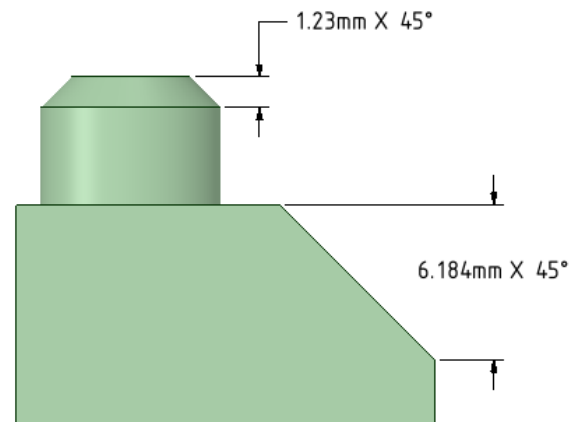

Chamfer Dimensions:

- o   Works for planar-planar and planar-cylinder chamfers
- o   **Flip Chamfer Dimension** property toggles the orientation of the dimension
- o   Example of each chamfer type is shown below

Create a Chamfer Dimension by selecting the Chamfer face and moving to an orientation that displays the dimension.
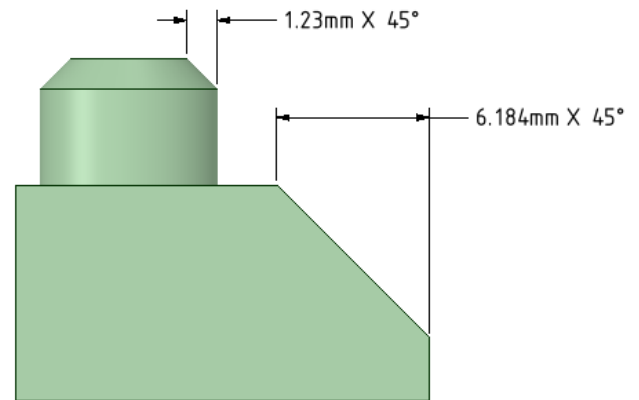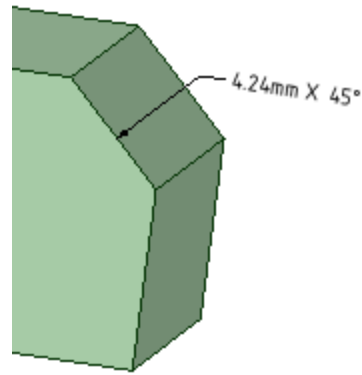

Set the **Flip Chamfer Dimension** property to True to flip the orientation.
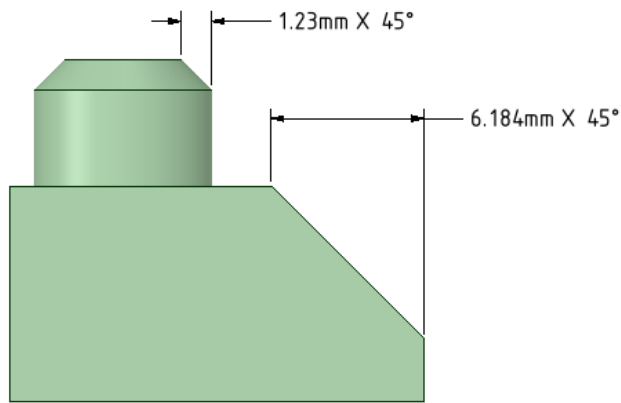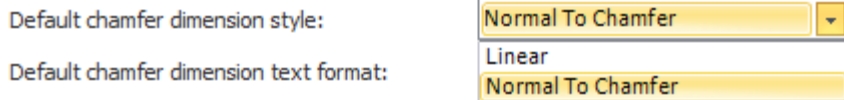
There are two visual styles for Chamfers.

- Normal to the Chamfer face

4.24mm X 45°

- Linear dimensions

1.23mm X 45°

6.184mm X 45°

Change the style in the General page of SpaceClaim Detail options.

Default chamfer dimension style: Normal To Chamfer ▼
                                        Linear
Default chamfer dimension text format: Normal To Chamfer

There is also a setting to change the Chamfer format to C<length> instead of <length>X 45.

C4.24mm

Default chamfer dimension style: Normal To Chamfer ▼

Default chamfer dimension text format: <length> X 45 ▼
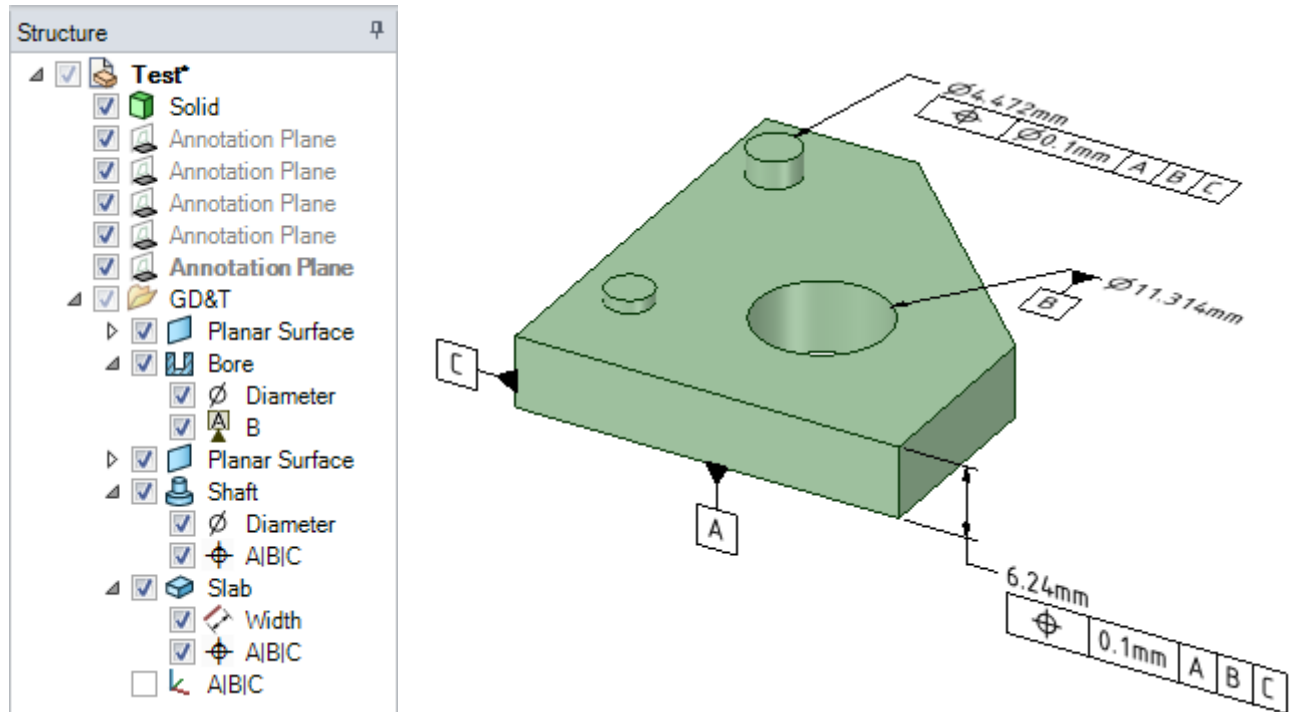   <length> X 45
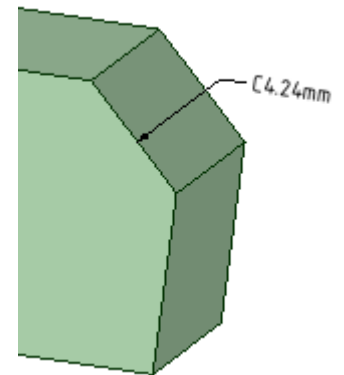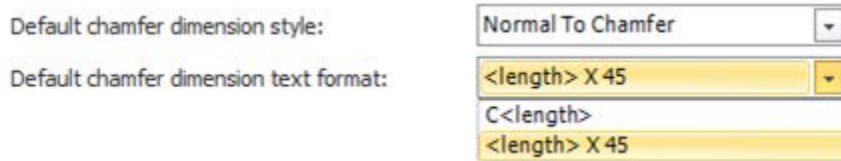
GD&T

The Structure Tree now has a GD&T folder.

Features of Size and their Dimensions are now shown in the Structure tree. The image below shows some different Features of Size displayed in the tree.

Dimensions for Features of Size are also represented in the Structure tree. Expand the Feature node to see the dimensions and other GD&T related items.

Feature of Size Dimensions shown in the tree are the Dimension Representation for the Feature, which is different than the graphical representation shown in the Graphics Window.

Deleting the dimension in the Graphics Window does *not* delete the Dimension Representation.

So, **Features of Size** can now only be deleted by explicitly deleting their dimension from the tree. They are no longer automatically deleted when their last Feature Control Frame and Datum Features are deleted.
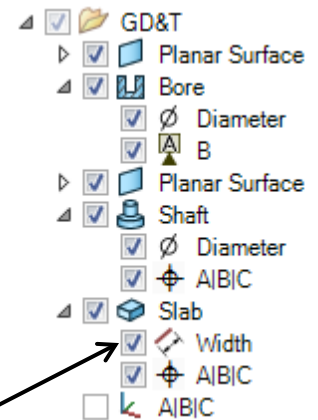

Deleting this Width dimension will delete the Slot Feature of Size.

You can now create Basic dimensions between Datum Reference Frames (DRF's) and Tolerance Zones – also referred to as Features.

For a selected Feature, the RMB contains a **Create basic dimensions** command, which automatically determines the Basic dimensions from the DRF to the Feature. All you need to do is place them.

Note that the command will not be available for every Feature in the tree. Features that define the DRF do not have basic dimensions. So, the Create basic dimensions command is not available when any of them are selected.
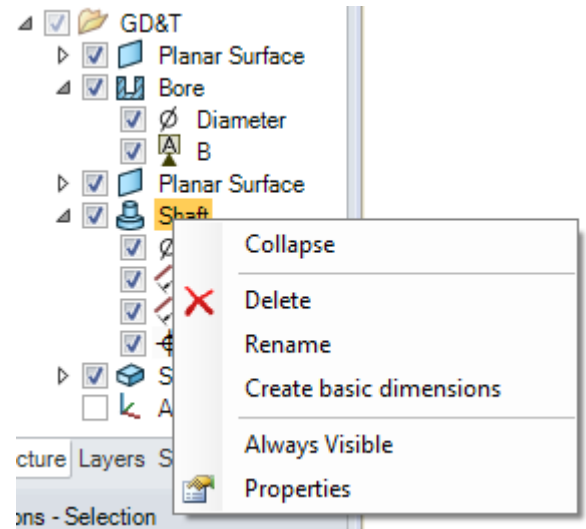
Depending on the selected feature, different sets of dimensions can be available for different coordinate directions. Tool guides allow you to switch between the directions while placing the dimensions.
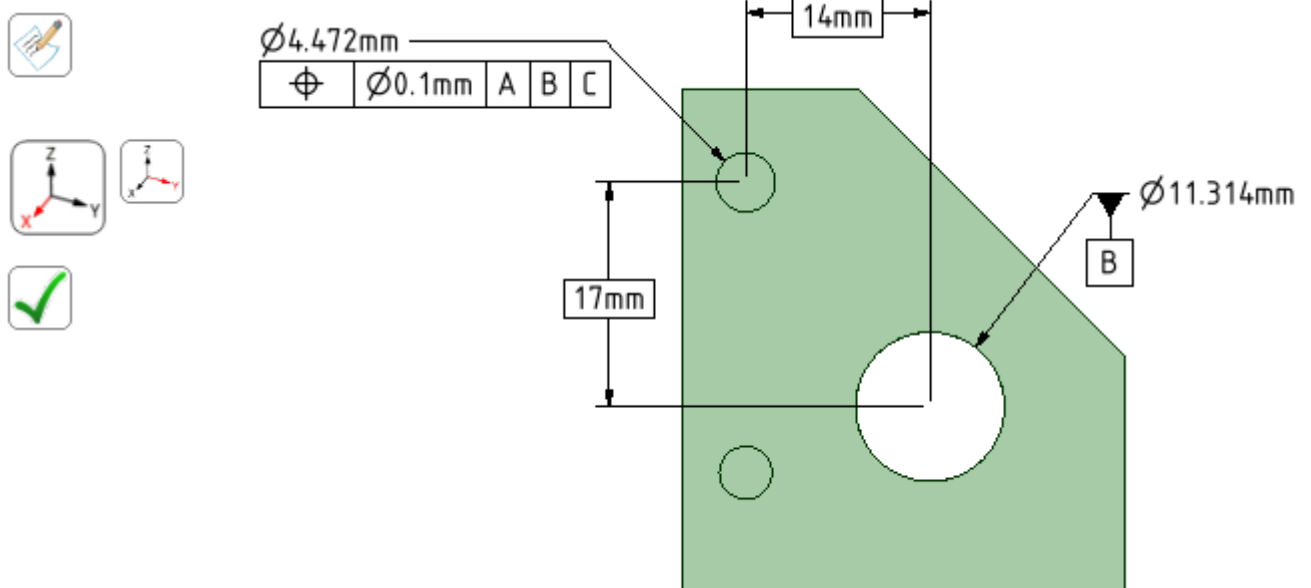
Upon entering the tool, if an annotation plane can be determined, it will become the default. If one cannot be determined, you will need to select one.

At any time during the process, you can change the annotation plane.
- Changing the plane may change how many dimensions are available for placement

You can also change which dimension to place while placing a dimension using the tool guides.

Composite Features - those which span a continuous set of faces and shown as Feature ▷ ☑ ⬛ Feature in the tree - require several basic dimensions. An example is shown below where four faces were selected to create a Profile Surface tolerance.
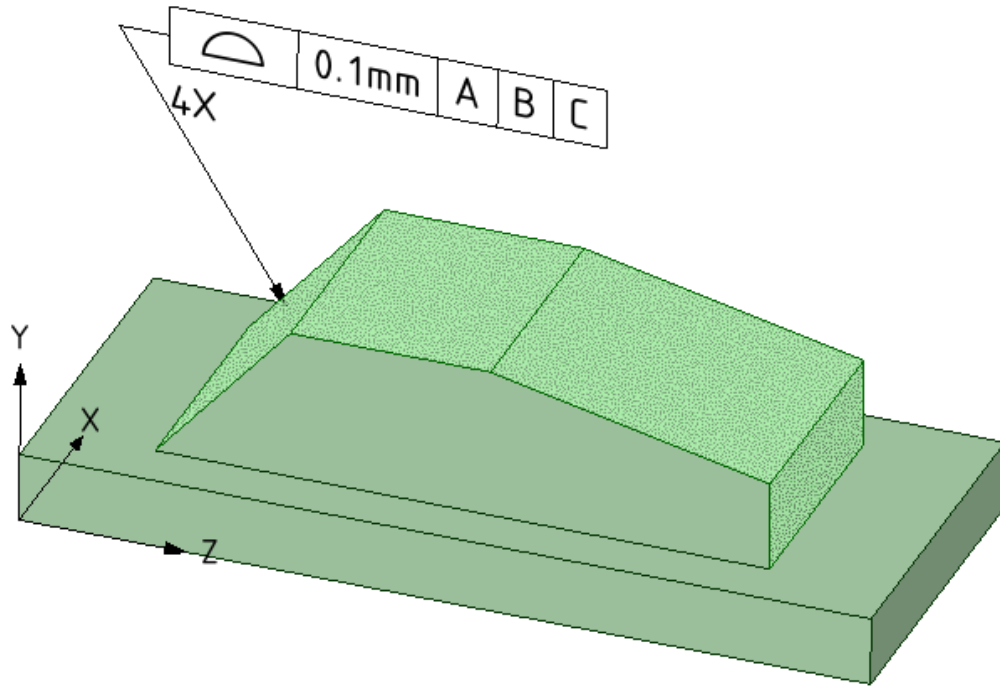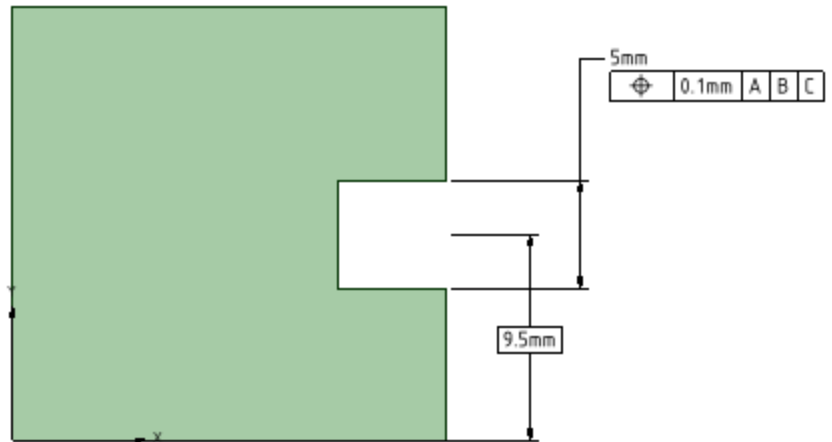

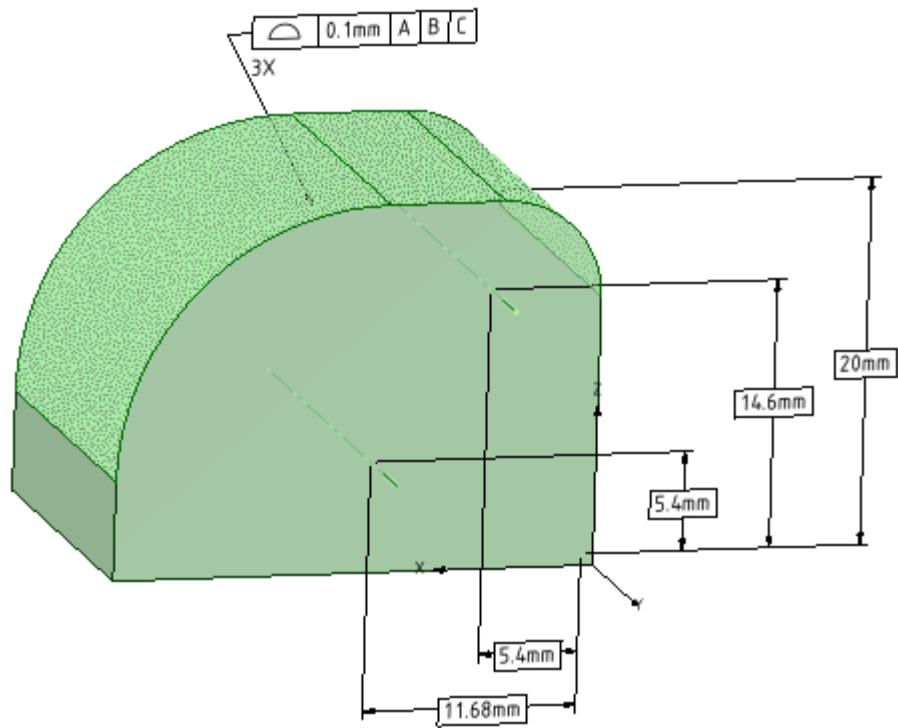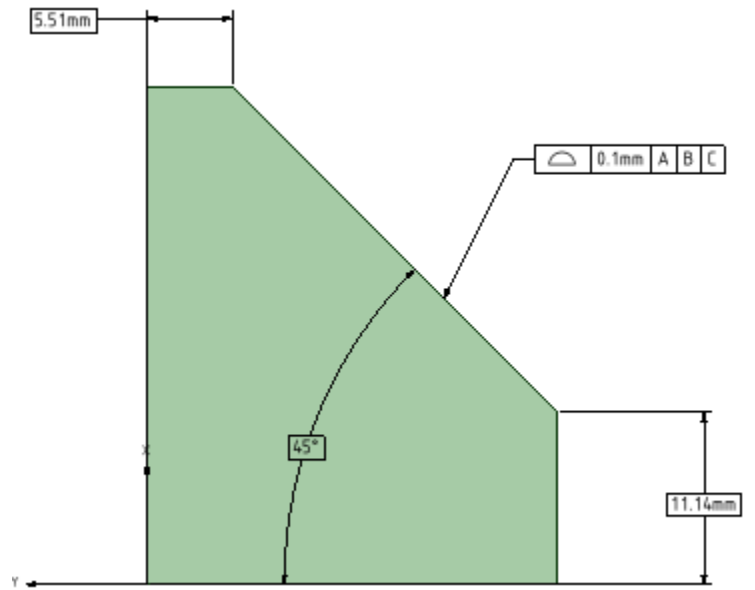Basic Dimensions for all the edges of the four faces are shown below.
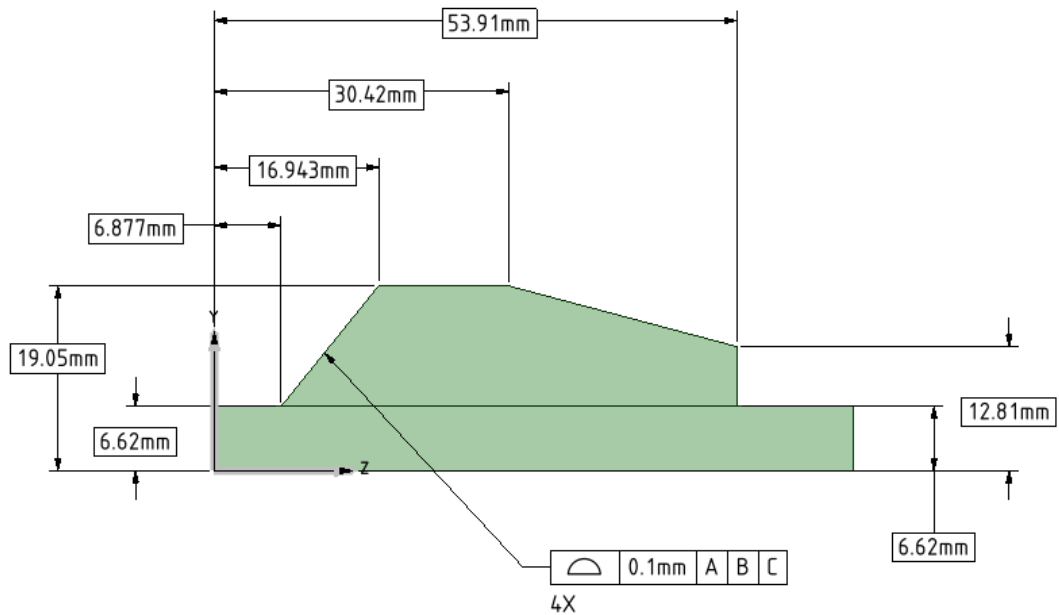
Basic dimensions for Slabs and
Slots dimension to the mid-plane.

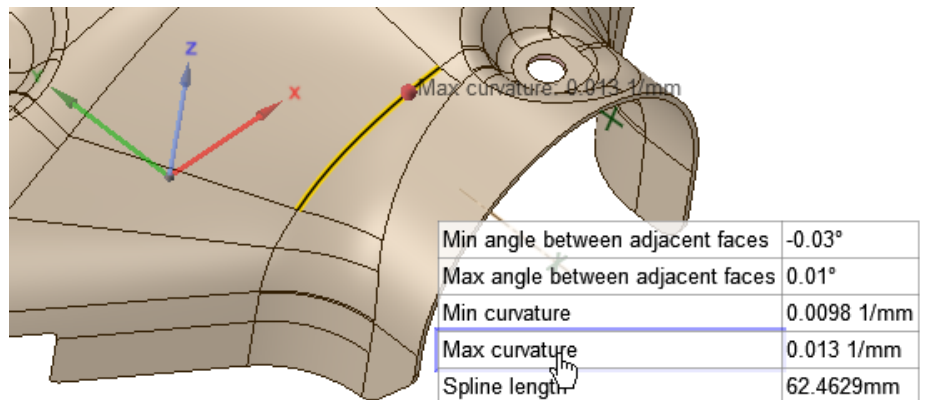Basic Dimensions can be made for cylindrical faces in Composite Features.

Angular Basic Dimensions are also supported.

Measure

When measuring Edges and Curves, you can see the **Minimum** and **Maximum Curvature** points by hovering over the measurement in the results popup table.

Min angle between adjacent faces	-0.03°
Max angle between adjacent faces	0.01°
Min curvature	0.0098 1/mm
Max curvature	0.013 1/mm
Spline length	62.4629mm

Min angle between adjacent faces	-0.03°
Max angle between adjacent faces	0.01°
Min curvature	0.0098 1/mm
Max curvature	0.013 1/mm
Spline length	62.4629mm
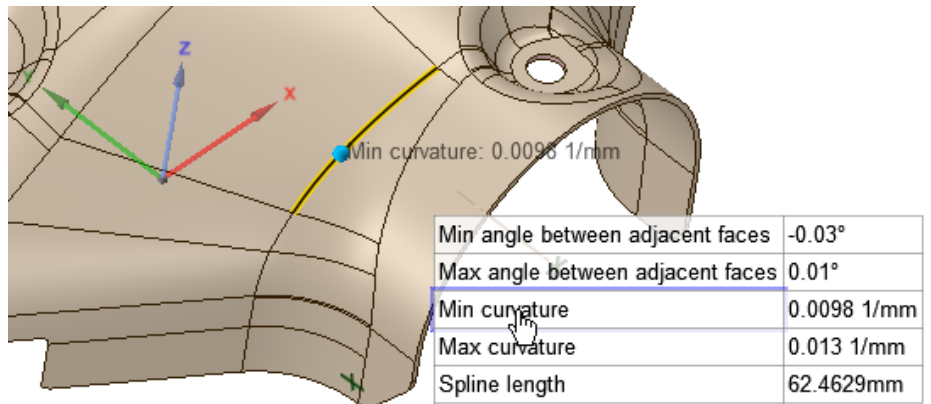
The exponent "^1" has been removed from the unit symbol when the exponent is 1. Some examples of where this occurs are Curvature, Centroid X & Y, and Shear Center measurements. The images below show the difference for Curvature.

Min angle between adjacent planes	-0.03°
Max angle between adjacent planes	0.01°
Min curvature	0.0098 1/mm^1
Max curvature	0.013 1/mm^1
Spline Length	62.4629mm

Min angle between adjacent faces	-0.03°
Max angle between adjacent faces	0.01°
Min curvature	0.0098 1/mm
Max curvature	0.013 1/mm
Spline length	62.4629mm

Minimum Distance and **Minimum Distance to Axis** now work for mesh vertices.
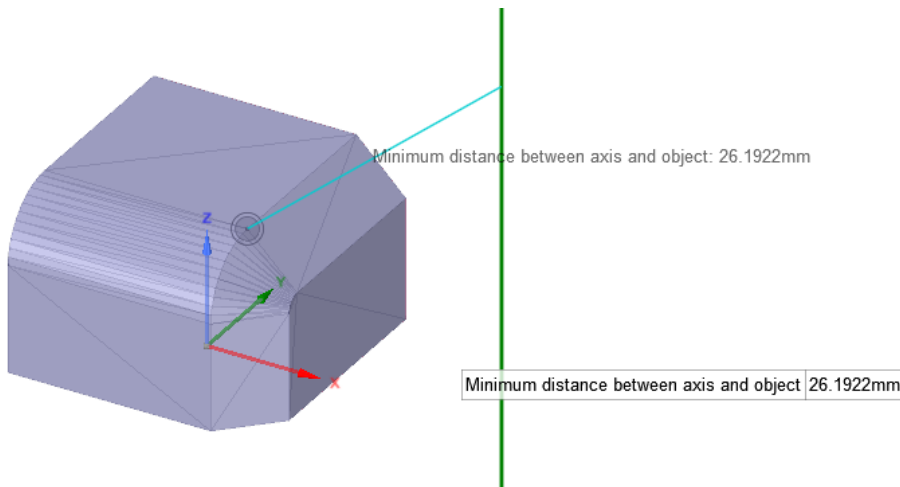
Repair

Two new tools have been added to the Fix group in the Repair ribbon.

Overlapping Faces finds faces that overlap but does not *Fix* them yet.

- False positives are filtered out

Bad Faces finds faces that are bad but does not *Fix* them yet.

- Bad edges flagged in facet creation are shown

The **Overlap Faces** tool now uses multi-threading. Depending on the processor, overlapping faces can be found up to 10X faster.

A **Clip Volume** checkbox has been added to the Navigate group.

- It acts to create Clip Volume spheres around all the current Repair errors.
- The forward and back arrows act the same as before, navigating between the found errors - optionally Zooming to Fit - but now they are smaller.
- The small Select icon converts the current error to a real selection for defining the Clip Volume (and necessarily exits the current find tool – for manual fixing)
- Eases the identification of errors buried deep within complicated models

Errors found
Navigate to an error   Clip Volume On   Zoom in
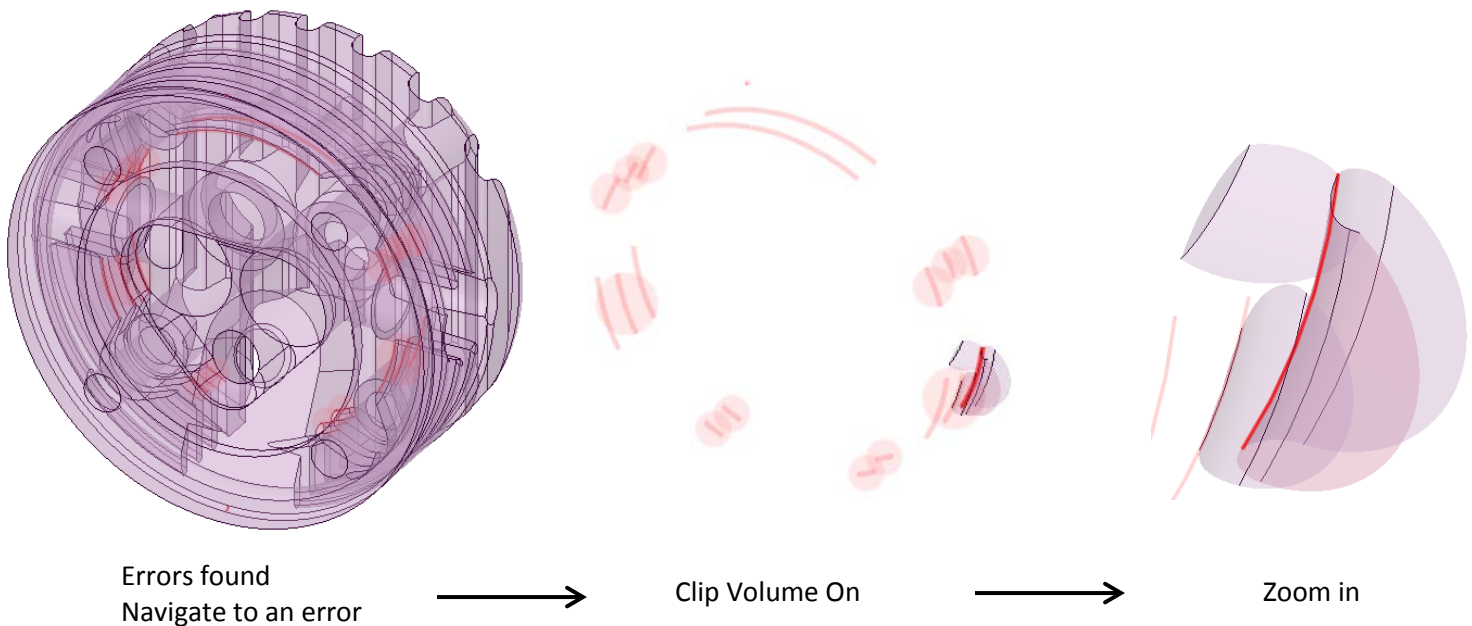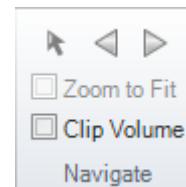
Fit Curves

The **Fit Curve** tool now has a **Correct tangency** option, which does a dramatically better job fitting Arcs, as shown in the example below.

Other examples can be found here:
\\192.168.2.17\depts\R&D\Justin\Mesh curve fit samples\
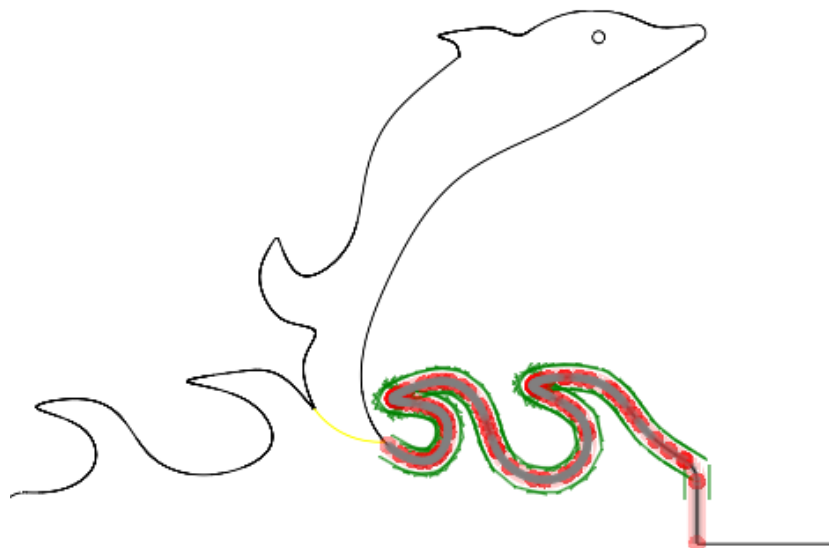
This example does *NOT* correct tangency. Notice that only one tangency is created at one end of the curves.
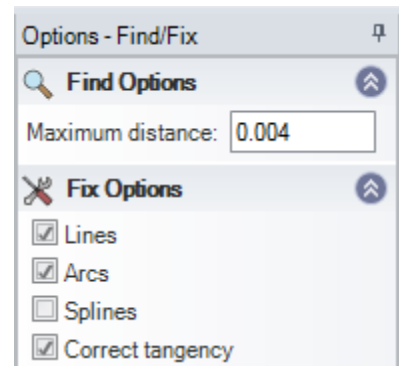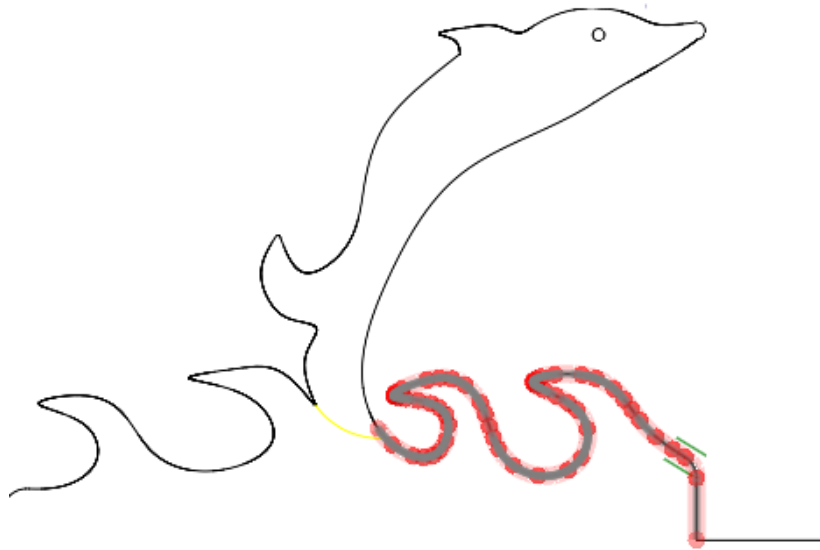
This example corrects tangency and produces tangency between all of the selected curves.

Prepare

A **Short Edges** tool has been added to the Analysis group in the Prepare ribbon. This is a partial implementation that finds edges smaller than a specified length. Small edges reduce the quality of meshes used for FEA and CFD applications.
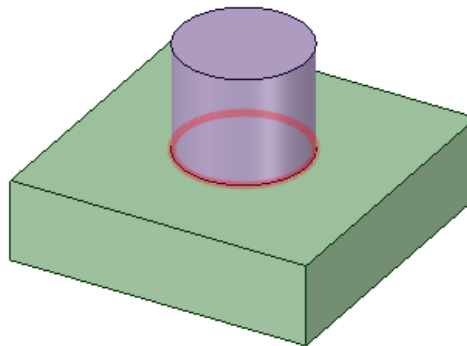
Midsurface Face Thickness has been added to Power Selection, for both faces and bodies.
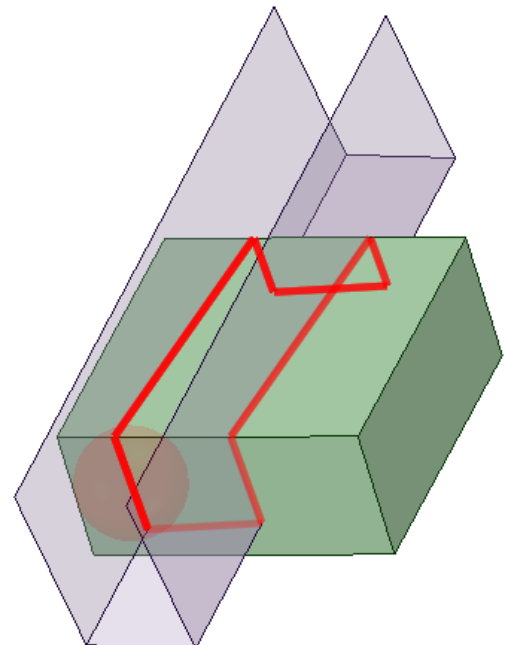
The **Imprint** tool now supports:

- **Body intersections** – coincident faces that belong to intersecting bodies
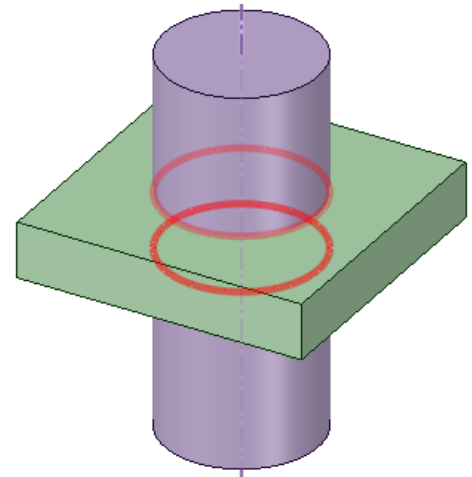
- **Body coincident faces** – Faces on separate bodies that are coincident

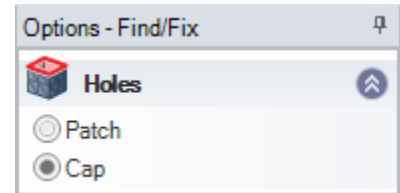- **Plane-Body coincident faces** – faces of a body that lie within a plane

Facets

The **Mesh** ribbon tab has been renamed to **Facets.**

Holes Tool:

There is a new **Holes** tool in the Cleanup group of the Facets tab.

- Find-and-fix workflow
- You can choose between Patch (smooth) and Cap (flat) filling of the holes

In the mouse example below, the button needs a smooth fill and the bottom simply needs to be capped.

The Holes tool finds to areas to fix.

Click the **Exclude Problem** tool guide and select the bottom hole.
- If you have multiple holes to exclude, you can use box-selection
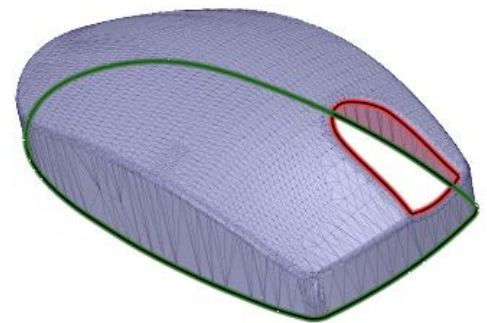
Choose Patch in the options and click the Complete tool guide. The button area is filled with a smooth patch.

The bottom is found again. Select the Cap option and click Complete to fill the bottom with a flat mesh.

Separate All Tool:

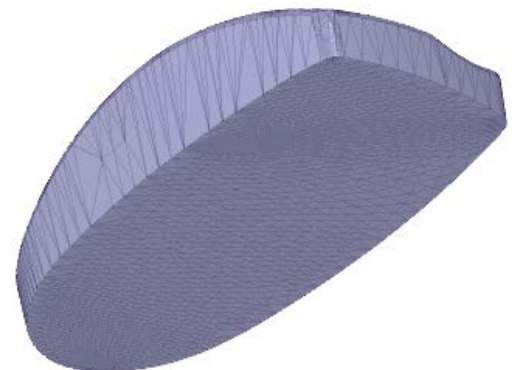The **Separate All** tool has been added to separate all shells in a mesh. It works like the **Separate** tool but converts all shells into separate mesh bodies. The separate tool only converts one shell at a time.

Separate All Pieces has been changed to **Separate Pieces** and has been removed from the RMB.

- o   Tool Guides have been added for **Separate Pieces** and **Separate All Pieces**
- o   Only meshes that can be separated are selectable

To separate pieces
1. Click Separate Pieces in the Cleanup group of the Mesh tab
   - o   Tool Guides appear
   - o   Options are available (placeholders for now) for Connected and Over-Connected meshes
2. Choose Separate Pieces (default) or Separate All Pieces
3. Select the mesh
4. The mesh separates and the pieces display in the tree

The Organize group has been rearranged.

- Split All has been renamed to Separate All
- 24 x 24 icons for Separate, Separate All and Join
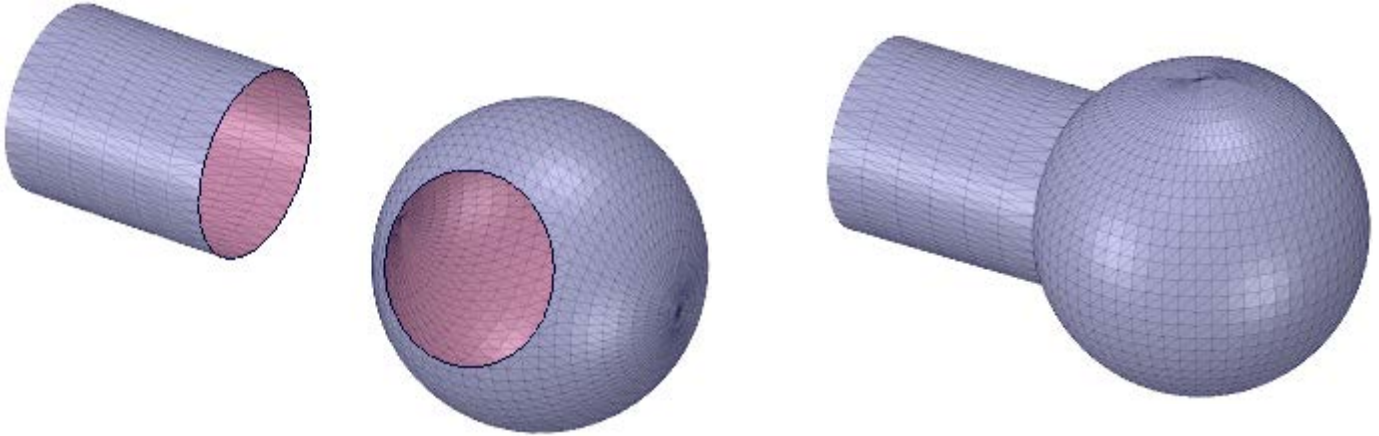- Separate All has been put in a drop-down menu under Separate
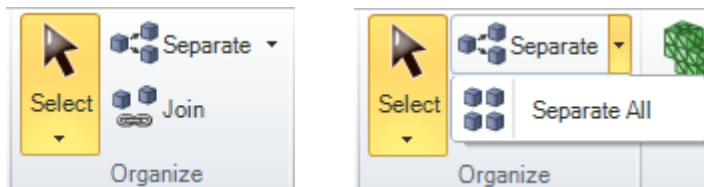
Join Tool:

The **Join** tool has been added to the Organize group. It joins meshes that have been separated using **Split All** and/or **Separate.**

Merge Tool:

The **Merge** tool now works on two *Open* meshes to automatically create the union that results in a single, closed mesh.

Split and Boolean Tools:

Split and the Mesh Boolean tools (**Merge**, **Subtract**, and **Intersect**) now allow selection of objects in the Structure Tree.
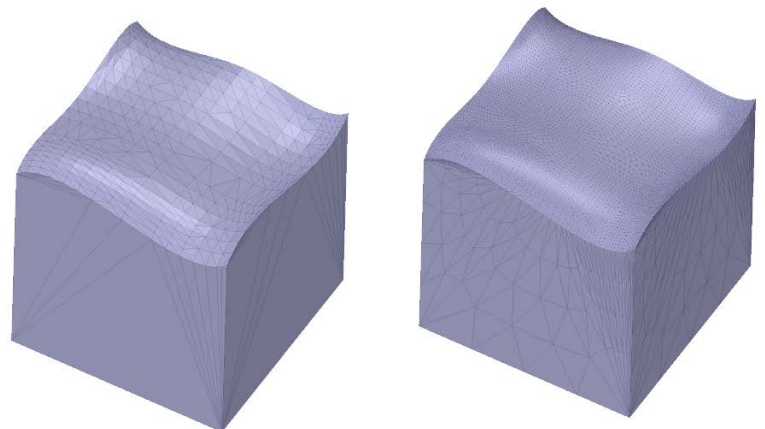
Smooth Tool:

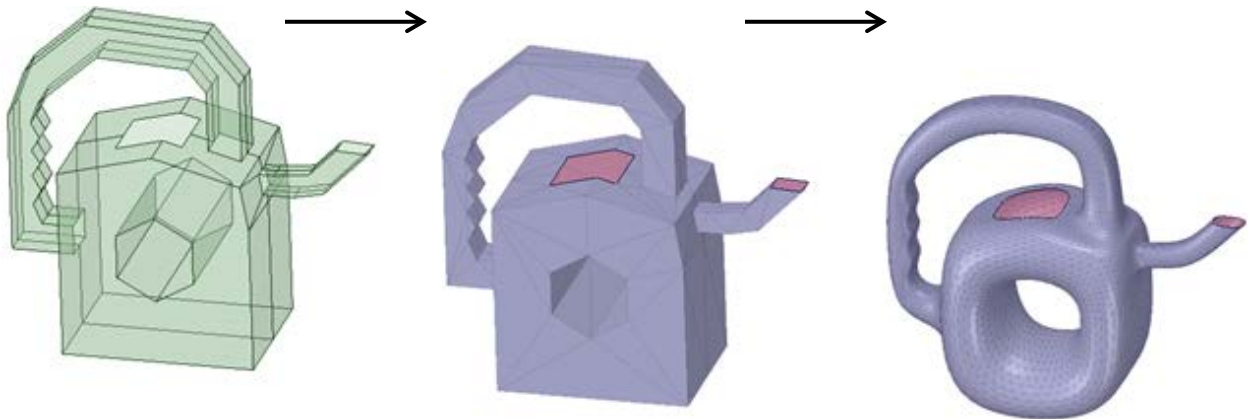The new **Smooth** tool, and its options, can be used to smooth out an existing mesh.

Smooth is not a one-and-done operation. You can execute the command multiple times, producing a finer mesh with each click.

This technique has two different applications:

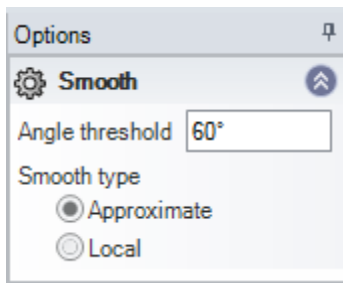- Use it to smooth a mesh that is only slightly rough

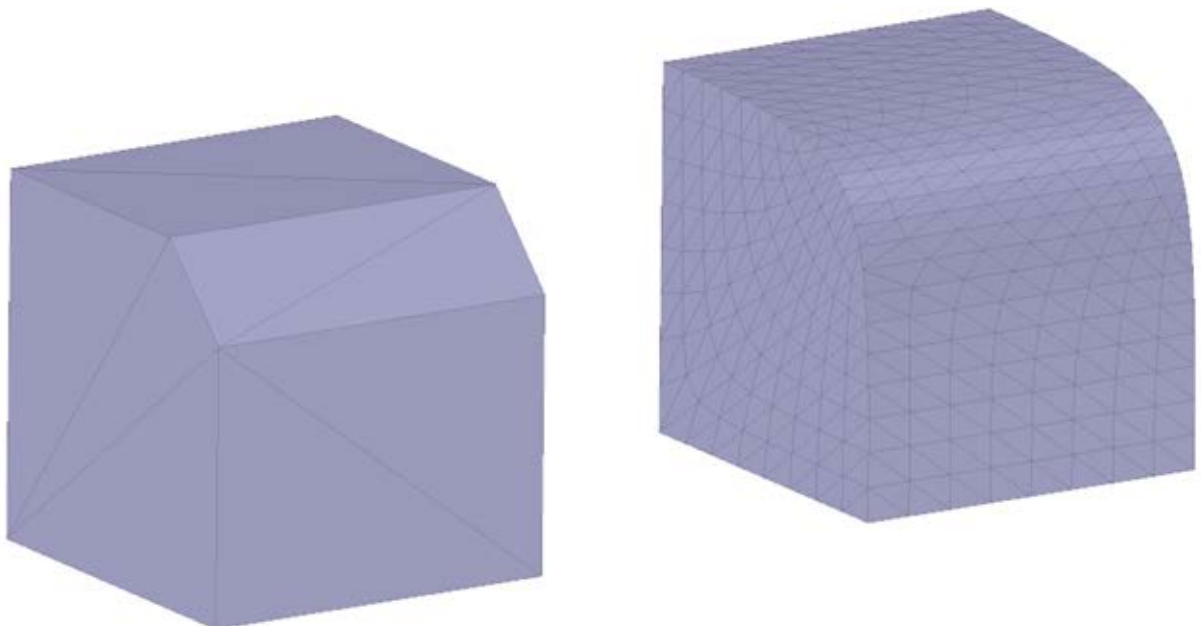- Begin with a rough control mesh and produce something very smooth

Smooth Options

**Angle Threshold** determines if an angle stays sharp or gets "smoothed".

The example below shows an example of a block with all 90° edges except for two which are 45°. With an Angle Tolerance of 60°, only the two 45°edges will get smoothed and the 90°edges are left alone.

**Smooth type** determines which smoothing algorithm is used.

Approximate:
- The original vertex positions and the new vertex positions are allowed to move
- The amount of movement depends on how smooth the surroundings are
- In sharp areas, the amount of movement will be large
- In smoother areas, the amount of movement will be barely noticeable
- Because of the movement, the Approximate algorithm will, in general, produce smoother results

Three Approximate operations
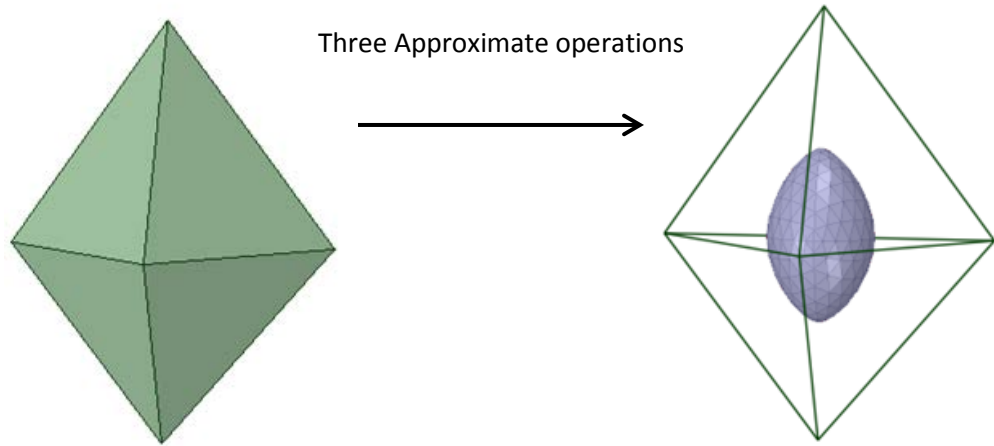
Notice that, with the **Approximate** method, vertex movement allows the final shape to deviate more from the original shape.

Local:
- No new vertices/facets are created
- The algorithm operates locally to "knock down" spikes

Convert to more facets

Three Local operations
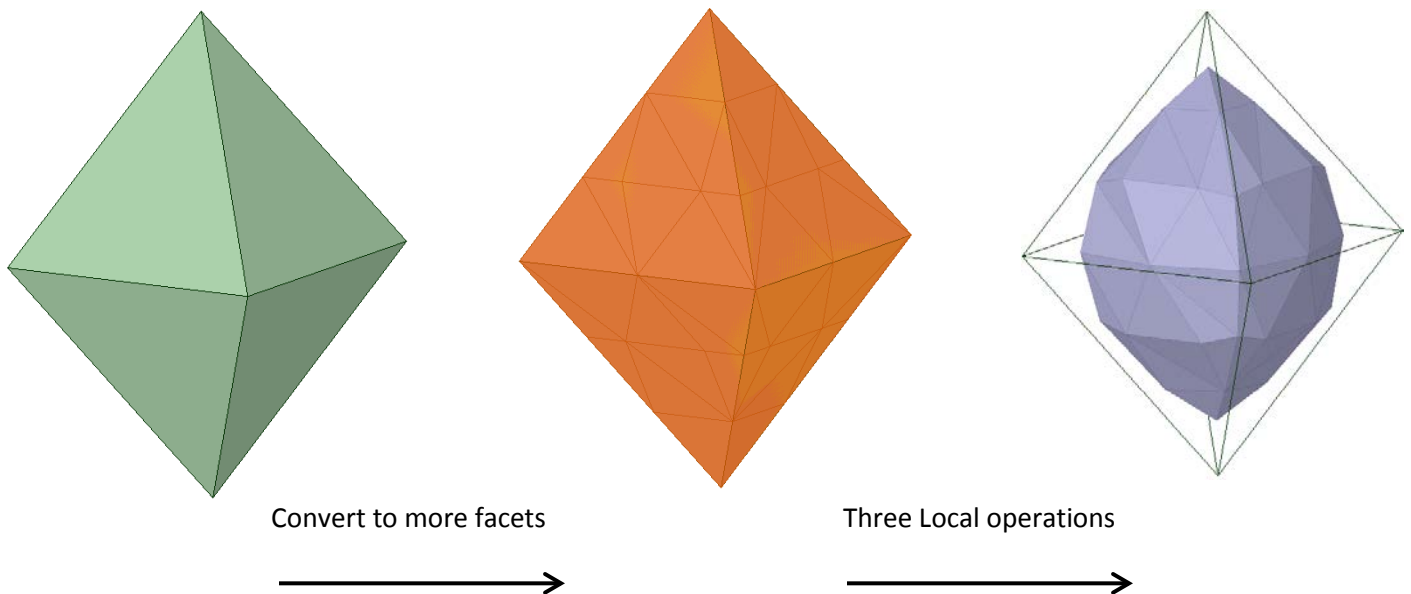
Regularize Tool:

The **Regularize** tool has been added for making aspect ratios more consistent in mesh bodies and selected facets. An example of using the tool is shown below. The aspect ratios of all of the triangles are brought closer to each other to reduce the differences between facets in the selection.
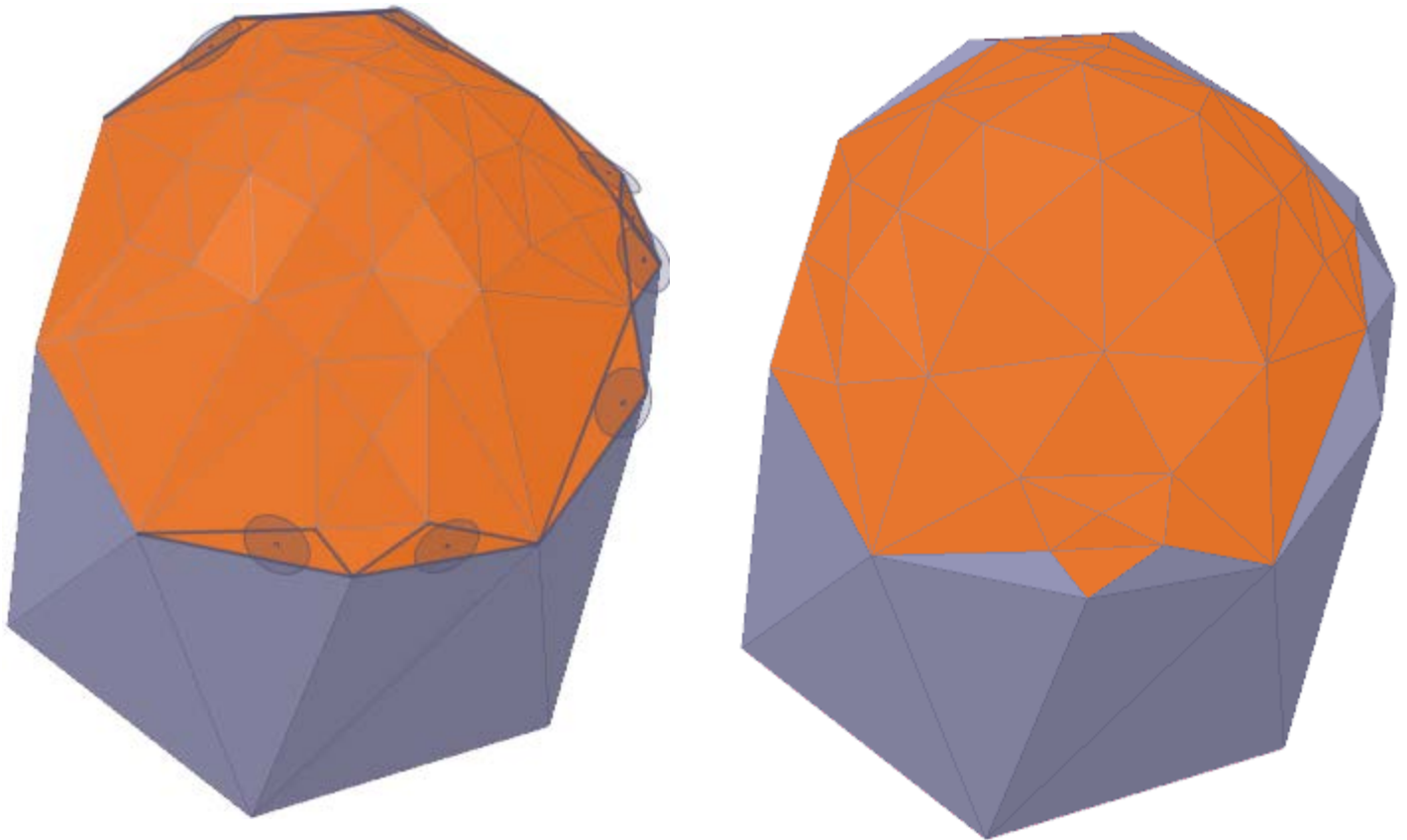
Thickness Tool:

The Thickness tool now has a **3D Printing Direction** tool guide

> o Use it to set the normal direction representing the machine vertical

Click a mesh to analyze.

3D Printing Direction
Click a line, edge, or axis to specify 3D printing direction.

Overhangs Tool:

A new **Overhangs** tool has been added to the Analyze group. It detects and highlights areas of the mesh that protrude out at angles up to 90°. You can set the angle to look for in the Options panel by entering a value between 0 and 90. The example below is at 65°. These overhangs need to be considered when setting the model up for 3D printing.
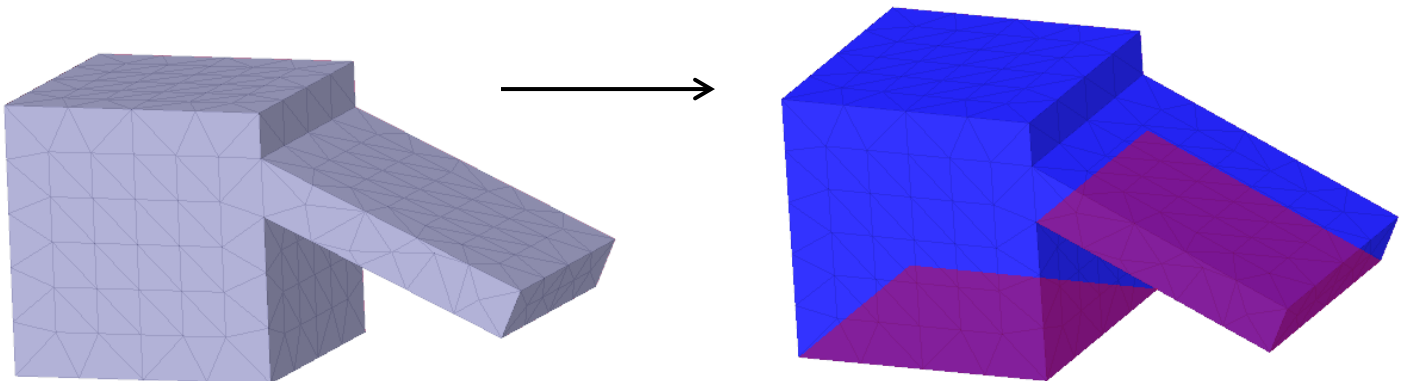
Convert Tool:

Multi-threaded faceting is now used for **STL export** and **Mesh > Convert**.

Mesh Highlighting:

The highlight scheme (Classic, Default and Custom) for Meshes has changed to match the scheme for Bodies.
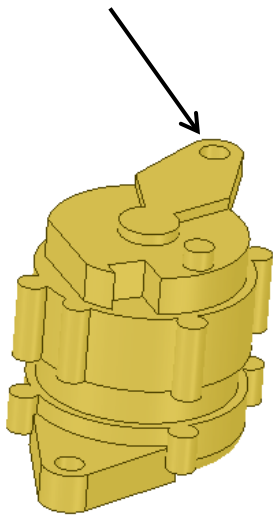
Body pre-highlight   Mesh Pre-highlight   Body pre-highlight   Mesh Pre-highlight

Edge and face preselect and select highlight have also changed.

Custom Highlight   Custom Selected
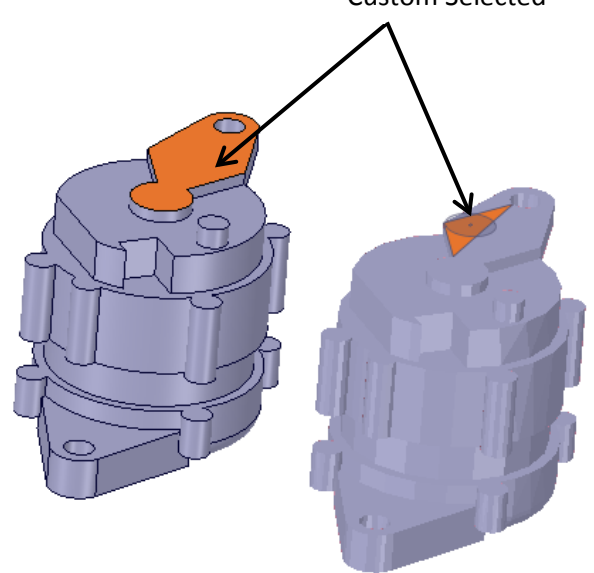
You can now use **Ctrl+Alt+Shift+H** as a shortcut to change the Mesh highlight scheme.

Selection:

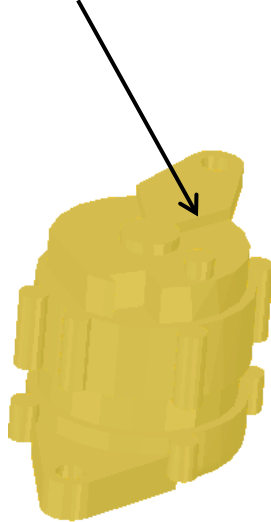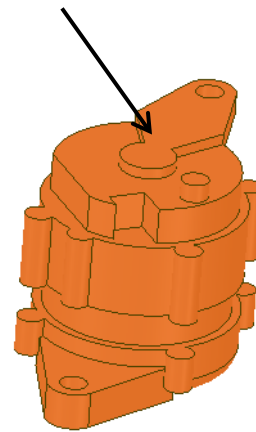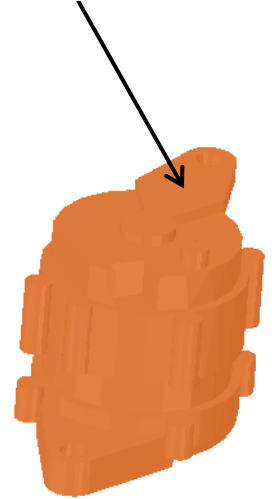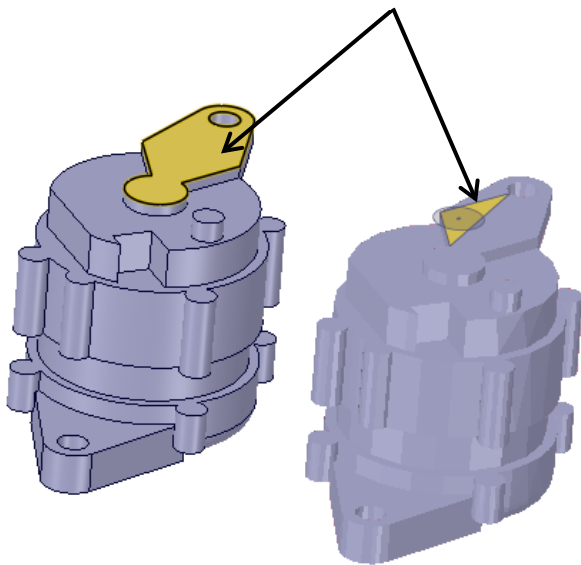- Use **Shift+Double-click** on sharp edges to select a set of mesh edges that form an isoline – edges that are nearly continuously tangent to each other as the line propagates.

  The selection will be the best polyline found and not necessarily a complete loop. As you can see here, the selection stops at the cylinder because it is adjacent edges are not within some tolerance of "tangent".
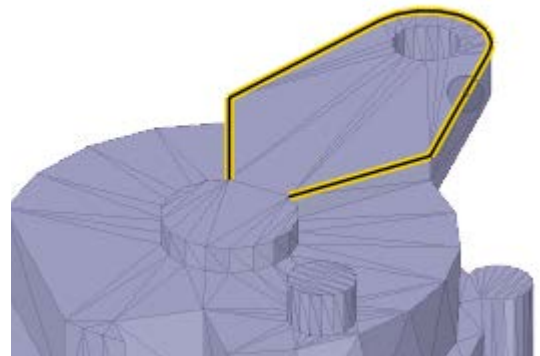
- Use **Ctrl+Shift+Double-click** on sharp edges to select multiple mesh edge isolines
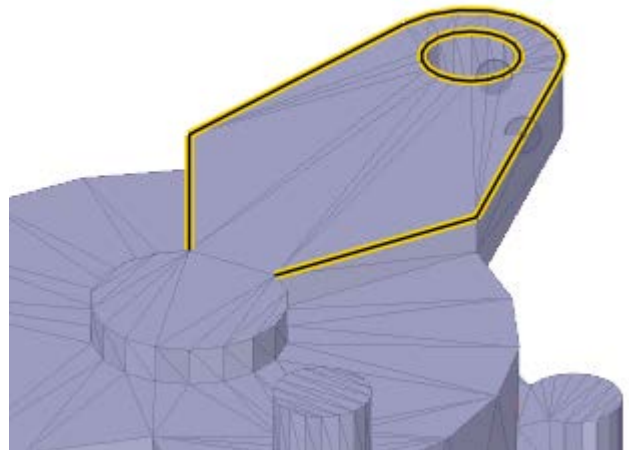
- Use **Double-click** on *Boundary* mesh edges to highlight and select the boundary edges
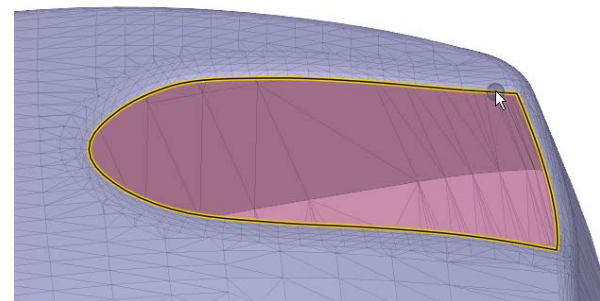
- You can now Select-All from a seed selection using **Ctrl+A**
- You can now Invert the selection using **Ctrl+Shift+I**

Pull/Move:

The **Pull** tool now works with selected mesh facets. Note that triangles at the boundary are simply stretched to connect the new and the old triangles in these cases.
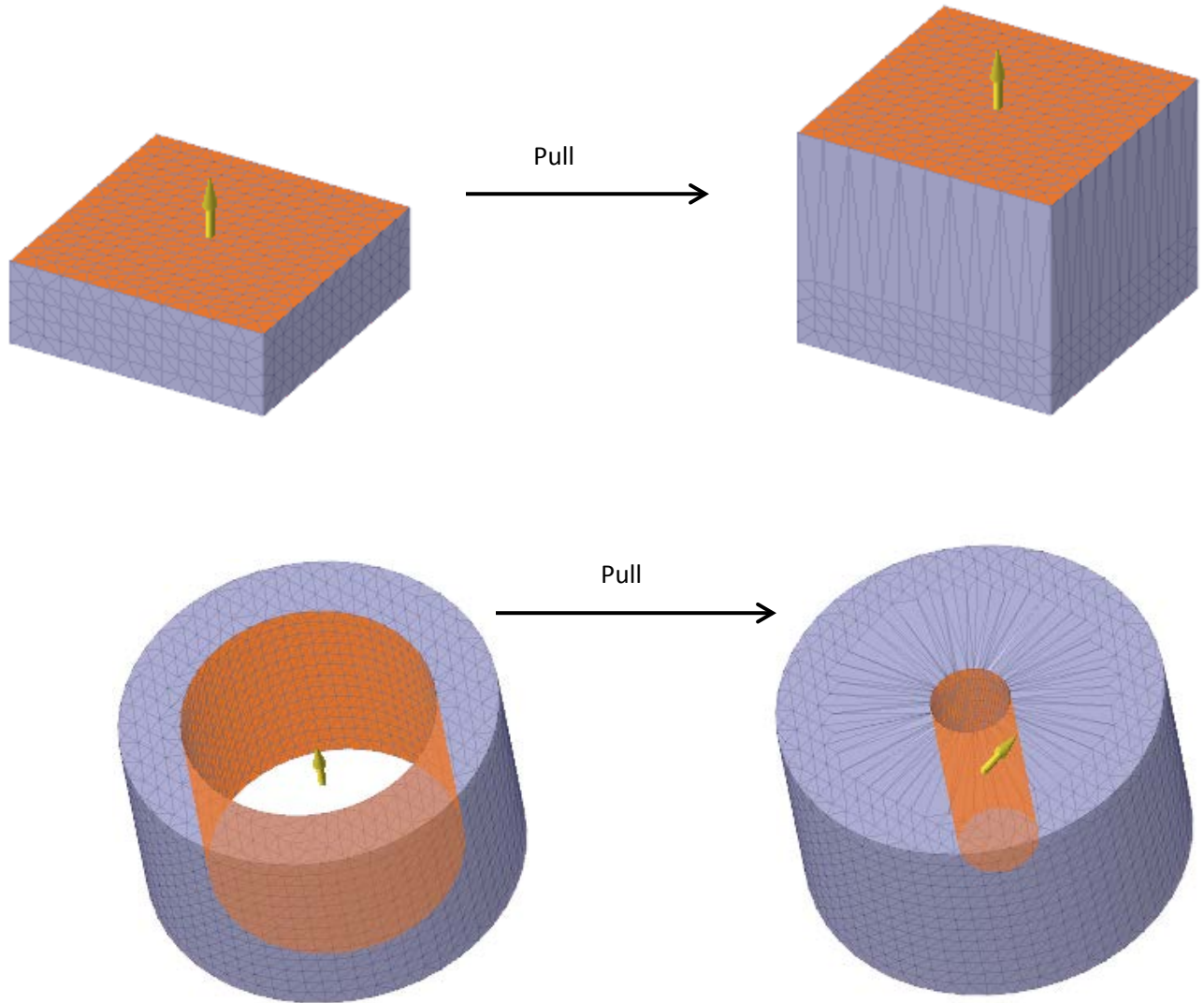
Non-uniform scaling of meshes now works by allowing the X, Y and Z directions to be toggled On/Off.
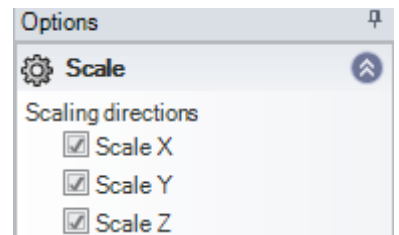
The **Pull** and **Move** handles now orient themselves to the face-wise normal of the selected facets.
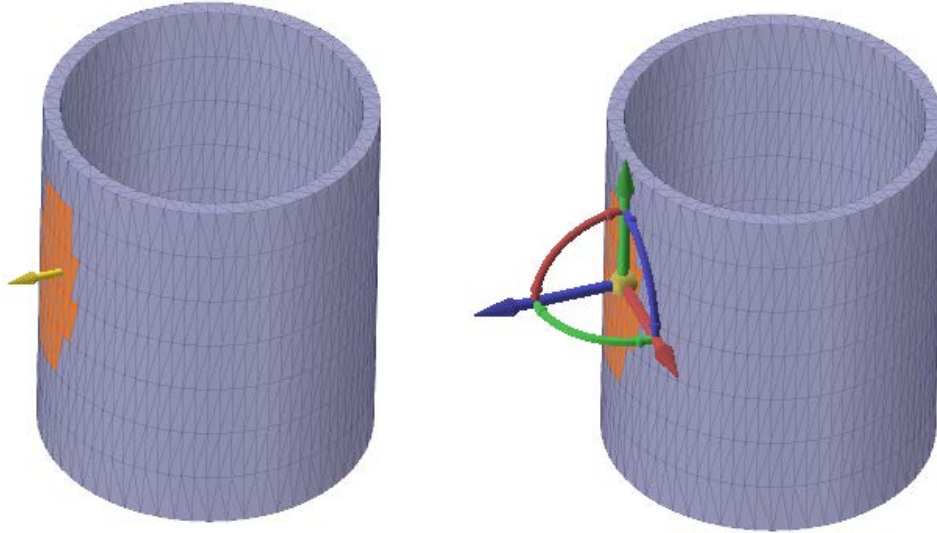
The **Move** tool axes now align on Alt-selection of a mesh edge.
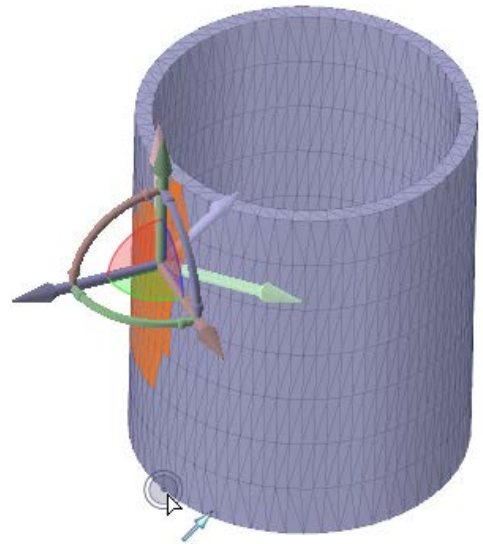
The **Move** tool can now be used on Faces, Edges, and Vertices. Previously, you could only use Move on mesh bodies to move and create copies of the body.

Alt-selection for direction, and **Up To** for distance, can also be used.
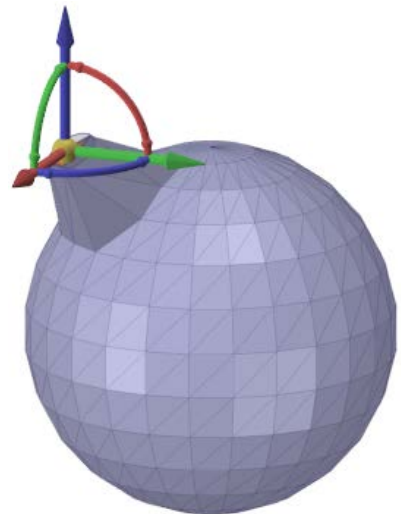
Moved mesh faces are recognized in copied components. This is the same behavior as solids and surfaces.
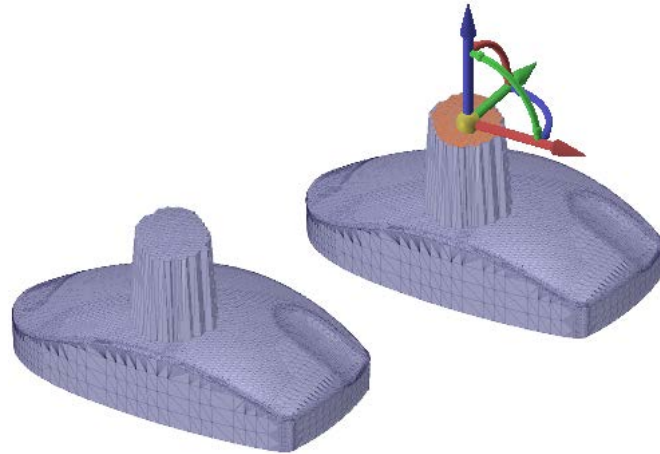
You can **Move** selected mesh edges, whether boundary loops or isolines.
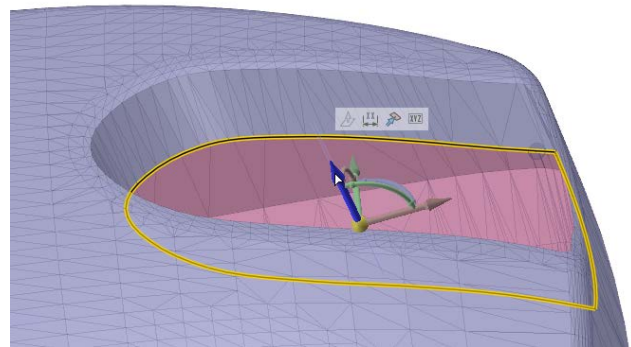
Measure:

- You can now use the Deviation tool to do mesh-to-mesh comparisons. Previously, you could only compare a mesh to a solid body.
- Mass Properties now works for meshes and is enabled through the **Options>Debug** panel in SpaceClaim Options
- The **Select Bodies** tool guide now selects mesh bodies and not mesh points
- Meshes are included in the analysis when selecting components in the Structure Tree
- Measurement values are now correctly rounded off
- Measure now treats open meshes like surface bodies
    - Only calculate surface area
    - Does not allow a mixed selection of solids and closed meshes
- You can now measure the area of selected facets
- The area of a selected facet or of an area selection of facets is now displayed in the Status Bar
- Selecting a mesh body in the Measure tool now reports the total area of the body

    - Works correctly for a mesh that has been uniformly scaled

    - Works for transformed components

Replace Tool:

You can now use the **Replace** tool on a selected set of facets with a surface. Previously, mesh facets could not be selected in the tool at all.

Replace
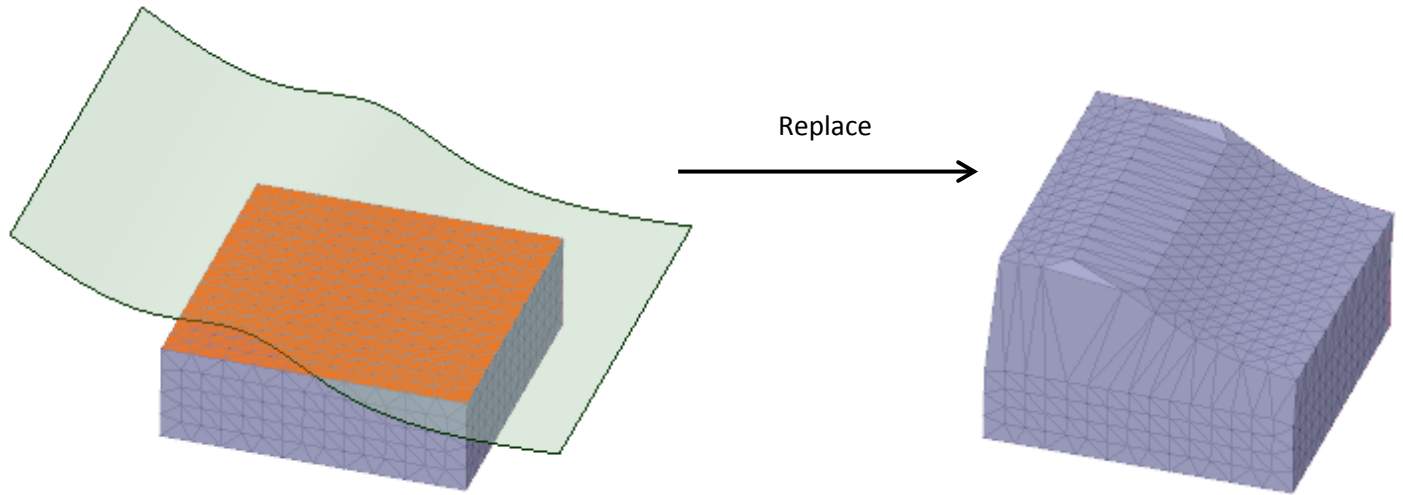
Fill Tool:

Fill now works on selected mesh boundaries.

Fill selected loop
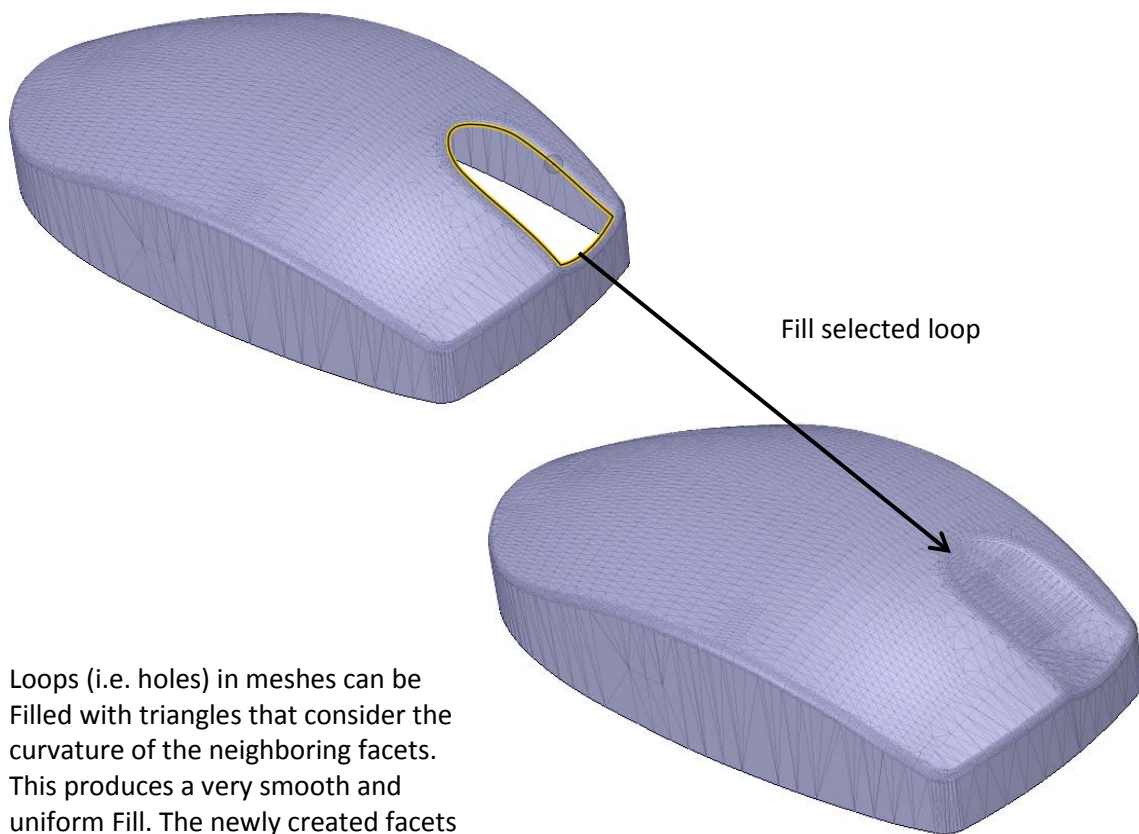
Loops (i.e. holes) in meshes can be Filled with triangles that consider the curvature of the neighboring facets. This produces a very smooth and uniform Fill. The newly created facets are added to the selection set afterwards.

Multiple loops can be filled at once in three cases.

o   Separate loops on the same mesh object that are *NOT* intended to be joined
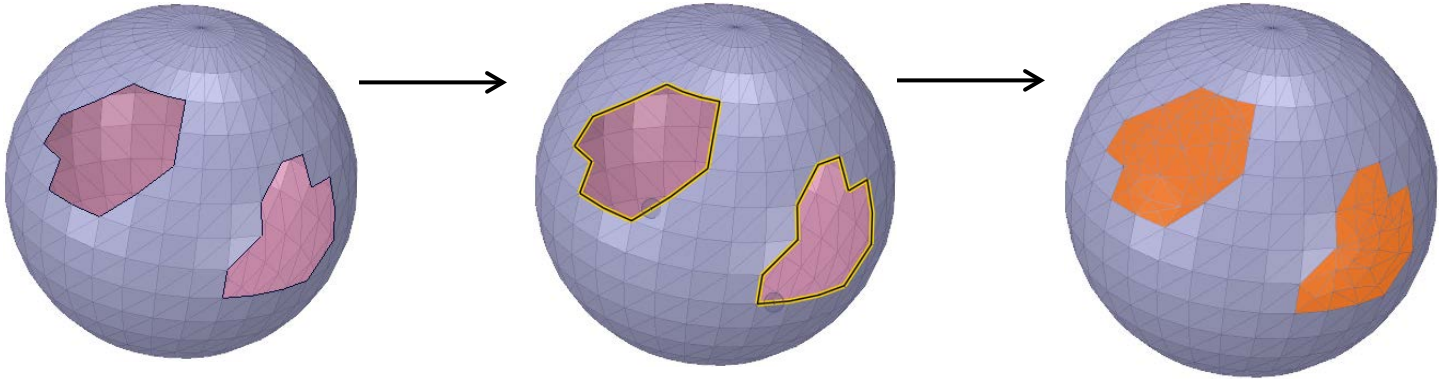
o   Separate loops on the same mesh object that *ARE* intended to be joined
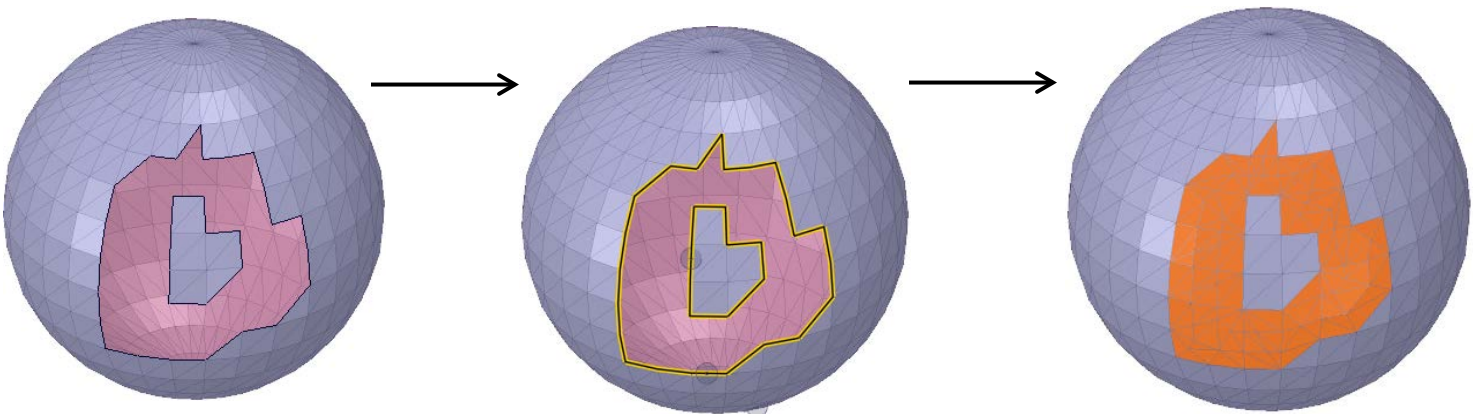
o   Separate loops on different mesh objects that *ARE* intended to be joined
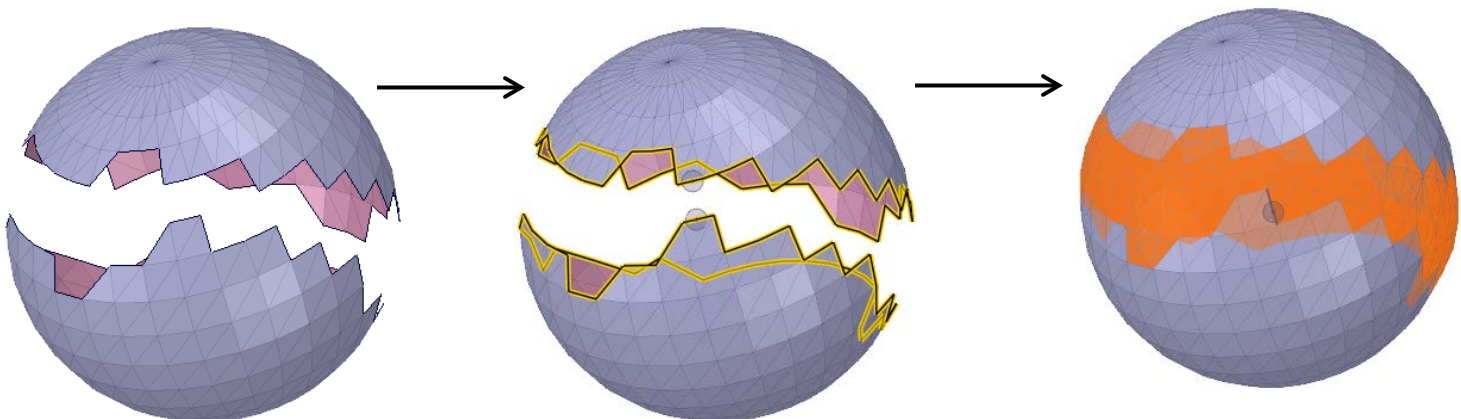
Loops on exterior and interior shells, when filled simultaneously, will create a wall thickness. Fill the interior and exterior loops separately to continue the shell.
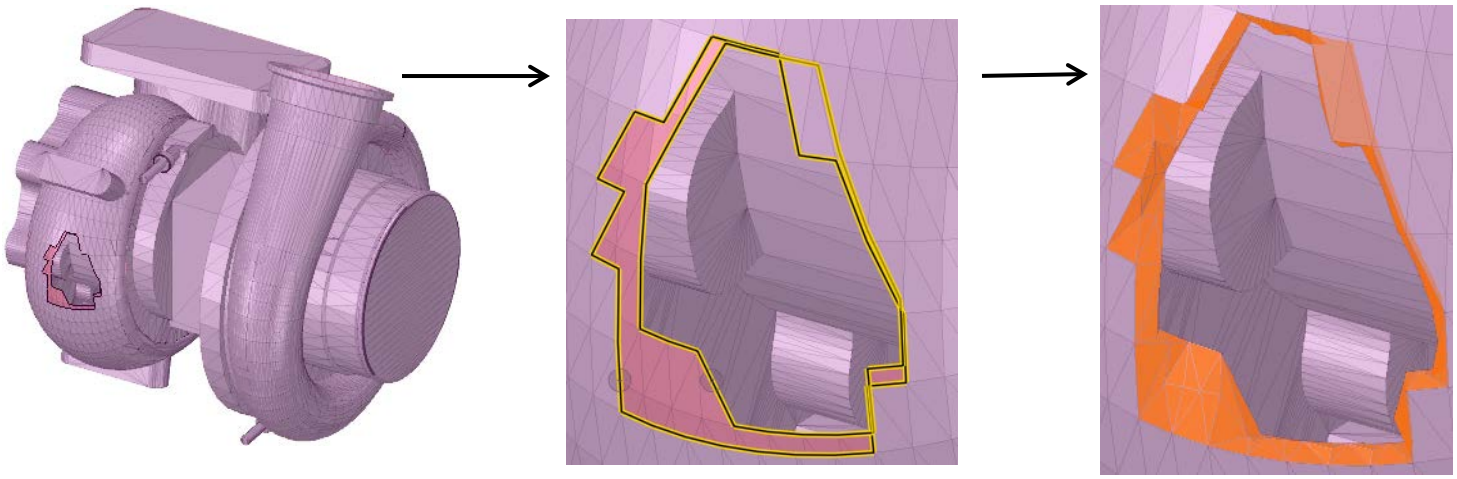
Enclosure Tool:

The **Enclosure** tool in the **Prepare** tab has been enabled for mesh objects. It has only been enabled for use with mesh objects. It does not work with the Update functionality.
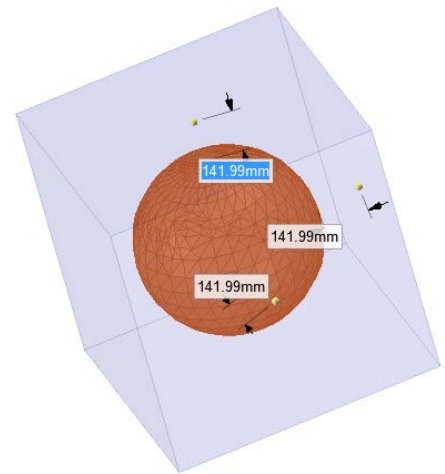
Volume Extract Tool:

The **Volume Extract** tool in the **Prepare** tab has also been enabled for mesh objects with color change and leak detection.
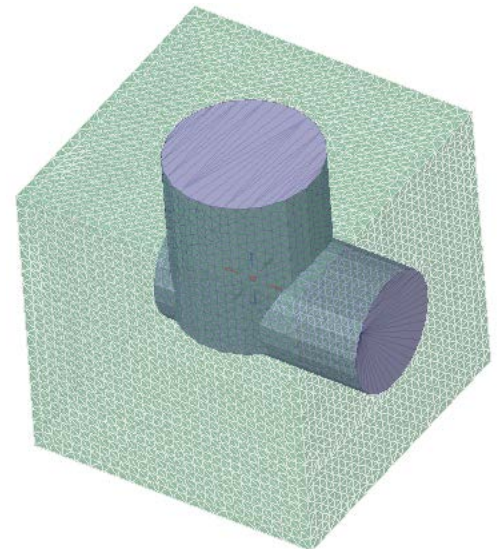
General:

All tessellation options in SpaceClaim File Options have been standardized.

- Angle deviations for Coarse, Medium, and Fine are now 16°, 8°, and 4° respectively
- A problem in OBJ options has been fixed so that the slider and text fields update correctly when a different preset is selected
- The VRML option page now displays the Angle the same as other formats (e.g. 16° where it was displayed as 16.0000)
- A problem has been fixed with VRML options so that now when you change the preset and then save the settings, the correct preset is written to the file

You can now assign Material Properties to selected mesh bodies.

Mesh material properties are preserved after operations such as:

- Copy/Transform
- Move to component
- Mesh Boolean operations
- Reduce

Material ID is maintained for the following:

- **Mesh > Export**
- Conversions between mesh and solid
- Converting from simple mesh to connected mesh

Manufacturing

Deburring Tool:

There is a new **Deburring** tool in the
Manufacturing group of the Insert tab.

- Creates an offset curve profile representing the center of a tool
  traversing a set of edges

- Using the tool options, you can change the tool radius and the
  width of the cut
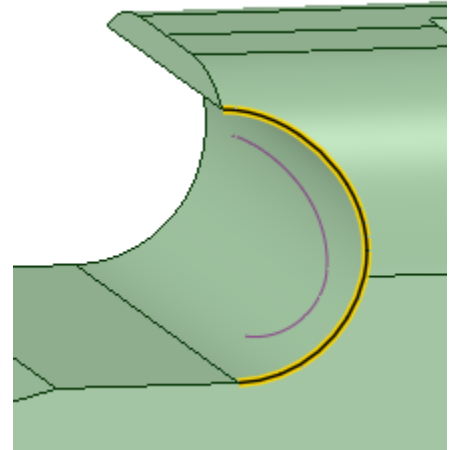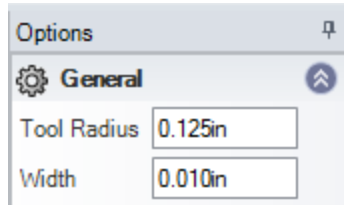
Wrap Tool:

There is a new **Wrap** tool in the **Manufacturing** group of the Insert tab.

- Allows wrapping of solids around other solids

- Preview is shown as curves

- Works with bodies and faces

  - With a face selected, the **Wrap Bodies** option will
    wrap the entire body that the face belongs to

  - With a face selected, the **Imprint on Target** option
    will imprint the face
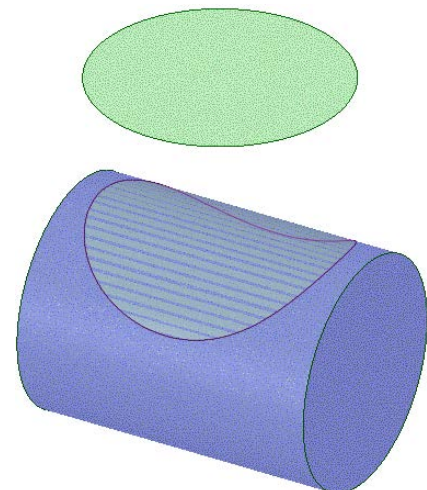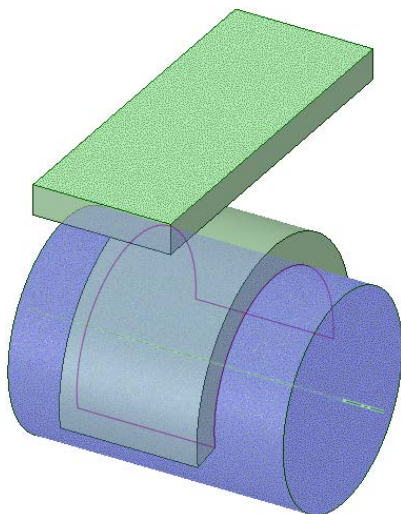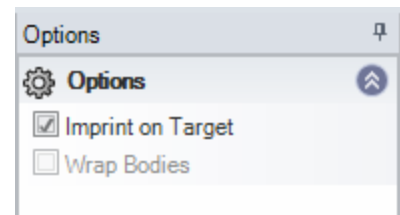
- Multiple target faces (or an entire other solid) can be selected to Wrap or Imprint
  - The image below shows an example of a solid wrapped around many faces
  - You can select multiple solids to wrap at once
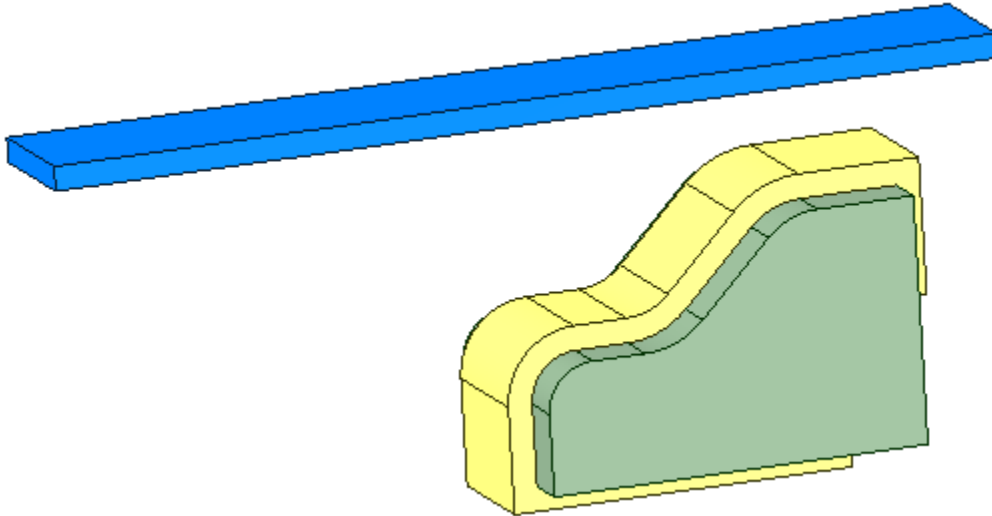
- Sketched Curves can be wrapped.
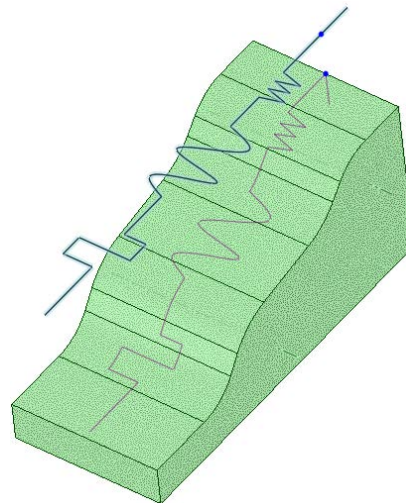
- Notes can also be wrapped.
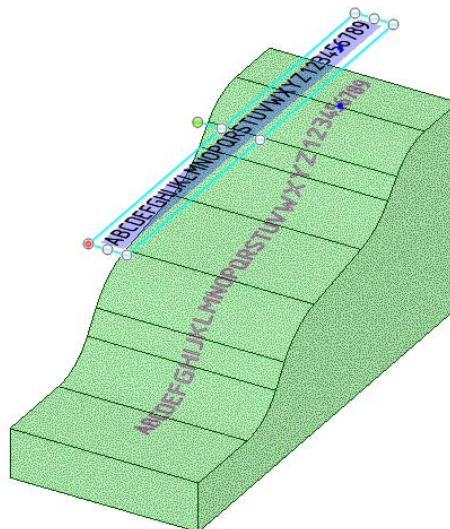
- Solids are created after wrapping even if the source faces are at different heights

    o The example below has three solids "sandwiched" together
    o All three solids are wrapped at the correct heights
    o Imagine a circuit board wrapped onto a solid

Three different heights
produce three different solids
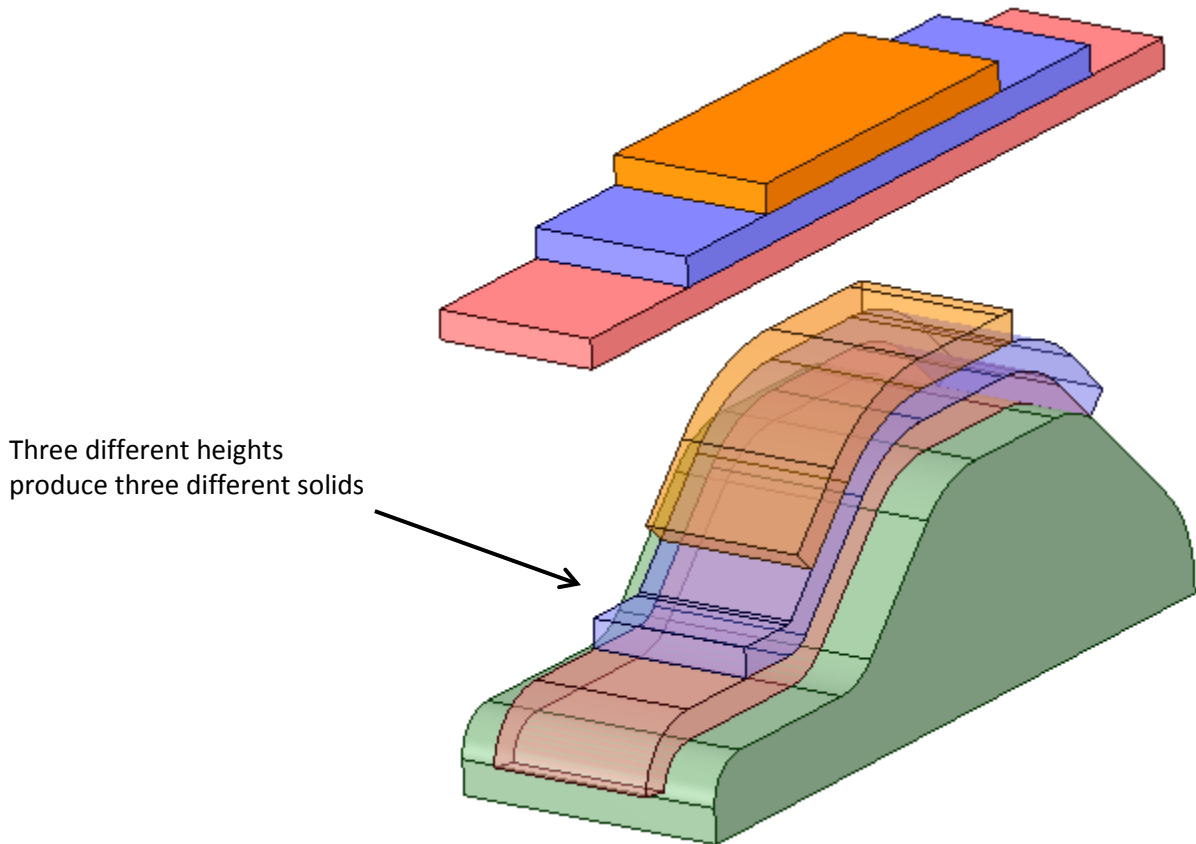
The **Wrap** tool allows manual alignment of Targets and Sources by dragging the Start Point on the Target. In the example below, the wrapped object needs to be positioned at the intersection of the two axes on the Target. So, the first step is to reposition the Start Point on the Source to a more convenient location, near the desired result. Drag the Start point and place it on the Source somewhere in the lower left corner.
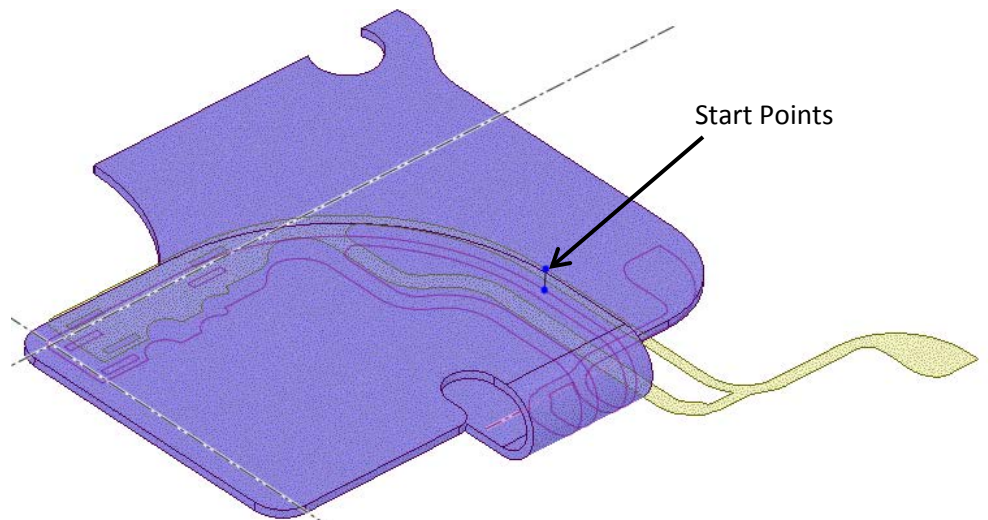
Start Points

The image below shows the repositioned Start Points.
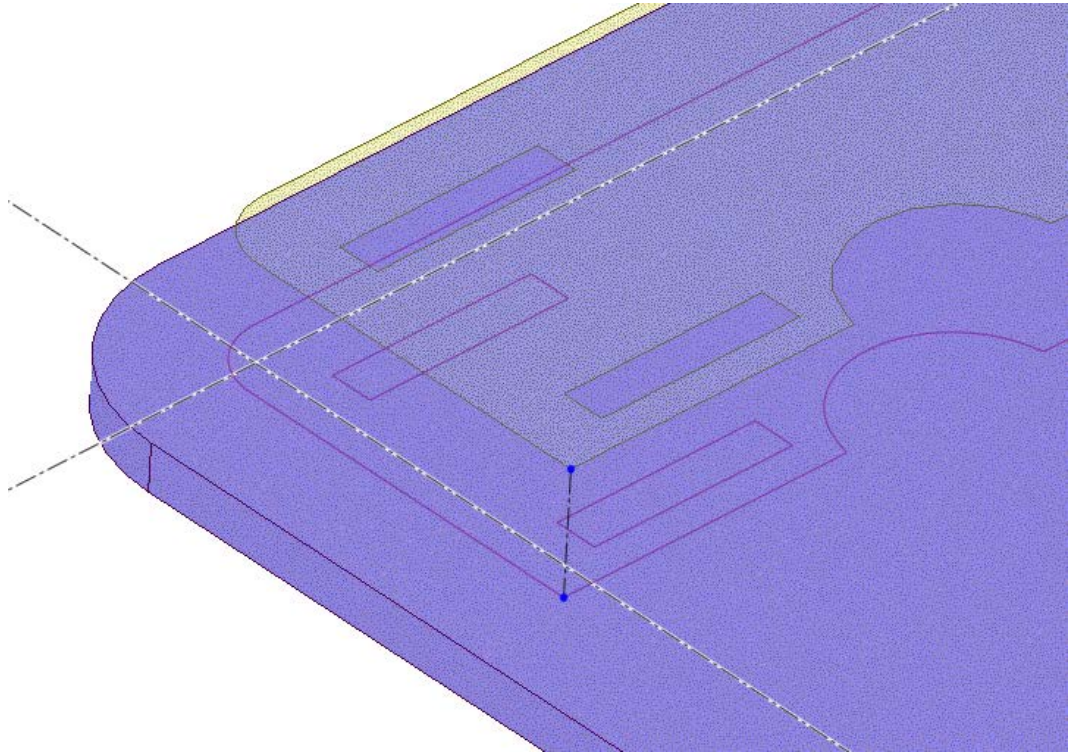
As you drag the Start Point on the Target, the Wrap preview dynamically updates to show where the Wrap will be created. In the image below, the Start Points have been dragged back and to the right to have the Wrap edges line up with the axes.
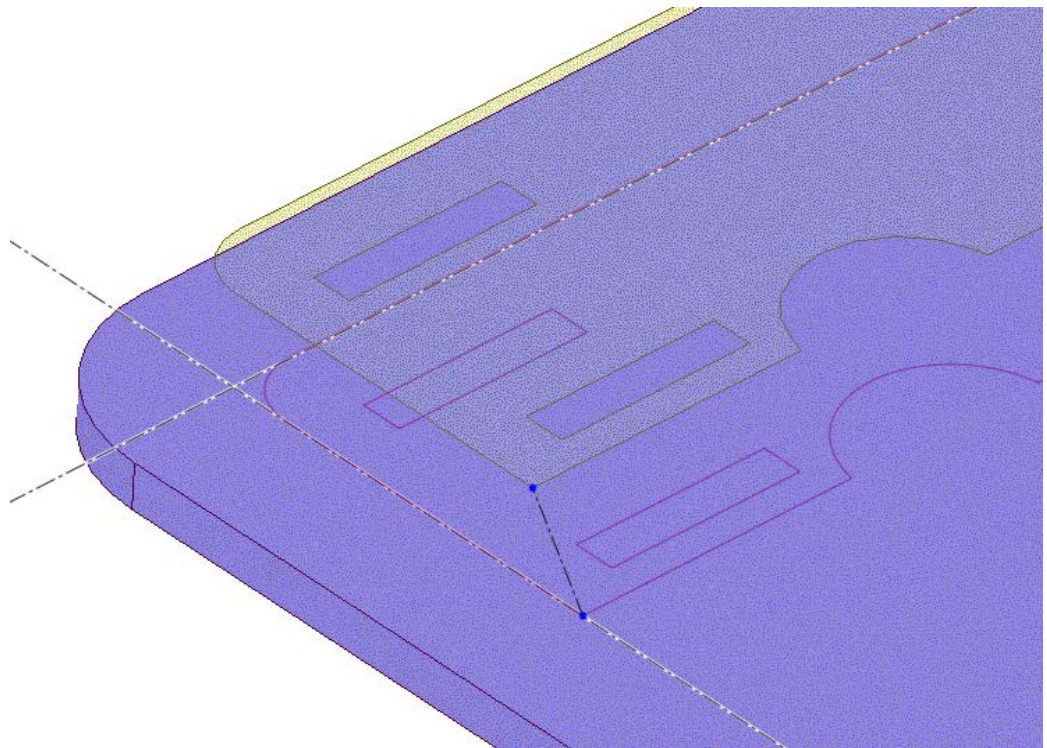
The image below shows the result, after clicking **Complete**.
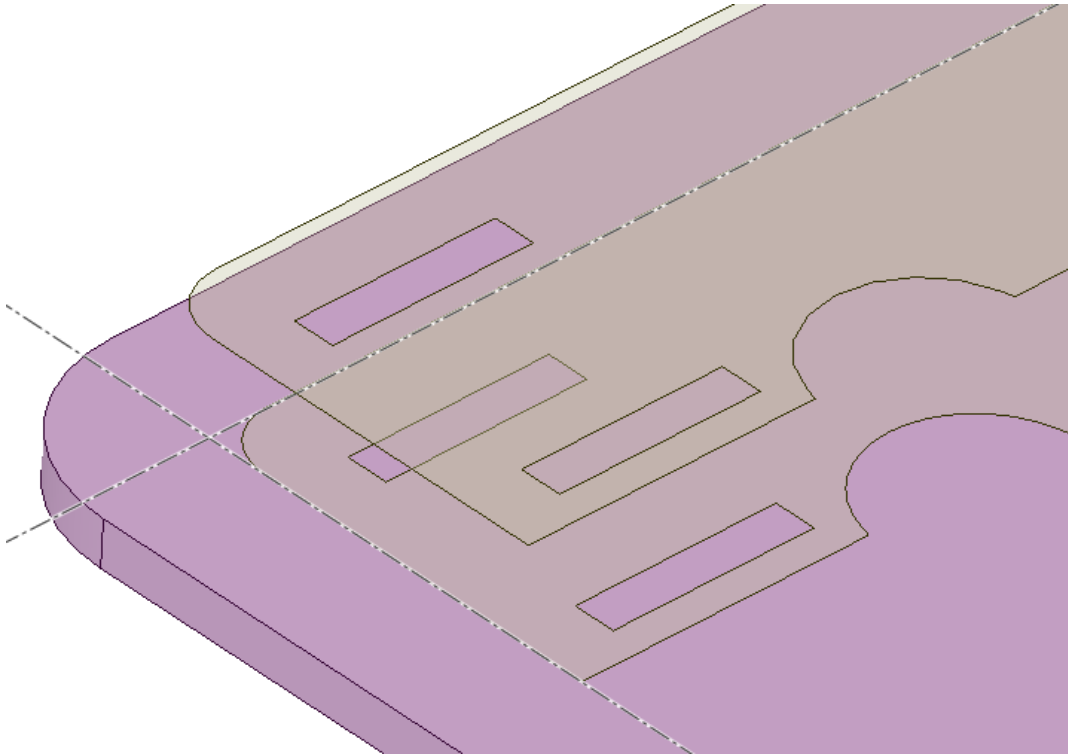
Turn Profile Tool:

There is a new **Turn Profile** tool in the Manufacturing group of the Insert tab.

- Works for cylindrical bodies

- **NOTE – You must align the cylindrical body along the Z-Axis**
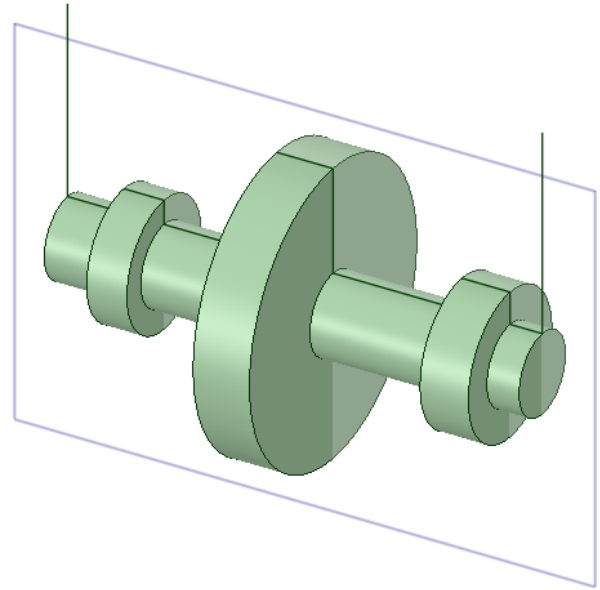
- Offset curves are supported
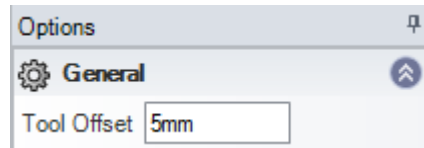
- After selecting a cylindrical face, the tool automatically previews the Profile
  - Click **Complete** to create the Profile curves

5mm offset
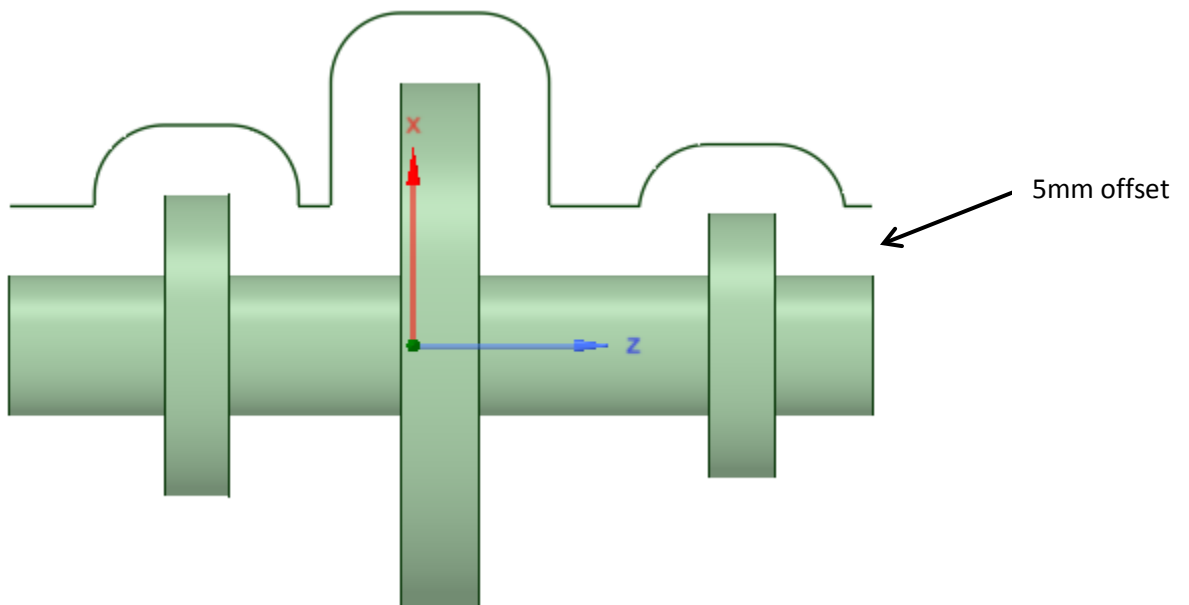
A **Curve Table** function has been added to the Illustrations group.

- Displays the data points for selected Manufacturing Curves

Select one or more sketch curves to be added to the Manufacturing curve profile table

Line	(0, -10, 33)mm
Arc	(0, -14.434, 26.04)mm
Line	(0, -14.434, 21.2)mm
Arc	(0, -10, 16.232)mm
Line	(0, -10, 9.76)mm
Line	(0, -18.868, 9.76)mm
Arc	(0, -23.868, 4.76)mm
Line	(0, -23.868, -0.93)mm
Arc	(0, -18.868, -5.93)mm
Line	(0, -10, -5.93)mm
Line	(0, -10, -8.25)mm
Line	(0, -10.817, -8.25)mm
Arc	(0, -15.817, -13.25)mm
Line	(0, -15.817, -17.92)mm
Arc	(0, -10.817, -22.92)mm
Line	(0, -10, -22.92)mm
Line	(0, -10, -29)mm
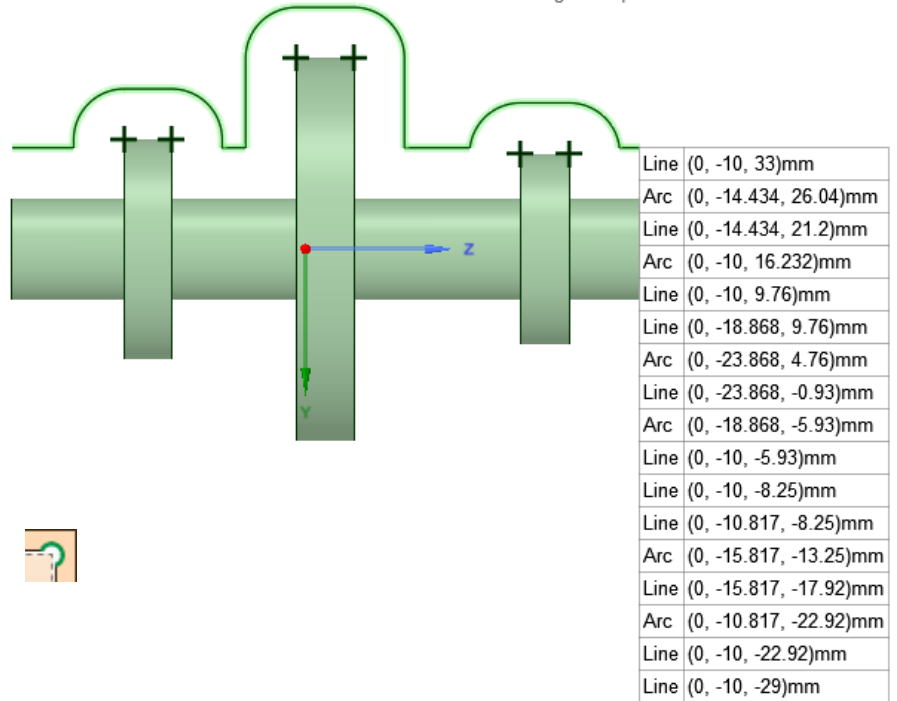
Relief Tool:

A new **Relief** tool has been added.

- Find and Fix tool
- Looks for thickness edges in concave situations
- Filters out edges that don not go through
- Finds edges on multiple pairs of thickness faces in the same body
- Edges found can have a cylindrical cutout made to provide relief
- The size and positioning of the relief can be changed in the Options

This example has two pairs of thickness faces.

- Notice that the pocket edges don't go through the part so they are filtered out
- The external corner is excluded unless you check **Include external corners** in the Options panel

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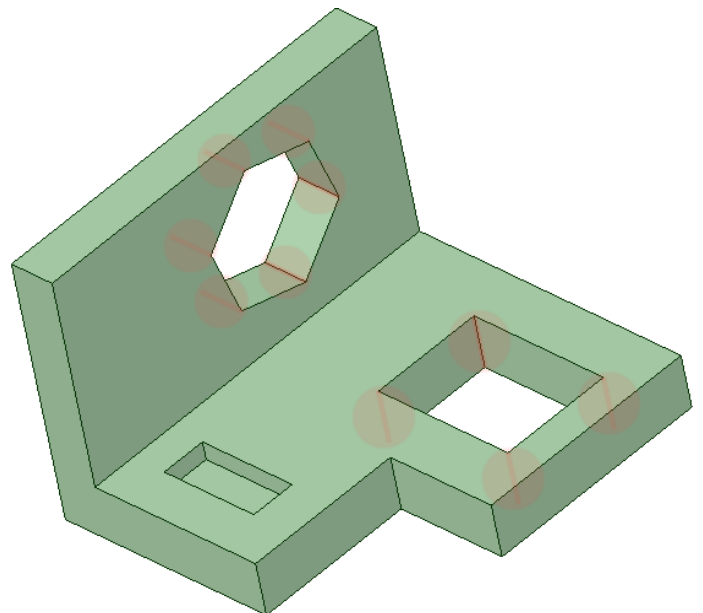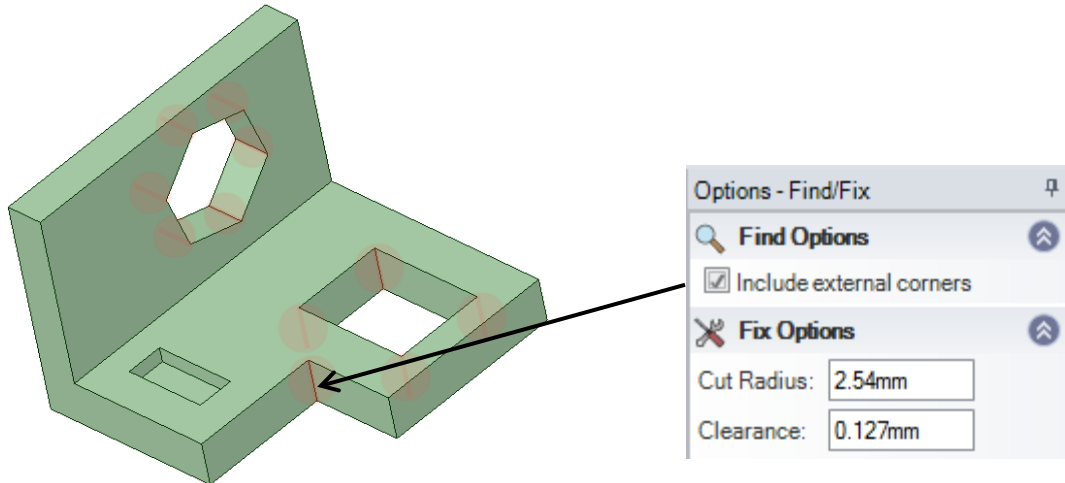
You can change the following options:

- **Cut Radius** – controls the size of the relief
- **Clearance** – controls the position of the Relief by specifying how far the edge of the Relief is from the original edge

1mm Radius
Default Clearance

1mm Radius
0mm Clearance

1mm Radius
1mm Clearance

1mm Radius
1.5mm Clearance
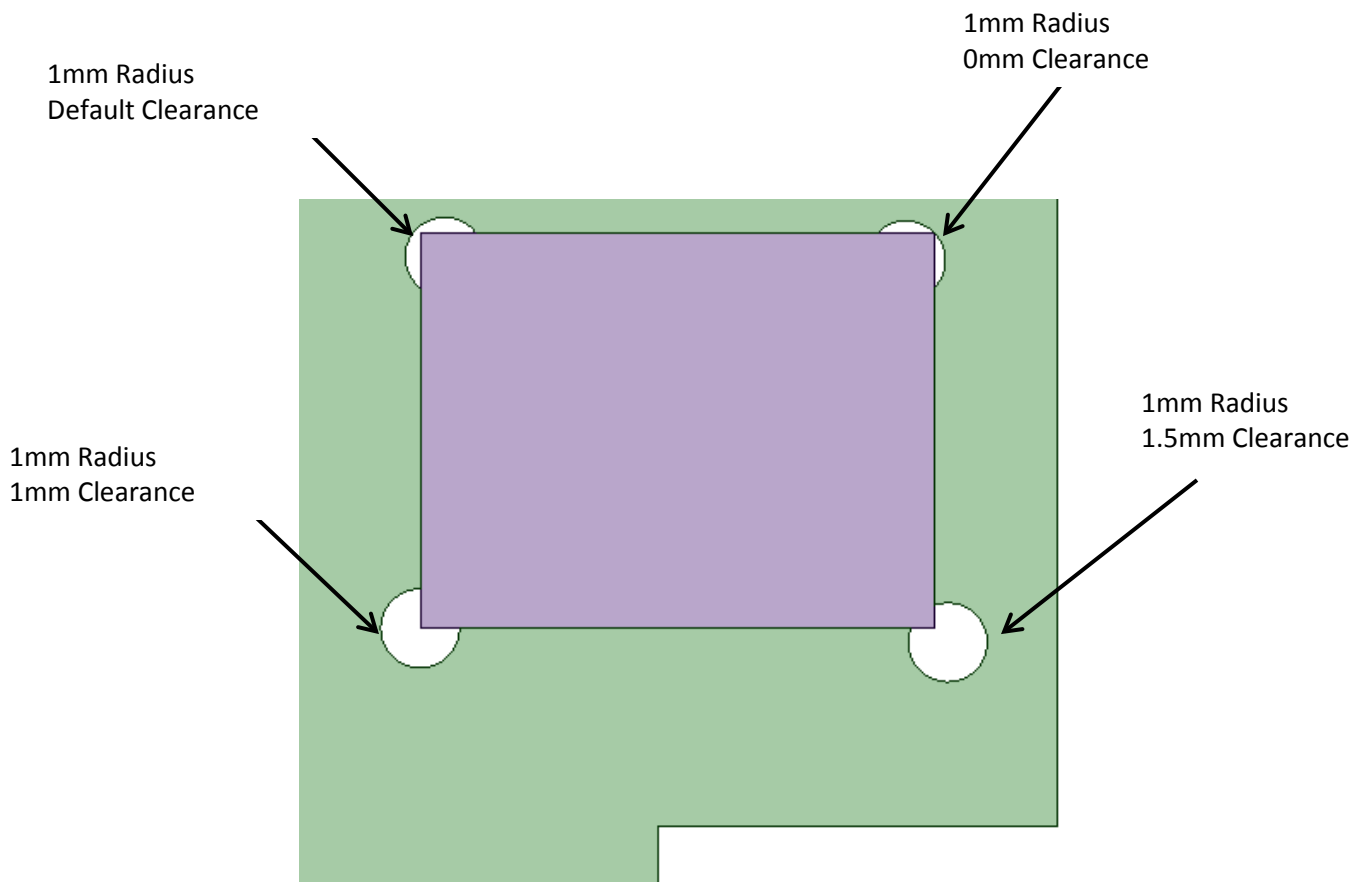
Graphics/UI/Selection

The **Dimension** tool now has status messages in the Status area of the SpaceClaim interface.

The messages are also shown in the Hint area of the Graphics window (upper left corner)

The Welcome Window has been redesigned with new icons.

- The What's New section now advertises major new functionality with specific links for:
    - LiveReview
    - Mesh
- All other links are the same as before except for the tutorial link in the Getting Started section
    - We now link to the Training page on our website
    - Previously, we linked directly to the Wrench tutorial
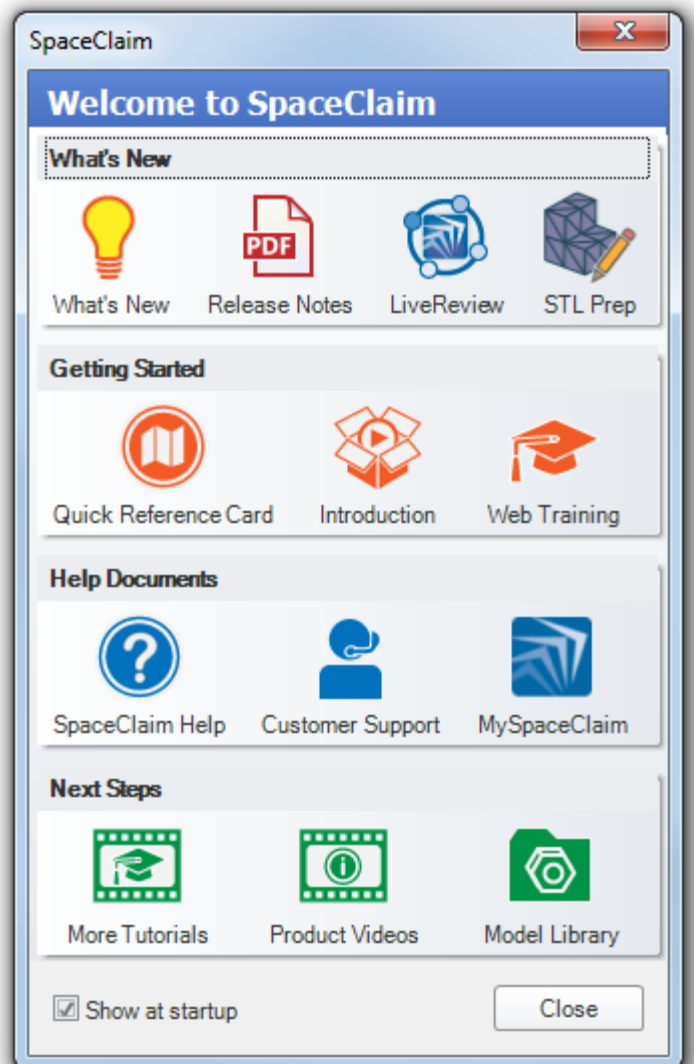
There have been changes to the correspondence between graphics card capability and the Anti-Aliasing settings in SpaceClaim Options > Popular Options.

- Essentially, the *old* High setting is the *new* **Low** setting
- SpaceClaim now estimates the video card memory available and decrements the Ant-Aliasing setting if the graphics card does not have enough capability
- Available graphics card memory is estimated based on the new screen size as it is reset by dragging
- For the High setting, if the memory limit is reached and exceeded (i.e. the device runs out of memory), SpaceClaim turns off Anti-Aliasing and does not crash
- The images below show the effect of Ant-Aliasing

Anti-Aliasing High

Anti-Aliasing Off
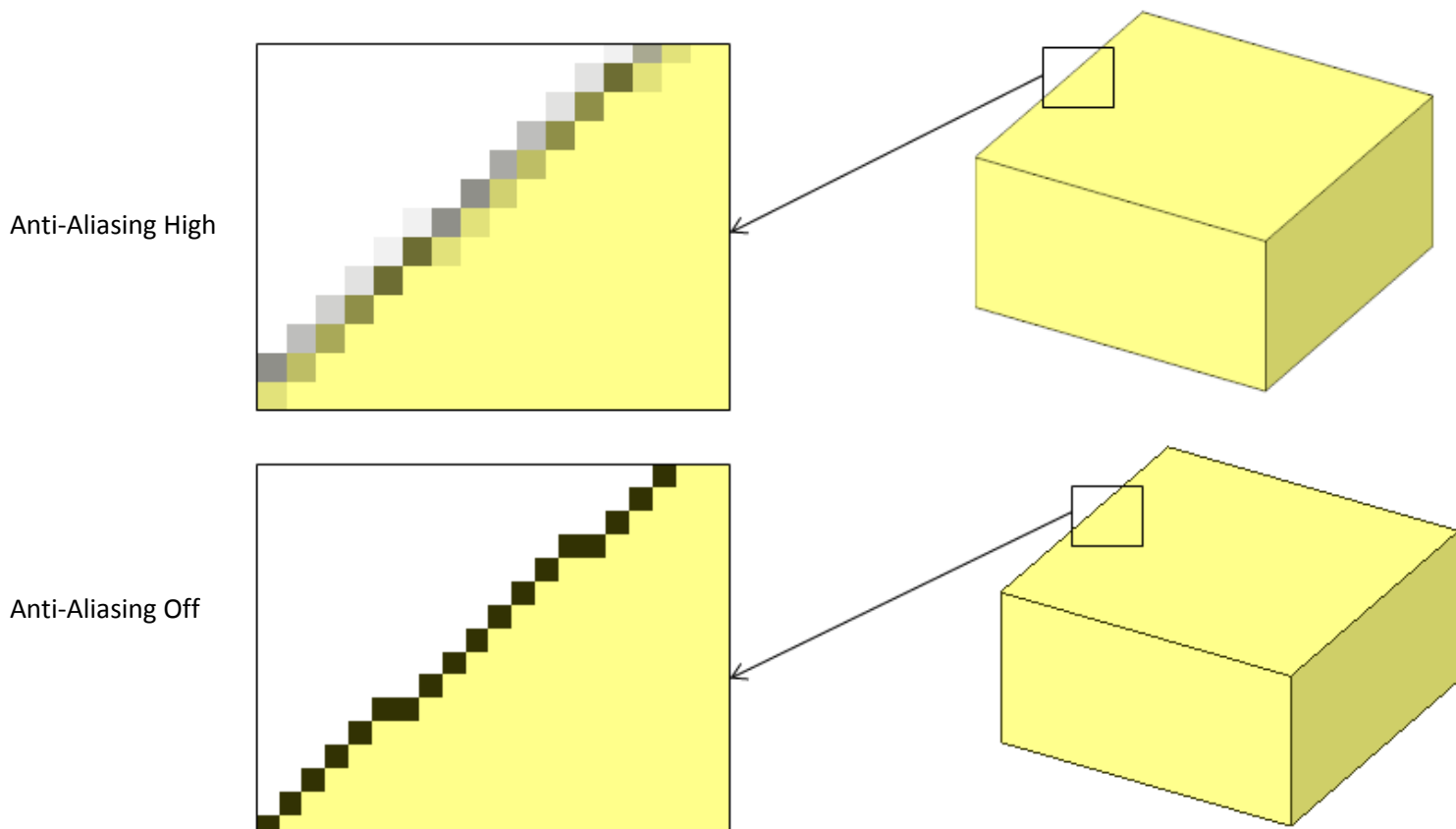
SpaceClaim Options > Popular now contains a checkbox for the Keyshot tab.

Adding "Unknown Material" to the material library no longer deletes library entries.

The "Unknown Material" name is no longer editable. Previously, you could type in a new name but it would revert back to "Unknown Material".

The cursor now changes when you are in **Turntable** spin mode.

- Picture the World Origin as a turntable
    - The model sits on the XY-plane
    - The Z-axis is the axis of rotation

- Spinning is based on cursor movement as follows:
    - Left-to-Right cursor movement spins about the Z-axis
    - Up-and-Down cursor movement spins the XY-plane

- You can see the difference by orienting the World Origin as follows:

    - Z-axis horizontal:
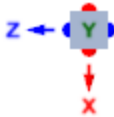
        Cursor movement Left-to-Right spins the model only about the Z-axis

        Cursor movement Up-and-Down *Also* spins the model only about the Z-axis

        A combination of both movements *Also* spins the model only about the Z-axis

- Turntable spin is enabled in the Navigation page of SpaceClaim Options

Pre-selection is hidden for any object under the cursor when a Box-select (rectangle, lasso, or polygon) is initiated. The previous behavior is shown in the image below left. The new behavior is shown below right. When the pre-highlighting was included, it could be unclear what the actual selection will be. Hiding the pre-highlighting as the selection box is being created makes it clearer what the selection will be.
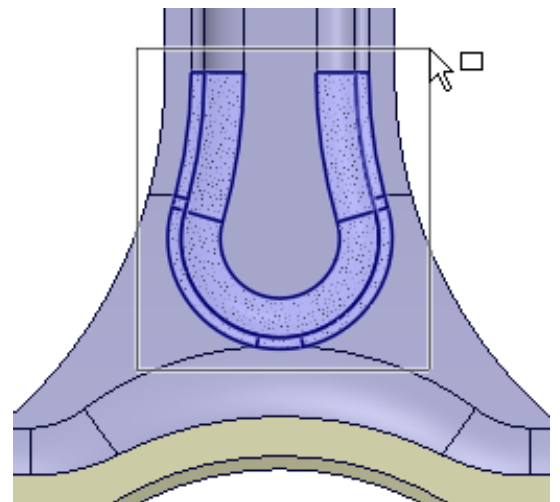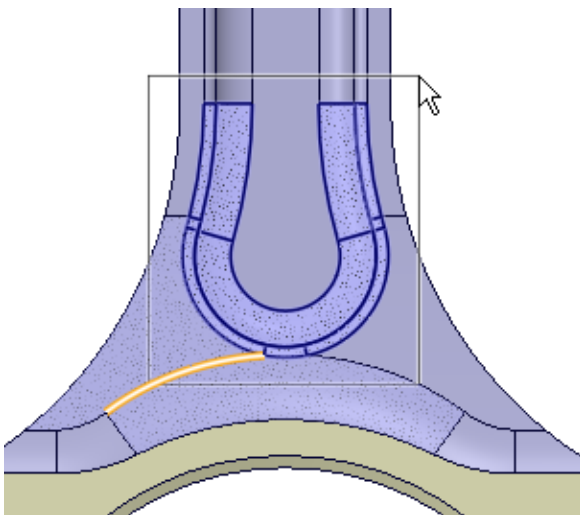
The default size for ALL undocked windows has been changed from 128 x 128 to 300 x 300.

Protrusions and **Depressions** can now be selected with the RMB in the **Mirror** tool for faces. This is a shortcut for selecting all the faces that make up a Protrusion/Depression.

To mirror a depression
1. Enter the Mirror tool
2. Select the Mirror Plane
3. Click the Mirror Face tool guide
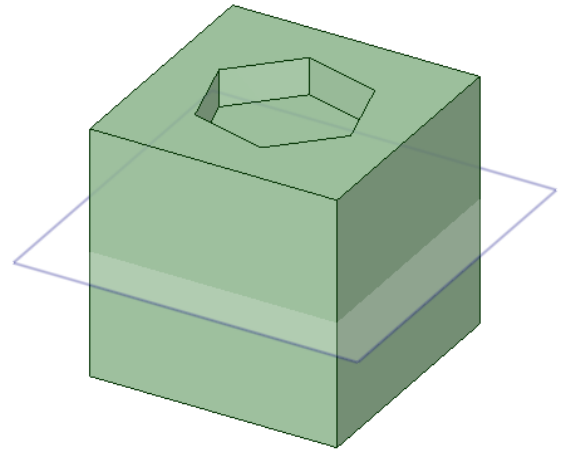
4. Select, or hover over, a face of a depression to mirror
5. RMB > Select > Depression
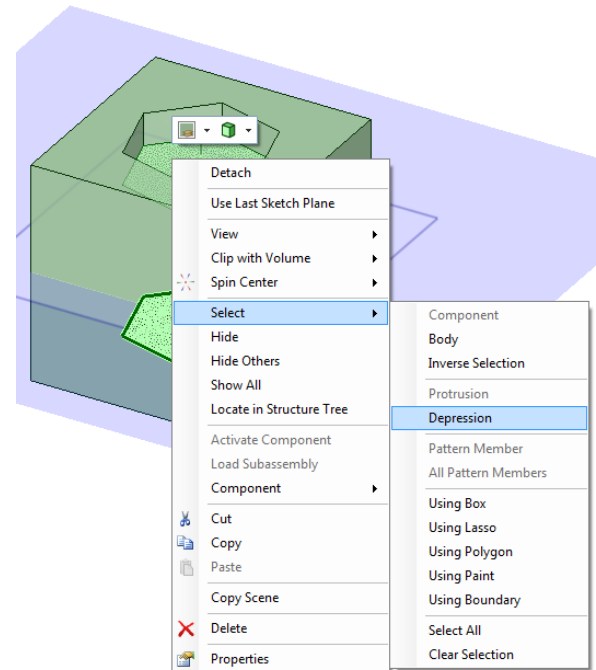
6. The Depression faces are selected and mirrored
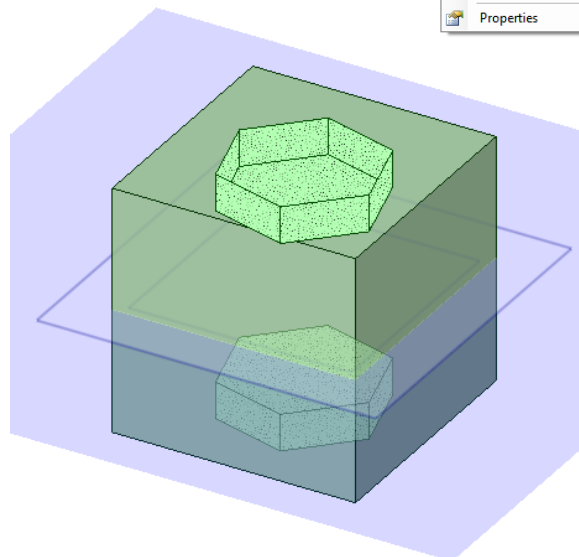
The process is the same for Protrusions.

Line Style, **Lineweight**, and **Line Color** are now maintained for copied Layout Curves. Previously, you could create Layout Curves in a drawing view, change their properties, and then Copy-Paste them but the new properties were not maintained.

Keyboard shortcuts are now processed even when the mouse is over the Structure Tree.

Opening files that are already open in a window tab now pops the opened document to the front. Previously, nothing happened and you had to look through the tabs to find the document.

When Pasting a copied component, there is now a second Paste option in the RMB.
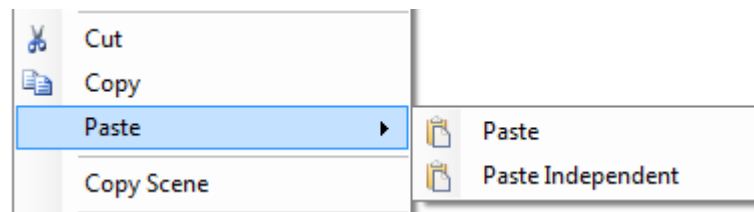
Paste Independent eliminates the need to use **RMB>Source > Make Independent** after Pasting, which could become tedious when repeated.
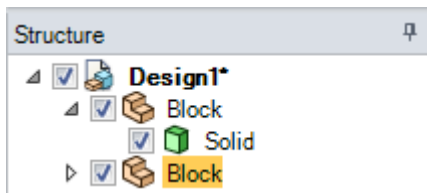
Paste
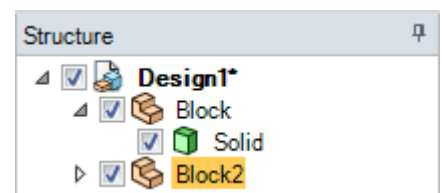
Paste > Independent

You can now store a selection (i.e. create a Group) containing *only* a Secondary selection. For example, if you enter the **Pull** tool and use **Alt** to select a face, you can create a group containing only the Alt-selected face. Previously, there had to be at least one Primary selected object, in order to store an Alternate object.

The graphics in the tool options panels has been cleaned up.

- Removed drop shadow around each group panel

- Group panels now fill the width of the container

- Tuned the color gradients

- Panel background is now white to be consistent with the structure and properties panels

- Added generic 16x16 "Gear" icon for tools' general options panel header - updated Pull, Move, Fill, Combine, Blend, Volume Extract

The new clean look is shown in the image below left and the previous look is shown in the image below right.
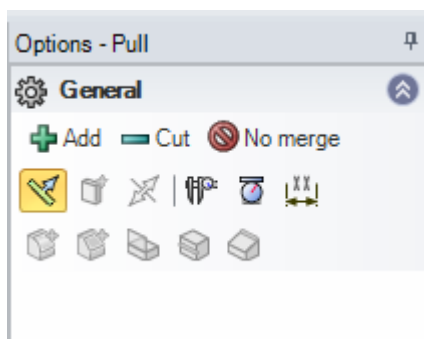
New
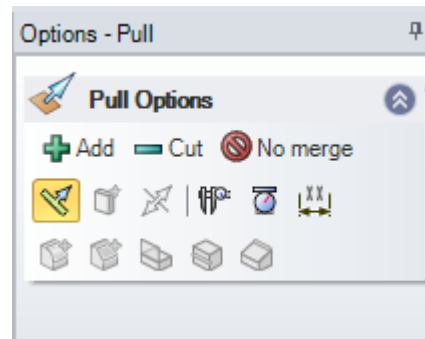
Old

In the Application menu, the SpaceClaim Options button now has a gear icon.

Insert Tab:

- The Manufacturing group has been moved from the Prepare tab to the Insert tab
- Standard Hole is now in the relocated Manufacturing group
- Orient Mesh has been moved to the Geometry group

Showing and Hiding large numbers of objects in the tree has been made dramatically faster. One test model went from 4 minutes and 32 seconds to just 0.745 seconds!

Power Selection has been added for Variable radius round *faces* and chains (of faces, even if they are composed of a mixture of constant and variable faces).
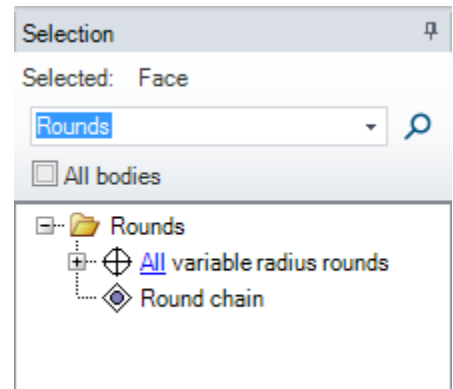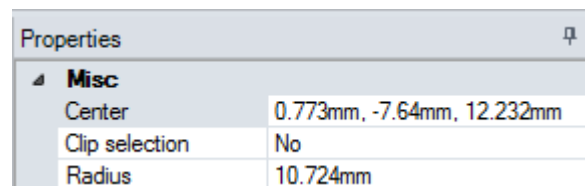
The **Table** and **Hole Table** tools have been moved from the Detail tab to the Illustrations group in the Insert tab.

The SpaceClaim Viewer can now use OpenGL 3.3.0.

- Can read SCDOCs
- Can set different edge thicknesses (by virtue of OpenGL) in a single pass

Clip with Volume (Sphere) has been enhanced.

- The Center coordinates (from World Origin) are now given in model units
- Coordinates are also truncated to the number of digits set in SpaceClaim Options
- You can now change the location and size of the sphere by entering new coordinate and Radius values

Source > Replace Component now works for multiple selected components. Previously, **Source** was not active when more than one component was selected.

Color Changed Faces now shows faces as changed if internal edge loops are changed. Previously, faces were colored when their *exterior* boundary edges changed. In the example below, the exterior boundary edges of the top and bottom faces remain the same while the edges of the interior holes have changed. Previously, only the holes' faces would have been colored. Now, the top and bottom faces are colored as well.
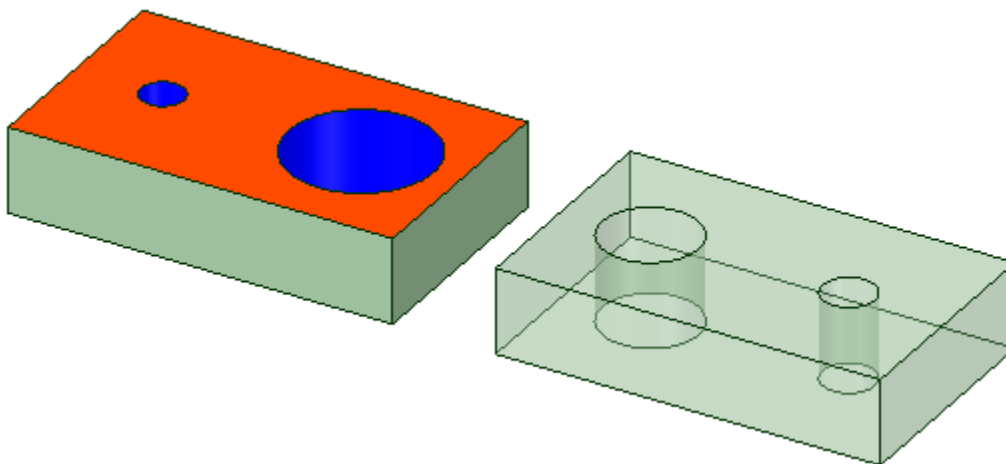
In the Find dialog, invoked with **Ctrl+F**, the order of Wildcards and Regular Expressions has been flipped. Now Wildcards is selected first.
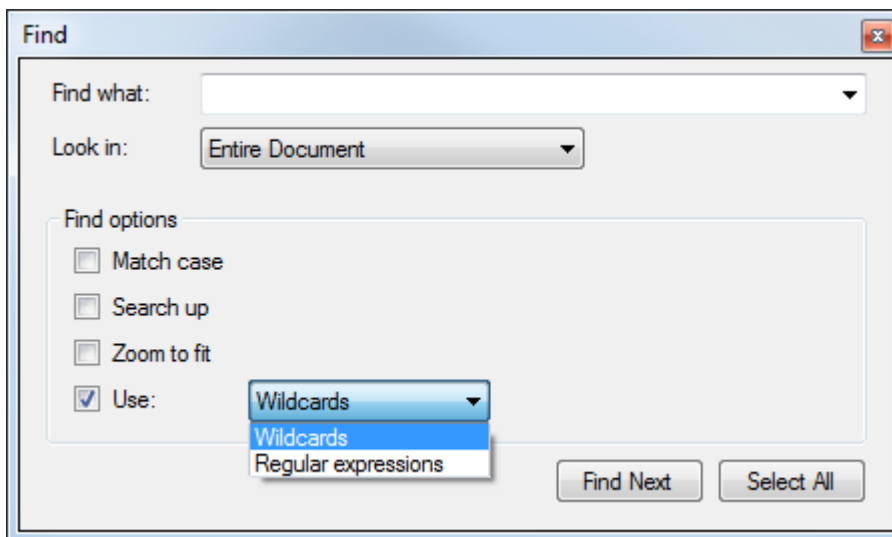
Import/Export

General Import:

We now import layer information for all supported file types.

ACIS:

We now import ACIS materials which are translated by Interop from another file format like CATIA V5.

ANSYS:
- The order of the Shared Topology settings has changed. **Share** is now before **Merge** in the dropdown list.
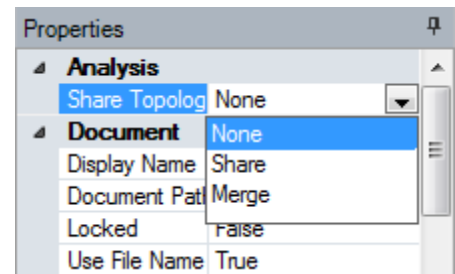
CATIA:

There is a new Import option for CATIA V5.

Map CATIA Geometric sets to SpaceClaim Groups will create SpaceClaim Groups that match Geometric sets in CATIA V5.

The images to the right show two Geometric Sets in CATIA V5

- Sketch.1 – contains the connected curves
- Extrude.1. – contains the surfaces extruded from the curves

When the CATIA V5 model is imported in SpaceClaim, the sets become Groups Sketch1 and Extrude1.
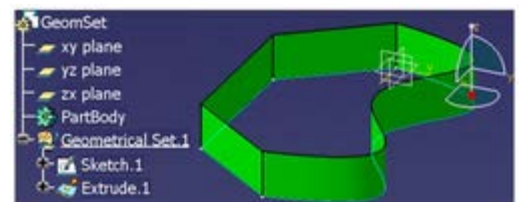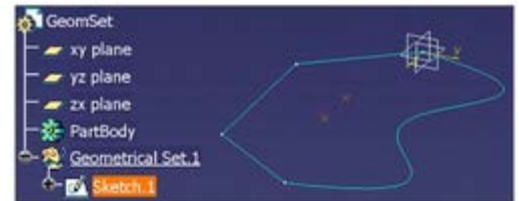
Multi-threaded Import is now enabled for CATIA CGR files. Without multi-threading, CGR components in CATIA .product files would not import.

IDF:

- IDF files can now be "synced" when certain operations occur
- Currently only Moved-around Components are supported for syncing
- Use **RMB > Update IDF** to sync the file after transforming components
    - If the current document was created by importing an IDF file
    - The source IDF file has to be present on disk at its original location

- A new IDF file is written that contains updated information
- The new file can be read back into the originating ECAD system and automatically update the components

Synced IDF files support these operations:

- Components moved
- Components deleted
- Drilled holes moved
- Board outlines modified
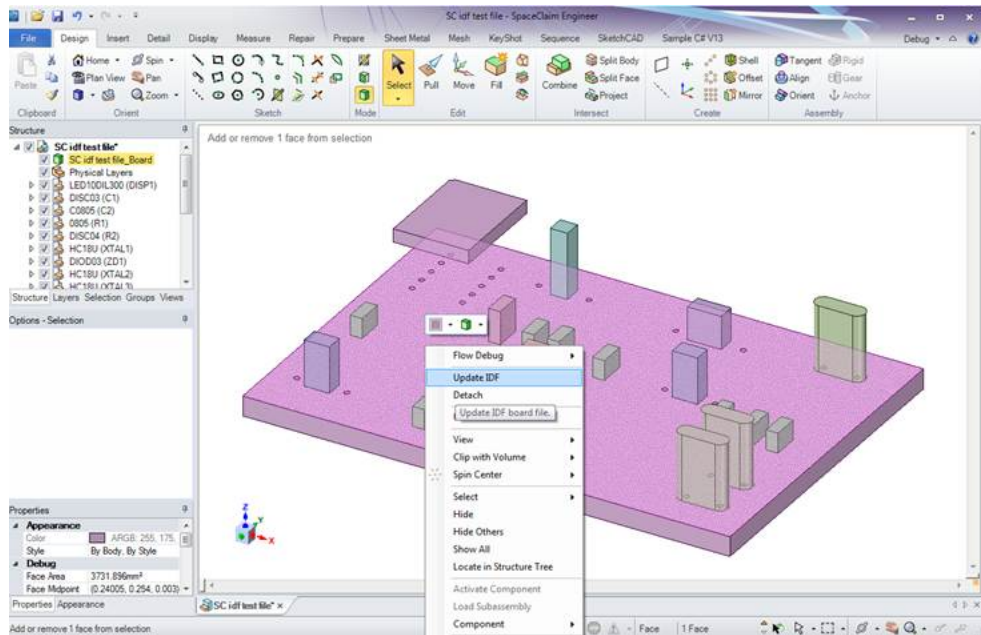
JT Import:

- JT Facets-Only files are now supported
- Multi-threading has been enabled in both stages of the import (thus showing the multi-threaded progress bar)
- Assembly information from a JT assembly file is uploaded even if the components referred to by the file are not present
- Assembly import now continues in the case of IOP Parasolid to ACIS translation failure

JT Export:

- Assemblies can now be exported to a JT faceted file

JT and PDF:

The JT and PDF pages of SpaceClaim Options now have the familiar import settings for faceted models.

The Lightweight option has been removed for CATIA, so all facet import options are as shown on the right.
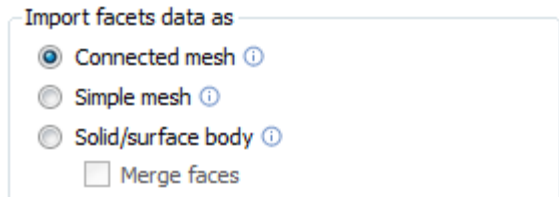
OBJ:

- UV's are now transformed according to a body or face, and texture information is exported with the file. An example is shown below.
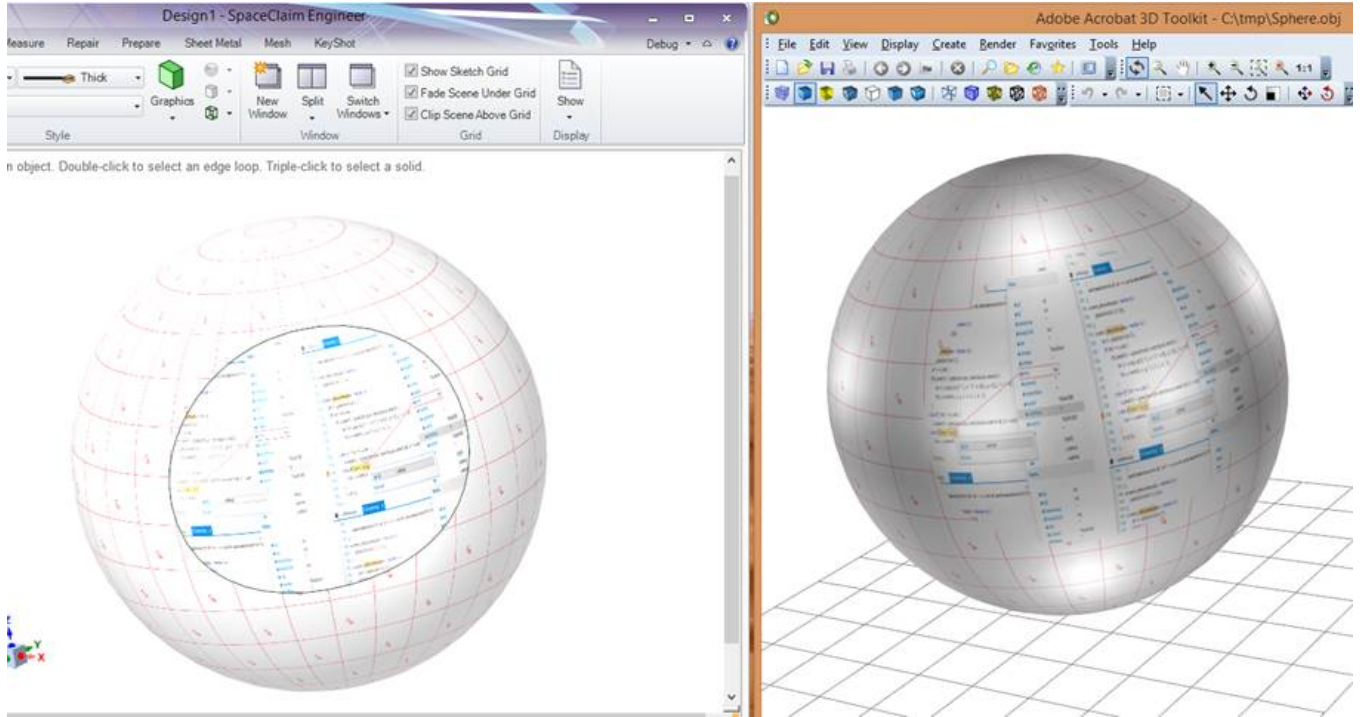
PDF:

3D PDF (U3D format) can now be *Imported* as facets in the same way as CATIA CGR.

A progress bar is shown during import.

Import of curves from faceted PDF file (U3D) has been implemented.

3D PDF, if geometry is not available (i.e. graphics only)

PMI:

SpaceClaim can now write (some) PMI to SAT/SAB files

Rhino:

We now automatically import Mesh from a Rhino file, when there are no B-rep contents present in the file.

Rhino import now automatically fixes tiny edges, when the **Improve imported data** option is checked.

SketchUp:

The SketchUp SDK has been upgraded to 2014, which includes 64-bit support. This will handle larger SketchUp files.

Outlook:

Share > As Outlook Attachment now starts a new Outlook email with the model attached and the Subject filled in. Previously, a bland, generic dialog opened.
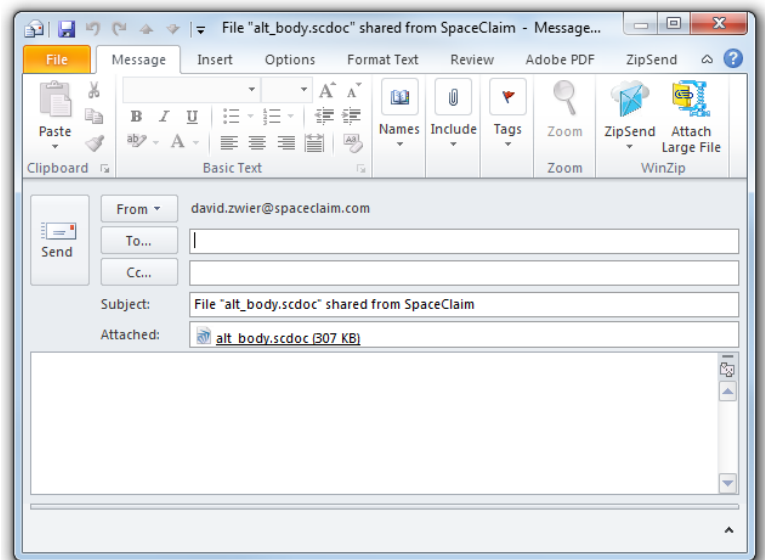
Converter

You can now specify a directory for converted files.

There is now a command line switch (for SpaceClaim) to check for add-ins

- **CommandLineAddInsOnly** will only load add-ins specified by the **AddInManifestFileName** switch
- For example: `/CommandLineAddInsOnly=true`
- The switch is primarily intended for use with the Converter to limit the add-ins loaded
- The switch can be used to start SpaceClaim without looking for add-ins in the usual locations

Options changes:
- You can now specify an Output directory
- Email notifications can now be sent for:
  - o   Errors
  - o   Convert completion

- You can now set a Failure Threshold to trigger an Abort

Process changes:
- **Pause** and **Skip** buttons have been added
  - o **Pause** temporarily stops the process and displays a message that the conversion is paused
    - Click the **Resume** button to continue the conversion
  - o **Skip** fails the current model and displays Status message naming the skipped model
- Cancel is renamed to **Abort**
- SpaceClaim status messages are shown at the bottom
- A running tally total successes and failures is displayed while the process is running
- The image below shows these changes
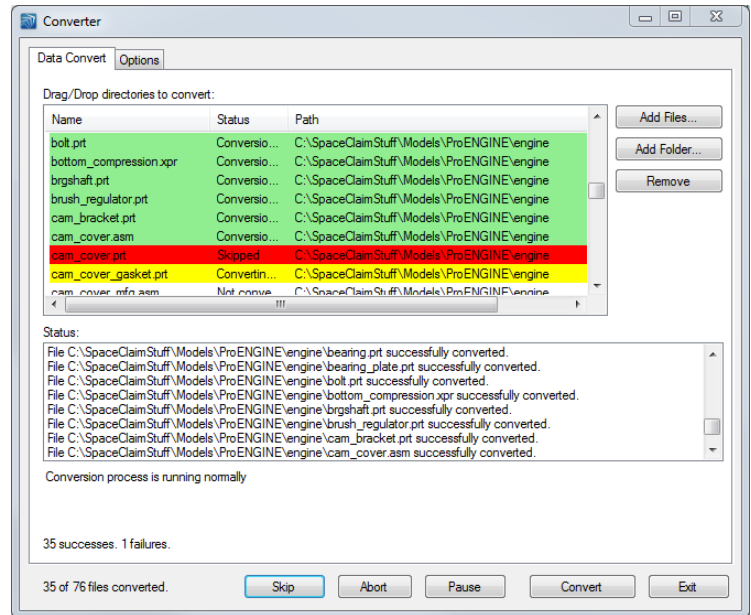
SC Options

UI for Customizing Command Shortcuts has been added.
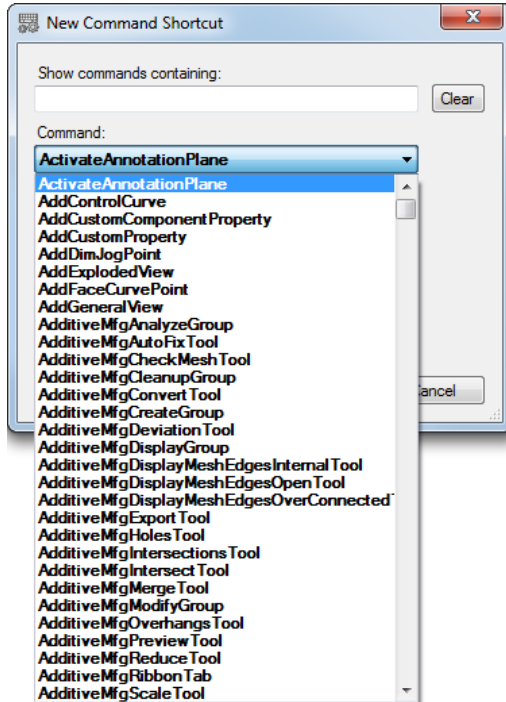
General Detail Options:

Hide annotations behind model has been added.
- Hides Annotations behind *solid* geometry
- Datum Reference Frames are not affected by the option as they need to be visible
- Default setting is Off
- Image below left is Off and right is On
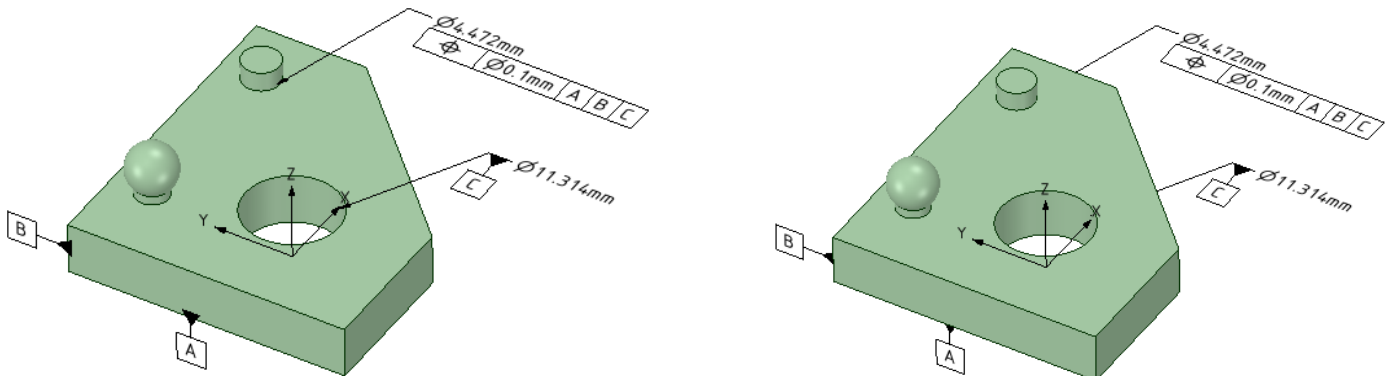
LiveReview

All participants now see markup Text as it is typed.

You can now work with Mesh models in LiveReview sessions.

Snapshots can now be deleted on the Client using the RMB.

On the Client and the Host, snapshots that are not related to the current model are grayed-out.
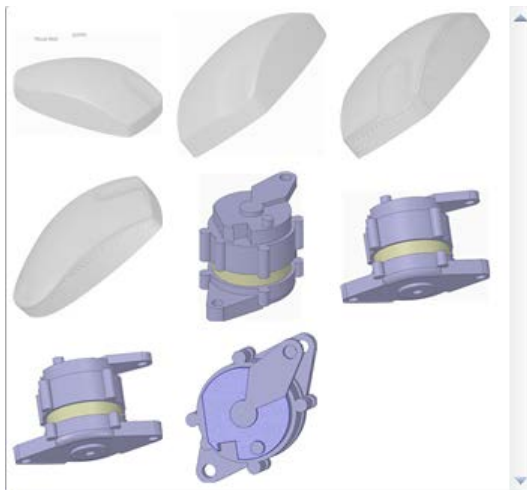
We now synchronize the view between Host and Clients when changing to Perspective mode on the Host.

We now use a better default size for the Chat window when it is undocked.

- o Changed from 128x128 to 300x300
- o If the you resize the undocked Chat window, the new size is respected in subsequent SpaceClaim sessions.

Share As Weblink

The website that **Weblink** connects to has changed. See the image below.

- Gradient background
- Added SpaceClaim logo
- Removed File Open button
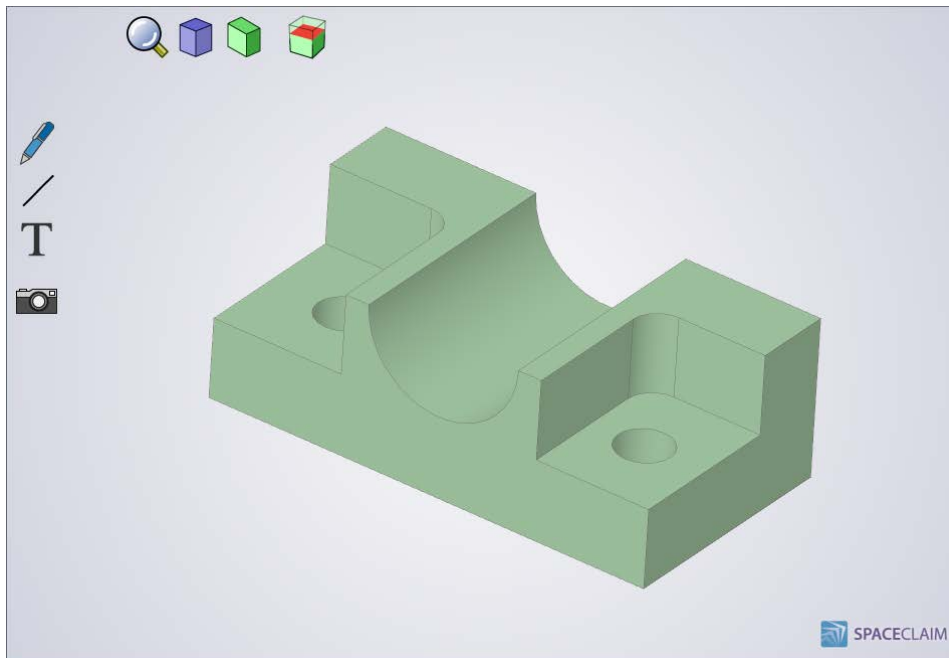
The **Snapshot** tool now displays the image in a popup frame for printing, copying, etc.

SC Viewer

Andrey.scfiles.com can now read analysis mesh files from Fluent (.msh) and any saved in GLTF format. For GLTF, all child files (.bin, .glsl, and .json) in the folder are retrieved.

ANSYS/SCDM

In the Repair tab, new buttons have been added for **Overlap Faces** and **Bad Faces**

- No functionality attached yet
- These will display overlapping and bad faces respectively

In SpaceClaim, to ensure that dimension updates from Workbench (WB) are robust, the Move handle needs to be anchored to s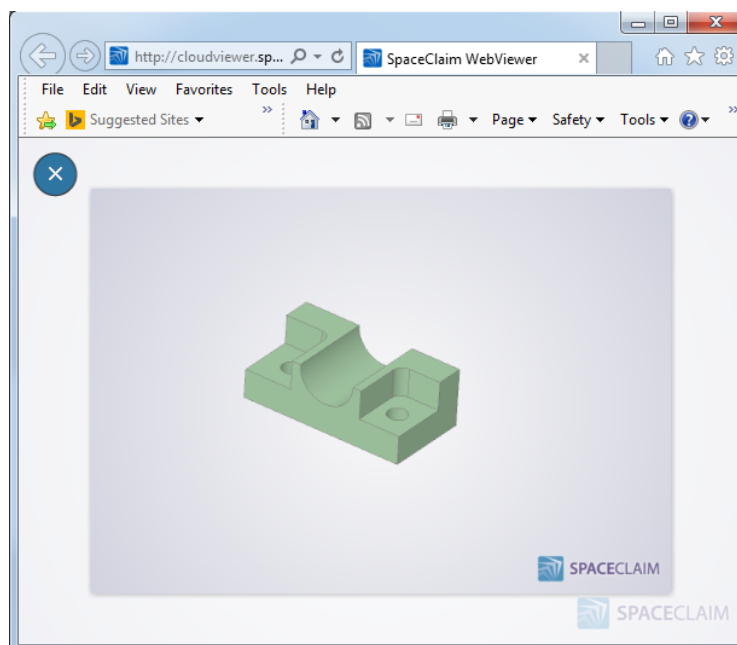omething meaningful (i.e. not just the default location). The Ruler Dimensions should also be attached to stable objects. If not, then you can get unexpected results or failures. Now, in SpaceClaim, a balloon warning/error message is displayed if the dimension update from WB is inaccurate or if it fails.

Models with many beams and shells now transfer dramatically faster to ANSYS 16. The improvement is on the order of 10X faster.

Converting surface bodies to mid-surfaces, by entering a thickness in the properties panel, is now up to 100X faster. A progress bar has also been added.

SpaceClaim can now open directly into the UI of AIM 160 in addition to ANSYS WB 160.

SpaceClaim–to—ANF is now available to WB Schematic.

When transferring models from SpaceClaim to Part Manager (PM), we protect access to PM. In other words, if PM is busy accepting a transfer no other models can attempt to transfer until PM is free.

Multi-threading is now used for SpaceClaim—to—PM transfers.

Actions such as tab switching, saving, and model manipulation are now dramatically faster for models with large numbers of beam bodies.

The SCDM splash screen is now suppressed when SpaceClaim is invoked directly from the WB schematic.

An option has been added to WB to flatten an assembly (i.e. create a single level of individual bodies) by exploding multi-body components and sub-assemblies. Below is a description of the situation:

- WB can only represent models by parts that contain one or more bodies

- To use multi-threaded part meshing, independent bodies (i.e. bodies that do not share topology with other bodies) need to be separated into their own parts upon transfer to WB

- Eventually, assembly handling and meshing will be enhanced to remove these limitations

Transfer speed to ANSYS has been improved by another 10X for a flat structure tree of only surfaces.

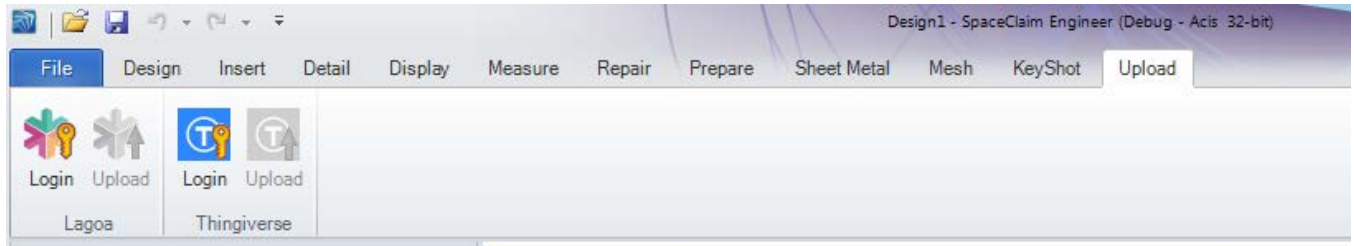The ANSYS 16.0 watermark is now displayed in the graphics window.
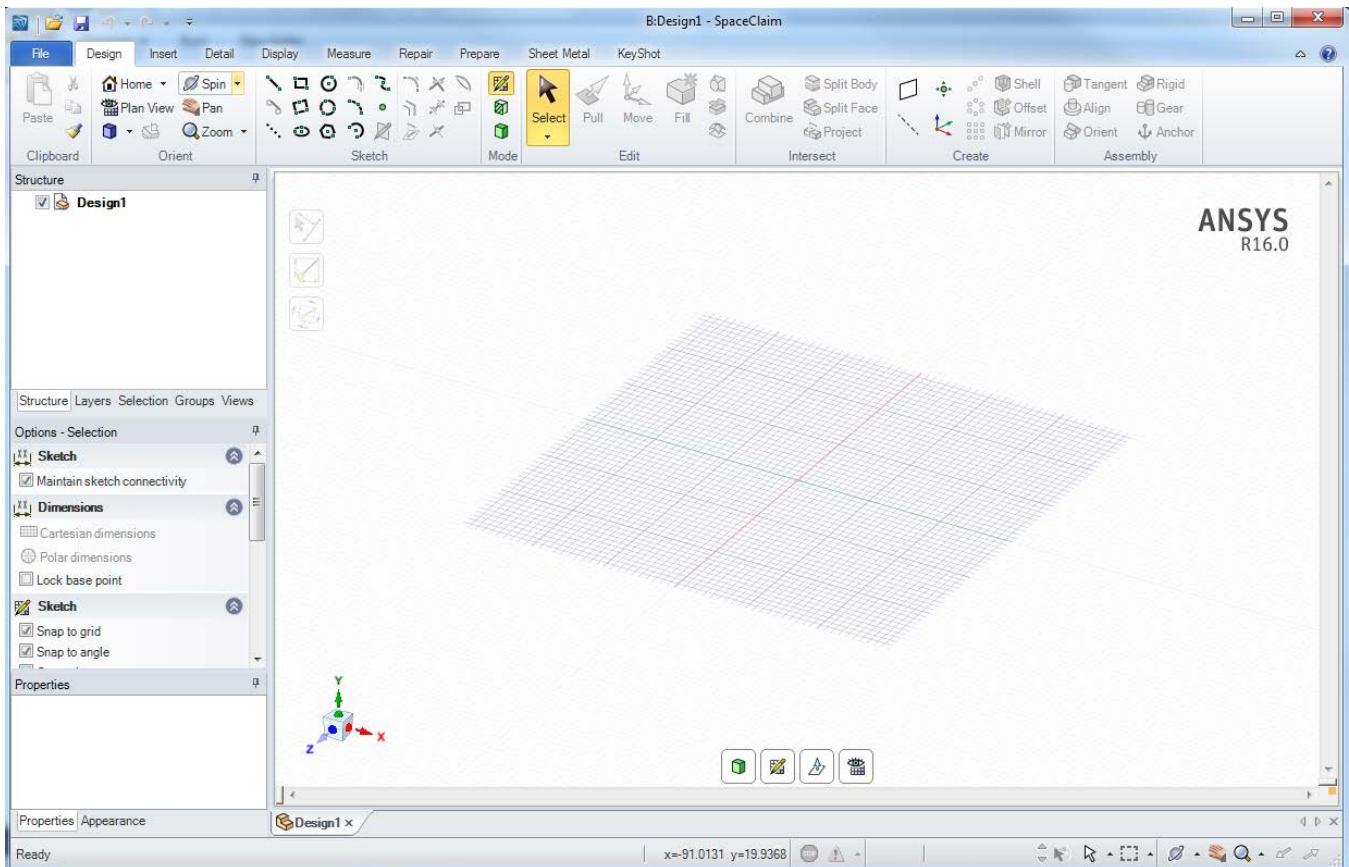
Designs with hundreds of named selections now transfer up to 3X faster to ANSYS.

API

There is a new add-in for uploading mesh models to **MakerBot's** design community called **Thingiverse**.

To upload a model to Thingiverse:
1. Click **Login** to login to Thingiverse
2. A web page opens for you to authorize SpaceClaim to access your Thingiverse account
    - If you do not have an account, you will be redirected to the web page for creating an account
    - Then you will be asked to authorize SpaceClaim
3. SpaceClaim displays a message in the Status Bar when you are logged in
4. The **Login** button grays out
5. The **Upload** button is enabled
6. Click **Upload** to upload the current mesh model
7. A dialog opens to enter information Thingiverse needs to process your model
8. When you submit the information, Thingiverse opens the web page containing your model
    - It may take a few minutes for your model to appear as Thingiverse processes the information and generates thumbnail images, etc.

Add-ins

MakerBot Thingiverse Uploader:

- Opening the browser is now optional
- The Thing's link can be copied to the clipboard

ACIS

ACIS R25 is now included.