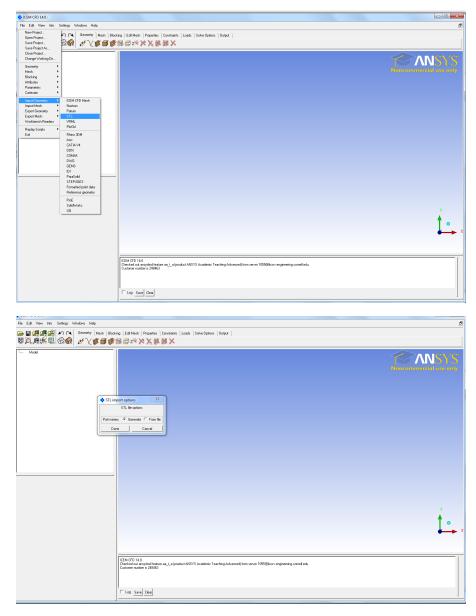
Nasal Airway Model - Mesh

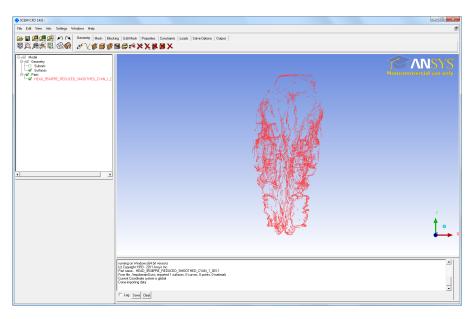
Author: Brandon Kovarovic, Cornell University Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Numerical Solution 6. Numerical Results 7. Verification & Validation Exercises Comments

Mesh

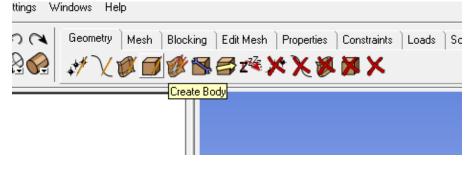
Open up ICEM CFD in the ANSYS 14 meshing folder. After it loads, select File -> Import Geometry -> STL. Locate the STL created in MIMICs. Select the Generate option in STL import options that appears.



Now that the STL is imported, rotate the model to confirm the geometry is correct from the wire frame.



Next a 'Body' has to be created. This designates where the volume cells will be and where the fluid will flow.



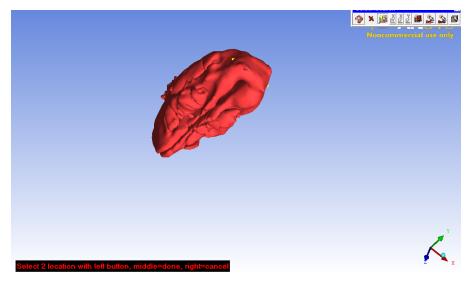
You'll need to turn on the body view seen in the image below to view the surface of the STL.



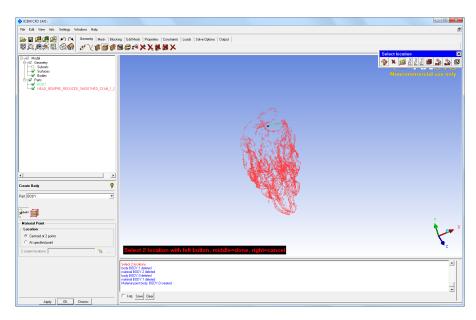
In the Create Body command, you'll need to select two point on the surface of the geometry where the centroid of those points is inside the cavity. The easiest place to do this is in the back of the nasal cavity after the passages have joined together.

Create Body		<i>?</i> ?	
Part BODY		•	
KatP1 Body			
- Material Point			
Location			
 Centroid of 2 points 			
O At specified point			
2 screen locations		⊗	
		Select loc	ation(s)
			running on Wir (c) Copyright 1 Part name H From file ./tmp Current Coordin Done importing
Apply OK	Dismiss		□ Log Sav

Select one point on the right side surface of the back cavity geometry and then rotate the model (Crtl + Left Mouse) and select another point on the left side.



Rotate the model and confrim that the Body point is inside the cavity geometry.



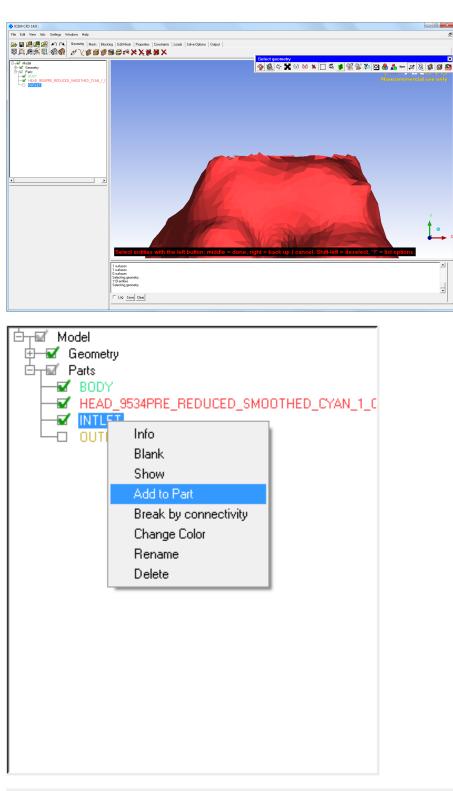
Now the next step is to set up the meshing of the geometry. This is done by setting a Global Element Size or a Surface Element size. Adjusting this properties will greatly change the final number of elements and the quality of the mesh. ICEM CFD has significant amount of options and settings to get a quality mesh at the walls and coarsening in the 'farfield' regions. In this tutorial we will choose an edge criteria in the terahedral elements of 0.2 and smooth the mesh to a quality of 0.4 with 5 iterations. This will give a coarse mesh for the purposes of the tutorial. Additionally prism elements and tetrahedral elements can be combined and after the creation of the global mesh refinement of the global and localy refinement/ coarsening can be performed based on the geometry complications.

After inputting the Global Mesh Settings, click Apply and the command window will inform you that the new parameters are set.

ICEM CFD 14.0 : project1		
File Edit View Info Settings Windows Help		
Ceonety Mesh		ing Edit Mesh Properties Constrain/ で
Model Geometry Geometry For 2007 For		
	Þ	
Global Mesh Setup	?	
₹ ₹		
Volume Meshing Parameters		
Mesh Type Tetra/Mixed		
Tetra/Mixed Meshing		
Mesh Method Robust (Octree)		
Run as batch process		
Fast transition		
Edge criterion 0.2		Done saving tetin file. Saved family boco data to project1.fbc
Define thin cuts		
		Saved attribute data to project1.atr Saving solver parameters to project1.par
Smooth mesh		Saved attribute data to project1.atr
Iterations 5 €		Saving solver parameters to project1.atr Saving solver parameters to project1.par Saving project settings to "H:/meshing s Project saved.
Smooth mesh	-	Saved attribute data to project1.atr Saving solver parameters to project1.par Saving project settings to "H:/meshing s

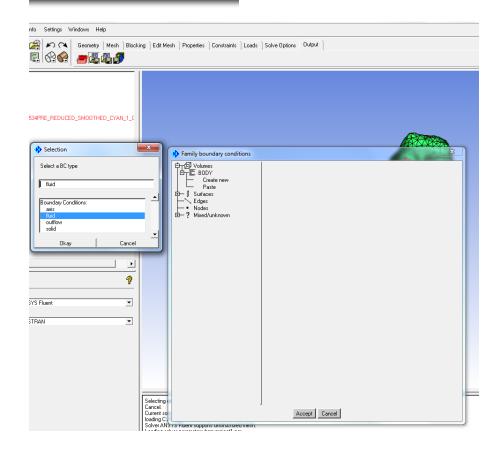
ICEM CFD 14.0 : project1			
	ocking Edit Mesh Properties Constraints Loads Solve Options Output		5
A M M R AND			
Select Select	Drane saving Mith Re. Seved ImmB book data to project I flos Saving software parameters to project 1 part Saving software parameters to project 1 part Saving software parameters to project 1 part Project saved.	1	
ComputeDismiss	Log Save Dee	<u> </u>	
Hodel	etry Create Part Create Assembly Show All Hide All Reverse Blank All Expand All Collapse All Blank Selected Restrict Selected Delete Empty Parts Expose Component Parts Edit Attributes Part Mesh Setup "Good" Colors Color By Material	DOTHED_CYAN_1_C	

Create Part		9	
Part outlet			
Create Part			
🚴 🌋 🌄			
Create Part by S	election		
Entities	× …		
🗖 Adjust Geometry	Names		
L			
Apply	OK Dismiss		
ICEM CFD 14.0 : project1 File Edit View Info Settings Windows Help			
Image: Construction of the second	enh Popentes Countraies Loads Salve Opriors Output Solect must referenceds	102 02 11 0 0 2 0 1	×
Draf Model Draf Genetaty Draf Meth Draf Boty WEAD_SSUPPE_RECUCED_SMOOTHED_CYAL_LC		No. 10 10 10 10 10 10 10 10 10 10 10 10 10	a commercial use only
Cicate Part			
Pat joster Create Pat			ž
Create Part by Selection Er/thes Adjust Georetry Names			
Sole Asset in	act elements with the left button; middle = done, right = back up / cancel, ?? which growty or mah wold report where down it where	= list options.	
	ster germany or neh nodel Man Assert Man Assert Sere Case∫ Σεν Case∫		I
Apply DK Dismiss			1





File Edit	View Info	Settings	Windows	Help		
				ry Mesh		ng È Edit Mesh È Propert
	sometry ash	RE_REDUC	ED_SMOOT	HED_CYAN	_1_0	
 iolver Setu	ıp				•	
	ver ANSYS I				•	
Set As D						
						Selecting elements: elem 26 elements Done Selecting mesh. Selecting elements: elem Cancel.
						Log Save Clear



Family boundary conditions	X
Volumes BDDY Create new Paste Boundary Conditions fluid BACK Create new Paste Boundary Conditions wall Create new Paste Boundary Conditions velocity-inlet Create new Paste Boundary Conditions velocity-inlet Boundary Conditions	wall Delete Copy
Geometry Mesh Blocking	〕Edit Mesh
)_SMOOTHED_CYAN_1_C	Save Save Save Current project first?

Go to Step 4: Physics Setup

Go to all FLUENT Learning Modules