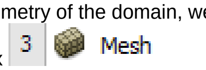
Supersonic Flow Over a Wedge - Mesh

Author: Rajesh Bhaskaran, Cornell University

Mesh

Launch the Mesher

Now that we have completed creating the geometry of the domain, we are ready to mesh it. Return to the *Project Schematic* Window. In the *Project Schematic* window, double-click the Mesh box 

to launch the mesher.

Note: If you created the geometry in SpaceClaim, you will need to perform an additional step here. Once the mesher is open, look in the tree outline and highlight Geometry > FFF\Surface, which will be your domain. Under *Details,* find *Material* > *Fluid/Solid* and set it to *Fluid.*

Mapped Face Meshing

First, we will apply a mapped face meshing (in later versions this is simply called face meshing instead), which will give us a regular mesh. First, in the *Outline* window, click 

to show the Mesh menu in the menu bar. In the Mesh Menu, select **Mesh Control > Face Meshing**. In the *Graphics* window, hold down CRTL, and select *both* domain faces to select it, then in the *Details* window, click ***Geometry > Apply***. (Note: If you created the geometry in SpaceClaim, you will only have one domain face, which is okay. The reason we split the domain in older versions of ANSYS was because the mesher had trouble giving a high quality mesh if we didn't.)

Body Sizing

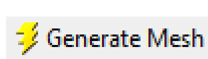
Next, we will create a body sizing for the elements that will make up the domain. In the Mesh Menu, select **Mesh Control > Sizing**. Next, select the body selection filter in the menu bar:

Next, select the surface in the graphics window. In the *Details* window, select ***Geometry > Apply***. Now, we want to change the element size. In the *Details Window*, select ***Element Size > Default*** and change the value to `0.05 m`.

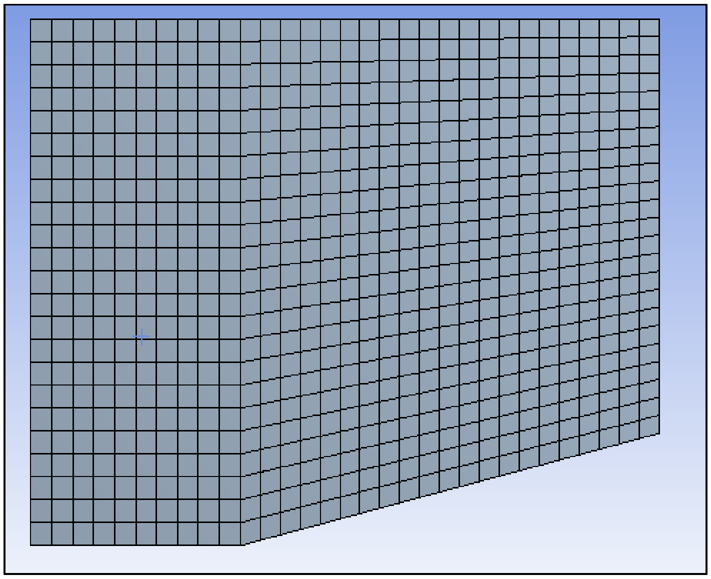
Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	5.e-002 m
Behavior	Soft
Curvature Normal Angle	Default
Growth Rate	Default

Generate the Mesh

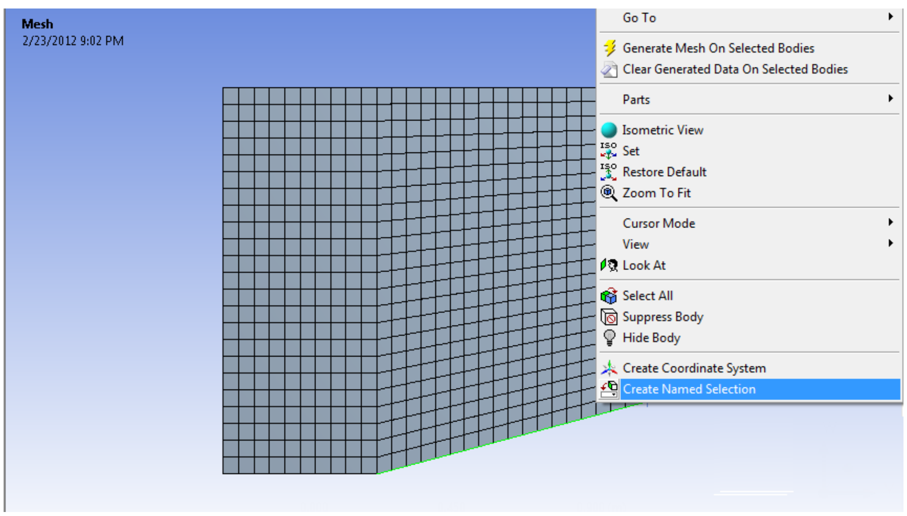
Now, we are ready to generate the mesh. Generate the mesh by clicking 

in the menu bar or by going to **Mesh > Generate Mesh**. The final mesh should resemble the one in the figure below.

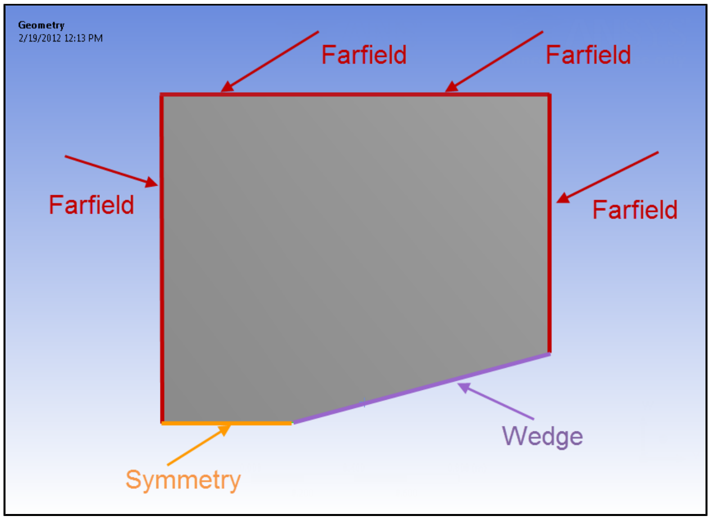


Named Selections

Now, we need to create named selections to use when we set boundary conditions. To create a named selection, first ensure that the edge selection filter  is selected. Next, left-click on the desired edge you wish to name (multiple edges can be selected while holding down CTRL), then right click on the edge and select ***Create Named Selection***.



Once you select ***Create Named Selection***, a dialogue box will appear where you will enter the desired name of the boundary. Use the diagram below to name all of the boundaries of the geometry.



There are 4 edges that make up the farfield, and they can all be named at once by holding down CTRL, left-clicking all of the edges while holding down CTRL, then right-clicking and selecting "Create Named Selection."

Once the selections are all named and the mesh is created, you may save the project and close the mesher.

Go to Step 4: Physics Setup

Go to all FLUENT Learning Modules