SpaceClaim User's Guide

Version 2008 SP1

A **SPACE**CLAIM Document

SPACECLAIM
CORPORATION

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Introduction

SpaceClaim Professional 2008 is the 3D productivity tool for engineers who need to focus on their core competencies while also benefiting from working in 3D. The software provides a highly flexible design environment coupled with a modern user experience that speeds contributions to the product development process. SpaceClaim Professional 2008 is for those who collaborate in the design and manufacture of mechanical products across a broad range of industries.

The online help, tutorials, and training materials are provided to help you become productive with SpaceClaim as quickly as possible. We strongly recommend that you review the Getting Started section and step through the tutorial provided in the online help before beginning your own work. Additional self-paced tutorial videos are available on MySpaceClaim.com. You can also begin by exploring a library of SpaceClaim models.

User's Guide

This User's Guide begins with a focus on the basic tools and on simple concepts. SpaceClaim is all about adding and manipulating the faces of a design model, primarily through pull and move operations. If there is a face, you can pull on it. If you need a new face, draw an edge or copy an existing one. Design clutter is minimized wherever possible. This guide communicates these simple, but powerful concepts so that you can extrapolate them to your real-world designs. This guide also provides useful shortcuts to use as you progress, as well as animations of tools in action to help you understand their function.

SpaceClaim is different, and we encourage you to open your mind and enter into a world where you can focus on the design, not the software. SpaceClaim appreciates your feedback, so let us know where we have succeeded and what we can do better. Thanks for your purchase and we look forward to working with you!

- Get started using SpaceClaim
    1. Sketch and pull to create a part, or open an existing model from any modeling software.
    2. Edit the part using SpaceClaim's 2D and 3D editing tools.
    3. (Optional) Customize SpaceClaim and your workspace to your working style.
    4. Detail the part with notes, measurements, and geometric tolerances.
    5. Submit the part for review using 3D Markup.
- Watch tutorials
- Review designs in the MySpaceClaim library

Getting started

Tutorials

Tutorials and demos are available on the SpaceClaim website. Step-by-step tutorials are also available in this Help file. Working through each of the tutorials will allow you to quickly grasp the basics of using SpaceClaim. We strongly recommend that newcomers to 3D design run through the tutorials. You will gain competency with the functionality featured and the experience will help you master the remaining features more easily.

Text tutorials

The following tutorials are available in this Help file:

Bracket and knob

In this tutorial, you will:

- Create a bracket using SpaceClaim's sketching and 3D editing tools
- Create an assembly by importing and modifying a knob to fit into your bracket
- Create a drawing sheet to detail your design

Turbine wheel and housing

In this tutorial, you will:

- Create a turbine wheel using SpaceClaim's sketching and 3D editing tools
- Create a sheet metal housing for the turbine wheel

Self-paced training tutorials

Self-paced, animated training tutorials are available on MySpaceClaim.com, a personalized web portal for easy access to everything SpaceClaim. On MySpaceClaim.com, you can:

- Gain access through a unique user name and password
- Directly download SpaceClaim software, including purchased new products, updates, and upgrades
- Learn from self-paced training tutorials
- Submit a new idea

To access MySpaceClaim.com, select the **Login** link at the top of the SpaceClaim.com home page.

Bracket and knob assembly

In this tutorial, you will:

- Create a bracket using SpaceClaim's sketching and 3D editing tools
- Create an assembly by importing and modifying a knob to fit into your bracket
- Create a drawing sheet to detail your design

Please note that as you move back and forth between this help window and the SpaceClaim application, you may need to click once in SpaceClaim to make it the active window before following the directions in the step.

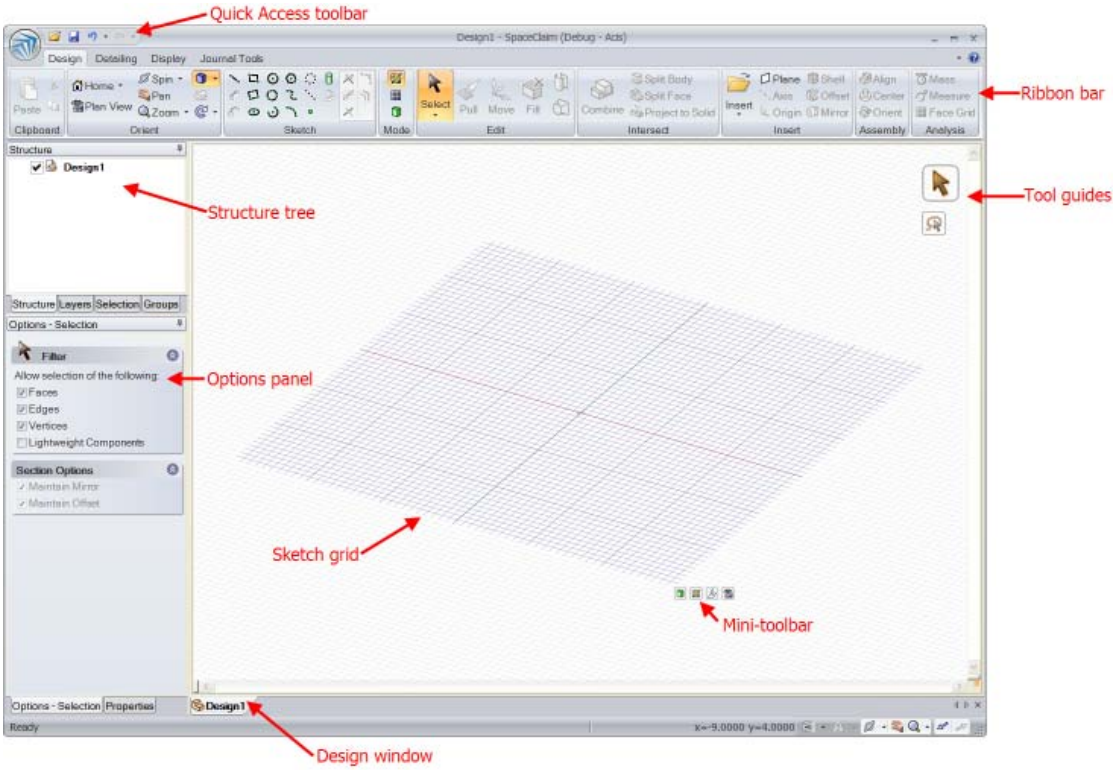
Getting started

1 Create a new design document.

a Select **New > New Design** from the Application menu to create a new design.

A blank design containing the sketch grid is displayed in a new Design window. The mode is set to Sketch, since that is usually the first step to create a new design.

The following figure shows some of the interface elements referred to in this tutorial.



b Select **Save** from the Application menu to name and save your design.

The name of your design appears as the top-level component in the Structure tree.

2 Set your design preferences.

a Click **SpaceClaim Options** in the Application menu.

b Click **Units**.

c Select **Imperial** from the Type drop-down.

Inches appear in the Length drop-down, the minor grid spacing changes from .1mm to 1/8 in, and the minor grid lines per major changes from 10 to 8. This means that you can dimension in inches, and that the sketch grid lines are now spaced 1/8 inch apart, and the darker grid lines appear every inch.

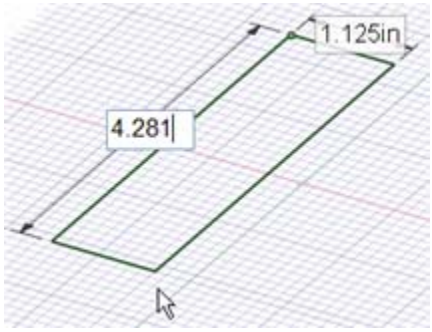
d Select **Decimal** from the Decimal/fraction drop-down.

e Click **OK**.

Creating the bracket

3 Create the bottom of the bracket using the sketch and pull tools.

a Sketch a rectangle that will become the bottom piece of the bracket.



1 Click the Rectangle tool in the Sketch ribbon group on the Design tab.

2 Click to set the first corner of the rectangle. (Start at the upper left.)

As you move your mouse, a preview of the rectangle is drawn, and dimension fields appear.

3 Enter **1.125**, then press **Tab** and dimension the second side by entering **4.281**.

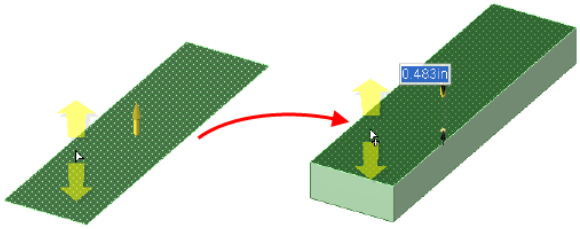
If you make a mistake, click a dimension to edit it. Or click the Select tool in the Edit ribbon group and double-click the rectangle to select it. Then press **Delete** to delete the rectangle and redraw it.

3 Press **Enter** to complete the rectangle.

b Pull the rectangle into 3D to create the bottom of the bracket.

1 Switch to 3D mode by clicking the 3D mode tool in the Mode ribbon group.

The Pull tool in the Edit ribbon group is activated, your sketched rectangle now appears as a rectangular surface, and the surface appears in the Structure tree.



2 Click on the face of your rectangular surface to select it.

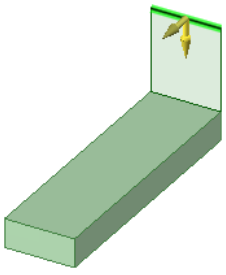
The faint yellow cursor arrows show you the directions in which you can pull the rectangle.

3 Drag to begin adding thickness to the rectangle.

You can drag with your cursor anywhere in the Design window – you do not have to drag on the Pull arrow itself. We recommend that you move your mouse off to the side when pulling to make it easier to see your changes.

4 Enter **.483** and press **Enter** to dimension the pull.

The surface in the Structure tree is replaced by a solid.



4 Create the back of the bracket by extruding an edge and adding thickness to the resulting surface.

a Extrude an edge to form a surface.

1 Click the back edge of the solid with the Pull tool to select it.

The edge is highlighted and edge options appear in the Options panel and mini-toolbar. Move your mouse closer to the mini-toolbar to make it more opaque.

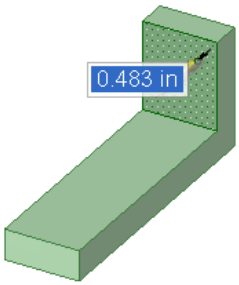
2. Select the extrude edge option in the Options panel.

   (You can hover over any option to display a tooltip that explains the option.)

   The Pull arrows change to indicate the two default directions in which you can extrude the edge.
3. Click the vertical arrow and drag the edge upward to begin creating a surface.
4. While dragging, press and release the spacebar to display a dimension field.
5. Enter **1.4**.
6. Press **Enter** to complete the surface.

   This surface now appears in the structure tree, below the solid.

b. Pull the surface into 3D.

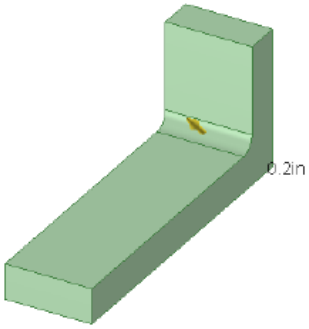


1. Click the surface you just created to select it.
2. Drag toward the front of the solid.
3. Enter **.483** to match the thickness of your first solid.
4. Press **Enter** to finish pulling and create the surface.

   In the Structure tree, the surface disappears and this new solid is automatically merged with the first.

5. Round the corners.

a. Round the inside corner of the bracket.

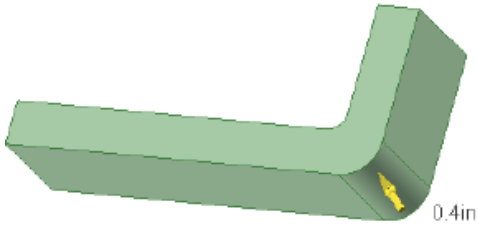


1. Click the edge on the inside corner of the bracket with the Pull tool.
2. Click the Round Edge pull option in the Options panel.
3. Drag away from the solid to round the edge.
4. While dragging, press and release the spacebar to display a dimension field.
5. Enter **.2**.
6. Press **Enter** to finish pulling and create the round.

b. Round the outside corner of the bracket.

1. Turn the bracket so you can see the bottom by clicking the Spin tool in the Orient ribbon group and dragging to spin your design.

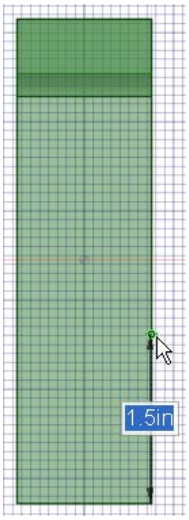
   Another way to spin is to mouse over an edge in your design. Then press **Alt** and drag with the middle mouse button to spin your design around that edge. Spinning in this way lets you keep the Pull tool active.



2. Click the Pull tool, then click the edge that forms the outside corner of the bracket.
3. Drag into the solid to round the edge.
4. While dragging, press and release the spacebar to display a dimension field.
5. Enter **.4**.
5. Press **Enter** to finish pulling and create the round.
6. Click Home in the Orient ribbon group to view the design in trimetric view.

6 Remove material by sketching and pulling.

a Sketch a dimensioned point on the top face of the bottom piece of the bracket.



1. Click the Select tool in the Edit ribbon group and click the top face of the bottom piece of the bracket to select it.
2. Click the Point tool in the Sketch ribbon group.

   You are now in Sketch mode. The sketch grid appears and the Sketch mode tool is active in the Mode ribbon group.

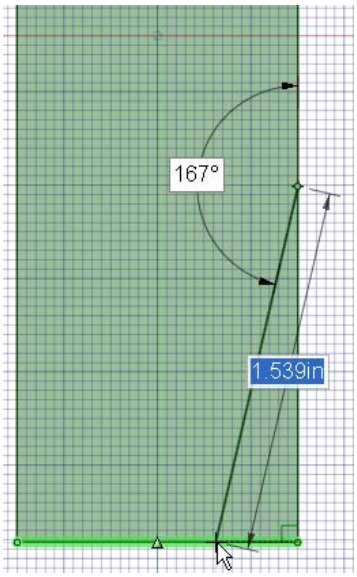
   Because you entered Sketch mode with a face selected, SpaceClaim assumes you want to sketch on that face, and orients the sketch grid along that face.
3. Click Plan View in the Orient ribbon or the mini-toolbar to view the sketch grid head-on.
4. Place the cursor over the bottom vertex of the face (as shown in the figure), press and release **Shift**, then move your mouse toward the back of the bracket along the right edge. Do not press the mouse button.

   A dimension field appears. (You can "Shift+touch" any object in any tool to dimension from that object.)
5. Press and release the spacebar to dimension the point's distance from the vertex.

   Enter **1.5**. If you need to, press **Tab** to switch dimension fields.
6. Press **Enter** to create the point.

   If the point was created at the wrong place, you can press **Ctrl+Z** or click in the Quick Access toolbar (on the left side of the SpaceClaim title bar) to try again.

b Draw an angled line.



1. Click the Line tool in the Sketch ribbon group.
2. Click the point you created in the previous step.
3. Move the mouse towards the end of the bracket. Do not hold a mouse button.

   Two dimensions appear, one for the line's length, and one for the angle formed between the sketch grid's axis and the line.
4. Press and release the spacebar to dimension the line. Press **Tab** to switch to the angle dimension.

   Enter **13**. (You may need to enter a different value if the angle dimension is drawn to a different axis than is shown in the figure.)
5. Press **Tab** and continue moving the mouse until it intersects with the end of the bracket.

   The edge that forms the end of the bracket highlights when the line intersects with it.
6. Double-click to end the line.

   If the line tool continues to draw line segments, press **Esc** or right-click and select **Finish Line**.

c Remove material from the bracket.

1 Click Home in the Orient ribbon group to view the design in trimetric view.

2 Click the Pull tool in the Edit ribbon group.

You are now back in 3D mode. The 3D mode tool is active in the Mode ribbon group.

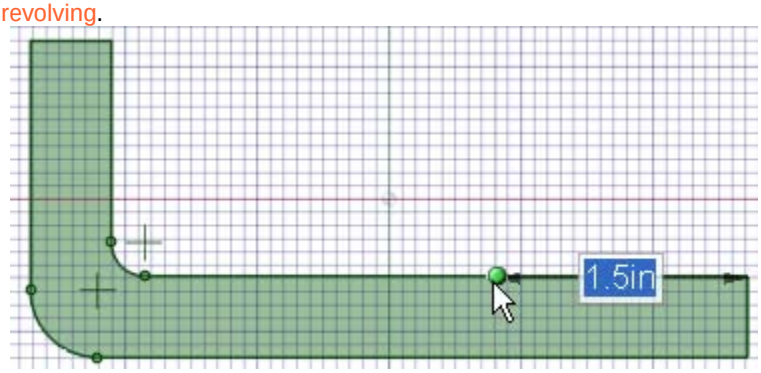
3 Click the triangular region created by the line and the edge of the bracket.

4 Drag downward until all the material is removed.

Notice that as you pull through the solid of the bracket, the Pull tool assumes that you want to remove material, and the cursor changes to indicate that the pull is subtractive.

7 Remove material by pivoting and revolving.

a Sketch a dimensioned point on the side face of the bracket.



1 By dragging with the middle mouse button, turn the bracket so you can see the side opposite the side you just removed material from. (You can also click the Spin tool in the Orient ribbon group and drag to spin your design or select **Right** (or **Left**) from the Trimetric tool menu.)

Click Pan in the Orient ribbon group and reposition the bracket within the Design window.

2 Click the Select tool in the Edit ribbon group and click the side face of the bracket to select it.

3 Click the Point tool in the Sketch ribbon group.

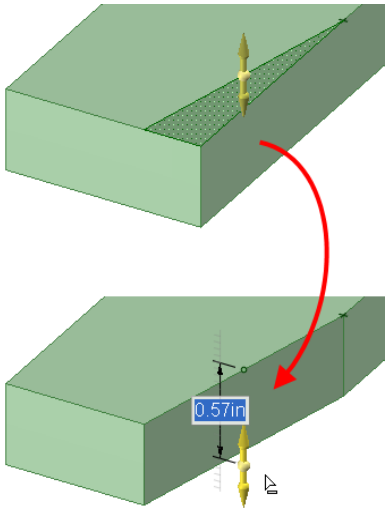
You are now in Sketch mode. You can tell you are in Sketch mode because the sketch grid appears, and the Sketch mode tool is active in the Mode ribbon group.

Because you entered Sketch mode with a face selected, SpaceClaim assumes you wanted to sketch on that face, and orients the sketch grid along that face.

4 Click Plan View to view the sketch grid head-on.

5 Place the cursor over the vertex at the end of the bracket, press and release **Shift**, then move your mouse toward the back of the bracket along the edge. Do not press the mouse button.

A dimension appears. (You can "Shift+touch" any object in any tool to dimension from that object.)



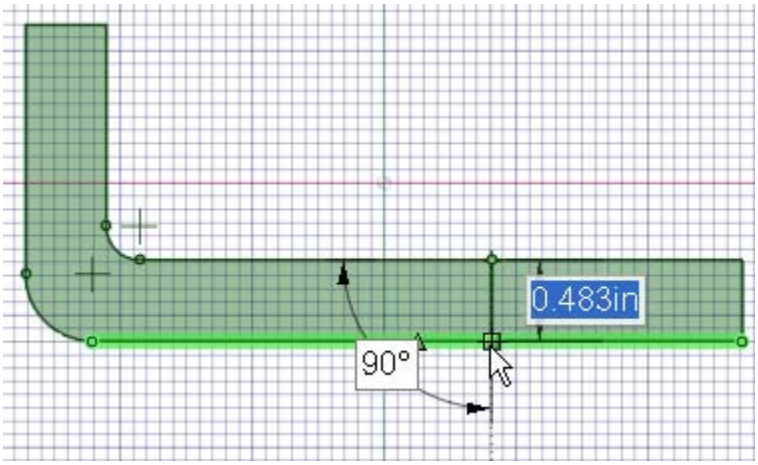
6 Press and release the spacebar to dimension the point's distance from the vertex.

Enter **1.5**.

7 Press **Enter** to create the point.

If the point was created at the wrong place, you can press **Ctrl+Z** or click in the Quick Access toolbar (on the left side of the SpaceClaim title bar) to undo and try again.

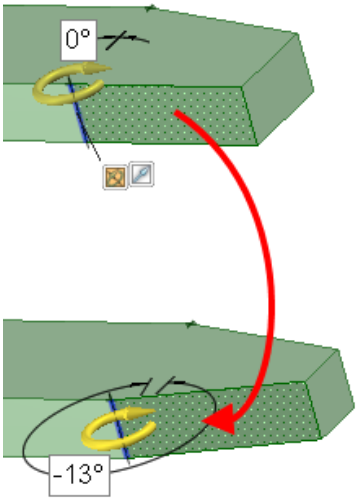
b Draw a line that will become the edge to pivot around.



1. Click the Line tool in the Sketch ribbon group.
2. Click the point you created in the previous step.
3. Move the mouse until the line is drawn at 90 degrees from the top to the bottom of the bottom piece of the bracket.
4. Double-click to end the line.

   If the line tool continues to draw line segments, press **Esc** or right-click and select **Finish Line**.

c Revolve the face to match the angled face on the other side of the bracket.



1. Click the Pull tool in the Edit ribbon group.

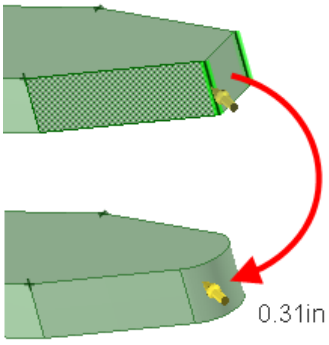
   You are now in 3D mode. The 3D mode tool is active in the Mode ribbon group.
2. Turn the bracket so you can see the side and top of the bracket by clicking the Spin tool in the Orient ribbon group and dragging a small amount to spin your design.
3. Click the Pull tool again and click the face between the pivot line and the end of the bracket.
4. Click the Revolve tool guide (located on the right side of the Design window).
5. Click the pivot line.

   The line is highlighted in blue and the Pull arrow changes to show that pulling will revolve the selected face.
6. Drag to begin revolving the face.
7. While dragging, press and release the spacebar to dimension the revolve.

   Enter **-13**.
8. Press **Enter** to finish the revolve.

8 Pull to round the angled end.

a Round the angled end of the bracket.



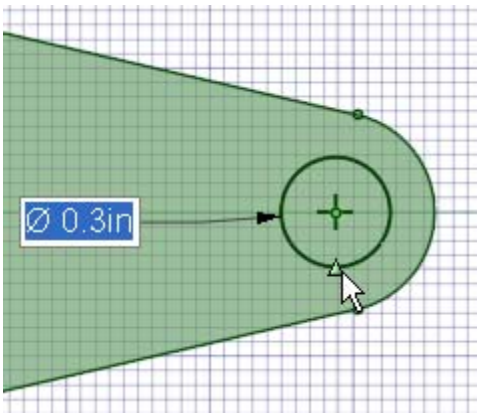
1. Click in the empty space in the Design window to clear your previous selections.
2. Click one of the edges at the end of the bracket to select it.

   (You can drag with the middle-mouse button to quickly spin your design.)

   Notice that the round edge option is selected by default, since the Pull tool assumes from your selection that you want to round the edge.
3. Ctrl+click the other edge to add it to your selection.
4. Drag into the solid to round both edges simultaneously. Continue dragging until the rounds meet in the center to form a full round.

9 Sketch and pull to create a hole.

a Create a circle at the end of the bracket.



1 Click the Select tool in the Edit ribbon group and click the top face of the bottom piece of the bracket to select it.

2 Click the Circle tool in the Sketch ribbon group.

You are now in Sketch mode. You can tell you are in Sketch mode because the sketch grid appears, and the Sketch mode tool is active in the Mode ribbon group.

Because you entered Sketch mode with a face selected, SpaceClaim assumes you wanted to sketch on that face, and orients the sketch grid along that face.

3 Click Plan View in the Orient ribbon group to view the sketch grid head-on.

The center of the arc created by the full round is shown with a cross.

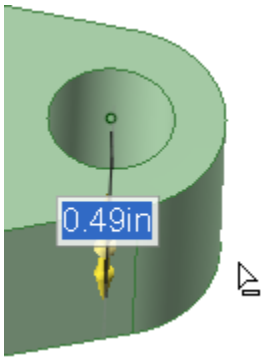
4 Click the center of the arc and drag to begin sketching a circle.

5 While dragging, press and release the spacebar to dimension the pull.

Enter **.3**.

6 Press **Enter** to create the circle.

2 Pull the circle to create a hole in the end of the bracket.



1 Click the Pull tool in the Edit ribbon group.

You are now in 3D mode. The 3D mode tool is active in the Mode ribbon group.

2 Turn the bracket so you can see the side and top of the bracket by dragging slightly with the middle mouse button.

3 Click on the circular region to select it.

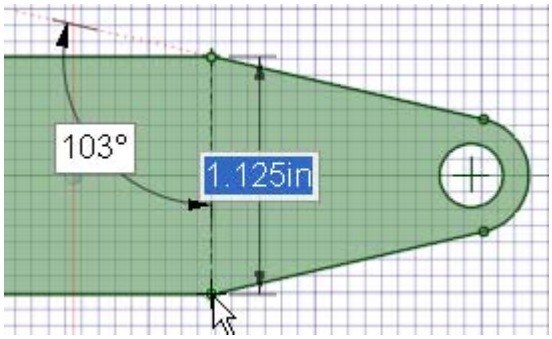
4 Drag downward until all the material is removed.

Notice that as you pull through the solid of the bracket, the Pull tool assumes that you want to remove material, and the cursor changes to indicate that the pull is subtractive. If you pull away from the solid, it will add material.

5 Drag with the middle mouse button to spin the bracket so you can see through the hole.

10 Create a precisely placed hole.

a Sketch a construction line so you can center a hole on the angle points.



1 Click the Select tool in the Edit ribbon group and click the top face of the bottom piece of the bracket to select it.

2 Click the Construction line tool in the Sketch ribbon group.

You are now in Sketch mode. The sketch grid appears and the Sketch mode tool is active in the Mode ribbon group.

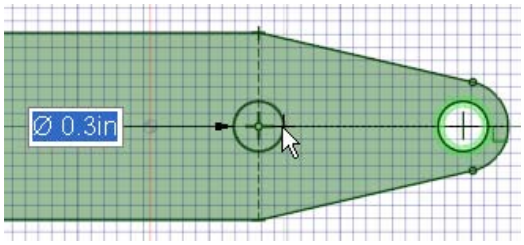
Because you entered Sketch mode with a face selected, SpaceClaim assumes you wanted to sketch on that face, and orients the sketch grid along that face.

3 Click Plan View to view the sketch grid head-on.

4 Click the vertex on one side of the angled part, then click the vertex on the opposite side.

The cursor snaps to each vertex and it is highlighted in green. A dotted construction line appears.

b Sketch a circle at the midpoint of the construction line.



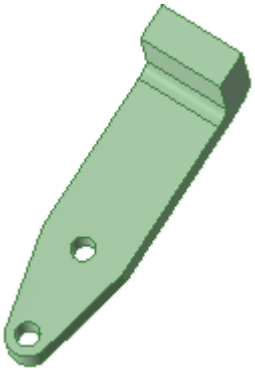
1 Click the Circle tool in the Sketch ribbon group.

Mouse over the construction line. A triangle indicates the line's midpoint.

2 Click the triangle and move the mouse slowly, until the existing hole is highlighted.

3 Release the mouse button to create a circle that matches the diameter of the first hole.

c Pull the circle to create the second hole.



1 Click the Pull tool in the Edit ribbon group.

2 Click Home in the Orient ribbon group to view the design in trimetric view.

3 Click on the circular region to select it.

4 Drag downward until all the material is removed.

5 Drag with the middle mouse button to spin the bracket so you can see through both holes.

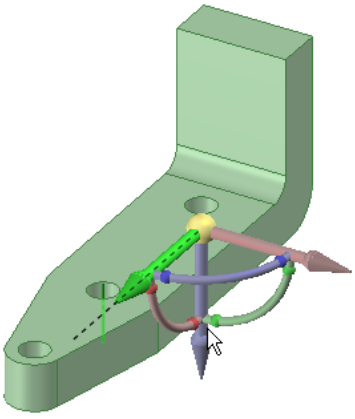
Notice that the construction line is converted to an axis, and now appears in the Structure tree.

6 Uncheck the axis in the Structure tree to hide it.

You can also remove the sketched points by clicking them with the Select tool, then pressing **Delete**.

11 Create a pattern of holes.

a Create a pattern with the Move tool.



1 Click the Move tool in the Edit ribbon group.

2 Check the **Create Patterns** option in the Options panel.

Zoom into your design by selecting **Zoom Extents** from the Zoom tool menu to make the next step easier.

3 Click the inside surface of the hole that is centered on the angle points.

The Move handle is aligned along the axis.

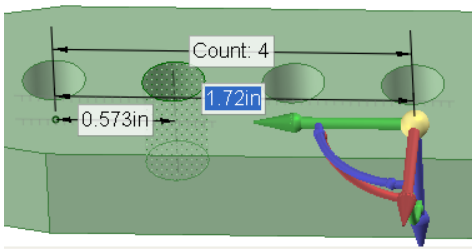
4 Click the Move handle axis that is in line with the long dimension of the bracket.

5 Press **Ctrl** and drag almost to the round at the back of the bracket.

6 Release **Ctrl** and the mouse button.

A pattern count parameter is displayed, along with the dimension from the original hole to the copied hole.

a Edit the pattern.



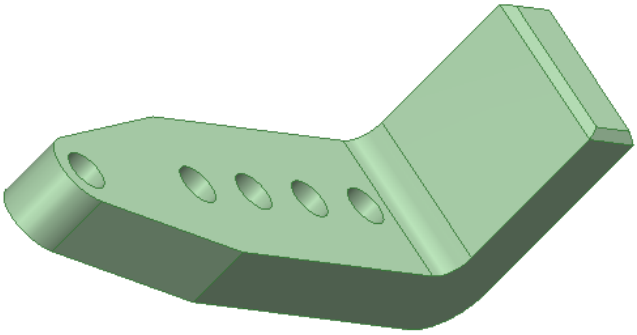
Enter **4** as the count.

Four identical, equidistant holes appear on the bracket surface. These holes are now part of a pattern. The dimension between each hole in the pattern is now displayed as well as the other parameters.

Changes made to one hole will now affect all the others in the pattern. For example, if you click the Pull tool and select an axis of one of your holes, then drag, you can see all the holes change to slots. (Press **Ctrl+Z** to undo your change.)

12 Pull to chamfer the top edge.

a Chamfer the top edge.



1 Click the Pull tool in the Edit ribbon group.

2 Select the chamfer edge option.

3 Double-click one of the top edges to select the edge loop.

If the wrong edge loop is selected, double-click to select an alternate loop.

4 Drag into the solid to begin chamfering the edge.

5 While dragging, press and release the spacebar to dimension the chamfer's setback.

Enter **.1**.

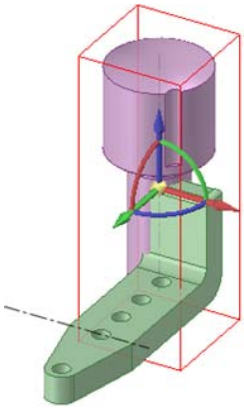
6 Press **Enter** to finish pulling and create the chamfered edges.

b Press **Ctrl+S** or select **Save** from the Application menu to save your design.

Importing, modifying, and aligning the knob

13 Import the knob.

a Get the knob model.



1 Select **SpaceClaim Options** from the Application menu.

2 Click **Resources**.

3 Click **Get Models** to display the SpaceClaim Model Library on the SpaceClaim website.

4 Find the **TutorialKnob.scdoc** file and click **Download**.

b Insert the knob component.

1 Click **Home** to orient your bracket.

2 Click the Insert tool in the Insert ribbon group to display the Open Design window.

3 Navigate to and select **TutorialKnob.scdoc** and click **Open**.

The knob appears in the Design window inside the outline of a red box with the Move tool active to move it to a better position.

c Move the knob so its small end is pointing at the back of the bracket.

1 Turn the bracket and knob so you can see their sides by clicking the Spin tool in the Orient ribbon group and dragging to spin your design. (You might find it easier to use the trimetric views.)

2 Click an arrow of the Move tool and drag the knob until it is far enough past the bracket so you can turn it on its side.

3 Click the curved arrow of the Move tool and drag the knob until it is pointing toward the back of the bracket.

d Activate the bracket component.

1 Mouse over the top-level structure in the Structure panel.

2 Right-click and select **New Component**. A new component, Component1, appears in the structure tree and is in bold, indicating it is activated.

3 Right-click the new component, **Component1**, click **Rename**, and name the new component **Bracket**. It appears in bold to indicate that it is the active component.

4 In the Structure tree, drag the first object, **Solid**, and drop it on the new Bracket component. You now have an ordered structure for your design.

5 In the Structure tree, right-click the top-level component and select **Activate Component**. The top-level component is now active, making both subcomponents active.

14 Modify the knob.

a Activate the knob as a component, preparing to work on it by itself.

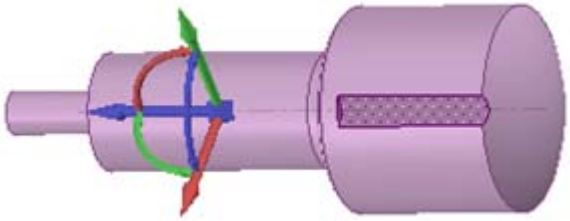
1 In the Structure tree, mouse over the knob component.

A red box appears around the knob.

2 Right-click and select **Activate Component**.

The knob component appears in bold to indicate that it is active.

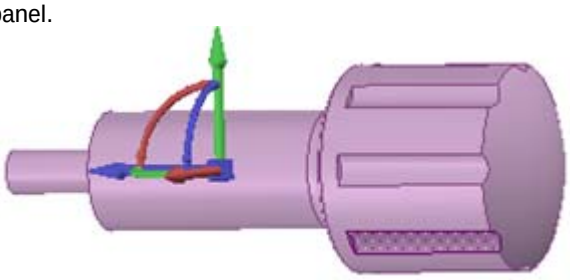
b Create a pattern of grooves with the Move tool.



1 Turn the knob so you can see the whole groove by clicking the Spin tool in the Orient ribbon group and dragging to spin your design.

To make this easier, zoom into your design by selecting **Zoom Box In** from the Zoom tool menu in the Orient group.

2 Check **Create Patterns** in the Move Options panel.



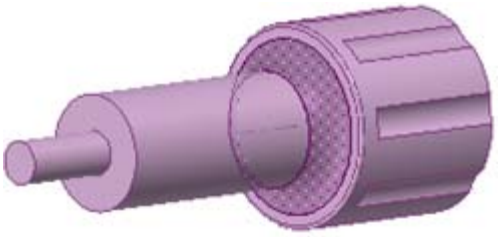
3 Click the Select tool in the Edit ribbon group and click both surfaces of the groove. Ctrl+right-click on both surfaces.

4 Re-anchor the Move tool to a central axis so you can copy the groove by dragging the center of the Move handle, or using the Anchor tool guide - select the Anchor tool guide (on the right of the Design window), then click on the axis in the center of the knob.

5 Press **Ctrl** and drag with the right mouse button slowly along the blue rotate arrow.

SpaceClaim gives you its best idea of what you would like for a pattern. It stops at 45°. If you kept going it would snap to a 60° pattern.

c Remove unwanted space with the Fill tool in the Edit ribbon.



1. Rotate the knob so you can see the fillet under the head of the knob.
2. Click on the Select tool in the Edit ribbon group then click on the fillet.
3. Click on Fill tool in the Edit ribbon

   The fillet becomes flat.

15 Fit the knob to the bracket.

a Measure the small cylinder on the end of the knob.

1. Zoom out of your design by selecting **Zoom Out** from the Zoom tool menu.
2. Click the Measure tool in the Analysis ribbon, then click on the small cylinder.

   You will see measurements for the circle diameter (3/8 in), circle perimeter (1.178 in) and angle between adjacent surfaces (90°).

   Click on other parts of the model to see their dimensions.

b Create a hole in the back of the bracket so you can insert the knob into it.

1. Right-click **Bracket** in the Structure tree and select **Activate Component** from the drop-down menu.

   A red box appears around the Bracket.
2. Pan over to the bracket and spin it so you can see the back of its wall and the knob.
3. Click the back of the bracket then click the Sketch Mode tool in the Mode ribbon. The grid appears on the back of the bracket.
4. Click the Display tab above the ribbon.

   In the Grid group, click **Clip Scene Above Grid**. The knob temporarily disappears.
5. Return to the Design tab and click the Trimetric tool from the Orient ribbon, and select **Back** from the menu. The back of the bracket faces you.

c Use a construction line to locate the hole in the center of the back.

1. Click the Construction line tool in the Sketch ribbon group.

   Move the cursor over the top line. It snaps to the center of the line. Be careful to click on the top of the back and not on the top of the chamfer.
2. Drag to draw a construction line to the bottom of the surface.

   The line snaps to the center of the bottom edge. Press **Esc** to exit the tool.
3. Click the Circle tool in the Sketch ribbon group and move the mouse over the top of the construction line. (Do not press a mouse button.)
4. Press and release the Shift key to dimension from another point, then move the mouse slowly in the direction you want to place the circle. In the dimension box, type the value where you want to locate the circle center (**.742**). Press **Enter**. The cursor is moved to center of the hole.
5. Move the mouse slowly and a second dimension box appears into which you can type the diameter of the circle (**.376**). Press **Enter**. Press **Esc** to exit the tool.
6. Click the Pull tool in the Edit ribbon group tool then click on the center of the hole so the direction arrows are over the hole.
7. Select the Up To tool guide on the right of the Display window. Move the mouse to the side of the model and roll the middle button to select the hidden surface. When it is highlighted, click. The circle goes to the surface and becomes a hole.

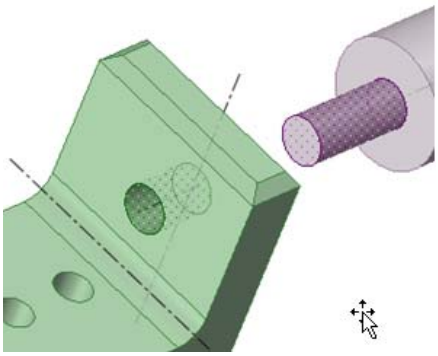
d Create an assembly.

1 Click the knob in the Structure tree to make it reappear in the display.

2 Drag with the middle mouse button to spin the model so the knob is on the right and the bracket is on the left.

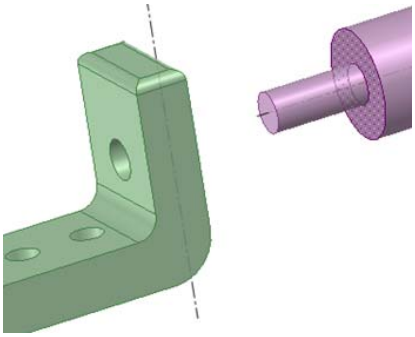
3 Click the Select tool in the Edit ribbon group, select the surface of the small end of the knob, then Ctrl+click the inside surface of the hole. Click the Center tool in the Assembly ribbon.

The knob moves so its end is aligned with the hole.



4 Click the flat surface of the knob just below the revolve surface.

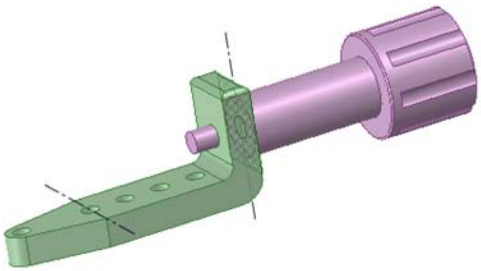
Move the mouse to the side of the bracket, turn the scroll wheel to highlight the back of the bracket. Ctrl+click the side.



5 Select the Align tool on the Assembly ribbon. The knob slips through the hole in the bracket.

The surfaces that move are those of the model you pick first.

6 Spin the assembly around to see that you assembled the bracket and the knob.



Creating the drawing sheet

16 Create a drawing sheet for your design.

a Shut off the display of the knob by unchecking it in the Structure tree.

b Click the Application Menu and select **New > Drawing Sheet**.

A new window appears with a drawing sheet form that includes the parts of your model in three orthogonal orientation for third angle projection and a format for entering dimensions and general information. Click the Select tool and move the parts of the model so they are closer together.

c Make the sheet smaller by clicking on the Detailing tab above the ribbon.

1 Click on the Format tool in the Sheet Setup ribbon.

2 Select **Format A Portrait** size. The bracket drawing sheet moves to the center of the window in portrait form.

You can toggle between the model and the drawing sheet by using the tabs below the Design window.

17 Annotate the design.

a Enter width of bracket back.

1 Click on the Design tab above the ribbon. To make this easier, zoom into the bracket components by selecting **Zoom Box In** from the Zoom menu in the Orient ribbon.

2 Click on the Detailing tab again then click on the Dimension tool in the Annotation ribbon.

3. Click on the left edge of the wall of the bracket (bottom left of drawing) then click on its right edge. Do not use the Ctrl key.

   A dimension box displays the width of the wall.

4. Move the mouse up (no buttons should be pressed) until the dimension box is located where you want it, then click.

b Enter the width of the bracket base.

1. Click on the top edge of the bracket base (bottom right of drawing) then click on its bottom edge. Do not use the Ctrl key.

   A dimension box displays the width of the base.

2. Move the mouse over (no buttons should be pressed) until the dimension box is located where you want it, then click.

   Notice the dimension is the same that you used to create the bracket

c Enter the height of the bracket back.

1. Click on the bottom edge bracket base (bottom right of drawing) then click on the top edge of its back. Do not use the Ctrl key.

   A dimension box displays the height of its back.

2. Click and move the mouse (no buttons should be pressed) until the dimension box is located where you want it then click again.

d Enter dimensions for the top view of the bracket (top of drawing).

1. Click on the edge of the hole at the end of the bracket.

   Move the mouse (no buttons should be pressed) until the diameter of the hole is located where you want it, then click. You may need to click on the arrow to move it to the edge oh the circle.

2. Click on the top of the back of the bracket.

   Move the mouse (no buttons should be pressed) to the bottom of the rounded edge. When you move the mouse, many dimensions appear, including the tangent point of the bottom edge. Click and move the dimension until you have placed it where you want it.

   Notice the dimension is 4.281, which is what you entered to create the solid.

e Create a note.

1. Click on the Note tool in the Annotation ribbon.

   MB1 where you want the note to be on the drawing and begin to type your note. Press **Esc** to exit the tool.

2. Click on the Note Leader tool  in the Annotation ribbon.

   Mouse over the note to see where you can attach the leader. Click on one of those places and drag. When the leader reaches where you want it to end, release the mouse and press **Esc** to exit the tool.

3. Create a Material Finish Symbol.

   Select the Material Removal Required tool from the Surface Finish list in the Annotation ribbon. Place the mouse where you want to place the symbol, click, then drag to the end of the symbol. Double-click to end the line.

4. Change the design from the drawing.

   You can see the solid models in the drawing by pressing the middle mouse button and spinning the drawing.

18 Modify the design from the drawing sheet.

a Change the design from the drawing.

1 Change the size of the .3 diameter hole at the bottom of the bracket.

Turn the drawing so you can see inside the hole by clicking the Spin tool  in the Orient ribbon group and dragging to spin your design.

Click on the Design tab above the ribbon.

Enlarge the hole by selecting **Zoom Box In** from the Zoom menu in the Orient ribbon.

Select the Pull tool in the Edit ribbon group then select the inside diameter of the hole. Press and release the spacebar and enter a radius of **.2**.

Zoom out. The dimension has been changed to .4 on the drawing.

2 Two views of the changed drawing.

To see the drawing view, right-click and select **View > Flat View**.

To view the whole drawing, right-click and select **View > Home**.

Turbine wheel tutorial

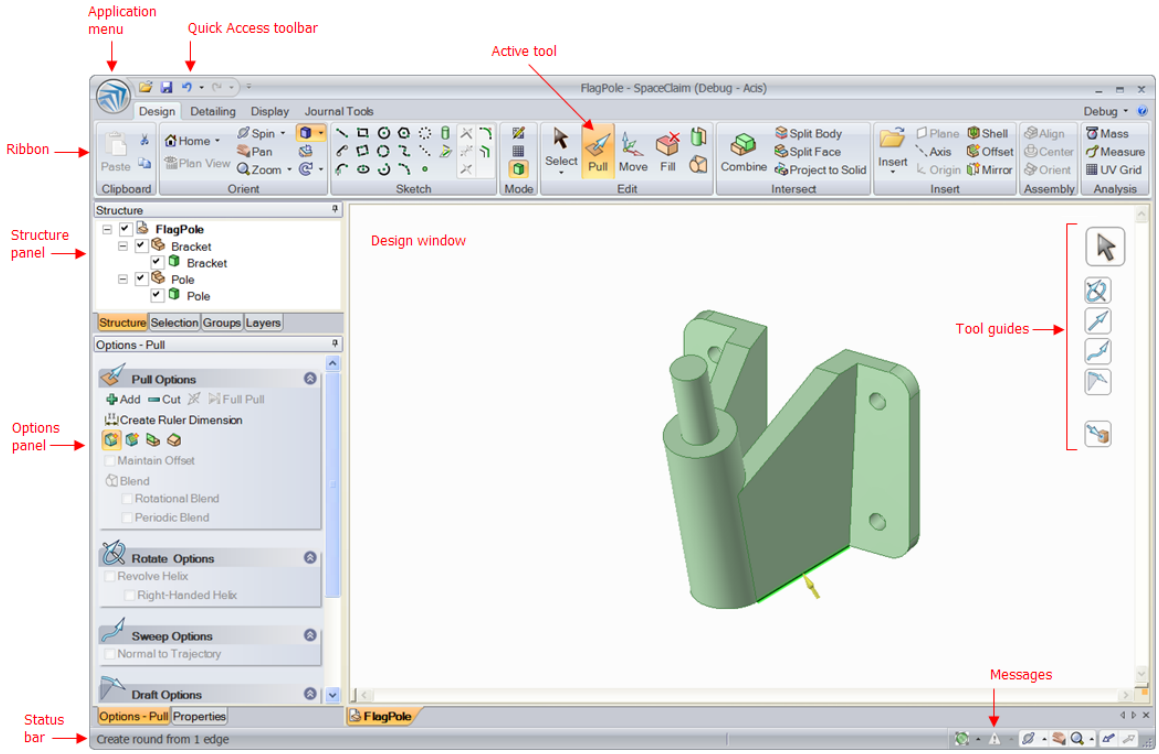
This tutorial will be inserted in the next version of the Help.

SpaceClaim interface

SpaceClaim's graphical user interface (GUI) was designed to conform (within reason) to Microsoft Vista standards and contains the toolbars, buttons, and windows associated with a Vista-compliant graphical application. As a result, only those features of the GUI that relate to performing SpaceClaim-specific tasks are explained in this guide. We assume, for example, that you are familiar with standard Windows conventions, such as dragging a window's title bar to move the window, or clicking the close button to close the window.

To take advantage of the full range of SpaceClaim features, we recommend using SpaceClaim with a scroll wheel mouse or with a 3D Connexion SpaceBall or SpaceNavigator. However, SpaceClaim is also fully operational with a laptop's touchpad and integrated mouse buttons. You can use the nub as a scroll wheel, and configure the laptop so that pressing both buttons simultaneously behaves the same as pressing a middle mouse button.

This image shows the major interface elements in the SpaceClaim application:



The Application menu contains file-related commands and options to customize SpaceClaim.

The Quick Access toolbar can be customized so that it contains the file-related shortcuts you use most often.

The Ribbon contains all the tools and modes you need to design, detail, and display models, drawing sheets, and 3D markups.

The Design window contains your model. If you are in sketch or section modes, it also contains the sketch grid to show the 2D plane on which you are working. The tool guides for the selected tool appear on the right side of the Design window. The cursor also changes to indicate the selected tool guide. A mini-toolbar places commonly used options and actions close to the cursor.

The status bar displays messages and progress information about your actions on the current design.

The Message icon displays error messages as they occur. Click the icon to display all the messages currently relevant to your design. Click a message to highlight the object referenced by the message.

Panels

The panels initially appear along the left side of the application window. You can dock and detach these panels.

The Structure tree contains the Structure tree, which shows you each of the objects in your design. You can quickly show or hide any object using the checkbox next to the object's name. You can expand or collapse the nodes of the tree, rename objects, create, modify, replace, and delete objects, as well as work with components.

The Layers panel allows you to group objects and set their visual characteristics, such as visibility and color.

The Selection panel lets you select other objects related to the one currently selected.

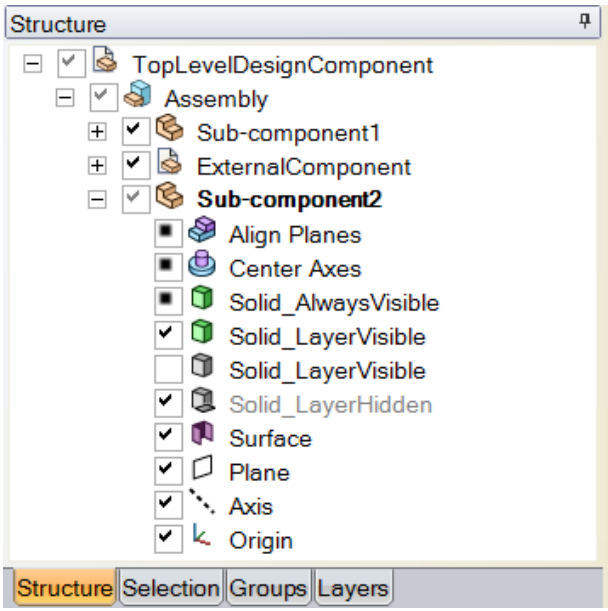
The Groups panel stores groups of selected objects. Selection, Alt+selection, and move anchoring, axis, and ruler dimension information is all stored with the group.

The Options panel allows you to modify the functions of the SpaceClaim tools. For example, when you use the Pull tool, selecting an edge and then selecting the Chamfer Edge option creates a chamfer instead of a round when you pull the edge.

The Properties panel displays details about the selected object. You can change the property values to change the object.

Working with objects in the Structure tree

The Structure tree contains the Structure tree, which shows you each of the objects in your design. You can quickly show or hide any object using the checkbox next to the object's name. You can expand or collapse the nodes of the tree, rename objects, create, modify, replace, and delete objects, as well as work with components.



The top-level design (called **StructureTree** in the image on the right) is also a component. The figure on the right shows some objects that can appear in a Structure tree.

When you select a solid or surface (or other object) in the Design window, it is highlighted in the Structure tree.

You can Ctrl+click or Shift+click objects in the Structure tree to select multiple objects at once.

To set the visibility of objects

There are now three methods for setting the visibility of objects in the Design window:

- Right-click an object in the Structure tree and select **Always Visible** from the context menu.
- Uncheck the box in the Structure tree to hide the object in the Design window. The object icon is displayed in gray. You can also right-click an object in the Design window and select **Hide** (or select it in the Design window and press **Ctrl+H**) to turn the visibility of the object off.
- Check the box next to the object in the Structure tree to set the visibility of the object to the layer visibility.

If the layer visibility is on, the icon appears normally. If the layer visibility is off, the icon appears like the Solid_LayerHidden icon in the figure on the right. You cannot work with hidden objects in the Design window.

Do it faster Shift+click and Ctrl+click multiple objects to work with them as a group.

To find an object in the Structure tree

Right-click any solid, surface, plane, axis, or other object in the Design window and select **Locate in Structure Tree** to display the object in the Structure tree. If the Structure panel is not open, it is displayed.

To expand or collapse components

Click ⊞ or press + on the number pad to expand a component. Click ⊟ or press - on the number pad to collapse it. Right-click any component (including the top-level component) and select **Expand All** or press * on the number pad to expand the component and all its subcomponents.

To rename objects

Right-click an object in the Structure tree and select **Rename** or press **F2** to rename the selected object.

Once you save a file, the top-level design component's name is set to the file name and it cannot be renamed.

To move objects into components

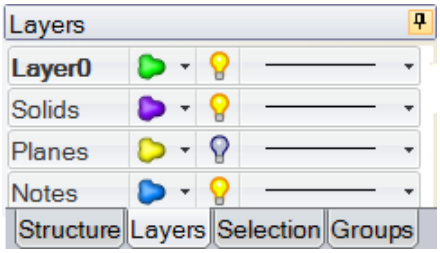
Drag any object or component to move it into another component.

To use an object as a secondary selection for a tool

Alt+click an object in the Structure tree.

For example, if you want to revolve an object, you can click to select the face to Pull, then Alt+click an axis in the Structure tree to set the revolve axis for the pull.

Working with layers



A layer can be thought of as a grouping mechanism for visual characteristics. Visual characteristics include visibility and color. Layers can be managed in the Layers panel and accessed and modified in the Layer tool in the Style ribbon group on the Display tab.

Layers are especially useful when you want to hide annotation planes. Any objects created are automatically placed on the active layer.

To create a layer

Right-click in the Layers panel and select **New**.

This layer becomes the activate layer. Any objects created are automatically placed on this layer.

To rename a layer

Right-click the layer in the Layers panel and select **Rename** or click the layer name and slowly drag to the right.

Layer0 cannot be renamed.

To delete a layer

Right-click the layer in the Layers panel and select **Delete**.

Layer0 cannot be deleted.

To place an object on a layer

1 Select the solid, surface, or component.

The Layer tool in the Style ribbon group on the Display tab displays the layer of the selected object. If no object is selected, it displays the layer on which new objects are placed. It is blank if selected objects are on different layers.

2 Select a different layer from the drop-down list to place the selected object(s) on that layer.

You can also create a new layer to place the selected object onto that layer.

To set layer visibility

1 Select a layer in the Layers panel.

2 Click  to show the objects on the layer. Click  to hide them.

If an object is located on a layer with the visibility turned off, and the object in the Structure tree is set to show visibility by layer, the object is not visible in the Design window, and cannot be acted on by the design tools. Layer visibility can be overridden in the Structure tree.

To set the visibility of layout lines and imported, DWG and DXF lines

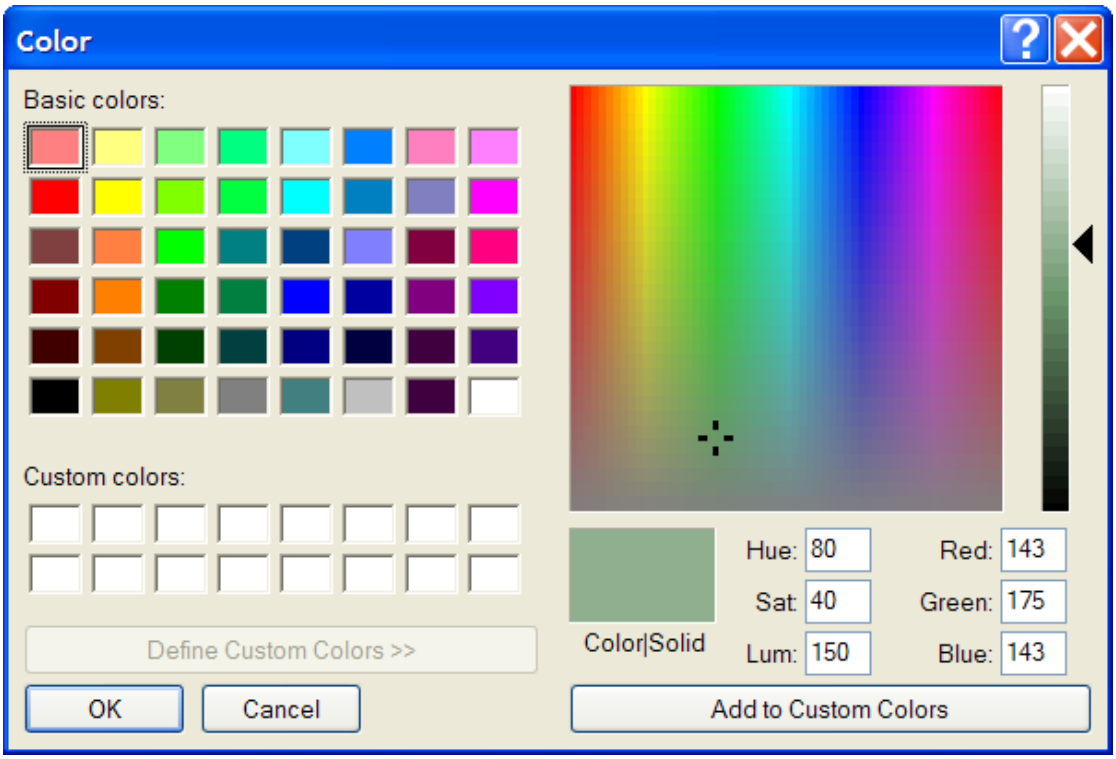
Select **Solid** or **Hidden** from the layer's line drop-down in the Layers panel.

To modify the layer color

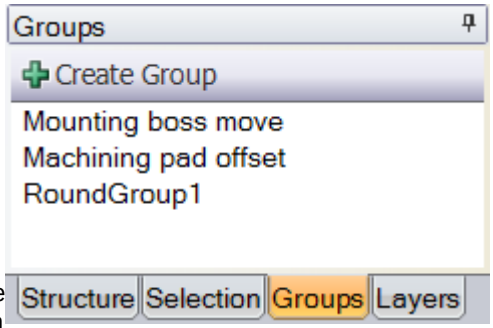
1 Select a layer in the Layers panel.

2 Select a color from the  drop-down.

You can also specify a custom color by selecting **Custom Color** and using the Color window.



Groups



You can create a group from any set of selected objects. Selection, Alt+selection, Section plane location, move anchoring, axis, and ruler dimension information is all stored with the group. You can use groups in combination with the SpaceClaim API to change these parameters, or use them as a way to indicate to others your design intentions about which sort of changes you expect them to make to your design.

A Round Group is also created each time you fill a round. You can reattach a group of rounds as long as some portion of the original edges (or faces that bordered the edges) still exists in your design.

To create a group

1. Select any set of 3D objects.
2. Click **Create Group** in the Groups panel or press **Ctrl+G**.

   The group appears in the list. A status message reports the number of faces in the group. Mouse over the group to highlight the objects in the group and click the group to select them. Properties and other information, such as the selected axis for the last Move rotation, are saved with the group.

To reattach a round

1. Right-click the Round Group in the Groups panel.
2. Select **Reattach Round** from the context menu.

To rename a named group

1. Right-click the group in the Groups panel.
2. Select **Rename** from the context menu.
3. Enter the new name for the group and press **Enter**.

Do it faster Click the group, then click it again to enter a new name.

To delete a named group

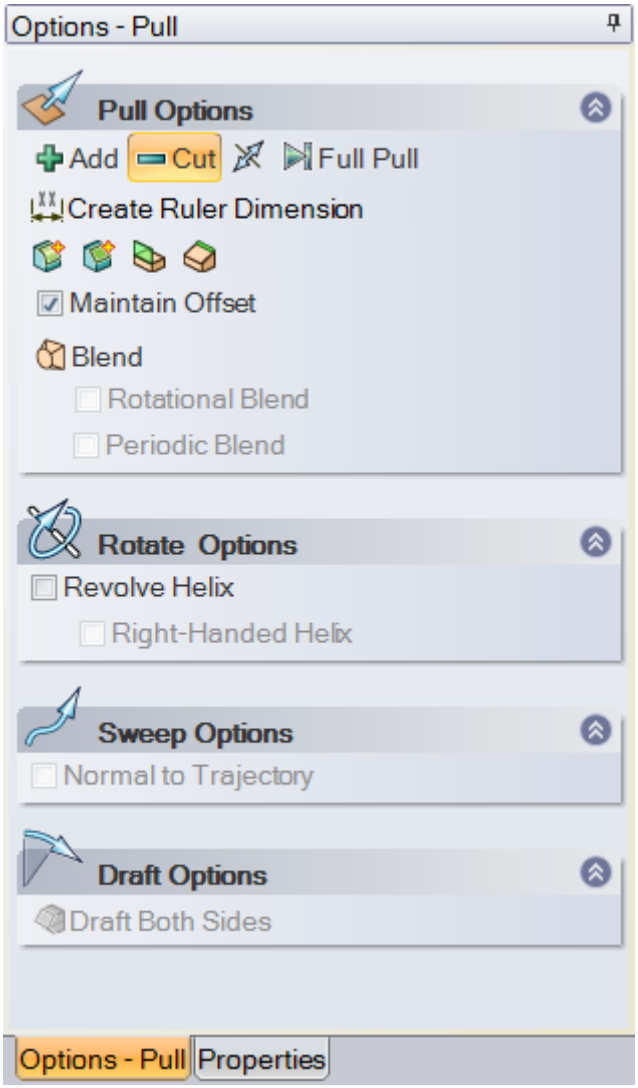
1. Select the group in the Groups panel.
2. Click **Delete Group**.

   You can also right-click the group and select **Delete Group** from the context menu.

Options panel

The Options panel allows you to modify the functions of the SpaceClaim tools. For example, when you use the Pull tool, selecting an edge and then selecting the Chamfer Edge option creates a chamfer instead of a round when you pull the edge.

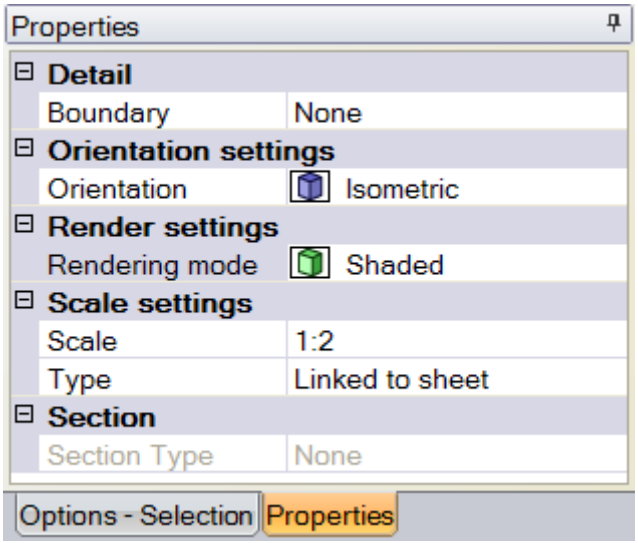
Options are enabled when the relevant geometry is selected.



Properties

Components, surfaces, and solids selected in their entirety (that is, with triple-click in the Design window, or selected in the Structure tree) display their properties in the Properties panel. The Properties panel initially appears on a Properties tab on the Options panel. When detailing your design, we recommend making Properties a separate panel and placing it above the other so that you can see object properties and tool options simultaneously.

The figure on the right shows the properties for a general view on a drawing sheet.



To modify an object's properties

Select an object in the Structure tree or right-click the object in the Design window and select **Properties**.

To create a custom property for your design

Document properties are displayed when you select the top-level design in the Structure tree. Right-click in the Properties panel and select **Add Property** to create a custom property. Expand the property to display its value. Enter a name for the property, select its type (date, Boolean, number, or string), and enter its value.

To create or specify a material for a component

1. Select a component in the Structure tree.
2. Select the Properties panel.
3. Enter the name of the material in the Material Name property.
4. Press **Enter**.
5. Enter the density of the material in the Density property.

   If you specified the density for the material elsewhere in the design, or specified it in the same SpaceClaim session, SpaceClaim displays that value in the Density property for you.

SpaceClaim shortcuts

You can use the following shortcuts to quickly access tools, tool guides, and other SpaceClaim commands. You can display these on the ribbon bar by selecting **Show tool KeyTips** in the SpaceClaim Popular options.

Bend	B
Circle	C
Escape	Esc
Fill	F
Home	H
Line	L
Pull	P
Rectangle	R
Select	S
3D Mode	D
Section Mode	X
Sketch Mode	K
Up To tool guide	U
Move	M
Spin	Drag with middle mouse button
Pan	Shift+drag with middle mouse button
Zoom	Ctrl+drag up and down with middle mouse button
Snap View	Ctrl+Shift+middle mouse button
Zoom Extents	Z

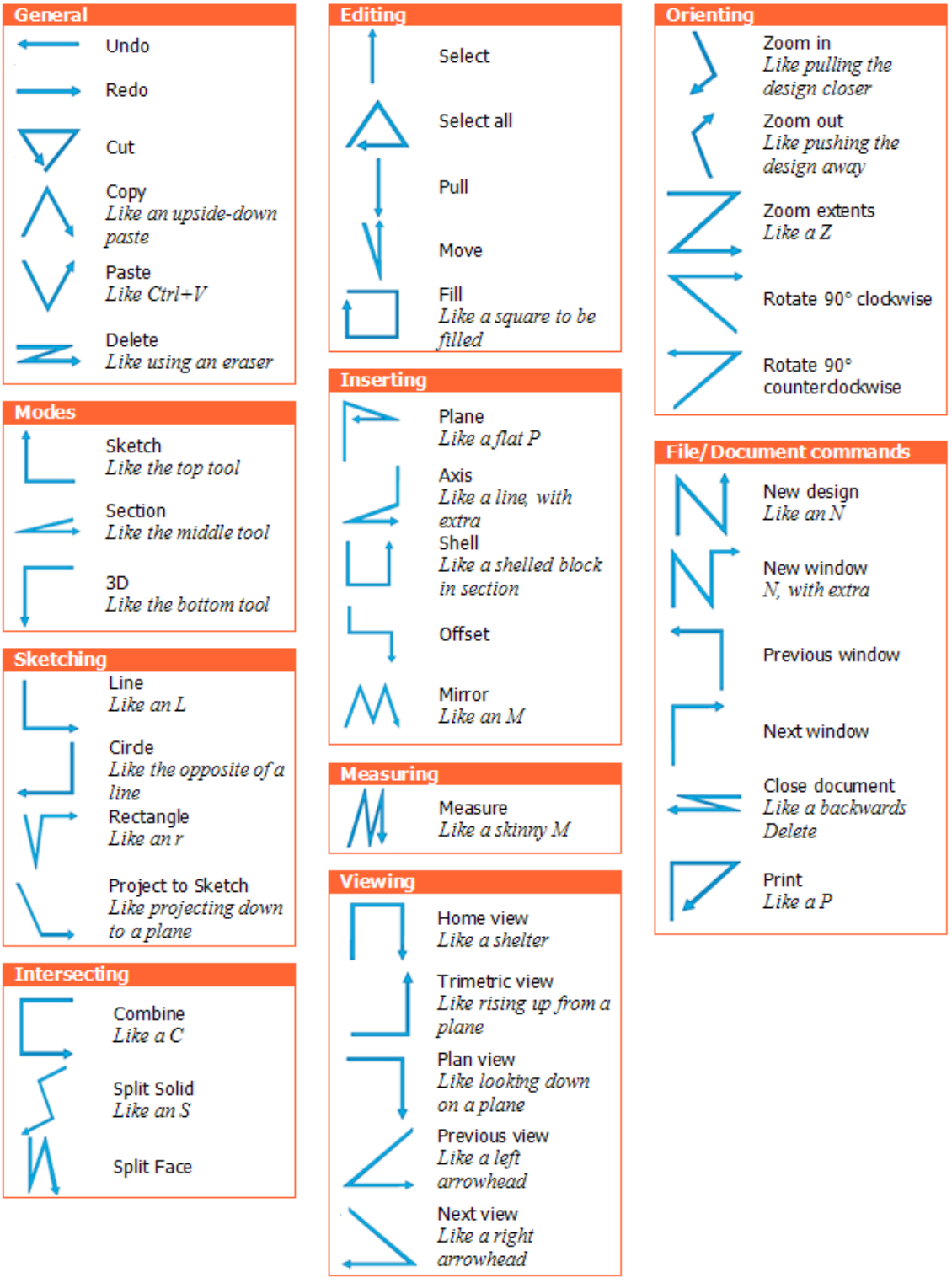
Application menu	Alt+F
Close document	Ctrl+F4
Complete	Ctrl+Enter
Copy	Ctrl+C
Copy Special	Ctrl+Alt+C
Create Group	Ctrl+G
Cut	Ctrl+X
Cut Special	Ctrl+Alt+X
Delete	Del
Detach Face	Ctrl+D
Exit	Alt+F4
Font	Ctrl+Shift+F
Font Size	Ctrl+Shift+P

Invert Selection	Ctrl+Shift+I
Display next Design window	Ctrl+Tab
Display previous Design window	Ctrl+Shift+Tab
Move Sketch Grid in	Ctrl+right arrow
Move Sketch Grid out	Ctrl+left arrow
Display the previous view of your design	Alt+left arrow
Reapply the last view to your design	Alt+right arrow
New	Ctrl+N
Open	Ctrl+O
Paste	Ctrl+V
Print	Ctrl+P
Print Preview	Ctrl+F2
Redo	Ctrl+Y
Save	Ctrl+S
Select All	Ctrl+A
Bold Text	Ctrl+B
Italicize Text	Ctrl+I
Underline Text	Ctrl+U
Toggle Visibility	Ctrl+H
Undo	Ctrl+Z
Zoom In	Ctrl+ +
Zoom Out	Ctrl+ -
Expand entire node in Structure tree	* on number pad
Expand selected node in Structure tree	+ on number pad
Collapse selected node in Structure tree	- on number pad

Mouse gestures

You can use mouse gestures in the Design window as shortcuts to common actions and tools.

You can make the following gestures while holding down the right mouse button. To cancel a gesture, pause for one second.



SpaceClaim objects

The SpaceClaim interface describes objects slightly differently than other modeling software you might be familiar with.

Document

A SpaceClaim .scdoc may contain any combination of design versions, associated drawing sheets, and 3D markup slides.

Design

A design is a 2D or 3D model, which contains at least one top-level component.

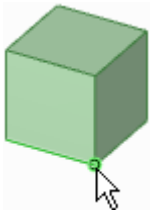
Component

A component consists of any number of objects, such as solids and surfaces. You can think of a component as a "part." A component can also contain any number of sub-components. You can think of a hierarchy of components and subcomponents as an "assembly."

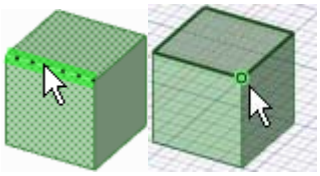
Object

An object is anything recognizable by SpaceClaim tools. For example, 3D objects include vertices, edges, faces, surfaces, solids, layouts, planes, axes, and origins. 2D objects include points and lines.

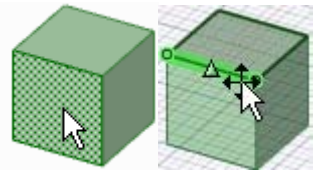
Examples of some object types are shown below:



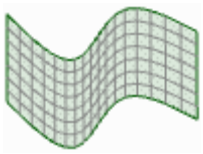
Vertex



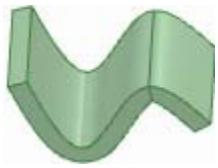
Edge in 3D mode and the same edge in Section mode



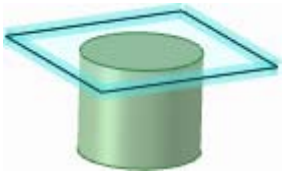
Face in 3D mode and the same face in Section mode



Surface

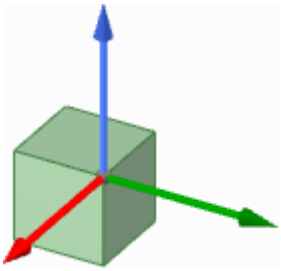


Solid



Plane

Axis



Origin

Body

In SpaceClaim, a body is a solid or surface.

Mating condition

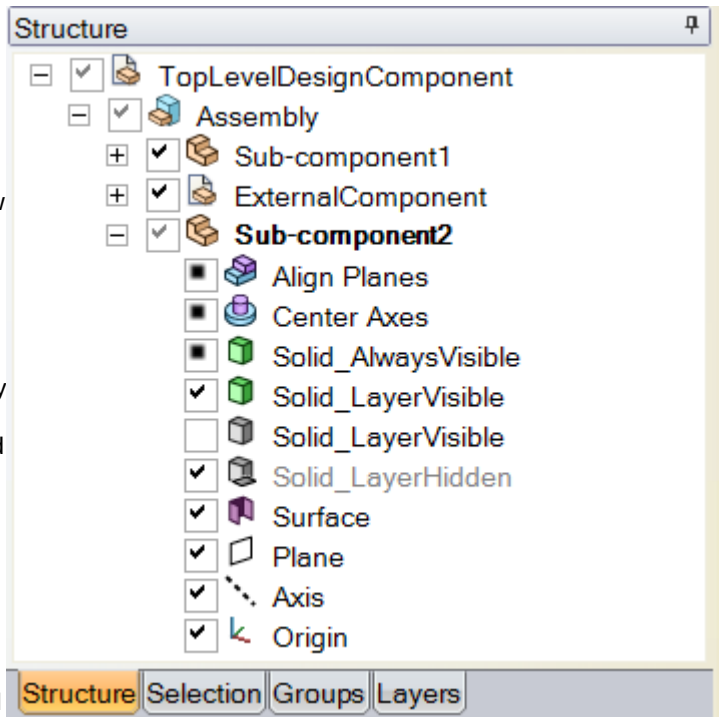
Components are aligned using mating conditions.

Curve

An imported Curve file.

Working with components

The Structure tree contains the Structure tree, which shows you each of the objects in your design. You can quickly show or hide any component using the checkbox next to the object's name. You can expand or collapse the nodes of the tree, rename objects, create, modify, replace, and delete objects, create components, copy components, make components independent, open a component in a new design window, set the component to be a sheet metal component, make a component active, make a component independent, delete, rename, or display properties.



Offset, mirror, and shell relationships stay with a solid when it is moved to another component, unless the relationship would link two components when it is moved.

The top-level design (called **StructureTree** in the image on the right) is also a component.

If you are working with a single instance of an external component, make that instance independent to prevent your changes from being made to the external component file. Once you make a copied sub-component independent, you can modify it without changing any of the other instances of that sub-component. Or you can modify one of the other instances to change all the copied sub-components except the one you made independent.

If your design includes multiple copies of an external component, making one of them internal does not affect the other copies. Making another copy of the same external component internal creates a second instance of the same internal component.

Do it faster	Shift+click and Ctrl+click multiple objects to work with them as a group.

About lightweight components

When you insert an external file into your design, if you have the Enable lightweight assemblies advanced SpaceClaim option enabled, only the component's graphic information is loaded. This allows you to quickly view the component with the Orient tools and load the geometry information when you are ready to work with it in SpaceClaim.

To create a component

Right-click top-level design (or another component) in the Structure tree and select **New Component** from the context menu to create a new component or sub-component.

To copy a component

1 Select a component and click the Copy tool or press **Ctrl+C**.

2 Select the component under which you want to create a copy and click the Paste tool or press **Ctrl+V**.

An instance of the component is created, which is linked to the original component. All changes made to the copied component are also made to the original unless you make the copy independent.

To insert a component or assembly

1 Select the Insert File tool  in the Insert ribbon group.

2 Navigate to the component and double-click to insert it.

The component is placed in the center of the workspace and its sub-components (if it is an assembly) appear in the Structure panel.

To activate a component

Right-click the component and select **Activate Component** from the context menu.

If the component is lightweight, it is also loaded. Any new objects are created within this component. A component must be active before you can cut or copy it for pasting.

To externalize a component

1 Right-click the component and select **Open Component** from the context menu.

The selected component appears in a new Design window.

2 Select **Save As** from the Application menu to save the component as a separate file. (DO NOT check the Save as copy box.)

The icon in the Structure tree of the original design changes to reflect that the component is now external.

To copy an external component into your design

Right-click the component and select **Use Internal Copy** from the context menu.

The selected component is copied into your design. Any changes you make to the component do not affect the original, external file.

To load a lightweight component

Right-click the component and select **Load Component** from the context menu.

The component and all its subcomponents' geometry information is loaded, and you can work on the components with any SpaceClaim tool.

To make a component independent

Right-click the inserted, dependent component in the Structure tree and select **Make Independent** from the context menu.

The icon in the Structure tree changes, and the component is renamed <OriginalComponentName>2.

If your design consists of multiple instances of the same external component, and that external component also contains multiple instances of another external subcomponent, making the subcomponent independent

makes both the subcomponent and its parent component independent. Any parent component in the tree, all the way up to the top-level design component, will also be made independent.

To create or specify a material for a component

1. Select a component in the Structure tree.
2. Select the Properties panel.
3. Enter the name of the material in the Material Name property.
4. Press **Enter**.
5. Enter the density of the material in the Density property.

   If you specified the density for the material elsewhere in the design, or specified it in the same SpaceClaim session, SpaceClaim displays that value in the Density property for you.

Getting help

We offer several resources you may find helpful when using SpaceClaim.

Online help

Detailed tooltips are provided for each tool within SpaceClaim. You may find that a careful reading of the tooltip provides all the information you need to use the tool.

If you need more information, click  in the tab bar or press **F1** while the tooltip is open to display the online help for the tool. The online help provides step-by-step instructions, animations, and examples.

Customer support

SpaceClaim is committed to providing you with every opportunity to communicate directly with us so you benefit by helping us continuously improve our products, services, and build our growing community. We want to help you apply SpaceClaim efficiently to solve your product development problems.

SpaceClaim's annual lease license includes subscription services, so you can be confident that you will always have access to expert technical resources and the latest software. Subscription services provide the latest product releases and upgrades, direct access to SpaceClaim technical resources via phone, email, and chat, and personalized access to our customer portal: MySpaceClaim.com.

To contact customer support:

- Email: support@spaceclaim.com
- Mail: SpaceClaim, 150 Baker Ave Ext., Concord, MA 01742
- Toll free phone: 1.800.636.4215
- Local/International phone: 1.978.482.2281
  Phones are staffed 8:30am - 6pm EST, Monday-Friday, excluding US national holidays

When contacting customer support, the following information may be needed to properly diagnose your issue:

- SpaceClaim version number
- Environment details (operating system, hardware, graphics card)
- Brief description of your issue
- Detailed steps to reproduce the issue
- Related files (journal files, data files)

Gathering this information before contacting customer support could help us find a resolution more quickly.

MySpaceClaim.com

MySpaceClaim.com is a personalized web portal for easy access to everything SpaceClaim. The web portal provides the ability to:

- Gain access through a unique user name and password

- Directly download SpaceClaim software, including purchased new products, updates, and upgrades
- Search the product knowledge base for answers to common questions
- Learn from self-paced training tutorials
- Check on the status of your license
- Submit a new idea to influence product development
- Find the status of outstanding issues

To access MySpaceClaim.com, select the **Login** link at the top of the SpaceClaim.com home page.

To check for updates

When you open the SpaceClaim application, it automatically checks for updates. If updates are available, a message appears in the status bar.

1. Select **SpaceClaim Options** from the Application menu and click **Resources**.
2. Click **Check for updates** to check for recent software updates.

To download models, contact us, or find the version number

1. Select **SpaceClaim Options** from the Application menu and click **Resources**.
2. Click:
   - **Get models** to download models from the SpaceClaim model library
   - **Contact us** to contact us through our website
   - **About** to find the version number of the SpaceClaim software you are running.

Designing

The tools you use for 2D and 3D sketching and editing are found in SpaceClaim's Design tab. With the design tools, you can sketch in 2D, generate and edit solids in 3D, and work with assemblies of solids.

In SpaceClaim, there are three modes you can use to design: Sketch, Section, and 3D mode. You can switch between these modes at any time.

When creating designs, you will use the following tools most often:


Use the Select tool to select 2D or 3D objects in your design for editing. You can select vertices, edges, axes, faces, surfaces, solids, and components in 3D. In 2D, you can select points and lines. You can also use this tool to change the properties of recognized or inferred objects.


Use the Pull tool to offset, extrude, revolve, sweep, draft, and blend faces; and to round or chamfer corner edges.


Use the Move tool to move any single face, surface, solid, or component The behavior of the Move tool changes based on what you have selected. If you select a face, you can pull or draft it. If you select a solid or surface, you can rotate or translate it.


Use the Combine tool to merge and split solids and surfaces.


Use Section mode to create and edit designs by sketching on and editing any cross-section through the design.

Design tools are grouped into the following ribbon groups:

Clipboard Cut, copy, and paste 3D objects.

Orient Spin, pan, and zoom your design. You can also select or create a view.

Sketch Create and edit lines, arcs, splines, rectangles, circles, and points in 2D.

Mode Switch between sketching in 2D, editing a cross-section, and creating and editing objects directly in 3D

Edit Edit 2D and 3D geometry.

Intersect Merge or split solids and faces.

Insert Create relationships between the components in your design.

Assembly Import other components and orient them within your design.

Analysis Find lengths, angles, and volumes.

Press and hold **Esc** to cancel out of any design action that is taking too long.

When using any tool guide, you can click an empty point in the Design window to clear the tool guide selection and return to the previous action within the tool.

2D and 3D design modes

In SpaceClaim, there are three modes you can use to design: Sketch, Section, and 3D. These modes make it clear when you are working in 2D vs. 3D, and clarify the difference between working in cross-section (where you can alter 3D objects) and sketching (where you cannot alter 3D objects)

To switch modes

Click a tool in the Mode ribbon group.

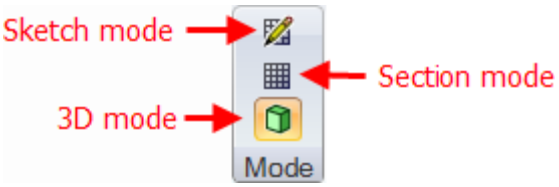
Detailed instructions

1. Click a tool in the Mode ribbon group.

   You can also right-click an empty area of the Design window and select **Sketch Mode**, **Section Mode**, or **3D Mode** from the context menu, or press **K** (for Sketch mode), **X** (for Section mode), or **D** (for 3D mode).

   If you are sketching, you can also click **Return to 3D mode** in the Sketch mini-toolbar.
2. If you are switching from 3D mode to one of the other modes, select a plane to sketch on or create the cross-section.

Mode ribbon group



The Mode ribbon group contains the following tools:

Sketch mode displays the sketch grid, so you can use any of the sketch tools to sketch in 2D.

Section mode lets you edit solids and surfaces by working with their edges and vertices in cross-section. You can also use all the sketch tools in Section mode to create and edit solids and surfaces in cross-section.

3D mode lets you work directly with objects in 3D space.

Editing in cross-section

Use Section mode to edit solids by working with their edges and vertices in cross-section. Since you are working with a cross-section of geometry in section mode, pulling a line pulls a face, and pulling a vertex pulls an edge. For example, to rotate a face around an edge, select the line that represents the face, Alt+click the vertex that represents the edge, and pull. Moving a sketched line in Section mode does not move the solid it is sketched on. You must move a section line (a line that represents a face) to modify a solid in Section mode.

Hatching is used to show the intersection of the cross-section plane and a solid. Arc centers are shown as small cross marks. Hatching appears bolder inside faces to indicate what is shown in a cross-section view. (See Examples, below).

You can use the following tools: Select, Pull, Move, Combine, Split Body, Shell, Offset, Fill, and all sketch tools. Use the Select tool to edit spline faces (represented by a spline in cross-section). You can also cut, copy, and paste. We recommend that you clip the scene above the grid to enhance the visibility of the cross-section

To edit in cross-section

1. Select or de-select options based on whether you want to maintain and view relationships while you edit in cross-section.
2. Select the face you want to use to create the cross-section.

   If you are in a drawing sheet with cross-section views, you do not have to choose a face, as the plane of the drawing sheet is automatically used as the section plane.
3. (Optional) Move or rotate the cross-section grid and click the Section tool when you are finished.

4 Click and drag the edges and vertices of the cross-section to edit them.

You can also bend edges with the Bend tool, and pull section points (edges) and section lines (faces) with the Pull tool. If you set the **Auto-extrude/revolve sketches in Section mode** Advanced SpaceClaim option, sketch made with the sketch tools are automatically extruded or revolved to form surfaces and solids when you begin the sketch on the edge of an existing surface or solid. If you do not begin the sketch on an existing edge, you are switched to Sketch mode.

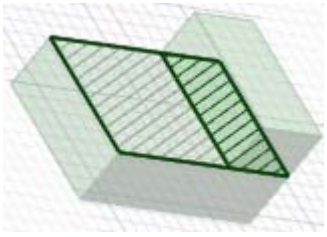
We recommend zooming into your design so that it is easier to select the correct entity. For example, if you are trying to select an edge, but your design appears very small in the Design window, it is possible to accidentally select a midpoint or endpoint of the edge instead. Moving the midpoint or endpoint of an edge will not be reflected by a mirrored entity.

Options

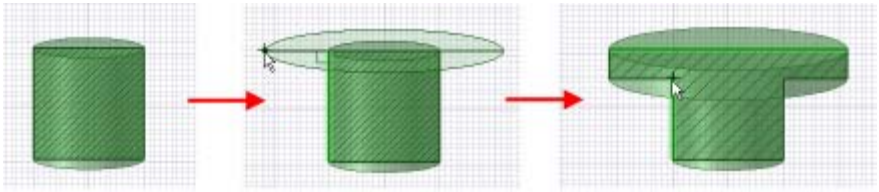
The following options are available in the Section tool:

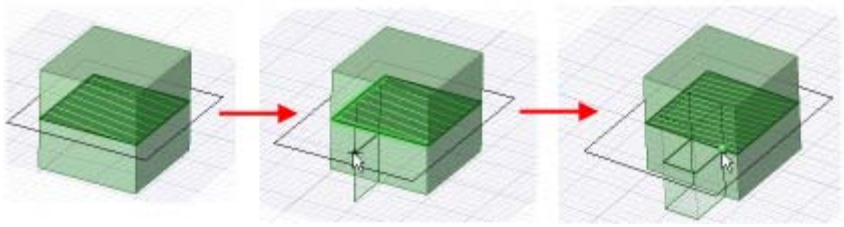
Maintain Mirror	Select this option to maintain the influence of mirrors in your design while editing.
Maintain Offset	Select this option to maintain the influence of baselines in your design while editing. Baseline faces are shown as blue edges when viewed in cross-section.

Examples



Bold hatching indicates the hatching that would be shown on a drawing sheet cross-section view





Extruding while sketching in Section mode

Cutting, copying, and pasting

Select a tool from the Clipboard ribbon group to cut, copy, or paste any 2D or 3D object. You can use these tools any time, even when you are designing with other 2D or 3D tools.

To cut an object

1. Select the object.
2. Select the Cut tool from the Clipboard ribbon group.

   You can also press **Ctrl+X** or right-click in the Design window and select **Cut** from the context menu. If you cut the face of a solid, it is converted to a surface.

To copy an object

1. Select the object.
2. Select the Copy tool from the Clipboard ribbon group.

   You can also press **Ctrl+C** or right-click in the Design window and select **Copy** from the context menu.

To paste an object into the Design window

1. Cut or copy the object.
2. Select the Paste tool from the Clipboard ribbon group.

   You can also press **Ctrl+V** or right-click in the Design window and select **Paste** from the context menu.

To paste an object onto a face

1. Select all the faces of the object.
2. (Optional) Alt+click the face of the object you want to attach to the face.

   The face is highlighted in blue.
3. Cut or copy the selection.
4. Click the face onto which you want to paste the object.
5. Paste the object.

   The object is pasted on the face at the location you clicked. The blue face is attached to the clicked face.

To paste a round onto another edge

1. Select the face of the round.
3. Cut or copy the face.
4. Click the edge onto which you want to paste the round.
5. Paste the round.

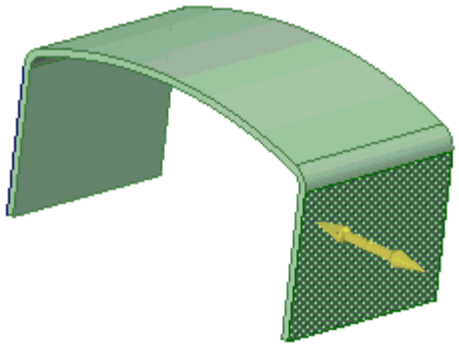
To delete an object

1. Right-click the selected object (or set of objects).
2. Press **Delete**.

   If you want to delete something and fill the gap with neighboring geometry, or create faces in the gap, you should use the Fill tool.

   If you delete the face of a solid, it is converted to a surface.

Dimensions



You can dimension every element in your design, from lines in sketches to faces of solids. In SpaceClaim, dimensions are not constraints. Rather, they are tools for precise control during the creation or modification of a design. In SpaceClaim, if you do want to save a dimension with your design, use the Ruler Dimension option when pulling or moving. You can save the ruler dimensions as Groups for later edits.

Whenever dimension fields appear, you can press the spacebar or click on them to enter a value, and press Tab to switch between fields. You can enter expressions as dimension values.

The following expression elements are available:

- Infix (dyadic) operators: **+ - * / ^**
- Prefix (monadic) operators: **+ -**
- Functions: **sin cos tan asin acos atan sqrt log log10 exp**
- Constants: **pi e root2 root3**
- Units: **m cm mm yd ft in ' " deg rad**

Normal precedence rules apply:

1 + 2 * 3 ^ 4 = 1 + (2 * (3 ^ 4)) = 163

Parentheses are required for expression arguments and optional for simple arguments:

- sqrt 2 = sqrt(2) = 1.4142...
- sqrt 2*2 = (sqrt 2) * 2 = 2.8284...
- sqrt(2*2) = 2

Missing operators are inferred:

- 1 1/2 = 1 + 1/2
- 1'6" = 1' + 6"
- 1ft 6in - 17in = 1ft + 6in - 17in
- 1 2 3 4 5 = 1 + 2 + 3 + 4 + 5 = 15
- (1)(2)(3)(4)(5) = (1) * (2) * (3) * (4) * (5) = 120
- 2(1 + 2) = 2 * (1 + 2) = 6
- sqrt 2 sqrt 2 = sqrt 2 * sqrt 2 = 2
- 4(4atan(1/5) - atan(1/239)) = 4 * (4 * atan(1/5) - atan(1/239)) = pi

Units are applied to previous terms if units were not specified and are applied to subsequent terms unless you override them:

- 1 + 1cm = 1cm + 1cm
- 1cm + 1 = 1cm + 1cm
- 1cm + 1 + 1mm = 1cm + 1mm + 1mm
- 1cm + 1 1/2 mm = 1cm + 1mm + 1mm / 2

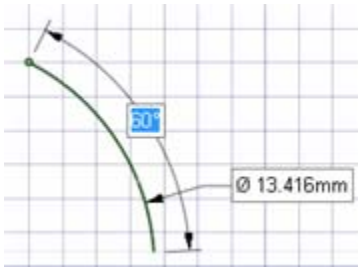
Trigonometry functions work in radians by default, but you can enter degrees:

sin(45 deg)

Numbers support standard form, but e is a built-in constant:

- 2e2 = 200
  2e 2 = 2 * e * 2 = 10.873...
- 2e-2 = 0.02
  2e – 2 = 2 * e – 2 = 3.436...
- 2e1 = 20
  2e = 2 * e

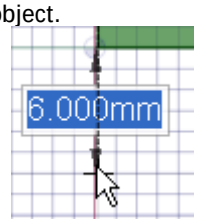
To dimension a sketch line during creation



1. Press the spacebar (or just type) to enter a value in the highlighted field.
2. Press **Tab** to switch between dimension fields.
3. Repeat step 2 until you have entered all the dimensions.
4. Press **Enter** to accept the values and return to sketching.

   The dimensions persist until you select another tool or begin drawing another sketch object.

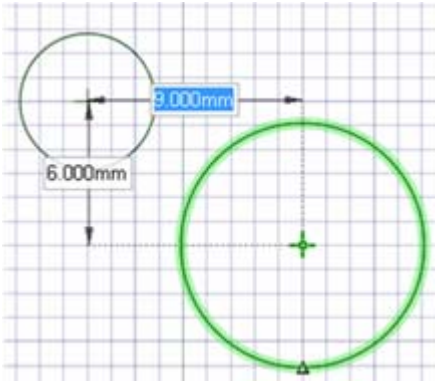
To dimension the start or end point of a sketch line from another point in your sketch



1. Hover the mouse over the point from which you want to dimension.
2. Press **Shift**.

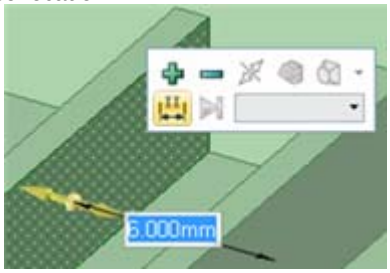
   As you move your mouse around the sketch grid, a dimension will appear from the point you indicated to the mouse location.
3. Press the spacebar (or just type) to enter a value in the highlighted field.
4. Press **Tab** to switch between dimension fields.
5. Repeat step 4 until you have entered all the dimensions.
6. Press **Enter** to accept the values and place the point that begins or ends your line.

To dimension existing sketch lines



1. Click the Select tool.
2. Select the sketch object you want to change.
3. Dimension the item's size or location by doing one of the following:
   - Press the spacebar (or just type) to enter a value in the highlighted field.
   - Drag the selected item to change its size or location.
   - Hover over a point in your design and press **Shift** to dimension between the selected object and that point.
   - Press **Shift** while dragging to dimension from the current mouse location.

To dimension while moving or pulling



1. Select a direction for the move or pull.
2. Press the spacebar (or just type) to enter a value in the highlighted field.
3. Press **Tab** to switch between dimension fields.
4. Repeat step 3 until you have entered all the dimensions.

5 Press **Enter** to accept the values and move or pull the selected object the distance you entered.

To create a ruler dimension

1 Select the faces or edges whose location you wish to specify.

2 Select a direction for the dimension.

3 Select **Create Ruler Dimension** from the Options panel or right-click and select it from the mini-toolbar.

The start point of the dimension is set as the location of the Pull arrow or Move handle.

4 Click an object to set the endpoint of the dimension.

Use the scroll wheel if multiple objects appear at the same point in the Design window.

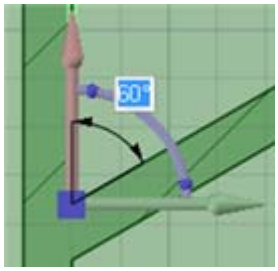
5 Enter a value.

6 Press **Enter** to accept the value and complete the move or pull.

Press **Esc** to hide the ruler dimension.

You can make more than one change per ruler dimension.

To create an angular ruler dimension



1 Select the Move tool and switch to Section mode.

2 Select the section line (that represents a face) that you want to rotate.

3 (Optional) Anchor the Move handle to the object around which you want to rotate by dragging the center sphere or using the Anchor tool guide.

4 Select the rotational axis of the Move handle.

5 Select **Create Ruler Dimension** from the Options panel or right-click and select it from the mini-toolbar.

An angular dimension indicator appears from the red linear axis of the Move handle.

6 Select the end reference for the angular dimension.

7 Enter a value for the dimension.

Detaching in 2D and 3D

You can detach individual pieces of a sketch, or detach objects or faces in 3D. You can detach protrusions to move them with the Move tool's Detach First option.

To detach in 2D

Alt+drag with the Select tool to detach the selected item when sketching. Use the 2D Move tool to detach items and move them.

To detach in 3D

1 Click the Select tool in the Edit ribbon group.

2 If you want to detach an object, Ctrl+click all its faces to select them.

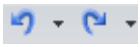
3 Right-click the object and select **Detach** from the context menu.

Undoing and redoing your actions

SpaceClaim stores all your actions from the moment you open the SpaceClaim application until you close it. This includes the use of all tools in all tabs, opening and closing files, loading and activating components, and changing settings. Every action is recorded and can be undone and redone.

The undo list is limited to 20 steps by default, but you can modify this number. We recommend that you set it to at least 50.

To undo or redo a tool action

Click the Undo and Redo tools  in the Quick Access toolbar or press **Ctrl+Z** to undo and **Ctrl+Y** to redo.

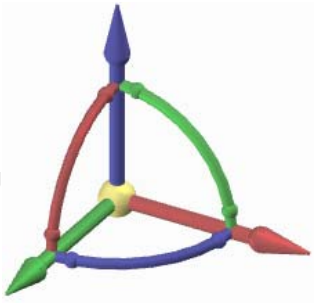
You can undo and redo actions until you have undone or redone every action in your session, or you can jump to a particular action by selecting that action from the Undo and Redo menus. If undoing an action will open or close a document or switch to a new Design window, a confirmation window is displayed.

When you undo a tool action, the view is also changed to the view you used to perform that action.

To undo or redo a view

Click the **Previous View** and **Next View** tools  on the status bar to undo and redo your design view changes.

Move handle



You will use the Move handle to move objects in 2D and 3D. The Move handle allows you to translate and rotate objects. You use the Move handle by clicking the axes of the Move handle and dragging to move the selected object.

When you select the object you want to move, and click one of the Move tools, SpaceClaim guesses at the anchor point and orientation of the Move handle. If either of these is incorrect, you can change them.

To translate objects using the Move handle

1. Click the axis that is aligned with the direction you want to move the selected object.
2. Drag in the direction of the axis to move the object.

To rotate objects using the Move handle

1. Click the rotational axis that is aligned with the direction you want to rotate the selected object.
2. Drag in the direction of the axis to move the object.

To realign the axes of the Move handle

You can realign the Move handle in the following ways:

- Drag the small balls on each rotation axis of the Move handle to reorient it. While dragging, you can also press the spacebar to enter the angle directly.
- Drag the center ball of the Move handle (or use the Anchor tool guide) to place it on another solid, face, edge, or vertex. If you use this method, keep in mind that the selected object is still the one that is going to move.

- Alt+click a point or line (or use the Move Direction tool guide) to orient the closest linear axis toward that point or along the line.

Sketching

Sketching is useful if you want to create a region that can be pulled into 3D. If you want to create a 2D layout, and have no immediate need to generate 3D objects from the lines in the layout, then you should create a layout.

Use the sketch tools to sketch shapes in 2D. When you exit the sketch, regions are formed by intersecting lines. These regions will become solids and lines become edges when you pull your sketch into 3D with the Pull tool. Even when pulled into 3D, a region can be decomposed back into its sketched lines for further editing as long as any remnant of the lines is still unused in 3D.

To use any of the sketch tools to sketch in 2D, you must first display the sketch grid. If you have a planar surface highlighted, and press a sketch tool shortcut (such as **L** for the Line tool), you can mouse over planar surfaces in the design to highlight surfaces for the sketch grid. (Press **Esc** while in this state to return to the Select tool in 3D mode.) You can adjust the units and spacing of the grid, and we recommend that you fade the scene under the grid to enhance the visibility of your sketch.

While you are sketching, you may need to orient your design. If you use the Spin, Pan, or Zoom tools to reorient it, click them again or press Esc to continue sketching where you left off.

Depending on your options, sketching in Section mode may automatically extrude your sketch to 3D. The extrusion depth is set to 10 times the spacing of your sketch grid. You can dimension this depth for any extruded sketch by entering a value in that dimension field.

To sketch

1. Select any sketch tool.
2. Choose where you want to sketch.
3. (Optional) Move or rotate the sketch grid.
4. Draw with the tool.
5. Repeat steps 1 through 4 until you are finished sketching.

Detailed instructions

1. Select Sketch mode  from the Mode ribbon group.
2. Choose where you want to sketch.

   Mouse over the planes and planar faces in your design to preview the location and orientation of the sketch grid.

   If you previously selected a set of references that define a plane, the sketch grid is placed on the defined plane. The sketch grid mini-toolbar allows you to switch from sketching on one plane to sketching on another without leaving the sketch tools.

   The sketch grid is placed automatically if an object in the active component is already selected.

   a. If the sketch grid is currently displayed, click **Select New Sketch Plane**  in the mini-toolbar, or right-click and select **Select New Sketch Plane** from the context menu.

   b. Mouse over any existing geometry to display existing planes.

   c. Click to select the highlighted plane and display the sketch grid. Any vertices or edges on the plane are drawn in the current layer color and bolded.

   d. (Optional) Click **Plan View**  in the mini-toolbar or in the Orient ribbon group to view the sketch grid head-on.
3. (Optional) Move or rotate the sketch grid.

   a. (Optional) Select any points, lines, or curves that you want to move with the sketch grid.

b Click **Move Grid** in the mini-toolbar.

c Use the Move handle to move or rotate the sketch grid.

4 Select any sketch tool from the Sketch ribbon group.

You can draw points, lines, tangent lines, construction lines, rectangles, three-point rectangles, circles, three-point circles, construction circles, ellipses, splines, tangent arcs, three-point arcs, sweep arcs, and polygons using the sketch tools. You can also create sketch lines by projecting the edges of 3D solids onto the sketch grid.

SpaceClaim's sketch tools also let you split, trim, and offset lines, as well as create corners and rounded corners.

5 Draw with the tool.

Mousing over the sketch grid snaps to points based on your snapping options. Press **Shift** to turn snapping on and off.

While mousing, SpaceClaim also provides extension lines when you are parallel to an edge or perpendicular to an endpoint. For certain drawing tools, it displays indicators of tangency, line midpoint, line endpoint, squares, and golden rectangles.

All tools let you enter dimensions while sketching. In some tools, you can press **Shift** at a reference point to see dimensions from that point to the cursor.

When you are done sketching with the tool, you can:

- Click another sketching tool.
- Click **Return to 3D Mode** in the mini-toolbar or **3D Mode** in the Mode group in the ribbon bar to pull your sketch to 3D.
- Press **Esc** or click the Select tool in the Edit ribbon group to edit the sketch.

To edit a sketch

1 Click the Select tool. (You can also press **Esc** if you are in a sketching tool.)

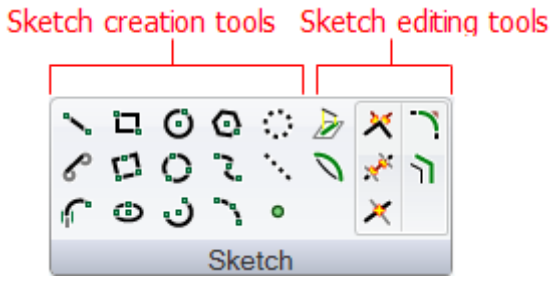
2 Click and drag the line or point you want to edit.

Alt+click and drag if you want to detach the line or point before moving it.

Ctrl+click and drag to create a copy.

Enter a value to dimension the move.

Sketch ribbon group



The Sketch ribbon group contains the following sketch creation tools:

Use the Line tool to sketch lines in 2D.

Use the Tangent Line tool to sketch lines tangent to any curves in your design.

Use the Tangent Arc tool to sketch an arc tangent to any single curve or line in your design.

Use the Rectangle tool to draw a rectangle along the axes of the sketch grid.

Use the Three-Point Rectangle tool to quickly sketch a rectangle at any angle in 2D.

Use the Ellipse tool to sketch an ellipse in 2D.

Use the Circle tool to sketch a circle in 2D when you know the location of the circle's center and radius, diameter, or a point on the circle's edge.

Use the Three-Point Circle tool when you don't know the center of the circle, but you know where the edge of the circle must be. This tool works with any combination of free points, known points, or tangent attachments.

Use the Sweep Arc tool to create an arc with a known center and endpoints.

Use the Polygon tool to sketch a polygon with up to 32 sides.

Use the Spline tool to sketch splines in 2D. A spline is a continuously curved line, without sharp boundaries (that is, without vertices).

Use the Three-Point Arc tool to create an arc by specifying its start and end points, and the radius or chord angle.

Use the Construction Circle tool to draw circles that help you create an accurate sketch. Construction circles cannot be pulled into 3D, and are removed when you select a new sketch plane or exit sketching.

Use the Construction Line tool to draw lines that help you create an accurate sketch. These lines become axes in 3D mode.

Use the Point tool to sketch points in 2D.

Use the Project to Sketch tool to copy an edge, vertex, or note text to 2D lines and points that you can manipulate with the sketch tools.

The Sketch ribbon group contains the following sketch editing tools:

Use the Bend tool to bend straight lines and edges to form an arc. You can also use the Bend tool to adjust the radius of arcs and arced edges. Bend works on straight lines when you are sketching, when you are editing in cross-section, and when you are editing blend sections with the Edit as Blend tool.

Use the Create Corner tool to trim back or extend two lines so that they meet at a corner.

Use the Trim Away tool to delete any line portion bounded by an intersection with a line or edge.

Use the Split Line tool to split one line with another line or point.

Use the Create Rounded Corner tool to trim back or connect two intersecting lines or arcs so that they meet with an arc tangent at both ends.

Use the Offset Line tool to create an offset of any line in the grid plane.

Sketching mini-toolbar

While you are sketching, the mini-toolbar provides quick access to the following actions:

Click **Return to 3D Mode** to switch to the Pull tool and pull your sketch into 3D. Any closed loops will form surfaces or faces. Intersecting lines will split faces.

Click **Select New Sketch Plane** to select a new face to sketch on.

Click **Move Grid** to move or rotate the current sketch grid with the Move handle.

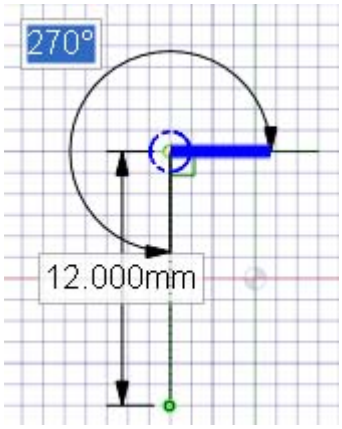
Click **Plan View** for a head-on view of the sketch grid.

Options

The following options are available in every sketching tool:

Snap to grid Select this option turn snapping on or off while sketching.

Editing a sketch



You can edit the dimensions of sketched lines, arcs, and points using the Select tool. Tangency is maintained when you edit a sketch.

To edit a sketch

1. Click the Select tool. (You can also press **Esc** if you are in a sketching tool.)

   Select the **Select Reference Curve** tool guide to dimension from another object, similar to the Shift+hover functionality in other tools.
2. Click and drag the line or point you want to edit.

   Alt+click and drag if you want to detach the line or point before moving it.

   Ctrl+click and drag to create a copy.

   Enter a value to dimension the move.

To edit the length and angle dimensions of a line

1. Click the endpoint of a line or arc to edit the length or diameter and angle or chord angle dimensions of the line or arc.
2. Mouse over the opposite endpoint of the line or arc to display a blue circle.
3. Drag to move the reference point for the dimension's length.
4. Enter the dimension for the length from the endpoint of the line to the new reference point.
5. Mouse over the line showing the end of the angle reference to highlight it in blue.
6. Drag the blue line to move the reference for the angle dimension.

   If you move the angular reference 360°, the angle measurement is changed from clockwise to counterclockwise.
7. Enter the dimension to adjust the angle of the line from the new reference line.

To edit the distance from another point

1. Click the midpoint of a line, the center of an arc, or any sketched point to display a dimension to a reference point.
2. Mouse over the reference point, then drag the blue circles to the location you want to dimension from.
3. Click to set the new reference location.

   To skew the X-Y lines, click the outer blue circle and enter the skew angle.

To change the properties of a sketch object

1. Click the Select tool. (You can also press **Esc** if you are in a sketching tool.)
2. Select the sketch object to display its properties in the Properties panel.

   You may need to adjust the width of the Properties panel or the columns within the properties panel. Place your mouse over the vertical column line and drag to widen the column. Place your mouse over the border of the panel and drag to widen the panel.
3. You can:
   - Select a new color from the Color property to color the object.
   - Switch between lines and construction lines by selecting a value from the Construction property.

- Switch between straight lines and mirror lines by selecting a value from the Mirror property.
- Enter the number of sides for a polygon in the Number of Sides property.

Copying a sketch

You can copy a sketch line in Sketch mode with the Select tool.

To copy a sketch line

Ctrl+drag (with the Select tool) while sketching to copy a sketch line.

Press the spacebar while Ctrl+dragging a sketch line to dimension the distance between the first line and the second, copied line.

The sketch grid

The sketch grid indicates that you are performing actions in a 2D plane. Selection, sketching, creating layouts, adjusting blend planes, cross-section editing, and annotating all use the sketch grid. You can adjust the units and spacing of the grid, as well as how solids are displayed when the grid appears.

To display a sketch grid

1. Select Sketch Mode  or any sketching tool.
2. Click a face, plane, axis, or drawing sheet.

   You can also select the combinations used to insert a plane to display a sketch grid at that location.

   If you do not see the grid, make sure that the **Show Sketch Grid** box is checked in the Display tab's Grid ribbon group.

To select a new location for the sketch grid

1. Click **Select New Sketch Plane**  in the mini-toolbar, or right-click and select **Select New Sketch Plane** from the context menu.
2. Mouse over any existing geometry to display existing planes.
3. Click to select the highlighted plane and display the sketch grid.

To move the sketch grid

1. (Optional) Select any points, lines, or curves that you want to move with the sketch grid.
2. Click **Move Grid**  in the mini-toolbar.
3. Use the Move handle to move or rotate the sketch grid.

To view the sketch grid head-on

Click **Plan View**  in the mini-toolbar or in the Orient ribbon group to view the sketch grid head-on.

Moving the sketch grid

Use the Move Grid tool to move the sketch grid. Make successive sketches by moving the grid after sketching closed line regions. These closed lines turn into regions when you move the grid.

To move the sketch grid

1. Click the Move Grid tool  on the mini-toolbar or click the Move tool and check the **Move Grid** option.
2. (Optional) Select any sketch entities that you want to move along with the sketch grid.
3. Select a Move handle axis.
4. Drag along the axis of the Move handle to move or rotate the sketch grid.

   Press **Shift** while dragging to snap the move to angular and linear increments based on your snap settings as well as to snap the move parallel to planes, edges, and axes. You can also right-click and select **Use Ruler Dimension**, and enter a value or press **Enter** to drag the grid. You can also use the standard Move tool guides when moving the grid.

   The Move handle can be moved around by dragging the center ball onto appropriate entities on the sketch grid.

Layouts

2D layouts are useful when you want to draw in 2D, and have no immediate need to generate 3D objects from the lines in the layout. If you want to create a region that can be pulled into 3D right away, create a sketch instead.

You can think of a layout as a pencil drawing made on your design. If you try to pull layout lines to 3D, they do not behave the same way that sketched lines do. Closed lines are not converted to regions, so if you pull a layout line, it creates a surface, not a solid. When you are ready to use your layout to create geometry, project the layout lines to a sketch. Projecting a layout line to a sketch is like inking the line.

Layouts always appear on planes in the Structure tree.

We strongly encourage you to use layers when working with layouts to help you organize your design. For example, you can color individual lines on each layout, show or hide the lines, or put the bounding planes on a separate layer and turn that layer's visibility off to declutter your design. (When you import files, they will initially appear in one color.)

To create a layout

1. Insert a plane.
2. Right-click the plane in the Structure tree and select **Edit Layout**.

   The icon on the Structure tree changes to reflect that the plane is a layout.
3. Sketch on the plane.

To convert an existing plane to a layout

1. Click the Edit Layout icon  next to the plane in the Design window.

   The icon on the Structure tree changes to reflect that the plane is a layout
2. Sketch on the plane.

To convert a sketch to a layout

1. Select the sketch entities that you want to appear on the layout.
2. Click the Plane tool in the Insert ribbon group.

   A layout plane is drawn around the selected sketch entities.
3. Click the Edit Layout icon  next to the plane in the Design window to continue editing the sketch objects on the layout.

To import a 2D AutoCAD DXF or DWG file as a layout

You can import a 2D AutoCAD file into your design in the following ways:

- Drag and drop the DXF or DWG file into the Design window to create a layout in the active component.
- Drag the file onto a plane in the Structure tree to place the drawing on that plane and convert it to a layout.

Edit a layout

1. Right-click the plane in the Structure tree and select **Edit Layout**, or click the Edit Layout icon  next to the plane in the Design window.

   A layout plane is drawn around the selected sketch entities.

Moving in two dimensions

Use the Move tool to move lines or points with the Move handle. When you move points or lines with this tool, they do not maintain their connections to other lines or points. If you want to maintain the connections in your sketch, use the Select tool to edit the sketch.

To move sketch entities

1. Click the Move tool.
2. Select the line or point you want to move.
3. Use the Move handle to move the line or point.

Detailed instructions

1. Select the Move tool  from the Edit ribbon group.
2. Select the line or point on the sketch grid that you want to move to display the Move handle.

   You can select multiple sketch entities.
3. (Optional) Drag the center point of the Move handle to anchor it to any endpoint or midpoint of any line on the sketch grid.

   This functionality is useful when you want to rotate a sketch entity around another point on the sketch.
4. (Optional) Click the Move Direction tool guide and click a line or edge to reorient the Move handle.

   You can also Alt+click the line or edge to reorient the Move handle.
5. Click an axis and drag in that direction to detach and move the selected sketch entity.

   To dimension the move, type the length of the move or the rotation angle and press **Enter**.

   The cursor does not need to be on the axis to move the selected object. In fact, you may find it easier to control the move if you drag some distance from the entity and the Move handle.

Dimensional sketching

SpaceClaim allows you to do precise, dimensional sketching internal to the current line and relative to other lines and points. If a dimension cannot be edited, it does not highlight on mouseover.

To dimension the current line

1. Press the spacebar to enter a value.
2. Press **Tab** to highlight and edit a secondary dimension.
3. Press **Enter** to accept the values.

SpaceClaim allows you to enter expressions as dimension values.

The following expression elements are available:

- Infix (dyadic) operators: **+ - * / ^**
- Prefix (monadic) operators: **+ -**
- Functions: **sin cos tan asin acos atan sqrt log log10 exp**
- Constants: **pi e root2 root3**
- Units: **m cm mm yd ft in ' " deg rad**

Normal precedence rules apply:

1 + 2 * 3 ^ 4 = 1 + (2 * (3 ^ 4)) = 163

Parentheses are required for expression arguments and optional for simple arguments:

- sqrt 2 = sqrt(2) = 1.4142...
- sqrt 2*2 = (sqrt 2) * 2 = 2.8284...
- sqrt(2*2) = 2

Missing operators are inferred:

- 1 1/2 = 1 + 1/2
- 1'6" = 1' + 6"
- 1ft 6in - 17in = 1ft + 6in - 17in
- 1 2 3 4 5 = 1 + 2 + 3 + 4 + 5 = 15
- (1)(2)(3)(4)(5) = (1) * (2) * (3) * (4) * (5) = 120
- 2(1 + 2) = 2 * (1 + 2) = 6
- sqrt 2 sqrt 2 = sqrt 2 * sqrt 2 = 2
- 4(4atan(1/5) - atan(1/239)) = 4 * (4 * atan(1/5) - atan(1/239)) = pi

Units are applied to previous terms if units were not specified and are applied to subsequent terms unless you override them:

- 1 + 1cm = 1cm + 1cm
- 1cm + 1 = 1cm + 1cm
- 1cm + 1 + 1mm = 1cm + 1mm + 1mm
- 1cm + 1 1/2 mm = 1cm + 1mm + 1mm / 2

Trigonometry functions work in radians by default, but you can enter degrees:

sin(45 deg)

Numbers support standard form, but e is a built-in constant:

- 2e2 = 200
  2e 2 = 2 * e * 2 = 10.873...
- 2e-2 = 0.02
  2e – 2 = 2 * e – 2 = 3.436...
- 2e1 = 20
  2e = 2 * e

To dimension relative to other lines and points

Hover the mouse over a line or point and press **Shift** to create a dimension relative to that line or point. If you click a spline, the dimension is always created from the start point of that spline.

To create an offset angular dimension

If the angular reference does not touch the arc center, parallel reference lines are created.

Points

Use the Point tool to sketch points in 2D. Points are useful as a dimensional reference, for splitting, and for creating a point on a line or curve through which you want to draw a three-point circle.

The sketch grid must be visible in the workspace before you can draw.

To draw a point

Click anywhere.

Detailed instructions

1. Select the Point tool from the Sketch ribbon group.
2. (Optional) Dimension the point relative to another sketch object.
3. Click or press **Enter** to place the point.

You can also project vertices in 3D to create points in the sketch.

To insert a point at the mid-point between two points

1. Select the Point tool.
2. Alt+Shift+click two points to create a temporary point at the mid-point.
3. Click the temporary point to place a point there.

Lines

Use the Line tool to sketch lines in 2D. These lines will become edges when you pull your sketch into 3D with the Pull tool.

The sketch grid must be visible in the workspace before you can draw.

To draw a line

Click to set each point of the line.

Do it faster Click and drag to draw one straight line.

Detailed instructions

1. Select the Line tool from the Sketch ribbon group or press **L**.
2. (Optional) Dimension the first point relative to another sketch object.

3. Click or press **Enter** to set the first point of the line.
4. (Optional) Dimension the line.
5. Click or press **Enter** to set the next points of the line.

   By default, the line is dimensioned to its start point. You can also dimension relative to another sketch object.

   If you want any section of the line to be an arc, right-click and select **Switch to Arc**, then click to set the radius. Right-click and select **Switch to Line** to return to drawing straight lines between points. You must create at least one line segment before using this option.
6. End the line.

   You can:

   - Double-click to set the end point of the line.
   - Right-click and select **Finish Line**.
   - Press **Esc**.
   - Connect the end point to the start point.
   - Click any tool (except the Clipboard and Orient tools).

You can also:

- Use a line as a mirror

  Right-click the line and select **Set as Mirror Line**.
- Toggle between a line and a construction line

  Right-click the line and select **Construction On/Off**.

Tangent lines

Use the Tangent Line tool to sketch lines tangent to any curves in your design. These lines will become edges when you pull your sketch into 3D with the Pull tool.

The sketch grid must be visible in the workspace before you can draw.

To draw a tangent line

1. Click a curve.
2. Click to set the line's end point.

Detailed instructions

1. Select the Tangent Line tool  from the Sketch ribbon group.

   Curves are highlighted as you mouse over them, and the tangency indicator appears at your cursor location. If there are no curves in your design, this tool is disabled; you must add a curve to create a tangent line.
2. Click the curve you want to draw tangent to.

   As you move the mouse, the start point moves so that the line remains tangent to the curve.
3. (Optional) Dimension the line with a length.

   You cannot dimension from another sketch object when creating a tangent line.
4. Click or press **Enter** to set the end point of the line.

   If you mouse over another curve, the line snaps so that it is tangent to the second curve. To stop this behavior, Alt+click the end point.

You can also:

- Use a tangent line as a mirror

  Right-click the line and select **Set as Mirror Line**.

- Toggle between a tangent line and a construction line

  Right-click the line and select **Construction On/Off**.

Construction lines

Use the Construction Line tool to draw lines that help you create an accurate sketch. Construction lines become axes in 3D. They are also useful for creating mirrors.

The sketch grid must be visible in the workspace before you can draw.

To draw a construction line

Click to set the start and end point of the construction line.

Do it faster Click and drag to draw a straight construction line.

Detailed instructions

1. Select the Construction Line tool  from the Sketch ribbon group.
2. (Optional) Dimension the first point relative to another sketch object.
3. Click to set the first point of the line.
4. (Optional) Dimension the line.
5. Click or press **Enter** to end the line.

   By default, the line is dimensioned to its start point, but you can dimension to another sketch object.

You can also:

- Use a construction line as a mirror

  Right-click the line and select **Set as Mirror Line**.
- Toggle between a line and a construction line

  Right-click the line and select **Construction On/Off**.

Rectangles

Use the Rectangle tool to draw a rectangle along the axes of the sketch grid. When you exit the sketch, any rectangles become surfaces, and the lines become edges of a rectangular solid when you pull your sketch into 3D with the Pull tool.

The sketch grid must be visible in the workspace before you can draw.

To draw a rectangle

1. Click to set the first corner of the rectangle.
2. Click to set the opposite corner of the rectangle.

Do it faster Click and drag to draw a rectangle.

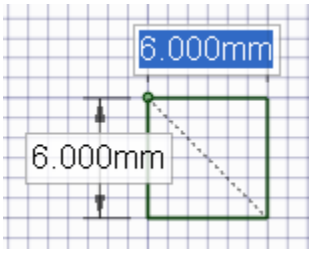
Detailed instructions

1. Select the Rectangle tool  from the Sketch ribbon group or press **R**.
2. (Optional) Dimension the first point relative to another sketch object.
3. Click to set the first corner.

   Mouse over the sketch grid to preview the rectangle. Indicator lines appear when you create a square.
4. (Optional) Dimension the rectangle from the first corner or relative to another sketch object.

5 Click or press **Enter** to set the opposite corner of the rectangle.

Example



Sketching a square

Three-point rectangles

Use the Three-Point Rectangle tool to quickly sketch a rectangle at any angle in 2D. These lines will become the edges of a rectangular solid when you pull your sketch into 3D with the Pull tool.

The sketch grid must be visible in the workspace before you can draw.

To draw a three-point rectangle

1 Click to set the first corner of the rectangle.

2 Click to set the length of the first side.

3 Click to set the length of the second side.

Do it faster	Click and drag to draw the first side, then click to set the length of the second side.

Detailed instructions

1 Select the Three-Point Rectangle tool from the Sketch ribbon group.

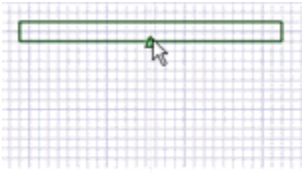
2 (Optional) Dimension the first point relative to another sketch object.

3 Click to set the first corner of the rectangle.

Mouse over the sketch grid to preview the rectangle. Indicator lines appear when you create a square or golden rectangle.

4 (Optional) Dimension the first side from the start point or relative to another sketch object.

5 Click or press **Enter** to set the length of the second side or dimension it.



Circles

Use the Circle tool to sketch a circle in 2D when you know the location of the circle's center and a point on the circle's edge, or the radius or diameter. The circle can become a cylinder or hole when you pull it into 3D with the Pull tool, or a sphere or torus if you rotate or sweep it.

The sketch grid must be visible in the workspace before you can draw.

To draw a circle

1 Click to set the circle's center.

2 Click to set the circle's diameter.

Do it faster	Click and drag to draw a circle.

Detailed instructions

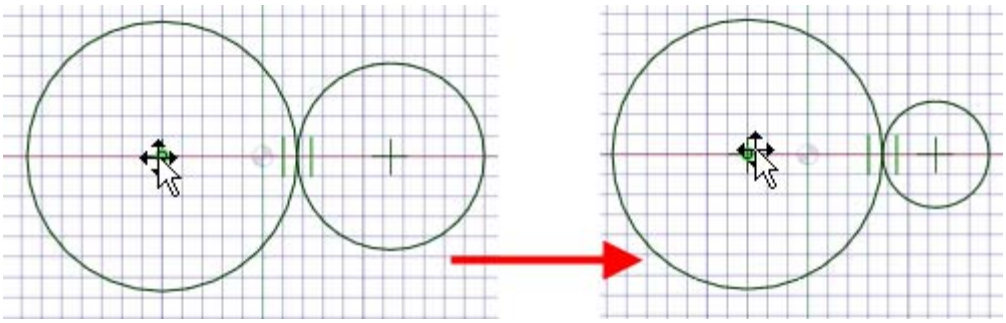
1. Select the Circle tool from the Sketch ribbon group or press **C**.
2. (Optional) Dimension the center of the circle relative to another sketch object.
3. Click to set the circle's center.
4. (Optional) Dimension the diameter.
5. Click or press **Enter** to set the circle's diameter.

   The circle will snap to existing sketches or determined circles and arcs in the plane of the sketch.

   If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

   If you drag the center of a circle that is tangent to another circle, the radius of the other circle changes to maintain tangency.

Example



Editing a tangent circle by dragging the circle's center; tangency is maintained

Three-point circles

Use the Three-Point Circle tool when you don't know the center of the circle, but you know where the edge of the circle must be. This tool works with any combination of free points, known points, or tangent attachments. The circle will become a cylinder or hole when you pull it into 3D with the Pull tool. You can also rotate the circle about a line to make a sphere or torus.

The sketch grid must be visible in the workspace before you can draw.

To draw a three-point circle

1. Click to set the first point on the circle's edge.
2. Click to set the second point.
3. Click to set the third point.

Detailed instructions

1. Select the Three-Point Circle tool from the Sketch ribbon group.
2. (Optional) Dimension the first point relative to another sketch object.
3. Click to set the first point on the circle's edge.

   If you click a curve or line, the circle will be drawn tangent to the curve or line, unless you click the midpoint or vertex.
4. (Optional) Dimension the second point relative to another sketch object.
5. Click or press **Enter**to set the second point on the circle's edge.

As you mouse over the sketch grid, if the circle disappears, the cursor location cannot be included in any circle drawn through the first two points and the current one. If you click a curve or line, the circle will be drawn tangent to the curve or line, unless you click the midpoint or vertex.

6 (Optional) Dimension the radius of the circle or dimension the third point relative to another point..

If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

7 Click or press **Enter** to set the last point on the circle's edge.

If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

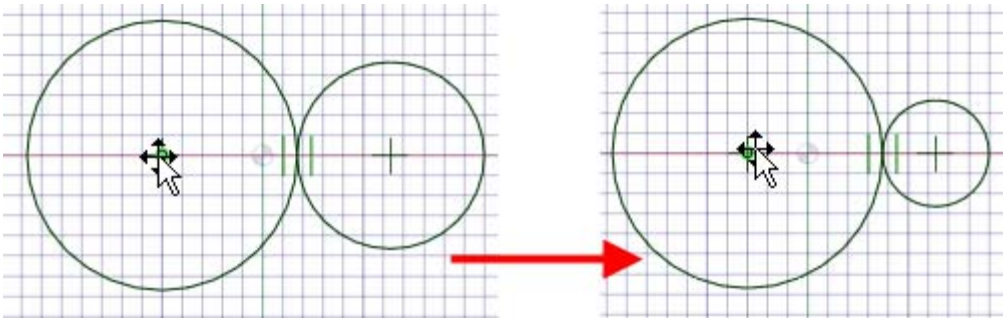
If you drag the center of a circle that is tangent to another circle, the radius of the other circle changes to maintain tangency.

Options

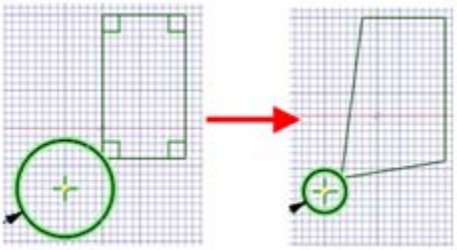
The following options are available in the Three-Point Circle tool:

Three-point circle segment	Check this option to create an arc that is a segment of a three-point circle. To create an arc with this option, click to set the first point, click to set the second point, then enter the diameter or click to set the final point.

Example



Editing a tangent circle by dragging the circle's center; tangency is maintained



Dragging (with the Select tool) a three-point circle drawn through a rectangle's vertex maintains the connection.

Construction circles

Use the Construction Circle tool to draw circles that help you create an accurate sketch. Construction circles cannot be pulled into 3D, and are removed when you select a new sketch plane or exit sketching. The construction circle is useful for dimensioning or indicating a circular relationship of objects that lie on a virtual circular line of centers. For example, bolt hole circles around a flange.

The sketch grid must be visible in the workspace before you can draw.

To draw a circle

1 Click to set the circle's center.

2 Click the set the circle's diameter.

Do it faster	Click and drag to draw a construction circle.

Detailed instructions

1 Select the Construction Circle tool from the Sketch ribbon group.

2 (Optional) Dimension the center relative to another sketch object.

3 Click to set the circle's center.

4 (Optional) Dimension the diameter.

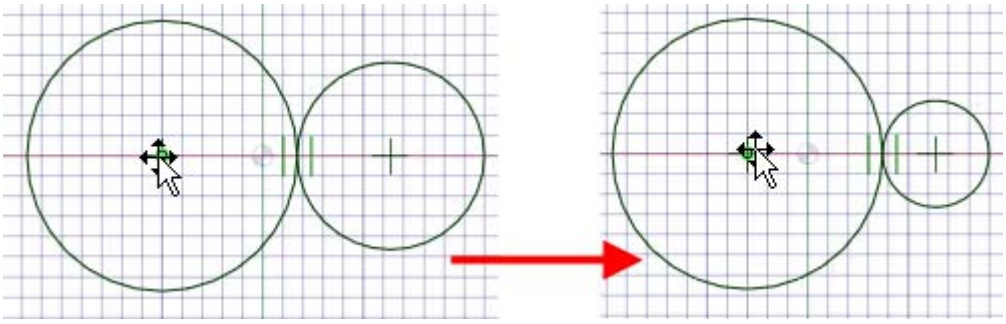
5 Click or press **Enter** to set the circle's diameter.

The circle will snap to existing sketches or determined circles and arcs in the plane of the sketch.

If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

If you drag the center of a circle that is tangent to another circle, the radius of the other circle changes to maintain tangency.

Example



Editing a tangent circle by dragging the circle's center; tangency is maintained

Ellipses

Use the Ellipse tool to sketch an ellipse in 2D. The ellipse can become an elliptical solid or hole when you pull your sketch into 3D with the Pull tool. You can also sweep the ellipse in 3D, or rotate it.

The sketch grid must be visible in the workspace before you can draw.

To draw an ellipse

1 Click to set the center of the ellipse.

2 Click to set the overall length and angular orientation of the first axis.

3 Click to set the overall length of the second axis.

Detailed instructions

1 Select the Ellipse tool from the Sketch ribbon group.

2 (Optional) Dimension the center relative to another sketch object.

3 Click or press **Enter**to set the center of the ellipse.

4 (Optional) Dimension the first axis.

5 Click or press **Enter** to set the overall length and angular orientation of the first axis.

6 (Optional) Dimension the second axis.

7 Click or press **Enter** to set the length of the second axis.

Tangent arcs

Use the Tangent Arc tool to sketch an arc tangent to a curve or line in your design. This arc will become an edge when you pull your sketch into 3D with the Pull tool.

The sketch grid must be visible in the workspace before you can draw.

To draw a tangent arc

1 Click to set the start point of the arc on a line.

2 Click to set the radius and chord angle of the arc.

Detailed instructions

1 Select the Tangent Arc tool from the Sketch ribbon group.

Mouse over the sketch grid to highlight curves and lines. If there are no curves or lines in the sketch, the tool is disabled. You must add a curve or line or move the grid to create a tangent arc.

2 Click the line or curve you want to draw tangent to.

Usually, this is done at the endpoint of a line, arc, or spline, but it can be on the line as well. If there are two lines sharing an end point, use the scroll wheel to set tangency to the other line.

3 (Optional) Dimension the radius and chord angle.

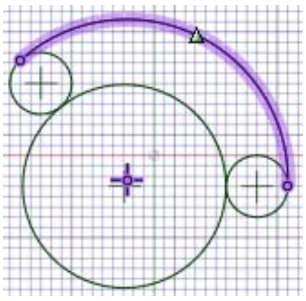
4 Click or press **Enter** to set the end point of the arc.

The end point cannot fall on the start point to make a circle or end on the same line as the start point.

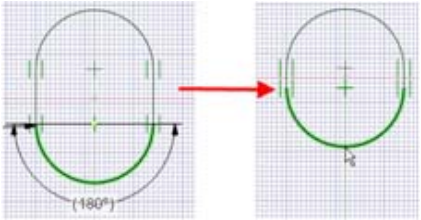
To edit a tangent arc

Drag the arc to edit it. When you drag a tangent arc, it increases its radius while maintaining its chord angle.

Example



Arc sketched tangent to two circles



Editing a tangent arc maintains its connections

Swept arcs

Use the Sweep Arc tool to create an arc with a known center and endpoints. Tangency is not a factor in the creation of this arc. The arc will become an edge when you pull your sketch into 3D with the Pull tool.

The sketch grid must be visible in the workspace before you can draw.

To draw a swept arc

1. Click the arc's center.
2. Click the start point to set the radius and begin the arc.
3. Click the end point.

Detailed instructions

1. Select the Sweep Arc tool  from the Sketch ribbon group.
2. (Optional) Dimension the center relative to another sketch object.
3. Click or press **Enter** to set the center of the sweep circle.
4. (Optional) Dimension the chord angle or create an offset angular dimension.
5. Click to set the start point and radius of the sweep circle.
6. (Optional) Dimension the chord angle or create an offset angular dimension.
7. Click or press **Enter** to set the end point of the arc.

Three-point arcs

Use the Three-Point Arc tool to create an arc by specifying its start and end points, and the radius or chord angle. This arc can be created tangent to another arc, line, or spline at its start. The arc will become an edge when you pull your sketch into 3D with the Pull tool.

The sketch grid must be visible in the workspace before you can draw.

To draw a three-point arc

1. Click the start point of the arc.
2. Click the end point.
3. Click to set the radius.

Detailed instructions

1. Select the Three-Point Arc tool  from the Sketch ribbon group.
2. (Optional) Dimension the start point of the arc relative to another sketch object.
3. Click or press **Enter** to set the start point of the arc.

   You can set tangency by clicking a point on another line, arc, or spline.
4. (Optional) Dimension the gap between the arc's ends with the linear distance and angle from the start point. You can also dimension the end point relative to another sketch object.
5. Click or press **Enter** to set the end point of the arc.

   If the start point is on another line, arc, or spline, then the arc will initially be drawn tangent to that line, arc, or spline.
6. (Optional) Dimension the chord angle or radius.
7. Click or press **Enter** to set the arc's radius.

Polygons

Use the Polygon tool to draw a polygon with up to 32 sides. You can dimension the location of the axis, the length of the radius, and the orientation angle. The lines of a sketched polygon maintain their relationship to each other. When you pull a polygon into 3D, faces with a polygon relationship are shown in blue. If you act on one face or edge, it affects all the faces in the relationship.

To draw a polygon

1. Select the Polygon tool in the Sketch ribbon group.
2. Click to set the center of the polygon.
3. Click to set the diameter and orientation of the polygon.

Detailed instructions

1. Select the Polygon tool in the Sketch ribbon group.
2. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.
3. (Optional) Dimension the center of the polygon relative to another sketch object.
4. Click or press **Enter** to set the center of the polygon.
5. (Optional) Dimension the diameter and orientation.
6. Click or press **Enter** to set the diameter and orientation of the polygon.

   The sides of the polygon are all related, and act as one object. When pulled in 3D, the edges and faces of the polygonal solid will also maintain this relationship.

Tip: If you trim a polygon sketch with the Trim Away tool, you can drag the original sides of the polygon with the Select tool to re-create the polygon.

To set the number of sides

1. In Sketch mode, select the polygon with the Select tool.
2. Right-click the polygon and select **Properties**.
3. Enter a value for the **Number Of Sides** property.

   Polygons can have a minimum of 5 sides, and a maximum of 64 sides.

To remove the polygon relationship from the faces of a polygon solid

Right-click a face of the polygon and select **Remove Association**.

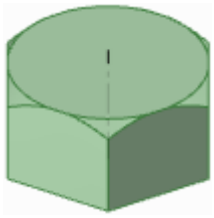
Any changes you make to the face of the polygon will affect only that face.

Options

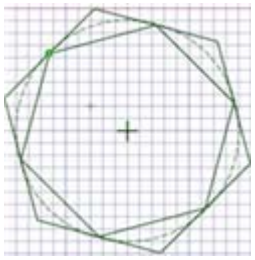
The following option is available in the Polygon tool:

Use internal radius	Select this option to dimension the polygon based on the diameter of a circle inscribed within the polygon. Uncheck it to dimension the polygon based on a circumscribed circle.

Examples



Hexagonal head nut



Two polygons. The outer polygon has the construction circle inscribed within it; the inner polygon has the construction circle circumscribed around it.

Splines

Use the Spline tool to sketch splines in 2D. A spline is a continuously curved line, without sharp boundaries (that is, without vertices). Splines can become edges when you pull your sketch into 3D with the Pull tool. Sweeping along a spline in 3D lets you create smooth, curvy shapes.

The sketch grid must be visible in the workspace before you can draw.

To draw a spline

1. Click to set each point of the spline.
2. Double-click to end.

Detailed instructions

1. Select the Spline tool from the Sketch ribbon group.
2. (Optional) Dimension the first point of the spline relative to another sketch object.
3. Click or press **Enter** to set the first point of the spline.
4. (Optional) Dimension the spline by entering the coordinate distance from the start point to each point, or dimension each point relative to another sketch object.
5. Click or press **Enter** to set the next points of the spline.

   .
6. End the spline.

   You can:

   - Double-click to set the end point of the spline.
   - Right-click and select **Finish Spline**.
   - Press **Esc**.
   - Connect the end point to the start point.
   - Click any tool (except the Clipboard and Orient tools).

You can also:

- Move a spline
  1. Click the Select tool in the Edit ribbon group.
  2. Mouse over the spline to highlight the spline and display its defining points.
  3. Click anywhere on the spline except for the defining points, and drag it to move it.
- Edit a spline
  1. Click the Select tool in the Edit ribbon group.
  2. Mouse over the spline to highlight the spline and display its defining points.
  3. Click and drag on any internal point to move it, leaving the other spline points fixed in space.
  4. (Optional) Edit the dimensions associated with that point.
  5. Click an endpoint to display its endpoint control handle.

     The endpoint control handle is a light blue point at the end of a dotted line extending outwards from the end of the spline. In some cases, these endpoints may be located some distance away from your sketch. Zoom out from the sketch until you can see the endpoint control handle.

     If another line, arc, or spline shares that endpoint, the endpoint influence may snap into tangency with that sketch entity. To adjust the endpoint influence in this case, move the mouse a short distance away from the endpoint to display the endpoint control handle.
  6. Drag the endpoint control handles to change the influence of that endpoint on the shape of the spline.

     The amount of the endpoint's influence can be controlled by dragging the endpoint control handle closer to or further away from the endpoint.
- Add spline points
  1. Click the Select tool in the Edit ribbon group.
  2. Mouse over the spline to highlight the spline and display its defining points.
  3. Right-click anywhere on the spline except on the defining points, and select **Add Spline Point** to create another spline point at that location.
- Remove spline points
  1. Click the Select tool in the Edit ribbon group.
  2. Click the spline to highlight it and display its defining points.
  3. Right-click the point you want to remove and select **Remove Spline Point**.

     The spline adjusts to accommodate the removal of the point.
- Create a closed spline

  You can create a closed spline in the following ways:
  - When drawing a spline, end it on the start point.
  - When editing the spline, drag one endpoint on top of the other point.

  Once you have created a closed spline, you cannot edit it into an open spline.

Splitting lines

Use the Split Line tool to split one line with another line or point. The segments of the split line can then be selected and edited independently.

The sketch grid must be visible in the workspace before you can split lines.

To split a line

1. Select the Split Line tool  from the Sketch ribbon group.

2 Click the line you want to split.

3 Click the line or point that will split the first line where they intersect.

Trimming lines

Use the Trim Away tool to delete any line portion bounded by an intersection with a line or edge.

The sketch grid must be visible in the workspace before you can trim lines.

To delete a bounded line

Click the line portion you want to delete.

Detailed instructions

1 Select the Trim Away tool  from the Sketch ribbon group or press **T**.

2 Mouse over a line to preview the section that will be deleted.

3 Click to delete the highlighted line section.

The portion of the line you clicked is deleted up to any intersection with another 2D line or edge of a solid.

Creating corners

Use the Create Corner tool to trim back or extend two lines so that they meet at a corner.

The sketch grid must be visible in the workspace before you can create corners.

To connect two lines with a corner

1 Click one of the lines you want to connect with a corner.

2 Click an intersecting line to trim the lines, or a non-intersecting line to extend the lines.

Detailed instructions

1 Select the Create Corner tool  from the Sketch ribbon group.

2 Click a line, arc, or spline.

3 Mouse over another line to preview the corner that will be created.

If you mouse over a line that is the cross-section edge of a solid or a silhouette edge, it will not change. Only a sketched line will change.

4 Click an intersecting line to trim the lines, or a non-intersecting line to extend the lines.

If the lines intersect, click the part of the intersecting line you want to keep. If the lines do not intersect, you can click anywhere on the line to extend the lines to form a corner.

Options

The following options are available in the Create Corner tool:

Trim / Extend curve	If you select this option and click the first line, then click a second, non-intersecting line, you extend the first line, but not the second. If the two lines intersect, the first line is trimmed by the second line.

Creating rounded corners

Use the Create Rounded Corner tool to trim back or connect two intersecting lines or arcs so that they meet with an arc tangent at both ends.

The sketch grid must be visible in the workspace before you can create filleted corners.

To connect two lines with a fillet

1. Click a line or arc.
2. Click an intersecting line to size and trim the arc.

Detailed instructions

1. Select the Create Rounded Corner tool  from the Sketch ribbon group.
2. Click a line or arc.
3. Mouse over another line to preview the fillet that will be created.

   If you mouse over a line that is the edge of a solid, that edge will not change. Only a sketched line will change. The radius of the arc is previewed as you move the mouse. Enter a radius to dimension it.
4. Click an intersecting line to trim the lines, or a non-intersecting line to connect the lines.

   Where you click on the intersecting line defines the radius of the tangent arc, or you can dimension the round by entering its diameter.

   If the lines intersect, click the part of the intersecting line you want to keep. If the lines do not intersect, you can click anywhere on the line to extend the lines to form a rounded corner. Circles are never trimmed.

To edit a rounded corner

Drag the rounded corner to change its radius. Tangency is maintained.

Offsetting lines

Use the Offset Line tool to create an offset of any line in the grid plane. If you are creating an offset spline, you will get the best results with an offset distance appropriate to the curvature of your spline.

The sketch grid must be visible in the workspace before you can create offset lines.

To offset a line

1. Click the line for which you want to create an offset.
2. Click to set the offset distance.

Detailed instructions

1. Select the Offset Line tool  from the Sketch ribbon group.
2. Click the line for which you want to create an offset.

   You can Ctrl+click to include other lines in the sketch that you want to have the same offset.

   Standard line selection techniques apply, so you can double-click to select connected loops of lines. Ctrl and Shift can also be used to extend or replace the selected items prior to offsetting.

   To clear the current selection, click an empty spot in the Design window.

   Hover the mouse over a line or point and press **Shift** to create an offset from that line or point.

   As you move the cursor away from the selected line, you can see a preview of the offset result. The offset defaults to one grid square.
3. (Optional) Select an option to specify how intersecting offset lines should meet.

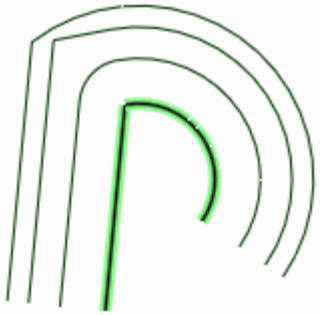
4 Enter a dimension or click to set the width of the offset.

Options

The following options are available in the Offset Line tool:

Close with corner	Close intersecting offset lines with a corner.
Close with arc	Close intersecting offset edges with an arc.
Close naturally	Close intersecting offset edges with a curve.
Offset both ways	Create two offset lines on either side of the selected line

Example



Highlighted line offset using each option. Shown from top to bottom: Close naturally, Close with corner, and Close with arc.

Projecting onto the sketch grid

Use the Project to Sketch tool to copy an edge or vertex of a solid to a 2D line or point that you can manipulate with the sketch tools. You can also project a note onto the sketch grid. Projecting an axis onto the sketch grid creates a construction line.

Except when explicitly selecting a new sketch plane, this is the only sketch tool that can select items that are not on the sketch grid. You can also select note text and project the text onto your sketch.

The sketch grid must be visible in the workspace before you can project edges onto it.

To project a face, edge, vertex, or note onto the sketch grid

Click any edge, vertex, or note text in your design. Click a face in Section mode (represented by a line) to project it onto the sketch grid.

If you have difficulty seeing a projected line, check that **Fade Scene Under Grid** is selected on the Display tab.

Detailed instructions

1 Select the Project to Sketch tool from the Sketch ribbon group.

Mouse over your design to highlight the objects available for projection.

2 Click the edge, edge chain, vertex, or note text you want to project into the sketch plane.

Standard edge selection methods apply, so you can double-click, Ctrl+click, and Shift+click to create or modify your selection.

You can also click and drag to draw a selection box enclosing the objects you want to project, including silhouette edges.

The edge, vertex, or text appears as a sketched line to show that it is now a line or point in the sketch plane.

Duplicate edges (that appear on top of each other) in the sketch are consolidated when you return to 3D mode.

Bending

Use the Bend tool to bend straight lines and edges to form an arc. You can also use the Bend tool to adjust the radius of arcs and arced edges. Bend works on straight lines when you are sketching and when you are editing in cross-section.

You can insert a point curve text file to make a complicated bend.

To bend a line or edge

1. Select the Bend tool  from the Sketch ribbon group.
2. If you are working in Section mode, select the face that contains the edge.
3. Click and drag a line or arc to bend it or change its radius.

   Enter a value to dimension the radius.

Editing

Use the editing tools to create and edit 2D and 3D solids. You can select, pull, move (in 2D and 3D), edit in cross-section, fill (heal), bend lines and edges, and edit faces as a blend.

In SpaceClaim, the line between creating and editing is blurred. Since there is no hierarchical feature tree, you have considerable freedom when designing. Create a box by pulling on a rectangular region. Edit the size of the box by pulling on one of its faces. Draw a rectangle to create a pull-able region. Draw a rectangle on a face to create a new face.

In general, you are either editing or creating with one of main tools (Select, Pull, Move, Combine, or sketching and editing in Section mode), the many secondary tools, or inserting relationships between faces in the design (Shell, Offset, Mirror). Combining objects (intersecting, merging, cutting, etc.) is handled by the tools in the Intersect ribbon group.

Selection is integral to face and edge manipulation. You can extend selections with standard controls (double-click, Ctrl, Shift), by right-clicking and selecting from the Select menu, or using SpaceClaim's power selection functions.

A quick note about solids and surfaces: SpaceClaim always converts a closed set of surface faces into a solid. Similarly, sketched lines that clearly create regions on faces are replaced with real edges. The change in face transparency and edge lines reflects this transformation.

You can cut, copy, paste, and detach objects in most tools.

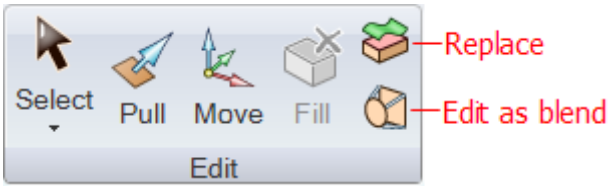
In SpaceClaim, there are three modes you can use to edit your designs:

Sketch mode displays the sketch grid, so you can use any of the sketch tools to sketch in 2D.l

Section mode lets you edit solids by working with their edges and vertices in cross-section.

3D mode lets you work directly with objects in 3D space.

Edit ribbon group



The Edit ribbon group contains the following tools:

Use the Select tool to select 2D or 3D objects in your design for editing. You can select vertices, edges, planes, axes, faces, surfaces, rounds, solids, and components in 3D. In 2D, you can select points and lines. You can also select circle and ellipse centers, the midpoints of lines and edges, and the internal points and endpoints of splines. The Select tool can also be used to edit a sketch.

Use the Pull tool to offset, extrude, revolve, sweep, draft, and blend faces; use it to round, chamfer, or extrude edges.

Use the Move tool to move any single face, surface, solid, or component. The behavior of the Move tool changes based on what you have selected.

Use the Fill tool to fill in the selected region with the surrounding surface or solid. The region must be completely surrounded to be filled.

You can use the Edit as Blend tool to edit any blended face or convert a regular face to a blended one. You can create a new blend section or delete one. You can move any blend section, or you can edit it by moving an edge, moving a vertex, or bending an edge.

Selecting objects

Use the Select tool to select 2D or 3D objects in your design for editing. You can select vertices, edges, planes, axes, faces, surfaces, rounds, solids, and components in 3D. In 2D, you can select points and lines. You can also select circle and ellipse centers, the midpoints of lines and edges, and the internal points and endpoints of splines. You can also select components and other objects in the Structure tree.

You can search for related objects to select on the Selection panel.

The Select tool can also be used to edit a sketch.

To select

1. Click to select the highlighted object. Double-click to select an edge loop. (Double-click again to cycle through alternate loops.) Triple-click to select a solid. Drag (or select **Using Box** from the Select tool menu) to create a selection box.
2. (Optional) Ctrl+click and Shift+click to add or remove items. Alt+click to create a secondary selection set.

Detailed instructions

1. Select the Select tool  from the Edit ribbon group.
2. Mouse over the vertices, edges, faces in the workspace to preview the selectable items in your design.

   If multiple objects occur at your cursor location, use the scroll wheel or arrow keys to preview each one.
3. Click to select a vertex, edge, or face in 3D; click to select a line or point in 2D.

To select:	Do this:
All the edges around a face or closed loop	Double-click an edge or line. Double-click again to select the next loop of edges. Repeat as necessary. You can also right-click the edge or line and choose one of the loop options in the Select menu.
All tangent faces	Double-click a face. (Tangent faces are created by rounds or when edges are drawn on a face.)
Contiguous edges or faces	Click one face or edge, then Shift+click another face or edge to select all the faces or edges between the two.
The sides (but not the top and bottom) of a solid	Triple-click the solid and Ctrl+click the top and bottom to remove them from the selection.
A solid or surface body	Triple-click the solid, or right-click on the solid and select **Select > Body**.
An entire sketch	Triple-click the sketch.
A component	Right-click on the component and select **Select > Component**. This option is available only when you right-click an object within the active component.
All the objects in the active component (except layout surfaces)	Press **Ctrl+A**, click **Select All** in the Select tool menu, or right-click and select **Select > Select All** from the context menu. The types of objects selected depend on whether you are in Sketch, Section, or 3D mode.
Layout surfaces	Click the object in the Structure tree.
Anything completely within a selection box	Click and drag from the upper left to lower right to draw a selection box. You can also click **Using Box** in the Select tool menu, or right-click and select **Select > Using Box** from the context menu, then click and drag.
A lightweight component	Check the **Lightweight Components** box in the Options panel. Then right-click and select **Select > Component**.
The inverse of the current selection (in the active	Right-click a selected object and select **Select > Inverse Selection**.

To select:	Do this:
component)	
A protrusion	Right-click on a protrusion and select **Select > Protrusion**.
A depression	Right-click inside a depression and select **Select > Depression**.
One member of a pattern	Right-click a pattern member and select **Select > Pattern Member**.
All members of a pattern	Right-click a pattern member and select **Select > All Pattern Members**.
Anything partially in a selection box	Click and drag from the lower right to upper left when drawing a selection box.

If any object was part of a group selection used to perform an action, the other parts of the group are highlighted when that object is selected. Click again to select the entire highlighted group.

When selecting within a view on a drawing sheet, you can only select those objects that are on the cross-section plane, or that are within the boundary of a detail view. Box selecting in Sketch mode selects only sketch lines. Box-selecting in Section mode selects only section lines.

4 (Optional) Ctrl+click and Shift+click to add or remove items from the selection set.

Ctrl+click to add or remove one item from the selection set. Shift+click to add everything between your first click and the Shift+click to the selection. You can add or remove items both in the workspace and on the Structure tree. You can also press **Ctrl** and drag to add the items within the selection box to the selection.

5 (Optional) Alt+click to create a secondary selection set.

Hold down the Alt key while performing any of the other selection techniques (double-click, triple-click, Ctrl, Shift) to create the second selection set. Alternate selections are shown in blue, and are used to guide the actions of the Pull and Move tools.

Click an unoccupied space in the Design window to clear the selection.

You can also use the Select tool to:

- Move protrusions and depressions
- Move solids and surfaces
- Edit a sketch
- Copy a sketch
- Pull
- Pivot

To clear a selection

Click any empty space in the Design window or select **Clear Selection** from the Select tool menu.

Selectable objects that are coincident in 2D and 3D

Sometimes objects appear at the same location in 2D. For example, a vertex and end point of a line are often located at the same point in space. When selecting, be sure to check that you have selected the correct object by using the scroll wheel without moving the mouse.

When you have an edge shared by two surfaces or solids, mousing over the edge shades the face that will be affected by an action to the edge if you select it. You can scroll with the mouse wheel to switch between the two faces. Mousing over a vertex shades the edge that will be affected by an action to the vertex if you select it. You can scroll with the mouse wheel to switch between the edges.

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.

Tool guides

Within the Select tool, there are two tool guides that offer you alternative behaviors:

The Select tool guide is active by default. This tool lets you click, double-click, triple-click, Ctrl+click, Shift+click, and Alt+click to select items.

Click a face or edge with the Select Bounds to Extend Selection tool guide to extend the current selection to include all objects bounded by that face or edge. Ctrl+click or double-click to add faces or edges to the boundary. The extended selection appears when you release the **Ctrl** key. Faces are selected if your initial selection was a face. Edges are selected if your initial selection was an edge.

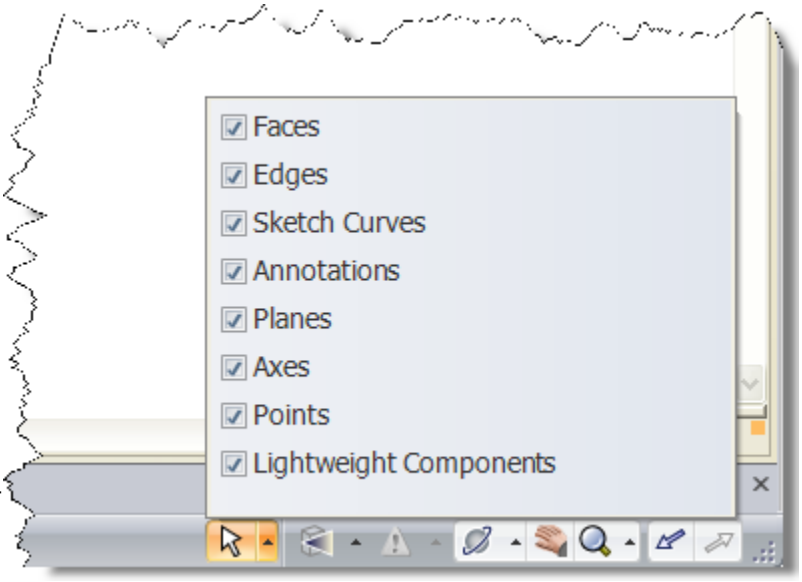
Options

The following options are available in the Select tool:

Cartesian Dimensions	Select this option to use Cartesian dimensions while editing sketches.
Polar Dimensions	Select this option to use polar dimensions while editing sketches.
Snap to Grid	Select this option to snap to the grid while editing sketches.

Filtering selections

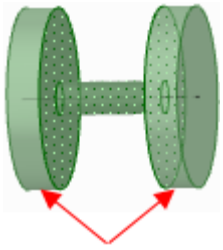
You can filter selections using the drop-down control in the status bar, as shown in the image on the right.



Selection filters apply to each tool. When you switch to another tool, the filter selections are reset to the default.

When you select objects, only those selected in the Filter options are selected. You can limit selection (on click and with box-select) to a number of different objects. When selecting by clicking, all checked objects will be selected. When using box-select, only the top-most checked object is selected. For example, in the figure above, box-selection would select only faces.

Example



Extending a selection with the Select Bounds to Extend Selection tool guide. Arrows indicate the two faces that were clicked to bound the selection.

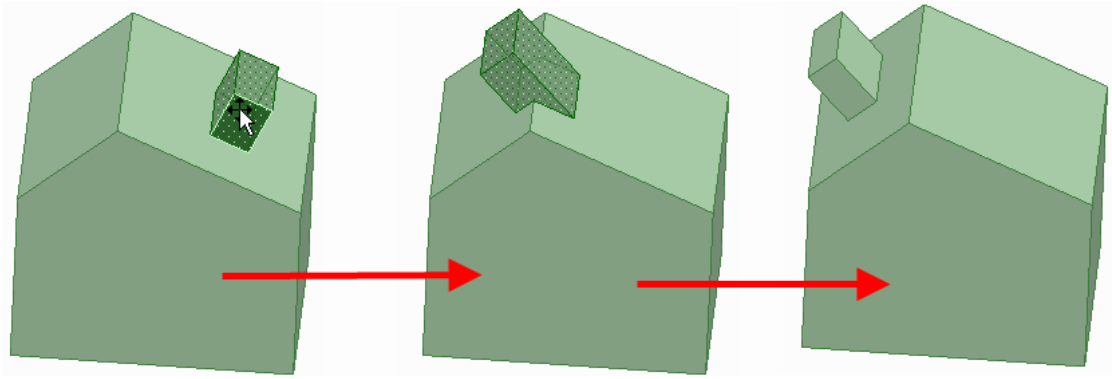
Moving protrusions and depressions

You can move protrusions and depressions with the Select tool or the Move tool.

To move a protrusion or depression

1. Select a protrusion or depression.
2. Drag the protrusion or depression.

   If two faces connect at an angle greater than 90 degrees, you can drag the protrusion across the two faces, as shown in the image below.



Moving solids and surfaces

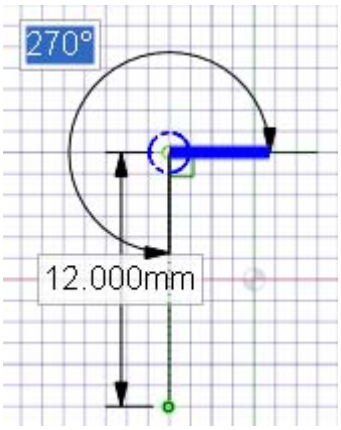
You can move solids and surfaces with the Select tool.

To move a solid or surface

You can do any of the following:

- Select a vertex of a solid or surface, then drag it to move the solid or surface. (Surfaces on the same plane are treated as a single surface and move together.)
- Alt+click a point, then drag the vertex to rotate in the plane of the screen.
- Alt+click an edge or two points, then drag the vertex to rotate the object around the axis defined by the line or points.
- You can place the dragged vertex on a plane or edge by dragging until the plane or edge is highlighted.

Editing a sketch



You can edit the dimensions of sketched lines, arcs, and points using the Select tool. Tangency is maintained when you edit a sketch.

To edit a sketch

1. Click the Select tool. (You can also press **Esc** if you are in a sketching tool.)

   Select the **Select Reference Curve** tool guide to dimension from another object, similar to the Shift+hover functionality in other tools.

2. Click and drag the line or point you want to edit.

   Alt+click and drag if you want to detach the line or point before moving it.

   Ctrl+click and drag to create a copy.

   Enter a value to dimension the move.

To edit the length and angle dimensions of a line

1. Click the endpoint of a line or arc to edit the length or diameter and angle or chord angle dimensions of the line or arc.
2. Mouse over the opposite endpoint of the line or arc to display a blue circle.
3. Drag to move the reference point for the dimension's length.
4. Enter the dimension for the length from the endpoint of the line to the new reference point.
5. Mouse over the line showing the end of the angle reference to highlight it in blue.
6. Drag the blue line to move the reference for the angle dimension.

   If you move the angular reference 360°, the angle measurement is changed from clockwise to counterclockwise.
7. Enter the dimension to adjust the angle of the line from the new reference line.

To edit the distance from another point

1. Click the midpoint of a line, the center of an arc, or any sketched point to display a dimension to a reference point.
2. Mouse over the reference point, then drag the blue circles to the location you want to dimension from.
3. Click to set the new reference location.

   To skew the X-Y lines, click the outer blue circle and enter the skew angle.

To change the properties of a sketch object

1. Click the Select tool. (You can also press **Esc** if you are in a sketching tool.)
2. Select the sketch object to display its properties in the Properties panel.

   You may need to adjust the width of the Properties panel or the columns within the properties panel. Place your mouse over the vertical column line and drag to widen the column. Place your mouse over the border of the panel and drag to widen the panel.
3. You can:
   - Select a new color from the Color property to color the object.
   - Switch between lines and construction lines by selecting a value from the Construction property.
   - Switch between straight lines and mirror lines by selecting a value from the Mirror property.
   - Enter the number of sides for a polygon in the Number of Sides property.

Copying a sketch

You can copy a sketch line in Sketch mode with the Select tool.

To copy a sketch line

Ctrl+drag (with the Select tool) while sketching to copy a sketch line.

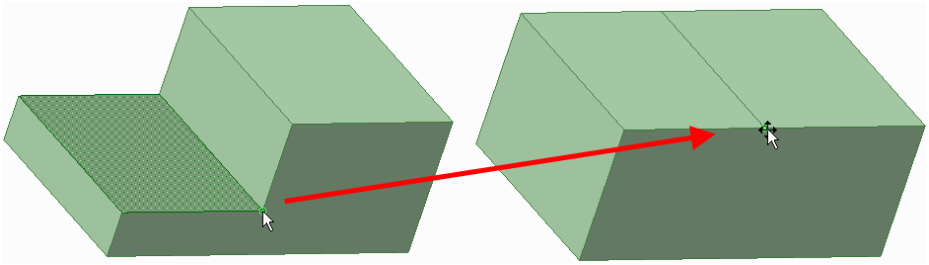
Press the spacebar while Ctrl+dragging a sketch line to dimension the distance between the first line and the second, copied line.

Pulling with the Select tool

You can pull with the Select tool or the Pull tool.

To pull with the Select tool

Drag the vertex of a selected face to another vertex to pull the selected face, as shown in the figure below.

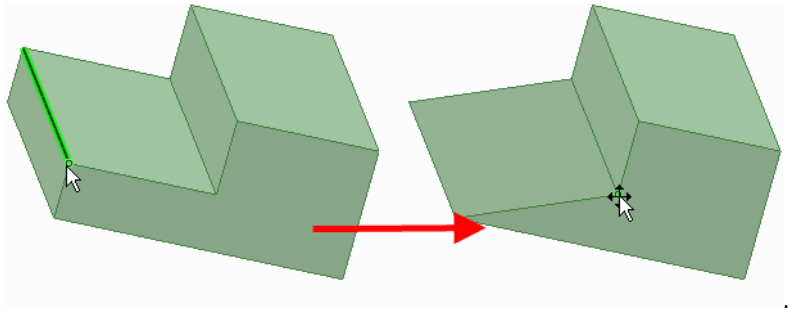


Pivoting with the Select tool

You can pivot an edge with the Select tool or the Pull tool.

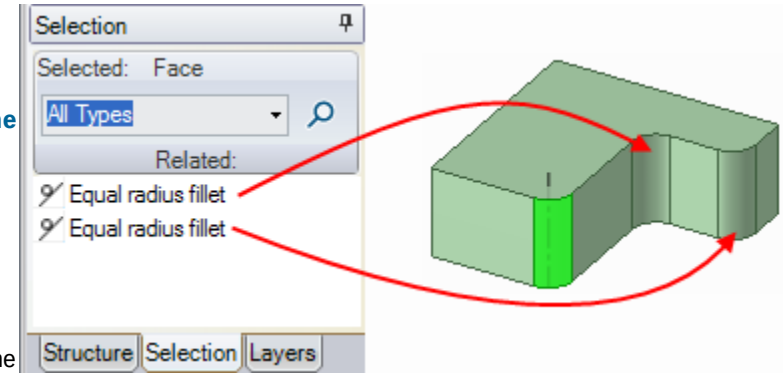
To pivot an edge

Drag the vertex of a selected edge to another vertex to pivot the selected face, as shown in the figure below.



Advanced selection techniques

SpaceClaim offers a powerful method for selecting objects related to the one currently selected in the Selection window that are in the same component.



To select objects similar to the currently selected object

1. Select any 3D object.
2. Click the Selection tab to display the Selection panel.

   The Selection tab is initially located at the bottom of the Structure panel. It displays the number and type of objects currently selected.

3. (Optional) Select a type from the drop-down list to show only those objects with the selected relationship.

   Select **All Types** to display all the geometry related to your selection.

4. Click to search for related objects in your design.

   Related objects are displayed in the Related list. Mouse over the objects in the list to highlight those objects in the Design window.

5. Click any number of the related objects in the list to add them to the current selection.

To find all the edge loops in a surface

1. Select a face on the surface that contains one edge loop.
2. Select **Surface Edge Loop** in the Selection panel drop-down.
3. Click to search for all the edge loops on the entire surface.

   This option is useful when edge loops appear on top of each other in the Design window.

To select and create potential patterns

1. Select a protrusion or depression.
2. Alt-click the planar face that contains the potential pattern.
3. Select **Recognized Pattern** from the drop-down.
4. Click to display potential patterns containing the selected protrusion or depression.
5. Right-click the potential pattern and select **Create Pattern** from the context menu to convert the selected protrusions or depressions into a pattern.
6. (Optional) Select a face of a pattern member to display the count and dimensions for editing.

To select faces that surround an internal void

1. Click a face of the solid with internal faces.
2. Select **Inner Faces** from the drop-down.
3. Click to display all the internal faces.

Pulling

Use the Pull tool to offset, extrude, revolve, sweep, draft, and blend faces; use it to round, chamfer, or extrude edges. You can select a face, then pull, dragging anywhere to act, or you can click, drag, and release a highlighted face. In general, the result of a pull stays selected or highlighted after the pull operation.

The action of the Pull tool depends on which faces and edges you select to work with, and which faces, planes, or edges you select to drive the change. For example, if you choose to work with a face, then select an edge to "drive" the pull, the Pull tool infers that you want to pivot the face around that edge. When multiple actions can be inferred, you can use the tool guides to correct the Pull tool's inference. The Pull tool maintains any offset, mirror, pattern, or coaxial relationships.

When you pull a face, there are two main decisions you need to make. The first is to determine the direction you want to pull in. A default direction is offered to you, but it can be overridden using the Direction tool guide. The second is to determine what is going to happen at the edges of the face. By default, the edges of the face are determined by its neighbors, but you can override this behavior by including the edges in your Pull selection to create an extrusion.

To create and edit solids

1. Select the faces and/or edges you want to work with.

2 (Optional) Alt+click the face or edge that will drive the pull.

3 Drag in the direction of the Pull arrow.

Detailed instructions

1 Select the Pull tool  from the Edit ribbon group.

Mouse over faces and edges in your design to preview the natural Pull direction. If your mouse is over multiple faces or edges, use the scroll wheel to preview the Pull direction for each one.

2 Select the faces and/or edges you want to pull to create 3D solids or surfaces.

You can right-click in the Design window and select **Anchor Pull Handle**, then click to anchor the Pull handle on another object. This command is useful when you want to dimension a Pull from a different location than the center of a face.

3 (Optional) Alt+click to select the face or edge that will drive the change.

You can also select the appropriate tool guide, then click the face or edge. The driving edge or face is shown in blue. If you select the **Show cursor arrows**SpaceClaim option, arrows appear to indicate the directions you can move your mouse to edit the selected object.

4 (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

5 Click and drag in the direction of the Pull arrow to create or edit a solid.

If the correct pull arrow is not highlighted, press **Tab** or click the Pull arrow you want to use.

Press **Shift** to snap while pulling.

To dimension the pull, type the distance you want to pull and press **Enter**. (If the dimension field does not appear editable, start typing *while you are pulling* to enter the dimension.)

You do not have to click and drag on (or even near) the Pull arrow. In fact, we recommend that you move your cursor away from the arrow for more precise control of the pull. The only important input to the Pull tool is the direction in which you are moving the cursor. When you pull, contiguous solids are automatically combined.

If you want to pull in a head-on view, use the ruler at the bottom right corner of the Design window to pull instead of the Pull arrow. Slide the gray bar to the left to subtract material, and to the right to add material.

You can also click the Up To tool guide, then click the object that sets the plane up to which you want to pull. (When pulling an edge, the face does not need to intersect the edge you are pulling.)

If you pull through another object in the same component, the smaller object is merged into the larger one, and receives the larger object's properties. If you pull multiple, touching surfaces, the smaller surfaces are merged into the largest one.

Press **Esc** to cancel the Pull.

See the topics in the Table of Contents under **Designing > Editing > Pulling** for the list of actions you can perform with the Pull tool.

Tool guides

Within the Pull tool, there are several tool guides that let you specify the behavior of the Pull tool:

The Select tool guide is active by default. When this tool guide is active, you can perform standard selection tasks, and create natural offsets and rounds. Select a face, parallel faces, or surface edges to offset them. Select a solid edge to round it. Alt+click to select the driving face or edge for revolves, directed extrusions, sweeps, and drafts. Alt+double-click an edge to select an edge loop. Alt+doube-click again to cycle through alternate edge loops. You can select objects across multiple components to pull.

Select a face to pivot or select a face and edge to revolve. Then use the Rotate tool guide to select the straight line, edge, or axis around which you want to pivot or revolve.


Use the Direction tool guide to select a straight line, edge, axis, origin axis, plane, or planar face to set the pull direction.


Use the Sweep tool guide to select the straight or curved lines or edges along which you want to sweep. Faces and edges can be swept, and the sweep trajectory cannot be in the same plane as the face.


Select any number of contiguous faces on the same body, then use the Draft tool guide to select the plane, planar face, or edge around which you want to pivot. None of the contiguous faces can be parallel to the neutral plane, face or edge around which you want to pivot.


Use the Up To tool guide to select the object that you want to pull to. The pulled object's face or edge will mate with the surface of the selected body or be pulled up to a plane through the selection.

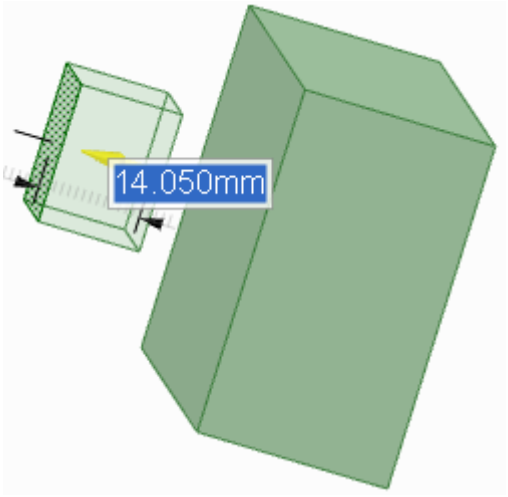
Options

The following options are available in the Pull tool. Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar:

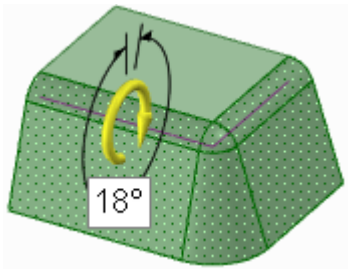
Add	Select the Add option to only add material. If you pull in a subtractive direction, no change will occur. You can combine this option with other Pull options.
Cut	Select the Cut option to only remove material. If you pull in an additive direction, no change will occur. You can combine this option with other Pull options.
Pull Both Sides	Select a single, detached edge, imprinted edge, or surface, then click this option to pull both sides of the edge or surface at once.
Full Pull	Once you select the edge about which to revolve or sweep, click this option to revolve 360 degrees or to the next face, sweep through the full trajectory, or blend through selected faces.
Create Ruler Dimension	Select this option, then click to connect a ruler, oriented along the pull axis, to an anchor edge or face. You can use the ruler to dimension the pull. The direction must be specified to successfully create a ruler dimension. Press **Esc** to cancel the ruler dimension.
Round	When you are pulling an edge, select this option to create a fillet.
Chamfer	When you are pulling an edge, select this option to create a chamfer.
Extrude Edge	When you are pulling an edge, select this option to extrude the edge into a surface.
Copy Edge	When you are pulling an edge, select this option to create a copy of the edge.
Pivot Edge	When you are pulling an edge, select this option to pivot the edge along the selected Pull arrow.
Maintain Offset	Select this option to maintain the offset relationship when pulling.
Blend	Select this option to create a blend between the selected faces, surfaces, or edges when you pull.
Rotational Blend	Select this option to create cylinders and cones whenever possible during the creation of a blend.
Periodic Blend	Select this option to go all the way around when blending.
Show Take-Off Vectors	Select this option to show the vectors created by maintaining tangency of neighboring faces and edges when blending. This option is especially useful when blending between edges.

Revolve Helix	Select this option to create a helix.
Right-Handed Helix	Select this option to determine the direction in which the helix is revolved around its axis.
Rotational Rib	Select this option to pull a rib in a rotational direction.
Normal to Trajectory	Select this option to keep every portion of the swept geometry normal to the sweep trajectory.
Draft Both Sides	Select this option to pivot the face on the opposite side of the reference face as well as the selected face.

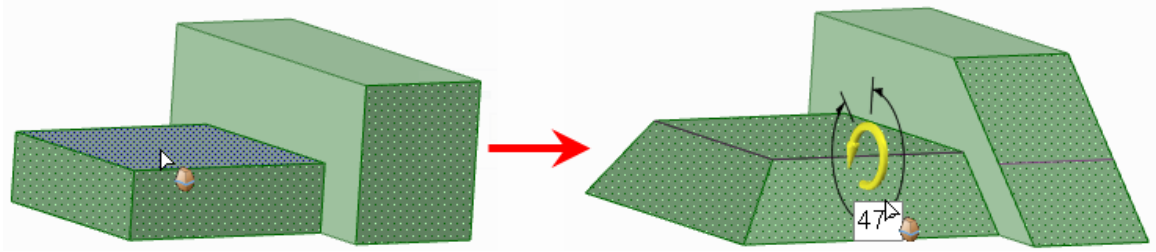
Examples



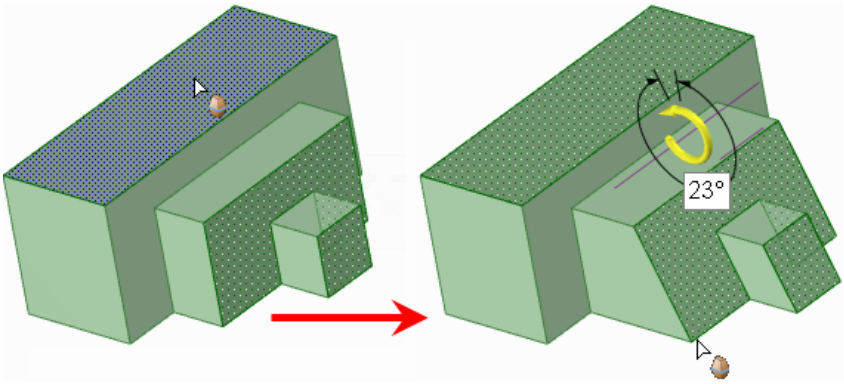
About to cut a solid with a surface using the Pull tool's Cut option



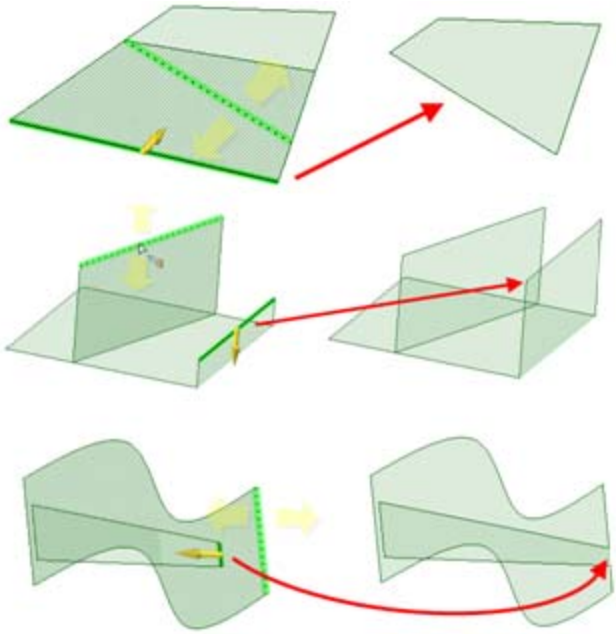
Drafting a face about a round



Drafting non-contiguous faces



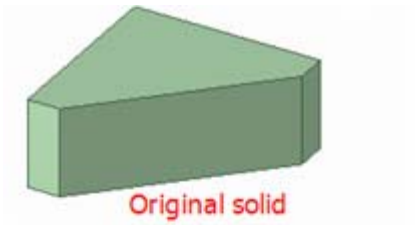
Drafting around a plane that does not touch the faces being drafted



Pulling edges up to other edges

Offsetting or extruding faces

When you offset a face with the Pull tool, the pull extends the neighboring faces without creating an edge. Extruding a face creates edges.



To offset or extrude a face

1. Make sure the Select tool guide is active.
2. Select the face or surface you want to offset or extrude.

   You can select multiple surfaces and faces of solids to offset or extrude them simultaneously.
3. (Optional) Add edges to your selection.

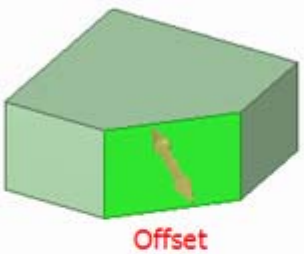
   Select the edges of the face that you want to extrude when you pull. (Any unselected edges are defined by the neighboring geometry during the pull, creating an offset instead of an extrusion.)
4. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

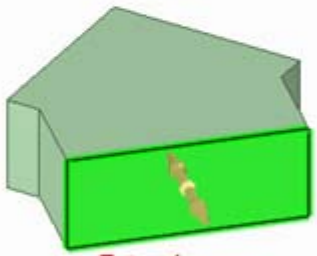
   Select:

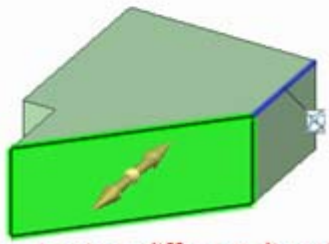
   - **Add** to add material during the pull.
   - **Cut** to remove material during the pull.
   - **Pull Both Sides** to pull both sides of a single surface or edge.
   - **Create Ruler Dimension** to dimension the pull to any reference point along the pull direction.
5. (Optional) Select the Direction tool guide, then click the straight line, axis, or edge if you want to offset or extrude in a different direction.

   You can also Alt+click a straight line, axis, or edge. Sometimes SpaceClaim guesses your intent incorrectly when you Alt+click the direction line. If this happens, just use the Direction tool guide to correct it.

   The pull direction is shown in blue.
6. Click and drag the face in the direction of the Pull arrow.







Dragging a:	Does this:
Solid face	Offsets the face in its natural offset direction
Solid face and all its edges	Creates an extrusion
Surface face	Thickens or thins the face
Corner edge of a solid	Creates a round, chamfer, or extrusion, depending on the selected option
Round or chamfer	Offsets the round or chamfer
Variable round	Uniformly offsets the variable round
Cylinder or cone	Offsets the cylinder or cone

To dimension the offset, type the distance you want to pull and press **Enter**.

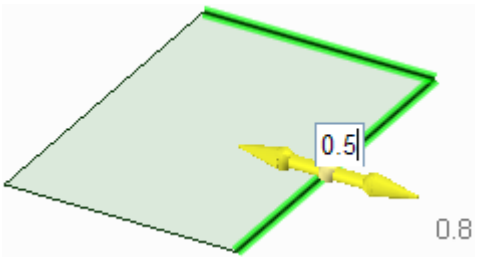
Press and hold **Ctrl** to offset a surface in both directions.

Neighboring faces automatically extend to bound the offset face.

You can also use the Up To tool guide to pull up to any edge, plane, surface, or face in your design.

Extending or extruding surface edges

You can extend or extrude the edges of any surface with the Pull tool. When you extend an edge, the pull extends the neighboring faces without creating a new edge. Extruding an edge creates edges.



To extend or extrude the edge of a surface

1. Make sure the Select tool guide is active.
2. Select the outside edge of a surface.

   Ctrl+click to select multiple edges.
3. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

   Select:

   - **Add** to add material during the pull.
   - **Cut** to remove material during the pull.
   - **Create Ruler Dimension** to dimension the pull to any reference point along the pull direction.
4. Click the Pull arrow oriented along the surface to extend the edge. Press **Tab** or click the other Pull arrow to extrude the edge in another direction.
5. (Optional) Ctrl+click the vertex of one or both neighboring edges to ignore their influence.
6. Drag in the direction of the Pull arrow to extend the edge or create a new surface perpendicular to the old one.

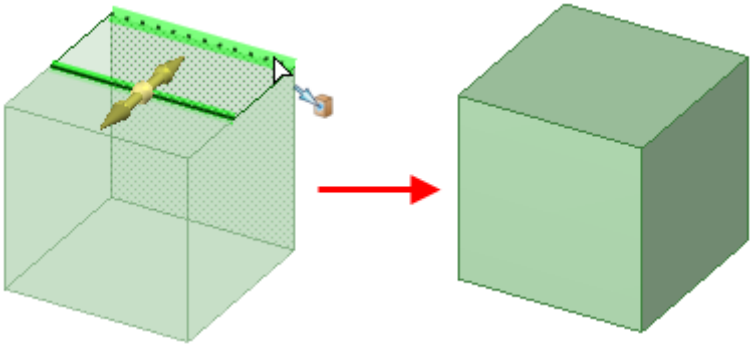
   If the correct pull arrow is not highlighted, press **Tab** or click the Pull arrow you want to use.

   The natural direction of a surface edge is in the plane of the surface.

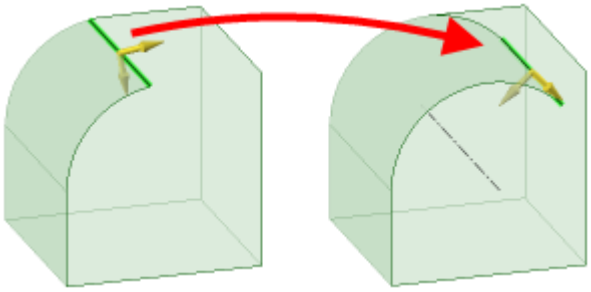
   You can use the Up To tool guide to pull a linear edge up to a face, surface, edge, or point. If the face or surface does not intersect the edge you are pulling, the edge will be pulled parallel to the selected object.

   To dimension the extension, type the distance you want to extend the surface *while you are pulling* and press **Enter**.

Examples



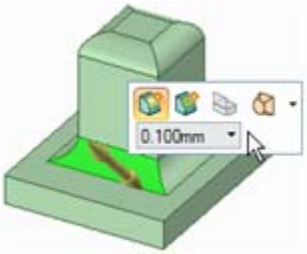
Pulling the edge of a surface up to another edge forms a solid



Pulling the edge of a cylindrical surface extends the surface along a cylindrical path

Rounding edges

You can round the edges of any solid by selecting the Pull tool's Fillet option.



To round an edge

1. Make sure the Select tool guide is active.
2. Select the edge or edges you want to round. Double-click to select a tangent chain.

   Rounds are generated on exterior corners, and fillets are generated on interior corners.
3. Select the Fillet option in the Options window, or from the mini-toolbar.
4. Click and drag the edge in the direction of the Pull arrow.

   To dimension the round, right-click and enter a radius length in the mini-toolbar, or type the radius length *while you are pulling* and press **Enter**.

   Once you create a round, pulling an adjacent face also pulls the round.

   Note: The face hidden by the round is remembered, so that if you fill the round it can be displayed. If you move a round, the face hidden by it also moved.

To create a full round by pulling edges

Select two edges that share a face, then pull until they meet to create a full round.

To create a full round by selecting faces

1 Select three faces: the face that will become the full round, and the two faces that will share edges with the full round.

All faces must be part of the same solid.

2 Right-click and select **Full Round** from the context menu.

To change a constant-radius round to a variable-radius round

1 Make sure the Pull tool's Select tool guide is active.

2 Right-click the round face and select **Edit as Variable Radius Round**.

3 Click the Pull arrow on the end of the round face and drag away from the face adjust the radius at the edge of the round. Click the Pull arrow pointing toward the center of the face and drag it along the face (or enter a length or percent) to set another point where you can adjust the round's radius.

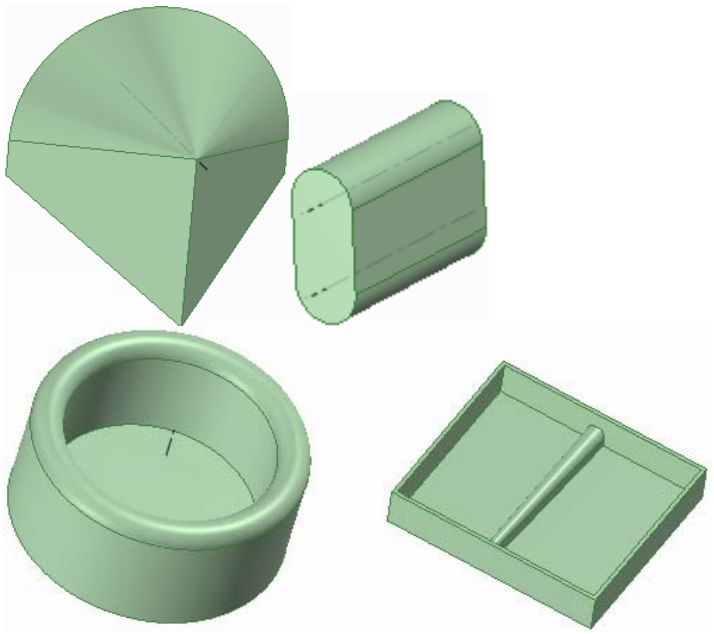
Two rounds that intersect at a shared edge can be made variable together by selecting the shared sharp edge for modification. Both rounds become variable at that point. They can be made variable independently by selecting the sharp end edge that is not shared by the rounds.

Capping round faces

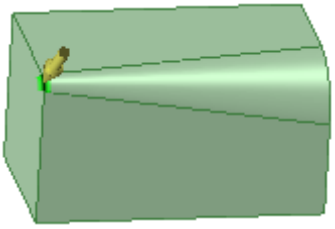
Use the Fill tool to remove and cap round faces. You might want to do this if you are trying to move a protrusion surrounded by round faces and it cannot be moved because the rounds create impossible geometry.

Usually you can just select all the round faces and click the Fill tool to remove and cap them all. However, sometimes this is not possible. In that case, select one round and fill it. If that works, undo and select that round and the next. Undo. Continue adding rounds to your selection and trying to fill them until the fill fails. Now you have identified one of the rounds that is causing the problem. Next, fill all the rounds that filled successfully. Finally, repeat this process in the other direction of the round tangent chain. Once you have filled all the rounds except for the one or two causing the problem, select the one causing the problem and its two neighbors. Then click Fill. This process allows more options for the extension of neighboring edges to intersect and cap the round.

Examples

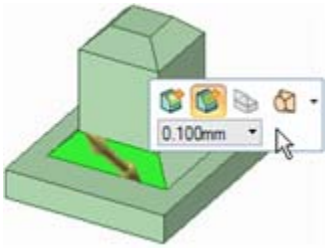


Full rounds



Editing a variable radius round

Chamfering edges



You can chamfer the edges of any solid by selecting the Pull tool's Chamfer option.

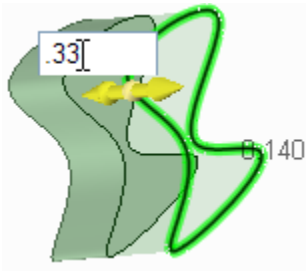
When you make a hole in a chamfered face, the face is no longer a chamfer. You can still pull the face, or the hole, but you cannot change the chamfer to a round or dimension the chamfer.

To chamfer an edge

1. Make sure the Select tool guide is active.
2. Select the edge or edges you want to chamfer. Double-click to select a tangent chain.
3. Select the Chamfer option in the Options window, or from the mini-toolbar.
4. Click and drag the edge in the direction of the Pull arrow.

   To dimension the chamfer, right-click and enter a setback distance in the mini-toolbar, or type the setback distance *while you are pulling* and press **Enter**.

Extruding edges



You can extrude the edge of any solid by selecting the Pull tool's Extrude Edge option. You can also extend and extrude surface edges.

To extrude an edge

1. Make sure the Select tool guide is active.
2. Select the edge or edges you want to extrude.

   Ctrl+click to select multiple edges. Double-click to select a tangent chain.
3. Select the Extrude Edge option in the Options window, or from the mini-toolbar.

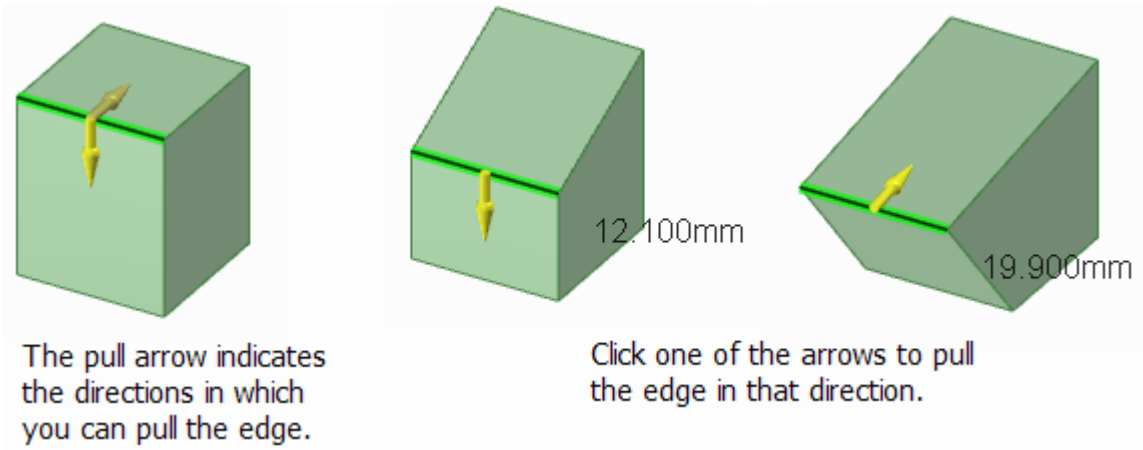
   The Pull arrow changes to show the two directions in which you can extrude the edge. One arrow is highlighted to show the primary direction.
4. If the arrow pointing the direction you want to pull is not highlighted, click the arrow or press **Tab** to change the direction.
5. Click and drag the edge in the direction of the Pull arrow.

   To dimension the extrusion, right-click and enter a distance in the mini-toolbar, or type the distance *while you are pulling* and press **Enter**.

   Press **Ctrl** to copy the edges.

You can click the Up To tool guide and click a face, edge, or point to pull up to. If the face does not intersect the edge you are pulling, the edge will be pulled parallel to the face. If you pull the edge of a surface up to another object, the result is automatically solidified if it creates a closed volume.

Pivoting edges



You can pivot the edge of any solid with the Pull tool's Pivot Edge option.

To turn a cylinder into a cone, select the Pull tool and the Pivot Edge option. Then click the edge of the cylinder and pull it until it forms a point. To turn a truncated cone into a cylinder, click the small edge and pull it out.

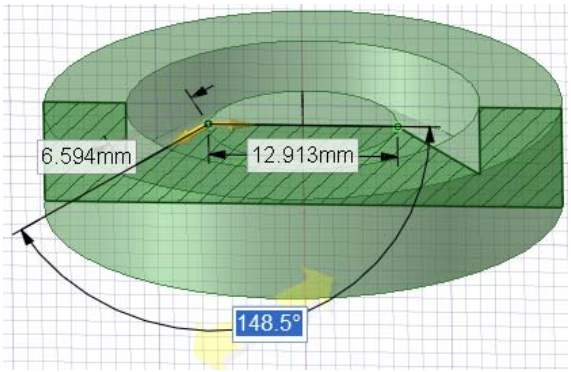
To pivot an edge

1. Make sure the Select tool guide is active.
2. Select the edge or edges you want to pivot. Double-click to select a tangent chain.
3. Select the Pivot Edge option in the Options window, or from the mini-toolbar.

   The Pull arrow changes to show the two directions in which you can move the edge to pivot the connected faces. One arrow is highlighted to show the primary direction.
4. If the arrow pointing the direction you want to pull is not highlighted, click the arrow or press **Tab**.
5. Click and drag the edge in the direction of the Pull arrow.

   During the pull, the distance you have moved the edge is displayed. You can dimension the distance in Section and 3D mode, and dimension the angle in Section mode.

Example



Dimensioning the angle of a pivot

Revolving faces

You can revolve any face or surface with the Pull tool.

To revolve a face

1. Make sure the Select tool guide is active.
2. Select the surfaces, faces, or solids you want to revolve.
3. Alt+click the straight line, axis, or edge to set the revolve axis.

   You can also select the Rotate tool guide, then click to set the revolve axis. The revolve axis is shown in blue.
4. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

   Select:

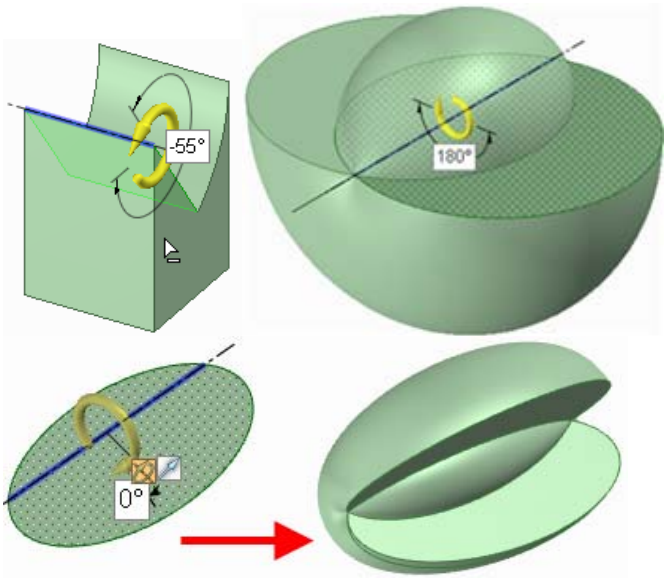
   - **Add** to add material during the pull.
   - **Cut** to remove material during the pull.
   - **Pull Both Sides** to pull both sides of a surface simultaneously.
   - **Full Pull** to revolve 360 degrees.
5. Click and drag in the direction of the Pull arrow to revolve the selected object, select the Up To tool guide and click an edge, face, or plane, or select **Full Pull** from the Options panel or mini-toolbar.

   Neighboring faces automatically extend to define the boundaries of the revolved faces of solids. Revolving from a flat surface face has no neighboring faces, so it makes new face boundaries.

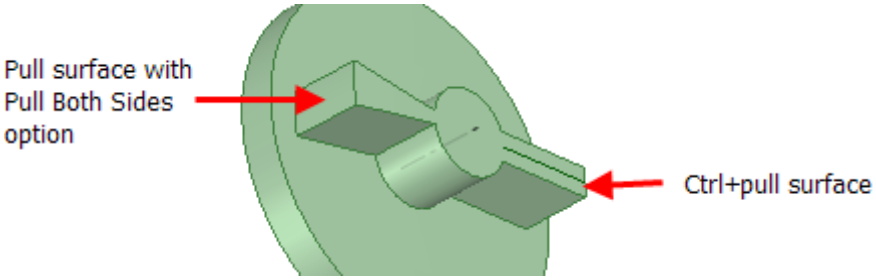
   Ctrl+drag to keep the faces parallel.

   To dimension the rotation, type the rotation angle *while you are pulling* and press **Enter**.

Examples



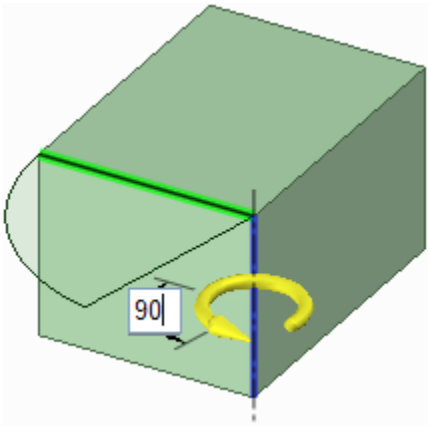
Revolve using cut, 180 degree revolve, and a full revolve of an ellipse around an asymmetrical axis



Ctrl+pulling vs. using the Pull Both Sides option to create ribs

Revolving edges

You can revolve an edge to form a surface with the Pull tool. You can revolve the edge of a solid or surface.



To revolve an edge

1. Make sure the Select tool guide is active.
2. Select the edges you want to revolve.
3. Alt+click the straight line, axis, or edge to set the revolve axis.

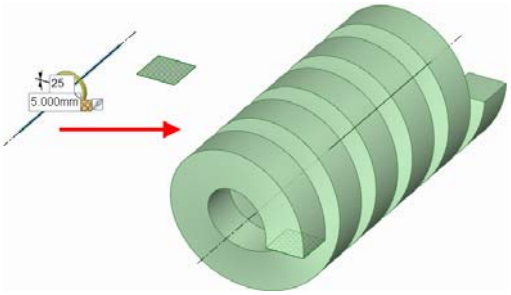
   You can also select the Rotate tool guide, then click the revolve axis. The revolve axis is shown in blue.
4. Click and drag in the direction of the Pull arrow to revolve the selected edge or select the Up To tool guide and click an edge or face to revolve up to.

   Ctrl+click to select edges of faces along with the face to force the pull tool to create new neighboring faces to contain these edges (essentially changing from a draft to a revolve). Unlike other CAD products, SpaceClaim allows revolving planar and non-planar edges and faces about lines that do not lie in those planes. This allows skewed rotational solids and surfaces.

   To dimension the rotation, type the rotation angle *while you are pulling* and press **Enter**.

Revolving helices

You can revolve a helix with the Pull tool.



To revolve a helix

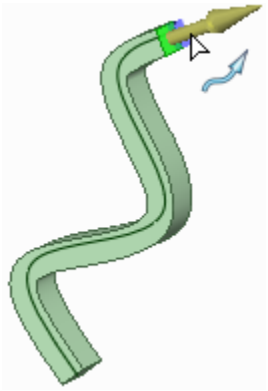
1. Make sure the Select tool guide is active.
2. Select the face or edge you want to revolve.
3. Select the Revolve tool guide.
4. Click the axis you want to revolve around.
5. (Optional) Set the handedness of the helix by checking or unchecking the **Right-Handed Helix** option in the Options panel.
6. Press the spacebar and enter the length and pitch or pull along the axis to create the helix dynamically.

   Pitch is the amount that the helix face shifts per 360 degrees of rotation. Length is the total length of the helix.

Sweeping faces

You can sweep a face along a trajectory with the Pull tool. Sweeping a face around a closed path creates a torus.

To sweep a face



1. Make sure the Select tool guide is active.
2. Select the surfaces or faces you want to sweep.
3. Alt+click the line or edge along which you want to sweep.

   You can also select the Sweep tool guide, then click the sweep trajectory. The sweep trajectory is shown in blue. Alt+double-click to select a tangent chain. Ctrl+click to add contiguous paths.

   Sweep works best when the sweep line is perpendicular to and touches the face you want to sweep. To quickly draw a sweep line, select the face, click the Line or Spline tool, select the Move Grid tool from the mini-toolbar, press **Shift**, and drag the Move handle's axis to rotate the sketch grid 90 degrees.

   To quickly sketch a face perpendicular to a desired trajectory, select the end of the trajectory and select a sketching tool to place the sketch grid at that point. Then draw the face.
4. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

   Select:

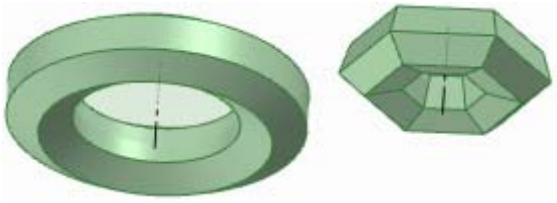
   - **Normal to Trajectory** to keep the swept surface normal to the sweep trajectory. If your sweep trajectory is perpendicular to the face you want to sweep, this option is enabled for you.
   - **Add** to add material during the pull. If you sweep through another solid, it is merged into the swept solid.
   - **Cut** to remove material during the pull. If you sweep through another solid, material is removed.
5. Click and drag in the direction of the Pull arrow to sweep the selected objects, or select **Full Pull** from the Options panel or mini-toolbar to sweep the entire length of the trajectory. If you select Full Pull and the face or surface being swept is in the middle of the trajectory, it will be swept in both directions.

   You can also use the Up To tool guide to select a face or surface on which you want to end the sweep.

To create a torus

1. Sketch a circle to describe the path.
2. Select the circle and click the **Insert Axis** tool.
3. Switch to 3D mode.
4. Select the axis.
5. Click the sketch tool you want to use to make the torus cross-section to automatically place the sketch plane perpendicular to the circle.
6. Draw the sketch centered on a point on the edge of the circle.
7. Sweep along the circle using the **Full Pull** option to create the torus.

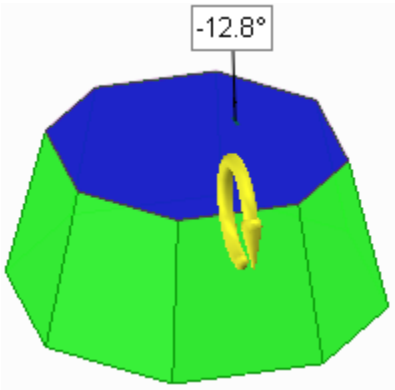
Example



Hexagons swept around circular and hexagonal sweep paths

Drafting faces

You can draft faces around another face or surface with the Pull tool.



To draft a face

1. Make sure the Select tool guide is active.
2. Select the face or contiguous faces you want to draft.
3. Alt+click the face (including rounds) or surface about which you want to draft.

   You can also select the Draft tool guide, then click the face or surface. The draft face or surface is shown in blue.

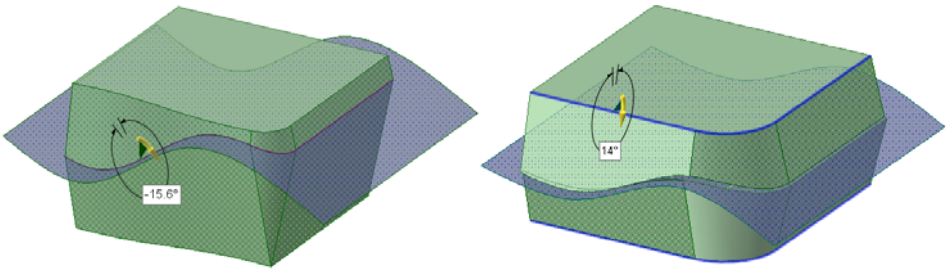
   If you select two edge loops instead, you can create a split draft.
4. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

   Select:

   - **Add** to add material during the pull.
   - **Cut** to remove material during the pull.
   - **Draft Both Sides** to pivot the face on the opposite side of the reference face as well as the selected face.
5. Click and drag in the direction of the Pull arrow to draft the selected faces.

   To dimension the draft, type the rotation angle *while you are pulling* and press **Enter**.

Examples

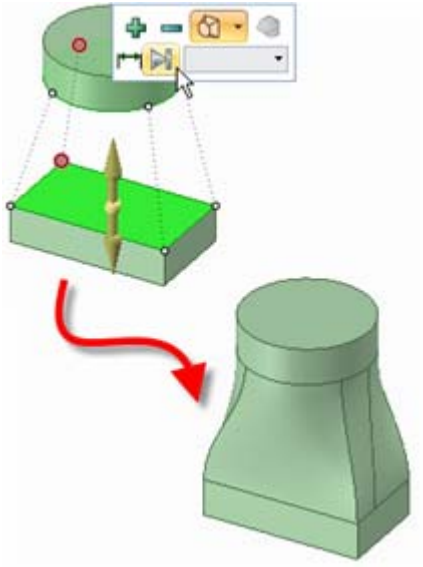


Drafting around a spline surface

Blending

Blending between faces

You can blend between two faces with the Pull tool.



To blend between two faces

1. Make sure the Select tool guide is active.
2. Select the surfaces or faces you want to blend through.

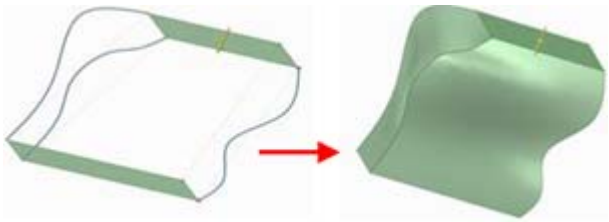
   When you blend between two faces, the blend surface begins tangent to the edges of the initial faces and ends tangent to the edges of the end faces. To prevent this behavior, select the edges you do not want to use for tangency. The figure on the right shows the blends created with and without the effects of the edges.

   The edges of the blend are displayed. If they aren't, check that the **Blend** option is selected in the Options window.
3. (Optional) Alt+click the edges or lines you want to use as guides for the blend.
4. (Optional) Click and drag the blend points to modify the blend edges.
5. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

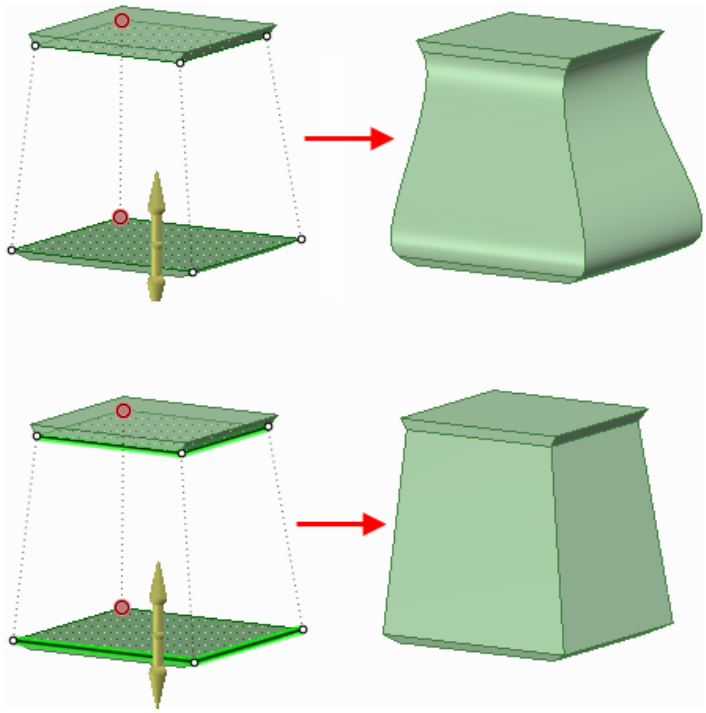
   Select:

   - **Rotational Blend** to create cylinders and cones whenever possible during the creation of a blend. You must have selected faces rotated around a common axis.
   - **Periodic Blend** to go all the way around when blending. You must have selected three or more faces rotated around a common axis, that also span an arc greater than 180 degrees. (Blending between 3 equal-radius circle faces creates a torus.)
6. Click, drag, and release in the direction of the Pull arrow or select **Full Pull** from the Options panel or mini-toolbar to create the blend.

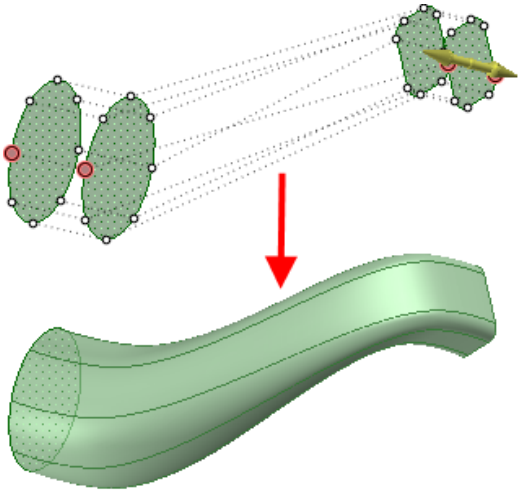
Examples



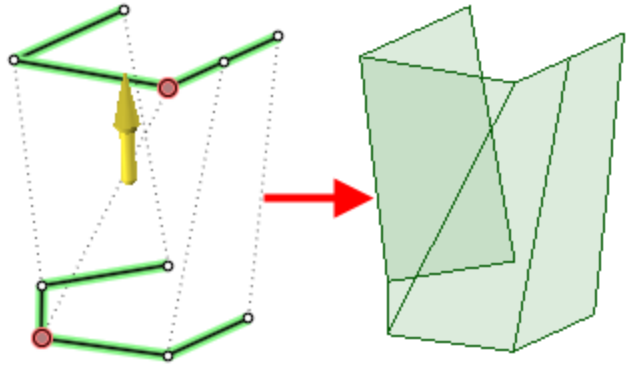
Blend between two surfaces using splines to guide the blend



Blend with edge tangency (no edges pulled) and blend without edge tangency (edges pulled)



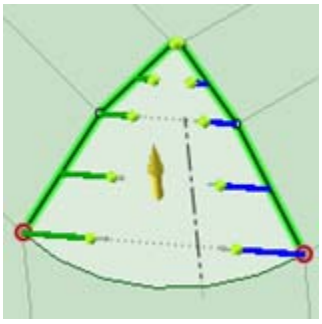
Blend between multiple surfaces, automatically ordered to prevent self-intersection



Blend between multiple lines in space (profiles) with take-off vectors

Blending between edges

You can blend between two edges with the Pull tool  in the Edit ribbon group on the Design tab.



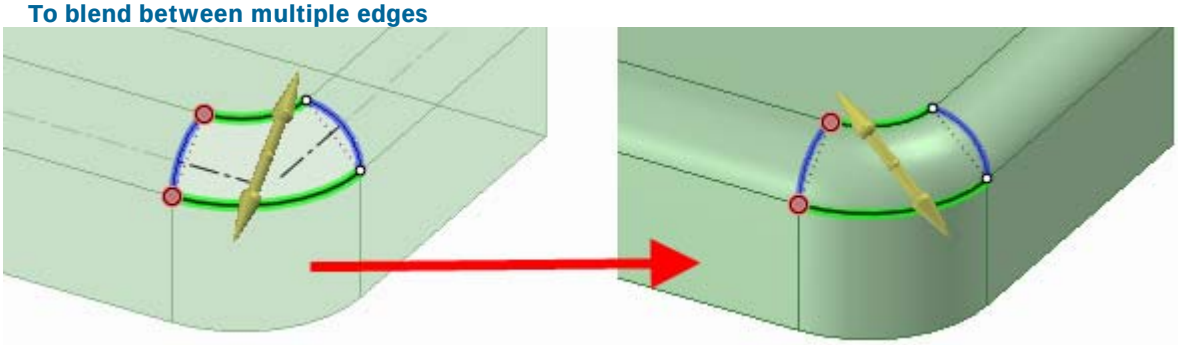
To blend between two edges

1. Make sure the Select tool guide  is active.
2. Click the  **Blend** option in the Options window.
3. Click the starting edge of the blend.
4. Ctrl+click the ending edge of the blend.
5. (Optional) Check the **Show take-off vectors** option to display and adjust the angle at which the new edge will be created if it is not set automatically by tangency.

   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.
6. Click, drag, and release in the direction of the Pull arrow or select  **Full Pull** from the Options panel or mini-toolbar to create the blended surface.

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

To blend between multiple edges

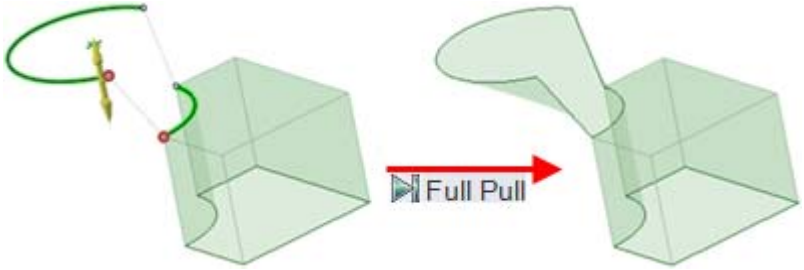


1. Make sure the Select tool guide  is active.
2. Select the two edges to blend between.
3. Press **Ctrl+Alt** and select the other one or two edges that will guide the blend.

   The edges of the blend are displayed. If they aren't, check that the  **Blend** option is selected in the Options window.
4. Select  **Full Pull** from the Options panel or mini-toolbar to create the blended face.

   If you create a blend that results in a surface contained within a solid, the surface remains a separate object.

Example



Blending between a surface edge and line in space using the Full Pull option

Blending between points

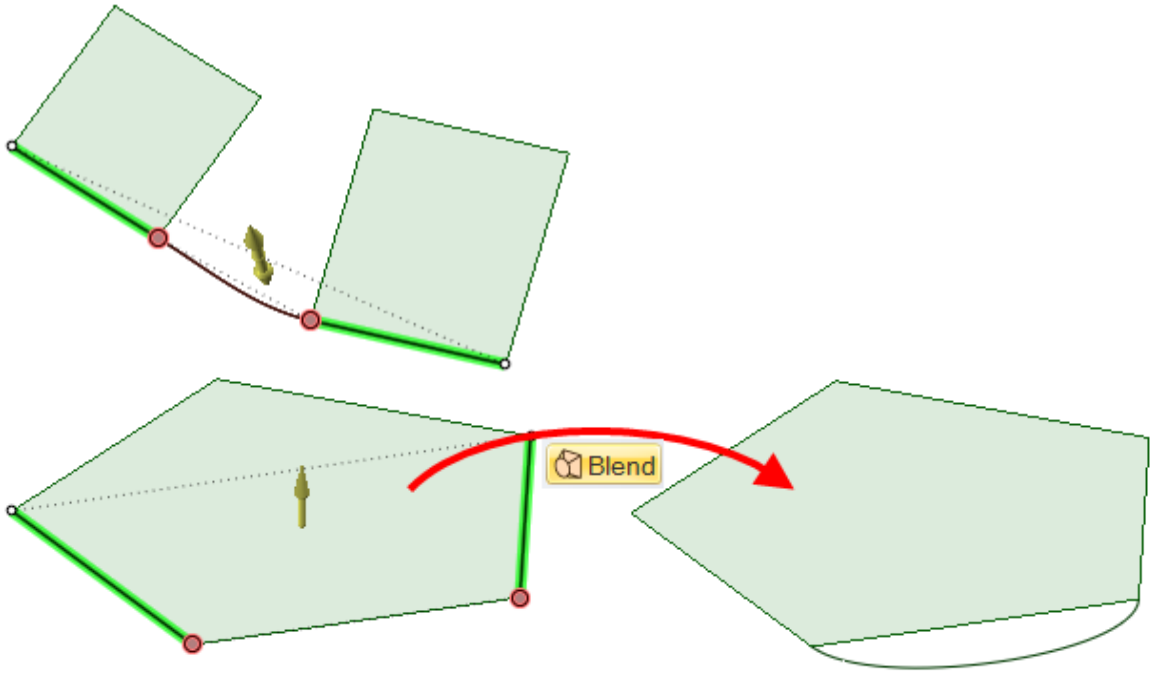
You can create a curve with the Pull tool by using the Blend option to blend between points. You can blend between any two points to create a curve, or select multiple points to create a spline.

To blend between points

1. Ctrl+click to select multiple points or vertices.

   The Blend option is selected automatically.
2. Ctrl+click neighboring edges of the vertices if you want the blend curve to begin tangent to those edges.
3. Click the **Full Pull** option in the Options panel to create the curve.
4. If you want the curve to appear as a separate object in the Structure tree, right-click the curve and select **Name Sketch Curve**.

Examples



Blend curves tangent to neighboring edges

Creating slots

You can create slots from holes using the Pull tool. You can also edit slots. Slots maintain the relationship between their faces.

To create a slot

1. (Optional) Select the **Pull Both Sides** option to create the slot in both directions.
2. Pull the axis or face of a hole to create a slot.

To create a curved slot

1. Select the axis or face of a hole.
2. Select the Rotate tool guide and click the axis of the driving cylinder.

   You can also Alt+click the axis of the driving cylinder.
3. Pull the hole to create a curved slot.

   You can pull a curved slot 360 degrees to make a round cut.

To create a radial slot

1. Select the axis or face of a hole.
2. Do one of the following:
   - Alt-click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the Pull Direction tool guide and pull the hole to create a radial slot.
   - Ctrl-click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the Pull Direction tool guide and pull the hole to offset the cylinder and create a radial slot simultaneously.
   - Select the Pull arrow that points toward the axis and pull.

If you want to lengthen a radial slot while keeping it the same distance from a radial face, Ctrl+click the radial slot's axis and the face, then pull.

If you want to move a slot along with a radial face Ctrl+select the hole's axis and the face, then Ctrl+pull.

To edit a slot

You can modify a hole that was dragged to form a slot by pulling on any of the slot axes.

To move a slot radially

1. Select the Pull tool.
2. Select both axes of the slot.
3. Do one of the following:
   - Alt-click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the Pull Direction tool guide and Ctrl-drag to move the slot radially.
   - Ctrl-click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the Pull Direction tool guide and Ctrl-drag to offset the cylinder and move the slot.

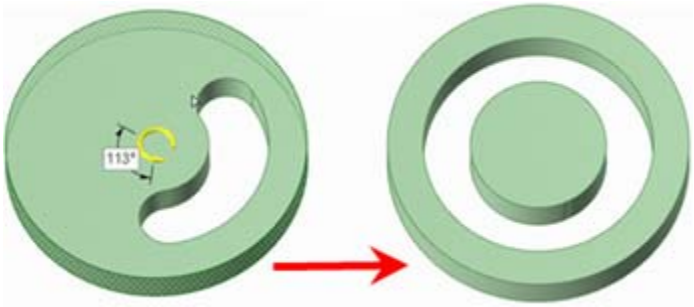
(You can move a hole radially by pressing **Shift** while dragging it with the Pull tool.)

To remove the slot relationship

Right-click a face of the slot and select **Remove Association**.

Any changes you make to the slot's face will affect only that face.

Example



Pulling a circular slot to make a round cut

Scaling

You can scale solids and surfaces with the Pull tool. You can scale multiple objects in different components.

To scale a solid or surface

1. Make sure the Select tool guide is active.
2. Select a solid or surface.
3. Click the Scale Body tool guide  .
4. Click a point, vertex, or origin to set the origin for scaling.
5. Pull to scale dynamically or press the spacebar and enter the scale.

   You can use enter expressions to calculate a scale factor.

The following expression elements are available:

- Infix (dyadic) operators: **+ - * / ^**
- Prefix (monadic) operators: **+ -**
- Functions: **sin cos tan asin acos atan sqrt log log10 exp**
- Constants: **pi e root2 root3**
- Units: **m cm mm yd ft in ' " deg rad**

Normal precedence rules apply:

1 + 2 * 3 ^ 4 => 1 + (2 * (3 ^ 4))) = 163

Parentheses are required for expression arguments and optional for simple arguments:

- sqrt 2 == sqrt(2) = 1.4142...
- sqrt 2*2 => (sqrt 2) * 2 = 2.8284...
- sqrt(2*2) = 2

Missing operators are inferred:

- 1 1/2 => 1 + 1/2

- 1'6" => 1' + 6"
- 1ft 6in - 17in => 1ft + 6in - 17in
- 1 2 3 4 5 => 1 + 2 + 3 + 4 + 5 = 15
- (1)(2)(3)(4)(5) => (1) * (2) * (3) * (4) * (5) = 120
- 2(1 + 2) => 2 * (1 + 2) = 6
- sqrt 2 sqrt 2 => sqrt 2 * sqrt 2 = 2
- 4(4atan(1/5) - atan(1/239)) => 4 * (4 * atan(1/5) - atan(1/239)) = pi

Units are applied to previous terms if units were not specified and are applied to subsequent terms unless you override them:

- 1 + 1cm => 1cm + 1cm
- 1cm + 1 => 1cm + 1cm
- 1cm + 1 + 1mm => 1cm + 1mm + 1mm
- 1cm + 1 1/2 mm => 1cm + 1mm + 1mm / 2

Trigonometry functions work in radians by default, but you can enter degrees:

sin(45 deg)

Numbers support standard form, but e is a built-in constant:

- 2e2 = 200
  2e 2 => 2 * e * 2 = 10.873...
- 2e-2 = 0.02
  2e – 2 => 2 * e – 2 = 3.436...
- 2e1 = 20
  2e => 2 * e

Do it faster	Select the solid or surface, Alt+click a vertex to anchor the scale, and pull.

To convert an existing solid or surface from millimeters to inches

1. Change the units to inches as described above.
2. Select the Pull tool in the Edit ribbon group on the Design tab.
3. Select the object you want to convert.
4. Scale the object by 25.4.

Copying edges and faces

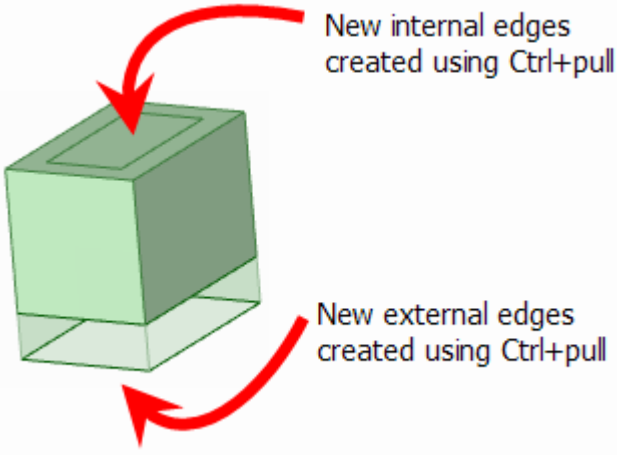
You can copy edges and faces by selecting the Pull tool's Extrude Edge option. You can also copy edges and faces with the Move tool.

To copy an edge or edges

1. Make sure the Select tool guide is active.
2. Select the edge or edges you want to copy. Double-click to select a tangent chain.
3. Select the **Copy Edge** option on the Options panel, or from the mini-toolbar.

   You can also Ctrl+drag with the Pivot Edge or Extrude edge options selected to copy the edge.

   The Pull arrow changes to show the two directions in which you can create copies of the edge. One arrow is highlighted to show the primary direction.

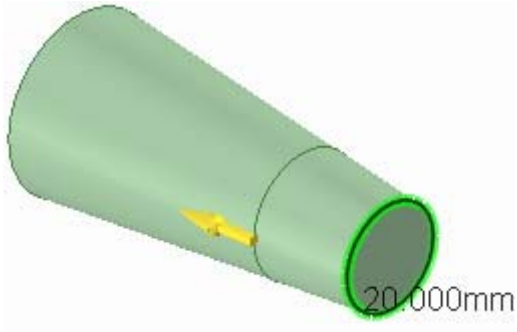


4. If the arrow pointing the direction in which you want to copy the edge is not highlighted, click the arrow or press **Tab** to change the direction.
5. Drag the edge in the direction of the highlighted Pull arrow.

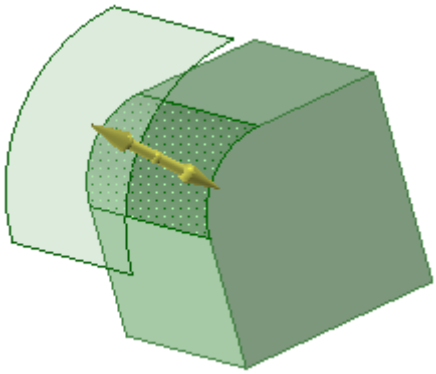
   During the pull, the distance between the copied edge and the new edge is displayed. When you create an external edge, a surface is created between the copied edge and the new one.

Do it faster Press **Ctrl** and drag to copy the selected face.

Example



When copying an edge, the edge adjusts based on the solid's geometry



Copying a round face

Moving

Use the Move tool to move any object, including drawing sheet views. The behavior of the Move tool changes based on what you have selected. If you select a solid or surface, you can rotate or translate it. If you select a face or edge, you can pull or move around it. You can move 2D and 3D objects.

When moving long distances, we recommend that you make a series of small movements to get there. When moving many faces, make sure to manually select all the faces that should be moved, instead of relying on SpaceClaim to automatically grab the interstitial faces.

You can also use the Select tool to move objects by dragging them.

To move an object

1. Select the object you want to move.
2. Use the Move handle to move the object.

Detailed instructions

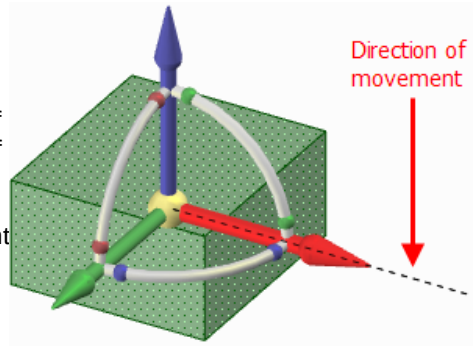
1. Select the Move tool  from the Edit ribbon group.
2. Select the object that you want to move to display the Move handle.

   You can select multiple objects. (Use the Select Component tool guide to select the solid or component to which a clicked object belongs.)
3. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.
4. (Optional) Drag the center point of the Move handle to anchor it to any face or edge in your design.

   You can also use the Anchor tool guide to select the face, edge, or vertex on which to place the Move handle. The yellow center sphere turns into a blue cube when the Move handle is anchored.
5. (Optional) Re-orient the Move handle.

   Drag one of the small balls on the rotational axis to reorient the Move handle, or dimension the orientation by typing the rotation angle *while you are dragging*, thenpressing **Enter**.

   You can also Alt+click a point or line (or click the Move Direction tool guide, then click a point or line) to orient one of the Move handle's axes toward that point or along that line. If you Alt+click a trajectory, you can move along the trajectory. (Ctrl+Alt+click to add contiguous lines or edges to the trajectory.) If you Alt+click a plane, the direction of movement is set perpendicular to the plane.



6 Click an axis and drag in that direction to move the selected object.

A dotted line extends from the Move handle axis to indicate the direction you selected for movement.

You can press **Ctrl** to copy the object selected for movement and place it at the location at which you drag or dimension the move. Press Ctrl+spacebar to make a copy and place it at the dimension you enter.

To dimension the move in Section mode, type the length of the move or the rotation angle (the angle can be positive or negative) and press **Enter**. You can also right-click and select **Create Ruler Dimension** from the mini-toolbar or the Options panel to dimension the move to any reference point along the move direction that you click.

The cursor does not need to be on the axis to move the selected object. In fact, you may find it easier to control the move if you drag some distance from the entity and the Move handle.

You can click the Up To tool guide and click a face or plane to move the selected object until the center of the Move handle is at the face or plane. You can also use this tool guide to rotate a solid, face, or surface until it is parallel to a face clicked with the Up To tool guide. Or use it to move to a point along a trajectory.

The Move tool respects offset, mirror, and coaxial inferred relationships, if they exist.

If you move a face that is in the same plane as another face, the face is detached.

If you move an object into another object in the same component, the smaller object is merged into the larger one, and receives the larger object's properties.

Moving a component moves all its subcomponents and objects.

If the move fails, the Move handle is repositioned to the last valid location and orientation. If you are trying to move a protrusion surrounded by round faces, you may need to fill the rounds.

You can also use the Move tool to:

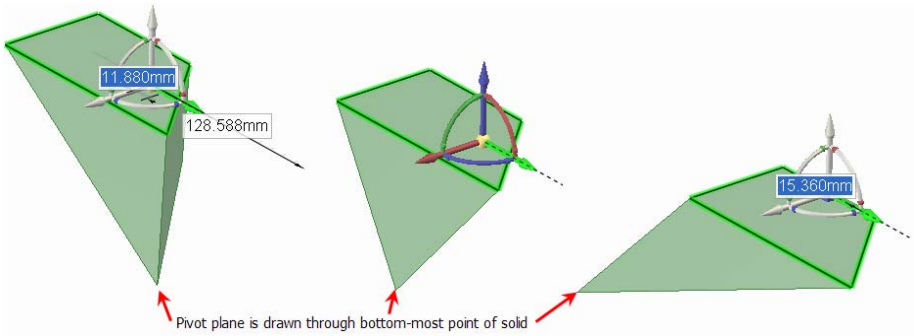
- Pivot solids

  1 Select an edge loop on the solid that you want to pivot.

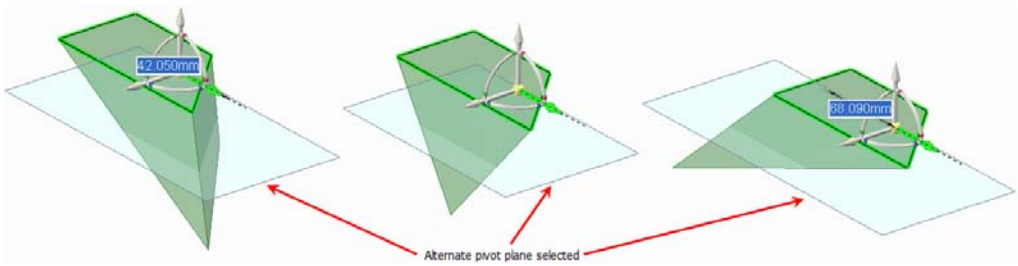
  2 Click a linear axis of the Move handle.

  3 (Optional). Click the Fulcrum tool guide and click an alternate plane to pivot around.

  4 Drag to pivot the solid around a plane drawn through a point opposite the selected edge loop or to pivot the solid around the plane you selected with the Fulcrum tool guide.



If the Move tool cannot maintain a planar or cylindrical face while pivoting, it will create a blended face.



- Pull solids

  You can use the Move tool to pull solids. If you select a face and move it, the solid is extended in the direction of the move.
- Create patterns
- Explode an assembly

Tool guides

Within the Move tool, there are several tool guides that let you specify the behavior of the Move tool:


The **Select** tool guide is active by default. When this tool guide is active, you can select faces, surfaces, solids, or components within the Move tool.


Click any object with the **Select Component** tool guide to select the solid to which the object belongs. Click again to select the component to which the object belongs.


Select a point, vertex, line, axis, plane, or planar face with the **Move Direction** tool guide to orient the Move handle and set the initial direction of the move. (The object will not move until you drag.)


Select a set of lines or edges with the **Move Along Trajectory** tool guide to move the selected objects along that trajectory. For best results, perform Moves along trajectories in small increments. If the object to be moved is a protrusion, it will be detached, then reattached in the new location. When you move a protrusion along a trajectory, rounds are automatically removed.


Select an object, then use the **Anchor** tool guide to select the face, edge, or vertex that will anchor the move. You can anchor the Move handle to a temporary object, such as the intersection between two axes by Alt+Shift+clicking the two objects.


Select an object, then use the **Fulcrum** tool guide to move other objects around it. Select a pattern member to anchor it, or select a component to explode an assembly.


Once you select the object to move and a Move handle axis, use the **Up To** tool guide to select the object you want to move up to. If a Move handle axis is selected, the Move is limited to that direction. (The axis must be selected to move up to the axis of an origin.) In a linear move to an intersecting object, the center of the Move handle is moved to the selected object. If the two objects do not intersect, the first object is moved parallel to the second. You can also select a point along a trajectory or the axis of an origin to move up to. You can also use this tool guide to move the sketch grid in Sketch and Section modes.


Once you select the object to move and a Move handle axis, use the **Orient to Object** tool guide to click an object. Your selected object will move until the selected Move handle axis is aligned with the clicked object. You can also use this tool guide to move the sketch grid in Sketch and Section modes.

Options

The following options are available in the Move tool:

Move Grid	Select this option to move the sketch grid.
 **Create Ruler Dimension**	Once you select an axis on the Move handle, select this option and click an edge or face to anchor the ruler. The ruler is oriented along the selected Move handle axis. Enter a value to use the ruler to dimension the move.

Maintain Orientation	Select this option to maintain the orientation of the object when rotating or moving along a trajectory.
Create Patterns	Select this option if you want to create a pattern by Ctrl+dragging selected objects with the Move tool. Ctrl+dragging creates a copy of the selected object and moves it to a new location. If this option is enabled, a pattern relationship is also created.
Detach First	Select this option to detach a protrusion, move it, and reattach it at the new location.

Moving in two dimensions

Use the Move tool to move lines or points with the Move handle. When you move points or lines with this tool, they do not maintain their connections to other lines or points. If you want to maintain the connections in your sketch, use the Select tool to edit the sketch.

To move sketch entities

1. Click the Move tool.
2. Select the line or point you want to move.
3. Use the Move handle to move the line or point.

Detailed instructions

1. Select the Move tool  from the Edit ribbon group.
2. Select the line or point on the sketch grid that you want to move to display the Move handle.

   You can select multiple sketch entities.
3. (Optional) Drag the center point of the Move handle to anchor it to any endpoint or midpoint of any line on the sketch grid.

   This functionality is useful when you want to rotate a sketch entity around another point on the sketch.
4. (Optional) Click the Move Direction tool guide and click a line or edge to reorient the Move handle.

   You can also Alt+click the line or edge to reorient the Move handle.
5. Click an axis and drag in that direction to detach and move the selected sketch entity.

   To dimension the move, type the length of the move or the rotation angle and press **Enter**.

   The cursor does not need to be on the axis to move the selected object. In fact, you may find it easier to control the move if you drag some distance from the entity and the Move handle.

Creating patterns

You can create a pattern of protrusions or depressions (including slots), points, or components. You can also create a pattern from a mix of object types, such as a pattern of holes (faces) and bolts (imported components) In SpaceClaim, any pattern member can be used to modify the pattern after you create it. If the change cannot be made to all pattern members, the member that cannot change is still part of the pattern.

Any of the Move tool's tool guides are available to you while creating a pattern.

To create a linear pattern

1. Select a protrusion or depression to be the first member of the pattern.
2. Select the Move tool.
3. Check the **Create patterns** box in the Options panel.
4. (Optional) Alt+click to set the direction for the pattern (or use the Direction tool guide).

5. Ctrl+drag the first member to copy it to the location of the last member of the linear pattern. All pattern members will be created between the first member and this member in a straight line. You can use the Up To tool guide to set the position.
7. Press **Tab**, enter the number of members you want to have in the whole pattern in the **Count** field, and press **Enter** to create the linear pattern.

To create a rectangular pattern

Follow the steps for the linear pattern, but select a linear pattern as the first member of the pattern.

To create an arc, circular, or cylindrical pattern

1. Select a protrusion or depression to be the first member of the pattern.
2. Select the Move tool.
3. Check the **Create patterns** box in the Options panel.
4. Alt+click to center the Move tool on the center of the circular pattern.
5. Ctrl+drag the first member to copy it in a rotational direction.

   As soon as you pass a certain rotation threshold, you can preview the possible patterns. If you want a more closely-packed pattern, edit the pattern after you create it. If you want to create an arc pattern, mouse up at a location that does not form a circular pattern.
6. Mouse up to select the circular pattern.
7. Press **Tab** to enter the number of members you want to have in the whole pattern in the **Count** field.

To create a radial pattern

1. Select a protrusion to be the first member of the pattern.
2. Select the Move tool.
3. Check the **Create patterns** box in the Options panel.
4. Use the Direction tool guide to set the direction of the Move tool toward the axis.
5. Ctrl+drag the first member to copy it in a radial direction.

   As soon as you pass a certain rotation threshold, you can preview the possible patterns. If you want a more closely-packed pattern, edit the pattern after you create it. If you want to create an arc pattern, mouse up at a location that does not form a circular pattern. You can edit the incremental rotation angle.
6. Mouse up to select the radial pattern.
7. Press **Tab** to enter the number of members you want to have in the whole pattern in the **Count** field.

To create a radial circular pattern

1. Select all radial pattern members with the Move tool.
2. Re-anchor the Move tool on the circular axis.
3. Check the **Create patterns** box in the Options panel.
4. Ctrl+drag the radial pattern to form a circular pattern.

To create a pattern of points

1. Click a vertex with the Move tool.
2. Check the **Create patterns** box in the Options panel.
3. Click the Move Along Trajectory tool guide.
4. Click one of the edges that connects to the vertex.

5. Click the Move handle axis.
6. Ctrl+drag to create the end pattern member and create the pattern.
7. Click the new point with the Select tool to display and edit the count, the length along the edge, and the percentage of the edge that is between the first point and the end point. (For example, a point with a dimension of 50% appears at the midpoint of the edge.)

   Modify the count, length, and percent fields to edit the pattern of points. All points are associated to the edge, so that when the edge moves, the points move with it.

   When you create a pattern of points along an edge, points that are coincident to vertices are not included in the pattern.

To edit a pattern's properties

1. Select the face of one pattern member to display the pattern count and dimensions.
2. Edit the pattern's properties.

   Press **Tab** to switch between the fields.
3. Press **Enter**.

To move a radial pattern in a linear direction

1. Select all the members of the pattern.
2. Select the Direction tool guide.
3. Click an object to set the direction of the move.
4. Drag the pattern.

To move a pattern member

1. Select a pattern member.
2. Move the pattern member with the Move handle.

   If you move one of the middle members of a pattern, all the pattern members move, unless one is anchored.

   If you move a member at one end of a pattern, the member at the other end of the pattern automatically anchors itself, allowing you to adjust the spacing of the pattern.

   If you anchor a different member than the member opposite the direction you are moving, Move skews the pattern.

   If you have a linear pattern in a radial direction and you move a middle member without setting an anchor, then the entire pattern shifts in the selected direction.

To anchor a pattern member

1. Select the Fulcrum tool guide.
2. Click a face of the pattern member you want to anchor.

   You can anchor any member of a linear or rotational pattern.

To adjust linear pattern spacing

1. Anchor a middle pattern member.
2. Move another pattern member to adjust the spacing of the pattern around the anchored member.

To adjust circular pattern spacing

1. Select a member.

2 Drag the member with the Select tool to adjust the distance from the center of the pattern and the spacing between members.

To add rounds or chamfers to a pattern

1 Select the edge or edges of a pattern member.

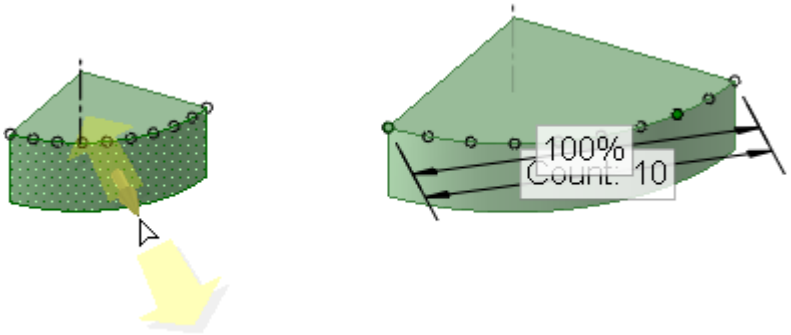
2 Right-click the selected edge and click **Select > Pattern Edges** to select all the corresponding edges in the pattern.

3 Modify the edge(s) to create a round or chamfer.

To remove a pattern member from the pattern

Right-click the face of the pattern member and select **Unpattern Member**.

Example



Pattern of points

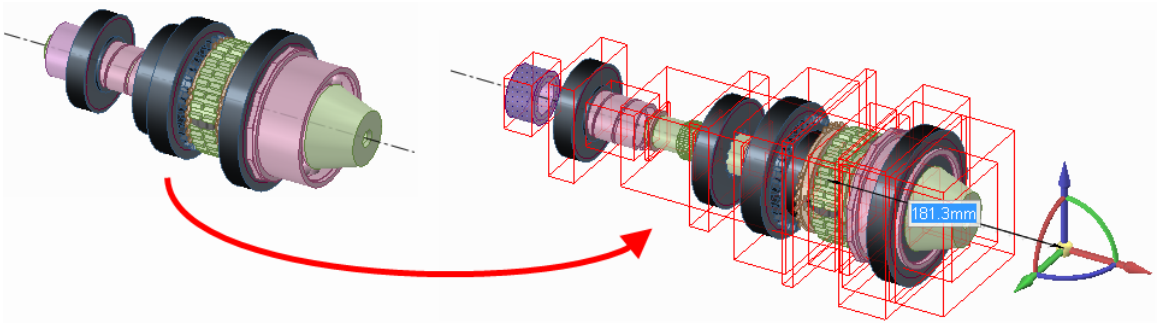
Exploding an assembly

1 Select all the components in the Structure tree that are part of the assembly you want to explode.

2 Anchor the Move handle on one component.

3 Select the **Fulcrum** tool guide and click another component.

4 Select an axis on the Move handle and drag to explode the assembly in that direction.



Filling

Use the Fill tool to fill in the selected region with the surrounding surface or solid. Fill can "heal" many cuts made into geometry, such as chamfers and rounds, subtractive revolves, protrusions, depressions, and regions removed by removing regions in the Combine tool. The Fill tool can also be used to simplify surface edges and cap surfaces to form solids.

You can use the Fill tool in Sketch mode to fill a sketch line that is almost closed, but that has multiple small gaps. If the gaps are too large, multiple error message appear to show you where the gaps are. You can also use it when editing a layout. Fill functionality is useful when you sketch faces across section lines, but do not want the section lines to split the surfaces when you switch to 3D.

To fill a region

1. Select the edges that define a surface region, or the faces that define a region within or on a solid.

   You can select an object in the Structure tree to simplify it.

2. Click the Fill tool  or press **F**.

To fill sketch or layout lines

1. Select a closed or almost closed loop of sketch lines.

2. Click the Fill tool  or press **F**.

   If a gap is 1.5 times the length of the minor grid spacing on the sketch grid, the edges are extended to close the gap. If the gap is larger, a message appears in the status bar and the gap's endpoints flash.

   The mode is switched to 3D mode, and the filled loop becomes a surface.

   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. If you fill lines in a layout mode, you can then pull the surface into 3D from the layout, but remain in edit layout mode after this action.

   You can fill lines and edges whether or not the sketched lines you want to fill were sketched in the same plane as the edges. (If the lines are imprinted on a face and become edges, filling those edges deletes them.)

Do it faster	Click the Fill tool in Sketch mode to fill any closed or almost closed loops and switch to 3D mode.

You can use the Fill tool to:

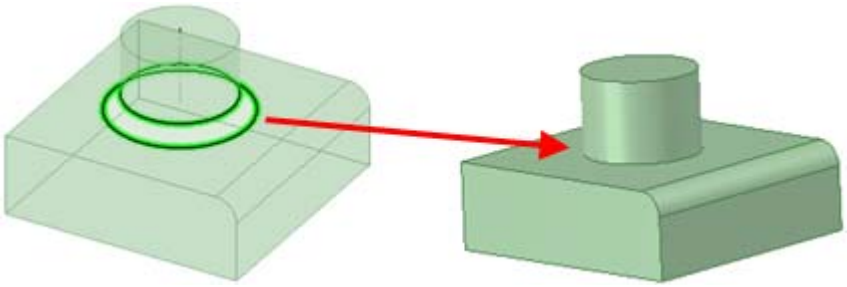
- Cap a surface

  Select the edges of the surface you want to cap and click the Fill tool.

  For example, to fill an uncapped cylinder, select the open edge and click the Fill tooll.

- Patch a surface

  Select the edges of a void in the middle of a surface or the edges of a void on the edge of a surface and click the Fill tool.



If you select two surface edge loops and select the Fill tool, the surfaces are extended until they intersect.

- Fill a round or chamfer

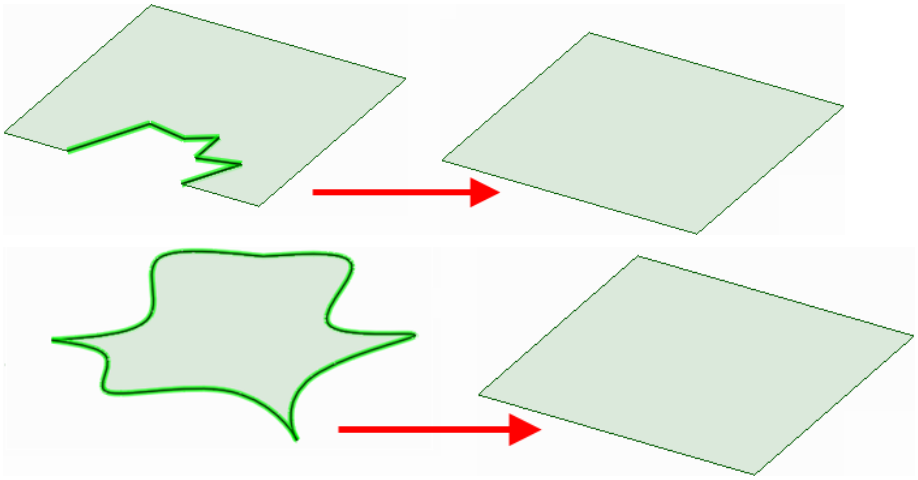
  Select the round or chamfer and click the Fill tool to fill the round and create caps if necessary.

  A RoundGroup is created in the Groups panel each time you fill a round.

  (You can fill any round created in SpaceClaim, even a round that removes the underlying face, until the round is changed by some other action.)
- Remove a protrusion or depression

  Select the protrusion or depression and click the Fill tool.
- Simplify edges

If you fill the edges of a face, the edges are simplified into straight lines. You can also press **Delete** to simplify the edges.



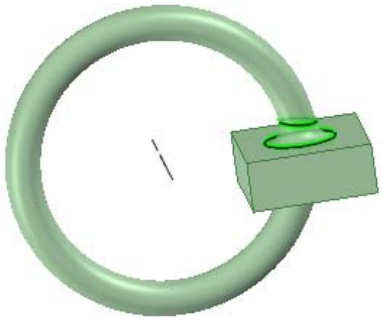
You can also fill all the edges of a spline surface to extend all the edges to form a simpler, larger surface. You can also press **Delete** to simplify the edges.

You can also fill complex edges on any other surface, including cones.

You can simplify multiple 3D edges across multiple surfaces or solids into one edge. Just select the edges you want to merge and click Fill.

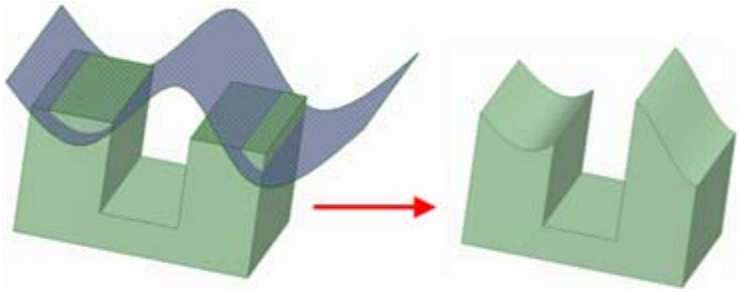
If you want to keep internal lines, select them before clicking the Fill tool.

- Fill multiple edge loops



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.

- Replace a face



1. If you are replacing a single face, click the face you want to replace.
2. Alt+click the replacement face.
3. Select the Fill tool.

- Replace multiple faces

Ctrl+click any number of faces with the Select tool. Then Ctrl+Alt+click any number of additional faces. Click the Fill tool to replace the first set of faces with the second set of faces.

- Remove rounds

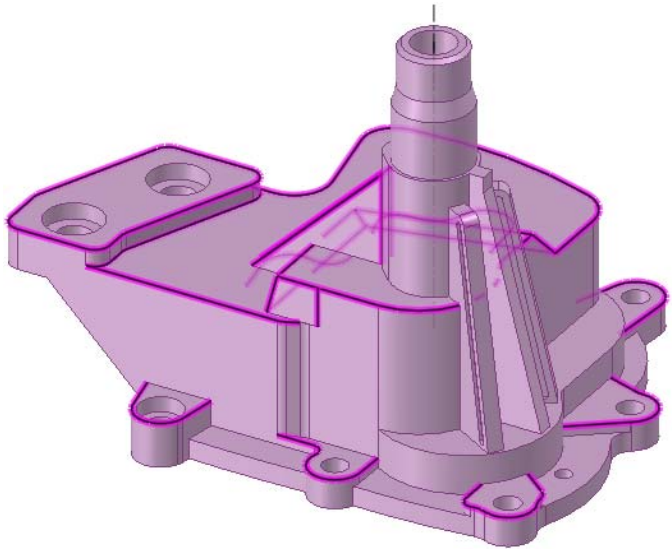
To fill a round

1. Select the round face.
2. Click the Fill tool  or press **F**.

When multiple 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 image on the right shows edges from which rounds were removed by the Fill tool.

If removing a face makes an invalid solid, the solid is converted to a surface.

If a round face is difficult to remove, try Alt+clicking it with the Fill tool, or using the Replace tool.



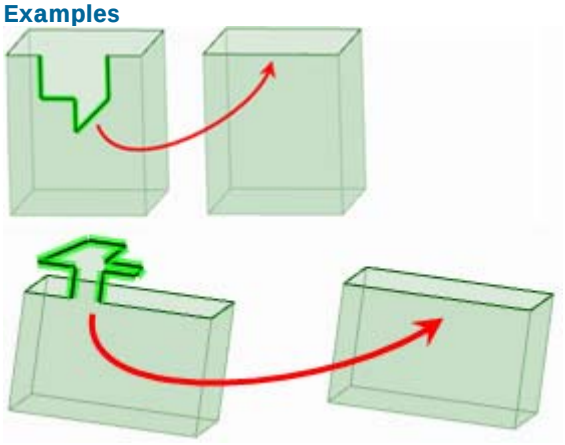
Capping round faces

You can use the Fill tool to remove and cap round faces. You might want to do this if you are trying to move a protrusion surrounded by round faces and it cannot be moved because the rounds create impossible geometry.

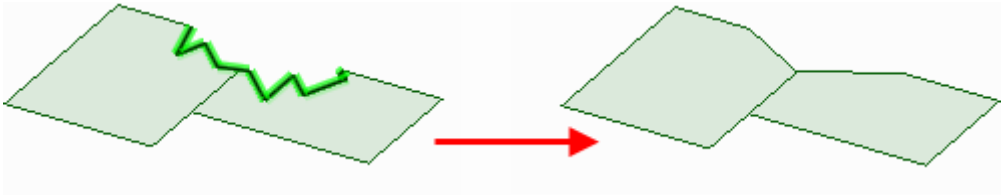
Usually you can just select all the round faces and click the Fill tool to remove and cap them all. However, sometimes this is not possible. In that case, select one round and fill it. If that works, undo and select that round and the next. Undo. Continue adding rounds to your selection and trying to fill them until the fill fails. Now you have identified one of the rounds that is causing the

problem. Next, fill all the rounds that filled successfully. Finally, repeat this process in the other direction of the round tangent chain. Once you have filled all the rounds except for the one or two causing the problem, select the one causing the problem and its two neighbors. Then click Fill. This process allows more options for the extension of neighboring edges to intersect and cap the round.

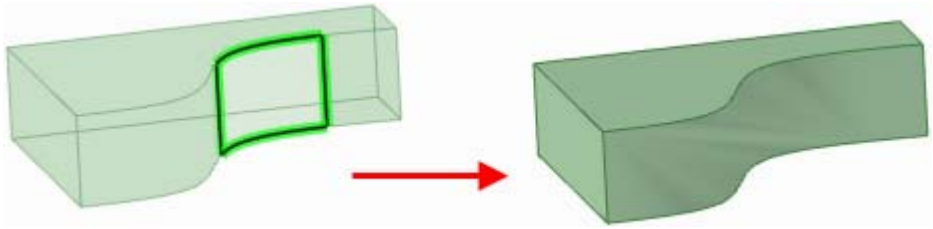
Examples



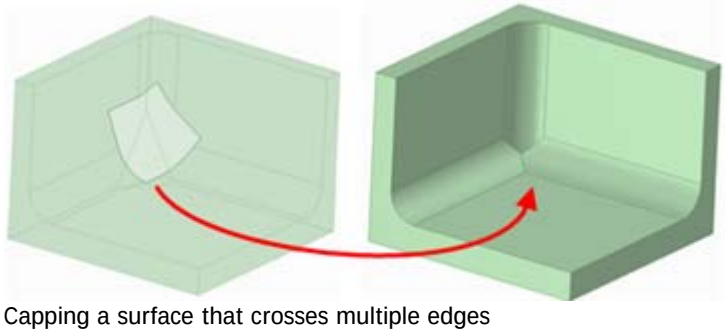
Simplifying edges



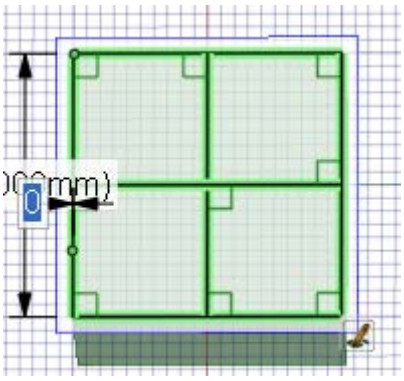
Simplifying edges across multiple faces



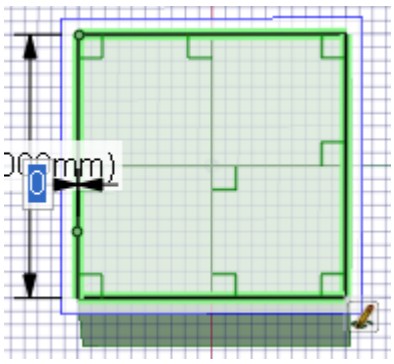
Capping a surface



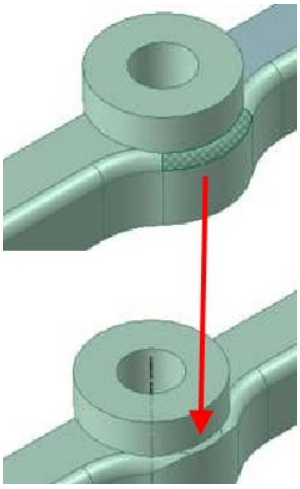
Capping a surface that crosses multiple edges



Selecting internal edges to keep them after filling.

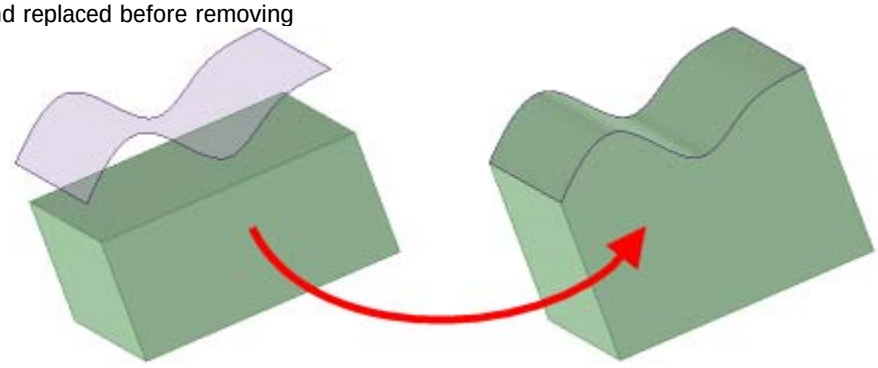


Selecting lines to simplify a surface by filling. Internal edges are removed.



A difficult-to-remove round replaced before removing

Replacing faces



Use the Replace tool to replace one face (or multiple faces) with another face (or faces). You can also use it to simplify a spline face that is very similar to a cylinder, or align a set of planar faces which are almost aligned.

To replace a face

1. Click the Target tool guide.
2. Select the face you want to replace.

   You can also select multiple faces.
3. Click the Source tool guide.
4. Click the face (or faces) you want to replace the target face with.

5 Click the Complete tool guide to replace the target face with the source face.

Do it faster	Click one face, then the next to replace the first with the second.

To simplify or align faces

1 Click the Target tool guide.

2 Ctrl+click each face.

The faces are displayed with a pulsating red glow to indicate that they will be simplified or aligned.

2 Click the Complete tool guide.

Tool guides

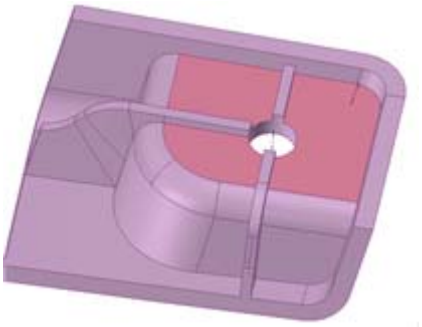
Within the Replace tool, there are several tool guides that help step you through the editing process:


If you did not pre-select the face that you want to replace, you can select it from within the Replace tool using the **Target** tool guide.


Click the face you want to replace the target face with using the **Source** tool guide.


Click the **Complete** tool guide to replace the target face with the source face, or to simplify or align the target face.

Examples



Faces that are almost planar targeted for alignment

Editing faces as a blend

You can use the Edit as Blend tool to edit any blended face or convert a regular face to a blended one. You can create a new blend section or delete one. You can move any blend section, or you can edit it by moving an edge, moving a vertex, or bending an edge.

If a face was originally created as a blend, the original blend surfaces will appear for editing, even if the faces originated as imported geometry. Any two planar faces can be edited as a blend, but a single face cannot be edited as a blend.

You may find it useful to display a grid on the face when editing the face as a blend.

To edit a face as a blend

1 Select the faces you want to convert to blended surfaces, or select existing blended faces.

2 Use the Edit as Blend tool to display the blend sections.

3 Move, rotate, or edit the interior blend sections.

You can also add or delete interior blend sections.

Detailed instructions

1 Select the faces you want to convert to blended surfaces, or select existing blended surfaces.

We recommend that you use the more powerful Select tool to do this, but you can also select faces within the Edit as Blend tool after the next step.

2 Click the Edit as blend tool  in the Edit ribbon group to display the blend planes.

If you did not pre-select the faces, you can select them in the Edit as Blend tool using the Select tool guide  .

Click a blend plane to select it.

3 Move, rotate, or edit the interior blend planes.

To move or rotate the blend plane

Click the Move tool guide  and drag the Move handle to move or rotate the blend plane.

To move a vertex or edge

Click the Edit tool guide  , click a vertex or edge of the blend section, and drag it to edit the blend plane.

To bend an edge

Click the Bend tool guide  , click an edge, then drag in or out to bend the edge.

4 Add or remove interior blend planes.

To add a blend plane

Click the Add Blend Plane tool guide  and click the face to insert a new blend plane through the face at that location. The orientation of the new plane is determined by the orientation of its neighboring blend planes.

To remove a blend section

Click a blend plane, then click the Remove Blend Plane tool guide  to delete it. The faces are redrawn between the remaining blend planes. If you delete the wrong blend plane, you can always press **Ctrl+Z** to undo.

Tool guides

Within the Edit as blend tool, there are several tool guides that help step you through the editing process:

If you did not pre-select the faces to edit, you can select them from within the Edit as Blend tool using the Select tool guide.


Select the Move Blend Plane tool guide to move and orient the selected blend plane using the Move handle.

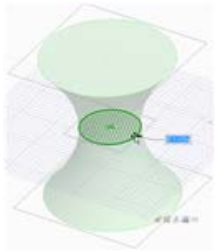

Make edits to any blend section with the Edit Blend Section tool guide. You can move, bend, and replace edges or move vertices. Just click to select the vertex or edge you want to edit.


Select the Bend tool guide and select a linear or arced edge to bend it.


Select the Add Blend Plane tool guide to add new blend sections by clicking on the face to insert a new blend section through the face. You can modify your new blend sections just like the others.


Select the Remove Blend Plane tool guide to click any blend section to delete it.

Examples



"Waisting" a cylinder

Moving the sketch grid

Use the Move Grid tool to move the sketch grid. Make successive sketches by moving the grid after sketching closed line regions. These closed lines turn into regions when you move the grid.

To move the sketch grid

1. Click the Move Grid tool  on the mini-toolbar or click the Move tool and check the **Move Grid** option.
2. (Optional) Select any sketch entities that you want to move along with the sketch grid.
3. Select a Move handle axis.
4. Drag along the axis of the Move handle to move or rotate the sketch grid.

   Press **Shift** while dragging to snap the move to angular and linear increments based on your snap settings as well as to snap the move parallel to planes, edges, and axes. You can also right-click and select **Use Ruler Dimension**, and enter a value or press **Enter** to drag the grid. You can also use the standard Move tool guides when moving the grid.

   The Move handle can be moved around by dragging the center ball onto appropriate entities on the sketch grid.

Intersecting

You can use the intersect tools to merge and split a solid or surface in your design with another solid or surface. You can merge and split solids or surfaces with other solids or surfaces, split a solid with a face, and split a face with another face. You can also project the edges of a face onto other solids and surfaces in your design.

SpaceClaim's intersection capabilities include the full suite of geometry combination, all done with one major tool (Combine) and two minor ones (Split Solid and Split Face). Combine always takes two or more objects. The split tools always act on one object and that object is automatically selected from the cutter or projection face.

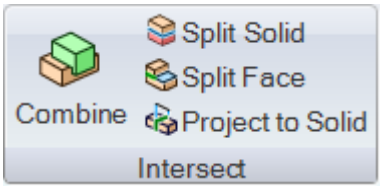
To understand what the Combine tool can do, the first step is to know which objects can be handled. Solid objects have faces that meet at corner edges. Edges that lie on faces can be deleted, but corners cannot. Surface objects have faces that come together at internal edges, and are surrounded by outside edges. Outside edges can be extended. Internal edges can be deleted if they are bounded by planar faces. For the purposes of combine, planes can be thought of as surface faces that extend across the design.

When surfaces enclose a volume, they automatically change into solid objects. When edges of the same surface become coincident, they will automatically merge. Planes cannot be split by any combine operation, but they can be used to split with. In general, layer, color, and visibility in the structure tree propagates from the first selected item (the target) to the result. What happens as a result of the combination can be overridden with the options (both in the panel and the mini-toolbar). When you use the Combine or Split tools, the newly created objects have the layer properties of the previous objects.

You have complete control over the pieces that solids and surfaces get cut into. When appropriate, SpaceClaim prompts you to remove regions, but you can choose to keep or remove those regions.

When using the intersect tools, the original, individual layers and colors of the objects are maintained.

Intersect ribbon group



The Intersect ribbon group contains the following tools:


Use the Combine tool to merge and split solids and surfaces.


Use the Split Solid tool to split a solid by one or more of its faces or edges. Then select one or more regions for deletion.


Use the Split Face tool to create an edge on a face by splitting it with another face or surface.


Use the Project to Solid tool to create edges on a solid's face by extending the edges of another solid or surface.

Combining and splitting

Use the Combine tool to merge and split solids and surfaces. These actions are sometimes known as Boolean operations.

The Combine tool 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. This mode works just like any other tool. You can box-select multiple targets and cut them without automatically merging them.

  For example, if you click the Select Target tool guide, you can click an object to add it to the selection and click a selected object to remove it from the selection. You can also use box-select. (Nothing is merged.)

  If you click the Merge tool guide, the selected targets are merged. You can click more objects to merge them to the targets or use box-select.

  If you click the Cutter tool guide, every object you click becomes a cutter and cuts the target immediately. You can also use box-select to create cutters.

  To delete regions created by the cuts, you must click the Delete Regions tool guide then click the region you want to delete.

- "Fast mode" automatically activates tool guides and moves you through the workflow. This predictive mode exhibits the same behavior as the Combine tool had in previous SpaceClaim releases, although you can now box-select multiple cutters to add even more efficiency.

  In both modes, if you select objects, then click the Combine tool, the objects are automatically merged. The procedures in this section describe Fast mode behavior.

To merge solids and surfaces

1. Select the Combine tool.
2. Click the first solid or surface.
3. Ctrl+click additional solids or surfaces to merge them.

Do it faster	Ctrl+click the solids or surfaces you want to combine in the Structure tree, then click the Combine tool to merge them.

Detailed instructions

1. Select the Combine tool  from the Intersect ribbon group or press **I**.
2. Click the first solid or surface.
3. Ctrl+click additional solids or surfaces to merge them.

   The Structure tree shows the merge.

Solids can be merged with solids, and surfaces with surfaces. Solids and surfaces can only be merged if the surfaces make a region that can be added to or cut out of the solid. For example, if you select the face of a cylinder, copy and paste it, then you can merge it with the solid used to create it.

To split solids and surfaces

1. Select the Combine tool.
2. Select the target.
3. Select the cutter.
4. (Optional) Select the region of the target to delete.

Do it	If you want to split a body by a face of the body, select the Split Solid tool.

faster If you want to create an edge on a face, select the Split Face tool.

Detailed instructions

1. Click the Combine tool  from the Intersect ribbon group or press **I**.
2. Select the solid or surface you want to cut.

   The Select Cutter tool guide is activated.
3. (Optional) Control the behavior of the cut by selecting options.
4. Click the surface you want to use to cut the solid.

   Depending on the option you chose, the cutting surface will be kept or deleted. Review the information in the Structure panel to see the solids or surfaces created by the cut.

   Mouse over the target solid to see the regions created by the cut.
5. Click each region you want to delete.

   When you are finished selecting the areas to be deleted, click the Select Target tool guide to combine some more, or select another tool.

Tool guides

Within the Combine tool, there are several tool guides that help step you through the splitting process.


The Select Target tool guide is active by default. If you did not pre-select the target solid or surface, you can select it from within the Combine tool using the Select tool guide.


Click the Select Bodies to Merge tool guide to select multiple solids or surfaces to merge together. Tool guides that appear with a double outline are "sticky," and allow you to perform the same action repeatedly. For example, when this tool guide appears with a double outline, just keep clicking objects to merge each object with the previous one. To "unstick" the tool guide, you can click it again, click another tool guide, or click an empty place in the Design window


The Select Cutter tool guide activates once you select a target. When this tool guide is active, click to select the solid or surface you want to use to cut the target. You can Ctrl+click when this tool guide is active if you need to add other solids to your cutter selection.


The Select Regions tool guide activates once the target is cut. When this tool guide is active, mouse over the target to preview the regions created by the cut. Click a region to delete it.

Options

The following options are available for the Combine tool. Select these options from the Options docking panel, or right-click and select them from the mini-toolbar:

Merge when done	Select this option to merge all touching solids or surfaces when you exit the Combine tool. Hidden objects are not merged.
Make curves	Select this option to create edges at intersections instead of selecting regions. You will not be able to preview regions for deletion. As soon as a region is selected for deletion, this option is disabled.
Keep cutter	SpaceClaim assumes that you created a cutter object to be used only for cutting. If you want to keep the cutting surface in your design, select this option. If this option is not selected, then the cutting surface is automatically deleted as soon as you select it. In other words, cutter objects are normally "used up" unless you select this option. A kept cutter can be a surface or solid, but either way only the regions of the target can be removed. If you are splitting surfaces, check this option to prevent the cutter object from being split by the target object.

Make all regions

Select this option to cut the target object with the cutter object and the cutter object with the target object. Target and cutter must be the same type of object, either both solids or both surfaces. Because this option can create a large number of regions, we recommend using this option along with the **Merge When Done** option to quickly merge all remaining regions when you click another tool or press **Esc** to finish using Combine.

Merging

Use the Combine tool to merge solids and surfaces.

To merge solids and surfaces

1. Select the Combine tool  from the Intersect ribbon group or press **I**.
2. Click the first solid or surface.
3. Ctrl+click additional solids or surfaces to merge them.

   The Structure tree shows the merge.

Do it faster	Ctrl+click the solids or surfaces you want to combine in the Structure tree, then click the Combine tool to merge them.

Solids can be merged with solids, and surfaces with surfaces. Solids and surfaces can only be merged if the surfaces make a region that can be added to or cut out of the solid. For example, if you select the face of a cylinder, copy and paste it, then you can merge it with the solid used to create it.

Using the Combine tool, you can:

- Merge two solids

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the first solid.
3. Ctrl+click the second solid to merge it with the first solid.
   The second solid takes on the color and visibility properties of the first solid. The Structure tree shows the merge.

- Merge many solids

1. Select the solids you want to combine using box selection.
2. Select the Combine tool  from the Intersect ribbon group.
   The solids take on the color and visibility properties of the merged solid that appeared top-most in the Structure tree. The Structure tree shows the merge.
   You can also select the Combine tool, select the first solid, then Ctrl+click each individual solid you want to merge.

- Add material to a solid with a surface that forms a protrusion

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the first solid.
3. Ctrl+click a surface that could create a protrusion on the solid to merge it with the solid.
   The surface becomes a protrusion on the solid and takes on the color and visibility properties of the first solid. The Structure tree shows the merge.

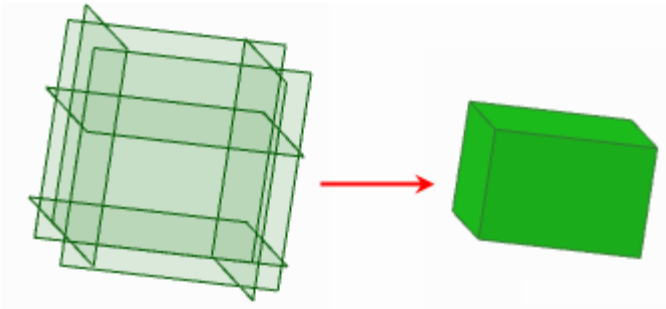
- Merge two surfaces that share edges

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the first surface.
3. Ctrl+click the second surface to merge it with the first surface.
   The second surface takes on the color and visibility properties of the first surface. The Structure tree shows the merge.

- Merge surfaces that intersect to form an enclosed region

1. Select the surfaces that intersect to form an enclosed region using box selection.
2. Select the Combine tool  from the Intersect ribbon group.
   The enclosed region becomes a solid and takes on the color and visibility properties of the merged surface that appeared top-most in the Structure tree. The Structure tree shows the merge.

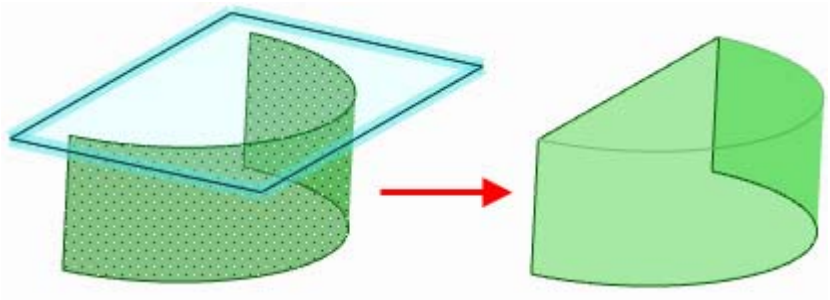
Example



Six surfaces enclosing a volume, box-selected and combined with the Combine tool

- Merge a surface and a plane

Ctrl+click a surface and a plane to cap the surface.



Merging solids

You can merge two or more solids with the Combine tool.

To merge two solids

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the first solid.
3. Ctrl+click the second solid to merge it with the first solid.
   The second solid takes on the color and visibility properties of the first solid. The Structure tree shows the merge.

To merge multiple solids

1. Select the solids you want to combine using box selection.
2. Select the Combine tool  from the Intersect ribbon group.
   The solids take on the color and visibility properties of the merged solid that appeared top-most in the Structure tree. The Structure tree shows the merge.

You can also select the Combine tool, select the first solid, then Ctrl+click each individual solid you want to merge.

Merging surface protrusions

You can merge a surface that forms a protrusion to a solid.

To merge a surface protrusion

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the first solid.
3. Ctrl+click a surface that could create a protrusion on the solid to merge it with the solid.
   The surface becomes a protrusion on the solid and takes on the color and visibility properties of the first solid. The Structure tree shows the merge.

Merging surfaces

You can merge surfaces with the Combine tool. Surface must share edges to be merged. If surfaces enclose a volume, you can use the Combine tool to quickly convert the enclosed volume into a solid and trim the extra.

Sometimes, if you manipulate a surface so that it becomes self-intersecting, you may need to create a solid from the surface that remains.

To merge surfaces that share edges

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the first surface.
3. Ctrl+click the second surface to merge it with the first surface.
   The second surface takes on the color and visibility properties of the first surface. The Structure tree shows the merge.

To merge surfaces that enclose a volume

1. Select the surfaces that intersect to form an enclosed region using box selection.
2. Select the Combine tool  from the Intersect ribbon group.
   The enclosed region becomes a solid and takes on the color and visibility properties of the merged surface that appeared top-most in the Structure tree. The Structure tree shows the merge.

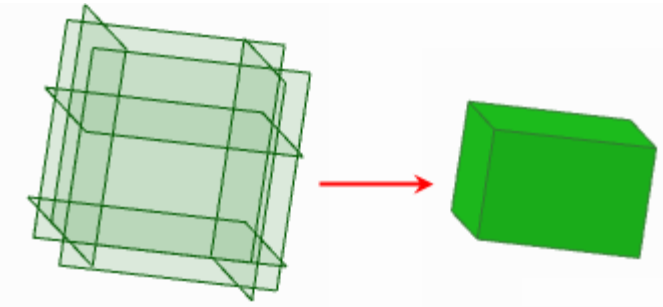
To create a solid from a self-intersecting single surface

1. Click the Select tool in the Edit ribbon group.
2. Select the "open" edge of the set of surfaces, shown in black.
3. Click the Combine tool.

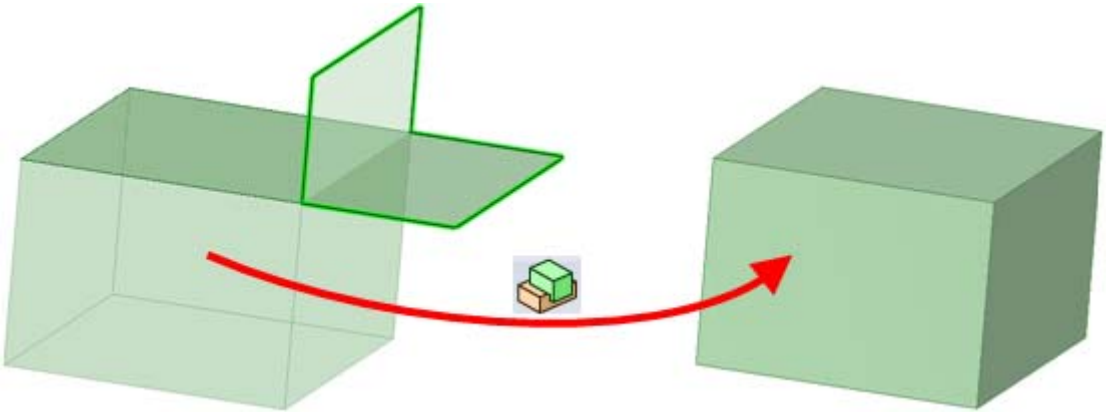
To repair a self-intersecting surface

1. Click the Select tool in the Edit ribbon group.
2. Double-click the edge loops.
3. Click the Combine tool.

Example



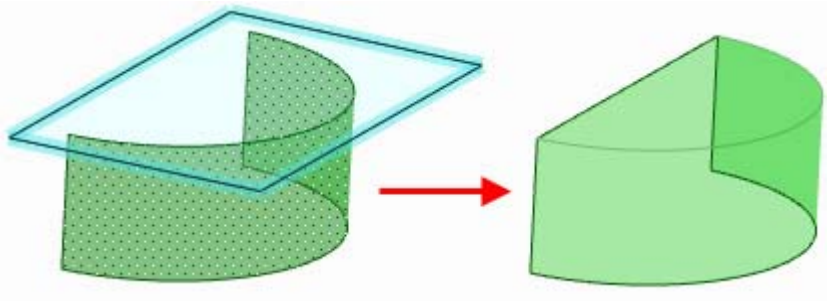
Six surfaces enclosing a volume, box-selected and combined with the Combine tool



Repairing a self-intersecting surface with the Combine tool

Capping a surface with a plane

Ctrl+click a surface and a plane to cap the surface.



Splitting

Use the Combine tool to split solids and surfaces.

To split solids and surfaces

1. Click the Combine tool  from the Intersect ribbon group or press **I**.
2. Select the solid or surface you want to cut.
   The Select Cutter tool guide is activated.
3. (Optional) Control the behavior of the cut by selecting options.

4 Click the surface you want to use to cut the solid.

Depending on the option you chose, the cutting surface will be kept or deleted. Review the information in the Structure panel to see the solids or surfaces created by the cut.

Mouse over the target solid to see the regions created by the cut.

5 Click each region you want to delete.

When you are finished selecting the areas to be deleted, click the Select Target tool guide to combine some more, or select another tool.

Do it faster	If you want to split a body by a face of the body, select the Split Solid tool. If you want to create an edge on a face, select the Split Face tool.

Using the Combine tool, you can:

- Split a solid with a surface or plane

1 Select the Combine tool  from the Intersect ribbon group.
2 Click the solid you want to cut.
3 Click the surface you want to cut the solid with.

You can Ctrl+click multiple surfaces that together completely intersect a solid.

4 Mouse over the solid to see the regions created by the cut.
5 Click the region you want to delete.

- Split a solid with a solid

1 Select the Combine tool  from the Intersect ribbon group.
2 Click the solid you want to cut.
3 Click the solid you want to cut with.
4 Mouse over the solid to see the regions created by the cut.
5 Click the region you want to delete.

- Split a surface with a solid or plane

1 Select the Combine tool  from the Intersect ribbon group.
2 Click the surface you want to cut.
3 Click the solid or plane you want to cut the surface with.
4 Mouse over the surface to see the regions created by the cut.
5 Click the region you want to delete.

- Split a surface with a surface

1 Select the Combine tool  from the Intersect ribbon group.
2 Click the solid you want to cut.
3 Click the surface you want to cut the solid with.

You can Ctrl+click multiple surfaces that together completely intersect the target surface.

4 Mouse over the solid to see the regions created by the cut.
5 Click the region you want to delete.

- Remove material from a solid with a surface that forms a depression

1 Select the Combine tool  from the Intersect ribbon group.
2 Click the solid you want to cut.

3. Click the surface you want to use to create a depression.
4. Mouse over the solid to see the regions created by the cut.
5. Click the region you want to delete.

- Remove an enclosed volume from a solid

1. Create the exterior solid and interior solid in two different components.
2. Select the Combine tool  from the Intersect ribbon group.
3. Click the exterior solid.
4. Click the interior solid to use it as the cutter.
5. Click the interior solid to delete it.

Splitting a solid

Use the Combine tool to split solids. Solids can be split by surfaces, planes, and other solids.

To split a solid with a surface or plane

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the solid you want to cut.
3. Click the surface you want to cut the solid with.
   You can Ctrl+click multiple surfaces that together completely intersect a solid.
4. Mouse over the solid to see the regions created by the cut.
5. Click the region you want to delete.

To split a solid with another solid

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the solid you want to cut.
3. Click the solid you want to cut with.
4. Mouse over the solid to see the regions created by the cut or click another solid to cut with.
5. Click the region you want to delete.

Splitting a surface

Use the Combine tool to split solids. Surfaces can be split by solids, planes, and other surfaces.

To split a surface by a solid or plane

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the surface you want to cut.
3. Click the solid or plane you want to cut the surface with.
4. Mouse over the surface to see the regions created by the cut.
5. Click the region you want to delete.

To split a surface by another surface

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the surface you want to cut.
3. Click the surface you want to cut the surface with.

   You can Ctrl+click multiple surfaces that together completely intersect the target surface to fully cut the surface. You can box-select surfaces that only partially intersect the target surface to partially cut the surface..
4. Mouse over the surface to see the regions created by the cut.
5. Click the region you want to delete.

Removing material from a solid

Use the Combine tool to remove material from a solid based on the intersection of a solid or surface. You can remove the material defined by a surface that forms a depression, or remove a solid completely enclosed within another solid.

To remove material from a solid using a surface that forms a depression

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the solid you want to cut.
3. Click the surface you want to use to create a depression.
4. Mouse over the solid to see the regions created by the cut.
5. Click the region you want to delete.

To remove an enclosed volume from a solid

1. Create the exterior solid and interior solid in two different components.
2. Select the Combine tool  from the Intersect ribbon group.
3. Click the exterior solid.
4. Click the interior solid to use it as the cutter.
5. Click the interior solid to delete it.

Splitting a solid

Use the Split Solid tool to split a solid by one or more of its faces or edges. Then select one or more regions for deletion. The Split Solid tool expects that you have already selected the cutter objects. From these faces or edges, the single target body is inferred, since a face or edge can only belong to one solid or surface. If only one solid face is selected as a cutter, the default action is to extend that face to cut as far as it can through the solid. If a face of a surface is selected, then it is automatically removed.

The Split Solid tool works in two modes:

- "Slow mode" is activated by clicking a Split Solid tool guide. In this mode, once a tool guide is clicked, it remains selected until another tool guide is clicked. This mode works just like any other tool.
- "Fast mode" automatically activates tool guides and moves you through the workflow. This predictive mode allows you to box-select multiple cutters to add even more efficiency.

To split a solid by one of its faces

1 Click the Split Solid tool from the Intersect ribbon group.

2 Click the faces or edges you want to use to cut the body.

Mouse over the solid to see the regions created by the cut.

2 (Optional) Click a region to delete it.

When you are finished deleting regions, select another tool.

If the selected edges do not completely encircle a portion of a solid or surface, no region selection can occur.

Do it faster	Select a face with the Select tool, then select the Split Solid tool to cut the solid with the face.

You can use the Split Solid tool to:

- Split a solid by its edges
- Split a solid by its faces

Tool guides

Within the Split Solid tool, there are several tool guides that help step you through the splitting process:

The Select Cutter tool guide is active by default. When this tool guide is active, click to select the face you want to use to cut the solid.

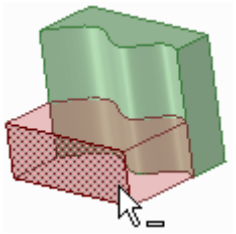
The Select Regions tool guide activates once you once the solid is cut by the face. When this tool guide is active, mouse over the target to see the regions created by the cut.

Options

The following options are available in the Split Solid tool. Once you select the edge or face to pull, select these options from the Options docking panel, or right-click and select them from the mini-toolbar:

Merge when done	Select this option to merge all touching solids or surfaces when you exit the Split Solid tool. Hidden objects are not merged.
Extend faces	Extends the selected cutter face to cut through the target solid.

Example



Solid split by a spline face

Splitting a face

Use the Split Face tool to create an edge on a face or surface by splitting it with another face or edge.

To create an edge on a face

1. Click the Split Face tool  from the Intersect ribbon group.
2. (Optional) Select a tool guide.

   Mouse over faces or edges in your design to preview the edge that will be created on the target.
3. Click to select the face or edge you want to use to split the selected face.

Detailed instructions

Split a face with another face

1. Click the Split Face tool  from the Intersect ribbon group.
2. Click to select the face you want to split.

   You can Ctrl+click to select multiple faces.
3. Click the **Select Cutter Face** tool guide.

   Mouse over faces in your design to preview the edge that will be created on the target.
4. Click the face or surface to split the selected face with an edge.

Split a face using a point on an edge

1. Click the Split Face tool  from the Intersect ribbon group.
2. Click to select the face you want to split.

   You can Ctrl+click to select multiple faces.
3. Click the **Select Cutter Point** tool guide.

   Mouse over the edges of the face to preview the edges that will be created.
4. Click a point on the edge to split the selected face.

   The percent distance along the edge is displayed.

Split a face using two points on edges

1. Click the Split Face tool  from the Intersect ribbon group.
2. Click to select the face you want to split.

   You can Ctrl+click to select multiple faces.
3. Click the **Select Two Cutter Points** tool guide.
4. Click a point on an edge.

   Mouse over edges of the face to preview the edges that will be created.
5. Click a point on another edge to split the selected face.

Tool guides

Within the Split Face tool, there are two tool guides that help step you through the splitting process:

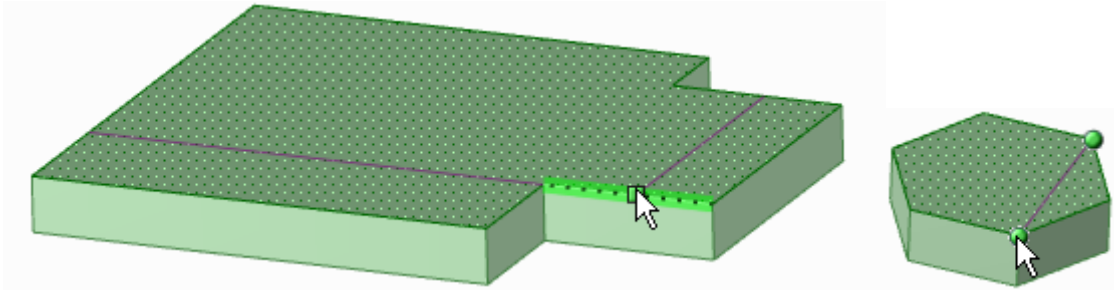
The Select Target tool guide is active by default. If you did not pre-select the target face or surface, you can select it from within the Split Face tool using the Select tool guide. Ctrl+click multiple surfaces or solid faces in the same plane to split them all.

The Select Cutter Face tool guide activates once you select a target. When this tool guide is active, click to select the face or surface you want to use to create an edge on the target.

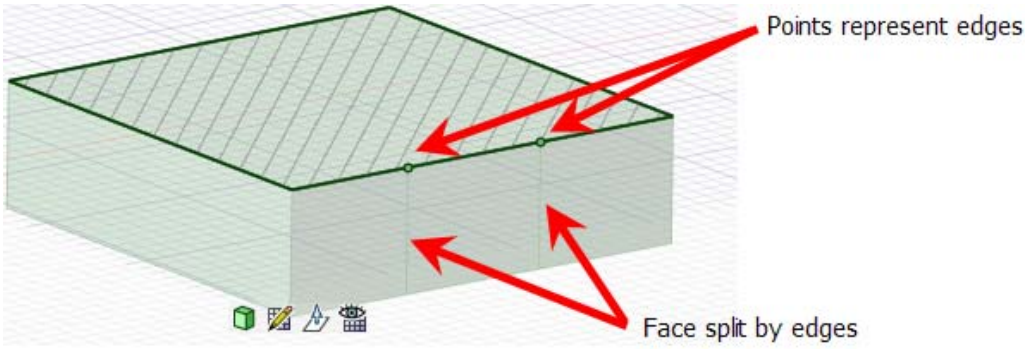
The Select Cutter Point tool guide activates once you select a target. Mouse over an edge to preview the new edges that will be created. Click to create the edge on the selected face. You can mouse over an edge with this tool guide to display and edit the length along the edge, and the percentage of the edge that is between the first point and the end point.

The Select Two Cutter Points tool guide activates once you select a target. Click to select the first point on an edge, then mouse over another edge to preview the new edge that will be created. Click to create the edge on the selected face. You can mouse over an edge with this tool guide 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.

Examples



Previewing edges that can be created using the Select Cutter Point and Select Two Cutter Points tool guides



Splitting a face in section mode by selecting points on a section line

Projecting to a solid

Use the Project to Solid tool to create edges on a solid's face by extending the edges of another solid, surface, sketch, or note text.

To project the edges of a face, surface, sketch, or note text to a solid

1. Select the Project to Solid tool from the Intersect ribbon group.
2. Click the face, surface, sketch or note text whose edges you want to project.

   The projection occurs perpendicular to the selected object onto the nearest solid faces. The nearest solid faces are determined automatically.
3. (Optional) Alt+click a face or edge to set another direction for the projection.

To project edges to selected faces of a solid

1. Select the Project to Solid tool from the Intersect ribbon group.
2. Click the face, surface, or note text whose edges you want to project perpendicular to the face.

The nearest solid faces are determined automatically.

3 Select the **Use Selected Faces** option in the Options panel.

The purple edges disappear.

4 Select the faces you want to project onto individually.

Options

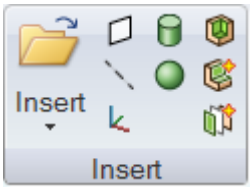
The following options are available for the Project to Solid tool. Select these options from the Options docking panel, or right-click and select them from the mini-toolbar:

Use Selected Faces	Select this option to project only onto the faces you choose.
Project Through Solids	Select this option to project edges on all faces through the entire solid instead of just the faces closest to the "projector" face.
Extend Projected Edges	When a projected edge does not completely span across a face, this option extends the lines until another edge is reached.

Inserting

The Insert tools allow you to insert components, images, planes, axes, origins, cylinders, and spheres, and create relationships between the solids and surfaces in your design.

Insert ribbon group



The Insert ribbon group contains the following tools:

 Use the Insert tool to insert a component or an image into your design.

 Use the Plane tool to insert a plane into your design.

 Use the Axis tool to insert an axis into your design.

 Use the Origin tool to insert an axis into your design at the location of the Move handle.

 Use the Cylinder tool to quickly create a cylinder.

 Use the Sphere tool to quickly create a sphere.

 Use the Shell tool to convert a solid to a shell.

 Use the Offset tool create an offset relationship between faces in your design.

 Use the Mirror tool to designate a face or plane as a mirror.

Inserting a component

You can insert another design created in SpaceClaim or another application as a component of your design. The design is inserted as an external component, linked to the external file. You can make this component internal to your own design. Lightweight components are displayed with solid edges and transparent faces until you right-click the component and select **Load component** from the context menu.

To insert another design as a component

1. Click the Insert tool  in the Insert ribbon group.
2. Navigate to and select the design you want to include as a component, then click **Open** on the Open Design window.

   Progress is displayed on the status bar until the design is inserted as an external component. You can insert multiple copies of the same component. Modifying one of these components will modify every copy unless you make them internal to your design.

Make an external component internal to your design

1. Right-click the external component in the Structure tree.
2. Select **Use Internal Copy** from the context menu.

If your design includes multiple copies of an external component, making one of them internal does not affect the other copies. Making another copy of the same external component internal creates a second instance of the same internal component.

Inserting an image

You can insert an image into your design or onto a drawing sheet. Inserted images and their transparency colors, aspect ratio, and placement information are saved with the design.

To insert an image

1. Select **Image** from the Insert drop-down in the Insert ribbon group.
2. Select the BMP, JPG, PNG, or TIF image file you want to insert and click **Open**.
3. Click on the face that defines the plane on which you want to insert the image.

   An ImagePlane appears in the Structure tree and the image is placed on the plane.

To resize an image

Select the image and drag the control handles.

You can right-click the image and select **Properties** to set whether you can adjust the aspect ratio in the **Keep aspect ratio** property in the Properties panel.

To rotate an image

Drag the handle at the top of the image to rotate the image.

You can also drag the handle not connected to the image edge to change the center of rotation.

To flip an image

Drag handle across the other side of the image to flip the image.

To copy an image

1. Select the image.
2. Press **Ctrl** and drag the image to create a copy of the image.

To set the transparency of the image

Change the value in the **Transparency** property.

To set a color in the image to be transparent

1. Click the **Transparent Color Table** property drop-down to display the color selection window.

   Mouse over the pixels in the image to display their RGB values.
2. Click  and click a pixel in the image that is the color you want to become transparent.

   You can use the Zoom slider to zoom into the image and drag to pan the image within the window.

   The color you selected appears in the drop-down next to the eyedropper icon. You can add multiple transparent colors by repeating this process, and remove the selected color (or all colors) using the eraser icons. The image window previews the image with the transparency you selected.

Inserting a plane

You can define a plane by selecting various faces, edges, axes, or lines in your design. The plane will extend a small amount beyond the edges of the face to make it easier to work with. The plane will maintain this margin around all the objects in your design, extending and trimming itself as you add, remove, or move solids within your design.

Planes are useful for creating layouts and annotations. Planes are not linked to the geometry used to create them; they must be explicitly selected to move along with other items. You can also use a plane to quickly clip your design to show only the geometry behind the plane.

To insert a plane

1 Select one of the following:

Select a:	To insert a plane:
Planar face	That contains the face
Planar face and point	Through the point and parallel to the face
Planar face and edge	Through the edge and perpendicular to the face (useful when creating a sweep path)
Two parallel planar faces	At the midpoint between the two faces
Axial or cylindrical face	Tangent to the face at the selection point
Two axial faces with parallel axes	Tangent to both faces as close as possible to the selection points
Planar edges	That contains the edges
Axis	That contains the axis
Two parallel axes	That contains the axes
Two origin axes	That contains the axes
Axis (or line) and one point	That contains the axis (or line) and the point
Endpoint of any line	Through the endpoint and normal to the line at the endpoint
Three points	That contains those three points
Line sketched on a plane	Through the line and normal to the plane
Line and point on the line	Through the line and the point
Two planar lines	That contains those two lines

You can also select temporary objects to define planes.

2 Select the Plane tool  from the Insert ribbon group to insert the plane.

(You can also select the Plane tool first, then select an object.) An error message appears in the status bar if your selection does not define a plane.

To insert a plane tangent to a cylinder and parallel to a plane

Select a plane, Ctrl+click a cylinder, then click the Plane tool.

To insert a plane tangent to a cylinder and perpendicular to a plane

Select a cylinder, Ctrl+click a plane, then click the Plane tool.

To insert a plane through the mid-point between two points

1 Select the Plane tool.

2 In 3D mode or Section mode, Alt+Shift+click two points to create a temporary plane that includes the midpoint.

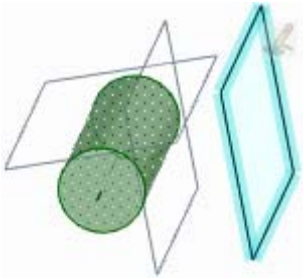
3 Click the temporary plane.

To clip your design with a plane

Right-click a plane and select **Clip with Plane**. To restore the view of your design, right-click the plane and select **Clip with Plane** again.

Planes, axes, and annotations are not clipped.

Example



Planes inserted parallel and perpendicular to the selected plane, and tangent to the selected cylinder

Inserting an axis

You can extend any axis in your design to make it available for revolves or rotations. The axis will extend a small amount beyond the edge of the space containing your design. It will maintain this margin extending and trimming it as you add, remove, or move objects in your design. Axes are not linked to the geometry used to create them (unless they are an extension of existing axes); they must be explicitly selected to move along with other items.

To insert an axis

1 Select one of the following:

Select a:	To insert an axis:
Axial face	That contains the face's axis
Two, non-parallel planar faces	At the intersection of the theoretical planes containing the two faces
Linear edge	That contains the edge
Cylinder and tangent plane	At the intersection of the cylinder and plane
Line	That contains the line
Origin axis	Along the origin's axis
Circle or arc	That contains the center point and is normal to the line
Two points	Through both points

You can also select temporary objects to define planes.

2 Select the Axis tool  from the Insert ribbon group to insert the axis.

To insert an axis through the mid-point between two points

1 Select the Select tool.

2 Alt+Shift+click two points to create two temporary axes that cross at the midpoint.

3 Click the temporary axis.

4 Select the Axis tool to create the axis.

Inserting an origin

You can insert an origin at any location in your design where you can anchor the Move tool. You can anchor a ruler to an origin, and snap to origins while in Sketch mode. This allows you to dimension from an origin and quickly move the sketch grid to an origin. You can also insert an origin at a solid's center of mass or volume.

To insert an origin

1. Select the Move tool.
2. Position the Move handle where you want the origin to appear.
3. Select the Origin tool to insert an origin.

   The origin appears in the Design window and in the Structure tree.

To insert an origin at the center of mass or volume

1. Select the Mass tool from the Analysis ribbon group.
2. Select the solid at whose center you want to place the origin.

   Axes appear at the center of mass or volume.
3. Select the Origin tool.

   The origin appears at the center of mass or volume.

To insert an origin at the mid-point between two points

1. Select the Move tool.
2. Alt+Shift+click two points to create a temporary point at the mid-point.
3. Click the temporary point to place the Move handle there.
4. Select the Origin tool.

   The origin appears at the midpoint.

Creating a cylinder

Use the Cylinder tool to sketch the cylinder's axis in 2D and create its diameter in 3D.

The sketch grid must be visible in the workspace before you can draw.

To draw a cylinder

1. Click to set the first endpoint of the cylinder's axis.
2. Click to set the other endpoint.
3. Click to set the diameter of the cylinder.

Do it faster Click and drag to draw the axis, then click to set the diameter.

Detailed instructions

1. Select the Cylinder tool from the Insert ribbon group.
2. (Optional) If you want to dimension the first endpoint of the axis, press **Shift** and hover the mouse over a line or point to create a dimension relative to that line or point.
3. Click or press **Enter** to set the first endpoint of the axis.
4. (Optional) Dimension the axis.

5 Click or press **Enter** to set the other endpoint of the axis.

By default, the axis is dimensioned to its first endpoint. You can also press **Shift** and hover the mouse over another line or point to create a dimension relative to that line or point.

6 (Optional) Dimension the cylinder's diameter.

7 (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

8 Click or press **Enter** to set the diameter of the cylinder.

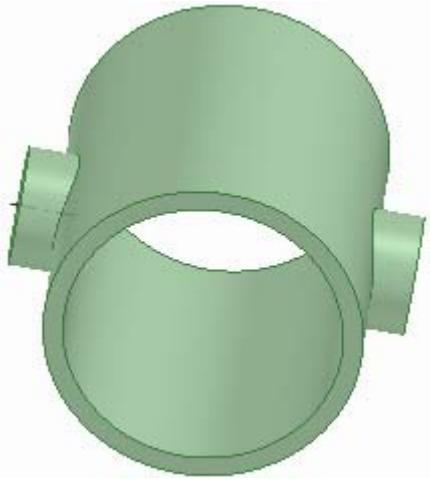
Options

The following options are available in the Cylinder tool.

Add Add material to create a solid cylinder.

Cut Remove material when the cylinder intersects another object.

Example



Tube with nubs

Creating a sphere

You can quickly create a sphere using the Sphere tool.

To create a sphere

1 Select the Sphere tool from the Insert ribbon group.

2 Click to set the center of the sphere and the plane in which the sphere's radius is dimensioned.

As you move the mouse, you can see a preview of the sphere. Sphere creation works best when you move the cursor in the x-y direction of the plane indicated by the first click.

3 Click to set the radius of the sphere.

To move a sphere

Drag the sphere's center with the Select tool in Section mode to move the sphere. The section must cut through the center of the sphere to move it.

Converting a solid to a shell

Use the Shell tool to remove one of the faces of the solid and create a shell of a specified thickness. You can then use the Shell tool to remove other sides of the shell. SpaceClaim automatically creates an offset relationship between the sides of the solid for you.

To create a shell

1. Select the Shell tool from the Insert ribbon group.

   Mouse over the solids in your design to highlight the faces that could be removed.
2. (Optional) Enter a value into the dimension field to change the thickness of the shell.

   Enter a negative number to create the shell thickness from the outside of the solid.
3. Click the face you want to remove.

   The face is removed and a shell is created. The baseline of the offset is shown in blue. If you did not change the thickness, the default thickness is set by the minor grid spacing.

   Continue clicking to remove multiple faces.

To create a closed shell

1. Select the solid.
2. Click the Shell tool to create an internal shell.

To edit a shell

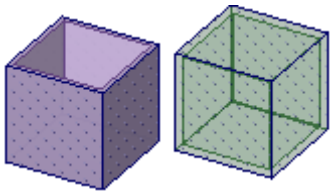
Right-click the shell to change its thickness.

Tool guides

This tool guide is active by default. Select a face of the solid to remove it and create a shell. Ctrl+click to remove multiple faces.

If you create a shell, then add a protrusion onto it, you can extend the shell through the protrusion by clicking the More Shell tool guide, then clicking the newly added protrusion.

Example



Open shell and closed shell

Creating an offset relationship

Use the Offset tool to create an offset relationship between two faces. This relationship will be maintained in the other 2D and 3D editing tools. For example, when you select an offset face with the Pull tool, the offset dimension is displayed. Baseline faces do not display a dimension.

To create an offset relationship

1. Click the Offset tool in the Insert ribbon group.

   Mouse over your design to see the faces eligible for offset relationships.

2. (Optional) Select options.

   Select the **Find All Same Offset** option if you want to select all contiguous face pairs that have the same offset distance as the pair you selected. If this option is not selected, the offset relationship is only created for the selected pair of faces.

2. Click the first face.

3. Click the second face.

   You have now established an offset relationship between the face pair and other contiguous face pairs with the same offset distance (if the All the Same option is selected). The Toggle Baseline tool guide activates.

4. (Optional) Click to select the baseline face(s).

   When you move one of the faces with a design tool, the other face in the offset pair moves to maintain the offset relationship.

Tool guides

Within the Offset tool, there are several tool guides that help step you through the editing process:

The Face Pair tool guide is active by default.

When the Toggle Baseline tool guide is active, click the faces that you want to be the baseline of the offset.

Mirroring objects

Use the Mirror tool to designate a face or plane as a mirror, or create a mirror plane between two faces. The plane becomes a persistent mirror once you create geometry using the mirror. You can quickly mirror solids, or you can use the Mirror Face tool guide to mirror faces. The mirror relationship is maintained when you use other 2D and 3D tools to edit your design.

Mirrored objects are created on the same layer as the original object. Mirrored points do not maintain their mirror relationship.

To mirror in Sketch mode

1. Draw a line or construction line.
2. Right-click the line and select Set as Mirror Line.
3. Sketch on one side of the line to mirror the sketch on the other side.

To mirror objects

1. Select the plane you want to be a mirror.
2. Click the object you want to mirror.

Detailed instructions

1. (Optional) Create the plane you want to use as a mirror using the Insert Plane tool and position it with the Move tool.
2. Click the Mirror tool in the Insert ribbon group.

Mouse over the solids in your design to preview the geometry that can be created by the mirror.

3 Click the object you want to mirror.

You can click solids or surfaces in your design or click a component in the Structure tree.

You can use the Mirror tool to:

- Mirror a body or face

  1 Click the Mirror Body  or Mirror Face tool guide  .

  Mouse over the solids or faces in your design to preview the solid or face that will be created on the other side of the mirror plane.

  2 Click the solid or face to create the mirrored solid or face.

  The mirror plane is created and will persist in other tools.

  If you mirror faces that enclose a volume, a solid is created.

- Set up a mirror relationship between two faces

  1 Click the Setup Mirror  tool guide.

  As you mouse over the faces in your design, the eligible faces are highlighted. A face is eligible if it is parallel to another face in your design.

  2 Click a face.

  3 Click a parallel face.

  The second face is highlighted in blue and the mirror plane appears halfway between the two faces. Certain actions performed on one face will now be mirrored on the other face.

- Remove a mirror face or plane

  1 Click the Remove Mirror  tool guide.

  2 Click the face from which you want to remove the mirror relationship.

  You can temporarily disable a plane in any tool. Select a mirrored face to display the mirror plane, then click the mirror icon to disable the mirror. Click the mirror icon again to enable the mirror.

- Find similar faces on the other side of a mirror plane

  1 Click the Setup Mirror  tool guide.

  2 Click a face.

  Ctrl+click to select multiple faces.

  3 Alt+click a mirror plane.

  Identical faces equidistant from the mirror plane are highlighted in blue, and a mirror relationship is created.

Tool guides

Within the Mirror tool, there are several tool guides that help step you through the mirroring process:


The Mirror Plane tool guide is active by default. Select a face or plane to use it as a mirror. (You can also use this tool guide to select another mirror plane to use if one is already selected.


The Mirror Body tool guide activates automatically once you select a mirror face or plane. Mouse over the solids in your design to preview the solid that will be created by the mirror. Click a solid to mirror it.


Once you select a mirror face or plane, use the Mirror Face tool guide to select the faces you want to mirror. Mousing over the faces before you click them previews the face that will be created by the mirror. Click a face to mirror it.


The Setup Mirror tool guide can be used any time to create a mirror plane between two faces. Only those two faces will be affected by the mirror. If you click a face then Alt+click a mirror plane, you can use this tool guide to create a mirror relationship between identical faces equidistant from the plane.


Use the Remove Mirror tool to remove the mirror relationship between two faces.

Mirroring a solid, surface, or face

You can mirror a solid, surface, or face.

To mirror a solid or surface

1. Click the Mirror Body  or Mirror Face tool guide  .

   Mouse over the solids or faces in your design to preview the solid, surface, or face that will be created on the other side of the selected mirror plane.
2. Click the solid or face to create the mirrored solid or face.

   The mirror plane is created and will persist in other tools.

   If you mirror faces that enclose a volume, a solid is created.

Set up a mirror relationship between two faces

You can create a mirror plane between two parallel faces in your design.

To set up a mirror relationship

1. Click the Setup Mirror  tool guide.

   As you mouse over the faces in your design, the eligible faces are highlighted. A face is eligible if it is parallel to another face in your design.
2. Click a face.
3. Click a parallel face.

   The second face is highlighted in blue and the mirror plane appears halfway between the two faces. Certain actions performed on one face will now be mirrored on the other face.

Remove a mirror face or plane

You can remove or disable a mirror.

To remove a mirror face or plane

1. Click the Remove Mirror  tool guide.
2. Click the face from which you want to remove the mirror relationship.

   You can temporarily disable a plane in any tool. Select a mirrored face to display the mirror plane, then click the mirror icon to disable the mirror. Click the mirror icon again to enable the mirror.

Find similar faces on the other side of a mirror

You can create mirror relationships by finding similar faces on the other side of an existing mirror.

To find similar faces on the other side of a mirror

1. Click the Setup Mirror tool guide.
2. Click a face.

   Ctrl+click to select multiple faces.
3. Alt+click a mirror plane.

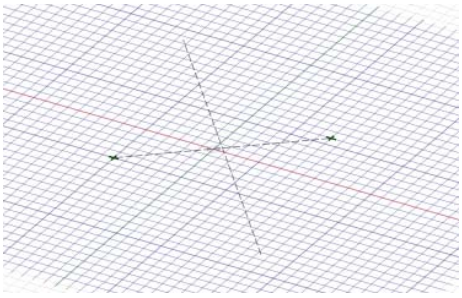
   Identical faces equidistant from the mirror plane are highlighted in blue, and a mirror relationship is created.

Inserting temporary objects

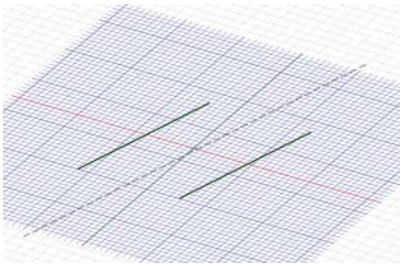
You can create temporary points, axes, and planes while working with other tools. Temporary objects can be used to dimension to and create other objects. For example, you can use them to create a ruler dimension, or anchor the Move handle on them. These temporary object disappear when you switch tools.

To create a temporary:	Alt+Shift+click:
Midpoint between two points and a bisecting line	Two points
Bisector lines and intersection point	Two non-parallel lines
Line midway between two lines	Two parallel lines
Circle that completes an arc	An arc
Plane midway between two planes	Two parallel planes
Bisecting planes and edge of intersection	Two non-parallel planes
Focus point	Cone
Major and minor foci	Ellipse

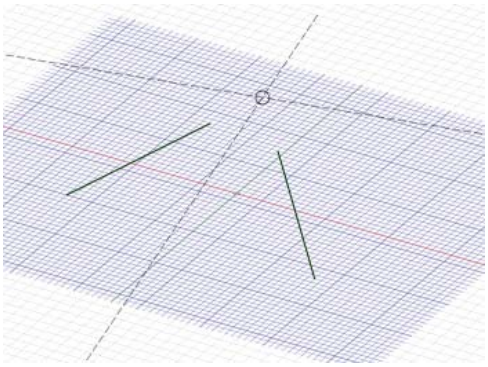
Examples



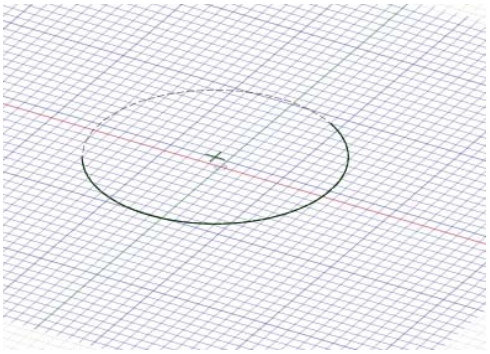
Midpoint between two points and a bisecting line



Line midway between two parallel lines



Bisector lines and intersection point



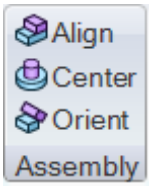
Circle that completes an arc

Assembling components

In SpaceClaim, a component consists of any number of objects, such as solids and surfaces. You can think of a component as a "part." A component can also contain any number of sub-components. You can think of a hierarchy of components and sub-components as an "assembly." The assembly tools work on components within the assembly; to enable the tools you must select two objects in different components. When working with components, you may want to specify how they are aligned with each other, that is, create a mating condition. Mating conditions that you create with the Assembly tools are shown in the Structure tree.

You can create multiple mating conditions for your components. If your components are not fitting together the way you expect, try clicking the checkbox next to the mating condition in the Structure tree to turn off the mating condition. A mating condition that cannot be satisfied is indicated with a different icon in the Structure tree. You can toggle the conditions on and off or delete them in the Structure tree. When creating mating conditions, the component clicked first will contain the mating conditions.

Assembly ribbon group



The Assembly ribbon group contains the following tools:

- Align the selected faces of objects in two different components with the Align tool.
- Align the selected axes of objects in two different components with the Center tool.
- Rotate components around their alignment axis so that the selected faces point in the same direction with the Orient tool.

To toggle a mating condition

Uncheck the mating condition checkbox in the Structure tree to disable the mating condition. Check the box to enable the mating condition.

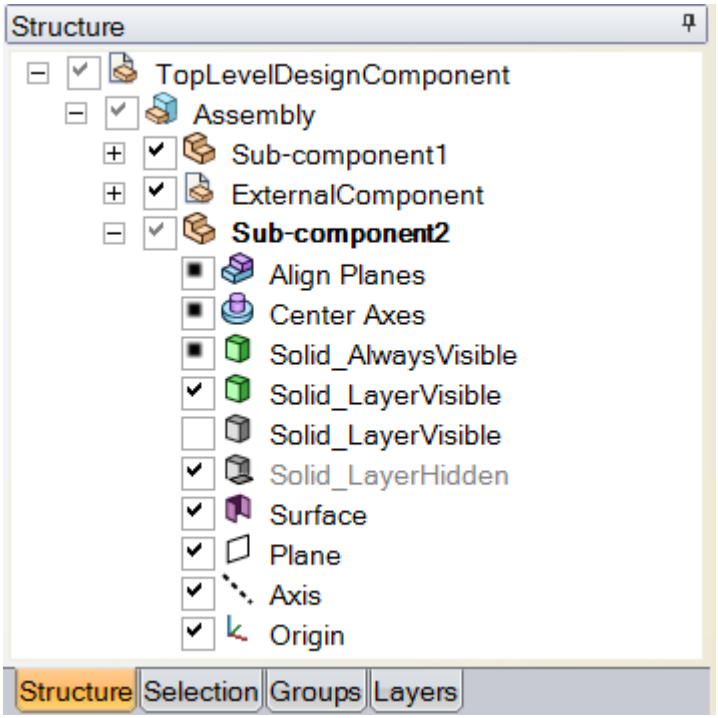
To delete a mating condition

Right-click the mating condition in the Structure tree and select **Delete Mating Condition**.

To reverse the sense of the mating condition

Right-click an Align or Orient mating condition in the Structure tree and select **Reverse Sense** to align the components to the opposite side of the alignment plane.

Working with components



The Structure tree contains the Structure tree, which shows you each of the objects in your design. You can quickly show or hide any component using the checkbox next to the object's name. You can expand or collapse the nodes of the tree, rename objects, create, modify, replace, and delete objects, create components, copy components, make components independent, open a component in a new design window, set the component to be a sheet metal component, make a component active, make a component independent, delete, rename, or display properties.

Offset, mirror, and shell relationships stay with a solid when it is moved to another component, unless the relationship would link two components when it is moved.

The top-level design (called **StructureTree** in the image on the right) is also a component.

If you are working with a single instance of an external component, make that instance independent to prevent your changes from being made to the external component file. Once you make a copied sub-component independent, you can modify it without changing any of the other instances of that sub-component. Or you can modify one of the other instances to change all the copied sub-components except the one you made independent.

If your design includes multiple copies of an external component, making one of them internal does not affect the other copies. Making another copy of the same external component internal creates a second instance of the same internal component.

Do it faster	Shift+click and Ctrl+click multiple objects to work with them as a group.

About lightweight components

When you insert an external file into your design, if you have the Enable lightweight assemblies advanced SpaceClaim option enabled, only the component's graphic information is loaded. This allows you to quickly view the component with the Orient tools and load the geometry information when you are ready to work with it in SpaceClaim.

To create a component

Right-click top-level design (or another component) in the Structure tree and select **New Component** from the context menu to create a new component or sub-component.

To copy a component

1. Select a component and click the Copy tool or press **Ctrl+C**.
2. Select the component under which you want to create a copy and click the Paste tool or press **Ctrl+V**.

An instance of the component is created, which is linked to the original component. All changes made to the copied component are also made to the original unless you make the copy independent.

To insert a component or assembly

1 Select the Insert File tool  in the Insert ribbon group.

2 Navigate to the component and double-click to insert it.

The component is placed in the center of the workspace and its sub-components (if it is an assembly) appear in the Structure panel.

To activate a component

Right-click the component and select **Activate Component** from the context menu.

If the component is lightweight, it is also loaded. Any new objects are created within this component. A component must be active before you can cut or copy it for pasting.

To externalize a component

1 Right-click the component and select **Open Component** from the context menu.

The selected component appears in a new Design window.

2 Select **Save As** from the Application menu to save the component as a separate file. (DO NOT check the Save as copy box.)

The icon in the Structure tree of the original design changes to reflect that the component is now external.

To copy an external component into your design

Right-click the component and select **Use Internal Copy** from the context menu.

The selected component is copied into your design. Any changes you make to the component do not affect the original, external file.

To load a lightweight component

Right-click the component and select **Load Component** from the context menu.

The component and all its subcomponents' geometry information is loaded, and you can work on the components with any SpaceClaim tool.

To make a component independent

Right-click the inserted, dependent component in the Structure tree and select **Make Independent** from the context menu.

The icon in the Structure tree changes, and the component is renamed <OriginalComponentName>2.

If your design consists of multiple instances of the same external component, and that external component also contains multiple instances of another external subcomponent, making the subcomponent independent makes both the subcomponent and its parent component independent. Any parent component in the tree, all the way up to the top-level design component, will also be made independent.

To create or specify a material for a component

1 Select a component in the Structure tree.

2 Select the Properties panel.

3 Enter the name of the material in the Material Name property.

4 Press **Enter**.

5 Enter the density of the material in the Density property.

If you specified the density for the material elsewhere in the design, or specified it in the same SpaceClaim session, SpaceClaim displays that value in the Density property for you.

Aligning components

Use the Align tool to align the planar faces of different components.

To align the planar faces of two components

1 Click the face of the component you want to move.

2 Ctrl+click the face of the component you want to remain in the same location.

3 Click the Align tool  in the Assembly ribbon group.

The two faces are aligned along the same plane, and an Align Planes mating condition appears under the moved component in the Structure tree. If you want to align to the other side of the plane, right-click the Align Planes mating condition in the Structure tree and select **Reverse Sense**.

Centering components

Use the Center tool to align the axes of two components.

To align the axes of two components

1 Click the axis of the component you want to move.

Mouse over an axial face to display that face's axis.

2 Ctrl+click the axis of the component you want to remain in the same location.

You can also select the faces defined by the axes.

3 Click the Center tool  in the Assembly ribbon group.

The two axes of the components are aligned and a Center Axes mating condition is created in the Structure tree under the moved component.

Orienting components

Use the Orient tool to rotate components around their alignment axis so that the selected faces point in the same direction.

To orient two components

1 Align the axes of the two components.

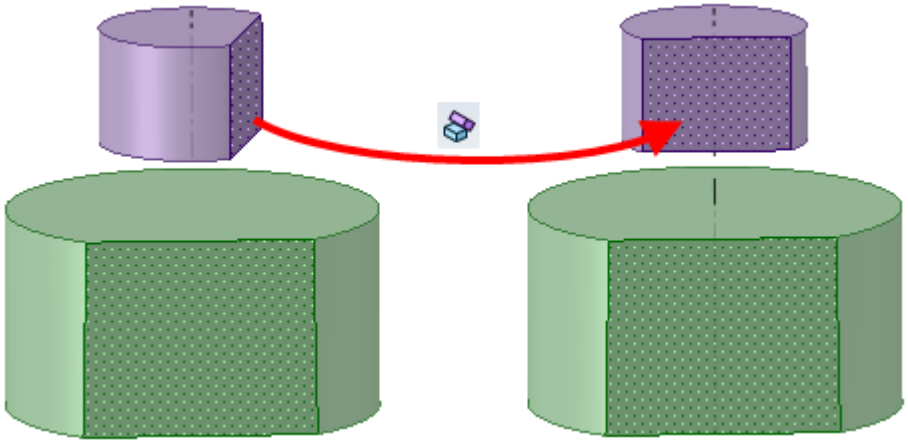
2 Click the face of the component you want to move.

3 Ctrl+click the face of the component you want to stay in the same location.

4 Click the Orient tool  in the Assembly ribbon group.

The second component rotates around the alignment axis until the two selected faces are oriented in the same direction, and an Orient Directions mating condition appears under the moved component.

Example

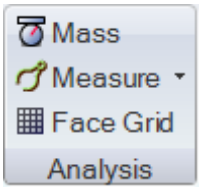


Components before and after orienting

Measuring and analyzing

Select a tool from the Analysis ribbon group to display measurements for the edges, faces, and solids in your design, or to display a u-v grid .

Analysis ribbon group



The Analysis ribbon group contains the following tools:

Use the Mass Properties tool to display volume information for the objects in your design.

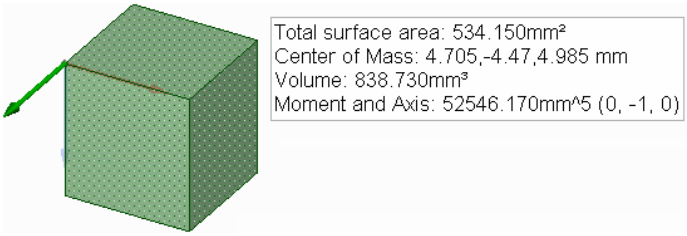
Use the Measure tool to display measurements of the edges and faces in your design. Select from this tool's menu to display edges and volumes of intersection.

Use the Face Grid tool to display a grid on any face or surface in your design.

Displaying mass

Use the Mass tool to display volume information for the solids and surfaces in your design.

If you use the tool on a surface, it will display the total surface area. If there are multiple surfaces on the same plane, the tool displays the total surface area for all the surfaces. To display the surfaces' individual surface areas, use the Measure tool.



To view mass properties of a solid

1. Select the Mass Properties tool from the Analysis ribbon group.
2. Click a solid in the Structure tree or by triple-clicking it in the Design window to display its volume, center of mass, and principle moments and axes.

   A small origin appears at the center of volume with its axes oriented in the direction of the principle axes. To calculate the principle moments and axes for another point, Ctrl+click an origin to add it to your selection.

   To calculate the actual moment of inertia, multiply the moment measurement by the density of the solid.

   The axis is indicated with x, y, z values; in the figure above, these values are (0, -1, 0). The first number is the red axis of the origin, the second is the green axis, and the third is the blue axis.

Displaying measurements

Use the Measure tool to display measurements of the edges and faces in your design. You can select different units for measurement by setting the SpaceClaim Units options.

To view a measurement

1. Select the Measure tool  from the Analysis ribbon group or press **E**.

   Mouse over your design to preview the faces and edges eligible for measurement.
2. Click an edge or face to display measurement information.

   Select two points, edges, or faces to display the distance and angle between them.

   You can select one or two objects to be measured with the Measure tool. If you try to add a third object to the selection, the object you selected first is deselected.

Displaying edges of intersection

Use the Quick Interference tool  to display the edges where solids intersect each other.

To display the edges of intersection

1. Select the Quick Interference tool  from the Measure tool menu in the Analysis Ribbon group.
2. Click intersecting solids to see their edges of intersection.

Displaying intersection volumes

Use the Volume Interference tool  to display volumes created by the intersection of solids in your design.

To display volumes of intersection

1. Select the Volume Interference tool  from the Measure tool menu in the Analysis Ribbon group.
2. Ctrl+click solids to display the volumes created by their intersection in red.

Displaying a face or surface grid

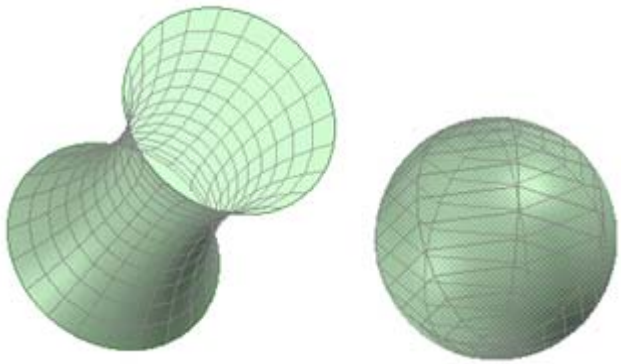
Use the Face Grid tool to display the curves that define any face or surface in your design. You may find it useful to display a face grid when working with the Edit as Blend tool.

To view a face or surface grid

1. Select the Face Grid tool  from the Analysis ribbon group.
2. Click a face or surface.

   Ctrl+click to display the grid on multiple faces and surfaces. Click a face again to hide the grid.

Examples



Face grids displayed on a blended face and on a sphere

Detailing

You can detail your designs to communicate with others or to submit your designs for review with the tools on SpaceClaim's Detailing tab. With the detailing tools, you can annotate your designs, create drawing sheets, and review changes to designs. You can customize detailing options to conform to standards or create your own custom style.

Detailing tools are grouped into the following ribbon groups:

Orient	Quickly display a particular view of your design.
Font	Format note text by adjusting the font characteristics.
Annotation	Create notes on your design with text, dimensions, geometric tolerances, tables, surface finish symbols, and datum symbols, center marks, center lines, and threads.
Views	Add views to a drawing sheet.
Sheet Setup	Format a drawing sheet.
3D Markup	Create markup slides to illustrate the changes made to a design.

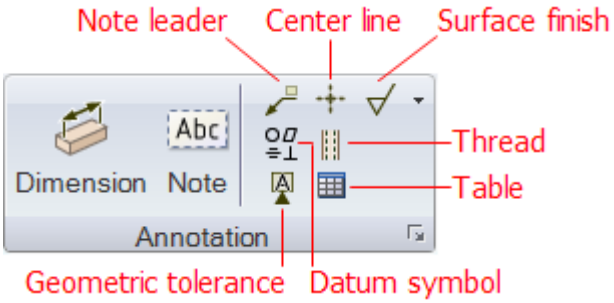
Press **Esc**, then **S** to end any detailing action and return to the Select tool.

Annotations

You can annotate your designs, drawings, and 3D markups with notes, dimensions, geometric tolerances, surface finish and datum symbols, as well as center marks, center lines, and threads. When you create annotations that are attached to the objects in your design, they stay attached, even when you modify those objects using the Design tools. Annotations created on a drawing sheet or 3D markup slide are part of that sheet or markup only; they do not appear on your design.

Each annotation has properties which you can modify in the Properties panel. When you create the first annotation, it is scaled so that it will be visible when your design is zoomed to its extents in the Design window. All other annotations use the same scale.

Annotation ribbon group



The Annotation ribbon group contains the following tools:

 Use the Dimension tool to create a measured dimension.

 Use the Create Note tool to select an annotation plane and enter text onto the plane.

 Use the Note Leader tool to connect a note to an object.

 Use the Center Line tool to add a center mark on any circle, arc, cylinder end, or sphere, and put center lines on any cylindrical face.

- Use the Surface Finish tool to create a surface finish symbol.
- Use the Geometric Tolerance tool to create a geometric tolerance.
- Use the Thread tool to create a threaded surface on any cylinder, cone, or hole.
- Use the Datum Symbol tool to insert a datum symbol.
- Use the Table tool to insert a table note.
- Click to display SpaceClaim's detailing options.

Press **Esc**, then **S** to end any detailing action and return to the Select tool.

Creating notes

Use the Create Note tool to annotate your designs, drawing sheets, and 3D markups. You can use this tool to create and edit notes. You can also project the note onto a sketch or onto a solid. Place the annotation plane on one layer and the note on another layer, then turn off layer visibility for the annotation plane to hide it.

To create a note

1. Select the Create Note tool Abc from the Annotation ribbon group in the Detailing tab.

   Mouse over the faces of your design to preview the eligible annotation planes. (In Sketch and Section mode, the sketch grid defines the annotation plane.)
2. Click a face to create the plane on which to place the note.

   To create an annotation plane for a cylindrical face, select the cylinder's axis.

   If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu. Then right-click the new place and click **Set As Annotation Plane**.
3. Click to place the note on the plane.
4. Enter the text of the note.

   Click Ω in the mini-toolbar to insert a symbol into your note at the cursor location.

   Click XXX in the mini-toolbar to insert a dynamic field. Dynamic fields include current values from a variety of properties.

   You can format the note text, and enter the text by cutting, copying, and pasting the text from other notes or dimension annotations.
5. Adjust the orientation of the note by dragging the rotation handles.

To create a note field

1. Click within the text of the note and place the cursor where you want the field to appear.
2. Right-click to display the mini-toolbar.
3. Click XXX in the mini-toolbar to display the Insert Field window.

   The Fields tab displays the properties available for insertion. (Document properties are those that appear in the Properties panel when you click the top-level design in the Structure tree.)
4. Select a value from the Category drop-down to filter the properties displayed in the Fields list.

   If you click **Selected Object**, you can click any object in the Design window or Structure tree to make its properties available.

   If you select **Formula**, you can enter an expression, and include any numeric fields within the expression.

The following expression elements are available:

- Infix (dyadic) operators: **+ - * / ^**
- Prefix (monadic) operators: **+ -**
- Functions: **sin cos tan asin acos atan sqrt log log10 exp**
- Constants: **pi e root2 root3**
- Units: **m cm mm yd ft in ' " deg rad**

Normal precedence rules apply:

1 + 2 * 3 ^ 4 = 1 + (2 * (3 ^ 4)) = 163

Parentheses are required for expression arguments and optional for simple arguments:

- sqrt 2 = sqrt(2) = 1.4142...
- sqrt 2*2 = (sqrt 2) * 2 = 2.8284...
- sqrt(2*2) = 2

Missing operators are inferred:

- 1 1/2 = 1 + 1/2
- 1'6" = 1' + 6"
- 1ft 6in - 17in = 1ft + 6in - 17in
- 1 2 3 4 5 = 1 + 2 + 3 + 4 + 5 = 15
- (1)(2)(3)(4)(5) = (1) * (2) * (3) * (4) * (5) = 120
- 2(1 + 2) = 2 * (1 + 2) = 6
- sqrt 2 sqrt 2 = sqrt 2 * sqrt 2 = 2
- 4(4atan(1/5) - atan(1/239)) = 4 * (4 * atan(1/5) - atan(1/239)) = pi

Units are applied to previous terms if units were not specified and are applied to subsequent terms unless you override them:

- 1 + 1cm = 1cm + 1cm
- 1cm + 1 = 1cm + 1cm
- 1cm + 1 + 1mm = 1cm + 1mm + 1mm
- 1cm + 1 1/2 mm = 1cm + 1mm + 1mm / 2

Trigonometry functions work in radians by default, but you can enter degrees:

sin(45 deg)

Numbers support standard form, but e is a built-in constant:

- 2e2 = 200
  2e 2 = 2 * e * 2 = 10.873...
- 2e-2 = 0.02
  2e – 2 = 2 * e – 2 = 3.436...
- 2e1 = 20
  2e = 2 * e

5 Click a property in the Fields list.

6 Click the **Format** tab to format the text within the field.

The formatting options are based on the type of the property value. For example, strings can be formatted with upper case, lower case, initial capitals, or title case.

7 Click **OK** to insert the formatted, dynamic field into the note at the cursor location.

If the field is empty, check to make sure that the property you selected has a value by selecting the appropriate object and viewing the Properties panel.

To copy a note

Ctrl+drag a note with the Move tool to copy it.

To edit a note

1. Select the note to move, size, or rotate it.

   To move the note box, mouse over the edge of the box until the cursor changes to  , then drag the note.

   To size the box containing the note, drag the handles of the note box (the white circles).

   To rotate the note, drag the rotation handle (the green circle). Press **Shift** to snap to angular increments.

2. Select the text of the note to reformat it.

   Click a field to edit it. Changing the value of a string also changes it in the Properties panel.

3. Modify the note's properties in the Properties panel. Modify the:
   - **Space** property to set the size of the note. Select **Model Space** to size the text based on the actual measurements of the objects in your design. Select **View Space** to size the text based on the view of the design in the Design window.
   - **Flagnote** property to create a border around the note. Select a shape from the drop-down. Enter a value in the **Minimum width** property to prevent the flag from resizing automatically to fit the content of the note.

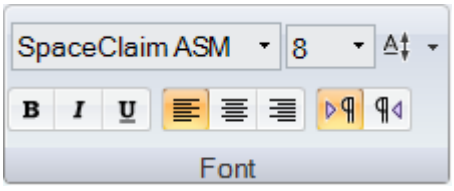
To display the text of an annotation and hide the plane

1. Create two layers, one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

Formatting note text

You can adjust the font, size, style (bold, italic, underline), alignment of the text within the box, and create superscripts and subscripts using the tools in the Font ribbon group, or by right-clicking the annotation and selecting from the mini-toolbar.

Font ribbon group



The Font ribbon group contains the following tools:

Family Select the font family.

Size Select or enter a font size.

Select an amount of offset to create a superscript or subscript.


Bold, italicize, or underline the note text.


Left, center, or right-justify the note text.

 Set the text direction from left-to-right or right-to-left.

To format text

1. Select the text in the note.
2. Use the tools in the Font ribbon group to format the selected text.

   To create a superscript or subscript, select a preset amount from the **Vertical Text Offset** drop-down, or select **Custom** and enter a custom amount to raise or lower the text

Creating note leaders

Use the Note Leader tool to create an arrow from your note.

To create a note leader

1. Select the Note Leader tool from the Annotation ribbon group in the Detailing tab.

   Mouse over notes on the active annotation plane to see the possible ways to connect the note leader to the note.
2. Click a note's connection point to draw the first segment of the note leader.

   Mouse over your design to highlight the geometry to which you can attach the end of the note leader. You can also attach leaders to temporary objects.
3. For a segmented line, click to set each point of the note leader line.
4. End the note leader.

   Click a vertex, edge, or face to attach the end of the note leader to it, or double-click to end the note leader at any point. The end of the note leader is an arrow unless you attach it to a face.

To insert an all-around symbol

Right-click the note leader and select All Around.

To change a note leader's segments

Drag a segment of the note leader line to move it and its surrounding boundary points. Right-click the note leader and select **Add Jog Point** to create a new segment.

You can delete note leader line segments by deleting the jog points that border the segment.

To change a note leader's properties

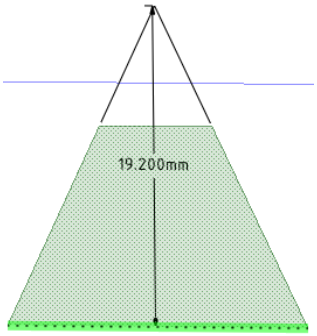
1. Right-click the note leader and select **Properties**.
2. Modify the Arrow **Style**, **Length**, and **Width** values.
3. Set the **All Around** value to **True** to display an all-around symbol. Select **False** to hide it.

To attach a note leader to a virtual sharp

1. Select the Note Leader tool from the Annotation ribbon group in the Detailing tab.
2. Ctrl+click a line.
3. Ctrl+click a converging line.

   The head of the note leader is attached to the virtual sharp. You can also 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.

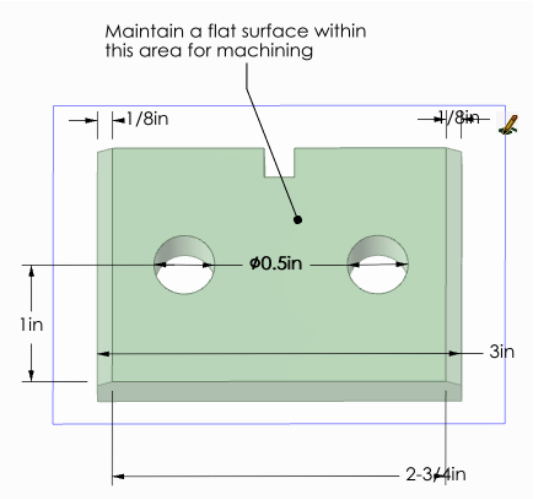
Example



A virtual sharp

Creating dimension annotations

Use the Dimension tool to add a measurement to your design, drawing sheet, or 3D markup.



To create a dimension annotation

1. Select the Dimension tool  from the Annotation ribbon group in the Detailing tab.

   Mouse over the faces of your design to preview the eligible annotation planes. (In Sketch and Section mode, the sketch grid defines the annotation plane.) If multiple objects occur at your cursor location, use the scroll wheel or arrow keys to highlight each one.
2. Click a face to create the plane on which to place the dimension.

   To create an annotation plane for a cylindrical face, select the cylinder's axis.

   If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu. Then right-click the new place and click **Set As Annotation Plane**.
3. Click an edge or face.

   Where you click on a circle determines whether you will measure from the circle's center, near, or far edge. To select the center click the top, bottom, left, or right side of the circle.
4. Mouse over your design to preview the possible dimensions.
5. Click to create the dimension.

To edit a dimension annotation

1. Select the dimension annotation to move, size, or rotate it.

   To move the dimension note, mouse over the edge of the box with the Select tool until the cursor changes to  , then drag the note.

   To size the box that contains the dimension note, drag the handles of the note box (the white circles).
2. Select the text of the note to reformat it.
3. (Optional) Right-click the dimension and select text formatting options from the mini-toolbar.

   Click XXX to select a tolerance format, then edit the text of the tolerance.

Click to insert a field. You can select a field type and format from the Insert Field window.

Select from the Ω drop-down to insert a symbol.

4. Click an arrowhead to cycle through alternative leader styles.
5. Click the note leaders to modify them.

   You can right-click a leader and select **Add Jog Point** to add a new point.
6. Modify the dimension note properties in the Properties panel. Modify the:
   - **Arrow Length** and **Width** properties to set the length and width of the arrowheads
   - **Measurement** property to change the measurement type. For example, you may want to display the radius of a hole instead of the diameter.
   - **Precision** property to change the number of decimal places.
   - **Upper Limit**, **Lower Limit**, and **Type** of tolerance property to change the format of the dimension and enter upper and lower tolerance values.

To attach a dimension annotation to a virtual sharp

1. Select the Dimension tool from the Annotation ribbon group in the Detailing tab.
2. Click a line.
3. Ctrl+click a converging line.

   The head of the note leader is attached to the virtual sharp. You can also 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.

To display the annotation and hide the plane

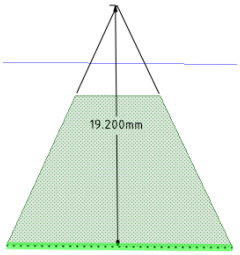
1. Create two layers, one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

To dimension to a virtual sharp

1. Select the Dimension tool from the Annotation ribbon group in the Detailing tab.
2. Ctrl+click a line.
3. Ctrl+click a converging line.

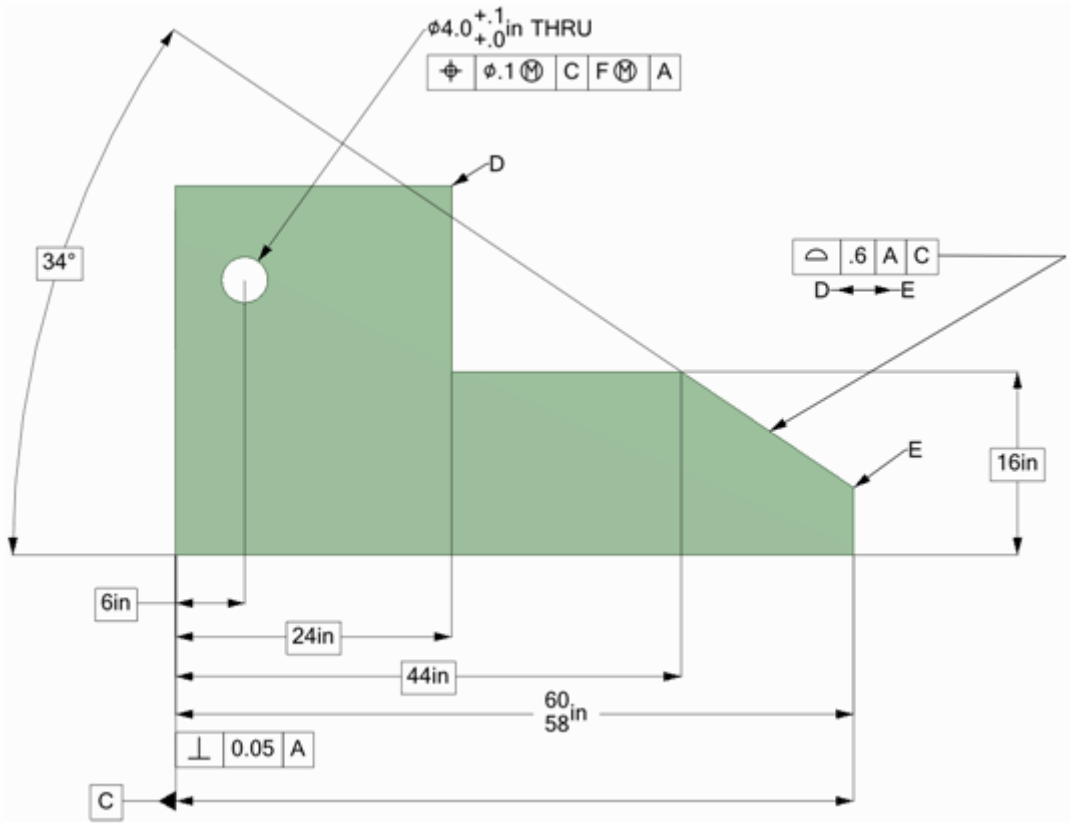
   The head of the note leader is attached to the virtual sharp. You can also drag the virtual sharp's end point, draw another leader to the virtual sharp, create virtual sharps in cross-section, for rounds, and between an angled and straight edge.

Example



A virtual sharp

Creating geometric tolerance annotations



Use the Geometric Tolerance tools in the Annotation ribbon group in the Detailing tab to add a tolerance to your design, drawing sheet, or 3D markup slide. In SpaceClaim, geometric tolerances are not created automatically. You can enter anything in a geometric tolerance, so we recommend that you read the geometric tolerance tooltips carefully to create intelligent geometric tolerance annotations.

To create a geometric tolerance annotation

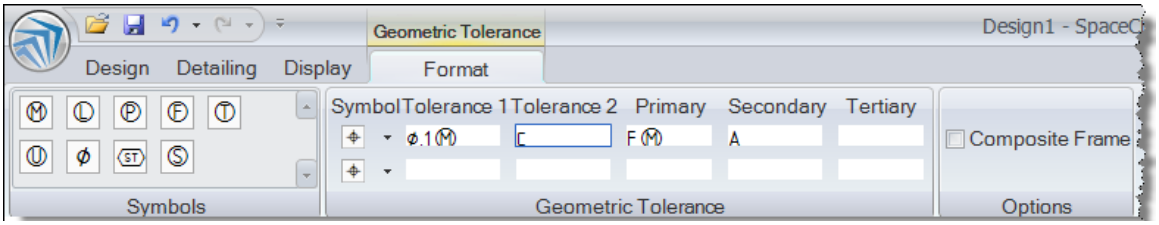
1. Click the Geometric Tolerance tool ⌖⊥.
2. Click to place the geometric tolerance annotation on the appropriate annotation plane and display the Format tab.
3. Select the characteristic symbol from the **Symbol** drop-down in the Geometric Tolerance ribbon group.
4. Click in one of the Tolerance fields and enter any combination of text and modifying symbols from the Symbols ribbon group.

   To create multi-row geometric tolerance annotations, select a characteristic symbol, then enter text and modifying symbols in the second row. To combine the two rows so that they have one characteristic symbol, check the **Composite Frame** option.
5. Press **Esc** to exit the annotation and close the Format tab.

To display the text of an annotation and hide the plane

1. Create two layers, one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

Geometric Tolerance Format tab



Datum symbols

You can insert datum symbols onto your design, drawing sheet, or 3D markup slide.

To add a datum symbol

1. Select the Datum Symbol tool.

   Mouse over the faces of your design to preview the eligible annotation planes.
2. Click to place the datum symbol on the appropriate annotation plane.
3. Enter a letter.
4. Use the Note Leader tool to create the note leader.

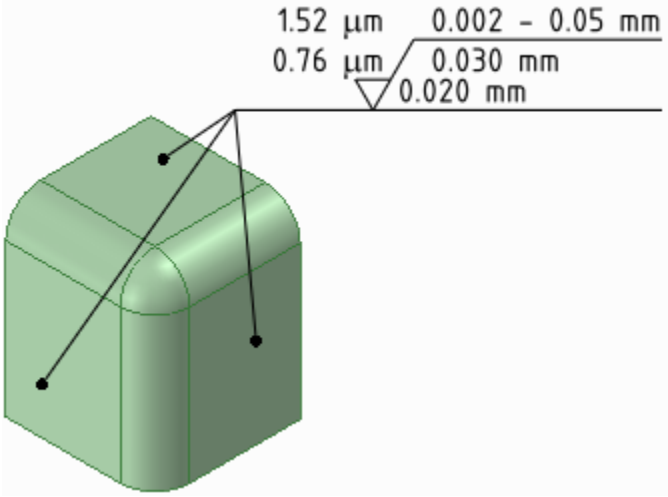
To replace a dimension with a datum

1. Click the Datum Symbol tool.
2. Click the dimension text.

To display the text of an annotation and hide the plane

1. Create two layers, one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

Surface finish symbols



You can insert surface finish symbols onto your design, drawing sheet, or 3D markup slide. Surface finish symbols move along with the surface they are attached to.

To add a surface finish symbol

1. Select the type of symbol you want to use from the Surface Finish tool drop-down.

   Mouse over the faces of your design to preview the eligible annotation planes. If an annotation plane already exists, right-click and select **Select New Annotation Plane** to choose a different plane.
2. Click a face to place a leader on the plane.

   You can click as many faces as you'd like to add leaders to those faces.
3. Click in an empty area of the Design window to place the surface finish symbol.

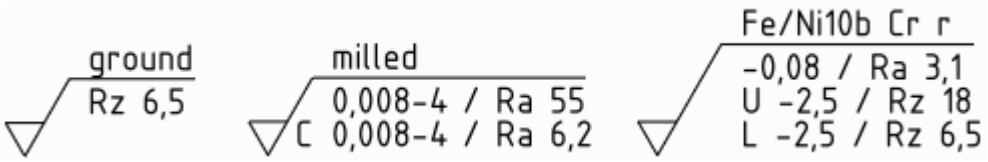
4. Modify the values in the Properties panel. Modify:
   - **Font Size** to change the font size for all the text fields on the symbol
   - **Template** to change the number of fields
   - **Show All Around Symbol** to add that symbol to the surface finish symbol
   - **Type** to change the type of surface finish symbol
5. Enter text or symbols in the field.

   You can press **Tab** to switch between the fields. Right-click and select Ω from the mini-toolbar to insert a symbol.

To display the text of an annotation and hide the plane

1. Create two layers, one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

Examples



Surface finish symbols using the Standard, Standard 2, and Standard 3 templates

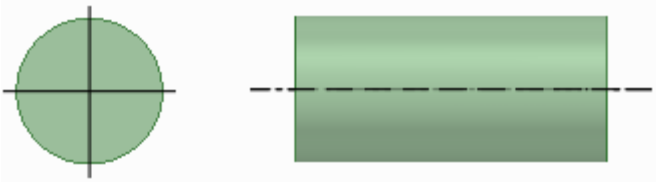
Center marks and center lines

You can put a center mark on any circle, arc, cylinder end, or sphere, and you can put center lines on any cylindrical face on a drawing sheet.

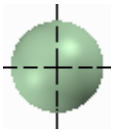
To add a center mark or center line

1. Click the Center Line tool .
2. Click an:
   - Edge of the end of the cylinder or hole to add a center mark.
   - Cylindrical face to add a center line.
   - Sphere to add a center mark.

Example



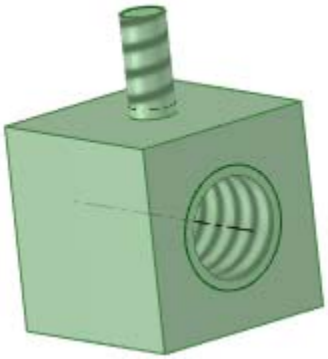
Center mark on the end of a cylinder and center line along the face of a cylinder



Center mark on a sphere

Threads

Use the Thread tool to create a threaded surface on any cylinder, cone, or hole.



To create a threaded surface

1. Click the Thread tool in the Annotation ribbon group on the Detailing tab.
2. Click the edge of a cylinder, hole, or cone.

   The threads are indicated as a texture on the surface, and the thread depth is shown when you view the threaded object in Section mode. An inner thread or outer thread object also appears in the Structure tree. (Broken threads appear with a small yellow triangle over the icon.)
3. Click the surface to edit the properties of the threaded surface in the Properties panel.

   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.

   Set the Thread Depth Type property by selecting **Blind** or **Full Thread**. Full Thread is the default for through holes and Blind is chosen automatically when the thread placement cylinder or cone ends in a plane at a convex edge.

To customize the thread properties

The thread data XML files that drive the drop-down menus for Standard thread properties are located in the SpaceClaim **Library/Threads** directory. You can:

- Add or files to the directory to display the name in the **Series** property drop-down when you click the threaded surface with the Thread tool. Remove files to remove them from the property.
- Add rows to an individual file to display them in the **Size** property drop-down. Remove rows to remove them from the property.
- Specify a different directory for thread files by adding a different directory to the support file options.

Tables

You can place a table on an annotation plane using the Table tool in the Annotation ribbon group.

To add a table

1. Add an Annotation plane or activate an existing plane.
2. Click the Table tool.
3. Drag to create the table.

   You can:

   - Select the table, then drag to select multiple cells. Once multiple cells are selected, you can right-click and select **Merge Cells** or **Unmerge Cells**.
   - Select one or more cells and right-click to access the other context menu functions, which allow you to add and remove columns and rows.

- Click within a table cell to edit its contents.
- Drag a column or row boundary to resize the column or row.
- Select rows, columns and cells by positioning the cursor to the left of the row, to the top of the column, and in the internal left side of the cell.
- Apply font and paragraph properties to selected text.
- Press **Tab** to navigate through table cells.
- Drag the left side of the table to resize it.

To move or rotate a table

1. Click the outer, dotted border of the table to select it.
2. Use the:
   - Arrow keys to move the table.
   - Ctrl+arrow keys to move the table by a very small amount.
   - Round handle at the top of the table to rotate the table.

To delete text within a table

1. Select all the cells in the table.
2. Press **Delete**.

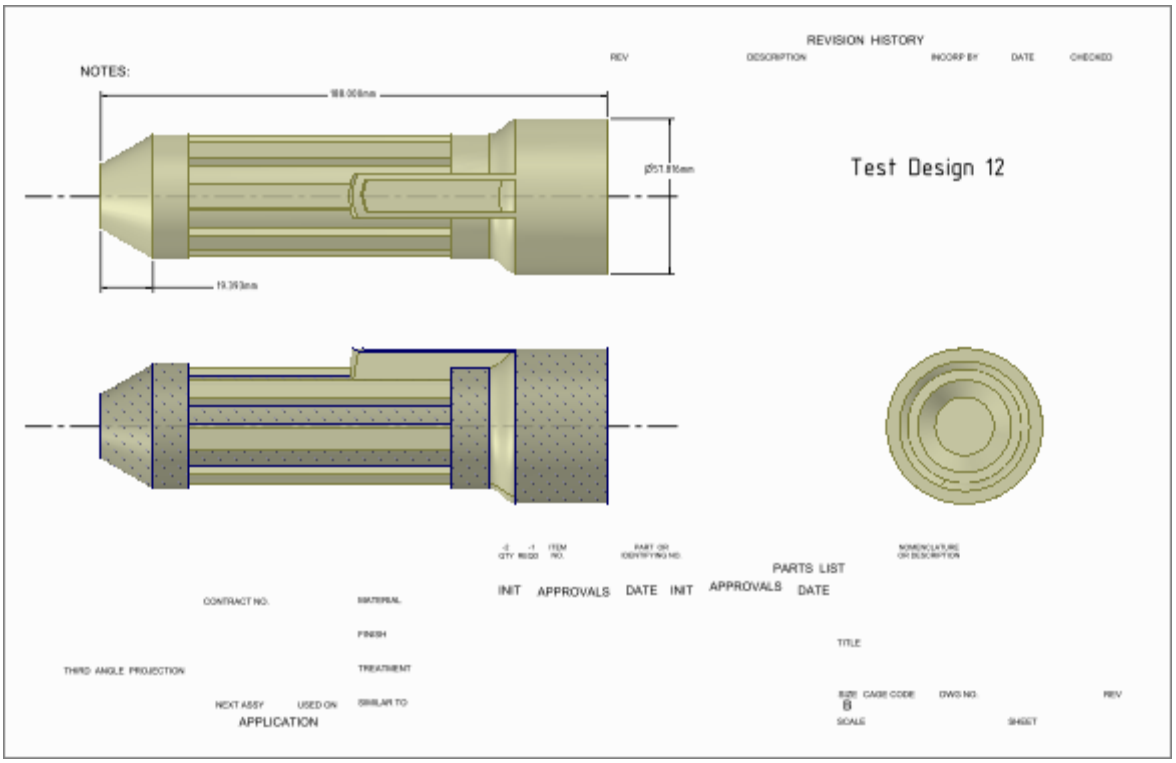
To delete a table

1. Select all the cells in the table.
2. Right-click and select **Delete Rows** or **Delete Columns** to delete the entire table.

To set table properties

1. Click the outer, dotted border of the table to select it.
2. Modify the number of columns and rows, row height, column width, and cell alignments and margins.

Drawing sheets



SpaceClaim helps you create drawing sheets. When you add a new drawing sheet to a design, views of the design are created for you automatically. You can then add, remove, and edit these views, as well as move them around on the sheet using the tools in the Detailing tab. Drawing sheets are saved within your design.

Normally, you will create the drawing sheet, set up the sheet, apply a format, add and modify views, then annotate the views.

Drawing sheets and their views appear in the Structure tree. You can place drawing sheet views on layers.

You can edit your design directly while viewing the drawing sheet using the tools on the Design tab.

To create a new drawing sheet

Select **New > New Drawing Sheet** from the application menu.

A drawing sheet containing top, front, and right-side views appears in the Design window, and the Drawing Sheet appears in the Structure tree. The Detailing tab is displayed. You do not need to create a design before creating a drawing sheet. SpaceClaim allows you to create and edit geometry within the drawing sheet itself. When you create a new drawing sheet for an empty design, the sheet contains the view outlines, with handles you can use to control the size and position of the view. You can delete the handles and move the view using the outline.

When sketching on a drawing sheet, you can:

- Click within the view boundary with a sketch tool to display a sketch grid.
- Use the scroll wheel to select a face of a solid in the drawing sheet when only the edge is displayed.
- Override colors for solids and surfaces. Changing the colors on the drawing sheet does not affect the colors in the design.

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.

Modifying one view changes the related views as appropriate.

To display a drawing sheet in the Design window

Right-click the sheet in the Structure tree and select **Open Sheet**.

To delete a drawing sheet

Right-click the sheet in the Structure tree and select **Delete**.

To view the drawing sheet head-on

Right-click anywhere in the drawing sheet and select **View > Flat View**.

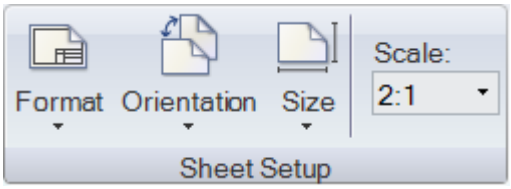
To edit design annotations

Right-click the annotation plane in the Structure tree and select **Show all dimensions** to display any design annotations on that plane. Click an annotation to edit it. Changes made on the drawing sheet will also appear in the design.

Setting up a drawing sheet

You can format the drawing sheet using a template, orient the page, and select a paper size for the sheet. When you select the format and size of the drawing sheet, SpaceClaim automatically sets the scale, but you can modify it.

Sheet Setup ribbon group



The Sheet Setup ribbon group contains the following tools:


Select a default or custom format, or remove the formatting from the drawing sheet with the **Format** tool.


Select a portrait or landscape orientation for the page from the **Orientation** drop-down list.


Select the page size from the **Size** drop-down list.

Scale Enter a scale in the **Scale** field. For example, enter 1:1 to display views of your design at life size.

Formatting a drawing sheet

You can apply SpaceClaim formats to your drawing sheet or create your own custom formats. You can show or hide the format lines on the sheet.

To apply a format to the drawing sheet

Select a default drawing sheet from the **Format** tool drop-down.

To apply a custom format to the drawing sheet

1. Select **More Formats** from the **Format** tool drop-down.
2. Navigate to the SCDOC file containing the format and click **Open**.

Remove the formatting from the drawing sheet

Select **Remove Current Format** from **Format** tool drop-down.

Create a custom drawing sheet format

1. Create a new drawing sheet.
2. Set the orientation, size, and scale of the drawing sheet.
3. Draw on the drawing sheet using the Sketching and Annotation tools.

   You can also insert an AutoCAD file to create the format. When inserting, select the **Import onto drawing sheet** option.

   Tip: If you insert fields based on document properties into annotations on the drawing sheet, you can create a drawing sheet format that automatically updates when it is applied to a drawing sheet in a design.
4. Save the drawing sheet into a support directory.

   The drawing sheet appears as a format within the Format tool's drop-down list.

Views

You can add and remove views from the drawing sheet, move them around the sheet, and modify their properties. You can create general views, projected views, cross-section views, and detail views. These views are all related to the view used to create them, and inherit properties from that view.

To add a view to the drawing sheet

1. Select one of the following tools from the Views ribbon group on the Detailing tab:
   - **General View** tool  to add a new, independent view.
   - **Projected View** tool  to create a view projected from one of the other views on the drawing sheet.
   - **Cross Section** tool  to create a cross-section view using one of the other views on the drawing sheet.
   - **Detail View** tool  to create an enlarged view of a particular area.
2. Click to place the view on the drawing sheet.
3. Press **Esc** or **S** to exit the tool.

To add a new, independent view

1. Click the **General View** tool  .
2. Click the drawing sheet to add the view at that location.

To modify a General view

1. Select the view.
2. Modify the values in the Properties panel, or right-click and select a style from the mini-toolbar. Change the:
   - **Orientation** to change the orientation of the view to isometric, trimetric, or any side. If you change the orientation of a general view from which projected views were created, the orientation of the projected views also change.
   - **Rendering mode** to change the graphics style for the view. Select **Inherit** if you want to link the graphics style to the drawing sheet.
   - **Scale** to magnify or shrink the view. When you change the scale, the Type property changes to **Independent from sheet**. You can select **Linked to sheet** to set the scale to be the same as the scale used for the drawing sheet.
3. You can also adjust the appearance of your design on the drawing sheet by using the:
   - Snap View tool to correctly position the design.
   - Move tool to position the design more accurately.

   When you modify the orientation of a general view, any views dependent on the general view are oriented simultaneously. Other general views do not change.

To select a view on the drawing sheet

1. Press **S** to activate the Select tool.

   If you are in another tool, press **Esc** to cancel the current action, then press **Esc** again to exit the tool and activate the Select tool.
2. Mouse just outside the view on the drawing sheet, or scroll over the design shown in the view until a dashed box around the view is displayed.
3. Click the box to select the view.

To move a view on the drawing sheet

1. Select the view.
2. Drag the view with the Select tool to move it or click the Move tool and move it with the Move handle.

   Projected and cross-section views move together as a group.

To orient the design shown in a view

1. Select the component shown in the view.
2. Select the Move tool on the Design tab.
3. Use the Move handle to orient the component within the view.

   Views related to the current view also change their orientation.

To modify a view

1. Select the view.

   You can select multiple views to modify the properties they have in common.
2. Modify the view's properties in the Properties panel or right-click and modify the view using the mini-toolbar.
3. Control the visibility of the view in the Structure tree.

Projected views

Projected views show another side of the model.

To create an orthogonal projected view

1. Click the **Projected View** tool  on the Detailing tab's Views ribbon group.
2. Click within the view you want to use to create the projected view. Do not click a highlighted edge.

   If dotted gray lines do not appear around the view, it is not selected. If this occurs, press **Esc** and try again.
3. Move your mouse to preview the projected view.

   Orthogonal views are created as you move toward the top, bottom, left, and right of the current view.
4. Click to place the view on the drawing sheet.
5. Press **Esc** or **S** to exit the tool.

To create an auxiliary projected view

1. Click the **Projected View** tool  on the Detailing tab's Views ribbon group.
2. Mouse over the view until the edge you want to use to create an auxiliary projected view is highlighted, then click.

   If dotted gray lines do not appear around the view, it is not selected. If this occurs, press **Esc** and try again.
3. Move your mouse to preview the projected auxiliary view.

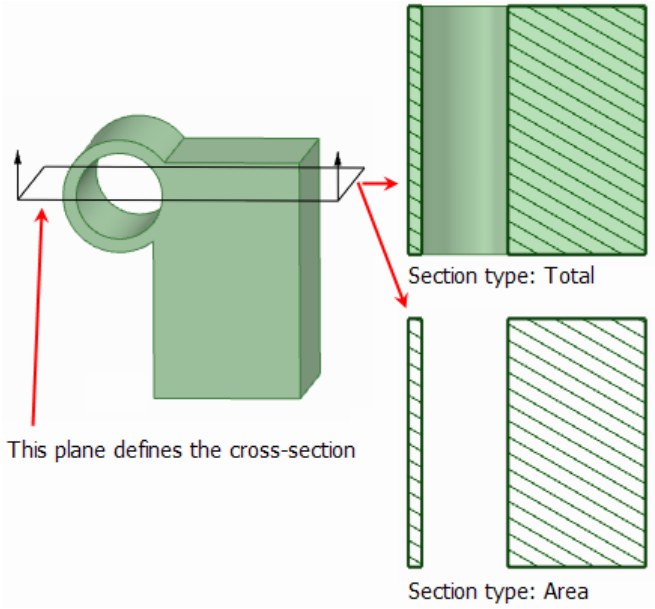
   Auxiliary views are created as you move the mouse perpendicular to the highlighted edge.
4. Click to place the view on the drawing sheet.
5. Press **Esc** or **S** to exit the tool.

To modify a projected view

1. Select the view.
2. Modify the values in the Properties panel, or right-click and select a style from the mini-toolbar. Change the:
   - **Orientation Type** to change the projected view to a General view. This makes the selected view independent of the view used to create it.
   - **Rendering mode** to change the graphics style for the view. Select **Inherit** if you want to link the graphics style to the parent view.
   - **Scale** to magnify or shrink the view. When you change the scale, the Type property changes to **Independent from sheet**. You can select **Linked to sheet** to set the scale to be the same as the scale used for the drawing sheet.

Cross-section views

Cross-section views show a cross-section through your design. To create a cross-section view, you must already have at least two views: one that will become the cross-section, and another that will be used to set the cross-section plane. You can add a cross-section to any view type.



To create a cross-section view

1. Click the **Cross Section View** tool on the Detailing tab's Views ribbon group.
2. Select the view you want to convert to a cross-section.

   If dotted gray lines do not appear around the view, it is not selected. If this occurs, press **Esc** and try again.
3. Mouse over a related view to display the cross-section indicator and preview the cross-section.

   The indicator line snaps to geometry in the view. Geometry in front of the cutting plane is not displayed.
4. Click to place the cross-section indicator and create the cross-section view.

   The view is labeled automatically.
5. Press **Esc** or **S** to exit the tool.

To modify a cross-section view

1. Move the cross-section indicator to change the cutting plane.
2. Select the cross-section view.
3. Modify the values in the Properties panel, or right-click and select a style from the mini-toolbar. Change the value in:
   - **Orientation Type** to change the cross-section view to a General view. This makes the selected view independent of the view used to create it.
   - **Rendering mode** to change the graphics style for the view. Select **Inherit** if you want to link the graphics style to the parent view.
   - **Scale** to magnify or shrink the view. When you change the scale, the Type property changes to **Independent from sheet**. You can select **Linked to sheet** to set the scale to be the same as the scale used for the drawing sheet.
   - **Section Type** to create a total or area cross-section. Select **Total** to display 3D geometry not on the cross section plane. Select **Area** to display only the geometry on the cross-section plane. You can select **None** to turn the view into a General view.
4. Select a region within the cross-section view.
5. Modify the values in the Properties panel. Change the value in the following Cross hatching properties:
   - **Exclude from sectioning** to remove the selected region from the cross-section view.
   - **Fill styles** to add or removing hatching from the region.
   - **Angle** to modify the angle at which the hatching lines are drawn.
   - **Spacing** to modify the space between hatching lines.
   - **Offset** to modify the start point of the first hatching line.

Detail views

Detail views allow you to magnify a portion of another view to show more detail. To create a detail view, you must already have at least one view on your drawing sheet.

If a detail view does not display correctly, make sure that you loaded any lightweight components shown in the view.

To create a new detail view

1. Click the **Detail View** tool  on the Detailing tab's Views ribbon group.
2. Select one of the following options:
   - **Create Detail of Selected View** to create a new detail view with the boundary drawn on an existing view.
   - **Convert Selected View to Partial** to convert an existing view.
3. Select the **Sketch Boundary Type** from the Options panel.
4. (Optional) Enter the **Scale** on the Options panel.
5. Click on the view that will create the detail to set the anchor point for scaling.

   Usually, you will create an anchor point very close to the detail that you want to show in the view.
6. Click to set the center of a boundary circle, a corner of a rectangle, or the first point of a closed spline.
7. Click again to define the boundary circle or rectangle, or click multiple times to define the boundary spline.

   The detail view is displayed.
8. Click to place the detail view on the drawing sheet.
9. Press **Esc** or **S** to exit the tool.

To modify a detail view

1. If you created a circular boundary, you can drag to expand or shrink it.
2. Select the detail view.
3. Modify the values in the Properties panel, or right-click and select a style from the mini-toolbar. Change the value in:
   - **Rendering mode** to change the graphics style for the view. Select **Inherit** if you want to link the graphics style to the parent view.
   - **Scale** to magnify or shrink the view. When you change the scale, the Type property changes to **Independent from sheet**. You can select **Linked to sheet** to set the scale to be the same as the scale used for the drawing sheet.
   - **Orientation type** to change the detail view to a General view.

3D markup

SpaceClaim allows you to create 3D markup slides so that you can highlight and communicate the differences between versions of a design.

Slides can be exported in PowerPoint and XPS formats.

Markup ribbon group

The Markup ribbon group contains the following tools:

Create a new 3D markup slide for the current design with the New Slide tool.

Display the dimensions of the previous version and the current version with the Original Dimension Values tool.

Apply colors to the design that indicate the type of change that occurred with the Color Changed Faces tool.

To create a 3D markup

1. Open the design you want to mark up in the Design window.
2. Select **Save As New Version** from the Application menu to create a new version of your design.

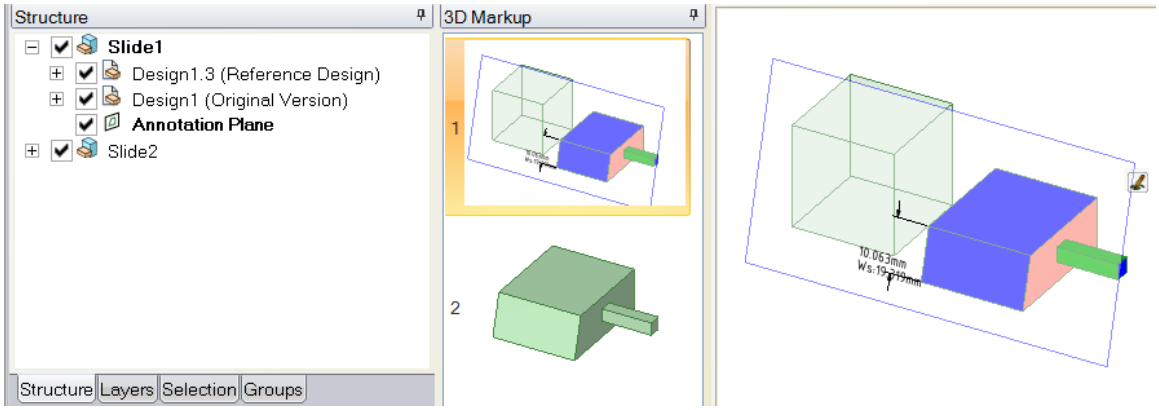
   The new version is saved with a version number appended to the file name, and becomes the active Design window.
3. Make changes to the new version of the design.
4. Select **New > 3D Markup** from the Application menu to create the first 3D markup slide.

   The slide is shown in the 3D Markup panel, the design window, and the Structure tree. In the Structure tree, the version you created in step 2 is labeled **Reference Design**.
5. Select the Insert tool from the Design tab and insert a previous version or the original version of the design into the slide.

   The design appears in the Structure tree, labeled either **Alternate Version** or **Original Version**. In the Design window, the previous version of the design is displayed in the wireframe transparent style and is placed directly on top of the new version. The Move tool is active.
6. (Optional) Move the alternate version to a new location by clicking on a Move handle axis and dragging.
7. Customize the slide using the tools in the Markup ribbon group in the Detailing tab.

   You can document the dimension changes with dimension annotations, color the faces based on the type of changes made, and apply any other annotations using the tools in the Annotation ribbon group in the Detailing tab. You can right-click a lightweight component and select **Load Component** to load it
8. Create as many slides as you need to effectively communicate your changes.
9. Select **Save As XPS** or **Save As PowerPoint** from the Application menu to export the 3D markup slides to a separate document.

Example



Structure tree shows contents of each slide. 3D Markup panel shows two slides in the slideshow. Design window shows reference and original versions compared with dimensions and coloring for changed faces.

Creating 3D markup slides

Use the New Slide tool in the Markup ribbon group on the Detailing tab to create a new 3D markup slide.

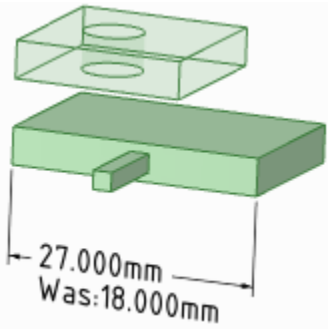
To create a 3D markup slide

1. Select the New Slide tool  or right-click in the 3D Markup panel and select **New Slide** from the context menu.

   A new 3D markup slide appears in the 3D Markup panel containing the reference design. (The reference design is the design that was active when you first created the 3D Markup document.

2. Insert a previous version of the design for comparison.

Displaying changes to dimensions



Use the Display Original Dimension Values tool to create dimension annotations for the current and original dimensions.

To display changes to dimensions

1. Select the Original Dimension Values tool  from the Markup ribbon panel on the Detailing tab.
2. Create dimension annotations with the Dimension tool.

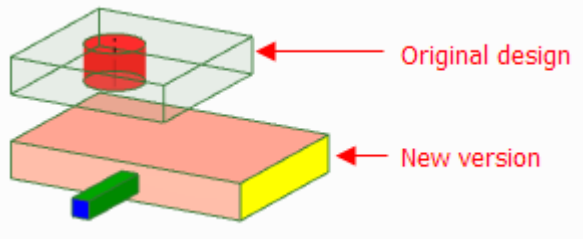
   The original dimension appears alongside the current one if that dimension changed between the two versions.

To edit a change dimension

Click within the dimension and edit the text.

To return the change dimension to its original values, select the note and click the Original Dimension Values tool  from the Markup ribbon panel on the Detailing tab.

Coloring changed faces



Use the Color Changed Faces tool in the Markup ribbon group on the Detailing tab to automatically color the faces that changed from one version to another based on the nature of the change.

We recommend creating a legend on your slide similar to the following to help communicate the meaning of each color:

New faces are colored green. Green faces did not exist in the previous version and will appear only on the modified version of the design.

New topology is colored blue. Blue faces have changes to both the face and its edges, but the face existed in the previous design.

Deleted faces are colored red. Red faces will appear only on the previous version of the design.

Faces with changed edges are colored pink. Pink faces are in the same spatial location, but are bounded differently because their edges have changed.

Faces with a changed spatial location are colored yellow. The edges of yellow faces have not changed from the previous version.

If you want to manually color the faces, you can apply colors to the individual faces on your 3D Markup slide.

To temporarily color changed faces

Click the Color Changed Faces tool . Click again to remove the colors.

The colors are shown on the current slide.

Detailing options

You can customize the style of your annotations in a single design or set a custom style as the default for all designs. You can quickly customize the style to conform to ASME or ISO/JIS standards, or you can create a style by customizing note leaders, dimensions, and geometric tolerances.

To customize SpaceClaim detailing options

1. Select **SpaceClaim Options** from the Application menu and click **Detailing** or click in the Annotation ribbon group on the Detailing tab.
2. Select one of the following from the **Detailing options for** drop-down list:
   - **All New Documents** to create a default detailing style for all your designs.
   - **This Document** to set options for the current design only.
3. Make your annotations, views, and linestyles conform to ASME, ISO, or JIS standards or Customize the default drawing sheet format.

   Select one of the following:

   - **Use external format** to select a pre-defined format provided by SpaceClaim or click **Browse** to select a custom format from any SpaceClaim file.
   - **No format** to use blank drawing sheets of a particular size and orientation.

If these options are disabled, select **All New Documents** in the **Detailing options for** drop-down.

Click **Reset to ASME Default** to customize annotations for ASME systems.

Click **Reset to ISO Default** to customize annotations for ISO systems.

Click **Reset to JIS Default** to customize annotations for JIS systems.

JIS defaults are the same as ISO, except that JIS uses third-angle views while ISO uses first-angle views. (A third-angle view is labeled by the object. Therefore, the front of the object is the "Front view" in JIS. A first-angle view is labeled by the direction you are looking. For example, if you are looking front, you see the back of an object. Therefore, the back of the object is the "Front view" in ISO.)

4 Customize drawing sheet views.

To customize General views

1 Modify the **Default view projection**. Select:

- **First Angle** to label the view by the direction you are looking. For example, if you are looking front, you see the back of an object. Therefore, the back of the object is the ”r;Front view.”
- **Third Angle** to label the view by the object. For example, the front of the object is the ”r;Front view.”

2 Modify the **Default front view position**. Select:

- **Top left** to place the front view at the top left corner of the drawing sheet. This is the ISO standard.
- **Top right** to place the front view at the top right corner of the drawing sheet
- **Bottom left** to place the front view at the bottom left corner of the drawing sheet. This is the ASME and JIS standard.
- **Bottom right** to place the front view at the bottom right corner of the drawing sheet. If you use third angle projection, ISO standards also allow this position.

To customize Cross Section views

Modify the following settings:

- **Section line arrow size** - Enter a value to set the size of the arrow shown on the end of the section indicator.
- **Section line length** - Enter a value to set the length of the section indicator.
- **Section cut line extension distance** - Enter the length of the arrows that extend from the section indicator.
- **Section line arrow direction** - Select whether you want the arrows to point toward or away from the section indicator.
- **Trim back section cut line interiors** - Check this box, then enter the length of the section line you want to appear connected to each arrow.
- **Default section name note prefix** - Select how you want the section label to appear on the drawing sheet.

To customize Detail views

Modify the following settings:

- **Detail view name text height ratio** - Set the ratio of a character's height to width. For example, a value of 1.4 sets the character height to 140% of its width.
- **Default view note layout** - Select **One Line** to display the detail name and scale on one line. Select **Two Lines** to display the scale below the detail name.
- **Default detail name note prefix** - Select how you want the detail label to appear on the drawing sheet.
- **Default view scale note prefix** - Select how you want the scale label to appear on the drawing sheet.

- **Detail view boundary note placement** - Select how you want the detail name and scale information to be positioned relative to the detail boundary.

To customize the display of threaded surfaces

Select a value from the **Cosmetic thread display standard** drop-down.

ASME Simplified is the same as ISO and JIS Conventional display standards.

5 Customize the annotation options**.**

Modify the following settings in the Annotation Options area:

- **Default text height -** Enter the height of the annotation text.
- **Leader circle size -** Enter the size of the circle that connects note leaders to faces.
- **Leader arrow length** - Enter the length of the arrow on note leaders.
- **Leader arrow width** - Enter the size of the arrowhead on note leaders.
- **Leader shoulder length** - Enter the length of the line from the note text to the note leader arrow.
- **Leader textbox gap** - Enterthe size of the margin between the note text and the beginning of the note leader.
- **Center line extend**- Enter the length that a center line will extend past the edge of the object.
- **Default arrow fill style** - Select the style you want to use for the arrowheads on note leaders from the drop-down list.
- **Default dimension text location** - Select how you want to align the note leader text with the note leader line from the drop-down list.
- **Default GTOL font name** - Select the font you want to use for geometric tolerance symbols from the drop-down list. The two fonts in this drop-down list contain all the geometric tolerance symbols necessary. These symbols will be used as necessary in annotations, even when you select a different font for the annotation text.
- **Virtual sharps rendering style** - Select the symbol you want to use to indicate the virtual sharp.
- **Enforce dimension line -** Check the box to use the European standard of keeping the dimension line when the extension lines are shown.
- **Tight gap between dimension line and text** - Check the box to shrink the gap between the dimension text and witness lines.
- **Horizontal dimension text** - Check the box to keep all annotation text oriented horizontally.
- **Override layer color for annotations** - Check the box to set all annotation text to the color selected shown in the Annotation color control. If you choose to override the layer color, select the color used for the override from the **Annotation color** control.
- **Extension line gap -** Enter the size of the margin between geometry and the end of dimension lines.
- **Extension line extend -** Enter the length that the dimension lines cross each other.
- **Dimension line extend -** Enter the length of the arrow that appears outside the dimension lines.
- **Dimension text offset** - Enter the distance between the dimension text and its leader line.

6 Customize the linestyle options**.**

Modify the following settings in the Linestyle Options area:

- **Default thick lineweight -** Enter the default width for thick lines.
- **Default thin lineweight -** Enter the default width for thin lines.
- Select an object type, then select the **Line style** and **Thickness** for that object.

7 Click **OK** to save all your changes and close the window.

Displaying designs

The tools you use to customize the appearance of your design within the design window are found on SpaceClaim's Display tab. You can customize your design by modifying which objects are displayed, the style in which solids and edges are displayed, and the color in which solids appear in your design. You can create layers to save different customizations and display characteristics. You can customize the workspace by creating windows or splitting the window to display multiple views of your design. You can also show or hide workspace tools. You can also configure the docking/detached location of all your workspace windows.

Display tools are grouped into the following ribbon groups:

Orient Quickly display a particular view of your design.

Style Determine how the solids in your design will be displayed.

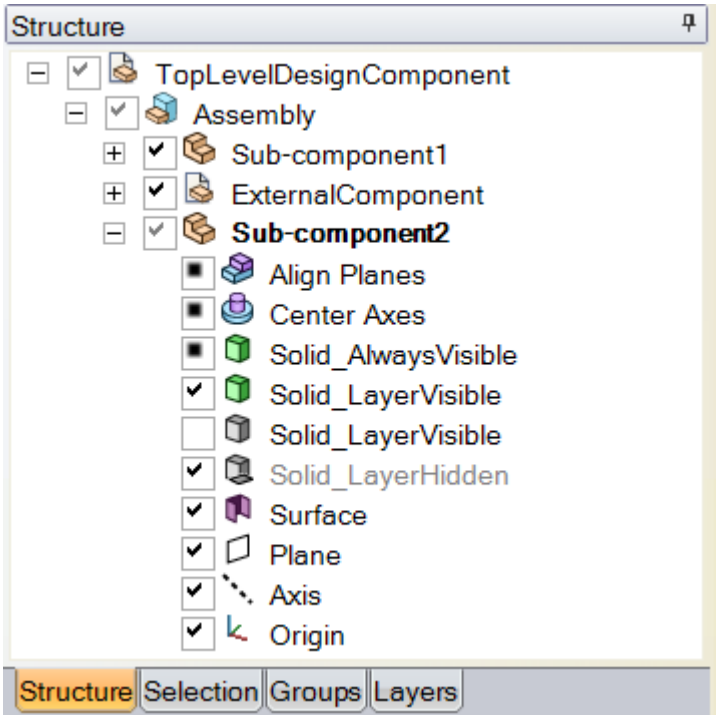
Window Create new design windows, split windows, and quickly switch between windows.

Grid Determine how the sketch grid and the geometry above or below the grid is displayed

Show Display or hide tools in the Design window.

Working with objects in the Structure tree

The Structure tree contains the Structure tree, which shows you each of the objects in your design. You can quickly show or hide any object using the checkbox next to the object's name. You can expand or collapse the nodes of the tree, rename objects, create, modify, replace, and delete objects, as well as work with components.



The top-level design (called **StructureTree** in the image on the right) is also a component. The figure on the right shows some objects that can appear in a Structure tree.

When you select a solid or surface (or other object) in the Design window, it is highlighted in the Structure tree.

You can Ctrl+click or Shift+click objects in the Structure tree to select multiple objects at once.

To set the visibility of objects

There are now three methods for setting the visibility of objects in the Design window:

- Right-click an object in the Structure tree and select **Always Visible** from the context menu.
- Uncheck the box in the Structure tree to hide the object in the Design window. The object icon is displayed in gray. You can also right-click an object in the Design window and select **Hide** (or select it in the Design window and press **Ctrl+H**) to turn the visibility of the object off.
- Check the box next to the object in the Structure tree to set the visibility of the object to the layer visibility.

If the layer visibility is on, the icon appears normally. If the layer visibility is off, the icon appears like the Solid_LayerHidden icon in the figure on the right. You cannot work with hidden objects in the Design window.

Do it faster	Shift+click and Ctrl+click multiple objects to work with them as a group.

To find an object in the Structure tree

Right-click any solid, surface, plane, axis, or other object in the Design window and select **Locate in Structure Tree** to display the object in the Structure tree. If the Structure panel is not open, it is displayed.

To expand or collapse components

Click ⊞ or press + on the number pad to expand a component. Click ⊟ or press - on the number pad to collapse it. Right-click any component (including the top-level component) and select **Expand All** or press * on the number pad to expand the component and all its subcomponents.

To rename objects

Right-click an object in the Structure tree and select **Rename** or press **F2** to rename the selected object.

Once you save a file, the top-level design component's name is set to the file name and it cannot be renamed.

To move objects into components

Drag any object or component to move it into another component.

To use an object as a secondary selection for a tool

Alt+click an object in the Structure tree.

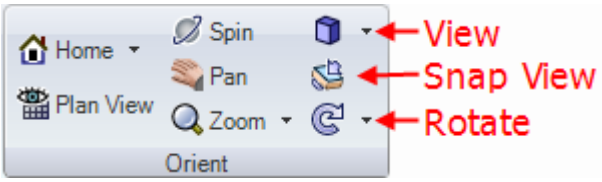
For example, if you want to revolve an object, you can click to select the face to Pull, then Alt+click an axis in the Structure tree to set the revolve axis for the pull.

Orienting designs

Select a tool from the Orient ribbon group to orient your design in the workspace. You can use these tools at any time, even when you are designing with other 2D or 3D tools.

Do it faster	Drag the middle mouse button to spin, Shift+drag it to pan, and Ctrl+drag it to zoom. You can switch between spinning, panning, and zooming by pressing and holding Ctrl (to zoom) or Shift (to pan). When you release the key, you will return to spinning.

Orient ribbon group



The Orient ribbon group contains the following tools:

Use the Home tool to return the orientation of your design to the default, trimetric view. You can customize the Home view to show your design with any orientation, location, and zoom level.

Click the Plan View tool to display a head-on view of the sketch grid or the select plane or planar face.

You can use the Spin tool to re-orient your design in any direction. Spinning your design allows you to view it from any angle.

Use the Pan tool to move your design within the Design window.

Use the Zoom tool to display your design closer or farther away in the Design window. You can zoom the design to fill the Design window, zoom into an area, or zoom in or out a preset amount.

Use the View tool to display a trimetric or isometric view of your design. You can also display a head-on view of the top, bottom, front, back, right, or left side.

Use the Snap View tool to display a head-on view of a face. You can also use the tool to "throw" the highlighted face to the top, bottom, right, or left.

Use the Rotate tool to rotate your design 90 degrees in the plane of the screen. You can rotate your design clockwise or counterclockwise.

Orient modes

When you click the Spin, Pan, and Zoom tools, they stay enabled until you click them again, press **Esc**, or click another tool.

Undoing and redoing views

You can undo and redo views using the **Previous View** and **Next View** tools  on the status bar.

Spinning your design

You can use the Spin tool to re-orient your design in any direction. Spinning your design allows you to view it from any angle. SpaceClaim uses standard arc-ball rotation; your design spins as if you could grab it with the cursor and adjust it like a real object.

When you click the Spin tool, it stays enabled until you click it again, press **Esc**, or click another tool.

To spin your design

1. Select the Spin tool from the Orient ribbon group or status bar.
2. Select one of the following from the Spin drop-down:
   - **On Center** to spin around the center of your design.
   - **On Cursor** to spin around the cursor location.
3. Click and drag to spin your design.

   If you start dragging on a highlighted line, edge, or axis, you can rotate your design around it. You can remove this feature by setting the advanced SpaceClaim option **Rotate about preselected object in spin**. (Press **Alt** and drag to rotate around a highlighted object whether or not this option is selected.)

   You may find it easier to reach a desired orientation if you use short mouse drags to spin the design a little bit at a time.

   If you double-click a face while using the Spin tool, the face is zoomed so that it fills the design window.

Do it faster	Set the **Rotate about preselected object in spin** option. Then, when working in any tool, position the middle mouse button on the face, edge, plane, or axis about which you want to spin your design, and drag to spin.

Panning your design

Use the Pan tool to move your design within the Design window.

When you click the Pan tool, it stays enabled until you click it again, press **Esc**, or click another tool.

To pan

1. Select the Pan tool  from the Orient ribbon group or status bar.
2. Drag to move your design around the Design window.

   If you double-click a face while using the Pan tool, the face is zoomed so that it fills the design window.

Do it faster	When working in any tool, Shift+click the middle mouse button and drag to move your design.

Zooming into and out of your design

Use the Zoom tool to display your design closer or farther away in the Design window. You can zoom the design to fill the Design window, zoom into an area, or zoom in or out a preset amount.

When you click the Zoom tool, it stays enabled until you click it again, press **Esc**, or click another tool.

To zoom in and out

1. Select the Zoom tool  from the Orient ribbon group or status bar.
2. Click where you want to center the zoom.
3. Drag down to zoom into your design; drag up to zoom out.

   You can also use the scroll wheel to zoom in and out.

Do it faster	When working in any tool, Ctrl+click the middle mouse button, then drag up and down to zoom. You can also press **Ctrl+** or **Ctrl-** to zoom in or out a preset amount.

To zoom the design or a selected face or edge to fit the Design window

Select **Zoom > Extents** or press **Z**.

The design or selected face or edge is zoomed so that it fills the Design window. When working with a drawing sheet, it will fit the drawing sheet to the Design window. If you resize the Design window, the design will also be resized until it again fills the Design window.

To zoom into a selected area

1. Select **Zoom > Zoom Box In**.

   You can also right-click in the Design window and select **View > Zoom Box In** from the menu.
2. Click and drag to select the area.

   Two rectangles appear as you draw. The dotted rectangle shows your selection; the solid rectangle shows what will be displayed in the Design window. When you mouse-up, the design pans and zooms until it fits within the area.

To zoom in and out a preset amount

Select **Zoom > Zoom In** to bring your design closer. Select **Zoom > Zoom Out** to move your design further away.

When working in any tool, press **Ctrl** and + or **Ctrl** and - to zoom in and out a preset amount.

Rotating your design

Use the Rotate tool to rotate your design 90 degrees in the plane of the screen. You can rotate your design clockwise or counterclockwise.

To rotate your design 90 degrees

Click the Rotate tool .

To change the rotation direction

Select **Rotate > Rotate 90 Counterclockwise**.

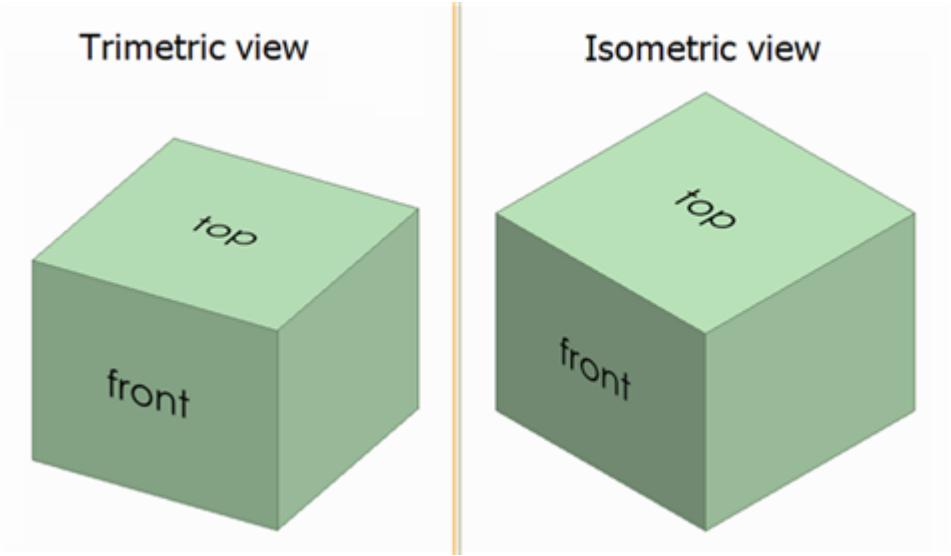
Your design rotates 90 degrees counterclockwise and the Rotate tool icon changes to . Clicking the Rotate tool will continue to rotate in the counterclockwise direction.

Select **Rotate > Rotate 90 Clockwise** to rotate your design in the clockwise direction and set the Rotate tool to rotate clockwise.

Your Home view

Use the Home tool to return the orientation of your design to the default, trimetric view. You can customize the Home view tool so that it displays your design with a specific orientation, location, and zoom level.

Compared to the isometric view, the trimetric view orients your design so that the front face is angled slightly towards you and less of the top is shown. Compare the two views in the image below.



To display the Home view

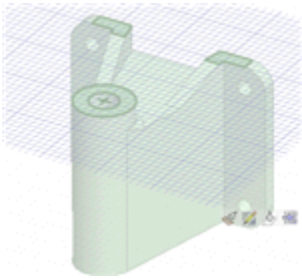
Click the Home tool in the Orient ribbon panel or press **H**.

To customize the Home view

1. Use the other Orient tools to set up a view of your design in the workspace.
2. Select **Home > Set As Home View** to make the view in the active Design window the Home view.

   Now, when you click the Home tool, your custom view is displayed. Your home view is saved with your design.

Click **Home > Reset Home View** to return the Home view to the default, trimetric view.



Display a head-on view of the sketch grid

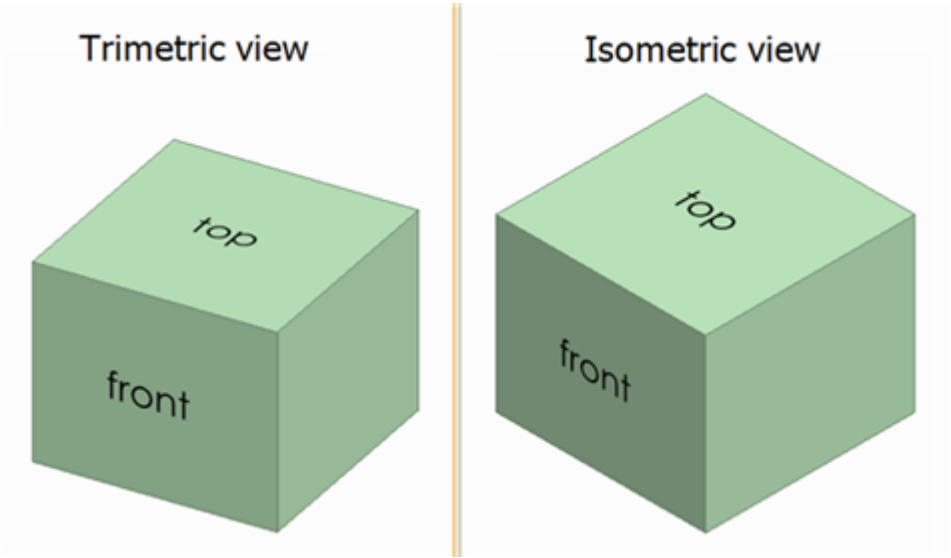
Click the Plan View tool in the Orient ribbon group or in the Sketching mini-toolbar to display a head-on view of the sketch grid or the selected plane or planar face.

If this tool is disabled, select a plane or planar surface or display the sketch grid.

Selecting a view

Use the View tool to display a trimetric or isometric view of your design. You can also display a head-on view of the top, bottom, front, back, right, or left side. Your design's orientation in the head-on views is determined by SpaceClaim's default coordinate system.

Compared to the isometric view, the trimetric view orients your design so that the front face is angled slightly towards you and less of the top is shown. Compare the two views in the image below.



To select a view

Select the view you want from the View tool menu in the Orient ribbon group.

To display a trimetric or isometric view of your design

Select **Trimetric** or **Isometric** from the View tool menu.

To display a head-on view of your design

Select **Top**, **Bottom**, **Front**, **Back**, **Right**, or **Left** from the View tool menu.

Snapping to a view

Use the Snap View tool to display a head-on view of a selected face. You can also use the tool to "throw" the highlighted face to the top, bottom, right, or left. On a drawing sheet, flipping one view also flips all related views.

You can use this tool only in 3D mode. It is disabled in Sketch and Section mode.

To snap the view

1. Click the Select tool  from the Edit ribbon group or press **S**.
2. Select the Snap View tool  from the Orient ribbon group.
3. Click a face to view it head-on.
4. Click, drag, and release the mouse towards the top, bottom, right, or left side of the design window to "throw" the face to that side.

   This gesture will work even with very small mouse movements.
5. Repeat steps 3 and 4 until you see the view you want.

Graphics style

SpaceClaim offers several different ways to style your design. You can apply styles to your entire design or drawing sheet, to individual layers, or to individual views in your drawing sheet.

To apply a graphics style to your design

Select an option from the Graphics Style tool  from the Style ribbon group on the Display tab to display your design with that option.

Select the option that works best for you for any given task. You can display geometry with a shaded or perspective shaded style, as wireframes, as wireframes with hidden lines displayed in light gray, and as wireframes with hidden lines removed. We recommend that you use the shaded style to most clearly indicate the difference between solids and surfaces.

To set a solid or surface to be transparent, opaque, or metallic

1. Select the solid or surface.
2. Select **Transparent**, **Opaque**, or **Metallic** from the Face Style  drop-down to apply that surface style to the selected solid or surface.

   If you apply the Opaque style to a surface, it will be returned to the transparent style if it becomes part of a new surface, such as after a merge.

Examples

Transparent bracket with opaque pole

Metallic style

Displaying edges

Use the Edges tool to customize which edges are displayed.

Select any combination of the following edge options from the Edges tool menu on the Display tab's Style ribbon group. The edge selections apply to your entire design.

Select:	To toggle the display of:
Tangent	Lines indicating tangent edges and edges that do not span a face
Surface	All edges on surfaces
Solid	All non-tangent edges on a solid
Silhouette	Lines indicating the silhouetted edges of all curved surfaces. This option affects only Wireframe, Hidden Line, and Hidden Line Removed graphics styles.
Layout Lines	Layout lines on the layout planes shown in the Structure tree

Applying colors to your design

Use the Color tool to apply color to the solids, surfaces, and components in your design. You can also apply color to individual faces on a 3D markup slide. Sketches are drawn in the color of the active component.

Normally the colors of solids, surfaces, and components are determined by the layer they are on. However, you can override the layer colors.

To apply the layer color

1. Select the object you want to color.

   Assemblies imported from other modeling software can have only one color applied to the entire assembly. You can Ctrl+click multiple components in the Structure tree to set their color simultaneously.

2. Select **Color by Layer** from the Color tool on the Display tab's Style ribbon group.
3. Select the layer from the Layer tool to place the solid on that layer and color it with the layer color displayed on the Color by Layer icon and next to the layer on the Layer panel.

To override the layer color for a solid

1. Select the solid you want to color.

   Assemblies imported from other modeling software can have only one color applied to the entire assembly.

2. Select a color from the Color tool on the Display tab's Style ribbon group.

If the color you want does not appear, select **Custom Color** and create the color.

To override the layer and solid color for a face in a 3D markup slide

1. Select the face you want to color.
2. Select a color from the Color tool on the Display tab's Style ribbon group.

   If the color you want does not appear, select **Custom Color** and create the color.

Line styles

You can apply a custom line style and line weight to the lines in your designs and drawing sheets. The line styles you choose apply to annotations, center marks, center lines, and drawing sheet cross-section view arrows, hatching, hatched area borders, and detail view boundaries. You can apply line styles to individual objects or to all the objects on a layer. The width that you set is exactly the width that will be printed when you print an unscaled drawing sheet.

You can set the default line styles for various objects in the SpaceClaim options. Setting the line styles individually overrides the default setting.

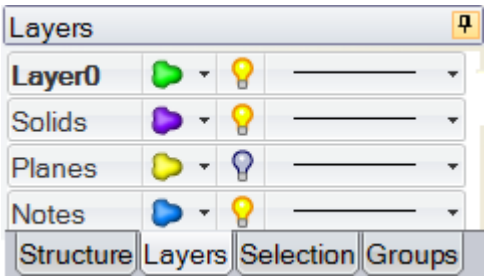
Linestyles can be assigned to layers so that you can have different line styles for sketch and layout lines.

To apply a line style

1. Select the objects or layer to which you want to apply the line style.
2. Select the line style from the Line Style tool menu.
3. Select the line weight from the Line Weight tool menu.

Working with layers

A layer can be thought of as a grouping mechanism for visual characteristics. Visual characteristics include visibility and color. Layers can be managed in the Layers panel and accessed and modified in the Layer tool in the Style ribbon group on the Display tab.



Layers are especially useful when you want to hide annotation planes. Any objects created are automatically placed on the active layer.

To create a layer

Right-click in the Layers panel and select **New**.

This layer becomes the activate layer. Any objects created are automatically placed on this layer.

To rename a layer

Right-click the layer in the Layers panel and select **Rename** or click the layer name and slowly drag to the right.

Layer0 cannot be renamed.

To delete a layer

Right-click the layer in the Layers panel and select **Delete**.

Layer0 cannot be deleted.

To place an object on a layer

1. Select the solid, surface, or component.

   The Layer tool in the Style ribbon group on the Display tab displays the layer of the selected object. If no object is selected, it displays the layer on which new objects are placed. It is blank if selected objects are on different layers.

2. Select a different layer from the drop-down list to place the selected object(s) on that layer.

   You can also create a new layer to place the selected object onto that layer.

To set layer visibility

1. Select a layer in the Layers panel.
2. Click to show the objects on the layer. Click to hide them.

   If an object is located on a layer with the visibility turned off, and the object in the Structure tree is set to show visibility by layer, the object is not visible in the Design window, and cannot be acted on by the design tools. Layer visibility can be overridden in the Structure tree.

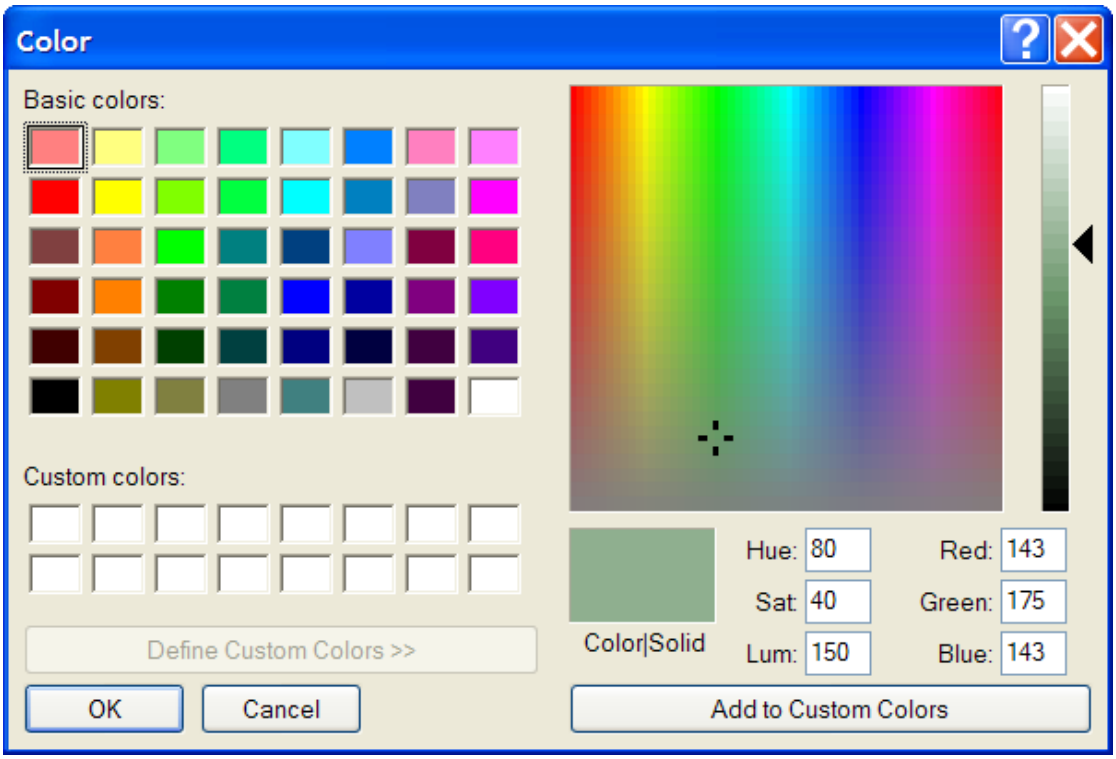
To set the visibility of layout lines and imported, DWG and DXF lines

Select **Solid** or **Hidden** from the layer's line drop-down in the Layers panel.

To modify the layer color

1. Select a layer in the Layers panel.
2. Select a color from the drop-down.

   You can also specify a custom color by selecting **Custom Color** and using the Color window.



Workspace windows

Displaying your design in multiple windows

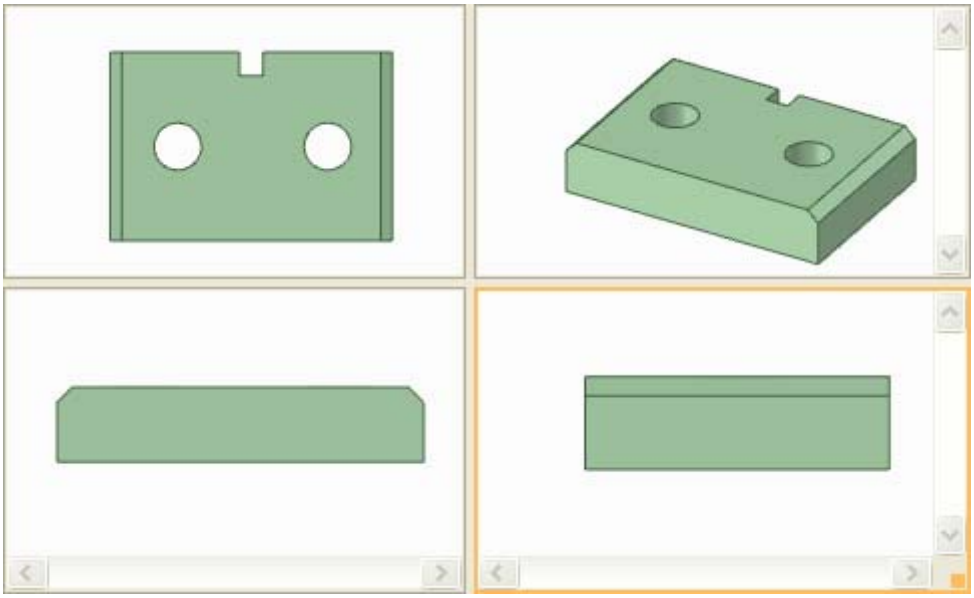
Use the New Window tool to create new workspace windows. Creating multiple windows allows you to set up several views of your design. Each window has a numbered tab at the bottom of the workspace.

To create a new workspace window

Select the New Window tool on the Display tab's Window ribbon group.

A new tab appears at the bottom of your workspace and a number is appended to the design, drawing sheet, or 3D markup name. You can click the middle mouse button on the tab of any design, drawing sheet, or 3D markup window to close that window

Splitting the workspace window



Use the Split Window tool to divide the workspace window into multiple windows. Splitting the window allows you to see multiple views of your design simultaneously.

Select how you want to split the workspace window from the Split Window tool menu on the Display tab's Window ribbon group. The window is split based on your selection.

Switching between windows in the workspace

If you have more than one window open, you can select the window you want to display from the tabs at the bottom of the workspace. If you have many windows open and the tabs do not fit in the workspace, you can use the Next Window and Previous Window arrows at the bottom of the workspace to switch between windows.

You can also use the Switch Window tool to select the window you want to display. Select the window you want to display from the Switch Window tool  menu on the Display tab's Window ribbon group.

Maximizing the Design window

Press F11 to maximize the design window to full screen. You can still switch between windows and use the tool guides when the window is maximized.

Press F11 to return to the standard user interface.

Displaying workspace tools

You can adjust the display of workspace tools and the display of your design in the Design window using the tools in the Show ribbon group on the Display tab, and by modifying SpaceClaim options.

To customize the tools displayed while you are working with your design

SpaceClaim offers the following tools on the Show ribbon group on the Display tab to assist you while creating, editing, and detailing your designs:

- Check the **World Origin** box to display the axes that set the default orientation of the design in the Design window.
- Check the **Spin Center** box to mark the center of the spin when using the Spin tool. (This is the same as the **Show Spin Center** SpaceClaim option.)
- Check the **Offset Baseline Faces** box to display offset relationships with blue shading.
- Check the **Coaxial Face Groups** box to display faces that share an axis with blue shading.
- Check the **Lineweight** box to switch the line style of lines (such as those displayed in Hidden Line, Hidden Line Removed, and Wireframe graphics styles) from thin to the thickness set by the Lineweight tool in the Style ribbon group.
- Check the **Adjacent Entities** box to display faint highlighting that appears on edges and faces adjacent when you hover over a point or edge. This feature is useful when you want to extrude the edge of a particular surface that meets another surface.

To display other workspace tools, modify the settings in the Popular SpaceClaim options.

You can also display journal-related tools by checking the **Show Journal Tab** option in the Advanced SpaceClaim options.

Sketch grid styles

You can modify whether the sketch grid is displayed, and how the geometry above or below the grid is displayed in each Design window. You may want to use one style when you are examining a component, and another when you are creating new geometry within a component. You can further customize the sketch grid by hiding section lines and faces using SpaceClaim options.

To show or hide the sketch grid

Check the **Show Sketch Grid** box in the Grid ribbon group on the Display tab to display the sketch grid. The sketch grid appears in all the sketching tools. Displaying the sketch grid allows you to snap to grid lines and provides a visual cue to the orientation of your sketch within your design

To determine how geometry above or below the grid is displayed

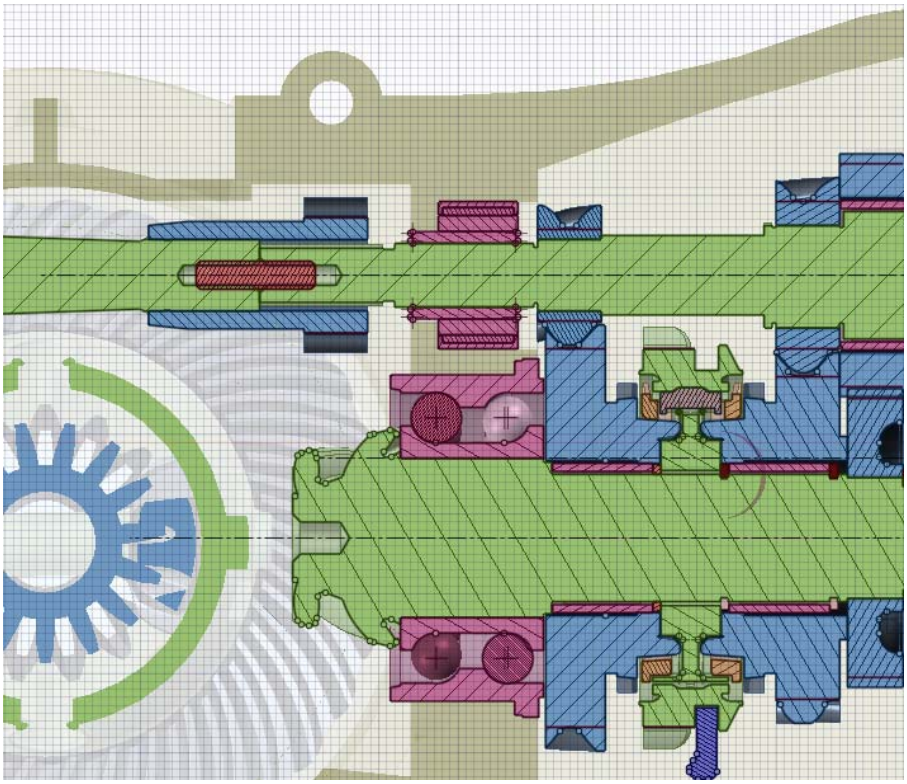
1. Check the **Fade Scene Under Grid** box to make the geometry under the sketch grid more transparent.
2. Check the **Clip Scene Above Grid** box to hide the geometry above the sketch grid.

To clip your design with a plane

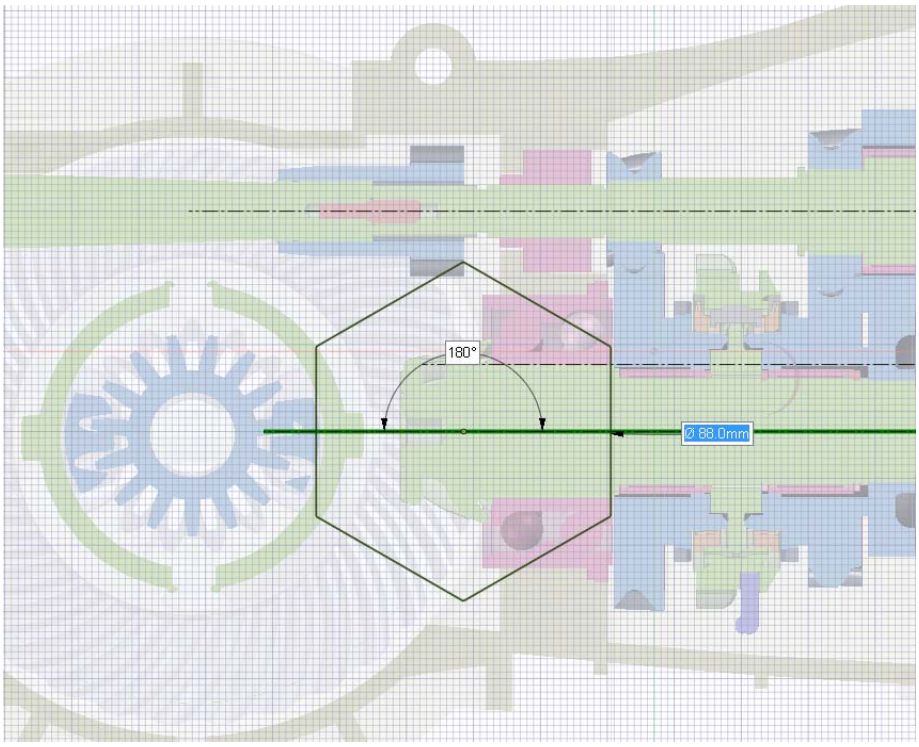
Right-click a plane and select **Clip with Plane**. To restore the view of your design, right-click the plane and select **Clip with Plane** again.

Planes, axes, and annotations are not clipped.

Examples



Examining a section with the scene clipped above the grid



Sketching on a cross-section with the scene faded below the grid

Displaying lightweight components

Lightweight components are a graphics-only representation of a design. You can adjust the transparency of the lightweight components in your design. If you do not see lightweight components when you open or insert a design, make sure your settings are configured to display them.

To adjust the transparency of lightweight components

1. Click the Transparency icon on the status bar.
2. Adjust the slider to set the transparency of the lightweight components in the Design window. All lightweight components are displayed with this transparency.

Working with SpaceClaim documents

Creating, opening, and saving designs, drawing sheets, and 3D markups

When you create a new design, it appears on a tab in the workspace. Designs can contain drawing sheets, multiple windows, and 3D markups. Each drawing sheet and 3D markup appears on its own tab in the workspace. You can edit your design directly using the drawing sheet.

You can create a drawing sheet for an existing design, or you can begin with an empty drawing sheet

Click a tab at the bottom of the SpaceClaim application window to display that design, drawing sheet, or 3D markup document, or click the arrow icons to cycle between them. Click the x icon to close the tab.

To create a new design

Select **New > Design** from the Application menu.

To create a new drawing sheet for the active design

Select **New > Drawing Sheet** from the Application menu to create a drawing sheet with three standard views of the design.

Select **New > Empty Drawing Sheet** to create a drawing sheet without a format or views.

If these menu options are disabled, create a new design.

To create a new design and associated drawing sheet

Select **New > Design and Drawing Sheet** from the Application menu to create a new design and an associated drawing sheet.

To open an existing design, drawing sheet, or 3D markup document

Select **Open** from the Application menu and select the file you want to open. The design and its windows, drawing sheets, and 3D markups open.

You can Ctrl+click or Shift+click to open multiple files at once. Mouse over a recent file to see an image of the design and the full path to the file.

To save a design

Select **Save** from the Application menu.

If you imported or opened non-SpaceClaim designs as multiple external documents, click **References** to specify where the documents are saved. Otherwise, opened design documents are stored in their original locations as .scdoc files, and inserted documents are saved in the same directory as your design.

To copy a design

1. Save any changes made to external components.
2. Select **Save as** from the Application menu and enter a new name for the design.
3. Check the **Save as copy** checkbox if you want to save a copy and continue working in your original design. Leave this box unchecked to close the original design and display the new design in the Design window
4. Click **Save**.

To save a version of your design

Select **Save as > New version** from the Application menu.

Saving a version allows you to create 3D markups.

To save a design as another file type

Please see Importing and exporting.

To close a design

Make sure its window is active in the workspace, then do one of the following:

- Select **Close** from the Application menu.
- Right-click the Design window tab and select **Close**.
- Click the **x** on the Design window tab bar (at the bottom of the application window).
- Click the **x** at the upper right of the Design window if you have undocked it.

To modify design properties

Document properties are displayed when you select the top-level design in the Structure tree. Right-click in the Properties panel and select **Add Property** to create a custom property. Expand the property to display its value. Enter a name for the property, select its type (date, Boolean, number, or string), and enter its value.

Importing and exporting components, designs, drawing sheets, and 3D markups

Use the Open command to open files created with SpaceClaim (.scdoc) or any other modeling application. Use the Save As command to export parts, assemblies, drawing sheets, and 3D markups to formats read by other applications. Your license type determines which of these actions are supported.

If you work frequently with non-SpaceClaim files, we recommend that you set your file options to optimize the importing and exporting process for your needs.

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.

Supported file type	Supported actions
ProE (prt, asm)	Open and insert parts and assemblies
Solidworks (sldprt, sldasm)	Open and insert parts and assemblies
Inventor (ipt)	Open and insert parts and assemblies (R12)
CATIA version 4 and 5 (model, CATPart, CATProduct)	Open and insert parts and assemblies, including product manufacturing information (PMI) for parts placed on the Imported Annotation Planes layer (with visibility turned off) Export v5 parts and assemblies
NX (prt)	Open and insert parts and assemblies
JT Open (jt)	Open and insert parts and assemblies Export parts and assemblies
Rhino (3dm)	Open and insert parts and assemblies Export parts and assemblies
ECAD Board (emn),	Open and insert files. Named components are created at the heights and contours

Library (emp)	defined by the ECAD file. Areas defined as "Keep in" or "Keep out" appear as open sketch lines.
Parasolid (x_t, x_b)	Open and insert parts and assemblies Export parts and assemblies
ACIS (sat, sab)	Open and insert parts and assemblies Export parts and assemblies (ACIS files flatten the assembly)
AutoCAD (dwg, dxf)	Open and insert drawings, parts, and assemblies. Drawings can be inserted as layouts. Export parts, assemblies, drawing sheets, and 3D markup slides
VDA (vda)	Open and insert parts and assemblies Export parts
STEP (stp, step)	Open and insert parts and assemblies Export parts and assemblies
IGES (igs, iges)	Open and insert parts and assemblies Export parts and assemblies
Point curve text (txt)	Insert curves
XAML (xaml)	Export part and assembly solids only
STL (stl)	Export parts and assemblies
VRML (wrl)	Export parts and assemblies
OBJ (obj)	Export parts and assemblies. Structure is not maintained.
Image (jpg, png, bmp)	Export parts, assemblies, drawing sheets, and 3D markup slides
PowerPoint (xps)	Export 3D markup slides

To import a design

1. Select **Open** from the Application menu or click  in the Quick Access toolbar, or click the **Insert** tool in the Insert ribbon group on the Design tab.
2. Depending on the selected file type, additional elements appear in the Open window. For descriptions of these options or to set their default values, click **Options**.

   The first line of a Point curve text file must be `Version=R2SP0`. The second line should be `Polyline=False` to use splines to connect points, and `Polyline=True` to use line segments to connect points. 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 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. 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.

   To import 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.

   When importing a Rhino file, multi-segmented curves are consolidated.
3. Navigate to and select the file you want to open or insert.

   If you are opening a file, it is displayed in a new Design window. If you are inserting a file, it appears as an external component within the active design.

   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.

To export a design or 3D markup

1. Select **Save as** from the Application menu.

   You can also press **F12** or **Ctrl+Shift+S**.
2. Select a file type from the **Save as type** drop-down.
3. Depending on the selected file type, additional elements appear in the Save As window.

Click:

- **Save as copy** if you want to save copies of external components referenced by the design with new names or replace external components with other external components. You must click **Resources** to do this.
- **References** to display all the external components referenced by the file. Select an external component and click **Browse** (or double-click the component) to rename or replace the component.
- **Options** to also set your default export options for the selected file type.

When saving as an .STL file, the quality is based your graphics quality setting. We recommend setting the option to enable the highest possible graphics quality if you want your design to be useful as an SLA rapid prototype for form, fit, and function purposes.

For CATIA, Parasolid, STL, and STEP files, you can select which version or protocol to save as. You can also set your default export options by clicking **Options**.

When you save an SAT file to an X_T file, edges with zero length are removed from the design.

Imported designs with identical filenames are given unique filenames when you save your SpaceClaim design. For example, if you imported `name.prt` and `name.asm`, these files are saved as `name.scdoc` and `name2.scdoc`.

When you save a design with a shaded graphics style as a DWG file, it is converted to the hidden line style.

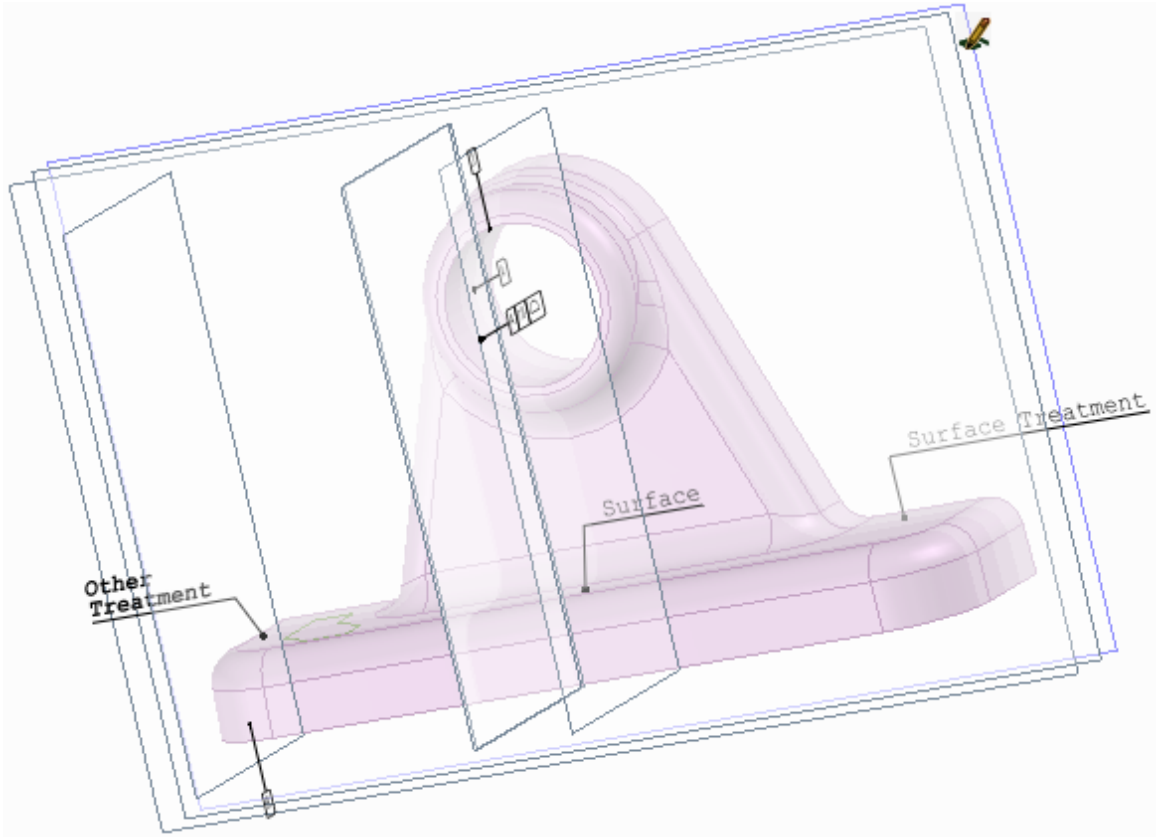
When you save a sheet metal design as a DXF file, notes and bend lines are saved on the same layer, and the overall unfold dimensions are removed.

When you save your design as an 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

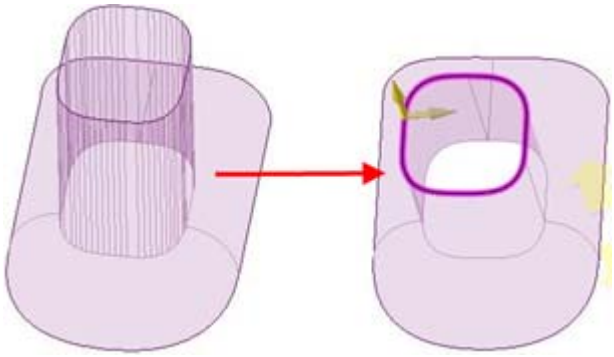
To export a design as an image

Right-click in the Design window and select **Copy Scene** to copy the contents of the Design window to a PNG file.

Example



PMI information from a CATIA file



Consolidated multi-segmented curves from a Rhino file

Printing drawing sheets and designs

Your zoom settings determine how a drawing sheet or design will print. For best results, select a paper size from the Print window that is the same as the drawing sheet format. If you changed the orientation of the drawing sheet, then the view that appears in the Design window is the one that will be scaled to fit the selected paper size. This can result in a clipped drawing.

Shaded designs are converted to hidden-line removed graphics style when you select Print from the application menu. Select **Shaded** from the Graphics Style drop-down after printing to return your design to the shaded style.

To print a drawing sheet

1. Select **Zoom > Zoom Extents** from the Orient ribbon group in the Design tab.

   You can also print the drawing sheet at different zoom levels by zooming in or out, but for the best results, we recommend you use Zoom Extents.
2. Select **SpaceClaim Options** from the Application menu and click **Detailing**.
3. Select **All New Documents** from the Detailing options for drop-down list.
4. Click **No format** and select the paper size and orientation.
5. Click **OK**.
6. Select **Print > Print Preview** from the Application menu to preview the print.

   You can zoom, set the page orientation, set print properties, and print the sheet from this window.
7. Select **Print** from the Application menu.

To print a design

1. Select **Zoom > Zoom Extents** from the Orient ribbon group in the Design tab.

   You can also print the design at different zoom levels by zooming in or out, but for the best results we recommend you use Zoom Extents.
2. Select **Print** from the Application menu.

   The design will be zoomed so that it fits the page size.

To adjust the design's size on the page

1. Select **Print** from the Application menu and click **Print Preview**.
2. Select one of the following from the Content ribbon group:
   - **Scene** to display the design based on the size shown in the Design window. This setting is used by default for designs.
   - **Extents** to expand the design until it fills the printable area of the page. This setting is used by default for drawing sheets.
3. If you selected **Extents**, select or enter one of the following from the Scale ribbon group:
   - **Scale to fit** to scale the design to fit the page. This setting is used by default.
   - Enter a value in the scale drop-down to scale your design by that amount.

To correct a clipped print preview

1. Close the print preview.
2. Select the Home tool from the Orient ribbon group.
3. Select **Print** from the Application menu and click **Preferences** to display your printer's Printing Preferences window.
4. Set your printer settings in the **Layout** area to match the format and orientation of the drawing sheet.
5. Set the paper size in the **Advanced** area to match the format paper size.

   If your printer only handles 8.5 x 11" paper, select the option that scales the image to fit the paper.
6. Click **OK** on the Advanced Options and Printing Preferences windows, and click **Apply** on the Print window.

To set page margins

1. Select **Print** from the Application menu and click **Print Preview**.
2. Click the Display tab and check **Margins** in the Show group.

3 Click the Print Preview tab and enter the margins in the Margins ribbon group.

Journals and logs

Journals record the actions you performed while creating your design, including file actions, such as closing a design. You must play a journal from a newly opened SpaceClaim application to avoid errors.

SpaceClaim also logs information automatically into the SpaceClaim.log file, which can be found in one of the following directories:

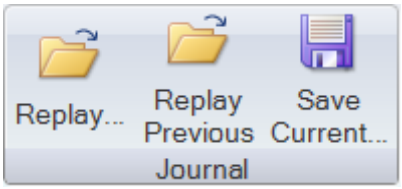
- `C:\Users\<User>\AppData\Roaming\SpaceClaim` on Vista systems
- `C:\Documents and Settings\<User>\Application Data\SpaceClaim` on XP systems

To display the journal tools

1 Select **SpaceClaim Options** from the Application menu and click **Advanced**.

2 Check the **Show Journal tab in the Ribbon** box and click **OK.**

The Journal Tools tab is displayed. It contains the Journal ribbon group.



To save all your actions in the current SpaceClaim session to a journal

1 Click **Save Current**.

2 Navigate to the directory where you want to save the journal, enter a name and click **Save**.

3 Save the design files used in the session.

To play the journal from your last session

1 Exit and restart SpaceClaim.

2 Click **Replay Previous**.

To play a saved journal

1 Exit and restart SpaceClaim.

2 Click **Replay**.

3 Navigate to and select the journal file you want to play and click **Open**.

4 Review the actions from the session.

SpaceClaim file format

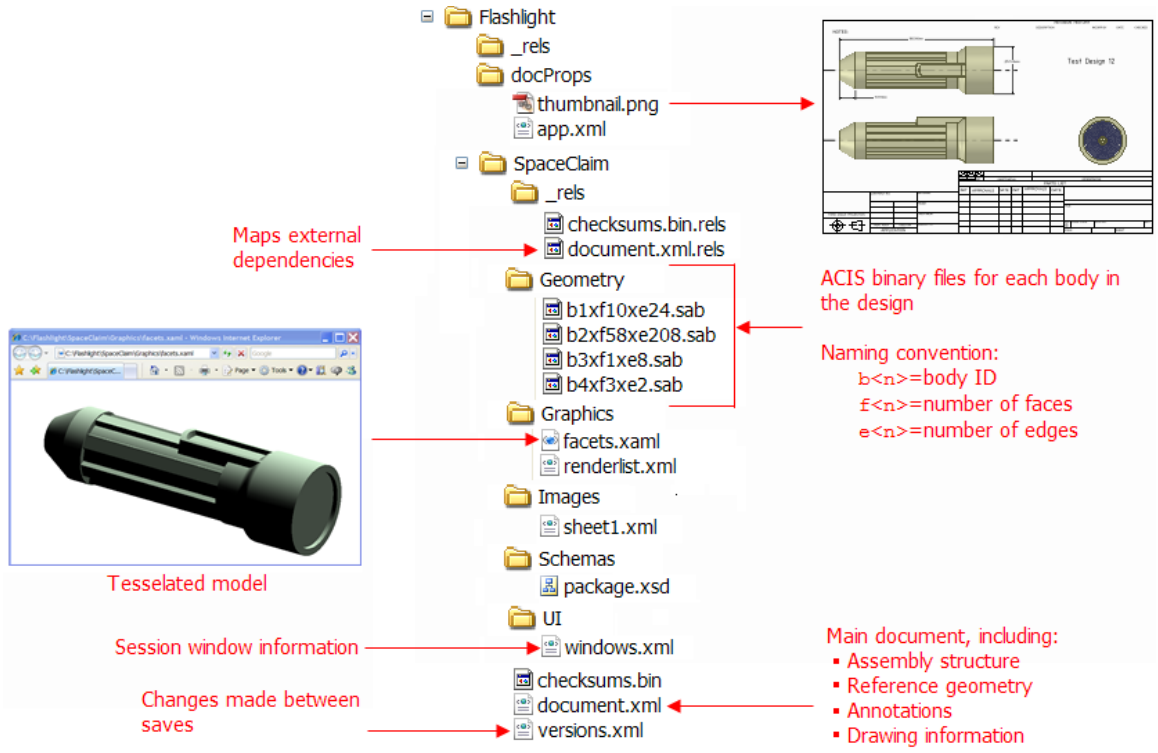
Our files comply with the Microsoft Open Packaging Convention, which is also used by Microsoft Office 2007. These files are actually zip archives with a special structure, the contents of which are primarily XML data.

This open format makes it possible for third parties to get the information they need from SpaceClaim without talking directly to the SpaceClaim API. For example, a PDM system could check in a SpaceClaim document with all external dependencies and show a 3D preview of the model. One could also create a batch translator that runs on a UNIX operating system.

Third parties planning on working with SpaceClaim data should contact SpaceClaim for advice on how to best integrate their solutions. In general, the benefit of working with SpaceClaim files is that there is no need for a license of SpaceClaim to be present to read SpaceClaim data. When creating SpaceClaim data or extending the SpaceClaim user interface, the API is a better choice. SpaceClaim provides sample code for extracting solids from SCDOC files.

Files contained within the SpaceClaim file

This figure shows the files contained within a sample `Flashlight.scdoc` file.



Files contained within the SpaceClaim file

The `document.xml.rels` file contains pointers to all the files required to load the design into SpaceClaim. The figure below shows this content for the Flashlight example.

```
<Relationships>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/versionHistory"
    Target="/SpaceClaim/versions.xml" Id="R158d51ac862c42dd"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/external/assemblyComponent"
    Target="C:\Documents and Settings\bcr\My Documents\Demos\XML scdoc\Standard Parts.scdoc#9532f9be-
    fdb3-401b-becb-02bfc5c15f8e,dc750e52-9196-4953-8a8e-6a60ecb32009,0,0" TargetMode="External"
    Id="R4a177fbfb52b43e0"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/external/drawingFormat"
    Target="C:\SpaceClaim2007\Library\DrawingFormats\B Landscape.scdoc#4c3d979a-a39a-4dd2-8d70-
    236f506016f4,cb727d5b-7b6f-4443-bd30-0a6c3d841650,5,3" TargetMode="External"
    Id="Rf1adf73efdf04791"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyGeometry#8c0e0f2a-fab9-43e7-
    bb6e-2b7bcd384cbb:109760" Target="/SpaceClaim/Geometry/b1xf10xe24.sab" Id="R270ce6cf44fc4122"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyGeometry#8c0e0f2a-fab9-43e7-
    bb6e-2b7bcd384cbb:67319" Target="/SpaceClaim/Geometry/b2xf58xe208.sab" Id="R1f12406a7bfa46b8"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyGeometry#8c0e0f2a-fab9-43e7-
    bb6e-2b7bcd384cbb:110505" Target="/SpaceClaim/Geometry/b3xf1xe8.sab" Id="R88923aed29904b69"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyGeometry#8c0e0f2a-fab9-43e7-
    bb6e-2b7bcd384cbb:7147" Target="/SpaceClaim/Geometry/b4xf3xe2.sab" Id="R2386a8b32a4a4c20"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/sheetThumbnail"
    Target="/SpaceClaim/Images/sheet1.xml" Id="Re64d81eec92c4b90"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyFacets"
    Target="/SpaceClaim/Graphics/facets.xaml" Id="R4ba09a6c64d14bfe"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/windows"
    Target="/SpaceClaim/UI/windows.xml" Id="R621f2c333b234fb2"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/renderlists"
    Target="/SpaceClaim/Graphics/renderlist.xml" Id="R06612e137a08469d"/>
</Relationships>
```

For example, you can use the information in this file to determine all the parts required for a BOM by looking at the `assemblyComponent` relationship. In this case, it points to another file called `Standard Parts.scdoc`. By opening this file and reviewing its components and subcomponents, you can generate a BOM report.

There are three types of external file pointers:

- `assemblyComponent` points to a file that contains subcomponents used in the assembly
- `drawingFormat` points to the file used to format the drawing sheet
- `redlineComponent` points to a 3D markup slide

There are also multiple internal file pointers to the geometry files, thumbnail, tessellated data, and window settings. The bodyGeometry Id provides the key to identifying the bodies described in the other XML and XAML files.

To view the data contained within SCDOC files

1. Copy the .scdoc file you want to view.
2. Replace the .scdoc file extension with the .zip file extension.
3. Extract the files within the zip file.
4. Open the folders and view the contents of the xml and xaml files with an XML viewer like XMLNotepad.

To edit the data contained within SCDOC files

You can safely modify some of the data contained within the .scdoc file; however, when creating SpaceClaim data or extending the SpaceClaim user interface, the API is a better choice. The following steps explain how to edit an annotation within the Flashlight\SpaceClaim\document.xml.

1. Once you extract the files, open the `document.xml` with an XML viewer like XMLNotepad.
2. Search for the text of the annotation you want to change.
3. Edit the text.
4. Save the file.
5. Create a new zip file containing all the previously-extracted files. Make sure that you do not include the top-level folder.

6 Replace the .zip extension with a .scdoc extension.

7 Open the file in SpaceClaim to check your edits.

To view a tessellated model

Open the facets.xaml file with Internet Explorer, or any other xaml viewer.

To view a thumbnail of the file

Open the thumbnail.png file with any graphics program. The thumbnail is displayed in Windows Explorer.

Customizing SpaceClaim

You can customize SpaceClaim to best suit your working style. Most customization is done on the SpaceClaim options window, accessible from the Application menu.

To customize SpaceClaim

1. Select **SpaceClaim Options** from the Application menu to display the SpaceClaim Options window.
2. Click the type of option you want to set from the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

Option types

You can set the following option types:

Popular	Change your application color scheme, set file options for importing and exporting, grid size, application performance (speed vs. graphics quality), and customize the tools that are displayed while you are working in the Design window.
Detailing	Change options for annotations and drawing sheets.
Color	Change the color of the SpaceClaim application window.
Snap	Change the objects snapped to while you are sketching and editing solids.
Units	Set the units for dimensions, the sketch grid, and text height.
Support Files	Set the search path for support files, such as standard thread size tables.
Sheet Metal	Set the thickness, bend, and relief defaults for sheet metal components.
Advanced	Modify how your design changes are displayed in the Design window, whether tools and hints are displayed, enable background loading, change the language, customize the Spin, Select, and Undo tools, reset the layout of the panels, and adjust the license warning.
Customize	Add or remove tools and commands from the Quick Access toolbar.
Add-Ins	Include or remove SpaceClaim add-ins.
Resources	Download sample designs, check for updates, contact SpaceClaim, or view information about this version of SpaceClaim.

Popular options

Change your application color scheme, set file options for importing and exporting, grid size, application performance (speed vs. graphics quality), and customize the tools that are displayed while you are working with your design.

To customize popular options

1. Select **SpaceClaim Options** from the Application menu to display the SpaceClaim Options window.
2. Click **Popular** from the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

To set importing and exporting options for each file type

1. Click **File Options**.
2. Select the file type for which you want to set importing and exporting options.
3. Modify the options for the file type.

To adjust graphics quality and application performance

1. Select a value from the **Image quality versus graphics speed** drop-down.

   Select a low number to increase the speed of the application; select a high number to increase the quality of the graphics.

   Increasing graphics quality may lower the SpaceClaim's responsiveness to actions in the Design window. If you notice a delay when working with your design, modify this option to increase application speed.
2. Check the **Anti-aliasing** box if you want to display text and lines with smooth edges.

   Leaving this box unchecked will help speed performance.

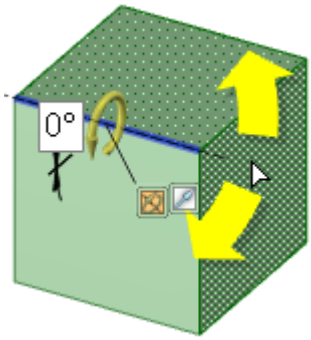
To customize the tools displayed while you are working in SpaceClaim

Modify the settings in the Display Options section. Check:

- **Show tool ruler** to display a ruler in the lower right corner of the sketch grid.
- **Show tool guides** to display icons in the upper right corner of the Design window that help guide you through the editing process and provide greater flexibility when using the tool. Although most tool guide functions can be performed using a combination of keystrokes, we recommend that you leave them displayed for ease of use.
- **Show tooltips** to display hints when you hover over tools, tool guides, and other icons. They briefly explain what will happen when you select the tool and provide some hints on how to use the tool.
- **Show splash screen on startup** to display the SpaceClaim splash screen when you open the application.
- **Show mini-toolbar on selection** to display a small toolbar near your cursor when you right-click. The contents of the mini-toolbar depend on the tool you are using. You can click or scroll the middle mouse button to hide the mini-toolbar, and it fades as you move the mouse away from it.
- **Show tool KeyTips** to display the keyboard shortcuts for each tool in the Ribbon bar.
- **Show arc centers** to display small crosses on the sketch grid at the center of circles, ellipses, polygons, and arcs.
- **Show spin center** to display the axis about which you are spinning your design when you use the Spin tool.
- **Show cursor arrows** to display arrows next to your cursor that indicate the directions in which you can move your mouse to edit the selected object. The arrows also convey the change in size that will occur if you pull in that direction. Adjust the **Cursor arrow transparency** slider to control the transparency of the cursor arrows. Move the slider to the right to make the arrows more opaque; move it to the right to make them more transparent.

Examples

Mini-toolbar showing Add, Cut, and Full Pull options for the Pull tool



Opaque cursor arrows indicating the directions to move the mouse for a revolve

File import and export options

You can open and insert files from many other modeling applications into SpaceClaim for editing, and save your SpaceClaim designs as many different file types. If you work frequently with non-SpaceClaim files, we recommend that you set your file options to optimize the importing and exporting process for your needs.

To customize importing and exporting options

1. Select **SpaceClaim Options** from the Application menu to display the SpaceClaim Options window.
2. Click **Popular** from the navigation panel on the left.
3. Click **File Options**.
4. Select the type of file options you want to change.
5. Click **OK** to save all your file-related changes and close the File Options window.
6. Click **OK** to save all your changes and close the SpaceClaim Options window.

To set general file options

1. Click **General**.
2. Check:
   - **Use SpaceClaim color tones when importing** to use the SpaceClaim color scheme for layers. This option is enabled by default to avoid fully saturated colors, on which highlighting is less visible.
   - **Create multiple documents when importing assemblies** to open or insert a non-SpaceClaim assembly as multiple documents. When you save your design, click **References** to adjust where the documents are saved.
   - **Use matching SpaceClaim documents for faster import** if a design contains an external component that was previously imported and converted to the SpaceClaim .scdoc format,and you want to re-use that previously imported file.
   - **Automatically save imported documents** to immediately save an opened or inserted non-SpaceClaim file to an SCDOC file. (If you do not select this option, then opened or inserted files are not saved until you save the design.)
   - **Improve imported data** 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 more quickly, but you may need to perform these actions manually.
   - **Load model in background** to orient large designs while they are loading.

Check **Use lightweight assemblies** to display a graphics-only representation of SpaceClaim files in the Design window. These files become components when loaded, opened, or inserted. Check **Import JT and CATIA models as lightweight assemblies** to load JT and CATIA models as lightweight assemblies. Check **Import assembly structure only** to display only the structure information about a SpaceClaim, JT, or CATIA, Pro/E, Solidworks, Inventor, or NX file in SpaceClaim's Structure tree.

When you import a non-SpaceClaim file as a lightweight assembly, you can save the unloaded lightweight component as an SCDOC file with only rendering data (that is, without geometry data). You can then open this SCDOC file in a new design and load it normally to include the geometry data.

- **Allow import of hidden components** to open or insert hidden components within CATIA v5, Parasolid, and Solidworks files and turn off their visibility in the Structure tree.
- **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.

To set ACIS file options

1. Select **ACIS**.
2. Select the ACIS version to set the format of exported designs.

To set AutoCAD file options

1. Select **AutoCAD**.
2. Select the AutoCAD version to set the format of exported designs.
3. Check the **Save as Black and White** box to remove any color information from exported designs.
4. Select one of the following import settings:
   - **Open DWG** to use SpaceClaim's translator. It will attach DWG dimensions to the geometry in your design.
   - **Autodesk Real DWG** to use Autodesk's translator. It will disconnect DWG dimensions from the geometry in your design.
5. Select one of the following export settings:
   - **Open DWG** to use SpaceClaim's translator. It will keep dimensions you created in SpaceClaim attached to the geometry in your design.
   - **Autodesk Real DWG** to use Autodesk's translator. It will disconnect dimensions from the geometry in your design.

To set CATIA file options

1. Select **CATIA**.
2. Check **Import part manufacturing information** to include product manufacturing information when you open or insert CATIA designs.

   If you select this option, and you previously selected Import assembly structure only in the General file options, that option will be deselected.
3. Select the CATIA version to set the format of exported designs.

To set Parasolid file options

1. Select **Parasolid**.
2. Select the Parasolid version to set the format of exported designs.

To set STEP file options

1. Select **STEP**.
2. Select the STEP protocol to use to format exported designs.

To set STL file options

1. Select **STL**.
2. Select one of the following output settings:
   - **Binary** to store the file data in binary format.
   - **ASCII** to store the file data in ASCII format.
3. Select one of the following to set the resolution of the output file:
   - **Coarse, Medium, Fine** to use the deviation and angle preset values.
   - **Custom** to use the **Deviation** and **Angle** sliders to set a custom resolution.

     When exporting to STL, resolution indicates the number of sides of a polygon used to represent a circle. The deviation is the difference in distance between the circle's radius and the polygon's radius. The angle is the angle between the edge of the polygon and a tangent drawn through the circle at the same point.
   - **System defined** to use the STL tolerance defined by the graphics tessellation.

Detailing options

You can customize the style of your annotations in a single design or set a custom style as the default for all designs. You can quickly customize the style to conform to ASME or ISO/JIS standards, or you can create a style by customizing note leaders, dimensions, and geometric tolerances.

To customize SpaceClaim detailing options

1. Select **SpaceClaim Options** from the Application menu and click **Detailing** or click  in the Annotation ribbon group on the Detailing tab.
2. Select one of the following from the **Detailing options for** drop-down list:
   - **All New Documents** to create a default detailing style for all your designs.
   - **This Document** to set options for the current design only.
3. Make your annotations, views, and linestyles conform to ASME, ISO, or JIS standards or Customize the default drawing sheet format.

   Select one of the following:
   - **Use external format** to select a pre-defined format provided by SpaceClaim or click **Browse** to select a custom format from any SpaceClaim file.
   - **No format** to use blank drawing sheets of a particular size and orientation.

   If these options are disabled, select **All New Documents** in the **Detailing options for** drop-down.

   Click **Reset to ASME Default** to customize annotations for ASME systems.

   Click **Reset to ISO Default** to customize annotations for ISO systems.

   Click **Reset to JIS Default** to customize annotations for JIS systems.

   JIS defaults are the same as ISO, except that JIS uses third-angle views while ISO uses first-angle views. (A third-angle view is labeled by the object. Therefore, the front of the object is the "Front view" in JIS. A first-angle view is labeled by the direction you are looking. For example, if you are looking front, you see the back of an object. Therefore, the back of the object is the "Front view" in ISO.)
4. Customize drawing sheet views.

To customize General views

1 Modify the **Default view projection**. Select:

- **First Angle** to label the view by the direction you are looking. For example, if you are looking front, you see the back of an object. Therefore, the back of the object is the ”r;Front view.”
- **Third Angle** to label the view by the object. For example, the front of the object is the ”r;Front view.”

2 Modify the **Default front view position**. Select:

- **Top left** to place the front view at the top left corner of the drawing sheet. This is the ISO standard.
- **Top right** to place the front view at the top right corner of the drawing sheet
- **Bottom left** to place the front view at the bottom left corner of the drawing sheet. This is the ASME and JIS standard.
- **Bottom right** to place the front view at the bottom right corner of the drawing sheet. If you use third angle projection, ISO standards also allow this position.

To customize Cross Section views

Modify the following settings:

- **Section line arrow size** - Enter a value to set the size of the arrow shown on the end of the section indicator.
- **Section line length** - Enter a value to set the length of the section indicator.
- **Section cut line extension distance** - Enter the length of the arrows that extend from the section indicator.
- **Section line arrow direction** - Select whether you want the arrows to point toward or away from the section indicator.
- **Trim back section cut line interiors** - Check this box, then enter the length of the section line you want to appear connected to each arrow.
- **Default section name note prefix** - Select how you want the section label to appear on the drawing sheet.

To customize Detail views

Modify the following settings:

- **Detail view name text height ratio** - Set the ratio of a character's height to width. For example, a value of 1.4 sets the character height to 140% of its width.
- **Default view note layout** - Select **One Line** to display the detail name and scale on one line. Select **Two Lines** to display the scale below the detail name.
- **Default detail name note prefix** - Select how you want the detail label to appear on the drawing sheet.
- **Default view scale note prefix** - Select how you want the scale label to appear on the drawing sheet.
- **Detail view boundary note placement** - Select how you want the detail name and scale information to be positioned relative to the detail boundary.

To customize the display of threaded surfaces

Select a value from the **Cosmetic thread display standard** drop-down.

ASME Simplified is the same as ISO and JIS Conventional display standards.

5 Customize the annotation options.

Modify the following settings in the Annotation Options area:

- **Default text height -** Enter the height of the annotation text.
- **Leader circle size -** Enter the size of the circle that connects note leaders to faces.
- **Leader arrow length** - Enter the length of the arrow on note leaders.
- **Leader arrow width** - Enter the size of the arrowhead on note leaders.
- **Leader shoulder length** - Enter the length of the line from the note text to the note leader arrow.
- **Leader textbox gap** - Enterthe size of the margin between the note text and the beginning of the note leader.
- **Center line extend**- Enter the length that a center line will extend past the edge of the object.
- **Default arrow fill style** - Select the style you want to use for the arrowheads on note leaders from the drop-down list.
- **Default dimension text location** - Select how you want to align the note leader text with the note leader line from the drop-down list.
- **Default GTOL font name** - Select the font you want to use for geometric tolerance symbols from the drop-down list. The two fonts in this drop-down list contain all the geometric tolerance symbols necessary. These symbols will be used as necessary in annotations, even when you select a different font for the annotation text.
- **Virtual sharps rendering style** - Select the symbol you want to use to indicate the virtual sharp.
- **Enforce dimension line -** Check the box to use the European standard of keeping the dimension line when the extension lines are shown.
- **Tight gap between dimension line and text** - Check the box to shrink the gap between the dimension text and witness lines.
- **Horizontal dimension text** - Check the box to keep all annotation text oriented horizontally.
- **Override layer color for annotations** - Check the box to set all annotation text to the color selected shown in the Annotation color control. If you choose to override the layer color, select the color used for the override from the **Annotation color** control.
- **Extension line gap -** Enter the size of the margin between geometry and the end of dimension lines.
- **Extension line extend -** Enter the length that the dimension lines cross each other.
- **Dimension line extend -** Enter the length of the arrow that appears outside the dimension lines.
- **Dimension text offset** - Enter the distance between the dimension text and its leader line.

6 Customize the linestyle options.

Modify the following settings in the Linestyle Options area:

- **Default thick lineweight -** Enter the default width for thick lines.
- **Default thin lineweight -** Enter the default width for thin lines.
- Select an object type, then select the **Line style** and **Thickness** for that object.

7 Click **OK** to save all your changes and close the window.

Color options

You can adjust the color of the SpaceClaim application. The Color options page also contains placeholder elements for future color customization options.

To adjust the color of the SpaceClaim application window

Select a color scheme from the Color scheme drop-down.

Select **Custom Color** if you want to choose a different color. You can mouse over the theme colors to display a preview, or click **More colors** and specify a unique color on the Custom tab of the Colors window.

The selected color appears in the background elements of the application, such as the title bar, tab bar, and ribbon bar.

Snap options

You can set snapping options for sketching and editing solids. Units for snap options are set by your Units options for all new documents.

To customize snap options

1. Select **SpaceClaim Options** from the Application menu to display the SpaceClaim Options window.
2. Click **Snap** from the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

To set snapping options for sketching on the sketch grid

1. Enter the angle increment you want to snap to when you press and hold **Shift**.
2. Check **Enable snapping in sketches** to snap to objects while sketching.
3. Check the box next to each item in the list to determine which objects you can snap to while sketching.

To set snapping options for editing solids

1. Check **Enable snapping to solids with the Shift key** to snap when you press and hold Shift using any tool.
2. Set the snapping interval for tool movements in the Incremental area.

   For example, if you set the **Linear increment** to 1 mm, you will pull to whole millimeters instead of fractions of millimeters.
3. Check the box next to each situation or object in the list to determine when a tool will snap while editing solids.

   For example, checking **Round radius** means that when creating a round, its radius will snap to the radii of existing rounds on the component.

Units options

You can set the units for dimensions, the sketch grid, and text height.

To customize units

1. Select **SpaceClaim Options** from the Application menu to display the SpaceClaim Options window.
2. Click **Units** from the navigation panel on the left.
3. Select one of the following from the **Units settings for** drop-down list:
   - **All New Documents** to create a default detailing style for all your designs. These settings will not affect any currently open documents.
   - **This Document** to set options for the current design only.
4. Modify the options on the page.
5. Click **OK** to save all your changes and close the window.

To change the default units

1. Select whether you want to use metric or the imperial system from the **Type** drop-down.
2. Select the default units to be used for length. (Angles are always shown in degrees.)
3. If you selected the imperial system, select whether you want to display fractions or decimals.
4. If you selected decimals, enter the number of decimal places.
5. Check the **Show symbol** box to display the units abbreviation.
5. Check the **Show trailing zero** box to display trailing zeros in decimals.
6. Check the **Show "-" separator** box to display a hyphen between whole values and fractions..
7. Review the preview.
8. Click **OK**.

To modify the sketch grid

1. Enter the spacing between the smallest grid lines in the **Minor grid spacing** field.
2. Enter the number of smaller grid lines between the larger, darker ones in the **Minor grid lines per major** field.

To set the units for text height

Select a unit type for note text from the **Text height units** drop-down.

To convert an existing solid or surface from millimeters to inches

1. Change the units to inches as described above.
2. Select the Pull tool in the Edit ribbon group on the Design tab.
3. Select the object you want to convert.
4. Scale the object by 25.4.

Support file options

You can specify the directories in which you store support files, such as drawing sheet formats or thread size tables. Drawing sheets contained in these directories are displayed within the Format tool.

To set the location of support files

1. Select **SpaceClaim Options** from the Application menu to display the SpaceClaim Options window.
2. Click **Files** from the navigation panel on the left.
3. Click **Add** or **Browse** and navigate to the file or directory you want to include. (You can also select the path and click **Remove** to remove it.)
4. Select a path and click **Move Up** or **Move Down** to order the file paths.
5. Click **OK**.

Sheet metal options

You can set the wall thickness, bend, and relief defaults for sheet metal components. These defaults can be changed for each component or bend by selecting the component or bend and modifying the property values in the Properties panel. Units for sheet metal options are set by your Units options for all new documents.

To customize sheet metal options

1. Select **SpaceClaim Options** from the Application menu to display the SpaceClaim Options window.
2. Click **Sheet Metal** from the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

To set basic sheet metal default properties

1. Enter the default thickness for sheet metal walls in the **Thickness** field.
2. Enter the default bend radius in the **Bend radius** field.
3. Enter the K-factor in the **K-Factor** field.

   The K-factor is a value between .25 and .50 that is used to calculate the bend radius. K-factor is a percentage of the metal thickness and depends on factors such as the material and type of bending operation.

To set bend relief default properties

1. Select the bend relief type that will be created by default.

   You can change this type for an individual bend relief by selecting the bend relief with the Pull tool and clicking a different option in the Options panel.
2. Select one of the following:
   - **Use ratio relative to thickness** to base the relief width and depth on the thickness of the wall.
   - **Use absolute value** to specify the exact width and depth of the bend relief.

Advanced options

Modify how your design changes are displayed in the Design window, whether tools and hints are displayed, enable background loading, change the language, customize the Spin, Select, and Undo tools, reset the layout of the panels, and adjust the license warning.

To set advanced options

1. Select **SpaceClaim Options** from the Application menu to display the SpaceClaim Options window.
2. Click **Advanced** in the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

To customize animation in the Design window

Check:

- **Animate changes to view projection** to animate the steps when you select a view.
- **Animate Full Pull** to animate all the steps when you select the Full Pull option to revolve, sweep, or blend your design.

To customize selection settings

1. Check the **Show previous selections with prehighlight hint** box to highlight all the geometry you last acted on that includes the selected vertex, edge, or face. If you click again with the Select tool, the previously selected group will be selected for you with that one click.
2. Enter how far away your cursor can be from the object you want to select in the **Hit radius** field.

To review a journal of the actions that created a design

Check the **Show Journal tab in the Ribbon** box to save and play back a record of actions used to create a design.

To customize update and license settings

1. Check the **Check for updates on startup** box to use your internet connection to check for updates each time you start SpaceClaim.
2. Enter the number of days before your license expires that you want to receive a warning message in the **License Expiration Warning** field.

To customize spinning

Check the **Rotate about preselected object in Spin** box to spin around the highlighted edge when you are using the Spin tool. You may want to uncheck this box if you are working with large or complicated designs. You can press **Alt** and to rotate around a highlighted object whether or not this option is selected.

To extrude when sketching in Section mode

Check the **Enable sketch auto-extrude/revolve sketches in Section mode** option**.**

As you sketch in Section mode, lines that begin on an existing edge are extruded to form surfaces, and closed surfaces form solids. If the line begins on a solid, the line is finished automatically when you click another point on the solid.

To change the language

1. Select the language from the **Language** drop-down.
2. Click **OK**.
3. Exit and restart SpaceClaim.

To set the number of stored undo actions

1. Enter the number of actions you want available for undo in the **Maximum undo steps** field.

   We recommend that this value be set to at least 50.
2. Click **OK**.
3. Exit and restart SpaceClaim.

To reset the layout of docking panels and windows

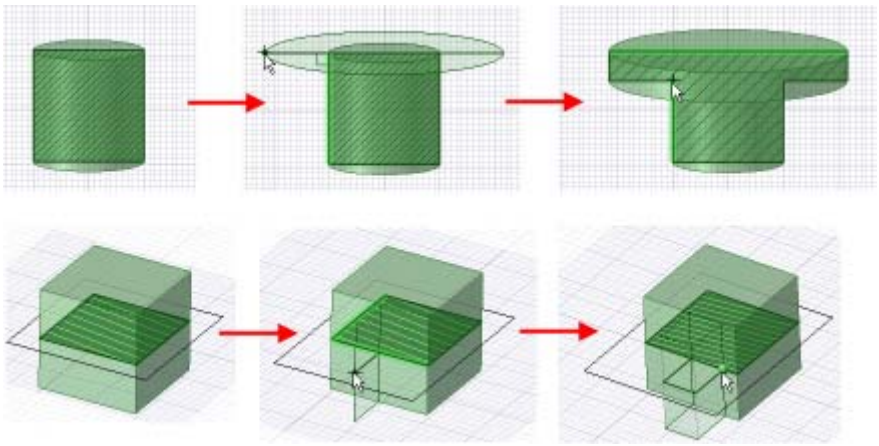
To return the layout of the docking panels and SpaceClaim windows to their original locations, click **Reset Docking Layout**.

To set display options for solids

1. Select when you want interim calculations performed when modifying solids from the **Preview before modifying** drop-down list. If calculations are turned off, previews appear as wireframes. Select:
   - **Auto** to allow SpaceClaim to determine when to perform interim calculations based on the size of the component and the capabilities of your graphics card.
   - **On** to always calculate previews.
   - **Off** to never calculate previews.
2. If you want SpaceClaim to help you when you are moving solids together so that the edges (such as rounds) converge perfectly, select **Enable geometry convergence**. If this option is enabled, when you are moving solids together, SpaceClaim will display a progress bar as it performs the convergence steps.

3. Check the **Show cross sections in sketch** box to display cross sections of solids with hatching. You might want to disable this option if you are working with complicated cross-sections.
4. Check the **Show cross section faces** box to display a cross section through a solid as a face when clipped with the sketch grid in Sketch or Section mode.

Example



Extruding while sketching in Section mode

Quick Access toolbar and ribbon bar options

The Quick Access toolbar (QAT) is located on the title bar. The Ribbon is the menubar that contains all the tools in groups. You can add or remove tools from this toolbar and control the placement and display of the Ribbon.

To customize the Quick Access toolbar and ribbon bar

1. Click next to the Quick Access toolbar.
2. Select the items you want to appear in the toolbar.
3. Select **Place Quick Access Toolbar below the Ribbon** to create a separate toolbar on the SpaceClaim interface.
4. Select **Minimize the Ribbon** to hide the ribbon when you are working in the Design window. Click a tab to display the ribbon bar temporarily.
5. Select **Customize Quick Access Toolbar** to display the SpaceClaim options window. (Any changes you made in the previous steps are saved automatically.)

   You can also select **SpaceClaim Options** from the Application menu and click **Customize**.
6. Select the ribbon group that contains the tool you want to include in the QAT from the **Choose commands from** drop-down menu.
7. Click the tool you want to include and click **Add**.

   Select a tool and click **Remove** to remove it from the QAT.
8. Click **OK**.

Add-in options

The following add-ins are currently available for SpaceClaim:

- **ANSYS Launcher** - Pass designs back and forth between SpaceClaim and ANSYS.

- **Conversion** - Batch convert files to the SpaceClaim format.
- **TraceParts** - Insert a component from an expansive library of standard parts.

You must install and activate each add-in before you can use it. If you want to use an add-in, but it is not available, contact SpaceClaim Customer Support.

To activate an add-in

1. Select **SpaceClaim Options** from the Application menu, then click **Add-Ins**.
2. Check the box next to the add-in to activate it.
3. Click **OK**.
4. Exit SpaceClaim and restart it.

To insert a Trace Part

1. Select **Insert from Trace Parts** from the Insert tool on the Design tab.
2. Select the part you want to insert.
3. Click **OK**.

   The selected part appears as a new component in your design.

Displaying workspace tools

You can adjust the display of workspace tools and the display of your design in the Design window using the tools in the Show ribbon group on the Display tab, and by modifying SpaceClaim options.

To customize the tools displayed while you are working with your design

SpaceClaim offers the following tools on the Show ribbon group on the Display tab to assist you while creating, editing, and detailing your designs:

- Check the **World Origin** box to display the axes that set the default orientation of the design in the Design window.
- Check the **Spin Center** box to mark the center of the spin when using the Spin tool. (This is the same as the **Show Spin Center** SpaceClaim option.)
- Check the **Offset Baseline Faces** box to display offset relationships with blue shading.
- Check the **Coaxial Face Groups** box to display faces that share an axis with blue shading.
- Check the **Lineweight** box to switch the line style of lines (such as those displayed in Hidden Line, Hidden Line Removed, and Wireframe graphics styles) from thin to the thickness set by the Lineweight tool in the Style ribbon group.
- Check the **Adjacent Entities** box to display faint highlighting that appears on edges and faces adjacent when you hover over a point or edge. This feature is useful when you want to extrude the edge of a particular surface that meets another surface.

To display other workspace tools, modify the settings in the Popular SpaceClaim options.

You can also display journal-related tools by checking the **Show Journal Tab** option in the Advanced SpaceClaim options.

Configuring SpaceClaim windows

SpaceClaim has several docking windows that are initially docked along the left side of the application. You can minimize these panels, detach them, or dock them to different sides of the application. You can also dock and detach your design windows.

To minimize a panel

Click the thumbtack icon to minimize the docked panel. Mousing over a minimized panel expands the panel while the cursor is over it. Once the cursor leaves the expanded panel, it returns to its minimized state.

To maximize a panel

Click the thumbtack icon to "stick" the panel to the application window.

To detach a panel or window

Drag a panel by its title bar to detach it. Drag a design window by its tab to detach it.

To dock a panel or window

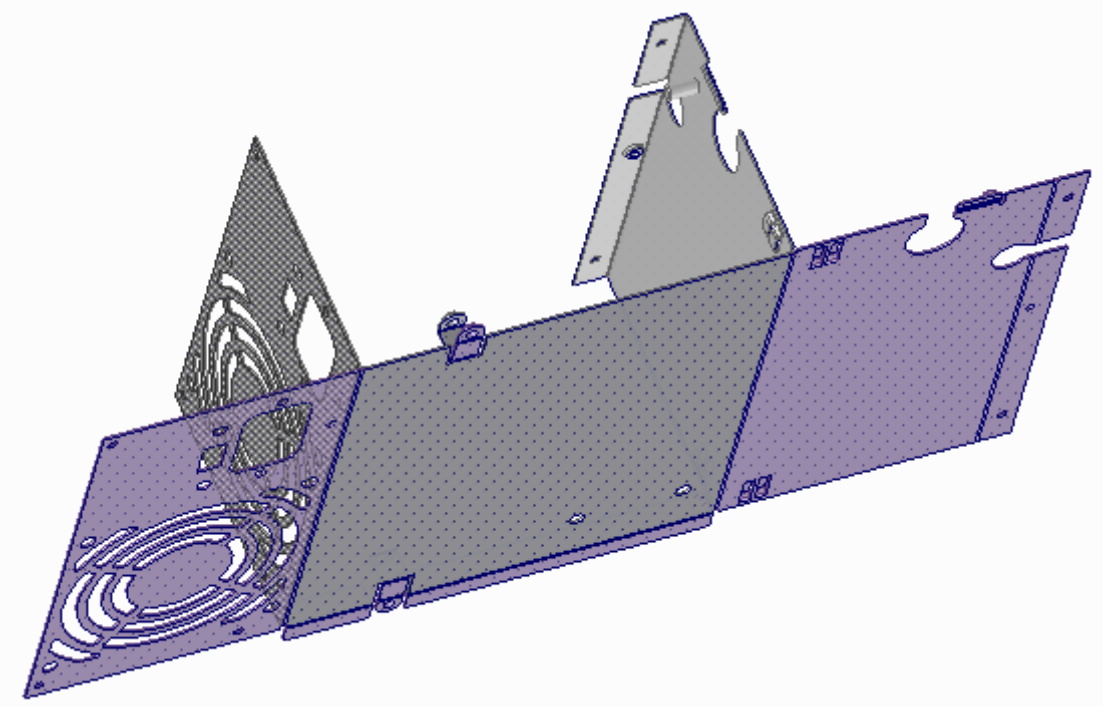
Drag a panel or window by its title bar or tab. As you move the panel or window over the application, icons indicate possible docking positions.

Mouse over an icon to preview the docked location. Release the mouse button to dock the panel at that location.

To return docked panels to their default layout

1. Select **SpaceClaim Options** from the Application menu and select **Advanced**.
2. Click **Reset Docking Layout**.
3. Click **OK**.

Sheet metal



If you purchased the SpaceClaim sheet metal license, you can use SpaceClaim to create sheet metal designs and components. You can unfold a sheet metal design; changes made to a sheet metal component appear simultaneously in the unfolded component and in the original design.

You can modify the thickness, inner radius, and K-factor of sheet metal components in the Properties panel. Modifying the component's properties affects all the objects in the component. Modified property values appear in bold. You can set the default values for sheet metal components by setting sheet metal options. Delete a modified property value to return it to the default value.

The thickness of sheet metal is maintained with an offset relationship. A sheet metal component retains its properties when you move it into a non-sheet metal component, but they are only visible if you move it into a sheet metal component unchanged.

SpaceClaim only recognizes a component as sheet metal if it consists only of planar and cylindrical faces.

You can create patterns of flanges and notches on sheet metal parts.

Sheet metal pull options

The following sheet metal options are available in the Pull tool in addition to the regular pull options when you are working with a sheet metal component:

Junction Type

Select the type of joint you want to create when you pull on an edge. The available options are no overlap, partial overlap, full overlap, bend, and flange. These options are also available in the mini-toolbar.

You can also:

Click to reverse the junction overlap.

Click to remove the rips and bends from the geometry and return to a shell, where there is no inside and outside radii. (You can also rotate a 180° "hem" to create a 90° flange.)

Click to remove the junction behavior, leaving only the geometry.

Bend Relief Type

When pulling a partial wall, select the bend relief type. You can edit the bend relief's depth, width, and type in the Properties panel. (Right-click the bend face and select **Properties**.) Bend reliefs are created automatically when they are required, including when you split a face. The available options are round, square, rip, and corner relief. These options are also available in the mini-toolbar.

If a relief cannot be made, such as when it is tool close to the side, a message appears in the status bar.

Bend Behavior

When you pull the edge of a piece of sheet metal, how the junction is created depends on the edge you select and the direction you pull. When you pull *across* the other edge, the selected edge becomes an outside corner. When you pull *away* from the other edge, the selected edge becomes an inside corner.

The length of the inside surface of the wall is maintained.

The length of the outside surface of the wall is maintained.

Sheet metal properties

The following sheet metal properties are available in the Properties panel when a sheet metal component is selected. Modifying the component's properties affects all the objects in the component. Modified property values appear in bold. Delete a modified property value to return it to the default value. A sheet metal component retains its properties when you move it into a non-sheet metal component.

Sheet Metal Set this property to **True** to convert the selected component to sheet metal.

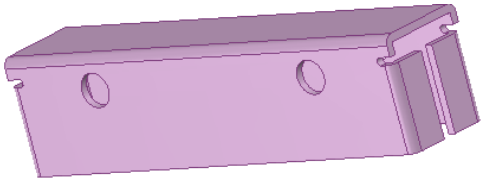
Thickness Enter the thickness for sheet metal walls. The thickness is maintained with an offset relationship.

Inner Radius Enter the default inner radius for bend junctions, or enter the bend radius for the selected junction.

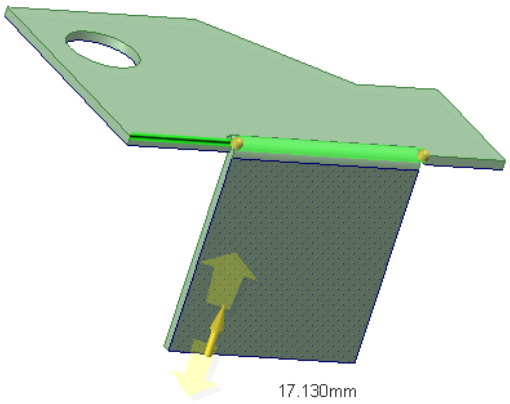
K-Factor K-factor is a fraction (usually 0.25 to 0.50) used to calculate bend allowance, or the material required to correct for the change in length caused by bending a flat sheet. In addition to K-factor, bend allowance is influenced by factors such as material thickness and bend radius.

K-factor relates to the depth of the neutral axis, a line within the sheet where the length does not change when the sheet is bent. The inside of the bend is under compression, the outside is under tension, and the neutral axis occurs somewhere between the midpoint of the material (K-factor=0.50) and a point closer to the inside of the bend (K-factor=0.25).

Examples



Example sheet metal part



Pulling a partial sheet metal wall

Creating a sheet metal design

You can create a sheet metal design using SpaceClaim's sketch and editing tools.

To create a sheet metal design

1. Right-click the component or top-level design and select **Sheet Metal**.

   The Structure tree icons change to indicate that the design or component is sheet metal.
2. Sketch the sheet metal wall.
3. Pull the face of the sheet metal wall.

   The sketch snaps to the thickness defined in the Properties panel.
4. (Optional) Edit the junction type, bend relief type, and bend behavior (while in the Pull tool) with which future junctions will be created by selecting a junction type from the Options panel or mini-toolbar. The type of an existing junction can also be changed by selecting an edge or face of the junction and then changing its type in the Options panel, mini-toolbar, or Junction Type drop-down on the Properties panel.

   To reverse a junction, select an edge or face of the junction and click the **Reverse Junction** option in the Options panel or mini-toolbar.

   If you are planning on pulling a partial wall, select the bend relief type from the Options panel to specify the type of bend relief that will be created when you pull it. You can edit the bend relief's type, depth, and width in the Properties panel by selecting one of the edges in the bend junction.
5. Select a linear edge of the sheet metal solid.

   The selected edge displays handles (yellow balls) at each end. The location of the handles can be adjusted to set the length of the flange or wall you want to create. (Drag the handles along the edge or press the spacebar to dimension the change in endpoints.)

   The pull arrows appear. (Do it faster: You can also mouse over the edge and pull.) If the pull arrows do not appear, then the selected edge cannot have material added to it. Select the arrow that shows the direction in which you want to pull by clicking on it or pressing Tab to switch between the two arrows.
6. Pull the edge to create a bent wall.

   If **Use Edge Location** is selected as the Bend Behavior option, when you pull the edge of a piece of sheet metal, the resulting junction is created depending on the edge you select and the direction you pull. When you pull *across* the other edge, the selected edge becomes an outside corner. When you pull *away* from the other edge, the selected edge becomes an inside corner.

When the **Inside** option is selected, the length of the inside surface of the wall is maintained. If the **Outside** option is selected, the length of the outside surface is maintained.

If you pull the top edge of a wall in the direction along the face with the **No Overlap** option selected, the new wall is offset by one thickness from the top. If you pull the bottom edge, the wall is offset one thickness from the bottom. If you select **Partial overlap** or **Full overlap**, the wall is extended without a junction. If you select **Bend**, an S curve is created. If you select **Remove Junction**, the junction no longer acts as a sheet metal junction.

You can use the Up To tool guide to pull the wall up to a face or edge of another object.

If you bend a wall back over itself to create a hem, a 0.0001 mm gap is created automatically.

7 (Optional) Select the sheet metal component in the Structure tree and edit the sheet metal properties in the Properties panel.

You can adjust the thickness, inside radius of bends, and K-factor for the sheet metal part. K-factor is a parameter of bends in a sheet metal part used to calculate the bend radius. K-factor is a percentage of the metal thickness and depends on factors such as the material and type of bending operation. The K-factor can range from .25 to .50.

For example, 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.

Continue editing your design using SpaceClaim's tools.

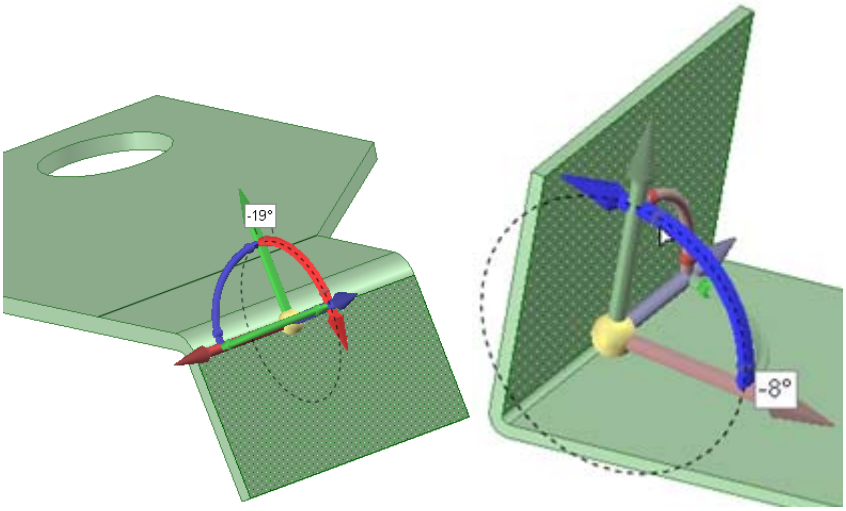
Editing a sheet metal design

In addition to the regular functions associated with the Pull and Move tools, several additional functions are available when you are working with sheet metal.

To rotate a sheet metal wall

Select one face of the sheet metal wall and rotate it with the Move handle.

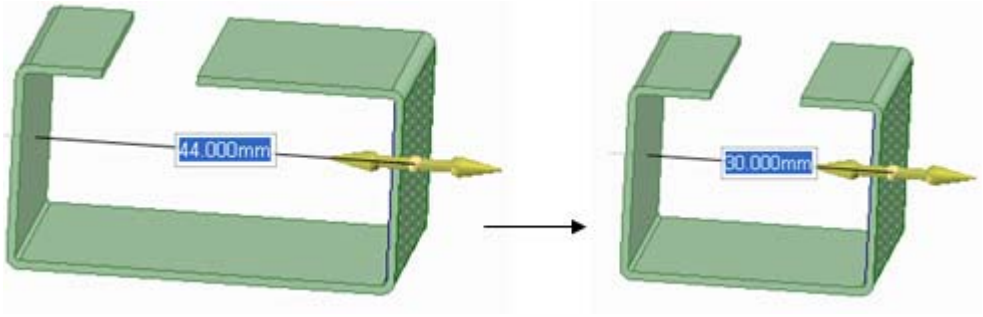
A default junction to rotate around is chosen based on which junction causes the smaller portion of the part to rotate. To rotate around the other junction, drag the Move handle anchor (the center ball) to an edge on the other junction.



To move or dimension the location of walls

Select the face of the wall to move. Pull the wall in the direction you wish to move it, dimensioning the change from its current location by typing or using the spacebar. To dimension the distance between two walls, use **Create Ruler Dimension** from the Options panel or the mini-toolbar to place a dimension

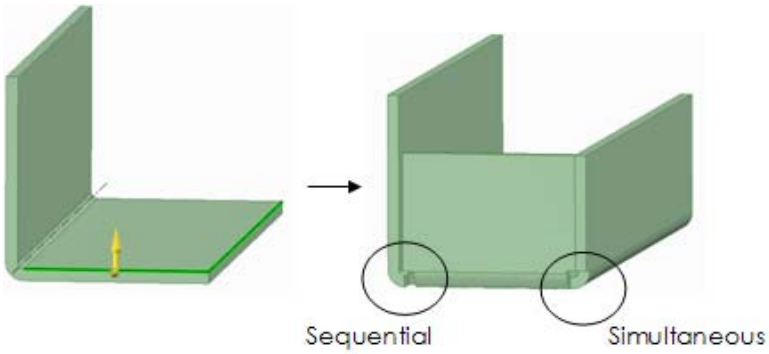
between the selected wall and another wall. Changing the value of this dimension will move the selected wall and update the surrounding geometry. The Move tool should be used to translate or rotate larger groups of walls. Bend reliefs move with the walls.



To create corner reliefs

Selecting multiple edges of a sheet metal wall and pulling them simultaneously creates the necessary relief in the corners in order to make the walls bendable. Pulling walls that form a corner simultaneously or sequentially produces different results. Both cases are shown below. When you create a corner junction, the No Overlap option is used by default.

Do it faster	Use the Selection filter in the status bar to select only vertices (by un-checking faces and edges), then box-select a sheet metal part to quickly select all the corners needing corner reliefs.



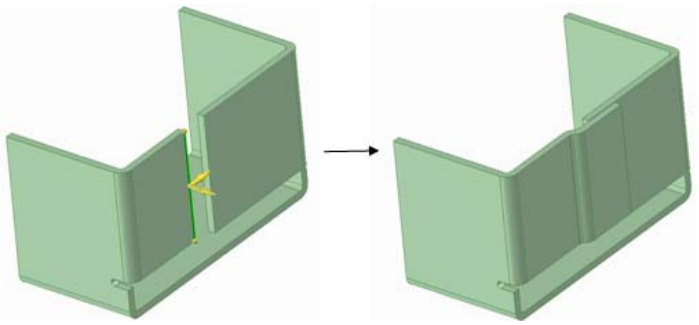
To fill a face with a sheet metal rip

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 to connect two corner reliefs, two points on two edges, or a combination.

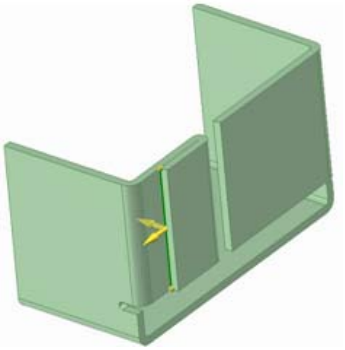
Click the face to fill the rip. The tool guide also snaps to 90 degree intersections with other edges.

To create a jog in a sheet metal wall

Select the edge of a sheet metal wall and then Pull it along the direction of the wall with the Bend joint type selected in the Options panel. The wall will jog up one wall thickness and continue in the same direction. An example is shown below.



Pulling the selected edge along the wall but opposite its direction creates a hem offset by one wall thickness as shown below. You can also create a partial hem.



Converting a design to sheet metal

You can convert existing designs to sheet metal within SpaceClaim.

To convert an existing component to sheet metal

Set the **Sheet Metal** property of the component to **True** to convert it to sheet metal.

To create a sheet metal box from a shelled box

1. Right-click the component containing the shelled box in the Structure tree and select **Sheet Metal**.
2. Create corner reliefs.
3. Classify all edges as rips or bends, depending on how you what the part to unfold.

To quickly convert a protrusion to sheet metal

Click the Shell tool's More Shell tool guide, then click the protrusion to create a shell with the same thickness as the sheet metal component.

To convert a plastic part to sheet metal

1. Eliminate rounds to create sharp corners.
2. Remove draft and internal features, such as ribs and punch-outs.
3. Convert the component to sheet metal.
4. Correct the wall thickness.
5. Create corner reliefs.

6 Classify all edges as rips or bends, depending on how you want the component to unfold.

Unfolding a sheet metal component

Introductory paragraph. Insert animated image to the left of the topic title and align it to the right.

To unfold a sheet metal component

1 Right-click a face of a sheet metal component with at least one bend junction.

The clicked face sets the orientation of the unfolded design.

2 Select **Unfold Part** from the context menu.

An unfolded version of the component is displayed in a new Design window with its overall dimensions, as shown in the figure below. It also appears in the Structure tree as an unfolded part . The visibility of the unfolded part in the original design is initially set to Off in the Structure tree. The unfolded part is saved as part of your design.

Overall dimensions on an unfolded part are measured based on the orientation of the sketch grid.

Bend lines for an unfolded part are placed on a Bends layer with the visibility off. Turn the visibility on in the Layers panel to view the bend lines.

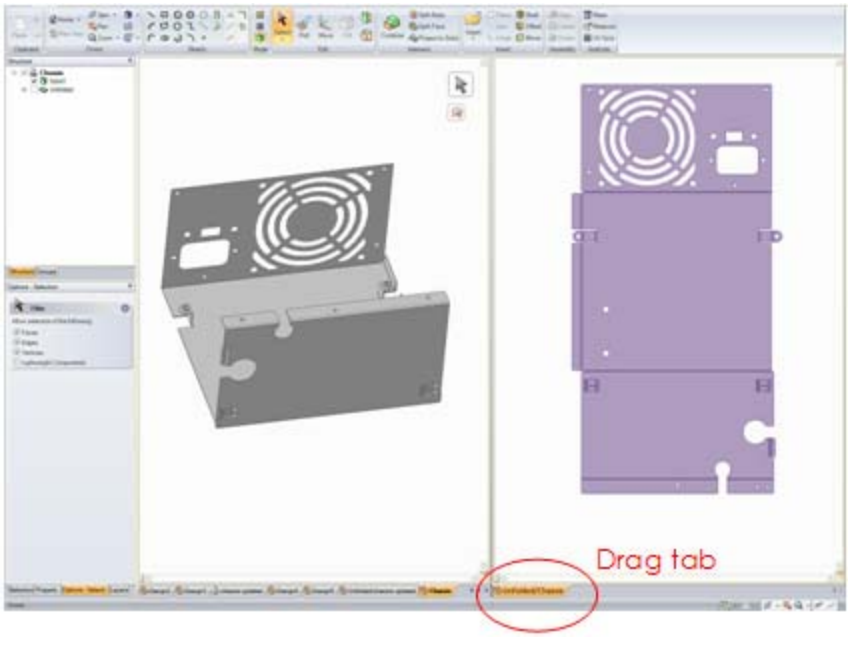
If an unfolded component has conflicting geometry, the conflicting geometry is made a separate surface in the Structure tree and highlighted in red, as shown in the figure below. The edge that prevents the unfold is also highlighted.

You can also right-click the part and select **Validate Body** to display non-manifold areas in red.

To work simultaneously on the folded and unfolded view of a design

The unfolded view of a sheet metal part can be placed next to the folded view in order to work simultaneously on both views. Changes made to the part in one view update on the other.

To place views next to each other, click and drag the tab for a Design window and drop it next to another view.



To export an unfolded part as .dxf

The unfolded sheet metal part can be exported as a dxf file to be used in manufacturing. After placing desired dimensions and notes on the unfolded view using the tools in the Detailing tab, set the Graphics Style in the Display tab to be Wireframe. This step prepares the model for dxf export and shows you what the resulting image will look like. Next, select Save As from the SpaceClaim menu and specify AutoCAD files (*.dxf) as the export type. The unfolded model will be saved as a 2D dxf file with the included annotations.

Calculations for unfolded lengths

SpaceClaim can now use the sheet metal bend deduction to calculate unfolded lengths. You can edit this value in the **Bend Deduction** property for the selected bend. The bend deduction is 2 times the distance from the outside mold line to the beginning of the bend (that is, the "set back") minus the bend allowance.

The **Bend Allowance** property contains the length of the arc through the bend at the neutral axis of the sheet metal wall being bent. Bend allowance and Bend deduction are linked. If you enter the value for one property, the other is calculated automatically.

To use a sheet metal bend table to calculate unfolded lengths

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.

Version	1.00					
Type	BendDeduction					
Units	mm					
Thickness	1.000					
AngleValues		90	60	45	120	135
Radius	0.500	0.300				
Radius	1.000	0.300				
Radius	2.000	0.300	0.500	0.660	0.280	0.200
Radius	3.000	0.300	0.500	0.660	0.280	0.200
Radius	4.000	0.300	0.500		0.280	0.200

You can clear the sheet metal bend table assignment from a component by selecting the blank value from the **Bend Table** property drop-down.

The table on the right shows a simple bend table as it would appear in a spreadsheet editor. Keywords are shown in bold. The table of values for a given thickness is shown with a shaded background: angle values in blue, radius values in green, and bend deduction values in yellow.

When you unfold a sheet metal design, the thickness, inner bend radius, and angle from the design is used to determine the value in the selected table.

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.

Bend table files are stored as comma-separated value (CSV) files which can be edited in a spreadsheet editor, such as Microsoft Excel, or in a plain text editor.

```
Version,1.00
Type,BendDeduction
Units,mm

Thickness,1.000
AngleValues,,90,60,45,120,135
Radius,0.500,0.300
Radius,1.000,0.300
Radius,2.000,0.300,0.500,0.660,0.280,0.200
Radius,3.000,0.300,0.500,0.660,0.280,0.200
Radius,4.000,0.300,0.500,,0.280,0.200

Thickness,2.000
VeeDieWidth,5
AngleValues,,90,60,45,120,135
Radius,1.000,0.300
Radius,2.000,0.300,0.500,0.660,0.280,0.200
Radius,3.000,0.300,0.500,0.660,0.280,0.200
Radius,4.000,0.300,0.500,,0.280,0.200
```

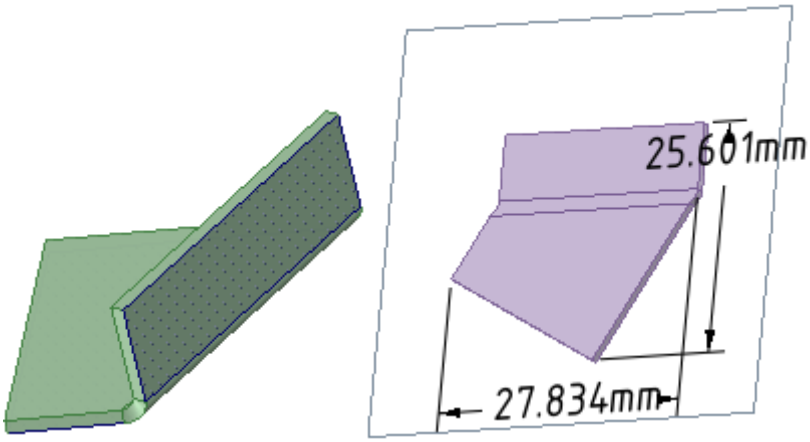
This file shows the same data as in the previous table, but as it is stored in the comma-separated value file. Each line consists of a keyword, followed by one or more parameters. This example also shows how data for multiple thicknesses would be stored by including two bend deduction tables.

Keyword description parameters for bend tables

Keyword	Description	Parameters
`Version,<version-number>`	Specifies the file format version.	The file format version number is provided. The current version is 1.00.
`Type,<type-name>`	The type of data contained in the file.	A single string representing the type of data contained in the tables. Currently the only supported value is `BendDeduction`.
`Units,<units-type>`	The default units are used for numeric values which do not have units provided explicitly.	Must be `in`, `mm`, or `cm`.
`Thickness,<value>`	Begins a bend table definition.	A single parameter specifying the thickness for which the bend deductions in the table are defined.
`VeeDieWidth,<value>`	Optional. Specifies an optional tooling parameter, allowing multiple sets of bend deduction values for a given thickness. Must occur after a `Thickness` keyword.	A single parameter specifying the width of the die tool.
`AngleValues,,<value>,…`	Defines the columns of the bend deduction table. Note the extra empty cell to aid in table alignment. The values defined by the table are associated with the preceding `Thickness` and `VeeDieWidth` keywords.	A list of angles, specified in degrees.
`Radius,<radius>,` `<deduction-value>,…`	Defines the body of the bend deduction table. Individual deduction values may be omitted.	The first parameter (radius) provides the inner bend radius for the table row. The following parameters all provide the deduction values

Keyword	Description	Parameters
	The total number of deduction values provided cannot exceed the number of values provided above for the `AngleValues keyword.`	for each combination of angle and radius.

Example



Folded and unfolded skewed bend

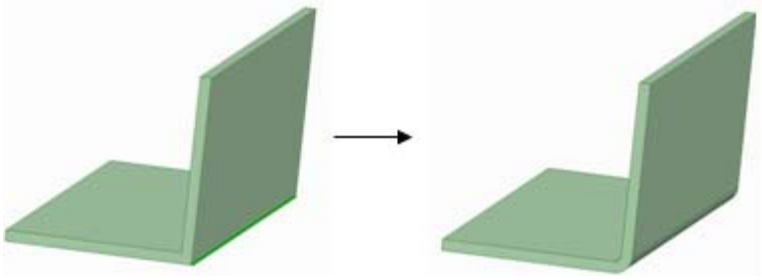
Correcting a sheet metal component

SpaceClaim's sheet metal functionality can be used to fix components that are intended to be made from sheet metal but which do not conform to the sheet metal standards required for production. There are a number of typical errors in sheet metal creation that can be easily remedied in SpaceClaim.

To correct a sheet metal component

1 Fix joints that are not bendable.

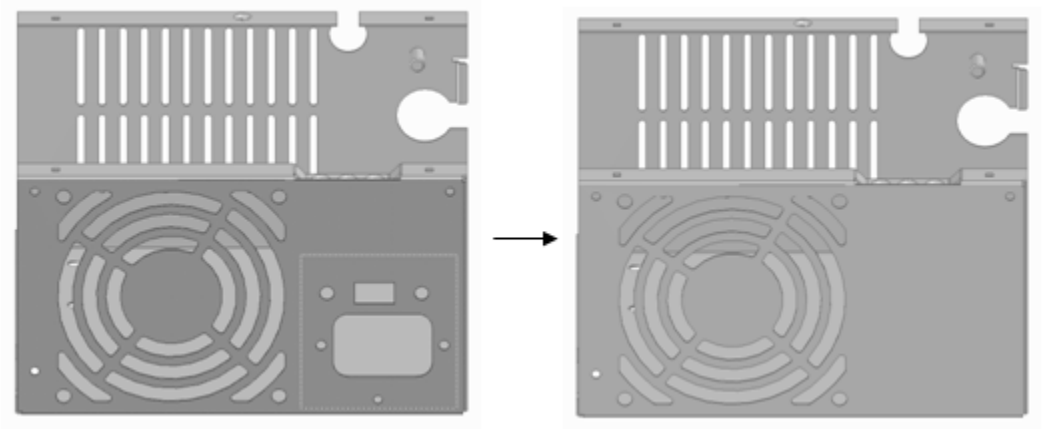
Select the edge of an improper joint with the Pull tool and then pick a joint type in the Options panel to have SpaceClaim remedy the problem. Once a proper joint has been created, it will update with changes to part thickness, bend radius, and K-factor as well as show the joint's bend allowance on the unfolded design. An example of this change is shown below.



2 Fill in extraneous geometry.

A sheet metal component often has a number of details (such as punches, louvers, and vents) that may not be relevant to each person involved in the manufacturing process. SpaceClaim allows these details to be quickly and easily filled in order to simplify parts and retain focus on only the elements necessary for each step of the manufacturing process.

To fill in geometry, use Select to box select desired features and then Fill to remove them. An example is shown below.



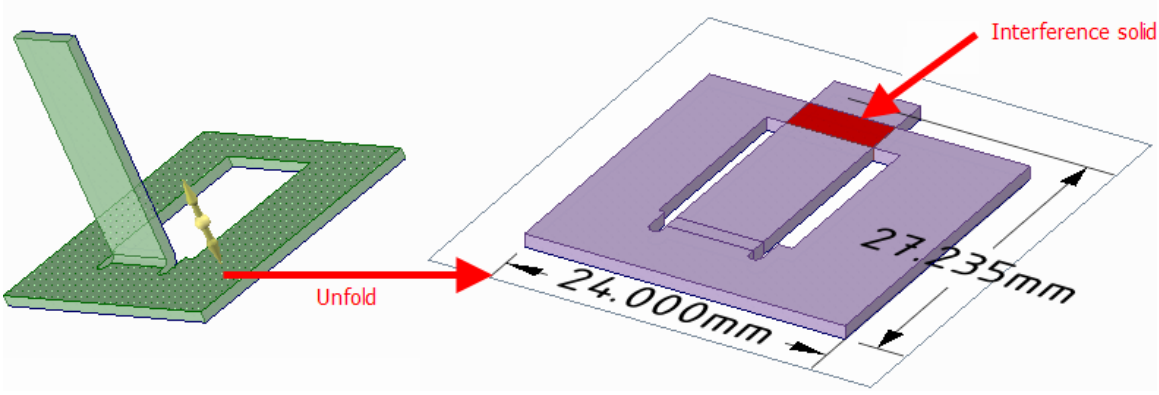
To identify non-manifold parts

Right-click a sheet metal component and select **Validate Body** to highlight all non-manifold parts in red. A non-manifold part is one in which you cannot tell the inside of the sheet metal from the outside.

An example of a manifold part is a sheet metal tank for holding water. It is always clear that the inside of the wall is where the water is and the outside of the wall is where the air is.

This is especially useful tool when troubleshooting a part that has been imported into SpaceClaim.

For example, if you create an L-shaped wall in SpaceClaim, then use the Move tool to rotate the short wall all the way around to make a hem, SpaceClaim inserts a small gap so that the whole gap remains manifold. If a part imported from another program is not manifold, the L-shaped wall appears as a thick wall folded over, with no indication of inside and outside. Highlighting in red will help to troubleshoot the non-manifold parts that have been imported into SapceClaim.



Unfolded sheet metal component, showing the non-manifold area in red

SpaceClaim add-ins

The following add-ins are currently available for SpaceClaim:

- **ANSYS Launcher** - Pass designs back and forth between SpaceClaim and ANSYS.
- **Conversion** - Batch convert files to the SpaceClaim format.
- **TraceParts** - Insert a component from an expansive library of standard parts.

You must install and activate each add-in before you can use it. If you want to use an add-in, but it is not available, contact SpaceClaim Customer Support.

Developing SpaceClaim add-ins

The SpaceClaim Application Programming Interface (API) allows you to create add-in applications that extend the functionality of SpaceClaim. An add-in application is a managed code DLL that uses Microsoft® .NET Framework 3.0 and the SpaceClaim API.

For more information about the SpaceClaim API, please review the Developer's Guide (English only). The Developer's Guide was written by developers, for developers. It is located in your SpaceClaim API installation directory. There is also class library documentation. To display these files, double-click the `SpaceClaim_API.chm` and `API_Class_Library.chm` files.