3D Convection through an Electronics Box - Geometry

Author: Ben Mullen, Cornell University Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Numerical Solution 6. Numerical Results 7. Verification & Validation Exercises Comments

Geometry

A 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.

🔥 Unsaved Project - Workbench							K
File View Tools Units Help							
📋 New 💕 Open 🛃 Save 🔣 Save	۸s	👔 Import		∉φ Reconnect 🙀	🖁 Refresh P	roject	>>
Toolbox 🔻 🕈	×	Project Sch	nema	tic		– Р	x
Analysis Systems							-
🔽 Design Assessment	1						
Electric		-		Α			
💹 Explicit Dynamics		1	C	Fluid Flow (FLUEN	п)		
🚱 Fluid Flow (CFX)		2	9	Geometry	?,		
S Fluid Flow (FLUENT)	Ξ	3		Mesh	2.		
Normal Marmonic Response		1		Cabus			
😥 Linear Buckling		7		Setup	<u> </u>		
🔘 Magnetostatic		5		Solution	2 a		н
😬 Modal		6	6	Results	? 🖌		
📶 Random Vibration				3D Convertion			
📶 Response Spectrum				50 001100001			
🛃 Rigid Dynamics							
Shape Optimization							
🤓 Static Structural			_	-		1	
📵 Steady-State Thermal			•	В			
Thermal-Electric			1	🥪 Geometry			-
main and the structural main and the structural main and the structural main and the structural structure			2	🞯 Geometry	1		
🔃 Transient Thermal	-			Geometry			
View All / Customize		•					•
Ready							

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 .

ANSYS Workbench		x		
Select desired length unit:				
	~			
C Meter	• Foot			
C Centimeter	Inch			
C Millimeter				
C Micrometer				
 Always use project unit Always use selected unit Enable large model support 				
СК				

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

 Generate to generate



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 📍						
Ξ	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 Select Caps to create the fill geometry.

De	tails View	4			
	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

Tree Outline			ņ	Gr
	olid		*	
	olid			
	blid			
	bild			
	DIIC			
	and	ſ		
	Jid		I.	
	Jid		=	
	lid		-	
	lid		I.	
				L
	\bigcirc	Measure Selection		
Sketching Modeling	₽ I	Hide Body		
<u></u>	P I	Hide All Other Bodies		
Details View	0	Suppress Body		
Details of Solid Bod	66	Form New Part		
Fluid/Solid Fluid		Suppress Solid Bodies	;	
	%	Generate		

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*

File View Tools Units Help		
🎦 New 💕 Open 层 Save 📓 Save As	👔 Import 🕹 Reconnect 🛿 Refresh Project 🍼 Update Project 🄇 Project	>>
Toolbox 💌 👎 🗙	Project Schematic	- 4 X
Analysis Systems		
Design Assessment		
() Electric	▼ A B	
Explicit Dynamics	1 🥪 Geometry 1 🕃 Fluid Flow (FLUENT)	
Fluid Flow (CFX)	2 🕅 Geometry 🖌 🚽 🕈 2 🚔 Mech 🏻 🥰	
Fluid Flow (FLUENT) =		
Marmonic Response	Geometry 3 🐨 Setup 💡 🖌	
Linear Buckling	4 Viii Solution	
0 Magnetostatic	5 😥 Results 🛛 😨 🖌	
Modal	3D Convection	
🔞 Random Vibration		
Response Spectrum		
Rigid Dynamics		
Shape Optimization		
5 Static Structural		
Steady-State Thermal		
Thermal-Electric		
rransient Structural		
🔃 Transient Thermal 👻		
View All / Customize		

Once the geometry has been connected, save the project.

Go to Step 3: Mesh

Go to all FLUENT Learning Modules