

Ansys Fluent Getting Started (New Fluent Experience)

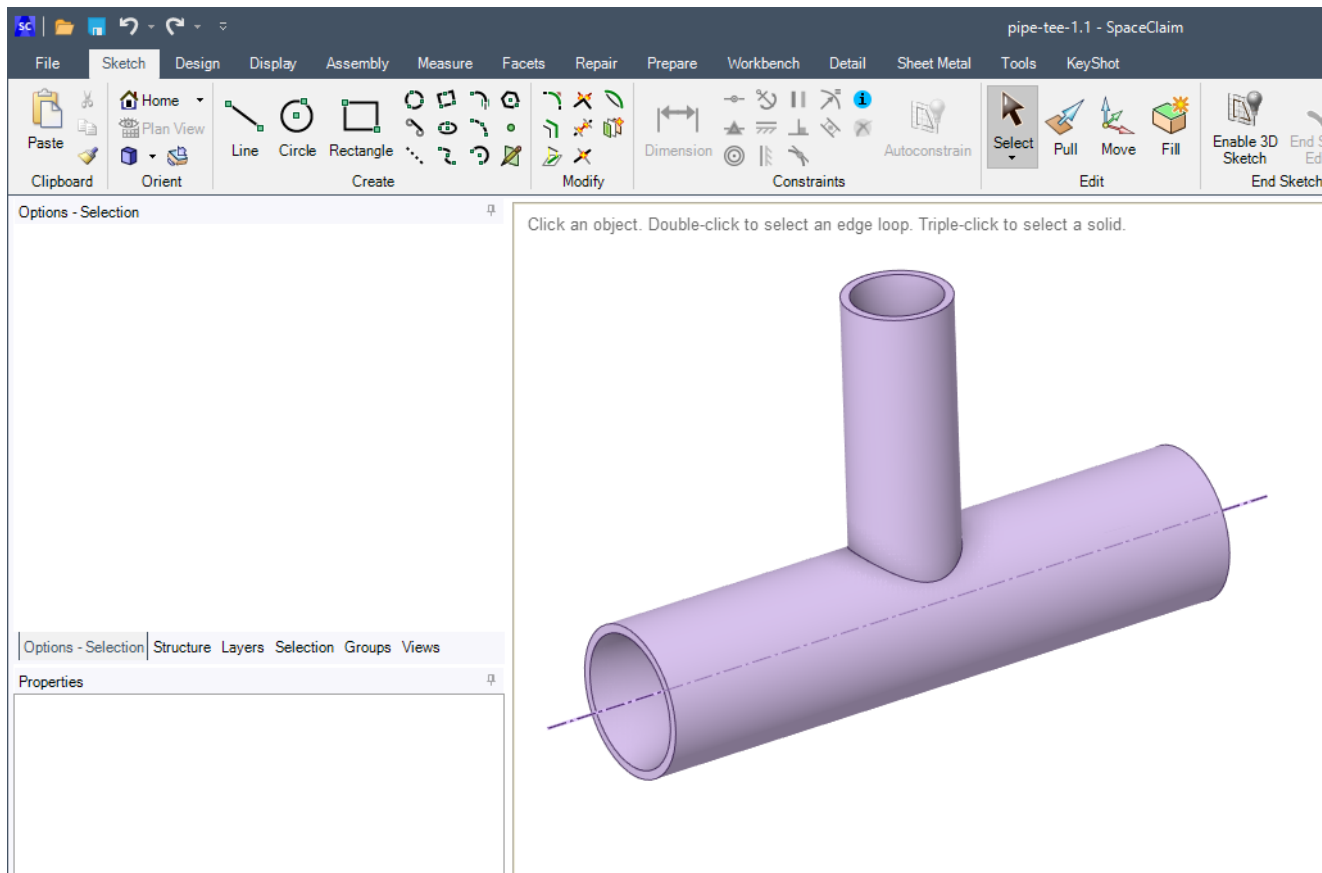
Workshop: Extracting a Fluid Volume in SpaceClaim

Release 2021 R1



Launch SpaceClaim and Import File

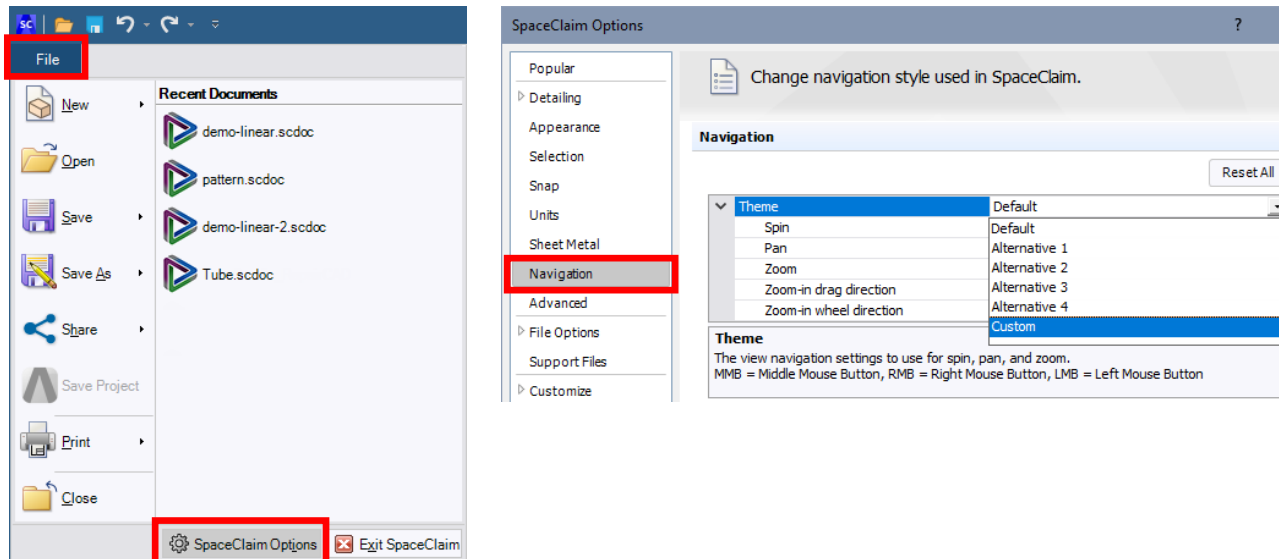
- Start SpaceClaim, go to File > Open and select "pipe-tee-1.1.scdoc"



Alternatively, after starting SpaceClaim, you can drag and drop the CAD file into the SpaceClaim window

Graphics Manipulation

- By default, the middle mouse button (MMB) is used to spin the model, Shift+MMB is used to pan the model and Ctrl+MMB is used to zoom
 - Often more convenient just to use the scroll wheel to zoom in and out
- Mouse Functionality can be customized in SpaceClaim Options

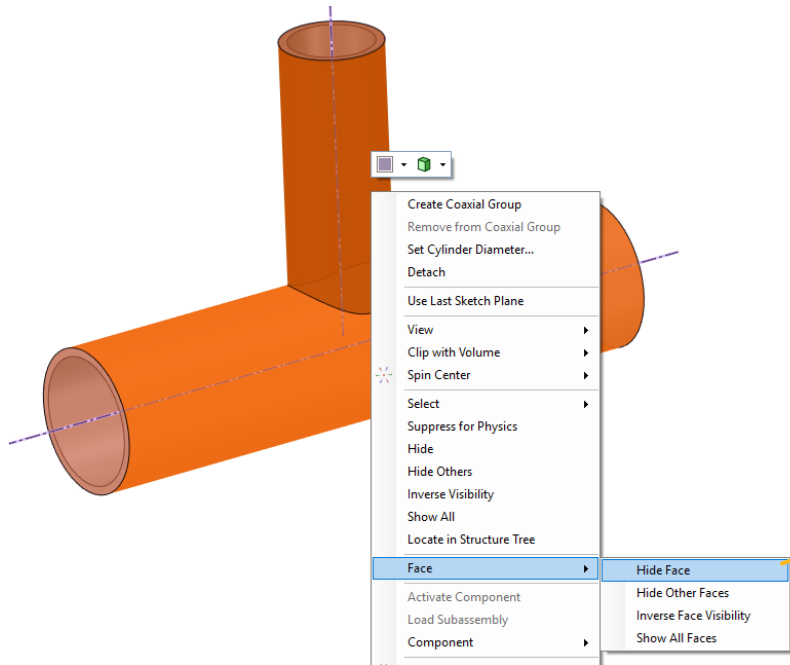


Select	Orient	Dimension
Single-click Click to select one object	Spin	Spacebar Edit dimension while dragging
Double-click Double-click to cycle through Face / Edge / Curve loops	Zoom	Tab Toggle between editable dimensions
Triple-click Triple-click to select all the faces of a body	Scroll	Shift + [Mouse Wheel] Hover over geometry to dimension from it while sketching
Ctrl + [Mouse Wheel] Add or remove an object from the selection	Pan	Enter Accept the dimension value Complete most tools
Shift + [Mouse Wheel] Select all objects between the original selection and this object	Snap View	Shift + Alt + [Mouse Wheel] Create temporary objects
Alt + [Mouse Wheel] Select a driving or alternate object for many tools (in blue)	Double-click	[Mouse Wheel] + [Mouse Wheel] Create a linear / angular ruler dimension
[Mouse Wheel] + [Mouse Wheel] Select all objects completely within the box	Home View	Ctrl + G Create Group and store ruler dimension
[Mouse Wheel] + [Mouse Wheel] Select all objects partially within the box	Plan View	[Mouse Wheel] + [Mouse Wheel] Drive a ruler dimension with an Annotation dimension
Scroll Select other objects under the cursor	Zoom Fit selection	U Select an Up To reference
[Mouse Wheel] + Drag Click to get command menus Drag to invoke gesture shortcuts	Previous view Next view	
[Mouse Wheel] + Drag Use the Select-Bounds toolguide to stop the propagation of selected faces and edges		
[Mouse Wheel] Revert to the last set of selected items		
Esc To exit current tool and return to selection		

Default Select and Orient guide from ANSYS SpaceClaim Documentation

/ Hide Face

- Use the left mouse button and Ctrl key to select the two large outer faces of the model, then right click and choose Face > Hide Faces from the context menu



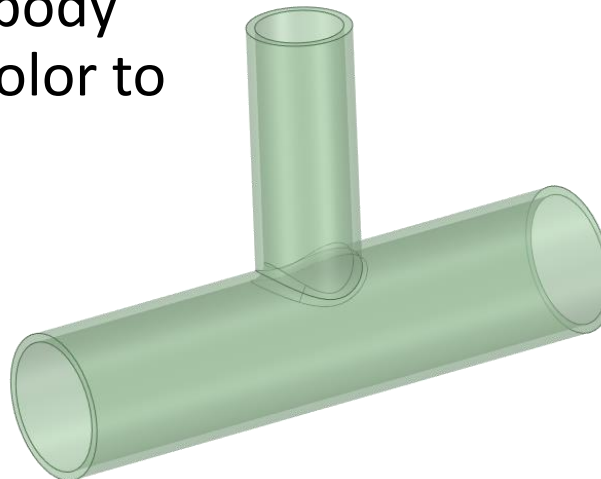
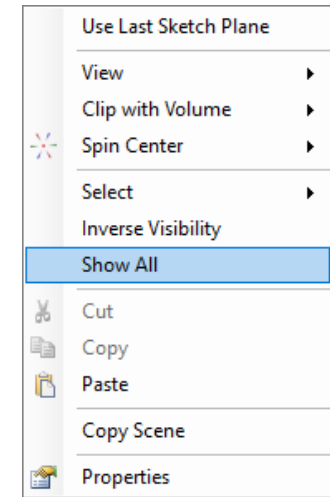
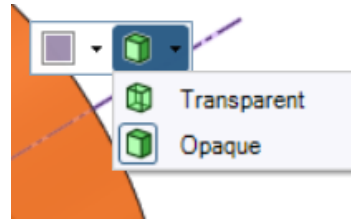
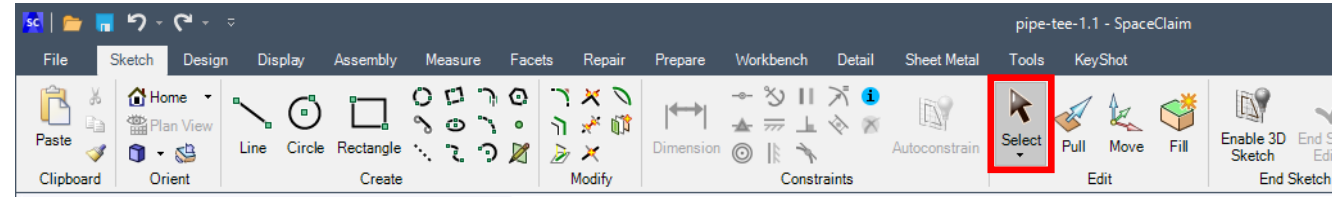
Hiding the outer faces reveals the presence of a round at the intersection of the pipes.

Current selection set is reported in the lower right corner of the SpaceClaim window



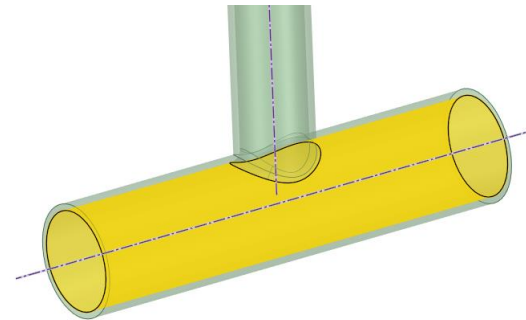
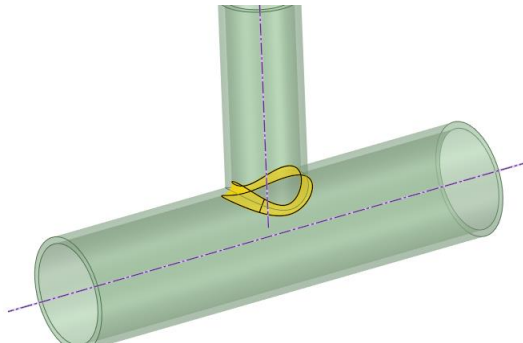
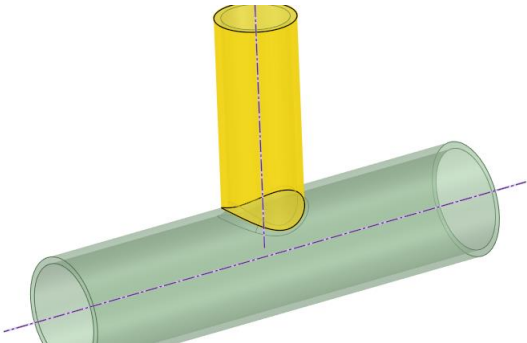
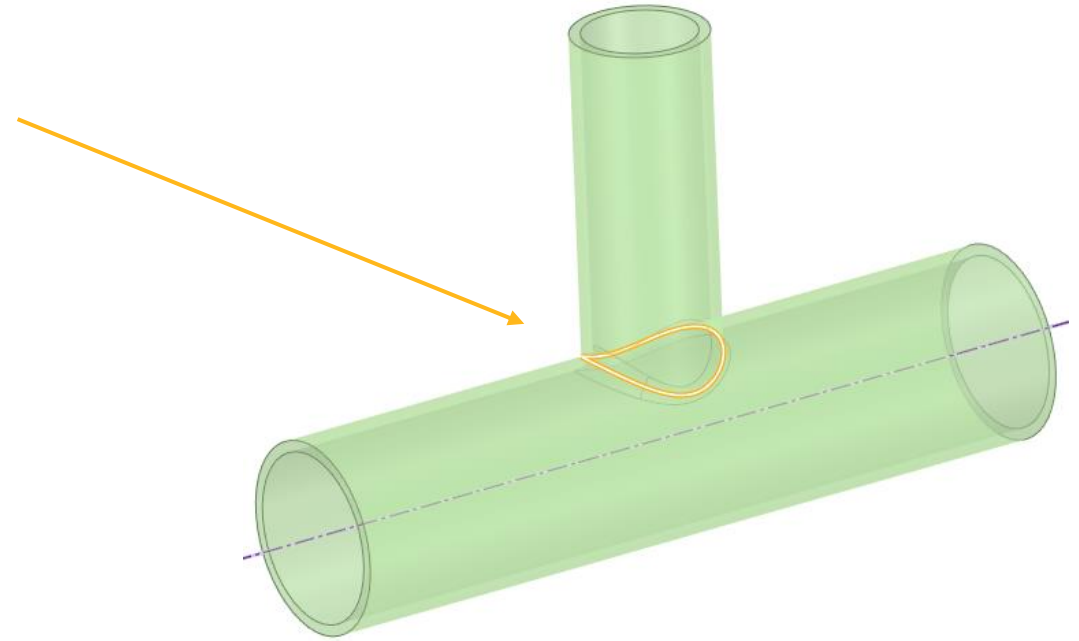
Transparency and Color Settings

- Ensure SpaceClaim is in Select mode
- Right click anywhere in the window and select Show All
- Left click anywhere on the body and note the Style Override menus
- Use the menus to make the body transparent and change its color to green



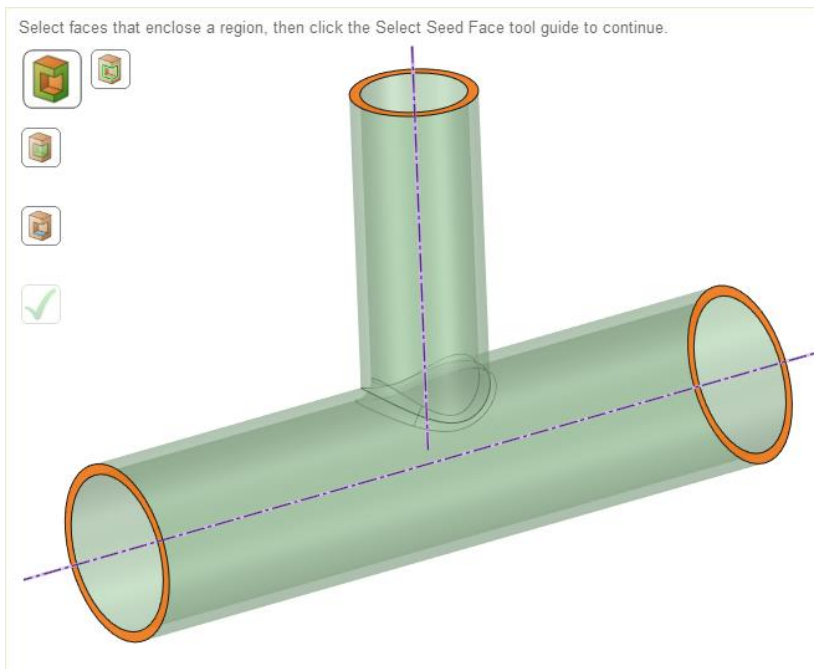
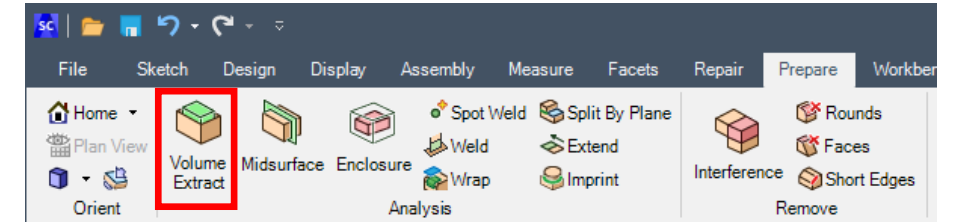
/ Selection

- Place the mouse pointer near the location shown below, but do not click anything
- Hold down the Ctrl key and move the scroll wheel
 - Note how nearby objects are cycled
 - Left clicking will select the currently highlighted object
 - This can be a convenient way to select faces and edges that are not on the outermost surface

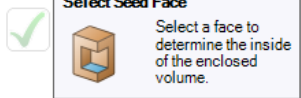


Volume Extract

- Click on the Prepare tab in the ribbon and select Volume Extract
- Select the faces shown in the figure
 - Remember Ctrl+scroll wheel can be used to select faces that are not in the foreground
- Click on the Select Seed Face tool guide
 - (continued on next slide)

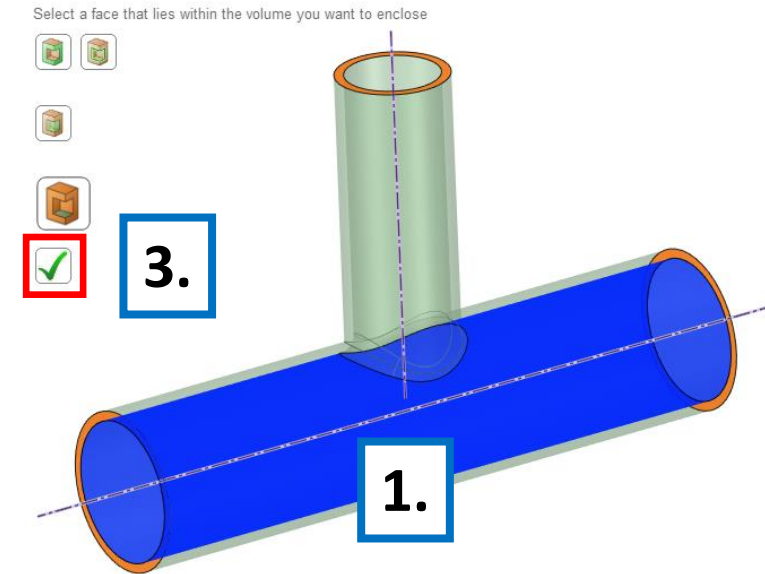
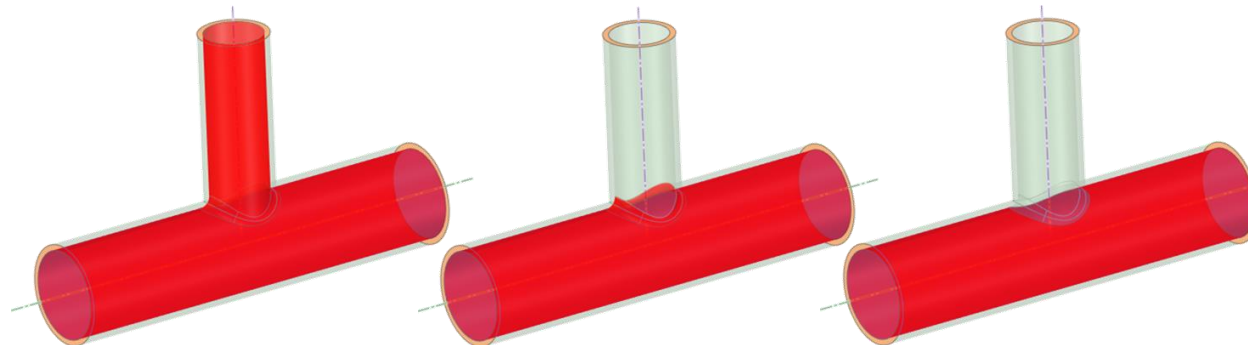
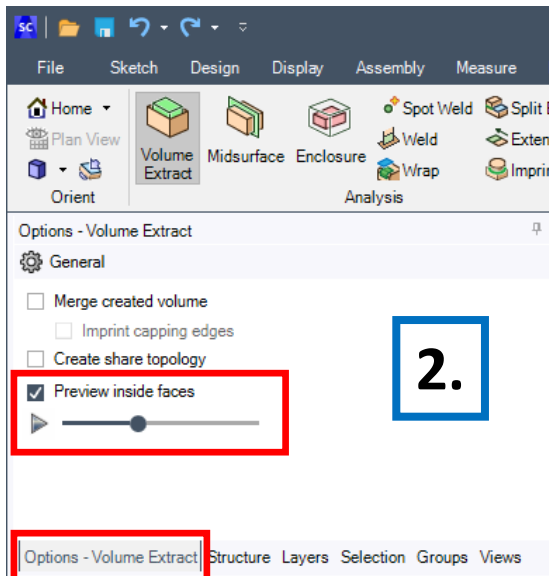


Select faces that enclose a region, then



Volume Extract

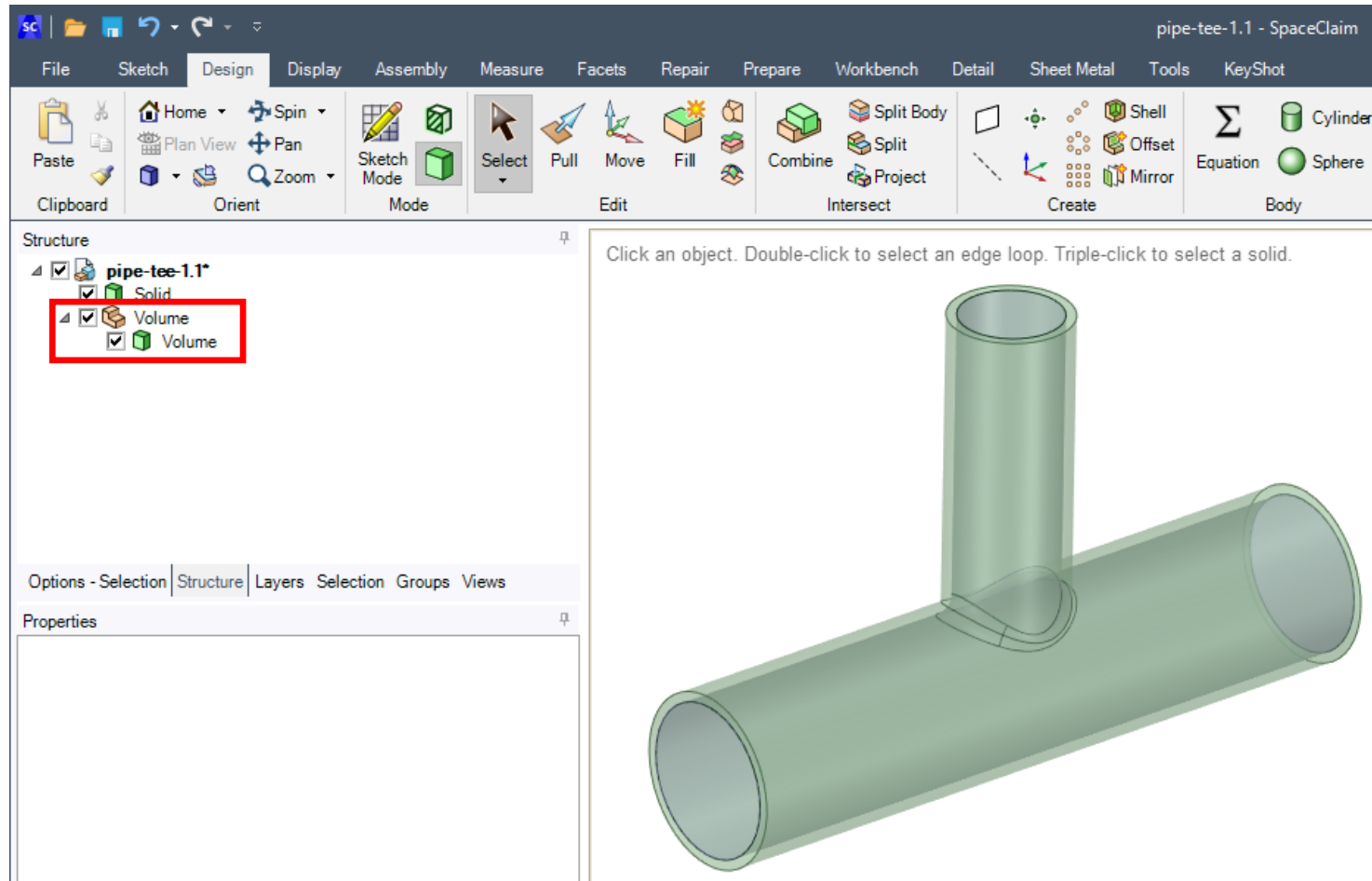
- Select the blue face as shown in the picture
- Go to the Options tab and tick Preview Inside Faces
 - Faces of the volume that will be extracted are highlighted in red
 - Drag the slider bar to view the progression of faces from the initial seed face
- Click the green check mark to complete the volume extraction



Hint: After completing the volume extraction, the Esc key can be used to clear the tool guides from graphics

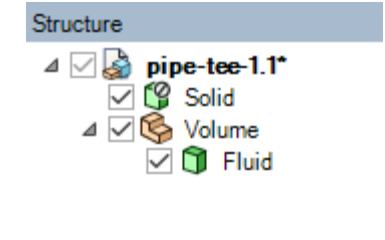
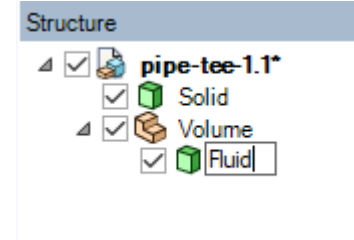
Structure Tree

- The newly created volume appears in the structure tree

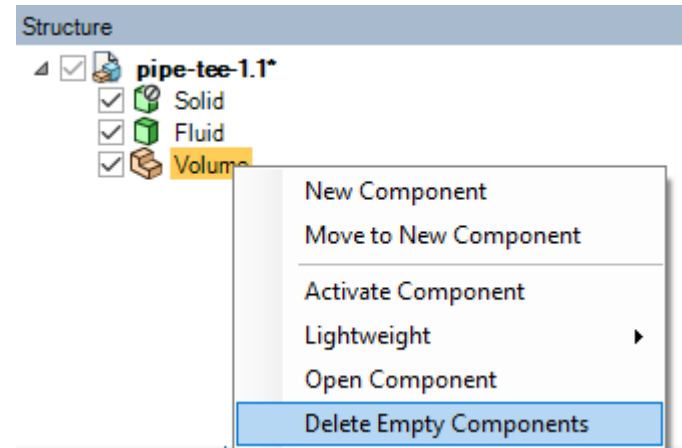


Rename Fluid and Suppress Solid for Physics

- Left click Volume once, then left click again (2 single clicks) and change its name to Fluid
- Right click the solid body in the tree and select Suppress for Physics
 - If desired to include the solid in the simulation model, for instance to calculate heat transfer in the solid, this step would be omitted
- There is no particular reason for the fluid body to be in its own component ... if desired, select in the structure tree and drag it into the upper level assembly with the solid
 - If desired, the empty component can be deleted using the right click context menu

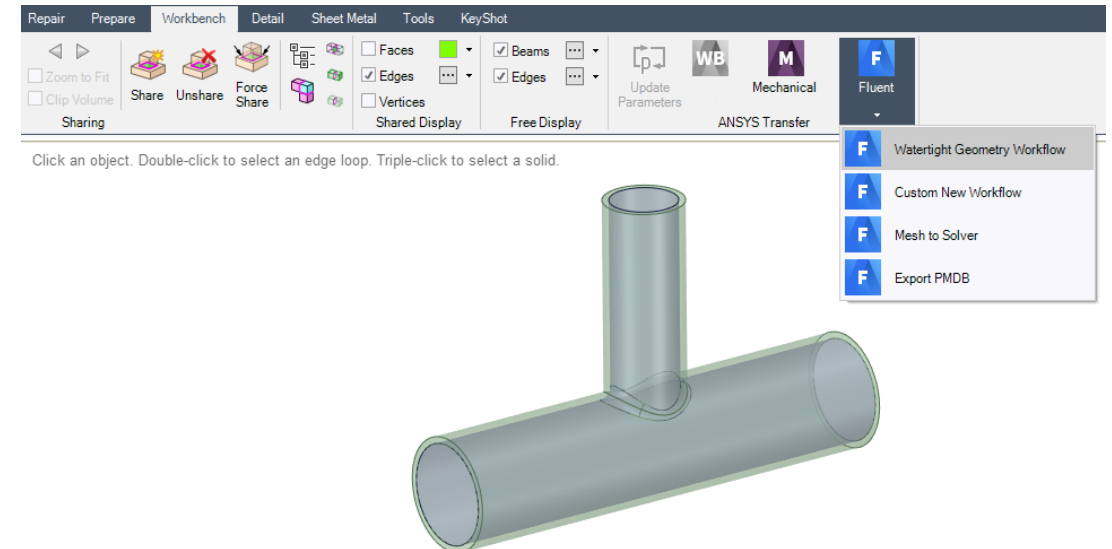
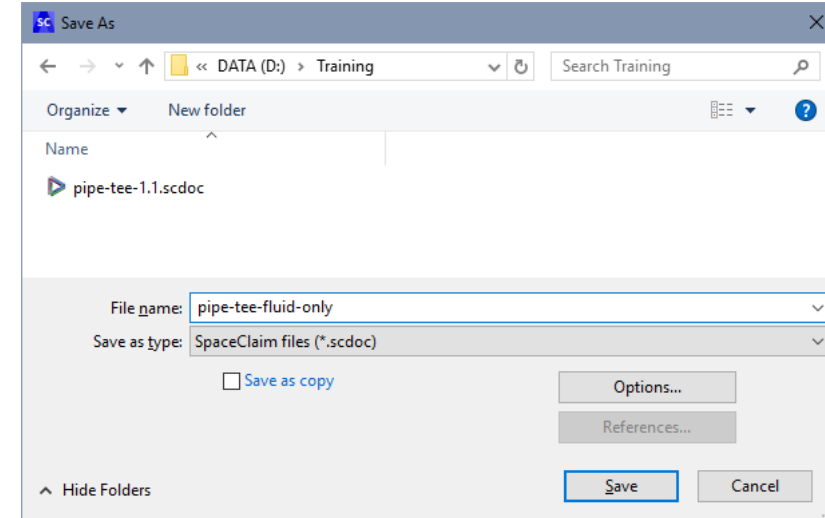


The symbol across the part icon indicates suppressed bodies.



/ Save Model With New Name and Exit

- Under File > Save As, save the model with a new name
- Note at any point, the model could be sent directly to Fluent from the Workbench tab
- Exit SpaceClaim after saving





End of presentation