3D Convection through an Electronics Box - Geometry

Author: Ben Mullen, Cornell University

Geometry

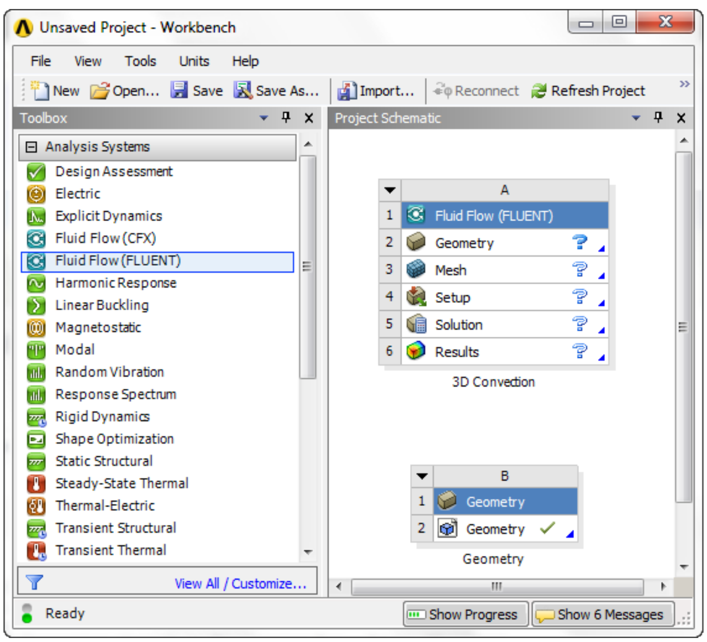
For users of ANSYS 15.0, please check this link for procedures for turning on the Auto Constraint feature before creating sketches in DesignModeler.

Download Geometry Files

Click here to download the geometry. Save the file somewhere convenient, such as your desktop.

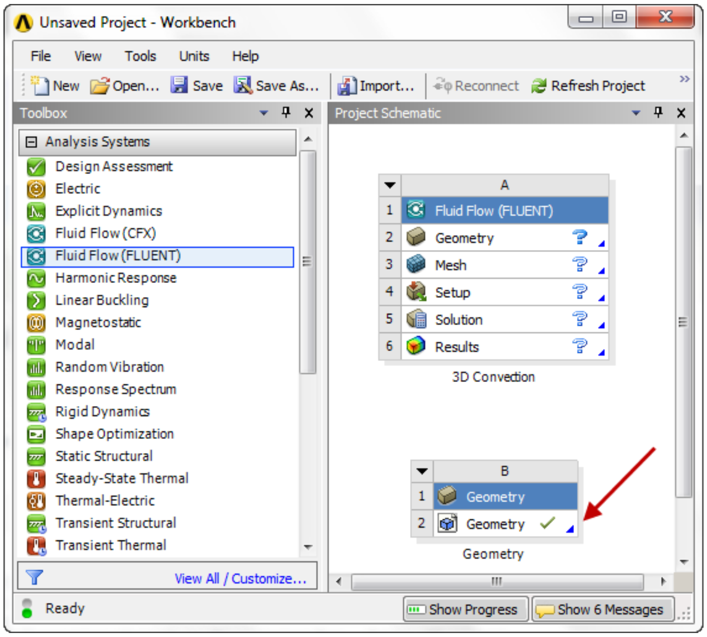
Load the Geometry into ANSYS

In ANSYS, go to **File > Import**. This will allow us to import the geometry from outside ANSYS into our workspace. First, make sure the file extension box is set to ***Geometry File*** . Next, browse to geometry file, select it, and press ***OK*** . The geometry file should now appear in the *Project Schematic* window.

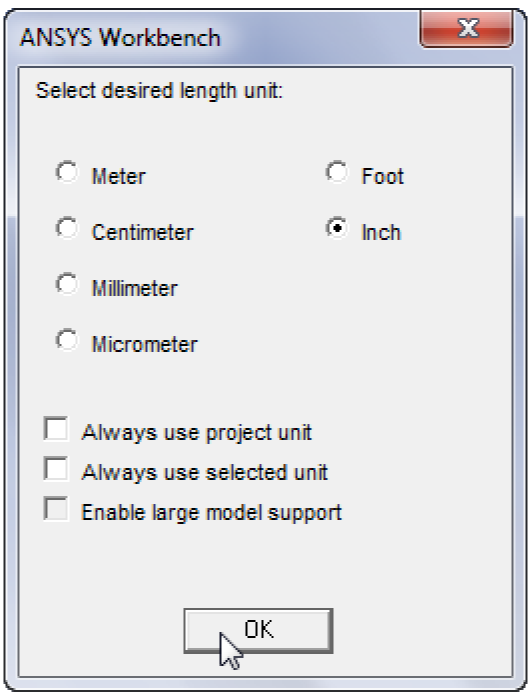


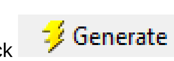
Open the Geometry

Next, open the geometry file by double clicking it in the *Project Schematic* window.

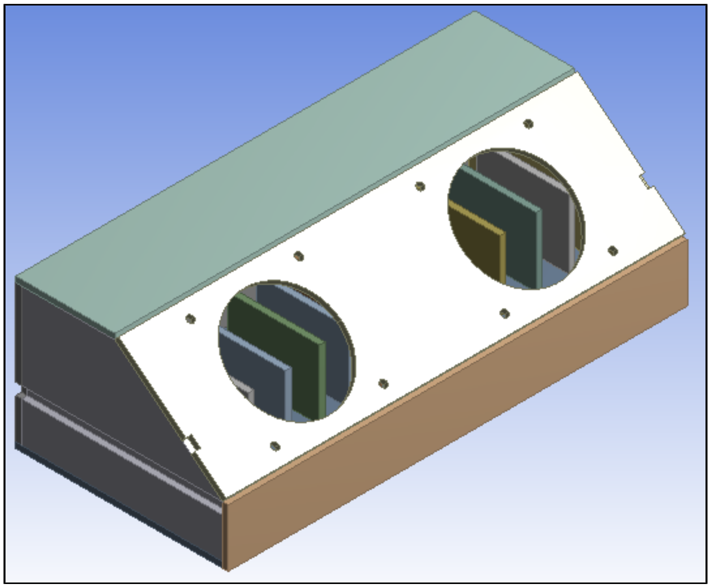


This will open the ***Design Modeler*** . When the design modeler loads, we will first need to specify a standard unit. Select ***Inch*** and press ***OK*** .



As you can probably tell, ANSYS does not generate the model automatically, so we need to generate the geometry. Click 

to generate the geometry.



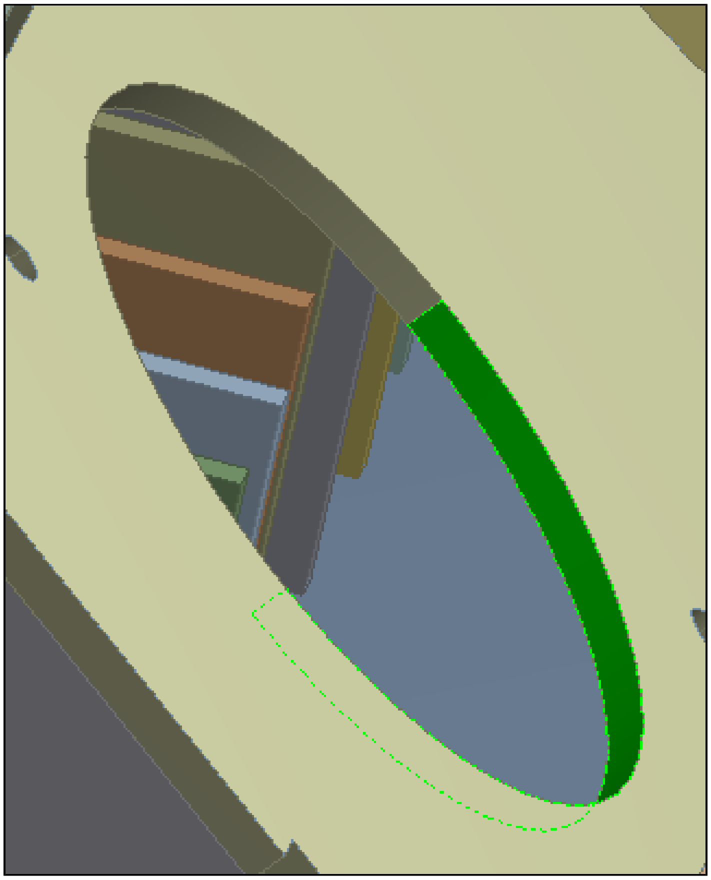
Create a Flow Geometry

Caps

The next step is for us to create a flow geometry. The air is going to flow through the inside of the box; however, we need to create a geometry for this empty space in order for ANSYS to be able to calculate the solution. We will accomplish this by creating ***Caps*** , then using those caps to create ***Fill*** . In the menu bar, select **Tools > Fill**. In the *Details* window, name the fill ***Caps*** , and make sure the extraction type is set to ***By Cavity*** .

Details View	
Details of Caps	
Fill	Caps
Extraction Type	By Cavity
Faces	0

We need to create caps for the two inlet holes, the two outlet holes, and all 16 connection holes. In order to create a cap, a cavity is identified by faces and ANSYS will fill the cavity with material. Most holes are identified by two faces. Notice in the figure below that in the large cavity, one section has been selection (green) and the other is not selected. In order to create a fill for the cavity, both surfaces need to be selected.



To select these surfaces, use the ***Face Selection Filter*** Hold down CTRL to select multiple surfaces at once. When all the surfaces have been selected, press ***Faces > Apply*** in the *Details* window. 40 surfaces should have been selected.

Fill

Next, we need to create the fill geometry. Once again, go to **Tools > Fill**. Name the fill, `FlowGeometry`. Also, we need to change the ***Extraction Type*** .

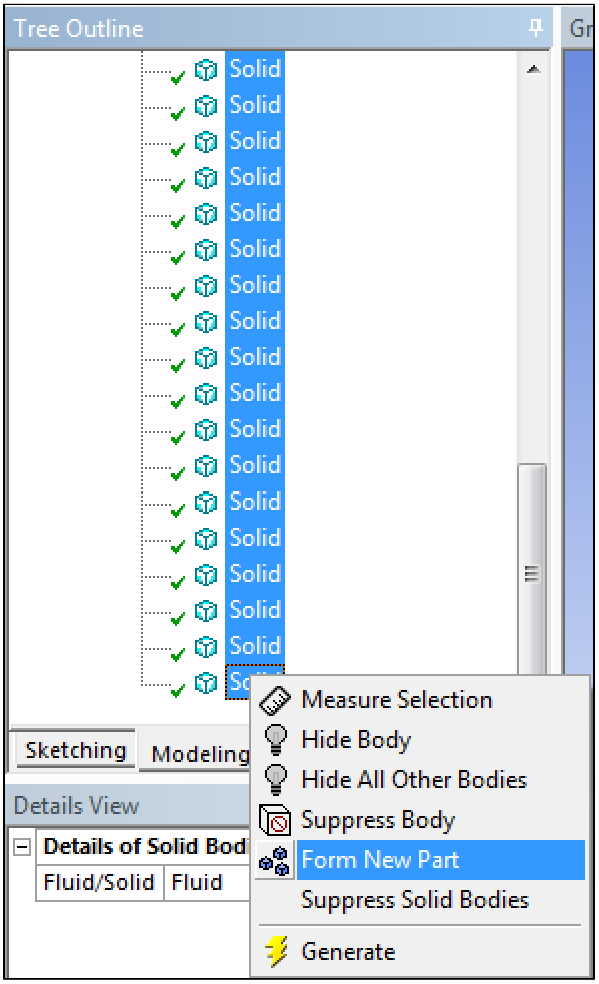
Select ***Extraction Type > By Caps*** . The other parameters should be left as their defaults. Press Generate to create the fill geometry.

Details View

Details of FlowGeometry	
Fill	FlowGeometry
Extraction Type	By Caps
Target Bodies	All Bodies
Preserve Capping Bodies	No
Preserve Solids	Yes

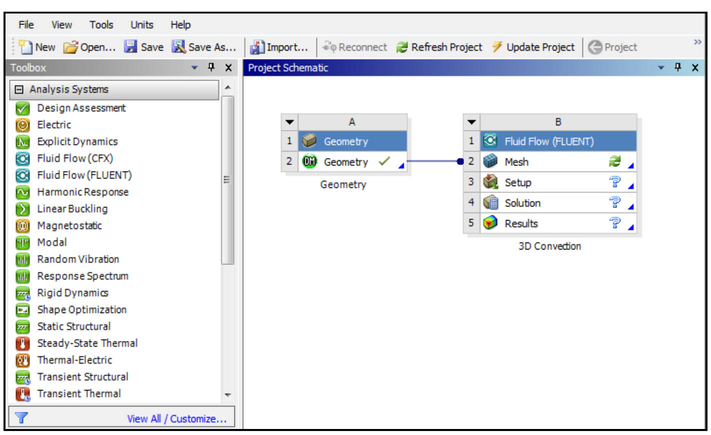
Form New Part

In the *Outline* window, select all of the parts in the tree, right click, and select ***Form New Part***



Connect Geometry to Project

Finally, we are ready to connect the geometry to the project. Close the design modeler, and return to the *Project Schematic*. Connect the geometry to the project by clicking and dragging the ***Geometry*** to ***Mesh***



Once the geometry has been connected, save the project.

Go to Step 3: Mesh

Go to all FLUENT Learning Modules