SpaceClaim 2009 SP1

Release Notes

SPACECLAIM
CORPORATION

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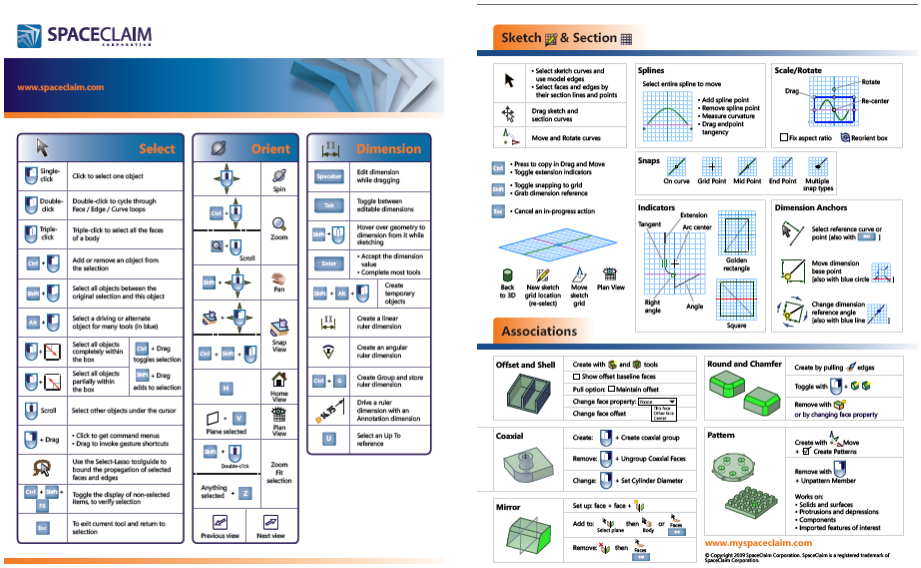
General

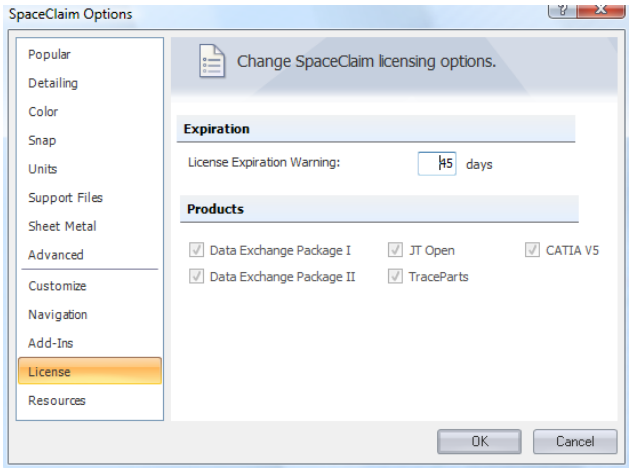
- The Quick Reference Card has been updated and added to the SpaceClaim web site, online help, and the Welcome Screen in all supported languages. You can find it at MySpaceClaim.com or under Getting Started in the online help.



- The Help translations in German, French, and Japanese are now available.
- A wait cursor is now displayed when you click Undo or Redo and it takes more than one second to complete.
- The icons have changed from paint blobs to squares in the Color menu and in the Layers panel.
- The default setting for Density units is now “Derived” to reduce the confusion arising from users that might mix unit systems unintentionally.

Licensing

- The Floating license configuration has been enhanced to allow users to release licenses for use by others, such as optional data exchange modules from within SpaceClaim. This can be done de-selecting the product check box as shown below. This is a dynamic update and is remembered between sessions.



Navigation

- New Mouse Wheel options have been added to SpaceClaim Options on the Navigation page:
    - **Zoom in Quick Zoom (Ctrl+MMB) and in Zoom mode**: The mouse wheel only zooms while you are in zoom mode or if you hold Ctrl and drag with the middle mouse button. Click Zoom in the Orient group on the Design tab (or let up on the Ctrl key) to toggle zoom mode. This is the default SpaceClaim behavior.
    - **Zoom while in Spin/Pan/Zoom (On Center) mode**: The mouse wheel zooms while you are in Spin, Pan, and Zoom modes. If you don't select this option, you can hold Ctrl to zoom while in these modes.
    - **Zoom always (hold Ctrl to query select)**: The mouse wheel always zooms. Hold the Ctrl key to select objects that are behind other objects (also known as query selection). This behavior is common in other CAD and 3D systems.
- New Spin Method options have been added to SpaceClaim Options on the Navigation page:
    - **Arcball**: Rotates the view using the spot where you start dragging as the rotation axis. This is the spin method that has always been used in SpaceClaim and is the default.
    - **Turntable**: Rotates the view using the top of the parts as the up direction and the location of your mouse as the rotation axis. This is the spin method common in architectural and ID 3D packages, like Rhino and SketchUp®.
    - To see the difference between these modes, make a rectangle and pull it into a 3D box. Now rotate it from right to left in a straight line.
- New Spin Center options in the right-click menu:
    - **Set**: Sets the spin center to the center of the object you have selected. You must select an object first. If you previously set the spin center, setting it again will change it to your current selection. You don't need to clear the spin center to set it again.
    - **Clear**: Clears the spin center. When you rotate your view in 3D, the spin will be centered based on the option you select in the Spin tool. If you select On Center, the view will rotate on the center of the current view. If you select On Cursor, the view will rotate on the spot where you clicked to start rotating.
    - **Locate**: Centers the view on the spin center, if one is set.
- New keyboard shortcuts have been added for Spin Center: Set (Ctrl+T), Locate (Ctrl+L), and Clear (Ctrl+Q).
- The spin axis displayed in the Design window looks different when you have the spin center set with right-click Spin Center > Set command. The axis is displayed when you select Spin Center in the Show section of the Display tab. The illustration to the right shows the default axis on the left and the axis for Spin Center > Set on the right.

Structure tree

- You can now drag and drop a curve folder from one component into another in the Structure tree.
- You can now select sketch curves, then right-click, and select the Move to New Component command. This command creates a new component and places the curves folder in the new component.
- Planes, axes, and origins are now highlighted as dimmed geometry in the Design window when you mouse over them in the Structure tree, even when they are hidden. This helps the user figure out which item should be shown, without having to turn them all on and off until the desired one is found.
- You can now use Internal Copy on many files at the same time. Previously this command had to be done one component at a time. Success or failure is reported for each file.

- Components have a new property named Component Name, which is appended to the Part Name and displayed in parenthesis in the Structure tree, as shown on the right. You can change this name in the Properties panel when you select a component.

- You can now right-click and select Delete to delete a curves folder in the Structure tree. The sketch curves in the folder will be deleted.
- Curves that belong to inactive components are now dimmed to indicate that they can't be selected in the Design window.

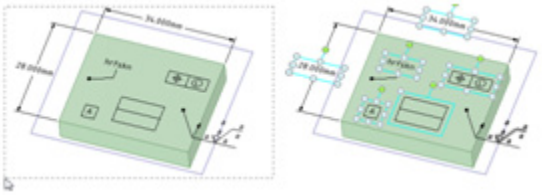
Copy and Paste

- Pasted objects are now positioned in their original location relative to the world origin unless you select a reference object before you paste. Previously, the pasted object was positioned at the world origin if you didn't select a reference.
- Copied objects that can be named (components, bodies, axes, or planes) will retain their names when pasted.
- Now when you paste an object and it is located outside of the visible area in the Design window, SpaceClaim will adjust the zoom to make the object visible.
- SpaceClaim now switches to 3D mode after you paste an object.

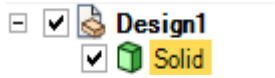
Select

General selection

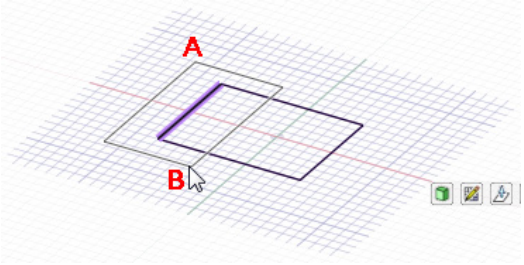
- You can now select annotations (notes, dimensions, symbols, and tables) on a drawing sheet and in the Design window by using box select, as shown on the right. This allows you to delete or change their display properties such as font size.

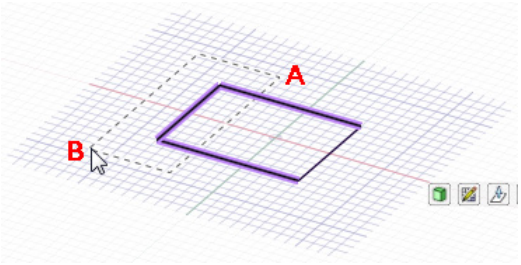


- Images can now be selected using box select in the Design window of a drawing sheet.
- Objects are now highlighted in the Structure tree when you move your mouse over them in the Design window. The lowest visible parent body or component of the pre-highlighted object is highlighted when the parent component is collapsed in the Structure tree. Primary selected objects are highlighted in blue, and secondary objects are highlighted in orange, as shown on the right.



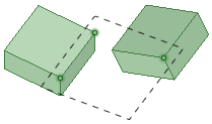
- Now when you draw a box to select objects in 2D mode, the direction that you draw the box is determined by the grid, rather than the orientation of the view. The examples below show box-selection from A to B, and the lines that would be selected are highlighted. Previously, the boxes below would have resulted in the same selection because they are both right-to-left selections *in the screen plane*.



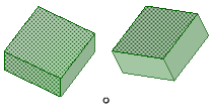


Select temporary objects

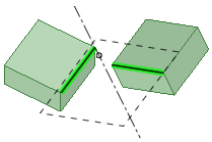
- You can now select a face or edge and use Ctrl+A to select all faces or edges of the same solid or surface.Temporary objects have been enhanced so you can
  - Select three points to make a plane.



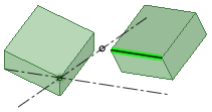
  - Select three faces or surfaces to make a point.



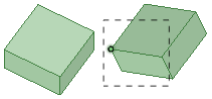
  - Select two skewed edges or axes to make an axis between the two points at the closest approach between each other, and a point and a midplane at the middle distance along that axis.



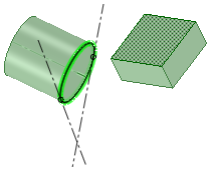
  - Select an edge and a point that is not in the line to make one axis from the point normal to the edge and a second axis parallel to the edge through the point.



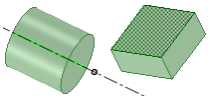
  - Select a point to make a plane that is normal to the view direction (or parallel to the plane of the screen).



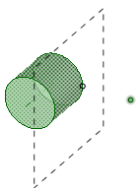
  - Select a curve and an intersecting face or surface to make points where the face would intersect the curve and axes that pass through the points, are tangent to the curve, and are on the same plane as the curve. One point and axis are created if the face would only intersect the curve at one point.



  - Select an axis and an intersecting face or surface to make a point at their intersection.



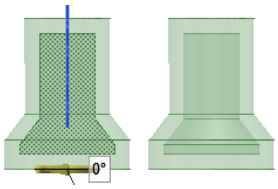
  - Select a point and a face or surface to make a plane that is tangent to the face and a point normal to the axis dropped normal to the face from the point.



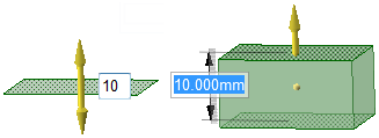
Designing and editing

Pull – General

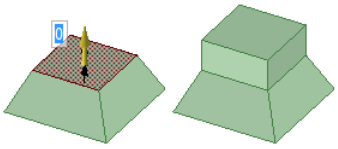
- You can now revolve a surface (to make a solid or a cut) that is on both sides of the revolve axis and inside the solid, as shown on the right. Pulling edges that are connected now produces the same results, whether the curves are smooth or sharp. Previously, sharp curves or corners could create multiple surfaces or no surfaces at all.



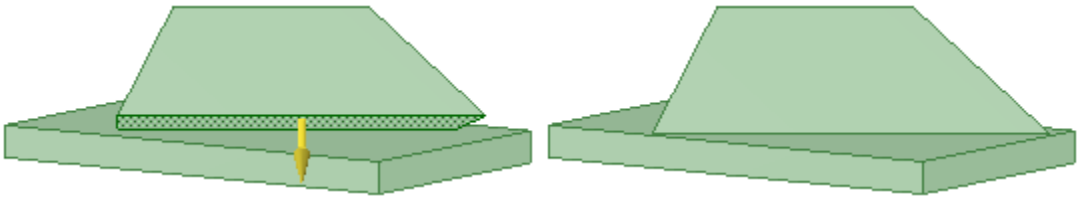
- When you select the Both Sides option, select a surface, then press the spacebar and enter a value, the value is the overall distance rather than the distance for each side, as shown on the right.



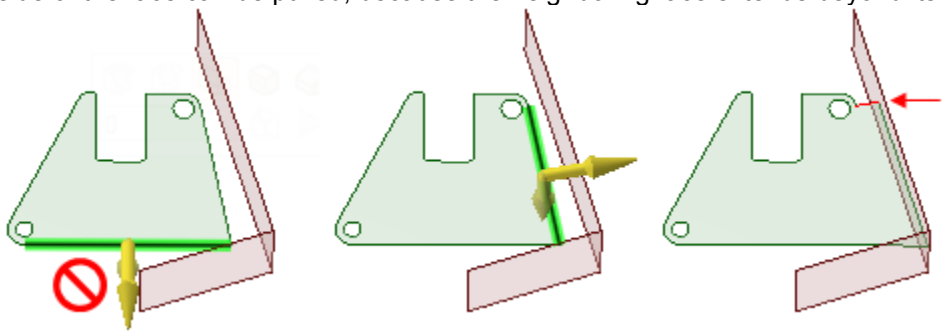
- If you copy and paste a solid face to make a surface and Pull it, it is no longer influenced by the solid neighbors of the source face. Previously, as shown in the example on the right, where the top face was copied and pasted as a surface, the surface would have been pulled into a pyramid shape.



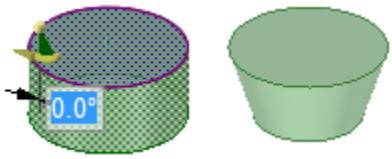
- You can now Pull with the Up To tool guide and the side influence of adjacent faces is correctly handled, as shown below. The Up To tool guide also works in more cases.



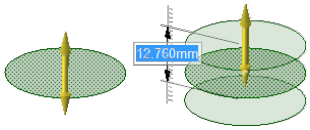
- Pull with the Up To tool guide no longer leaves the original edges behind.
- An offset dimension is now displayed when you pull a conical face and when creating a surface offset.
- You can now pull surface edges up to other objects. This is the natural extension for these edges, so the function is built into the Full Pull option for the Pull tool.
    - The faces you pull must be fully bound by a neighboring surface or face. If you select the lower edge of the green surface shown below, you will receive an error because the neighboring surface does not extend past the end of the selected edge. The edge on the right side of the face can be pulled, because the neighboring face extends beyond its length:



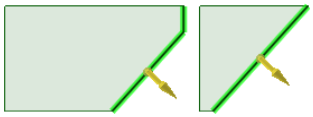
    - The original surfaces that the edges belong to are extended and new edges may be created; however, new faces are *not* created. The short edge highlighted in red in the image above was created when an edge is pulled with the Full Pull option.
- When you Pull with the Cut option using surface that is *inside* a body, a new solid is no longer created in the Structure tree.
- Now when you select a face, then Alt+select a non-tangent neighboring face, the Draft tool guide is automatically activated, as shown on the right. The faces do not need to be at a 90° angle to each other, so you can detect and adjust the draft angle on imported parts. Previously, you had to remove and reapply drafted faces to make a change to them.



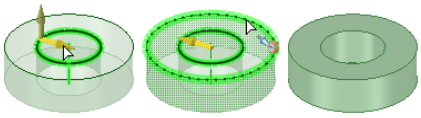
- You can now use annotation dimensions with Move or Pull to drive dimension changes for offset cylindrical faces.
- Now when you use the Pull tool with the Both Sides option and hold the Ctrl key to pull a surface, an offset copy is created on both sides, as shown on the right.



- Pulling edges now works even when an edge is removed during the pull, as shown in the right.



- You can now edit a circular slot by changing its angular dimension. Previously, the dimension field was not active until you pressed the spacebar, and the length indicator lines were not displayed when you changed the angle by typing a number.
- When you modify round faces they keep their layer membership, display state, color, name, and annotations.
- You can now pull circular edges radially, using the Up To tool guide, as shown on the right.

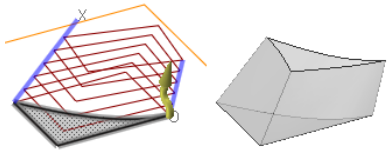


Blend

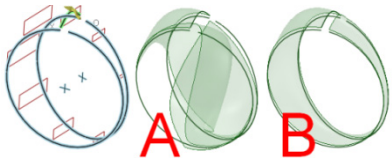
- You can now blend with the No Merge option and the result will not be merged with existing bodies.
- You can now blend between complex curves imported from Rhino.
- The new Ruled Surfaces option for blends now works when you select solid faces. When you use this option with faces, the blend is created as if you had all the edges of the face selected. Selecting edges (the manual override for the default tangency condition) will have no effect, because the option is essentially forcing all the edges to be formed non-tangent.
- The Pull tool now activates the Blend tool guide when you select profiles at both ends of a sweep trajectory.

Sweep

- You can now sweep a surface face and a loop of surface edges together along multiple trajectories to make a variable section sweep. Sweeping a loop of edges is shown on the right.

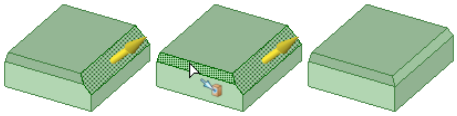


- Periodic, or nearly periodic, sweeps now create correct geometry. In the image on the right, figure A shows the sweep before and figure B shows the sweep after the fix.



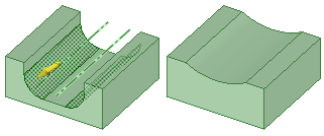
Chamfers

- Now when you pull a chamfer with the Up To tool guide, the size of the chamfer will change to the same size as the chamfer you select for the tool guide, as shown on the right. This change makes chamfering behave the same as Up To for rounds and cylinder diameters.

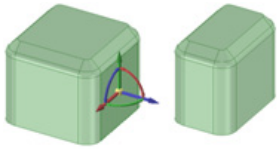


Rounds

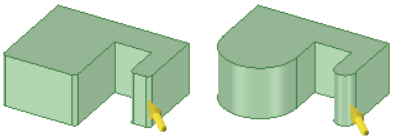
- Now when you Pull a pair of symmetrical round faces so they intersect, the intersection line remains symmetrical, as shown on the right. Previously, the intersection line was not centered between the faces for inside pockets rounding.



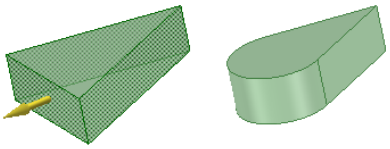
- You can now change a part that has a chamfer with a round edge and the rounded edge will no longer be lost, as shown on the right.
- You can now use the same technique to create a full round on the end of a cylinder as you do on other edges.



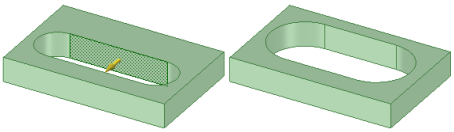
- The maximum radius for rounds on circular edges has been removed so you can make a rounded edge with as large a radius as you want.



- You can now create two or more full rounds by selecting multiple edge pairs, as shown on the right.
- You can now create a full round between multiple sets of three or more faces at the same time. Select sets of three or more faces, then right-click and select Full Round. The first face you select becomes the center of the round, as shown on the right.



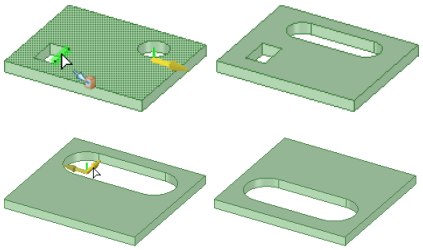
- Full rounds are now updated when you move adjacent faces, such as when you remove the association of a pulled slot and then move one of the side faces, as shown on the right. Previously, the face was moved but the radius of the full round was not increased.



- When you round more than one edge, and one or more of the rounds cannot be created, the successful rounds are created. Previously, all rounds would fail.

Holes and slots

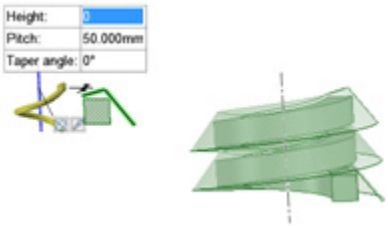
- You can now edit a circular slot dimension and the dimension field is no longer displayed as an uneditable field. Uneditable dimensions are displayed in parenthesis. You can now type a value to change the radius of a circular slot.
- You can now pull a slot up to a reference object using the Up To tool guide, as shown on the right.



- Now when you pull a slot axis perpendicular to the orientation of the slot, the slot will move in the direction you pull, as shown on the right.
- When you select a circular slot and Alt+select a circular face, and use the Pull tool with Ctrl to copy the slot radially, the dimensions are now displayed near the slot.
- The selection of slot axes is no longer blocked by the Pull handles.

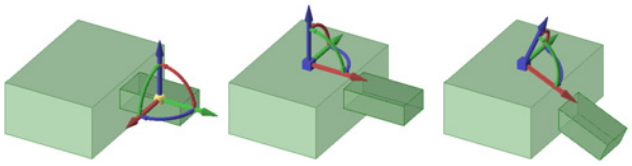
Helices

- You can now revolve a helix by selecting a combination of a face or surface and a curve, as shown on the right.

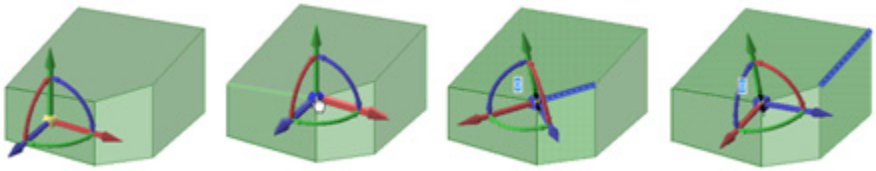


Move

- You can now use the Move tool with the Ctrl key to make a copy of a set of faces that include a face that is tangent to the edge of another face that is not copied.
- You can now create patterns of planes, origins, and axes.
- You cannot create patterns of threads. This capability has been explicitly disallowed.
- If you anchor the Move tool and move an object, the Move handle now remains aligned with the object, as shown on the right.



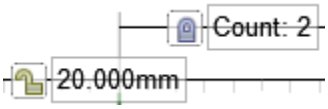
- You can now position the Move handle multiple times and the move handle will adjust to each position without overriding the previous position, unless it has to. You can combine repositioning selections to get exactly what you want.



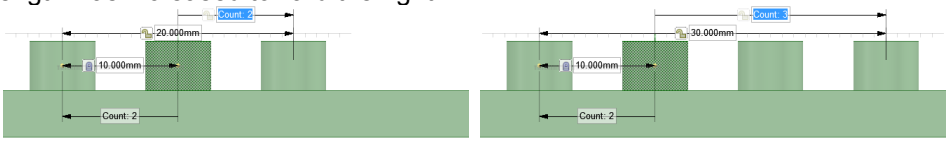
- The Ctrl key can now be used to make multiple copies of an object with the Move tool's Up To tool guide. You can also double-click the Up To tool guide to make it sticky so you can make multiple copies of an object *without* holding the Ctrl key. The tool guide icon has a double border when it is sticky, as shown on the right.

Patterns

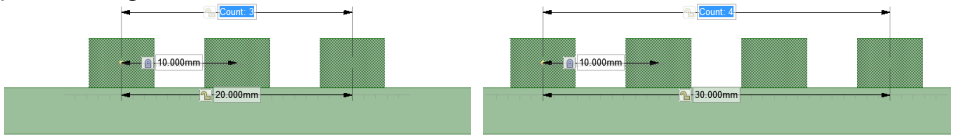
- Pattern dimension display has been improved to help you see the relationships and values for a pattern:
  - A pattern's dimensions now have icons to show which dimensions will be changed if you change the value of a highlighted field. The closed lock indicates that a dimension won't change and the open lock shows you that it will change based on its intrinsic relationship to the other, selected dimension.



  - You can override a lock on a dimension by clicking its icon to toggle it between locked and unlocked.
  - Dimension line placement, arrows, and dimension fields show you how the pattern will change if you change the value of a field.
- Your selection determines how changes to the distance and count affect the pattern's position:
  - If you select one member of the pattern, the change is centered on that member. For example, if you select a member of a pattern, you will see count values for both sides of the selected member. Changing one of these values will add or remove a member on that side. In the example below, we changed the highlighted field from 2 to 3, and the overall pattern length was increased toward the right:

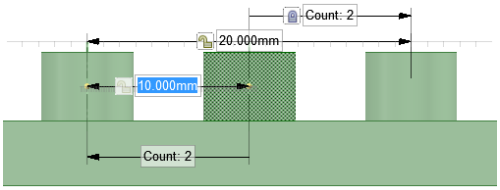


  - If you select all pattern members, the change is centered on the entire pattern, with the distance between members locked by default. For example, if you select all the pattern members and change the count, the pattern will shift so its center remains in the same location. In the example below, we changed the pattern count from 3 to 4, and the overall pattern length was increased from its center:



- You can control which direction the pattern will grow by selecting an internal pattern member:
  - Previously, the pattern change was centered on the member you selected.

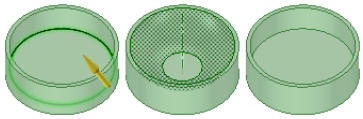
- Now you will see pattern count dimensions on either side of the pattern member, as shown on the right.
- Changing a pattern count in one direction does not change the distance between pattern members. This is indicated by the closed lock icon. Instead, the overall distance of the pattern will change.



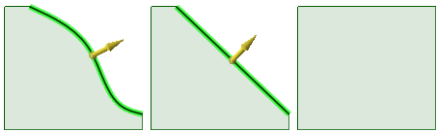
- You can now create and modify patterns of components that contain only sketch curves.
- You can now press the tab key and spacebar to dimension patterns while dragging around a rotation axis or the Move tool axis.

Fill

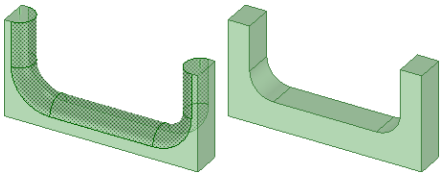
- Fill now works on face-to-face rounds between cylindrical faces. In the example on the right, we created a round between two cylindrical faces, and then filled it. Previously, Fill would have filled the entire bowl-shaped space to create a solid cylinder.



- When you use the Fill tool on a laminar edge, the edge is simplified if possible. If the edge is already straight (or simplified), then its neighboring edges are extended (or trimmed back). In the example on the right, you would first click Fill to simplify the edge, then click Fill again to extend the neighboring faces.



- Full rounds on U-shaped faces can now be removed with the Fill tool, as shown on the right.



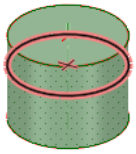
- Inside edges on a part are no longer removed when you use the Fill tool to simplify an outside edge.

Replace

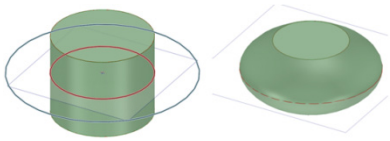
- The workflow for the Replace tool has been changed so it automatically advances, like the Split Solid tool. To use this tool,
  1. Click Replace.
  2. Select the face you want to replace. Hold the Ctrl key to select multiple faces.
  3. Select the face you want to replace with. You can click the Source tool guide to stick it and select more than one face.
  4. (Optional) To add faces that you want to replace, click the Target tool guide or hold Ctrl and select additional faces.
  5. Click the Complete tool guide to replace the faces.
- The face you have selected will now remain selected when you exit the Replace tool by clicking on another tool, and you can press the Delete key to remove the selected faces.
- You can now pre-select faces to be replaced before you start the Replace tool and you can box-select faces in the tool.
- Your selection is now cleared in the Replace tool when you click in empty space in the Design window.
- You can now select a plane object in the Structure tree when you use the Replace tool.

Edit as Blend

- Blend section curves are now highlighted in red so they are easier to see against the highlighted blend faces, as shown on the right.



- The Replace tool guide now allows you to replace an existing section curve with another curve on the same plane, as shown on the right.



Intersect tools

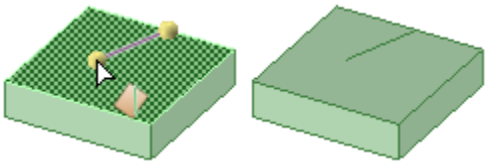
- You can now create and select temporary geometry in the Split Body and Combine tools.

Combine

- When you use the Combine tool, faces that are not combined are no longer colored as if they succeeded.

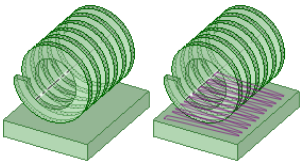
Split face

- You can now use Split Face with a plane from a different component.
- When you use the Two Cutter Points tool guide in Split Face, the second point is previewed at the cursor position instead of only on another edge (as before). That means you can select a point on an edge (to make completely-crossing edges) or any other point on a face (to make incomplete edges), as shown on the right. Previously, you could only select positions on two edges so no preview could have been shown until the second point was pre-highlighted.



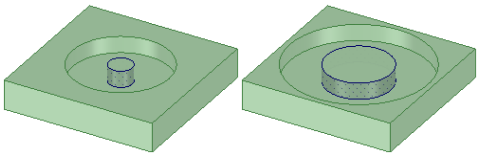
Project to solid

- You can now project helical faces onto a surface, as shown on the right.



Insert tools

- You can now select a point and a plane to place an axis through the point and normal (perpendicular) to the plane.
- When you cut and paste an object, the object retains its color and face style.
- You can now use the Offset tool to create an offset relationship between two cylinder faces in the same body when the cylinders are facing, as shown on the right.



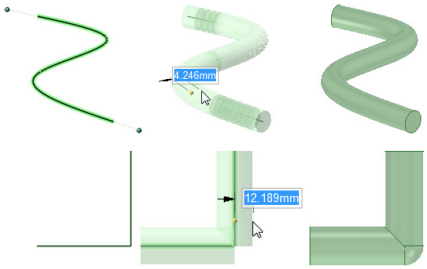
- When you select all of the offset faces of a shelled body, you can now change the Thickness property. Previously, changing the property didn't work and you could only change the thickness in the right-click menu.

Face Curve

- Sketch curves are now previewed when you add points, but the curve is not applied to the design until you click the Complete tool guide or press the Enter key.
- Now when you double-click or close a curve to finish adding a face curve, you will automatically go into curve editing mode with the curve and its points highlighted. The curve is applied to the model after you finish editing.
- You can now drag and delete points, and you can snap to other objects when you edit a sketch curve.
- The Periodic option is now available for the Face Curve tool.
- When you are finished drawing a face curve, Undo now takes you back to editing the curve.
- You can now make points on edges with the Face Curve tool.

- You can now delete points you make with the Face Curve tool by right-clicking and selecting Delete.
- You can now click Undo to remove a point you added with the Face Curve tool.
- You can now draw a face curve on a cylindrical face using the Face Curve tool.

Sphere and Cylinder tools



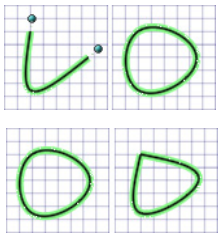
- You can now select 3D curves or edges and create a swept pipe with the Cylinder tool. Select a 3D curve or edges, or an edge chain, click Cylinder, and drag to set the cylinder diameter. A swept solid with a circular cross section is created along the curves or edges, as shown on the right.
  - If the curves or edges include angles, a spherical surface is added at the angle, as shown on the right.
  - If you use the Fill tool on the spherical joints, the corners are converted to sharp corners.
- The Sphere tool now has a No Merge option in the Properties panel.

Images and movies

- You can now cut, copy, and paste images and movies within your design. When you paste an image or movie, you can select the planar face where it will be placed.
- You can now place an image or movie on a datum plane, as shown on the right.
- Inserted images and movies will no longer be highlighted when you mouse over them if you select Location and Size for the Lock property. The image or video will no longer appear to flash as it is highlighted when you to trace over an image in Sketch mode.

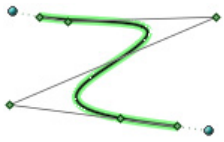
Sketch

- You can now snap to the center of a polygon.
- When you copy and paste a sketch curve, the curve is pasted in its original location. Previously, the curve was offset from its original location when it was pasted.
- Now when you use the Move tool on an object in Sketch mode you can use the Orient to Object tool guide.
- You can once again project an edge from one solid onto another solid. You can also select the object to project from in the Structure tree, and all the edges of the object will be projected.
- Sketched spline curves have a new Periodic property which controls the tangencies of end points:



  - If the property is set to True, the tangencies of the beginning and end of the spline curve will match to create a closed curve. If a spline curve is open and you set the property to True, the spline will be closed, as shown on the right.
  - If the property is set to False, the end tangencies of a closed curve don't match. If a spline curve is closed and you set the property to False, the curve will become teardrop-shaped because the end points will no longer be tangent, as shown on the right.
- Curves to be trimmed with the Trim Away tool are now easier to select. Previously, some curves had to be clicked several times to select them, and they were not pre-highlighted when you hover over them with the mouse.

- You can now edit a spline curve by moving its spline points or by moving its control points with the Move or Select tools while in 2D mode. Spline points are the actual points along the spline; control points are new, and are found outside of the spline and connected by dotted lines, as shown on the right. Both types of points can be selected with the Move or Select tools.



  - Control points are on top of the spline to make them easier to select and edit.
  - When you move control points, the points are not deslected after you drag them.

Scale tool

- You can now select spline points with which to use the Scale tool. Use this to scale part of a spline curve using the scale box.

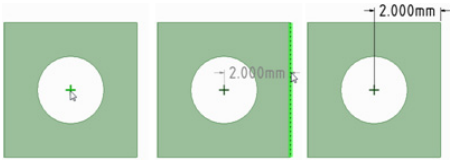
Detailing

General Detailing

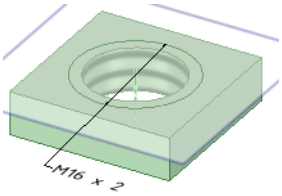
- You can now select the text belonging to a surface finish symbol to change its layer.

Dimension

- You can now click and drag to create the center, inner, and outer dimensions of a toroidal surface. Previously, you could only click to start, then click again to place the dimension.
- It is now easier to select the center of a circle while adding dimensions to a design, as shown on the right.



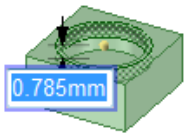
- Now when you create a diameter dimension to a thread in the ISO, JIS and CB standards, the text for the thread callout annotation displays according to standard.
- A new Show thread designation in thread diameter option has been added to SpaceClaim options, on the Detailing page. The choices for this option are:



  - **Never**: Show the diameter and not the thread designation.
  - **Always**: Show the thread designation instead of the diameter.
  - **Per thread table**: Use the value for `ShowThreadDesignationInDiameterDimensions` in the thread table XML file. The value can be set to true or false.
  - **If units match**: Show the thread designation if the system units match the units from the thread table.
  - **Per thread table if units match**: Show the thread designation if the units match and if the `ShowThreadDesignationInDiameterDimensions` value is true.
- A new property Show Thread Designation has been added for thread diameter dimensions, and you can set the value to Yes, No, or Default. Select the thread dimension field to change this value.

Threads

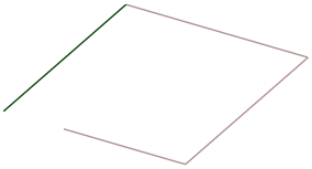
- You can now change the depth of a thread when it is part of a pattern.
- Thread properties are now displayed when you select a thread edge or face in the Design window.
- The depth of a threaded blind hole on a curved surface is now defined as the measured distance from the end of the thread to the closest point on the outer contour of the thread.



- You can now change the thread depth of a blind hole in a dimension field in the Design window, as shown on the right.
- Thread properties now include fields for the Major Diameter and the Minor Diameter.
- Thread depth dimensions are now displayed closer to the threaded face so they no longer fly out of the viewing area when you rotate the model.
- You cannot create patterns of threads. This capability has been explicitly disallowed.

Display

Colors and settings



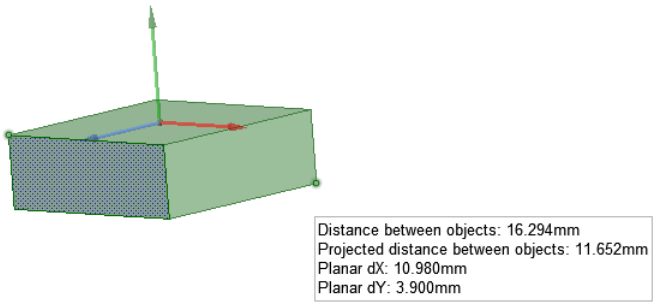
- When you add a Custom Color in the color palette, it is now remembered between SpaceClaim sessions.
- Now when you change the color of a curve, the curve is displayed in that color. Previously, the curve was displayed in black unless you selected it. In the image , three lines have been changed to red and one was left at the default color.
- A table of user-defined colors is now displayed at the bottom of the Color menu.

Layers

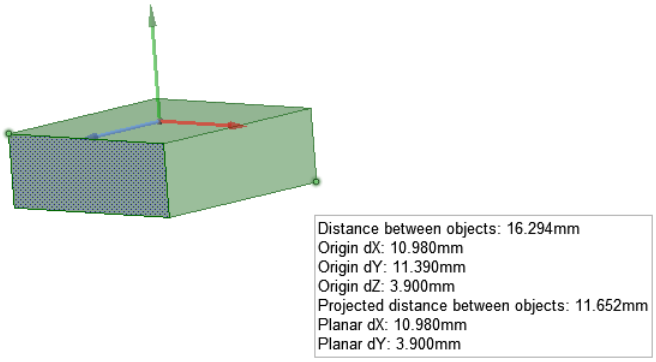
- You can now select an object, then open the Layers panel and click a layer to move the object to that layer.
- New layers now appear 1200% faster when you create them or turn their visibility off by clicking the light bulb to off.

Measurement

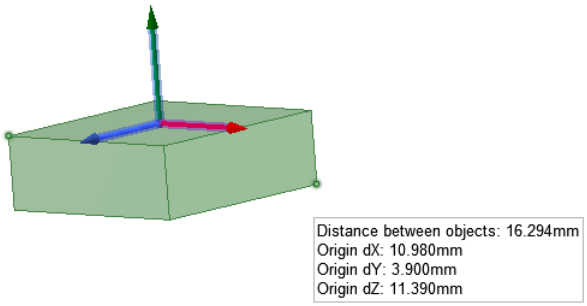
- You can now view the results of using the Measure tool in Cartesian coordinates when you select the new Enable XYZ Measurements option. Your selection must include one plane or origin axis, and you must Alt+select one plane or origin axis, origin, or global origin. The projected distance and dX, dY, and dZ values are reported. The distance from and to is determined by the order in which you select objects. This example shows two points selected and one face Alt-selected, so the projected distance is appended to the measurement.



- This example shows the same measurement as above, with the Enable XYZ Measurement option selected, so the distance line is reported in XYZ world coordinates.



- This example shows the measurement with the the Enable XYZ Measurement option selected and the origin on the top of the block Alt-selected, so the XYZ coordinates from the origin are reported.



Interference tools

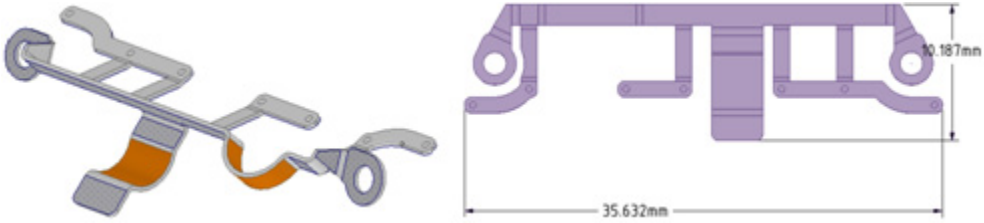
- You no longer need to hold the Ctrl key to select multiple faces with the Volume Interference tool. You can select multiple faces by default, and use the Complete tool guide to calculate the result.
- The Volumes tool no longer reports interference for thread intersections.

Quality tools

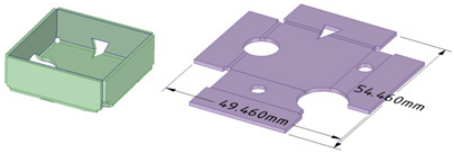
- The Dihedral tool now scales the fringe graph relative to the Design window when you select an edge, and all further selections are scaled accordingly. Previously, the scale was normalized for curvature on faces, but not for edges, which had their curvature calculation done individually.

Sheet metal

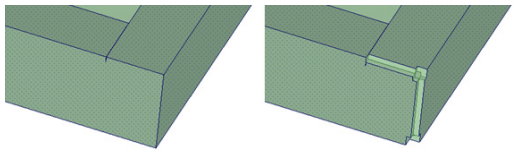
- You can now unfold a sheet metal part with rolled bends. The rolled bend faces are highlighted in orange on the unfolded part in the image below.



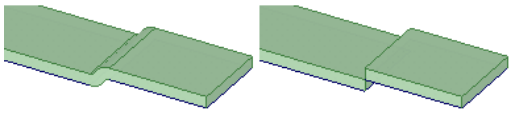
- Sheet metal bends that are intersected by multiple cut-outs that cross over the bends will now unfold correctly, as shown on the right.



- The default view for unfolded Sheet Metal parts is now the Plan view, like a drawing sheet, rather than a trimetric view that is used for the Home view.
- Rip reliefs that you create with the Split Face tool are now wide enough and deep enough to accommodate bend junctions, as shown on the right. This was correct with the 2008 release, but not with 2009 SP0.



- You can now convert an S junction into a no-overlap junction, as shown on the right. Click Pull, select an edge of the junction and click No Overlap in the Options panel.

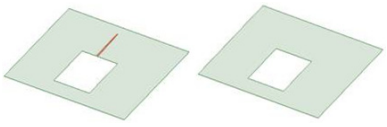


- The Rip Relief parameter on the Sheet Metal page of the SpaceClaim Options now works correctly. Previously, setting this option had no effect.
- The Split Face tool has been improved for sheet metal parts:
  - If you select the face of a sheet metal part, the Two Cutter Points tool guide is active by default.
  - If you select the side, or thickness, of a sheet metal part, all tool guides are enabled.
  - A rip relief is created when adjacent faces don't meet the split face at a 90° angle.
  - The Rip relief Width value in the SpaceClaim options Sheet Metal page is now used correctly.
  - You can now use the Select Perpendicular Cutter Point tool guide for sheet metal parts.
  - The ruler is no longer displayed for the second selection when you use the Two Cutter Points tool guide.

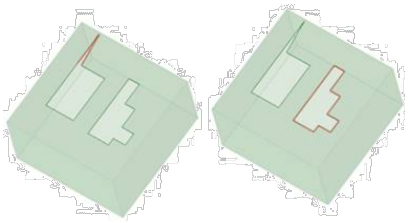
Prepare tools

Repair

- The Fix Gaps tool now fills only the edges that form a gap when they are adjacent to a larger hole, as shown on the right.



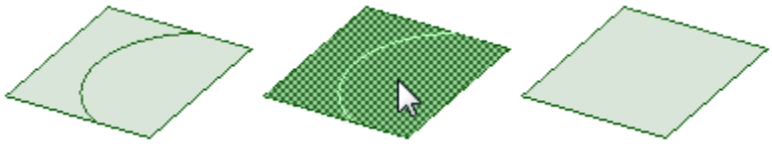
- The Fix Gaps tool now only works for edges that are paired. Paired edges are edges that are within the maximum distance along their length or that share an end point and are within the maximum angle you set in the tool's options. Use the Missing Faces tool if you need to repair a part with edges that are not paired. Problems found by Fix Gaps is shown on the left in the image on the right, and problems found by Missing faces is shown on the right.



- A new Simplify tool has been added to the Prepare tab. This tool examines a design and simplifies complex faces and curves into planes, cones, cylinders, lines, arcs, etc. This automates the one-by-one "Simplify" capability found in the Replace tool.

Adjust

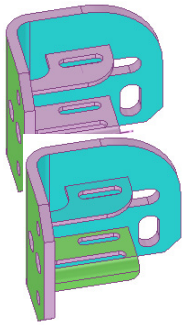
- New Stitch tool combines faces that are touching at their edges.



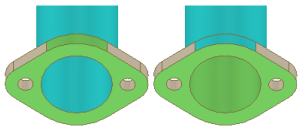
- The new Stitch tool repairs an object when you select it, rather than when you click Complete.

Define

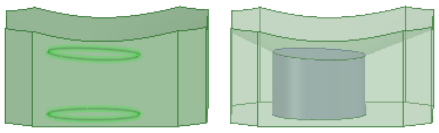
- New Midsurface tool on the Prepare tab creates a surface at the midpoint between two faces. You can use these surfaces for FE analysis.
- The Midsurface tool shows the thickness of the original model face offsets as a property named Thickness in the Midsurface section of the Properties panel. It is a face property, so you must select the face in the Design window rather than in the Structure tree, even if it is a single face.
- The Use Selected Faces option lets you use the selected faces to determine the offset distance. All face pairs with the same offset distance will automatically be selected.
- Color highlighting shows you face pairs that have been selected, as shown in the image on the right. The midsurface face will be offset from the cyan faces. Green indicates that a face is offset from a cyan face. Unselected faces and faces without offsets are shown in the original color (which is purple in the image).



- You can hold Ctrl and click on a face, then its offset, to select additional face pairs that have a different offset distance. Again, all face pairs with the same offset distance will be automatically selected, as shown in the image on the right.
- You can use Add/Remove Faces tool guide with the Midsurface tool to select additional faces to offset or remove detected face pairs from the selection.
- You can use Swap Sides tool guide with the Midsurface tool to switch the face pairs. You may need to do this when you detect pairs with more than one offset distance, and the offset relationships are incorrectly detected, as shown on the right.



- If the Midsurface tool fails to create mid-surface parts, the problem faces or edges are highlighted.
- The Volume Extract tool can now do a patch blend when the primitive fill fails. This makes it possible to create a volume when the boundary faces are not planar, as shown on the right.



Importing and exporting

General

- Allow import of hidden components has been renamed to Allow import of hidden components/geometry in the General page of the File Options dialog.
- New Allow import of free curves option has been added to the General page of the File Options dialog. This option is turned on (checked) by default. Previously, free curve entities were ignored when you imported certain types of files.
- When you save copies of external components, you can now select more than one component and choose a new folder to save the components. Previously this was only available for one component at a time with the References button on the Save As dialog box.
- When you import a file and it fails, the reason for the failure is now reported in the Status Log.
- The Stop button is now enabled for all imports.
- The name of an exported file is now displayed in the Status Log in the status bar.

Rhino and MoI

- You can now copy from Rhino or MoI (Moment of Inspiration) and paste directly into SpaceClaim. Simply copy an object in Rhino or MoI, then open SpaceClaim and click Paste.
    - Objects are placed in their original position and will not be placed relative to selected objects in SpaceClaim.
    - Objects from MoI do not have their units defined by default and could be scaled very differently when you paste them into SpaceClaim. To change the units in MoI, click Options, then set the Unit System field.

Pro/ENGINEER and NX

- A new Pro/ENGINEER page has been added to the File options. This page includes the option Allow import of Quilts. A Quilt in Pro/ENGINEER is roughly equivalent to a Surface body in SC, and is a collection of individual surface faces.
- When you import Pro/ENGINEER and NX assemblies and parts are missing, you will now be prompted to search for the missing files.

AutoCAD

- You can now import polyface meshes from AutoCAD files as 3D solids. To find this option, click SpaceClaim Options, open the Popular page, and click File Options. You can also click File Open, then click Options. The option is under *Insert polyface meshes into 3D as*.
- When you import AutoCAD files you can either select the Try to infer Model Space units option, or you can set the units of an imported AutoCAD file explicitly.
- You can now import DXF files with certain types of spline curves that were not previously supported.
- Text is now scaled correctly when you import a DWG file.

3D PDF

- You can now export 3D PDF with precise B-Rep geometry.
- When you export a PDF, the Allow export of hidden components option now works correctly. This option was previously ignored.

- When you import a PDF, the Allow import of hidden components and geometry option now works correctly. This option was previously ignored.
- A progress bar is now displayed when you import or export a PDF file.
- Adobe Acrobat® 9.1 Pro Extended can be used to export a SC file, and re-imported to Spaceclaim. Acrobat 9.0 must be used to export 3D PDF files from other applications, in order to successfully import the files into SpaceClaim in SP1.

STL

- STL files can now be imported as lightweight objects.

IDF

- When an IDF file is imported, a library of external documents used in the IDF assembly is created. Previously, new files were created even if they already existed. Now, if the library folder already exists and it contains files with the same names as those that would be created, the existing files are used. These files may exist if you previously imported the IDF data, or if different board layouts share the same components.
- The library folder name has been changed from Library to <filename> ECAD Library, where <filename> is the name of the IDF file. Each imported file must have its own library. SpaceClaim does not support a shared library for IDF files.

CATIA

- You can now import CATIA (.cgr) file formats into SpaceClaim as lightweight objects.
- You can now export CATIA V5 R19. This option is in the CATIA section of the File options dialog.

XAML

- Now when you save a design as XAML, the orientation and translation of the current view is saved.

SolidWorks

- You can now import SolidWorks version 2009.

Bunkspeed

- You can now save a design as a BIP file, for use by BunkSpeed rendering.

SpaceClaim API (V5-Beta 2)

Export

- The `DocumentExportFormat` enum has a new `Pdf` (3D PDF) value. The file is written in either Universal 3D format (containing facets) or PRC format (containing a B-Rep). This is controlled by a user setting.
- The `WindowExportFormat` enum has a new `Bip` (Hypershot scene) value.

Sheet Metal

- `Part.Unfold` has been added, which allows you to unfold a sheet metal part about an anchor face.
- `Part.FoldedSheetMetalPart` has been added, which allows you to obtain the folded part for an unfolded part.

- `IPart.SheetMetalPartType` tells you whether a part is a folded or unfolded sheet metal part.

Components

- `Component` now implements `IHasName`, which allows you to get or set the name of the component.

Design Bodies

- `DesignBody.Unite`, `DesignBody.Subtract`, and `DesignBody.Intersect` have been retired. `Body.Unite`, `Body.Subtract`, and `Body.Intersect` should be used instead.
- `DesignBody.GetDesignBody` has been added, which allows you to get the `DesignBody` that has the given `Body` as its `Shape`.

Modeler Bodies

- `Body.Unite`, `Body.Subtract`, and `Body.Intersect` have been added.
- `Body` now implements `IDisposable` and `ITransformable`.
- `Body.PieceCount` and `Body.SeparatePieces` have been added.
- `Body.ExtrudeProfile`, `Body.SweepProfile`, and `Body.CreatePlanarBody` now throw exceptions if the profile or path is invalid, rather than returning null. These methods now correctly handle profiles of more than one outer loop.
- `Body.LoftProfiles` has been added. This creates a body by lofting through open or closed profiles.
- `Body.RoundEdges` has been added. This creates fixed radius or variable radius edge rounds.
- `Face.Loops` now includes degenerate loops (e.g. the apex of a cone). A degenerate loop has no fins and a single vertex. `Loop.Vertices` has been added to provide access to this vertex.
- `Edge.Precision` has been added.

Geometry

- A new override of `CurveSegment.Create` has been added, which creates a curve segment for the entirety of a curve. This is useful for creating a complete circle or ellipse, since there is no need to supply a parametric interval.
- `Surface.IntersectCurve` and `ITrimmedSurface.IntersectCurve` have been added. These return the intersections between a trimmed curve and a surface or face, respectively.
- `Interval` now provides `GetProportion` and `GetParameter` to convert between a curve parameter and the proportion of the interval. The indexer is equivalent to `GetParameter`, so you can get the mid-parameter simply as `interval[0.5]`.