SpaceClaim Introduction
Training Session

A SpaceClaim Support Document
In this class we will walk through the basic tools used to create and modify models in SpaceClaim.

We will focus on:

- SpaceClaim User Interface
- Selection Methods
- Sketcher
- Pull/Move/Combine/Fill
- Spin/Pan/Zoom
- SpaceClaim Options
  - Import/Export
- Component/Structure Management
- Detail Drawing
- Help & Technical Support

Other learning resources

SpaceClaim provides online help (F1) tutorials, and other training materials to help you become productive with SpaceClaim as quickly as possible. At your convenience, you can check out the SpaceClaim Video Tutorials library.
Section One

Creating Geometry
Getting Started

Close the Welcome Screen if it appears. Go to the Application Menu (SpaceClaim logo in top left corner) and click on SpaceClaim Options.

Select 'Units' and change Metric to Imperial.
Sketch with Dimensions

Click the Rectangle Tool in the Sketch group. Click once anywhere to begin the sketch.

Note: Hovering over a tool will display a tooltip. This will give you more details on what the tool does and provides the keyboard shortcut in parenthesis.

Note: Clicking F1 while a tooltip is shown will give you context sensitive help on that tool.

Set the dimensions by typing '4'. Then press Tab to enter the next dimension and type '2'. Press enter to finish the sketch.

Note: Spacebar will also freeze the sketch and allow you to tab between dimensions.
Circle
Click the ‘Circle’ tool in the Sketch group.

Click once to place the center of the circle on the line at the midpoint marker △

Click a 2nd time at the endpoint of the line to snap the radius of the circle to the endpoint.

Note: the △ marker on the 2nd click indicates tangency
Three-Point Arc

Click the 'Three-Point Arc' tool in the Sketch group.

Click once to set the start point, a second time to determine the endpoint at the second corner of the rectangle, and a third time to set the arc's radius to 1in and chord angle to 180°.

Note: Just prior to the 3rd click to finish the arc, the circle on the left will become highlighted in green. This indicates the arc you are creating is snapping to equal radii with the highlighted circle.
Trim Away
To remove extra lines from a sketch click the ‘Trim Away’ button in the sketch group.

Select any lines you’d like to remove from the sketch. This will eliminate them from your sketch. Click the 3 line segments in the images to the right.

*Note:* You need a closed sketch to create a surface in 3D.

*Note:* The structure panel now has a “curves” folder with 2 lines and 2 arcs in it.
Pull

Click the 3D button to exit the sketch then click the Pull tool

Note: In the structure panel the ‘Curves’ folder has been replaced with a ‘Surface,’ and in the design window the closed loop of lines has been filled in.

Select the surface by clicking on it. Once a surface is selected it can be pulled to create a solid from anywhere on screen (you do not need to click on the small yellow arrow).

Hold the left mouse button and drag the surface upward. Use the Spacebar to freeze the dimension and type ‘.5’ to make the solid half an inch thick.

Click anywhere in the white space around the model in the design window to clear the selection

Note: In the structure panel the ‘Surface’ has been converted into a ‘Solid’
Sketch on Face

Click the ‘Circle’ icon in the Sketch group.

Note: Activating a sketch tool while in 3D mode automatically switches to Sketch mode.

Select the top face of the solid as the sketch grid.

Note: The grid icon attached to the mouse cursor indicates a grid location needs to be selected.

To bring the selected surface parallel to the screen, select the ‘Plan View’ button from the Orient group towards the left of the ribbon bar.

Note: The sketch minitoolbar will have common actions such as:
Return to 3D
Choose a new sketch plane
Move the grid
Go into a plan view.
Sketching Circles

Click on the axis of one of the full rounds (a small green circle will appear to indicate concentric alignment) and sketch a circle with a diameter of .75in.

Note: You can type “3/4” or any formula

Shift-Touch Dimensions

Hover the mouse cursor over the center of the circle and tap the shift key once. This Shift Touch procedure sets this reference point as the base point to place a second circle.

Type 1 to enter the horizontal x value, press tab to switch to the vertical y dimension, type .5 and press enter to place the center of the circle

Slowly move the mouse cursor away from the center of the 2nd circle. The 1st circle will become highlighted in green, indicating equal radii. Click to place the circle, snapping to equal radii.
Exit Sketch
Click the ‘Pull’ button to exit the sketch and enter 3D mode. Pull will normally be the first tool used when exiting a sketch. It is used to create and modify geometry.

Note: Notice there are no new surfaces in the structure panel. Sketching on a face creates imprinted regions instead of new surfaces.

Click the ‘Home’ button to obtain a trimetric view.
View Navigation
Take a moment to explore manipulating the view.

The middle mouse button (MMB) alone spins.

Shift+MMB pans.

Ctrl+MMB Zooms.

When you are finished press the ‘Home’ button to get back to a trimetric view.
Create Holes
Select the left circular imprinted region. Holding 'Ctrl' allows you to select on multiple entities. Hold Ctrl and select the 2nd circular face and click and drag to pull them down and through the model to create holes.
Move Holes
Click the Move Tool.

Select the cylindrical face of the hole on the right.

Click and drag the red straight move handle arrow to the right. A new dimension from the original location will appear. Press Spacebar to freeze the dimension and enter in a value of 1".

Create Ruler Dimension
Click and release the straight green move handle arrow and a mini toolbar will appear. Click the Ruler button in the middle.

Note: the Ruler button is also located in the Options Panel

Select the cylindrical face of the hole to the left. The axis through the hole can also be selected.

Type in 0" to align the center of the holes along the length of the solid.
Copy a Hole

With the Move tool still active hold the CTRL key and the straight red move handle selected, click the UpTo Toolguide .click the cylindrical face on the right side of the solid.

Note: Holding the CTRL key while moving or pulling in any direction creates a COPY of the selected geometry. The copy is made in the direction of the arrow.

Click the straight red move handle arrow and a mini toolbar will appear. Click the UpTo button on the right.

Note: the UpTo button is also located in the ToolGuide group

Select the cylindrical face on the far right side of the solid.

Note: In this situation the move handle was anchored to the center of the hole. The “Move UpTo” command translated the center of the hole “Up To” the axis through the center of the selected cylindrical face on the right side of the solid.
Fillet Rounds
Select the Pull Tool and click the edge of the right hole. Notice that it will highlight in bright green. Drag in the direction of the arrows to Pull the edge into a fillet round.

To enter a dimension, hit Spacebar and type in 0.2. Repeat this on the other hole and use the UpTo Toolguide to snap the second round to the first, by selecting the edge on the 2nd hole, click UpTo and select the first round.
Copy Edge

Select the top edge of the middle hole and the copy edge option from the mini toolbar or the options panel.

Press Tab until the horizontal arrow on the edge is highlight and the arrows around the mouse cursor are aligned to the length of the solid. Drag the mouse cursor in the direction of the arrow, press spacebar and type in .375"

*Note: Pressing ‘tab’ toggles between the 2 yellow perpendicular pull direction arrows.*
Create Protrusion
Click on the circular ring face in the middle of the solid.

Pull it up an arbitrary distance and click in white space to clear your selection.
Create Ruler Dimension

Select the top face of the cylinder as indicated and choose the ‘Create Ruler Dimension’ (either from the Options window on the left or the mini-toolbar popup).

Note: You can make the mini-toolbar popup just by clicking an object, such as a face. If you have multiple objects selected, i.e. 2 faces, release CTRL after selecting the objects to bring up the mini-toolbar.

Click the top face of the solid to dimension from.

Once the dimension is set up, enter ‘.75’ as a value and hit enter to set the height of the tube.
Sketch Mode
Select the front face of the model and click the ‘Construction Line’ button, even while the pull tool is active. (This will put you into Sketch mode).

Note: If a face is selected when you switch from 3D to 2D (either via the Sketch Mode button or any Sketch tool) that face will automatically become the plane for the sketch grid.

Click ‘Plan View’ in the upper left to orient the sketch.
Sketch Mirror
Create a construction line from the midpoint of the edge shown to the midpoint of the top edge.

Click Select from the edit group, select and right click on the construction line and click “Set as Mirror Line.”
Sketch Mirror

Click the 'Line' tool from the edit group and start a line at the base of the construction line. Make the line .375" to the left or right. Make a second line .625 upward perpendicularly. Finish the sketch with a third line back to the construction line.

Click Select in the edit group, select the construction line and press Delete on the keyboard.
Create Surface

Click ‘Pull’ to exit the sketch and click ‘Home’ to orient it in a trimetric view.

Note: In the structure panel a ‘Surface’ has been created because the sketch was not on the face of the solid.
Pull Up To
Select the new surface you've created and go to the toolguides on the right of the screen.

Click the bottom toolguide which is ‘Up To’. This allows us to snap a surface or face up to another.

After the Tool guide is selected select the outer cylindrical face behind the surface.
Fillet a Chain of Rounds
Double click one of the outer edges to grab a loop of edges. Place a round of .1 on the model. Click anywhere in white space to deselect the model.
Apply Full Round
Hold down ‘Ctrl’ and select on the three surfaces highlighted at right. Right click and select ‘Full Round’ from the pop-up menu to create a full round based on the three surfaces.

*Note: The context-sensitive RMB menu allows common tasks to be completed with little cursor movement.*

Pull Hole
Select the cylindrical face of the center hole and change the radius to .5in.
Save Part

Save the part as ‘Base.scdoc’ in the Introduction_Session folder.
Completed Section One
The base is now completed.
Workshop One (optional)

Using the image as reference, create a new part.

Use Sketch and Pull to make the model.
Section Two

Working With Assemblies
Open Assembly
Go to the SpaceClaim Application Menu and select Open.

Select ‘Assembly.scdoc’ and hit Open.
Insert File
Click the 'File' button from the Insert group towards the right of the Ribbon bar.

Select 'Base.scdoc' and hit Open to insert the document into your design.

The part is added to your Structure tree.
Create New Component
Select the Assembly in the Structure tree, right-click and select 'New Component' from the Context Menu.

Name the new component 'Ram_Screw' and hit Enter.

*Note: New components are active by default.*
Reordering Structure Tree

Select the three components with 'ram_screw' in the name.

Drag and drop these components into the active 'Ram_Screw' component.
**Active Components**

Only the solids inside the Ram Screw sub-assembly are now active. The other components appear greyed out.

To activate the whole assembly, right-click on the Assembly in the Structure tree and select ‘Activate Component’.
Assembly Conditions
Select the green face indicated on the base. Hold the Ctrl key down and add the light green face from the bracket to the selection.

Press ‘Align’ in the assembly group in the upper right corner of the screen.

Next select on the green cylindrical wall on the base then hold the Ctrl key and add the light green cylindrical wall of the bracket to the selection.

Press the ‘Align’ button.

*Note: The first selection always moves to meet the second selection.*
Open Component

Right-click on the green Y-shaped bracket and hover over the ‘Component’ submenu and select ‘Open Component’.

*Note: You can also Right-click the component in the Structure tree.*
Round Selection
The bracket should now appear in its own design window.

Select the round indicated at right in the bracket.
Fill
Once the round is selected hit the ‘Fill’ button to remove the round. Fill works by removing the selected geometry and attempting to extend the surrounding geometry to close the gap that is created.
Rotate
Rotate the model (using the Middle Mouse button - MMB) and Fill the matching round on the other side.

Note: There are also Spin/Pan/Zoom controls at the top left and bottom right of the screen.
Selection Tab
Go to the Selection tab which is below the Structure window. Select the round indicated in the picture below.

The Selection window will automatically search for geometry with similar characteristics.
Selecting Rounds
Select 'All rounds equal to or smaller than 0.100in' radius rounds.

Once they are selected as indicated in the picture, hit 'Fill' to remove them from the model.
Fill Conic Faces

Rotate the model (using MMB) so you have a view of the bottom. Hold Ctrl and select the four conical surfaces shown.

Use ‘Fill’ to remove them from the model.
Changing Design Windows

Use the ‘Home’ button to get back to a trimetric view.

Select the ‘Assembly’ tab at the bottom left of the screen to return to the top-level assembly.
Create an Axis
Select the cylindrical surface indicated.

Click the ‘Axis’ tool. This will place an axis through the part at the center of the cylindrical surface.
Section Mode
While the axis is selected press the Section button in the Middle of the Mode group.

Select the ‘Plan View’ button to orient the view as shown.

If the model is displayed differently, change the grid options in the Display tab to match the settings on the right.
Moving Components

Expand the Structure tree and select on base_screw.

Select the ‘Move’ tool.

Click and drag the horizontal move handle (red, straight & horizontal in the image to the right) to translate the base_screw 0.4 in along that axis.
Combine
Select the ‘Combine’ button and notice that ‘Keep Cutter’ from the Options window on the left is checked by default.

Note: In cross section a vertex represents a 3D edge, and an edge represents a 3D face.
Remove Material
Click an edge of the dark green base, and then the pink screw. This will cut away the intersecting geometry. Click on an edge of the intersection to delete it from the model.

Note: To select geometry in a section it is necessary to click the edge of the part. The area represented by the cross hatching is not selectable.
Change Hole Diameter
Click the 'Pull' tool. Select the green edge indicated in the picture. Pull the edge one way or the next to see how the geometry changes.
Pull Up To
Select the 'Up To' toolguide on the right of the screen. Select the light green edge of the bracket indicated to snap the geometry into place. Click anywhere in white space to deselect the model.
3D Mode
Click the ‘3D Mode’ button to exit section mode. Click the ‘Home’ button to get to a default view of the model.
Zoom
Zoom into the top of the model to see a screw that came in as separate parts.
Combine

Click the ‘Combine’ tool and uncheck the ‘Keep cutter’ option on the left under Combine Options. Select the pink solid then the green solid.

Note: This will generate red areas which you can delete by clicking on them.

Delete the red cylindrical depression created.
Merge Solids
Holding Ctrl switches the toolguide on the right side of the screen to merge solids instead of split them.

Hold Ctrl and click the purple solid and pink solid to combine the two together.
Save the Assembly
Click the ‘Select’ button to exit the Combine tool and the ‘Home’ button to center the model.

Save your document.
Completed Section Two
The assembly is now completed.
Workshop Two (optional)

Open the ‘Workshop2.scdoc’ file.

Using the techniques in this section, such as Structure Tree ordering, creating components, and Assembly conditions, reconfigure the model shown on this page to look as it does on the next.
Hint: You may need to use a Tangent Assembly condition to assemble the base_screw to the bracket. Or, since the base_screw and the bracket are correctly positioned together when you open workshop2, you can either Anchor the bracket or assemble everything TO the bracket (based on order of geometry selection).

Structure

Workshop2
- base_screw
  - Align
  - Tangent
  - Solid1
- bolt
  - Align
  - Align
  - Solid1
- bracket
  - Align
  - Align
  - Tangent
  - Solid1
- Ram Assembly
  - Align
  - Align
  - Align
- Nut
  - Align
  - Align
  - Solid1
- Ram
  - Align
  - Align
  - Solid1
- Screw
  - Align
  - Align
  - Solid1
Section Three

Detailing

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Revision History

- INDICATION
- DESCRIPTION
- REMARKS
- SIGNATURE

Parts List
New Drawing Sheet
To make a drawing, select the design window tab for the assembly and go to the Application Menu in the far upper left of the screen. Go to ‘New’, then ‘Drawing Sheet’.

This will open up a new tab with a drawing of the assembly.
Detailing Tab
Here you will be brought to the 'Detailing Tab' where you can add Dimensions, Cross Sections, General Views, and Annotations.

First, take a look at the Structure tree.
General View
Select the ‘General View’ button in the Views Ribbon group.

Place the general view in the empty area on the sheet.

Note: Select a View (inside the dashed grey outline) and drag to move it on the Drawing Sheet.
Cross Section View
Select the ‘Cross Section’ button.

Select View 3 (the Side view).

Drag the mouse over View 1 (the Front view). As you move the mouse, View 3 should appear as a cross section.

Click the center of the assembly to place the section plane.
Bill of Materials

Select the Bill of Materials button in the Annotation Ribbon group.

Select Indented from the dropdown.

Place the BOM in the corner of the Drawing Sheet. You may need to move the front view over to fit.
Add a Dimension

Select the ‘Dimension’ button.

Select one side of the ‘bracket,’ then the other.

Note: Placing a dimension with one reference will display the length of the edge, and with two references will display the distance or angle between them.

Click below the base to place the dimension.
Completed Section Three
The drawing is now completed.
Workshop Three (optional)

Open the file ‘Base.scdoc’ created in Section One. Create a Drawing Sheet that matches the image below.
Congratulations!

You’ve completed our Introduction Training Class.

If you have any questions or comments please contact Support@SpaceClaim.com