

MAE 157A: ANSYS Fluent Tutorial

I. SolidWorks

a. Creating computational domain for Fluent (make sure your dimensions are set to inches)

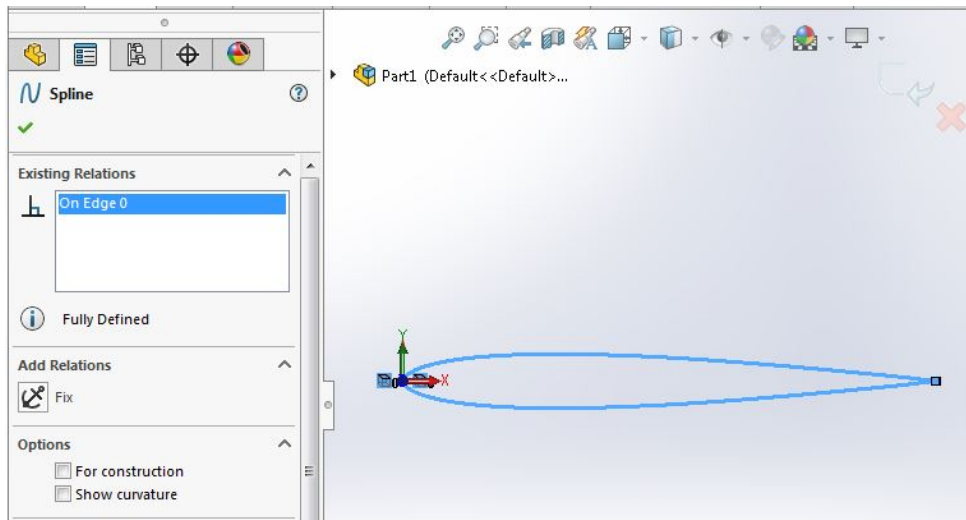
i. Import profile

1. For airfoils

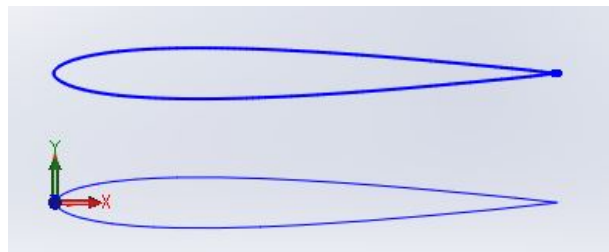
- Open a new part document
- At the top menu, go to *Insert* → *Curve* → *Curve Through XYZ Points...*
- Select *Browse* and open the .txt file of the desired airfoil
- Click *OK*
- Create a sketch on the *Front* plane
- Click on the airfoil profile (shown in blue) and select *Convert Entities*. You should now have something similar to the following



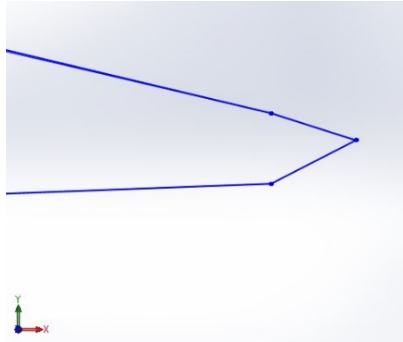
g. Click on the airfoil profile and delete the *On Edge* relationship



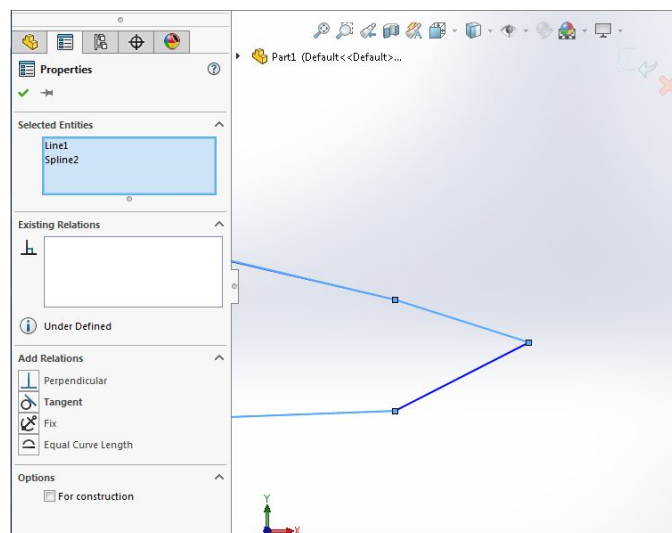
h. You should now be able to move the airfoil (you will be working with the profile that moves)



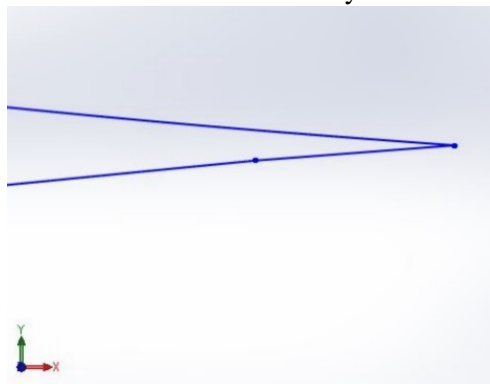
- i. Now zoom in to the trailing edge of the airfoil
 - i. **If the airfoil has 2 points that are almost vertical**
 1. Draw 2 lines to connect the profile



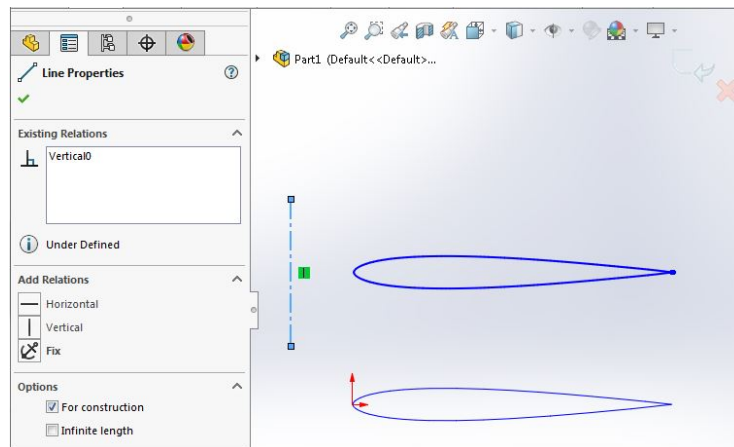
2. Now click one of the lines, hold the *Ctrl* key, and click the airfoil profile. Add a *Tangent* relation



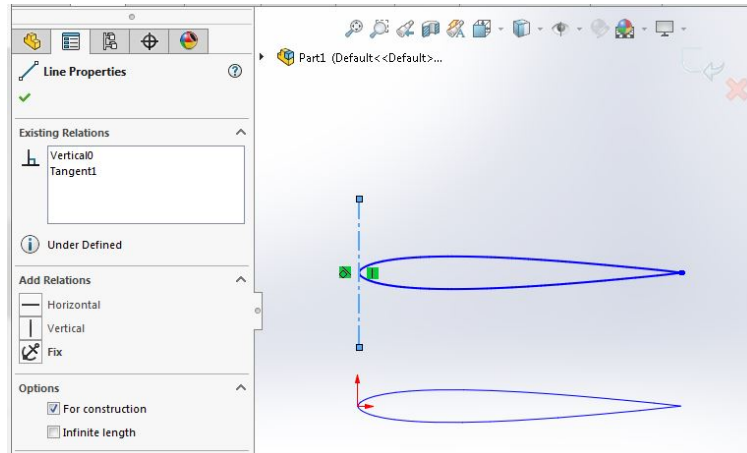
3. Repeat this for the other line
- ii. **If the airfoil has 2 points that are offset**
 1. Connect the profile using 1 line
 2. **Do not** add any relationships



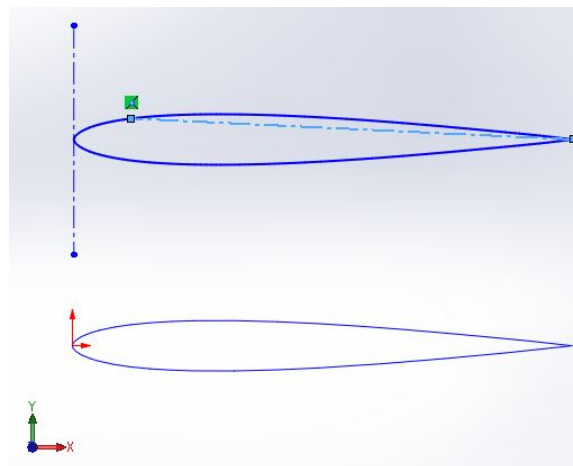
- j. Now zoom out and create a vertical line **for construction**



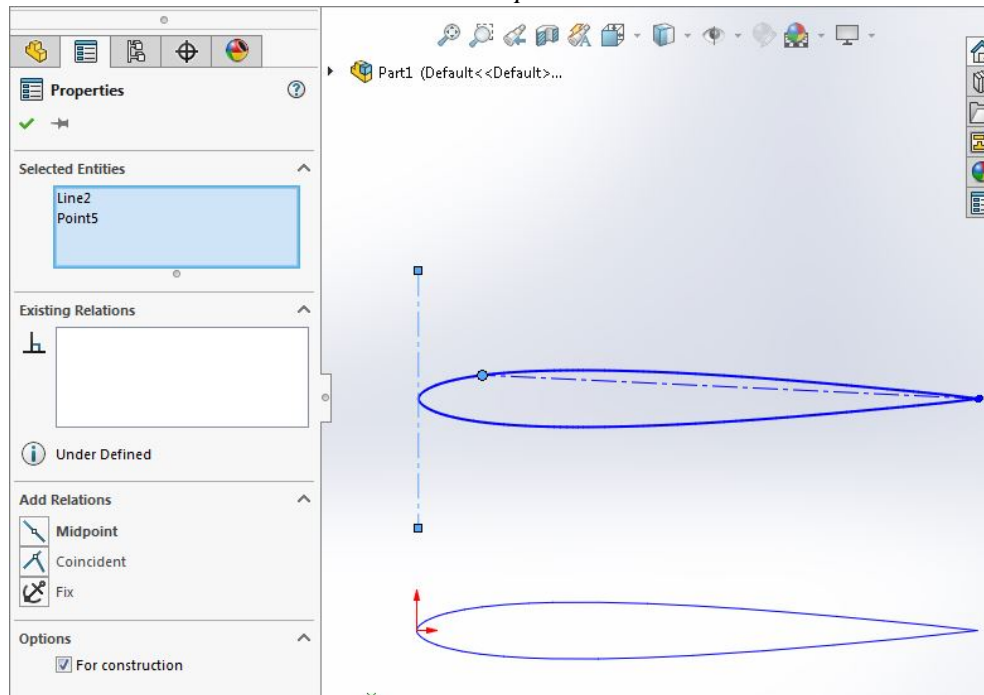
- k. Click the vertical line, hold the Ctrl key, and click the airfoil profile. Add a *Tangent* relationship



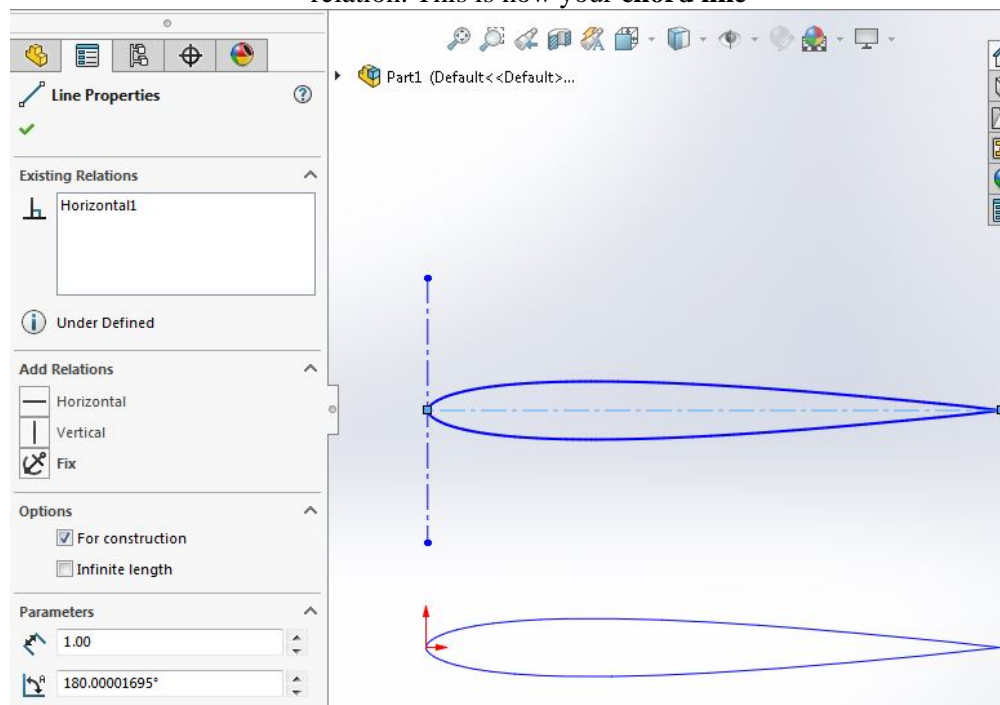
- l. Zoom into the trailing edge of the airfoil and create a **construction line** from the **trailing edge point** to anywhere else on the airfoil



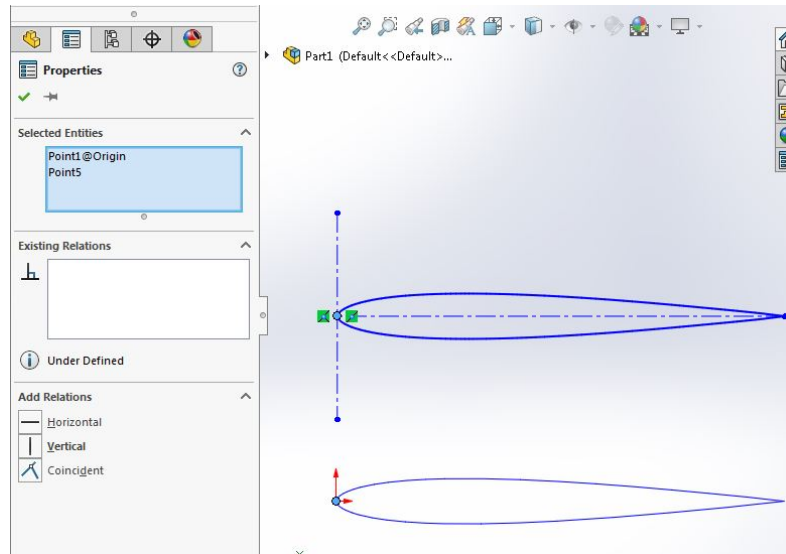
- m. From the line you just created in step (k), click on the left most point, hold the *Ctrl* key, and click on the vertical construction line. Add a *Midpoint* relation



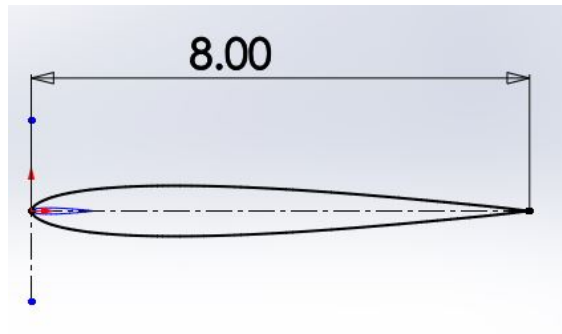
- n. Click on the line you created in step (k) and add a *Horizontal* relation. This is now your **chord line**



- o. Select the left most point of the horizontal line, hold the *Ctrl* key, and select the Origin (in red). Add a *Coincident* relation

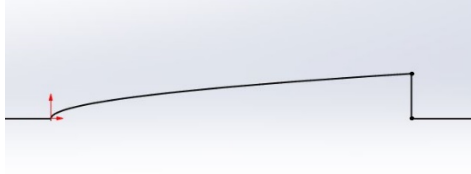


- p. Now select Smart Dimension and click on the horizontal construction line. Set the chord to 8". Your airfoil is now fully defined.



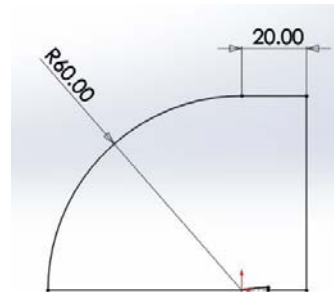
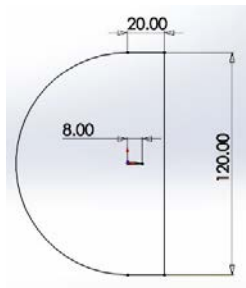
2. **For nose cones**, use *Equation Driven Curve*

- Open a new part document
- Create a sketch
- In the *Spline* drop-down menu, select *Equation Driven Curve*
- Enter the profile equation in y_x
- Set $x_1 = 0$, $x_2 = 8$ and click the green check mark. This is now your nose cone profile

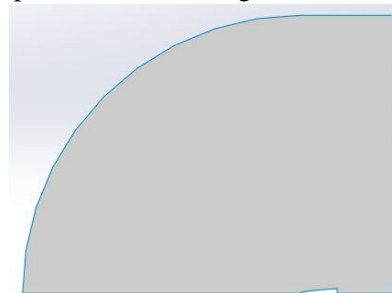
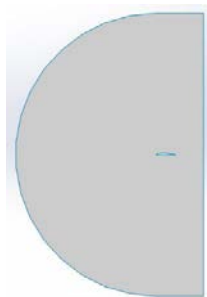


ii. Establish the computational domain as follows:

- In the same sketch you've been using, create the domain using lines and arcs
- Note: the center of the semicircle is located at the origin**
- Note: for nose cones, you only need half of the domain since the profile is axisymmetric**



iii. Once your sketch is fully defined, exit the sketch and create a surface using *Planar Surface*. You should end up with the following:



iv. Save the document as an *.IGS* file to import into Fluent

II. ANSYS Fluent

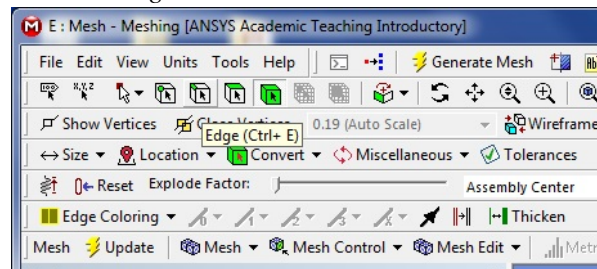
a. Open ANSYS Workbench 17.2

b. Importing Geometry

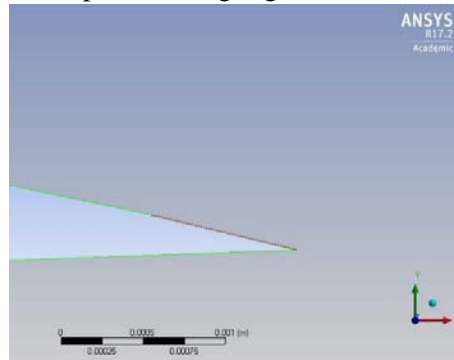
- i. From the *Component Systems* list, click on *Geometry* and drag it to your *Project Schematic*
- ii. Within *Project Schematic*, right click on *Geometry* → *Import Geometry* → *Browse* → select your *.IGS* file
- iii. Double click *Geometry* to open the *Design Modeler*
- iv. Once the *Design Modeler* is open, click *Generate*
- v. Your surface should now appear in the graphics area

vi. For Airfoils

1. Zoom in to view the trailing edge of the airfoil
2. Select the *Edge* selection filter



3. Holding the *Ctrl* key, select all the edges of your airfoil profile until the entire profile is highlighted

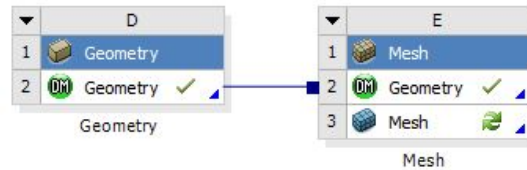


4. At the top, go to *Tools* and select *Merge*
5. Select *Apply*
6. Select *Generate*, and now your airfoil profile should consist of only one edge
7. Close the *Design Modeler*

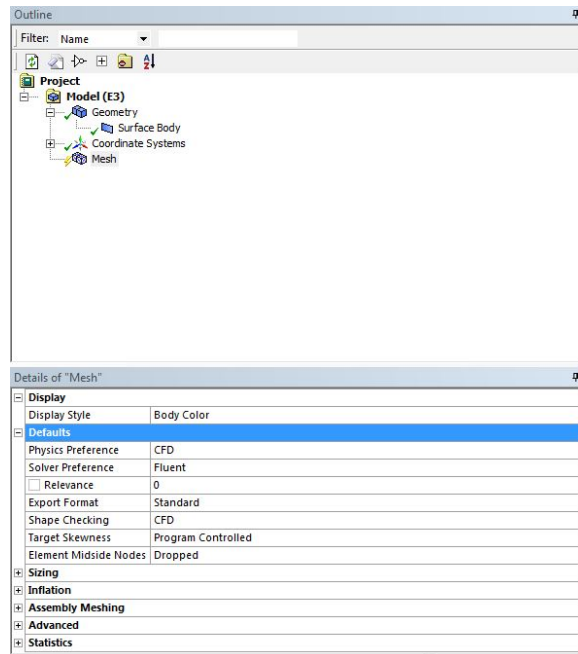
vii. For Nose Cones

1. Close the *Design Modeler*

- c. Meshing (make sure your dimensions in the Meshing program are set to inches)
 - i. From the *Component Systems* list, click on *Mesh* and drag it on top of (overlap) *Geometry* in your *Project Schematic*

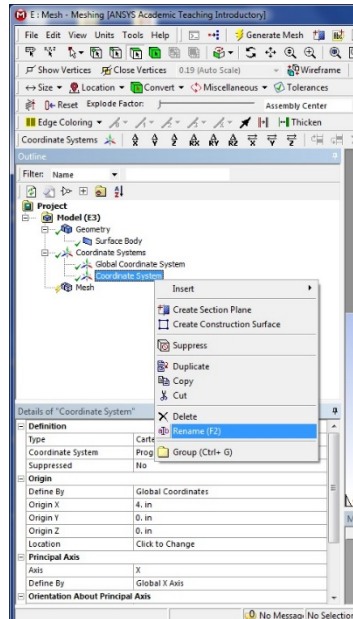


- ii. Double click *Mesh* to open the meshing program
- iii. Once the meshing program is open, click on *Mesh* under your project *Outline*

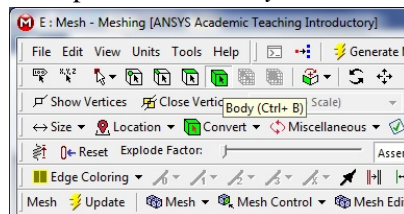


1. Under *Defaults*
 - a. Change *Physics Preference* to *CFD*
2. Under *Sizing*
 - a. Change *Size Function* to *Proximity and Curvature*
 - b. Change *Relevance Center* to *Fine*
 - c. Change *Smoothing* to *High*
 - d. Change *Max Face Size* to *2in*
- iv. Now under your project *Outline*, right click on *Coordinate Systems* → *Insert* → *Coordinate System*
 1. Under *Origin*
 - a. Change *Define By* to *Global Coordinates*
 - b. Change *Origin X* to *8 in.*
- v. Under your project *Outline*, right click on the *Coordinate System* you just created and select *Rename*

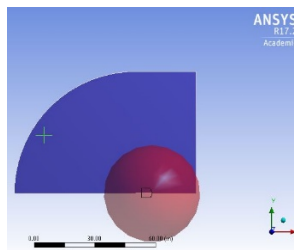
1. Rename the coordinate system to *Airfoil/Nose Cone End*



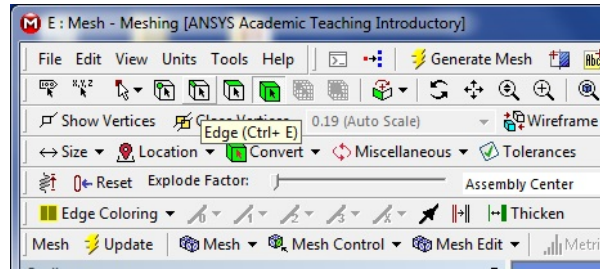
- vi. You should now have a new coordinate system located at the end of your airfoil/nose cone
- vii. Right click on *Mesh* under your project *Outline* and select *Insert* → *Sizing*
 1. At the top, select the *Body* selection filter



2. Click the airfoil/nose cone surface in your graphics window
3. Under *Scope*
 - a. Click geometry *Apply* to the geometry
4. Under *Definition*
 - a. Change *Type* to *Sphere of Influence*
 - b. Change *Sphere Center* to *Airfoil/Nose Cone End*
 - c. Change *Sphere Radius* to *16in*
 - d. Change *Element Size* to *0.25in*
5. You should now see the sphere of influence created in your graphics window



- viii. Right click on *Mesh* under your project *Outline* and select *Insert* → *Sizing*
 1. At the top, select the *Edge* selection filter



2. For airfoils:

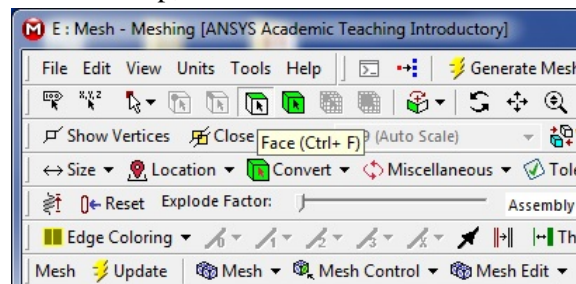
- a. Select the edge of the airfoil profile in your graphics window
- b. Under *Scope*
 - i. Click geometry *Apply* to the geometry
- c. Under *Definition*
 - i. Change *Type* to *Number of Divisions*
 - ii. Change *Number of Divisions* to 3000
- d. Under *Advanced*
 - i. Change *Behavior* to *Hard*

3. For nose cones:

- a. Select ONLY the parabolic/power/linear edge of the nose cone profile in your graphics window
- b. Under *Scope*
 - i. Click geometry *Apply* to the geometry
- c. Under *Definition*
 - i. Change *Type* to *Number of Divisions*
 - ii. Change *Number of Divisions* to 1200
- d. Under *Advanced*
 - i. Change *Behavior* to *Hard*
- e. Now REPEAT steps a-d, for the vertical edge of the nose cone, setting the *Number of Divisions* to 150

- ix. Right click on *Mesh* under your project *Outline* and select *Insert* → *Inflation*

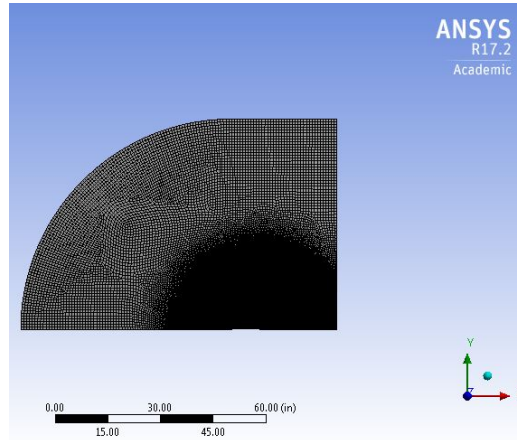
1. At the top, select the *Face* selection filter



2. Click the airfoil/nose cone surface in your graphics window
3. Under *Scope*
 - a. Click geometry and *Apply* to the geometry
4. At the top, select the *Edge* selection filter

5. For airfoils:

- a. Click on the edge of your airfoil profile
6. **For nose cones:**
 - a. Click on the power/linear edge of the nose cone
7. Click *Boundary* and *Apply* to the geometry
8. Under *Definition*
 - a. Change *Inflation Option* to *Total Thickness*
 - b. Change *Number of Layers* to 5
 - c. Change *Maximum Thickness* to 0.08in
- x. Right click on *Mesh* under your project *Outline* and select *Generate Mesh*. Wait a few minutes and you should end up with a mesh similar to this:



xi. For an airfoil:

1. At the top, select the *Edge* selection filter
2. While holding the *Ctrl* key, select the top edge, bottom edge, and radius of your domain (3 edges should be selected)
3. Right click and select *Create Named Selection*
4. Name the selected edges, *Inlet*
5. Now select the vertical edge of your domain
6. Right click and select *Create Named Selection*
7. Name the selected edges, *Outlet*
8. Now select the airfoil profile/edge of your model
9. Right click and select *Create Named Selection*
10. Name the selected edges, *Body*

xii. For a nose cone:

1. At the top, select the *Edge* selection filter
2. While holding the *Ctrl* key, select the top edge and radius of your domain (2 edges should be selected)
3. Right click and select *Create Named Selection*
4. Name the selected edges, *Inlet*
5. Now select the vertical edge of your domain
6. Right click and select *Create Named Selection*
7. Name the selected edges, *Outlet*
8. Now select the two symmetrical/bottom edges of your domain
9. Right click and select *Create Named Selection*
10. Name the selected edges, *Axis*
11. Now select all the nose cone profile/edges of your model

12. Right click and select *Create Named Selection*
13. Name the selected edges, *Body*
- xiii. Click *File* → *Save Project*
- xiv. You may now exit the Meshing software

d. Fluent

- i. From the *Component Systems* list, click on *Fluent* and drag it on top of *Mesh* in your *Project Schematic*



- ii. Right click on *Mesh* and select *Update*
- iii. Double-click on *Setup* under *Fluent*
 1. Under *Options*, make sure *Double Precision* is checked and press *Ok*
- iv. The Fluent software should automatically open up
- v. Wait for your model/mesh to load properly in the graphics window
- vi. Under *General*
 1. Change *Type* to *Density-Based*
 2. **For Airfoils:**
 - a. Change *2D Space* to *Planar*
 3. **For Nose Cones:**
 - a. Change *2D Space* to *Axisymmetric*
- vii. Under *Models*
 1. Double click on *Energy*
 - a. Make sure it is checked and press *Ok*
 - b. *Energy* should now say *On* in the *Models* list
 2. Double click on *Viscous*
 - a. Make sure *Spalart-Allmaras* is checked and press *Ok*
 - b. *Viscous* should now say *Spalart-Allmaras* in the *Models* list
- viii. Under *Materials*
 1. Double click on *Air*
 - a. Click on *Change/Create* and then click *Close*
- ix. Under *Boundary Conditions*
 1. Select *Inlet*
 - a. Under *Type* select *Velocity-Inlet* and click *Edit*
 - b. Change *Velocity Specification Method* to *Magnitude and Direction*
 - c. Change *Reference Frame* to *Absolute*
 - d. Set *Velocity Magnitude (m/s)* to 20
 - e. *X-Component of Flow Direction* = $\cos(\text{AoA}^\circ)$
 - f. *Y-Component of Flow Direction* = $\sin(\text{AoA}^\circ)$
 - g. Under *Thermal*, enter 300 for *Temperature (K)*
 - h. Press *Ok*

2. Select *Outlet*
 - a. Under *Type* select *Pressure-Outlet* and click *Edit*
 - b. Change *Backflow Reference Frame* to *Absolute*
 - c. Set *Gauge Pressure (Pascal)* to *0*
 - d. Under *Thermal*, enter *300* for *Backflow Total Temperature (K)*
 - e. Press *Ok*
3. Select *Body*
 - a. Under *Type* select *Wall*
4. **For Nose Cones:**
 - a. Select *Axis*
 - i. Under *Type* select *Axis*
- x. Under *Reference Values*
 1. Change *Compute From* to *Inlet*
 2. Ensure that reference values match your inlet parameters
3. **For Airfoils:**
 - a. Enter appropriate values for length, depth, and area (chord, span, and chord*span, respectively)
4. **For Nose Cones:**
 - a. Enter appropriate values for length and area (diameter and circular area of base, respectively)
- xi. Under *Solution Methods*
 1. Change *Formulation* to *Implicit*
 2. Change *Flux Type* to *Roe-FDS*
 3. Change *Gradient* to *Least Squares Cell Based*
 4. Change *Flow* to *Second Order Upwind*
 5. Change *Modified Turbulent Viscosity* to *Second Order Upwind*
- xii. Under *Monitors*
 1. Click *Residuals* and click *Edit*
 - a. In *Equations*, enter *0.01* for all *Absolute Criteria*
 - b. Click *Ok*
 2. Click *Create* and click *Drag...*
 - a. Under *Options* check *Print to Console* and *Plot*
 - b. Under *Wall Zones* click *Body*
 - c. Under *Force Vector*, enter:
 - i. $X\text{-Component} = \cos(\text{AoA}^\circ)$
 - ii. $Y\text{-Component} = \sin(\text{AoA}^\circ)$
 - d. Click *Save Output Parameter...* and name it *Cd-Case#*
 3. Click *Create* and click *Lift...*
 - a. Under *Options* check *Print to Console* and *Plot*
 - b. Under *Wall Zones* click *Body*
 - c. Under *Force Vector*, enter:
 - i. $X\text{-Component} = -\sin(\text{AoA}^\circ)$
 - ii. $Y\text{-Component} = \cos(\text{AoA}^\circ)$
 - d. Click *Save Output Parameter...* and name it *Cl-Case#*
- xiii. Under *Solution Initialization*
 1. Under *Initialization Methods* select *Standard Initialization*
 2. Change *Compute from* to *Inlet*

3. *Reference Frame* will be set to *Relative to Cell Zone*
4. Make sure your *Initial Values* match your *Inlet* conditions
5. Click *Initialize*
- xiv. Under *Run Calculation*
 1. Enter 2000 for *Number of Iterations*
 2. Click *Calculate*
 3. ANSYS Fluent is now iterating to converge on a solution. As it converges, you should be able to view the residuals in the graphics area. It may take up to 5 min to converge
- xv. Once a solution is found, you may plot and view results
- xvi. Under *Results*, select the *Graphics* tab
 1. Double-click *Contour* to create contour plots
 - a. Under *Options* select:
 - i. *Filled*
 - ii. *Node Values*
 - iii. *Global Range*
 - iv. *Auto Range*
 - b. Change *Contours of* to desired parameters
 - c. Click *Display*
 - d. Ensure that nothing is selected under *Surfaces* or *Surface Types*
 - e. You can now see the desired contour plots in the graphics area
- xvii. Under *Results*, select the *Plots* tab
 1. Double-click *XY Plot* to create plots
 - a. Under *Options* select:
 - i. *Node Values*
 - ii. *Position on X Axis*
 - b. Change *Plot Direction* to *X: 1 Y:0*
 - c. Change *Y Axis Function* to desired parameters
 - d. Change *X Axis Function* to *Direction Vector*
 - e. Click *Body* under *Surfaces*
 - f. Click *Plot*
- xviii. Under *Results*, select the *Reports* tab
 1. Double-click *Forces* to view forces
 - a. Under *Options* select:
 - i. *Forces*
 - b. Change *Direction Vector* to the following for drag:
 - i. $X\text{-Component} = \cos(AoA^\circ)$
 - ii. $Y\text{-Component} = \sin(AoA^\circ)$
 - c. Change *Direction Vector* to the following for lift:
 - i. $X\text{-Component} = -\sin(AoA^\circ)$
 - ii. $Y\text{-Component} = \cos(AoA^\circ)$
 - d. Click *Print*
- xix. Under *Parameters*
 1. Click *Output Parameters*
 - a. Right click *Cd-Case#* and click *Print to Console*
 - b. Right click *Cl-Case#* and click *Print to Console*