SpaceClaim 2008

Release Notes

A **SPACE**CLAIM Document

SPACECLAIM
CORPORATION

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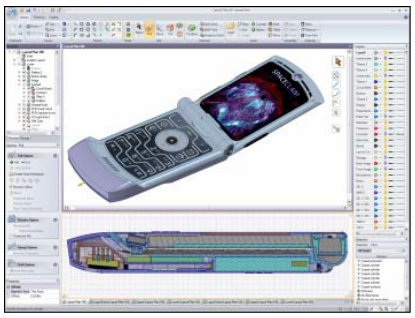
SpaceClaim 2008 enhancements overview

This section contains an overview of the enhancements made in the SpaceClaim 2008 release.

3D tool enhancements

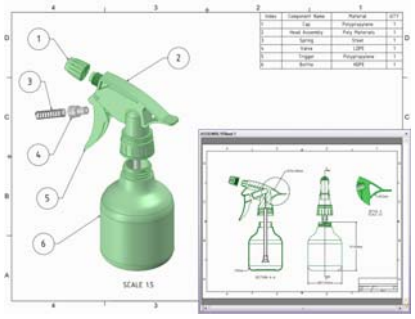
Enhancements to SpaceClaim's design toolset now allow you to:



Dynamically and precisely scale objects by pulling them ▪ Create blends between points, use guide curves, and manipulate each blend's take-off vectors ▪ Create patterns of points, faces, surfaces, and components ▪ Fill edges, surfaces, faces, and solids in both 2D and 3D ▪ Replace faces or quickly simplify them with a new Replace tool ▪ Combine objects using a simplified workflow ▪ Draft faces about spline surfaces ▪ Create and manipulate variable radius rounds by pulling on control handles ▪ Create cylinders and spheres with just a few clicks using the new Cylinder and Sphere tools ▪ Create and edit imported sheet metal designs for upstream and downstream applications

2D enhancements

Enhancements to SpaceClaim's sketch, cross-section, and drawing features now allow you to:



Create and edit 3D geometry from familiar, 2D drawings or cross-sections ▪ Edit sketches with on-the-fly, dimension-driven editing using polar, Cartesian, or relative coordinates ▪ Add and edit line styles by layer ▪ Copy and paste the contents of dimensions and notes ▪ Add table-driven and custom cosmetic threads to holes and cylinders with a new Thread tool ▪ Add symbols, intelligent centerlines, tables, and virtual sharps to create production-ready drawings

Streamlined workflows

Enhancements to SpaceClaim's user interface, data exchange, and toolset increase the speed with which you can:

Create designs using mouse gestures, heads-up interaction, and advanced 3D snapping ▪ Create temporary axes and planes for dimensioning or snapping ▪ Select objects using new selection filters and a same-length edge option ▪ Identify problem areas with new edge and volume interference tools ▪ Import assemblies using a Structure-tree only and lightweight options ▪ Collaborate on designs by saving them as 3D PDF files ▪ Restructure parts and assemblies by dragging and dropping them in the flexible Structure tree

Sheet metal enhancements

SpaceClaim's sheet metal feature provides intuitive 3D design and optimization for manufacturing of sheet metal parts and assemblies. You can:

Create sheet metal designs from scratch or import and edit designs using all of the familiar SpaceClaim tools, such as Pull, Move, Fill, and Combine ▪ Convert non-sheet metal designs to sheet metal so they can be unfolded and manufactured ▪ Change corner junctions, reliefs, bends, and bend allowances with one click ▪ Automatically create corner reliefs or add a rip between two points on a face ▪ Work on a folded or unfolded part and see your changes in both versions simultaneously ▪ Split a single part into multiple parts at critical junctions to save time and money during the manufacturing process

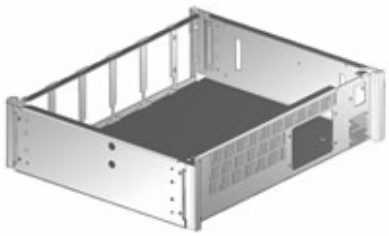


Image courtesy of catalCAD

Detailed enhancements list

This section contains a detailed list of individual enhancements organized by application area.

General

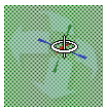
The following general enhancements were made to the SpaceClaim application:

- You can set the background color of the Design window.
- A message is displayed if an undo action will open or close a document or switch to a new Design window.
- SpaceClaim now fully supports the SpaceBall and SpaceNavigator with all license configurations.
- You can press Esc to cancel all operations after 2 seconds. This function is useful when SpaceClaim appears to "hang" or be "thinking too hard."
- You can now quickly create a new design and drawing sheet using a new application menu option.

SpaceBall

SpaceNavigator

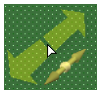
- The Show cursor-based interaction hints option was added to the Popular SpaceClaim options to display the following cursor icons:



Spin

Spin, when you are spinning around a highlighted object and the Rotate about preselected object in Spin advanced SpaceClaim option is selected

Pan

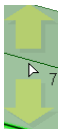


Pull (offset, scale, extend, pivot, extrude)

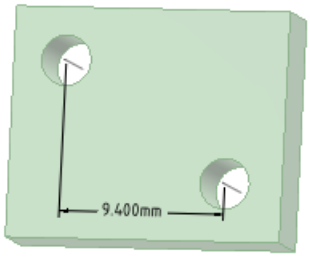
Pull arrows convey the object's change in size, such as when pulling a round, as shown in the upper figure. If the pulled object remains the same size, the arrows are the same size, as shown in the lower figure.

Zoom

Pull arrows convey change in size

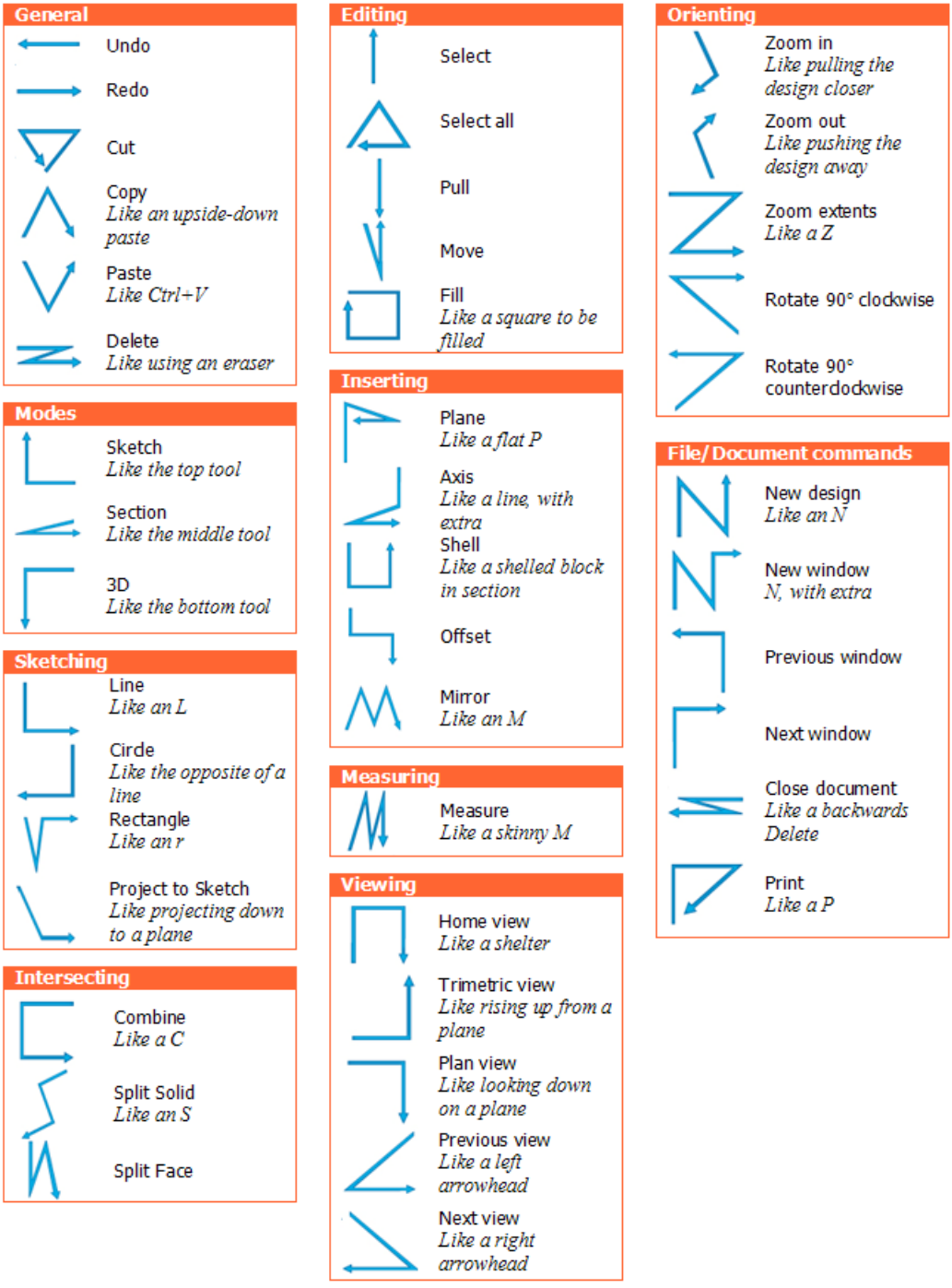


- You can click the middle mouse button on the tab of any Design, Drawing Sheet, or 3D Markup window to close that window.
- You can now right-click in the Design window and select Copy Scene to copy the contents of the Design window to a PNG file.
- The mini-toolbar is now displayed further away from the cursor. This prevents the mini-toolbar from obscuring a dimension or other visual information.
- A machine that is running SpaceClaim can now sleep, hibernate, or be undocked without adversely affecting the SpaceClaim application.
- Tooltip titles were added to all the icons in the status bar.
- Axes continue to be displayed, even when one of them is selected. This makes it much easier to dimension between two axes, as shown in the figure on the right.



- The Student and Home versions of SpaceClaim now allow add-ins.
- The default number of undo steps was changed to 50.

- SpaceClaim now has mouse gestures which can be used to control SpaceClaim. Move the mouse in the directions shown in the following table while pressing the right mouse (or touchpad) button:



Structure tree

The following enhancements were made to the Structure tree panel:

- You can right-click the top-level component in the Structure tree and click Expand All to expand all the components in the tree.
- You can now Shift+click multiple objects in the Structure tree to change their visibility with one click.
- Surfaces selected in the Design window are highlighted in the Structure tree.
- You can now Alt+click an object in the Structure tree to use it as a secondary selection. For example, if you are moving an object, you can Alt+click an axis in the Structure tree to set the direction of the move.
- You can now select an object in the Structure tree and press F2 to rename it.
- You can now hide origins in the Structure tree and drag them from one component to another.
- You can now name a 3D curve created by Ctrl+clicking multiple points with the Pull tool and blending. Right-click the curve and select Name Sketch Curve to display the curve in the Structure tree. Select the curve in the tree and press F2 to rename it.
- You can move a blended 3D curve from one component to another by dragging it in the Structure tree.
- You can now select multiple objects in the Structure tree, then drag the objects into another component.
- You can now drag and drop components in the Structure tree to move all of their subcomponents and contained objects.

Groups

The following enhancements were made to the Groups panel:

- Groups now contain ruler dimensions for cylinder faces.
- If you create an angular ruler dimension on an edge in Section mode, then create a group with that edge or its neighboring faces, the angular ruler dimension and the section plane are stored with the group and are displayed when you click the group on the Groups panel.
- The Delete Group button on the Groups panel now has a label and a tooltip.

Design tools

Enhancements were made to most SpaceClaim design tools.

Select

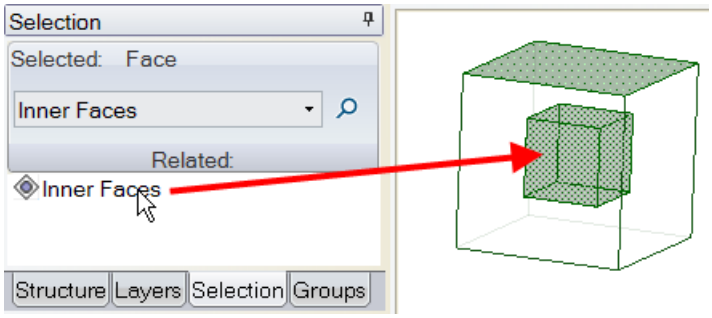
The following enhancements were made to the Select tool:

- When selecting by clicking, all checked objects will be selected. When using box-select, only the top-most checked object is selected.
- You can now Alt+double-click an edge to select an edge loop as the secondary selection. Alt+double-click again to select a different edge loop that contains that edge. You can Alt+double-click repeatedly to keep selecting edge loops.
- Box-select now works for very large solids.
- You can now box-select sketch lines drawing on a layout plane in 3D mode.
- You can again select sketch lines in Sketch mode.
- Selection highlighting now works with any linestyle.
- You can select the face of a solid when only the edge is displayed (such as in a drawing sheet view) using the scroll wheel. The edge becomes a slightly thicker line when the face is highlighted.

- You can now scroll to select references located at the same point in the Design window when setting ruler dimensions.
- You can now select Clear Selection from the Select tool drop-down. This function was added so you can clear a selection when the Design window has no empty space, such as when you are zoomed very far into your design.
- When you select a face to pull or move when only the edge is displayed (such as in a drawing sheet view), the Pull and Move handles now appear.
- You can now box-select planes, notes, and annotations.
- When box-selecting in Sketch mode, only sketch lines are selected, and when box-selecting in Section mode, only section lines are selected.
- You can now box-select 2D and 3D points, such as sketch points, sketch vertices, edge vertices, origins, and points that have been associated to edges with the Move tool.
- You can now drag to box-select open sketch lines that appear in a sub-component.
- It is now easier to select an axis using the scroll wheel to scroll past the face of a solid cylinder.

Selection panel

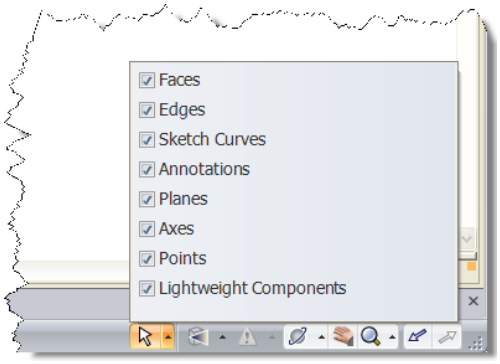
- You can now select Inner Faces from the Selection panel drop-down to find all the inner faces of a solid. Inner faces are defined as those that surround an internal void, as shown in the figure on the right.



- You can now select a solid, surface, face, or edge when using the Selection panel to find similar or embedded objects. You can select these objects from the Structure tree.
- You can now select the following new options on the Selection panel:
  - Surface edge loop finds all the edge loops in a component. This function is helpful when edge loops appear on top of each other in the Design window.
  - Edges with same length finds all the edges with the same length as the selected edge.

Selection filter

- You can now filter selections using the drop-down control in the status bar, as shown in the figure on the right.

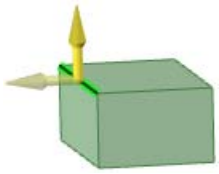


- The Selection filter is cleared when you switch from one tool to another.
- Filter options apply only to the Select tool, since all other tools and tool guides apply their own filtering based on the workflow and references needed when the tool or tool guide is active.
- Selection filters now apply to each Design window. (They used to apply to all Design windows in a session.)
- Filter options are now available in all tools and tool guides, and the filter options no longer appear in the Options panel when the Select tool is active.
- When you select, only the objects selected as Filter options are selected. You can now limit selection (on click and with box-select) to a number of different objects.

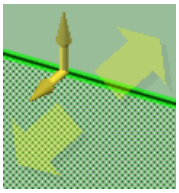
Pull

The following enhancements were made to the Pull tool:

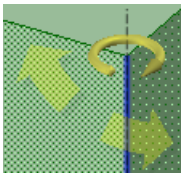
- You can now pull objects in multiple components.
- The Pull arrow was modified by removing the center ball and changing the coloring of the active Pull arrow, as shown in the figure on the right.



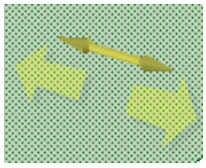
- Right-click in the Design window and select Anchor Pull handle to re-anchor the Pull handle.
- You can now press Shift while pulling to snap to edges and faces.
- You can now Alt+double-click an edge in the Pull tool to select an edge loop as the secondary selection. Alt+double-click again to cycle through alternate edge loops.
- The following cursor-based interaction hints were added to the Pull tool to help users understand the pull direction:



Pivot or extrude edge



Revolve



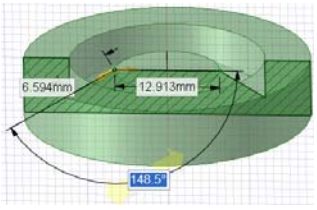
Pull Both Sides option selected

- Pull arrows appear differently depending on whether you are pivoting or extruding an edge, as shown in the figures on the right.

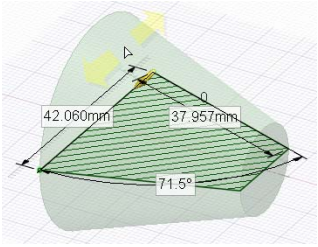
Extrude edge

Pivot edge

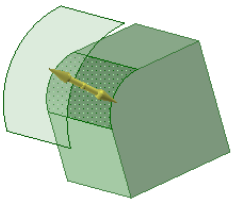
- Pulling a cylinder in Section increases or decreases the diameter based on the direction of the pull.
- You can now dimension a pivot in Section mode by setting the angle, as shown in the figure on the right.



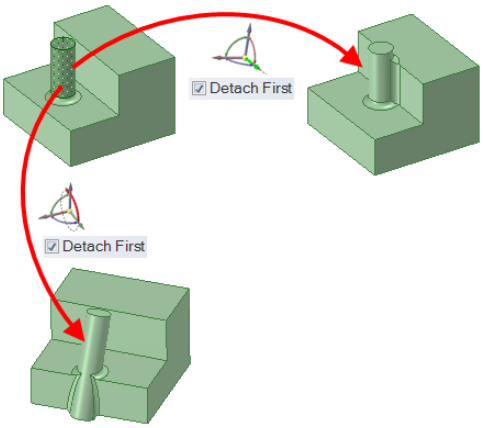
- You can create a ruler dimension when the Pull handle is moved from its default position.
- You can press Esc after pulling to hide ruler dimensions.
- You can now pull a rounded hexagonal solid.
- You can now edit a cone in Section mode by pulling the vertex where the edge of the cone intersects the Section plane, as shown in the figure on the right.



- Spheres now exhibit the same merge behavior as any other objects when intersecting other objects during a pull.
- You can now press the spacebar to enter offset dimensions for a conical face.
- Pulling the faces of a subcomponent no longer cuts a hidden component.
- If you pull a surface that intersects with a solid whose visibility is off, the solid is no longer deleted.
- The Pull arrow now appears perpendicular to a cone's surface instead of perpendicular to the cone's axis.
- You can now copy and offset a round's face with the Pull tool by pressing Ctrl while dragging the face. An example is shown in the figure on the right.

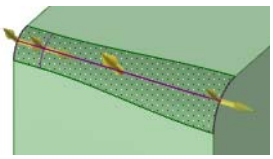


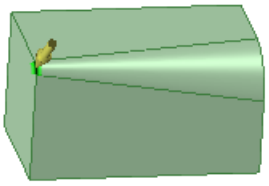
Rounds and chamfers



- You can now create a full round on a sub-component.
- SpaceClaim places a round wherever possible at the intersection between the selected protrusion and the solid when the protrusion is moved, as shown in the figure on the right.
- When you make a hole in a chamfered face, the face is no longer a chamfer. You can still pull the face, or the hole, but you cannot change the chamfer to a round or dimension the chamfer.
- The Round Edge and Chamfer Edge options are enabled only when an edge, rounded face, or chamfered face is selected.
- You can now switch a chamfer to a round when the object is within a component.

Variable radius rounds



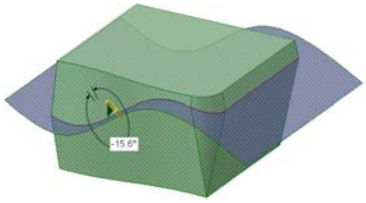


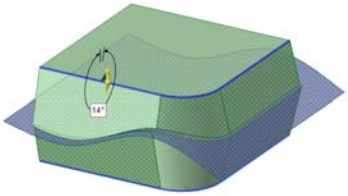
- You can right-click a round face and select Edit as Variable Radius Round to create a variable radius round.
- You can click a variable radius round face to display the Pull arrows, as shown in the figure on the right. Click a Pull arrow to set its location by entering a length or a percent.
- If you create a variable radius round with only two radii, a conical face is formed instead of a spline face, as shown in the figure on the right.
- When you pull a neighboring face of a variable radius round, the round follows the face as it is pulled.

Holes and slots

You can move a hole radially instead of turning it into a slot by pressing Shift while dragging with the Pull tool.

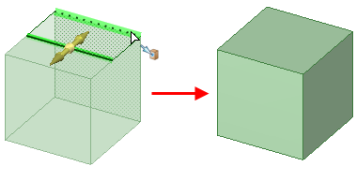
Draft tool guide





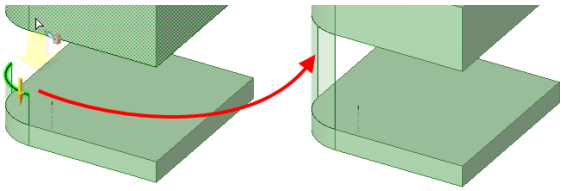
- You can draft when a plane or surface passes through a solid.
- You can now perform an edge-driven draft on an object within a subcomponent.
- You can now create a draft that is split at a spline surface with the Draft tool guide, as shown in the top figure on the right.
- You can also create a split draft split at a spline surface when you select two edge loops and pull with the Draft tool guide, as shown in the bottom figure on the right.

Up To tool guide



- You can now pull edges up to other edges and faces using the Up To tool guide.
- You can now extend or extrude an edge up to another edge. If this creates a closed volume, the result is automatically solidified, as shown in the figure on the right.
- You can now revolve the edges of a surface up to another object.

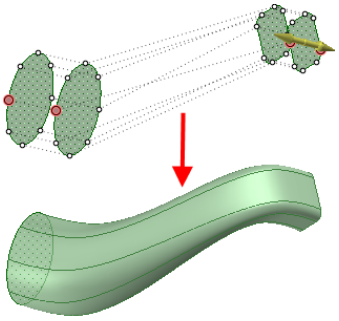
- You can now pull a cylinder up to another cylinder in a sub-component or across components.
- You can now pull circular planar edges up to a planar face, as shown in the figure on the right.



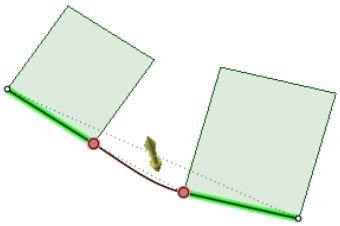
Scale tool guide

- To scale objects in Spaceclaim, you can now click the Scale Body tool guide.
- You can now scale a surface.
- You can now Alt+click an origin with the Pull tool to use it as the reference point for scaling.
- You can now use regular expressions to calculate a scale factor.
- You can now scale multiple solids or surfaces in different components simultaneously.

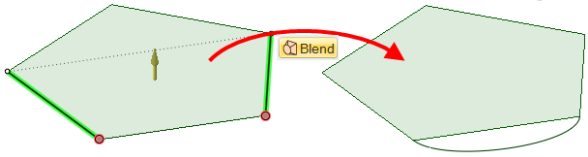
Blend option



- When you select two or more points with the Pull tool, the Blend option is selected automatically.
- When you pull multiple surfaces to form a blend, as shown in the figure on the right, the surfaces are arranged so that the blend does not intersect itself. As a result, faces can be selected (in this case) from right to left, or from left to right.
- You can now blend a line between more than three points.
- When blending between multiple points, you can Ctrl+click neighboring edges to control whether the blend should be kept tangent to those edges.
- In 3D mode, you can now blend between multiple 3D curves (that is, lines or surface edges not in the same plane) and lines or surface edges comprised of multiple line or edge segments.
- You can now blend through two or more points to create a line or spline.
- You can now pull a blended edge between two selected vertices with the Blend option, as shown in the figure on the right.

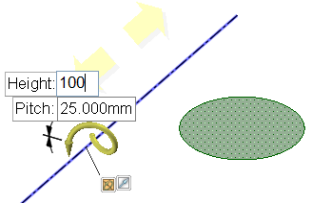


- You can now create a blended line between two points that has the tangency of neighboring edges, as shown in the figure on the right. The Blend option is selected automatically when you select the two vertices.



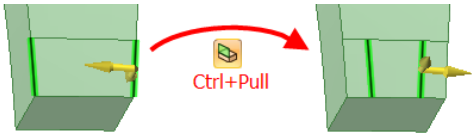
Other options

- A new Copy Edge option was added. Select this option to copy the selected edge when you pull it. If you select an edge, select the Pivot Edge or Extrude Edge option, then Ctrl+drag the edge, the new Copy Edge option is automatically selected and the edge is copied.
- There is a new Show take-off vectors Pull option.
- When creating a helix, cursor arrows now appear, as shown in the figure on the right. You can pull in either axial direction to create a helix dynamically.



- If the Pull both sides option is selected, the total distance from the far face to the near face is dimensioned, instead of the distance from the original surface to one face.
- You can now create a revolved cut with the Pull Both Sides option when the revolved face lies inside the solid to be cut.

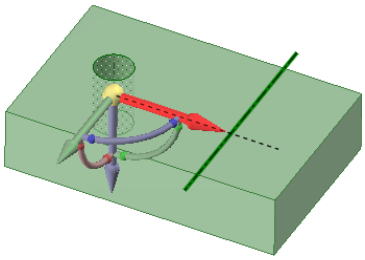
- If you Ctrl+pull edges with the Extrude option selected, as shown in the figure on the right, the edges are copied even if they are not part of an edge loop.
- You can continue a sweep using the Full Pull option if the face has not reached the end of the trajectory, or if the face is placed partway into the trajectory.



Move

The following enhancements were made to the Move tool:

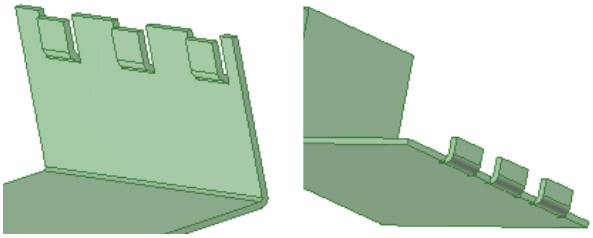
- You can now move both 2D and 3D lines and splines in 3D mode with the Move tool.
- When you move the sketch grid in Sketch mode or Section mode, press Shift to snap to parallel to planes, edges, and axes.
- You can now move along a path in Sketch and Section modes.
- You can now click a linear axis of the Move handle, then press Ctrl+spacebar to create a copy of the object at the specified dimension. Previously, you had to Ctrl+drag and then press the spacebar.
- If you select a linear axis of the Move handle, you can now move an object up to an origin axis.
- You can now use the Move tool's Up To tool guide when creating a pattern, as shown in the figure on the right.
- You can now use the Move tool's Orient to Object tool guide to orient an edge to an origin axis.
- The Move tool guides are now available when you are moving the sketch grid in Sketch and Section modes, including the Up To and Orient To Object tool guides.
- You can now anchor the Move handle to the intersection of two axes in 3D mode.



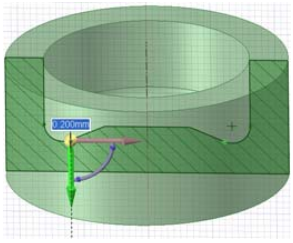
- You can now use the ruler to dimension the movement of the sketch grid.
- You can now select multiple components with the Move tool's Select Component tool guide.
- Once you anchor the Move handle, it is fixed. It no longer re-orients itself when you select additional objects.
- If you move a line that has a mirror relationship with the Move tool in Sketch mode, the mirrored line also moves.

Patterns

- Performance is enhanced when creating and editing patterns.
- You can now create a pattern of notches that add and remove material from multiple faces along a sheet metal edge, as shown in the figure on the right.



- Pattern dimensions are now displayed when you select a pattern member with the Move tool.
- You can create linear and rotational patterns of solid or surface components.
- When you place the Move handle on a tangent edge of a round in 3D or Section modes (as shown in the figure on the right) and move the edge, the Move handle can reposition itself on the new edge location. You can then modify the dimension of the edge again at the new location.

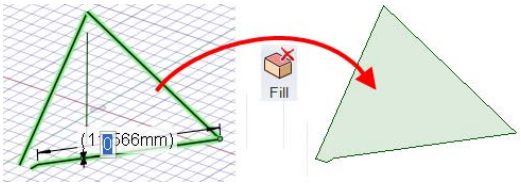


- You can now create a pattern with a mixed set of faces and components, such as a pattern of holes (faces) and bolts (imported components).
- The Move tool's Create Patterns option is now disabled by default.
- You can now pattern a vertex or points along an edge. All points are associated to the edge, so that when the edge moves, the points move with it.
- You can no longer re-anchor pattern dimensions.
- You can now dimension the incremental rotation angle in a rotational pattern to edit the pattern.
- You can now make a pattern of patterned components and pattern components along a trajectory.
- You can now maintain the orientation when patterning objects along a trajectory by checking the Maintain orientation checkbox in the Options panel.
- When creating the end pattern member, if you edit the distance dimension, it moves the new, end pattern member the specified distance from the original member.
- When creating a pattern of objects or components, the first object in the pattern is no longer pasted on top of the original object, and the count is modified so that it is correct.

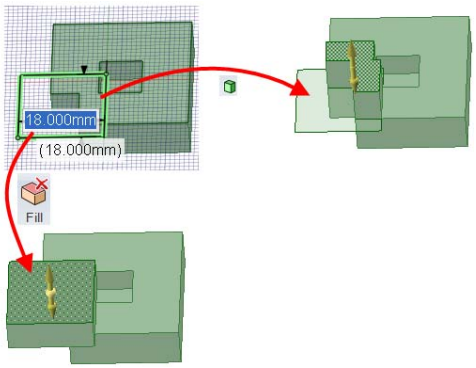
Fill

The following enhancements were made to the Fill tool:

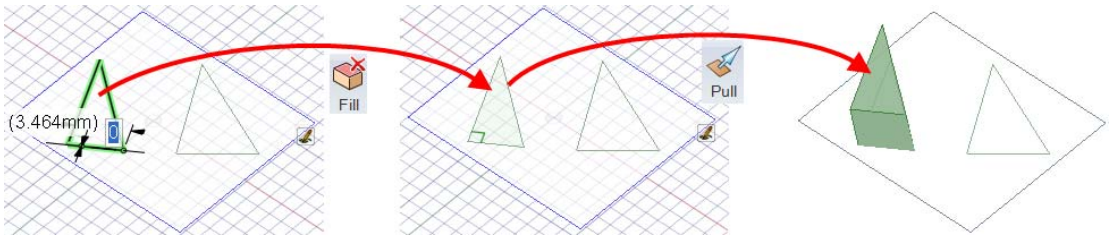
- You can now use the Fill tool in Sketch mode, as shown in the figure on the right.



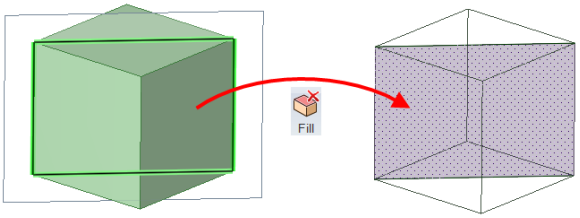
- This new Fill functionality is useful when you sketch faces across section lines, but do not want the section lines to create multiple surfaces when you switch to 3D, as shown in the figure on the right.



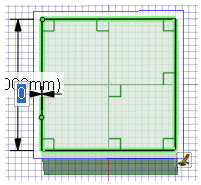
- You can now use the Fill tool while editing a layout to create a closed, flat surface (if possible) from a set of selected lines. If you fill lines in a layout mode, you can then pull the surface into 3D from the layout, as shown in the figure below.

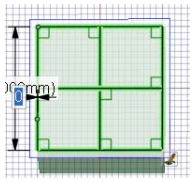


- You can now fill lines and edges, as shown in the figure on the right, even when the sketched lines you want to fill were not sketched in the same plane as the edges. (Please note that if the lines are imprinted on a face and become edges, filling those edges deletes them.)



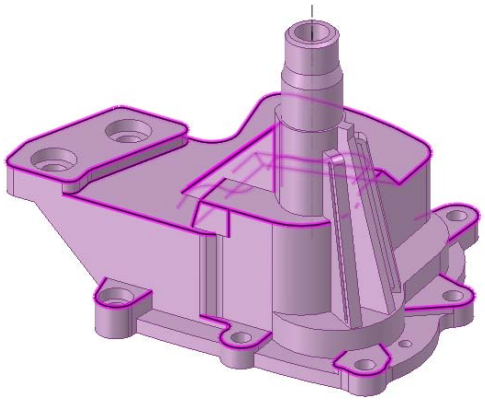
- You can now use the Fill tool when editing a layout. To simplify a surface, select the outside lines and click the Fill tool as shown in the top figure on the right. To keep internal line, select them, as shown in the bottom figure. (Using the Fill tool when editing a layout will not switch you to 3D mode.)



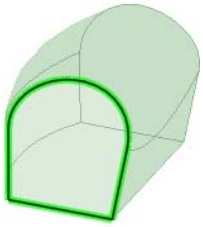


- The Fill tool now deletes redundant inner edges when you fill a closed edge.
- You can now fill planar lines that almost create a closed loop.
- If the Fill tool cannot fill a sketched line with multiple, larger gaps, it displays multiple error messages, one for each gap.
- When you try to Fill sketch or layout lines that almost make a closed loop, and the gap between the two endpoints is more than 1.5 times the minor grid size, the endpoints flash in red and an error message is displayed.
- You can now merge edges with the Fill tool. Select the edges you want to merge, then click Fill.
- You can select multiple edge loops in the same solid or surface then click Fill to fill them – even if, when extended, the faces containing the edge loops would connect to other faces, as shown in the figure on the right.

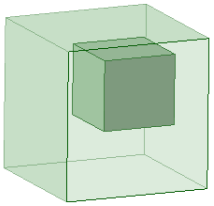
- If you delete the edge of a surface that contains a single face, the edge is simplified to its most natural u-v line.
- If you have a surface comprised of multiple faces, you can now click an outside edge of either face (or an edge that crosses both faces) and click the Fill tool to simplify the edge.
- When selected round faces are removed by the Fill tool, they are removed in a specific order so that they can be restored by right-clicking and selecting Reattach Rounds for each group, in reverse order. The figure on the right shows edges from which rounds were removed by the Fill tool.



- You can now fill 3D edges across multiple surfaces when the surfaces cannot extend and they are not in a single plane, as shown in the figure below.



- If you remove the outside face of a solid that contains an internal void, the internal void becomes a solid, as shown in the figure on the right.

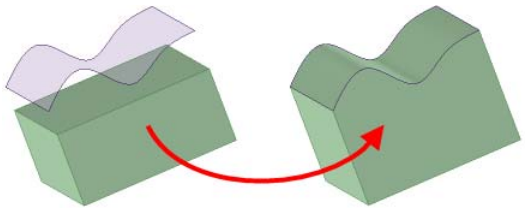


- On imported geometry with surfaces that have may have an “open” edge shown in black, you can manually create a solid from faces that enclose a volume by selecting the “open” edge (shown in black) and clicking the Fill tool.
- If you use Ctrl+X, Delete, or the Fill tool to delete faces and an invalid solid is created, the solid will be converted to a surface.

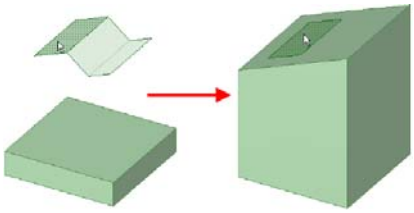
Replace

A new Replace tool was added to SpaceClaim 2008. It has the following functionality:

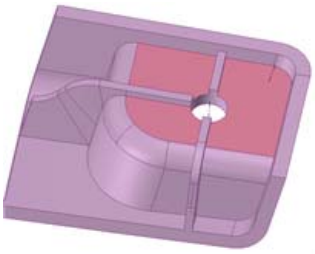
- The Replace tool was added to the Edit ribbon group to replace one face with another face or surface, as shown in the figure on the right.



- When you select a surface in the Structure tree as the source face, the target is extended to the entire selected surface.
- When you select only one face of a surface as the source face, the target is extended to meet the face, and the source face is extended as far as necessary to replace the target face, as shown in the figure on the right.



- When you select multiple, touching surfaces as the source, they are first merged, then the target is extended to the new, merged surface.
- When you select multiple faces of a surface as source faces, all the faces of the surface are automatically selected.
- You can use the Replace tool to align planar faces which are close to being aligned, as shown in the figure on the right.



- You can also simplify a spline face that is very similar to a cylinder with the Replace tool.

Edit as Blend

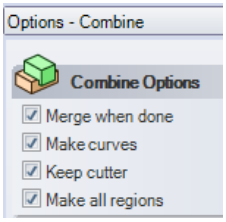
In this version of SpaceClaim, if you create a blended face using the Edit as Blend tool, then use another tool to extend the blended face, and then select the extended, blended face and open the Edit as Blend tool again, you can now insert a blend section plane anywhere within the extended portion of the face.

Combine

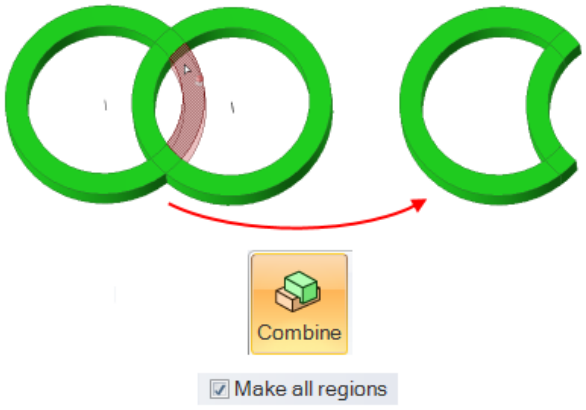
The following enhancements were made to the Combine tool:

- "Sticky" tool guides have been implemented in Combine. For example, if you want to select multiple targets to merge, click the Select Target icon after it is active to keep it active while you click all the targets to merge. To "unstick" the tool guide, you can click it again, click another tool guide, or click an empty place in the Design window.

- If you click the new Merge tool guide  , the selected targets are merged. You can click more objects to merge them to the targets or use box-select.
- Combine options are now displayed as shown in the figure on the right.



- The Make all regions option was added to the Combine tool. Use this option to cut the target solid with the cutter and the cutter with the target solid. This option is off by default. We recommend using this option along with the Merge When Done option to quickly merge all remaining regions, as shown in the figure below.



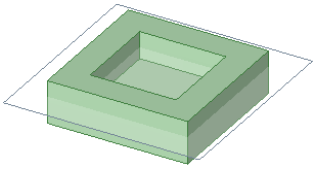
- The Combine tool now works in two modes: *Slow mode* is activated by clicking a Combine tool guide. In this mode, once a tool guide is clicked, it remains selected until another tool guide is clicked. In this mode, the Combine tool works just like any other tool. *Fast mode* automatically activates tool guides

and moves you through the workflow. This predictive mode exhibits the same behavior as the Combine tool had previously, although you can now box-select multiple cutters to add even more efficiency. In *slow mode*, you can now box-select multiple targets and cut them without automatically merging them.

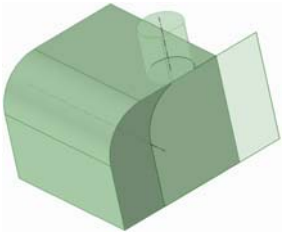
- When you cut a solid with another solid, you can immediately select another cutter. Previously, the Select Target tool guide was activated after a solid-solid cut. When you press Ctrl in the Combine tool, the cursor changes as shown on the right to indicate that clicked objects will be merged.

- You can now use the Combine tool to merge or split a disc intersecting a cylinder, as shown in the figure on the right.

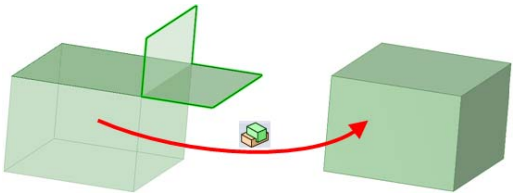
- You can now split a spline surface by a planar surface, then merge the spline surface regions back together.
- Merging multiple, intersecting solids and surfaces no longer deletes objects.
- If you combine a plane with a block that contains a hole, as shown in the figure on the right, the hole is now maintained instead of turning into a solid.



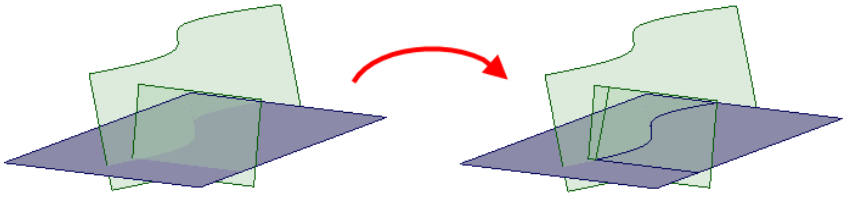
- You can now box-select when using the Combine tool.
- When you use the Combine or Split Body tools, the newly created objects have the layer properties of the previous objects.
- You can now use the Combine tool to merge or split a surface and solid when one of the surface's faces is coincident to one of the solid's faces, as shown in the figure on the right.



- You can now select edge loops of a self-intersecting surface and click the Combine tool to allow the surface to repair itself to a solid if possible, as shown in the figure below.



- Box-selection of surfaces that do not completely intersect will split each other and form new regions if possible, as shown in the figure below.



Split Solid

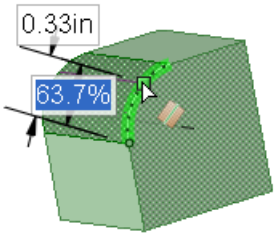
The following enhancements were made to the Split Solid tool:

- The Split Body tool was renamed to Split Solid, and now has sticky tool guides, *slow mode*, and works similarly to the Combine tool – the Select Cutter Face tool guide remains selected until another tool guide is clicked.
- You can now split the target solid by faces of another body or plane.
- In the Split Solid tool, you can only Ctrl+click splitter edges in the same solid.
- You can now select a cutter object from another component when using the Split Solid tool.
- The Split Solid tool now splits intersecting cylinders and cones.

Split Face

The following enhancements were made to the Split Face tool:

- The handles that appear when you use the Split Face tool's Select Cutter Point and Select Two Cutter Points tool guides are now colored yellow for consistency.
- When using the Split Face tool's Select Cutter Point or Select Two Cutter Points tool guides, mouse over an edge to display and edit the length along the edge, and the percentage of the edge that is between the first point and the end point, as shown in the figure on the right.
- If you select a point along an edge that is part of a pattern when using the Split Face tool, the percent distance along the edge is displayed.



Insert

In this version of SpaceClaim, inserted TIF images are rendered correctly.

Temporary objects

This version of SpaceClaim allows you to quickly create temporary objects for dimensioning and snapping. For example, you can use them to create a ruler dimension, or anchor the Move handle on them. These temporary objects disappear when you switch tools. You can:

- Create temporary planes, axes, and points by Alt+Shift+clicking a reference.
- Create temporary objects while working with other tools.

Plane

The following enhancements were made to the Plane tool:

- You can now select a temporary plane with the Plane tool to insert a plane.
- To create a plane, you can click the Plane tool, then click a plane reference. Or you can click the Plane tool, then press and release Shift on successive references to preview all the possible planes that can be made with that set of references, then click one plane to create it.
- If you click the Plane tool, then select the references to insert the plane, error messages appear if you select references that cannot define a plane.
- You can now create a plane located midway between a planar face and a plane.
- You can now create a plane through a point and parallel to a planar face by selecting a planar face and a point, vertex, or origin, then clicking the Plane tool.

Axis

In this version of SpaceClaim, to create an axis, you can click the Axis tool, then click an axis reference. Or you can click the Axis tool, then press and release Shift on successive references to preview all the possible axes that can be made with that set of references, then click one axis to create it.

Cylinder

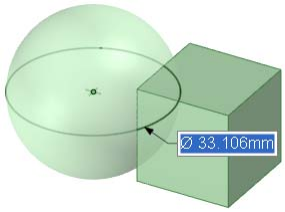
The following enhancements were made to the Cylinder tool:

- A new icon was added for the Cylinder tool to match the style of the Sphere tool's icon.
- The Cylinder tool was moved to the Insert ribbon group. It now works in all modes. In 3D mode, the two points selected define an arbitrary plane in which to place the axis. In Sketch and Section modes, the points are in the plane. You can select points on all faces, including cylinders and spheres.

Sphere

A new Sphere tool was added to the Insert ribbon group SpaceClaim 2008. It has the following functionality:

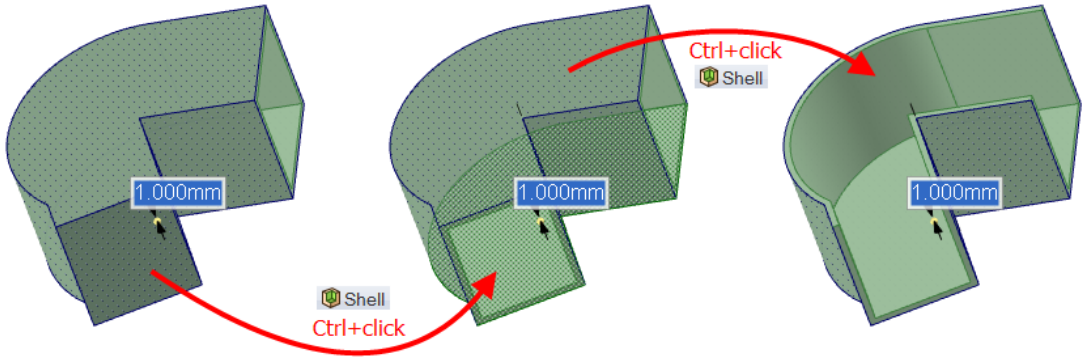
- You can quickly create a sphere in all modes. You can use the Add and Cut options to add or remove material.
- When you are creating a sphere, intersections with other objects are indicated with rendering changes, as shown in the figure on the right, so that the desired size of the sphere can be more accurately estimated.
- When creating a sphere or cylinder, the point snaps to edges and vertices.



Shell

The following enhancements were made to the Shell tool:

- You can now enter a dimension in the Shell tool. To edit the shell's thickness, click the dimension and enter a new thickness.
- To remove another face from a shell, just click it with the Shell tool.
- The dimension field in the Shell tool is now displayed near the face selected for removal, and you can press the spacebar or enter a value to edit the dimension.
- If you are using the Shell tool and Ctrl+clicking to remove an additional face, as shown in the figure below, and the removal of that face fails, the shell is kept in its last valid state.



- When you open or insert another file type and use the Shell tool, the first surface that cannot be offset because it is too curved for a region flashes red, and an error is displayed in the status bar.

Offset

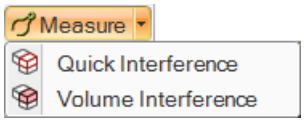
In this version of SpaceClaim, the Offset tool's Find All Same Offset option is not selected by default. If this option is not selected, the offset relationship is created only for the selected pair of faces.

Mirror

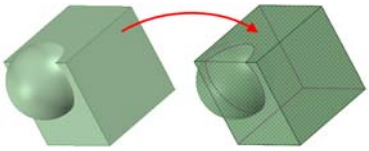
The following enhancements were made to the Mirror tool:

- You can now mirror components with the Mirror tool by selecting them in the Structure tree.
- If you are using the Mirror tool, and an action fails, a message appears.
- You can now use the Select Mirror Plane tool guide to select a different plane to use as the mirror plane, even if you already selected one within the Mirror tool.
- You can now mirror a component that is comprised of patterned solids.
- You can now set up a face-mirror relationship between cylindrical faces.
- 3D points (such as those created by a pattern) and sketched points no longer retain their mirror associations once they are created.

Measure



- New icons were added for the Measure tool's Quick Interference and Volume Interference sub-tools, as shown in the figure on the right.
- To see the volume created by the intersection of multiple objects, select the Measure tool's Volume Interference sub-tool. Then Ctrl+click the intersecting objects to display the volumes created by their intersection.
- The Quick Interference sub-tool was added to the Measure tool drop-down to allow interference detection. To see a quick interference sketch, select Quick Interference, then click two objects that intersect. The intersection is displayed with purple lines, as shown in the figure on the right.



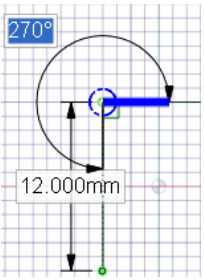
- The Measure tool now displays planar surface area in feet and inches if you selected that option within the Units page of the SpaceClaim options window.
- You can now measure from a 3D point to any other vertex in the design, regardless of which component or subcomponent contains the point or vertex.

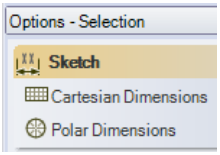
Sketch

The following enhancements were made to the sketch tools and editing functionality:

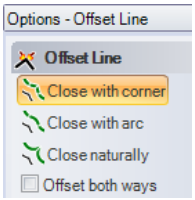
- Sketching in Section with the auto extrude option enabled now displays a preview of the 3D geometry that will be created as you sketch.
- You can now sketch on a drawing sheet with 3 views when you start a new design using the new option shown on the right.

- Dimensions persist after you create a rectangle, line, arc, tangent line, or offset line in Sketch mode, and you can click on a dimension to edit it, as shown in the figure on the right. They disappear when you select another tool or begin drawing another sketch.
- When editing sketch dimensions, you can right-click to display several new menu options.
- Sketch editing has been enhanced with more dimensional control. It also now has Cartesian and Polar dimension editing options, as shown in the figure below, and a new tool guide for dimension editing relative to other objects.



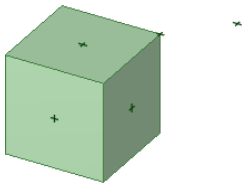


- The Three-Point Circle Segment option was added to the Three-Point Circle tool. Use this option to define an arc as a segment of a three-point circle.
- Multiple options were added to the Offset line tool, as shown on the right.

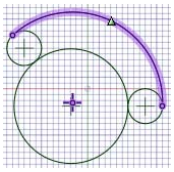


- The Trim/Extend Curve option was added to the Create Corner tool. If you select this option and click the first line, then click a second, non-intersecting line, you extend the first line. If the two lines intersect, the first line is trimmed by the second line.
- The Use Internal Radius option was added to the Polygon tool. Select this option to dimension the polygon based on the diameter of an inscribed circle. Uncheck it to dimension the polygon based on a circumscribed circle.
- The Bend tool was moved to the Sketch ribbon group.

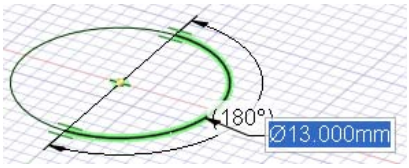
- The Snap View tool is now disabled in Sketch mode.
- You can now see sketched points in 3D mode, as shown in the figure on the right.



- The Point tool is disabled when you are editing a layout.
- To sketch on an empty drawing sheet (that is, one with no views) when no plane is selected, you must first select the plane on which to sketch. You can pre-select the plane before entering Sketch mode or you can right-click and select Select New Sketch Plane to select a new plane.
- You can now double-click to cycle through and select successive edge loops in the Project to Sketch tool.
- You can now box-select objects, including silhouette edges, within the Project to Sketch tool.
- You can now press Shift while hovering over an object with the Offset Line tool to dimension the offset from that object instead of from the line being offset.
- You can hide a closed sketch line in Sketch mode by placing it on a layer and hiding the layer.
- If you sketch two circles that are tangent to each other, changing the diameter of one circle by editing its dimension maintains tangency with the other circle.
- Annotations on a layout plane now move along with the plane.
- You can now sketch an arc tangent to two circles, as shown in the figure on the right.



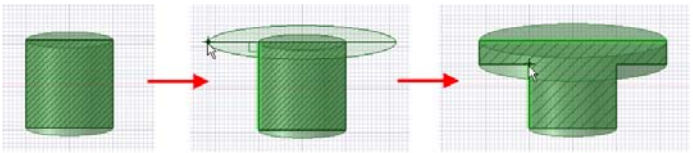
- When you copy a sketch line that is on a layer, the copied line is placed on the same layer.
- Dimensions remain at the same location when you are creating or editing a sketch line instead of moving around as you work with the sketch.
- If you create a circle by sketching two connected tangent arcs, you can now dimension the diameter of the circle they form, as shown in the figure on the right.

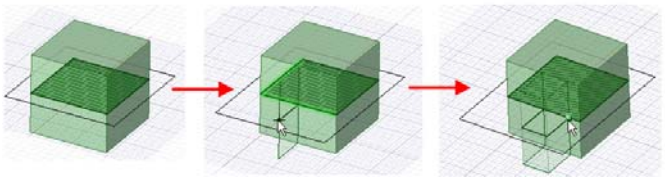


Section mode

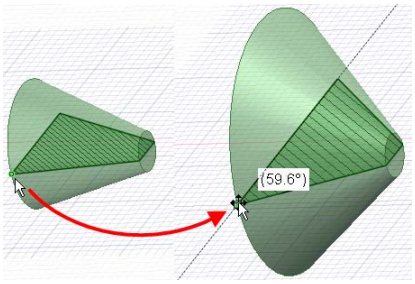
The following enhancements were made to Section mode:

- You can now control whether sketching in Section mode extrudes the sketched lines as you draw them. To control auto-extrusion, click the Application menu > SpaceClaim Options > Popular and select the Enable sketch auto-extrude option.
- You can sketch on a cylindrical face or edge to create a fully revolved profile, or sketch on a planar face to auto-extrude your sketch to the extent of the face. To revolve or extrude your sketch, make sure you begin your first sketched line on the existing face or edge. This first line will immediately form a surface, and closed regions add to the solid.
- When you sketch a rectangle in Section mode, the top and bottom faces are automatically closed to create a solid when the Auto-extrude/revolve sketches in Section mode option is selected, unless the rectangle begins on the edge of an existing object, as shown in the figure below.





- When you are sketching in Section mode, a depth is automatically assigned and displayed as a dimension. Press Tab to edit the dimension.
- Switching to Section mode in a drawing sheet with multiple cross-section views allows you to immediately modify any face by modifying its edges, because the plane of the drawing sheet as assumed to be the section plane.
- You can now pull up to objects in Section mode.
- You can now edit a cylinder by entering the length and dimension in Section mode.
- The Create Corner, Create Rounded Corner, Trim Away, and Split Line tools now work in Section mode.
- You can select a point on a cone in Section mode and drag to change the overall cone angle, as shown in the figure on the right.



- Performance when changing the view of large models in Section mode was significantly improved.
- You can now snap circles to arcs in Section mode when creating or editing them.
- The Project to Sketch tool now projects cylinder edges onto the sketch grid in Section mode.
- You can now select a face (by selecting an edge) in Section mode, then click the Project to Sketch tool to project the selected object onto the sketch grid. If you have difficulty seeing a projected line, check that Fade Scene Under Grid is selected on the Display tab.

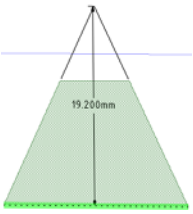
Detailing tools

Many new detailing tools were added, and enhancements were made to most existing detailing tools.

Dimension

The following enhancements were made to the Dimension tool:

- You can now set the number of decimal places for each dimension annotation by selecting the dimension annotation and modifying its Decimal Places property.
- You can now use the scroll wheel to select occluded objects when creating dimension annotations.
- You can now create virtual sharp for dimensions and notes, as shown in the figure on the right. You can drag the virtual sharp's end point, draw another leader to the virtual sharp, create virtual sharps in cross-section, for rounds, and between an angled and straight edge.

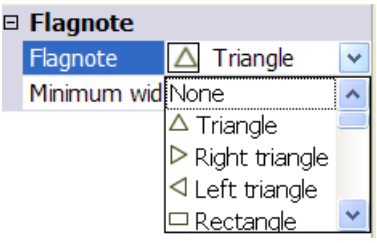


- You can now create annotations when the sketch grid is displayed. The annotations are placed on a temporarily-activated annotation plane that coincides with the sketch grid.
- You can now copy and paste the dimension text displayed on a drawing sheet into any annotation.

Note

The following enhancements were made to the Note tool:

- You can now resize the Insert Field window.
- Icons were added to the Flagnote drop-down menu, as shown in the figure on the right.

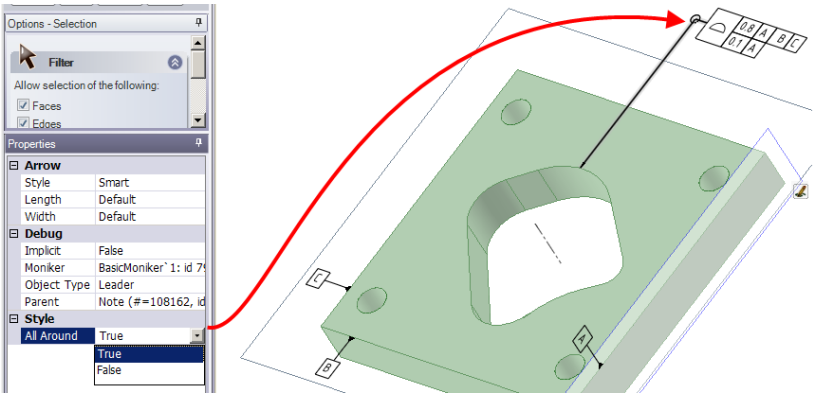


- Symbols in the drop-down menu now display the correct symbols based on the drawing standard selected in the SpaceClaim options (ASME, ISO, JIS).
- You can copy the content of one note and paste it into another note, then edit the pasted text.

Note Leader

The following enhancements were made to the Note Leader tool:

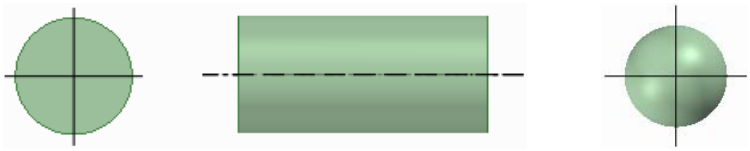
- A new icon was created for the Note Leader tool .
- You can now attach a note leader to a vertex.
- You can now add an all-around symbol on note leaders, as shown in the figure below.



- You can drag note leader line segments to change both bordering jog points.
- You can delete note leader line segments by deleting the jog points that border the segment.
- Note leader line segments highlight separately from the note leader when moved.

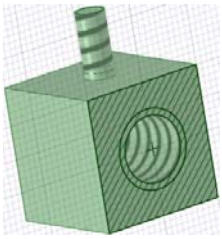
Center Line

A new Center Line tool was added to the Annotation group on the Detailing tab. Examples of center line annotations are shown in the figures below.

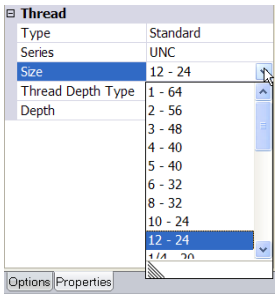


Thread

A new Thread tool was added to the Annotation group on the Detailing tab. It has the following functionality:



- You can now create threaded surfaces, as shown in the figure on the right. To create a threaded surface, select the Thread tool , then click the edge of a cylinder or cone. Depending on what you clicked, an inner thread or outer thread object also appears in the Structure tree. (Broken threads appear with a small yellow triangle over the icon.)
- The thread data XML files are in the installation directory's Threads subdirectory. You can also specify a different directory for the files by selecting SpaceClaim Options > Files from the Application menu.
- The visibility of the thread texture is controlled by the visibility of the surface.

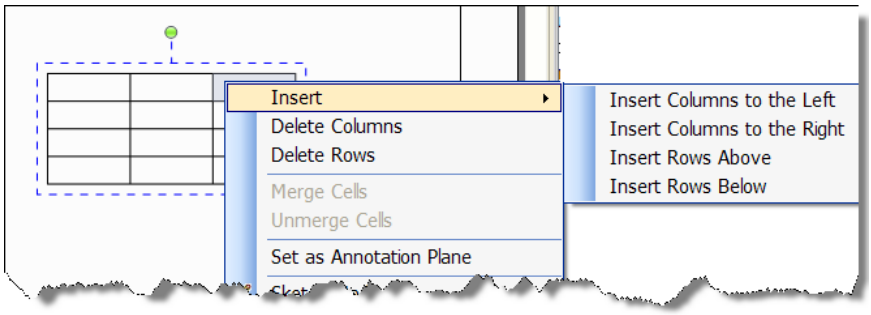


- Threaded surface properties are displayed in the Properties panel when the surface is clicked with the Thread tool, as shown in the figure on the right.
- You can set the Thread Depth Type property by selecting **Blind** or **Full Thread**.
- You can change the thread type from standard to custom which controls whether threads are selected from a table or entered manually.
- You can now change from the ISO to UNC thread series (or any other valid thread table in the correct directory) without changing the current thread size.
- You can now include the Thread Series and Thread Size property values when inserting a field into a note.
- The Type property controls whether threads are selected from a table or entered manually. For cylinder threads, select Standard to select values for the other properties from a drop-down list. The most likely values are selected by default (next smaller size for external threads, and the next larger size for internal threads). Select Custom to enter offset values. For tapered threads (threads made on conical surfaces) you can only use the Offset option

Table

A new Table tool was added to the Annotation group on the Detailing tab. It has the following functionality:

- You can now place a table on an annotation plane using the Table tool. To add a table, add an Annotation plane or activate an existing plane, then click the Table tool. Drag to create the table dynamically. You can:



- Table properties are displayed in the Properties panel when the table is selected. You can use these properties to set the number of columns and rows, row height, column width, and cell alignments and margins.
- You can copy and paste table annotations.

Surface Finish

In this version of SpaceClaim, surface finish symbols now move along with the surfaces they are attached to.

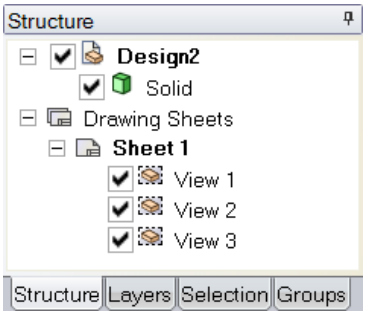
Drawing Sheets

The following enhancements were made to the Drawing Sheets tools:

- The SCDOC files used as the standard drawing formats were updated to fit the values in custom fields.
- You can now put a drawing sheet view on a layer.
- Dimensions associated with drawing views are now hidden when the visibility of the view is turned off in the Structure tree.
- When you create a new drawing sheet (for an empty design), the sheet contains only the view outlines. These handles can control the size and position of the view, and modifying one view changes the

related views as appropriate. You can delete the handles and move the view using the outline. Click within a view's boundary with a sketch tool to display a sketch grid.

- Drawing sheet views now appear in the Structure tree. New icons were added for each view, as shown in the figure on the right.
- You can select a face of a solid in the drawing sheet when only the edge is displayed by using the scroll wheel or by rotating the drawing sheet or selecting the face in another view.
- When beginning a design from an empty drawing sheet, you can right-click a design on the drawing sheet and select Open Component to display the design in a new Design window.
- You can switch from a rectangular boundary to a spline or circle boundary while creating the boundary of a detail view.



3D Markup

In this version of SpaceClaim, if you are working on a 3D Markup, you can right-click a lightweight component and select Load Component to load it.

Display

New display tools were added, and enhancements were made to existing display tools. The following enhancements were made:

- To view your design clipped by a plane, right-click the plane and select Clip with Plane. Planes, axes, and annotations are not clipped. To restore the view of your design, right-click the plane and select Clip with Plane again.

Orient tools

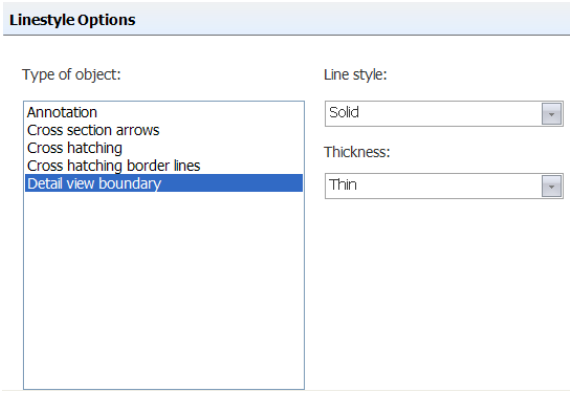
The following enhancements were made to the Orient tools:

- A new cursor  was developed for the Snap View tool.
- The On Cursor Spin option is enabled by default.
- The Snap View tool was added to the Orient ribbon group on the Display tab.
- You can now right-click in the Design window and select View > Zoom Box In to draw a box to zoom in to.
- The Rotate about pre-selected object in Spin advanced SpaceClaim option is no longer enabled by default. You can press Alt and highlight an object to rotate around it whether or not this option is selected.

Line Style and Line Weight

The following enhancements were made to the Line Style and Lineweight tools:

- Line style options were added to SpaceClaim Detailing options, as shown in the figure on the right.
- Line styles can now be assigned to layers so that you can have different line styles for sketch and layout lines.
- Line styles can override the Detailing option settings.
- The color of hatch lines and cross-section view borders match the body color when a thick line style is selected.



- All line styles are now displayed, whether or not Thick is selected as the line style on the Detailing page of the SpaceClaim Options window.
- Line weights are now applied to table borders and note border lines.

Face Style

In this version of SpaceClaim, if you apply the Opaque face style to a set of surfaces, and you delete a face or merge a face, the original set of surfaces stays opaque as long as no new surface or solid body is created.

Show tools

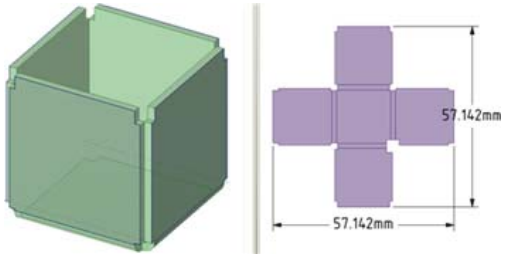
The following enhancements were made to the Show tools:

- The Ruler, Tooltip, and Tool Guide checkboxes were removed from the Show ribbon group on the Display tab. You can still modify the display of these tools in the SpaceClaim Options > Popular page, where they are called Show tool ruler, Show tool tips, and Show tool guide.
- The Use Line Weights option was added to the Show ribbon group on the Display tab. Select this option to switch the line style of sketched lines from thin to the thickness set by the Line Weight tool in the Style ribbon group.
- The Adjacent Entities tool was added to the Show ribbon group on the Display tab. Use this checkbox to turn on or off the faint highlighting that appears on edges and faces adjacent when you hover over a point or edge.
- The Show Spin Center SpaceClaim option and the Spin Center checkbox in the Show ribbon group on the Display tab are now linked so that when you select one, the other is also selected.

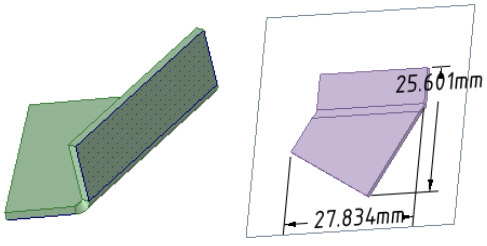
Sheet metal

The following enhancements were made to the SpaceClaim's sheet metal functionality:

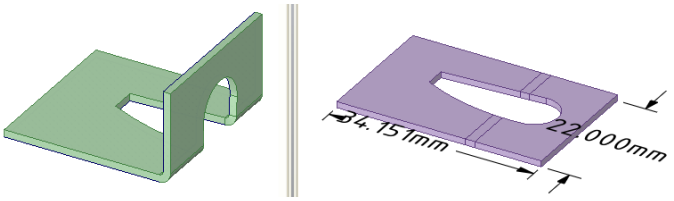
- A new, sheet metal junction option was added: Hard Bend. Use this option to remove the rips and bends from the geometry and return to a shell, where there is no inside and outside radii.
- A new, sheet metal junction option was added: Remove Junction. Use the option to keep the geometry, but remove the junction association between the edges, to allow for pulling the extents of side of a junction.
- You can now select a rip bend relief as a Sheet Metal option.
- A Create Relief tool was added. This tool lets you make cutouts on vertices of offset solids in preparation for rip and bend assignment to the edge. To create a corner relief, select a vertex and click the Corner Relief tool. The figure on the right shows corner reliefs created on all the outside corners of a shell.



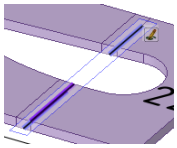
- You can now override the calculated bend allowance for a selected bend in the Properties panel. When you unfold a part, the edited bend allowance is used.
- You can now rotate a sheet metal "hem" (180 degree bend) to form a flange (90 degree bend).
- If there is an error unfolding a sheet metal part, both the face that is problematic and the edge that stops the unfold from succeeding are highlighted.
- You can now created a skewed bend in a sheet metal part and unfold it, as shown in the figure on the right.



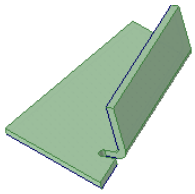
- You can now sketch regions on the unfolded part, use them to cut the part, and see the result on the folded part, as shown in the figure on the right.



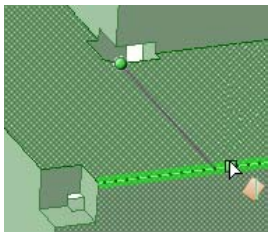
- Bend lines for an unfolded part are now placed on a Bends layer, with the visibility off. When you turn the layer's visibility on, the bend lines appear as shown in the figure below.



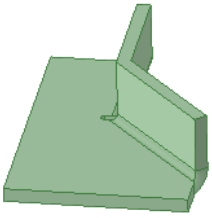
- A Failed to create relief message appears when a bend relief cannot be created, such as when it is too close to the side.
- A skewed bend face can now be rotated, as shown in the figure on the right. Handles are now positioned properly, and any necessary reliefs are created correctly.



- If you place an origin on a sheet metal wall, it now unfolds with the wall on which it is placed.
- You can now use the Selection filter to select only vertices (by de-selecting faces and edges), and then box-select a sheet metal part to quickly select all the corners needing corner reliefs.
- When you use a sheet metal part that has been unfolded, the Unfolded window is no longer displayed when the part is used as a subcomponent in another assembly.
- If you split a sheet metal face, a corner relief is created automatically, if it does not already exist.
- When you select the face of a sheet metal component with the Split Face tool and use the Select Two Cutter Points tool guide to create a rip across the sheet metal face, you can now click the face to fill the rip. The tool guide also snaps to 90 degree intersections with other edges.
- You can now select the face of a sheet metal component with the Split Face tool and use the Select Two Cutter Points tool guide to create a rip across the sheet metal face. This can connect two corner reliefs, two points on two edges, or a combination, as shown in the figure on the right.



- When you create a corner junction the No Overlap option is used by default.
- Bend relief dimensions were added to the Sheet Metal page of the SpaceClaim options window. You can set the default width, depth, and select which bend relief type to use by default.
- When you modify a sheet metal property, the value is displayed in bold.
- You can reset a sheet metal property to the default value by deleting the value.
- Right-click a sheet metal component and select Validate Body to display in red any areas that prevent the part from unfolding.
- You can now edit the depth, width, and type of a bend relief in the Properties panel. To edit the bend relief, right-click the bend face and select Properties. You can switch from one type to another, as well as edit the depth and width of square and rounded bend reliefs.
- You can now create a corner relief when there is a mix of convex and concave corner edges intersecting at the corner.
- You can now pull a sheet metal wall when the neighboring edge makes a non-90° intersection, as shown in the figure on the right.

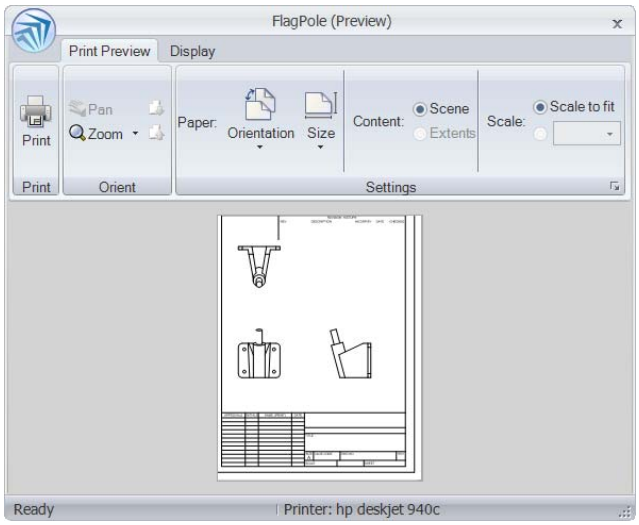


- When you are working with an unfolded sheet metal part, only the relevant tools are enabled.
- You can now pull multiple slots back into holes if the slots were created in a sheet metal part.

- You can now pull a sheet metal wall up, then reverse the pull direction and pull the wall back down.
- When the No Overlap junction option is selected, the creation of that junction type no longer depends on the order in which edges are pulled to create walls. Neighboring edges can be pulled up together, or one at a time, in either order, and the junction will be the same.

Printing

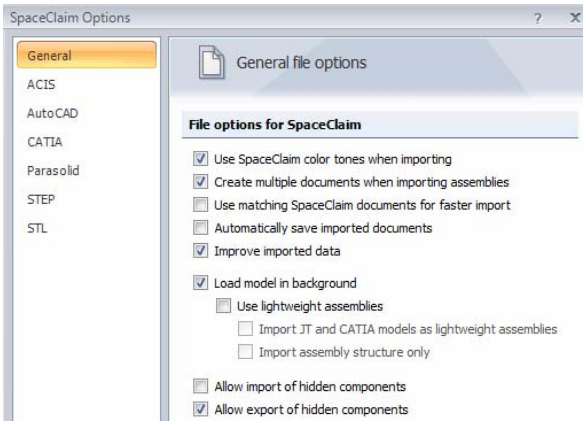
The following enhancements were made to the SpaceClaim's sheet metal functionality:



- A Print Preview window was added, as shown in the figure on the right. To access this window, select Print > Print Preview from the application menu. The following functions are available on this window:
  - You can zoom into or out of the page, or zoom the page to the extent of the preview window.
  - You can set the page's orientation to portrait or landscape.
  - Click in the Settings ribbon group to display and set the print properties.
  - Click Print to print the page as shown in the preview window.
- When printing, paper size is now selected by default.
- You can now print designs displayed in the Perspective Shaded graphics style.
- You can now view printer margins using the Display tab on the print preview window.
- You can now set the size of the printed design by selecting Scene or Extents in the Contents tool on the print preview window.
- You can now print a drawing with both shaded and non-shaded graphics style in different views.

Importing and exporting

The following enhancements were made to SpaceClaim's import and export functionality:



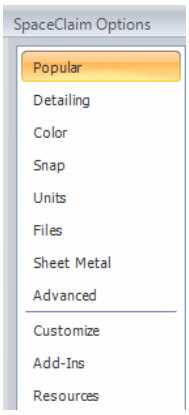
- The File Options window UI now appears as shown in the figure on the right.
- New options were added to the File options General page:
  - You can now select the Popular > File Options > General > Improve imported data option to improve a file when it is opened or inserted. De-selecting this option imports the file without stitching, healing, or other improvements. As a result, the file appears much more quickly.
  - A new Structure only option was added when you open or insert another file type. Select this option to import only the Structure tree.
  - You can now open or insert a JT or CATIA file as a lightweight component.

  - Select Allow import of hidden components to open or insert hidden components within files and turn off their visibility in the Structure tree. This option works for CATIA v5, Parasolid, and Solidworks.
  - Select Allow export of hidden components to save components that have their visibility turned off in your design as hidden components when you save them as any other file type.
- When you save a file as a 3D PDF file.
- You can now save a SpaceClaim design as a CATIA V18 file.
- When you import a JT file with the Use Lightweight Assemblies option enabled, you can save the unloaded lightweight component as an SCDOC file with only rendering data (that is, without geometry data). You can open this SCDOC file in a new design and load it normally to include the geometry data.
- The SCDOC file now has an external package relationship to the original, imported JT file. This relationship is used to know which file to re-import when you right-click a lightweight component and select Load Component.
- Object IDs for edges, faces, and bodies are now stored within the SCDOC file. Object IDs are preserved when other files are opened or inserted into SpaceClaim, and the IDs can also be exported. For example, if you export a design to an analysis company, and they tag geometry with load positions, boundary conditions, and so on, then when you re-import that design, make changes, and re-export to the analysis company, they will not need to recreate their tags on the new design.
- When you save a design with a shaded graphics style as a DWG file, it is converted to the hidden line style.
- Dimension annotations that have had the number of digits modified now retain that property when exported to AutoCAD. This property is also retained when an AutoCAD file is opened or inserted.
- Centerlines are now included when you save your design as an AutoCAD file.
- You can no longer select the Direct to AutoCAD file option.
- If there is an invalid character in the path of a file you are trying to open or insert, that character is replaced with a valid character to avoid errors.
- Point-curve text files opened or inserted in SpaceClaim now display a closed curve when the file has a repeated value.
- Point-curve text files with columns separated by commas can now be opened or inserted in SpaceClaim. This feature allows you to import any comma-separated value file into SpaceClaim.
- Point-curve text files with multiple curves read the first column of data as both the number of the curve and the z-value, allowing curves to appear at different heights.
- Table annotations are now saved when you save your design as a DXF or DWG file.
- Origins are now saved when you save your design as another file type.
- Patterns of points are now exported when you save your design as an SAT or SAB file.
- When you save a sheet metal design as a DXF file, notes and bend lines are saved on the same layer, and the overall unfold dimensions are removed.
- You can no longer rename the top-level component or external components after they are saved to a file. The name is locked as the name of the file.

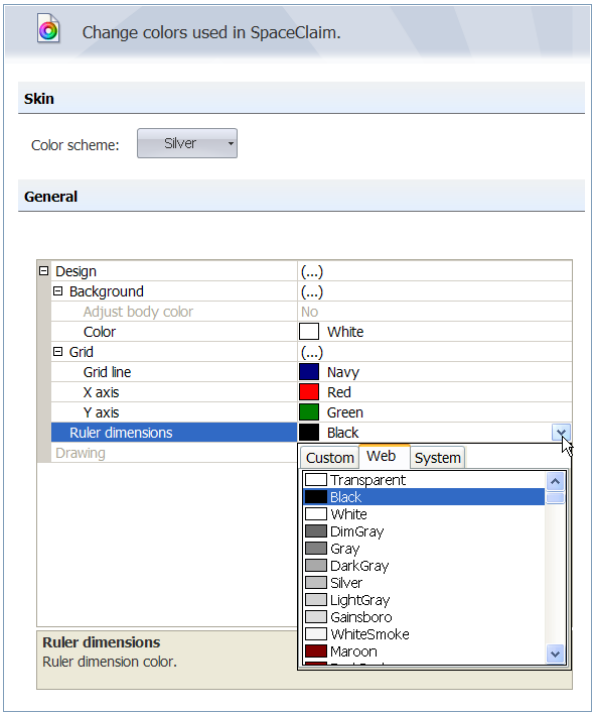
SpaceClaim options

The following enhancements were made to the SpaceClaim options:

- New Help icons were added to the SpaceClaim Options windows so that you can now get help on options without first closing these windows.
- The groups in the SpaceClaim options window are now ordered as shown in the figure on the right.
- Grid and other unit options are now stored with each design, and were placed on their own SpaceClaim options page.



- When you select Imperial as the unit type on the Unit options page, Decimal is selected from the Decimal/fraction drop-down by default.
- The options for customizing and changing SpaceClaim colors were placed on their own SpaceClaim options page, as shown on the right.
- You can set the background color of the Design window by selecting a Color value, set the colors of the sketch grid lines with the Grid options, and set the color in which ruler dimensions are displayed.
- The new Auto-extrude for sketches option is located on the Advanced options page.
- The Load model in background and Use lightweight assemblies options were moved from the Advanced options page to the General File options page.
- The Animate geometry option label was changed to Animate Full Pull in the Advanced SpaceClaim option group.
- A Linestyle option group was added to SpaceClaim detailing options. These options will allow you to modify the line style and thickness for each object type when it is complete.



- You can now control the transparency with which the cursor interaction arrows appear using the Transparency slider on the SpaceClaim Options > Popular page.
- The Linear snap option for solids gets its units from the settings for All New Documents on the Units page of the SpaceClaim options window. The units in which sheet metal property values are displayed are also set from there.
- A Snap to Grid icon was added to the Options panel for every sketch tool. Use this option to turn snapping on or off while sketching. It is selected by default.
- The Snap options page was rearranged into Sketch and Solid options, and an Angular snap option was added to the Sketch options.
- You can now set the text height for annotations in millimeters, inches and points.