SpaceClaim 2008 SP1

Release Notes

A **SPACE**CLAIM Document

SPACECLAIM
CORPORATION

These release notes cover what's new in this release and the issues resolved in this release.

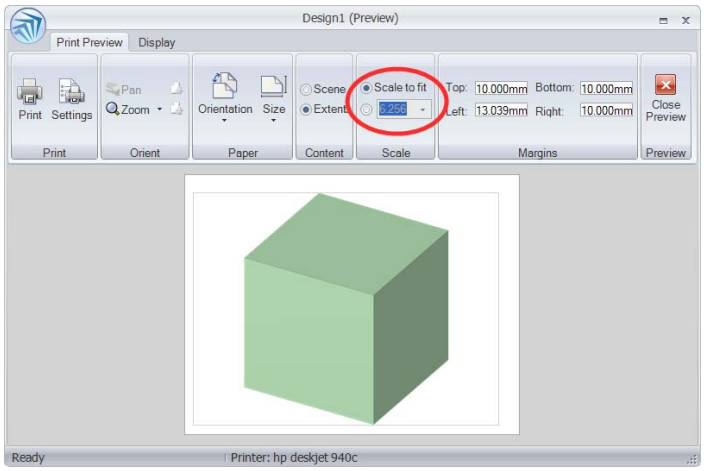
What's new

The following features have been added to SpaceClaim 2008 SP1.

General enhancements

The following general enhancements were made to the SpaceClaim application:

- The SpaceClaim user interface is available in Italian, Japanese, German, French, Korean, Chinese Simplified, and Chinese Traditional languages. SpaceClaim 2008 Help is available in French, German, Italian, and Japanese.
- SCDOC files display a thumbnail preview of your design in Windows Explorer on XP (SP2 and SP3), Vista, and Vista SP1.
- You can Shift+click multiple objects within a component in the Structure tree to select all the objects between them, whether or not they are visible in the Design window.
- You can use the Next and Previous Window controls on the status bar to navigate to Design windows that are hidden because too many documents are open.
- Press F11 in a release build to maximize the Design window. Press F11 again to return to the normal application window.
- You can switch seamlessly between spinning, panning, and zooming while dragging the middle mouse button and pressing Ctrl (to zoom) or Shift (to pan). Previously, you had to release the middle button and re-drag to switch.
- When you select Scale to Fit and Extents in the Print Preview window, the current scale is shown disabled in the scale value input drop-down, as shown in the figure on the right.
- If you do not have a printer installed on your machine, print options are disabled in SpaceClaim.



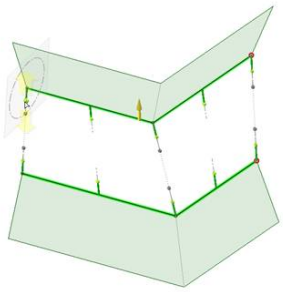
Select enhancements

The following enhancements were made to the Select tool:

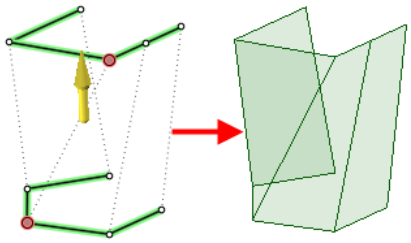
- When you edit an arc's radius with the Select tool so that it is tangent to another arc, the arc you are changing maintains its chord angle.
- You can move a sphere in Section mode by dragging its center with the Select tool.
- Double-click to select a set of sketch lines with tangent connections. Double-click again to cycle through alternate loops. Triple-click to select all connected lines (that is, the entire sketch).

Pull enhancements

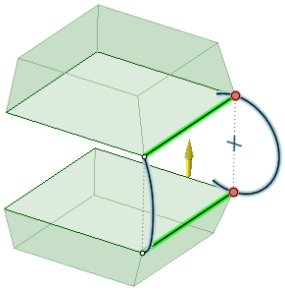
The following enhancements were made to the Pull tool:



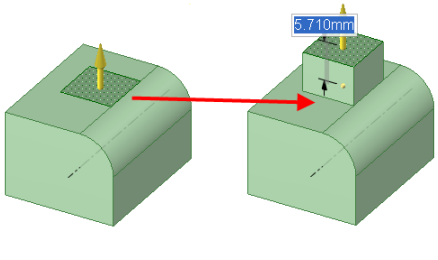
- When you select the edge of a blend, the blend take-off vector handles are attached and perpendicular to the edge, and tangent to the face that "owns" the edge, as shown in the figure on the right. You can adjust the length of a blend's take-off vector to adjust the magnitude of the take-off (non-tangent) or tangential influence at that point. You can also adjust the plane of rotation of a blend take-off vector. When you Alt+click a neighboring face or edge, the take-off vector handles snap to the selected object. The handles expand as you zoom so that they are easier to interact with.



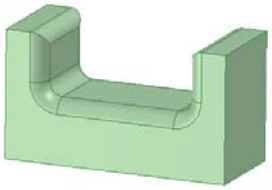
- You can blend between multiple lines in space (that is, profiles) with take-off vectors, even if the number of segments across the blend sections is not the same, as shown in the figure on the right.



- You can Alt+click internal guide lines for blends. (They used to have handles only on the outside edges of the sections.)
- Blend guides' endpoints do not have to be coincident with the endpoints of the blend sections, as shown in the figure on the right.



- You can select multiple surfaces and solids and pull them together to thicken the surfaces and offset the solids simultaneously. You can also pull a region with tangent edges to offset it, as shown in the figure on the right.
- When you Ctrl+Pull to copy a cylindrical face, the offset radius dimension is displayed.
- The offset dimension is shown as soon as an offset face with an offset relationship is selected with the Pull tool. Baseline faces do not show this value.



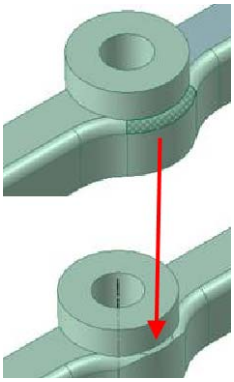
- Intersecting round faces meet with a torus instead of a sharp whenever possible, as shown in the figure on the right.
- The Right-Handed Helix option is selected by default when you select the Revolve Helix option.

Move enhancements

The following enhancement was made to the Move tool:

- You can move views on a drawing sheet with the Move tool.

Fill and Replace enhancements



The following enhancements were made to the Fill and Replace tools:

- You can Alt+click a round face with the Fill tool or use the Replace tool to replace a round that is difficult to remove, as shown in the figure on the right.
- When you select multiple faces of the surface to fill or replace, the entire surface is automatically highlighted to show that it will be used.
- You can select objects from the Structure tree to simplify them with the Fill tool.
- You can replace multiple faces or single faces with the Replace tool.

Intersect tool enhancements

The following enhancement was made to the Split Solid tool:

- When using the Split Solid tool, preview geometry appears as soon as you select the plane you want to split the solid with.

Other 3D tool enhancements

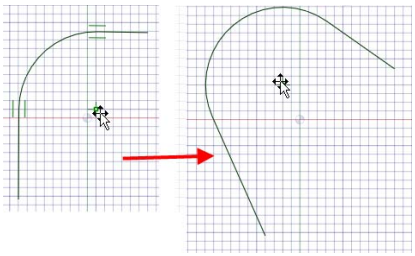
The following enhancements were made to the Plane tool:

- You can select a line sketched on a plane with the Plane tool to create a plane through the line and normal to the plane it was sketched on.
- You can select a point on a line and the line to create a plane with the Plane tool.

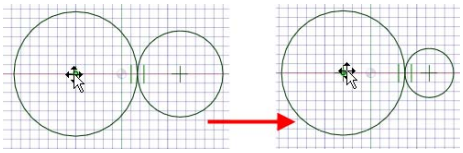
Sketch enhancements

The following enhancements were made to the Sketch tools:

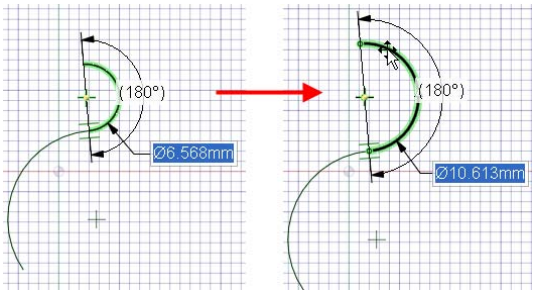
- When you drag the center of an arc created by the Create Rounded Corner tool, tangency is maintained, as shown in the figure on the right.



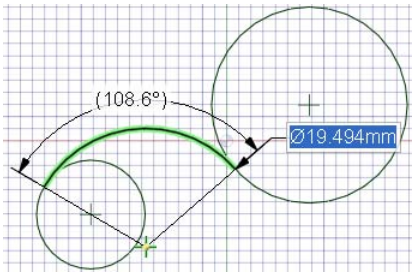
- When you drag the center of a circle that is tangent to another circle with the Select tool, the radius of the other circle changes to maintain tangency, as shown in the figure on the right.



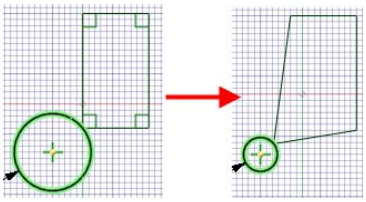
- When you drag a tangent arc, it maintains its chord angle, as shown in the figure on the right. (It used to maintain its original oriented start angle.
- When dragging the center of an arc, it is easier to stay on the dotted line of imaginary centers.



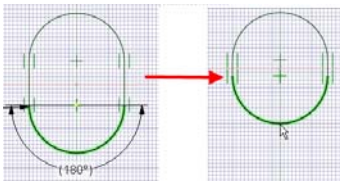
- You can drag an arc that is tangent on both ends (as shown in the figure on the right) to change its radius.
- If a sketch contains three tangent circles, and you drag one of the circles, the radii of the other two circles change to maintain tangency.



- When you drag a three-point circle drawn through a rectangle's vertex, the connection to the vertex is maintained as the radius changes. An example is shown in the figure on the right. If you drag the circle's center, the connection is also maintained.



- When you drag an arc with the Select tool, all connections in the sketch are maintained, as shown in the figure on the right.
- When you are sketching a line with the Line tool in Section mode with the Auto-extrude/revolve sketches in Section mode advanced SpaceClaim option enabled, if the line began on a solid, the line is finished automatically when you click another point on the solid.
- Polygon creation can have as many as 64 sides.
- Sketch dimensions can be edited if they highlight on mouseover.



Display enhancements

The following enhancements were made to the Display tools:

- Mirrored objects are created on the same layer as the original object. If you place the original object on a different layer, the mirrored object is automatically placed on that same layer.
- The Color By Layer icon displays the layer color.
- You can Ctrl+click multiple components with subcomponents in the Structure tree to set their color simultaneously.
- The Mini-toolbar option was removed from the Show ribbon group on the Display tab. You can adjust the visibility of the mini-toolbar by checking the Show mini-toolbar on selection option on the SpaceClaim Popular options page.

Annotation enhancements

The following enhancements were made to the Annotation tools:

- You can resize a table annotation by dragging its left side.
- You can box-select annotations on drawing sheets.
- When you create a new datum symbol with the Datum tool, the focus is placed inside the symbol so that you can enter text without needing to click the symbol first.
- The minor diameter for the insides of external threads was added to the thread tables, enhancing external thread creation automatically for the selected thread surface.
- You can press Esc to close the Geometric Tolerance Format tab.

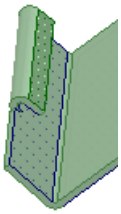
Drawing sheet enhancements

The following enhancements were made to the drawing sheet features:

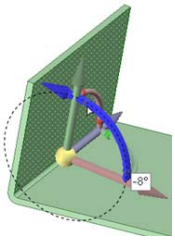
- When you print a drawing sheet, cross-section lines print at the default, thin lineweight.
- Lineweights and line styles are shown parallel to the plane of the drawing sheet.
- Performance was enhanced for modeling within a drawing sheet that contains multiple views.
- Line trimming of the extension lines was improved for the closed blank arrow head style for detail view boundaries.
- The JIS line thickness for the cross-section indicator for a cross-section view is thin by default.
- The sketch grid's intersection with the design only appears in the active cross-section drawing view.

Sheet metal enhancements

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



- You can create a partial hem in a sheet metal part, as shown in the figure on the right.
- You can make patterns using copies of flanges and notches on sheet metal components.
- If you change the thickness or inside bend radius properties for a sheet metal component, all the bend reliefs are updated.
- You can edit the radius of a single sheet metal bend by selecting the bend face and modifying the Inner Radius property in the Properties window.
- SpaceClaim can use the sheet metal bend deduction to calculate unfolded lengths.
- The Bend Allowance property contains the length of the arc through the bend at the neutral axis. Bend allowance and Bend deduction are linked. If you enter the value for one property, the other is calculated automatically.
- A new sheet metal bend table property was added for sheet metal components. To use a bend table, copy a CSV file to the SpaceClaim `Library/Bends` directory or into a SpaceClaim Support File directory. Set the bend table for a sheet metal component by selecting the component in the Structure tree and selecting the bend table from the Bend Table property. All values may then be obtained from the bend table instead of being calculated. Any values not obtained from the bend table may result in an error when you unfold the design. Once you select a bend table, you can edit the Vee Die Width property in the Properties panel. Vee die width sets the width of the tooling that produces the bend. You can clear the sheet metal bend table assignment from a component by selecting the blank value from the Bend Table property drop-down.

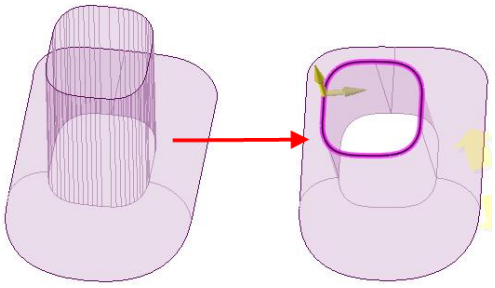


- When moving a sheet metal part in Section mode, the Move handle is aligned with the inside edges so that the Move dimension becomes the included angle, as shown in the figure on the right.

Importing and exporting enhancements

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

- You can open and insert ECAD files. When importing an ECAD file, first open the EMP file to generate a folder of library files in the same directory as the EMP file. Then open the corresponding EMN file to display the ECAD model in SpaceClaim.
- You can save a design as a Wavefront Object (OBJ) file. The current graphics tessellation is used for accuracy. You can modify the tessellation by setting the Image quality vs. graphics speed SpaceClaim option.
- You can open or insert 3D Rhino curves in SpaceClaim. When opening or inserting a Rhino file, multi-segmented curves are consolidated, as shown in the figure on the right.



- When you open or insert a Rhino file, object names are displayed, solids within a subcomponent that is nested over two levels down are displayed correctly, layer and material information is included, layer information for curves is maintained, and Rhino RGB colors are converted to SpaceClaim RGB colors.
- You can open or insert a file from another application that contains a variable-radius round and edit that round as if it was created in SpaceClaim.

Resolved issues

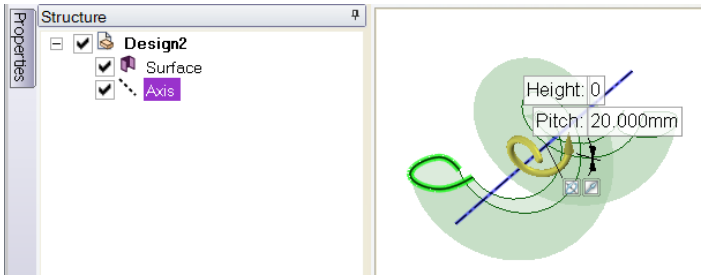
The following issues were resolved in SpaceClaim 2008 SP1.

General

- Right-clicking a component displays the Make Independent option in the context menu only if the component is a copied (dependent) component.
- You can adjust an object's visibility in the Structure tree and right-click to locate the object in the Structure tree when only some faces of the object are selected. This also works for multiple objects.
- When you drag an object in the Structure tree, the component into which the object will be placed is highlighted on mouseover.
- If you copy a face or solid as an unattached object, the face's or solid's layer and visibility state remain the same.
- Whenever you create a new object, such as when you paste an object, create a shell, split a solid by edges, drag using Select, or bend, it is placed on the same layer as the original object.

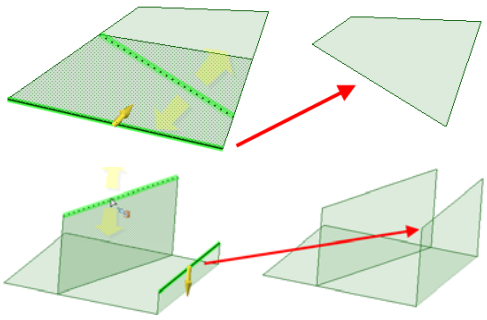
Pull

- You can pull an edge up to the face or edge of an inactive component. When you pull with the Blend option enabled, the blend is created immediately as soon as you begin pulling. There is no longer a delay. When you complete a blend between edges or lines, the edges are de-selected.
-
- If you undo a blend, then change the blend type and re-blend, the newly selected blend type is used.
- You can now view blend take-off vectors in subcomponents. Note that blend take-off vectors appear only for surface-to-surface blends, and do not work for closed loops.
- Cursor arrows (set by the Show cursor arrows setting in popular SpaceClaim options) no longer appear when you are adjusting blend take-off vectors.
- If you Alt+click an edge to select it as a guide for a blend, and the blend will self-intersect, then the blend will not occur, an error message appears, and the problematic edges flash red.
- Revolving offset faces with the Pull tool maintains their offset relationship.
- When you create a helix from a sketched line, the line is no longer maintained as a separate object. It is now included as part of the surface, as shown in the figure on the right. When a blend will create a self-intersecting curve, an error message is displayed, and the blend is not created.

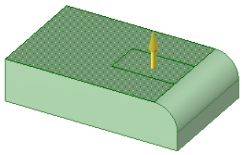


- When a blend between splines will self-intersect or almost intersect, the splines are modified slightly to prevent this from happening.

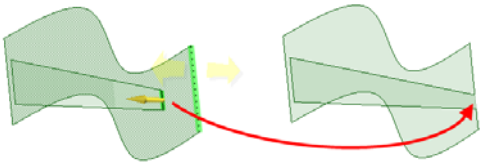
- You can now pull the edge of a surface up to a non-parallel edge, as shown in the figures on the right. In general, extending an edge works as long as the surface can be extended naturally. Then, you can split the surface with the Combine tool to trim it.
- You can now extrude an edge loop up to another object.
- You can now pull a cylinder up to another cylinder when the cylinders are in different components.



- A round face adjacent to a split face (as shown in the figure on the right) is pulled along with the split face if all the face pieces are selected.
- When you offset a face with the Pull tool, the Pull arrow is now anchored so that the dimension measures the offset correctly.

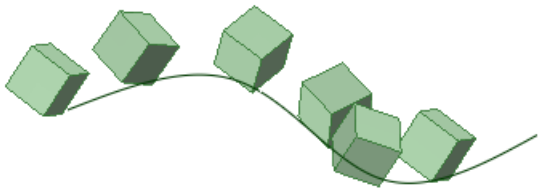


- You can now pull the edge of a planar surface up to the edge of a spline surface, as shown in the figure on the right.
- When creating a variable radius round, you can now tab between the % and length fields.



Move

- If you double-click an object obscured by the Move handle, that object is now selected.
- When you move a component, all objects within the component, such as planes, axes, origins, and sketch lines, are now moved as well.
- When you create a pattern of solids, the pattern members are all placed in the same component as the first member.
- When you edit a pattern, the selected member remains stationary.
- When patterning a component along a sketched spline, the components are correctly aligned, as shown in the figure on the right.
- You can now create a pattern of points in a component or subcomponent.
- When you edit a pattern of points along an edge, changing the percentage value now correctly updates the distance between each point.
- When moving mirrored faces or faces of a pattern member, they are updated immediately.



Fill and Replace

- You can now fill edge loops whose member edges simultaneously exist on different objects.

Intersect tools

- When using the Combine tool, if a target is displayed as metallic, the result of the merge will also be metallic.
- When splitting or combining multiple target solids, the original, individual layers and colors of the targets are retained.
- You can now split multiple solids or surfaces when they have conical faces.

- When using the Split Face tool's Select Cutter Point or Select Two Cutter Points tool guides, the cursor now snaps to points associated with an edge, such as a pattern of points along an edge.

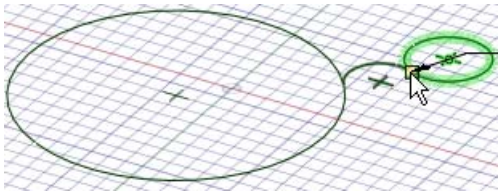
Other 3D tools

- Mirror relationships are maintained when drafting, revolving, and rotating faces.

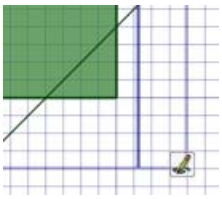
Section mode

- If you have the Auto extrude/revolve sketches in Section mode option set, and are sketching an extruded face in Section mode, the solid on which you are sketching is no longer moved into another component. Display
- When you check Clip Scene Above Grid, points above the grid (on the objects that have been clipped away) can no longer be selected.
- When sketching in Section mode, the mini-toolbar now appears.

Sketch



- When you create a rounded corner between two circles, as shown in the figure on the right, the previewed line now renders smoothly.
- Geometry edges and sketch lines are now displayed differently than plane boundaries and the intersection between two planes on the sketch grid. The figure on the right shows the edge of a box and diagonal sketch line in black, the boundary edge of a plane in light blue, and the intersection of two planes in dark blue.



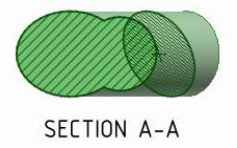
- Sketches no longer change when the linestyle is set to Thick. (Layouts do change.)

Annotations

- You can now use keyboard shortcuts when a note is selected or when you are manipulating the note. You cannot use keyboard shortcuts when you are editing the content of the note.
- Temporary axes are no longer displayed when you create a center mark or for surfaces that already have center lines, and if you delete a center line, the center line is no longer displayed as a temporary axis.
- If you select the Closed blank from the Default arrow fill style Detailing option for arrowheads, the arrowheads now appear correctly for dimension annotations.
- Cross-section lines now correctly reflect ISO standards, if you select that detailing option. When editing the text of a dimension, the text is now positioned correctly relative to the dimension lines. Note leader arrows now appear in the layer color.
- Line weights and font sizes now change in the Design window based on the zoom level of the design.

Drawing sheets

- When you have cross-section views on a drawing sheet and view the design in Section mode, the cross-hatching now displays correctly, as shown in the figure on the right.
- Spacing of dashed lines (such as those used for hidden lines) now appears correctly on the drawing sheet.

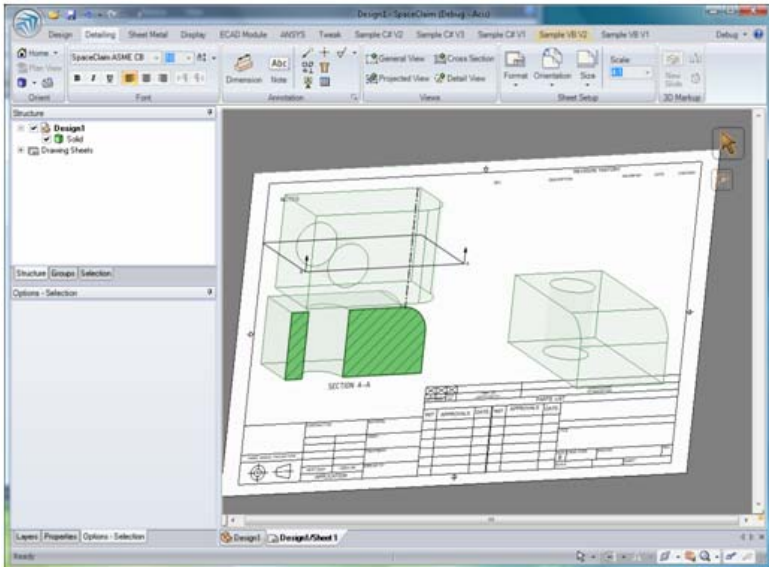


Sheet metal

- You can now unfold sheet metal boxes in more cases.
- When you create a new window by unfolding a sheet metal component, the window will fill all the space available in the Design window pane.
- Automatically-created bend angle notes on a sheet metal component are sized appropriately based on the size of the component.
- You can now create sheet metal corner reliefs with the Corner Relief option in more cases.
- You can now rotate sheet metal walls with the Move tool when they are located within a subcomponent.

Importing and exporting

- When you open or insert a DXF file, the colors within the design are now displayed correctly. When saving a sheet metal component as a DXF file, the bend angle dimensions and bend line axes are now included.
- When you open or insert a DWG file, it now displays correctly. When you save a design as a DWG file, diameter symbols are now saved correctly.
- When you open or insert an AutoCAD file, the global scale is now taken into account, so that annotations are displayed at the correct size. You can now save a drawing sheet as a DXF or DWG file when the sheet is not oriented to the plane of the screen. (For example, when you use the Orient tools to rotate the drawing sheet, as shown in the figure on the right.)



- When saving a SAT file to an X_T file, edges with zero length are removed from the design first to prevent faces from disappearing.
- You can now save designs as any file type when they have only one top-level component.
- When you save a design as a PDF file, colors set in SpaceClaim now appear correctly in the PDF file.