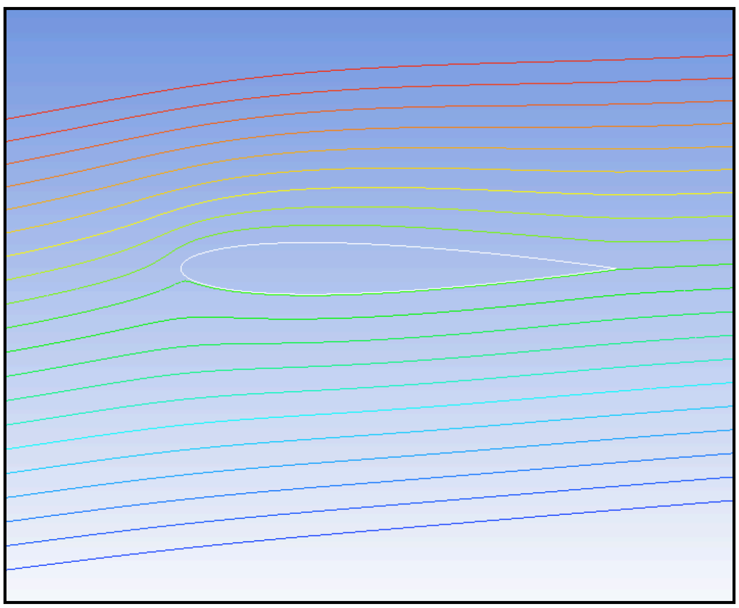
Fluent 18.2 Tutorial

Case Study: Flow Over an Airfoil



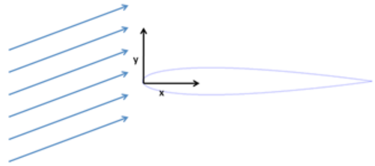
Presented by Aerodynamics Laboratory, Department of Mechanical Engineering, CCNY

The City College of New York

Table of Contents

NOTE: From page 3 onward, the bullet points are the step-by-step instructions and the paragraphs are explanations and additional information.

Fluent 18.2 Flow Over an Airfoil



Problem Specification

This case simulates a fluid flow over a NACA 0012 Airfoil at a 6-degree angle of attack. In this tutorial, a 2-D cross section of the airfoil will be used to analyze the fluid flow.

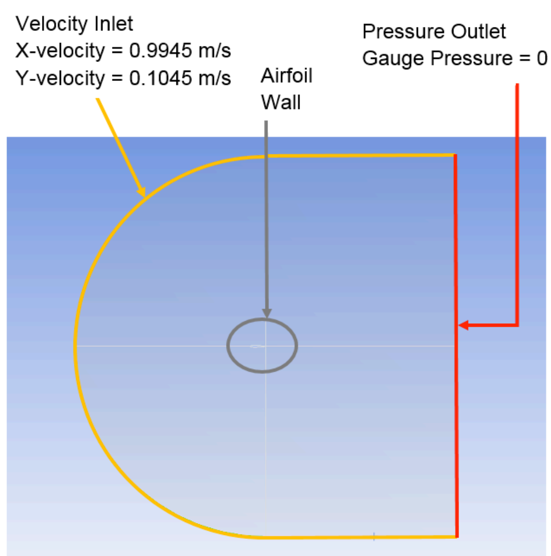
The airfoil coordinates are provided, with the distance from the leading edge (front of airfoil) to trailing edge (back of airfoil) spanning 1 m. The density of the fluid will be 1 kg/m^3 in order to simplify the computation. The air will be assumed to have no viscosity.

Solution Domain

Since the airfoil is rounded towards the leading edge and tapered straight towards the trailing edge, the outer boundary will be modeled using an arc in the front and a rectangle in the back. In order to minimize the effects of flow at the boundaries disturbing flow at the airfoil, the radius of the arc and width of the rectangle will be set to 12.5 times the length of the airfoil, or 12.5 m.

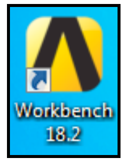
Boundary Conditions

In order to model fluid flow from the left to the right, at an angle of 6 degrees, boundary conditions must be specified. The boundary that covers the entire airfoil will be the velocity inlet, where the velocity will be 1 m/s at 6 degrees. The right side of the rectangle will be the pressure outlet, where the gauge pressure will be 0 Pa. Finally, the airfoil will be a wall, with a no-slip boundary condition.



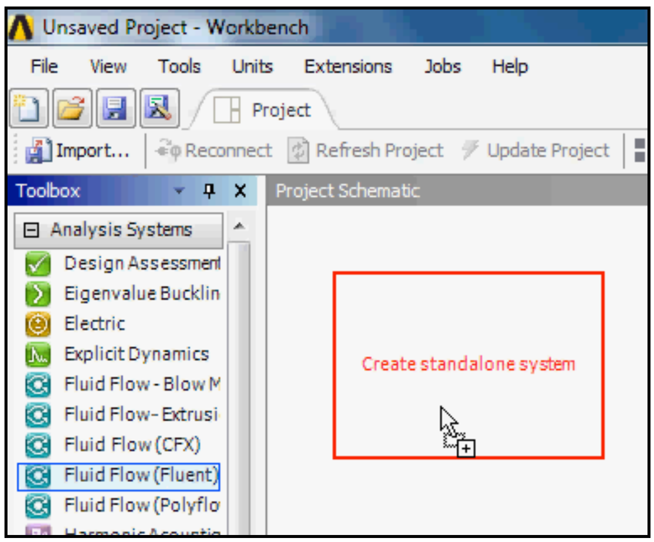
1. Start-Up

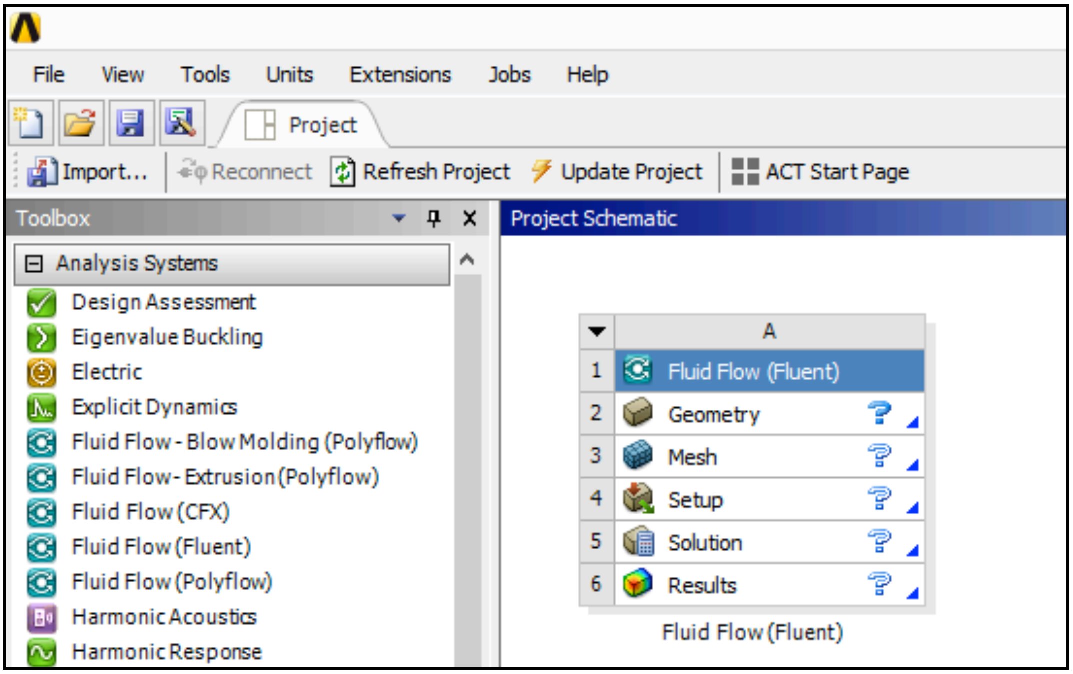
- Open ***Ansys WorkBench 18.2***



On the left hand is a ToolBox with different project options, including Fluid Flow (Fluent). To the right of the toolbox is the Project Schematic, where the current project progress is displayed.

- Drag ***Fluid Flow (Fluent)*** into the *Project Schematic* window





The Fluent project now appears in the *Project Schematic*.

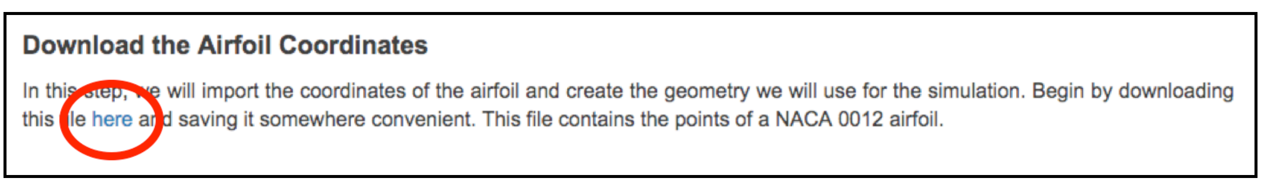
The project name can be changed by clicking below the box, where the default name is "Fluid Flow (Fluent)".

The steps 2 – 6 represent the progress of the project. The question marks to the left of each step indicate that the steps have not been worked on yet.

Downloading the Airfoil Coordinates

Before starting, download the airfoil coordinates from the Cornell University Flow Over an Airfoil tutorial in the following link:

https://confluence.cornell.edu/display/SIMULATION/Flow+over+an+Airfoil+-+Geometry



Download the Airfoil Coordinates

In this step, we will import the coordinates of the airfoil and create the geometry we will use for the simulation. Begin by downloading this file here and saving it somewhere convenient. This file contains the points of a NACA 0012 airfoil.

- Right click ***here*** > ***Save Link/Target as…*** to save the file

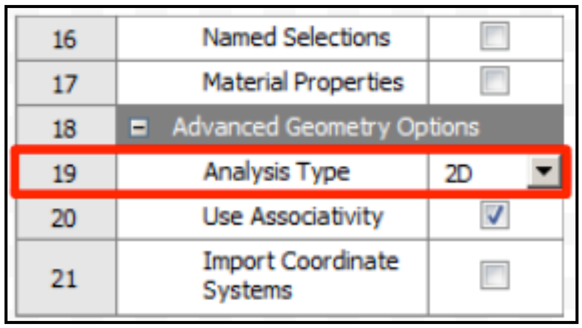
The file contains a text document with a list of all of the coordinates used to create an airfoil shape.

2. Geometry

In Geometry, the shapes and dimensions of the case will be modeled.

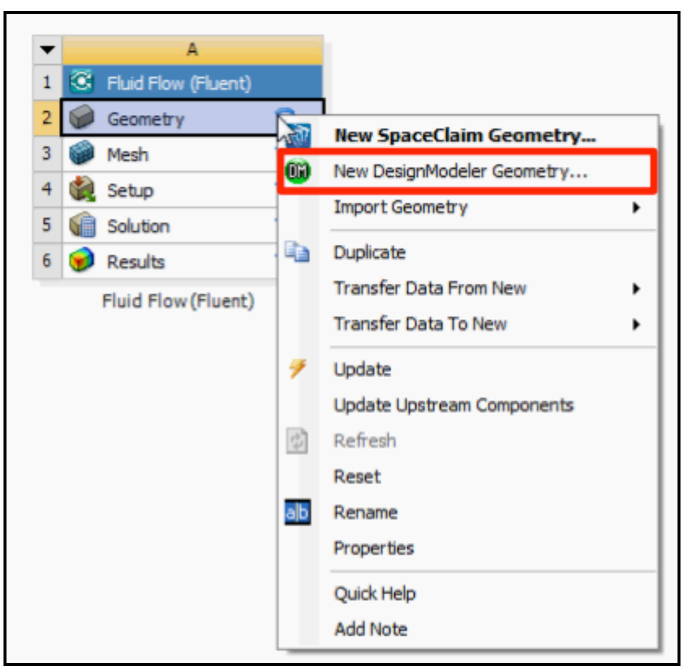
Analysis Type

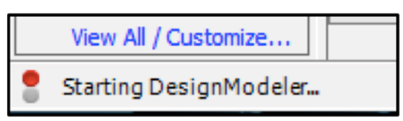
- Right click ***Geometry*** > ***Properties***
- On the Properties window to the right, set ***Analysis Type*** to ***2D***

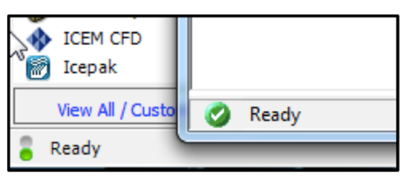


- Right click ***Geometry*** > ***New DesignModeler Geometry***

This opens up a new DesignModeler window. You can check the status of the WorkBench or any subsequently opened windows by looking at the bottom left corner of the window. Once the window is fully loaded, it will say Ready. 





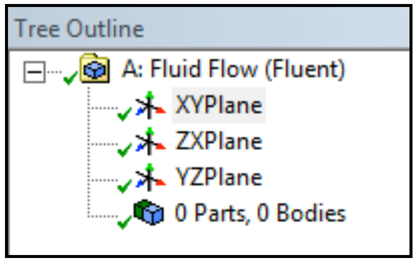


Downloading the Airfoil Sketch

First, check the units to make sure they are in meters.

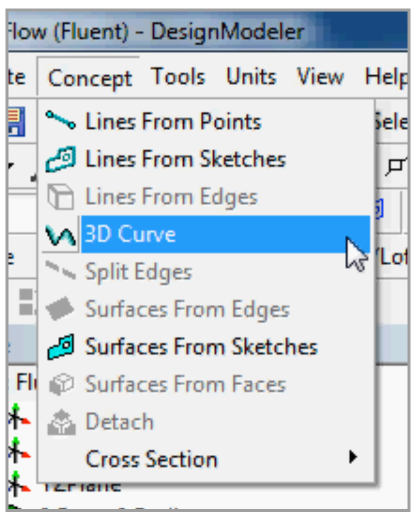
- In the toolbar at the top, click on ***Units*** > ***Meter***

On the left side is the *Tree Outline*, where all of the planes, sketches, and bodies are located.



- Click ***Concept*** > ***3D Curve***

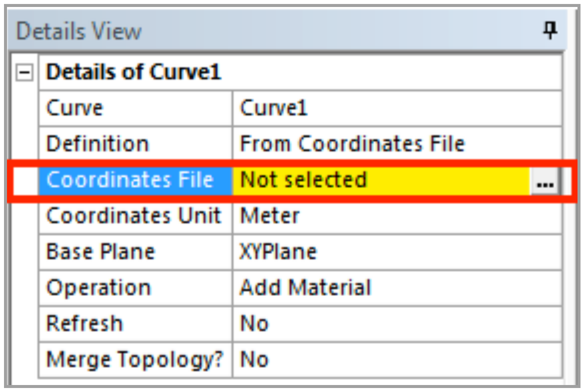
3D Curve creates line bodies using existing points or coordinates.



This will bring up a new *Details View* on the bottom left area.

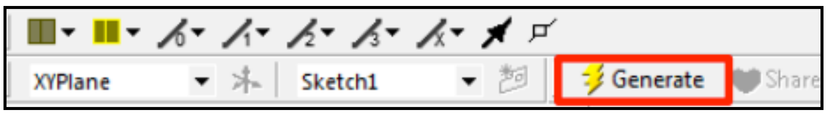
- In the *Details View*, for *Definition*, select ***From Coordinates File***

- Click next to ***Coordinates File*** and click on the ellipsis ...

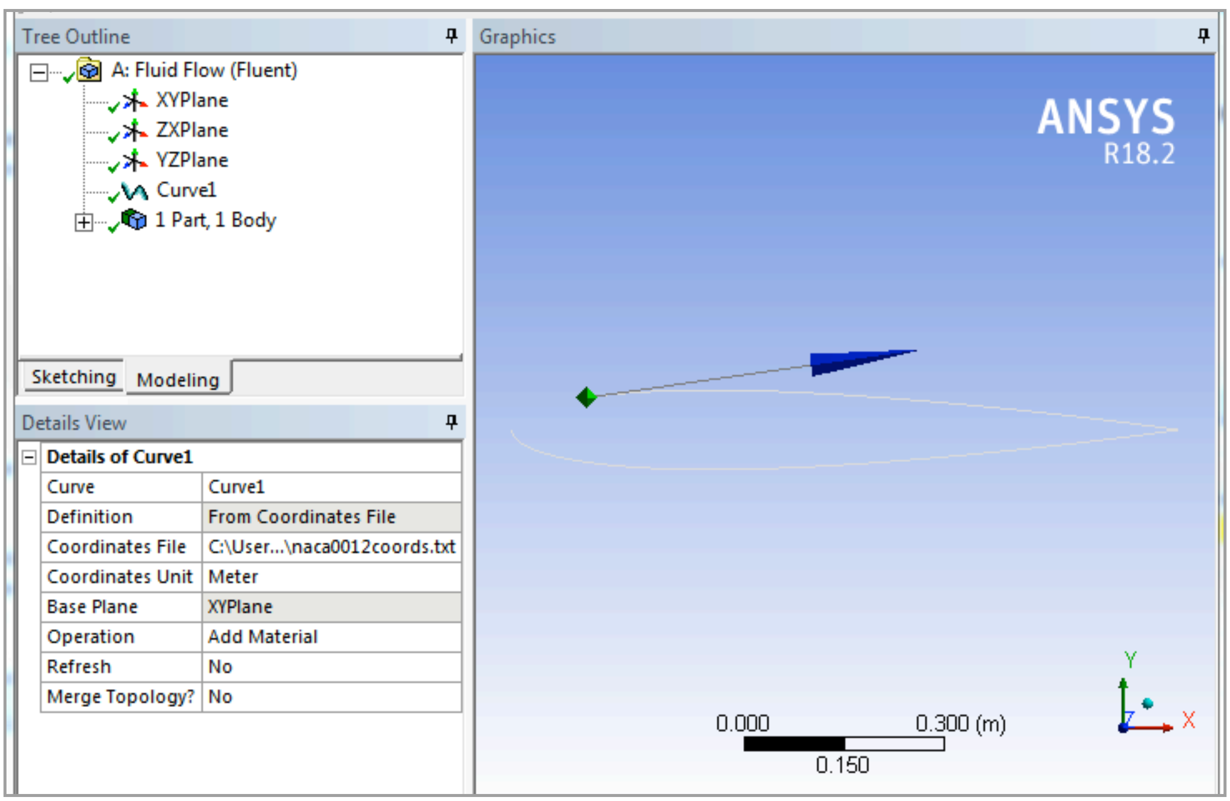


This will open up your file browser.

- Find the ***naca0012coords*** text document and double click to select
- Click ***Generate*** on the toolbar at the top of the window

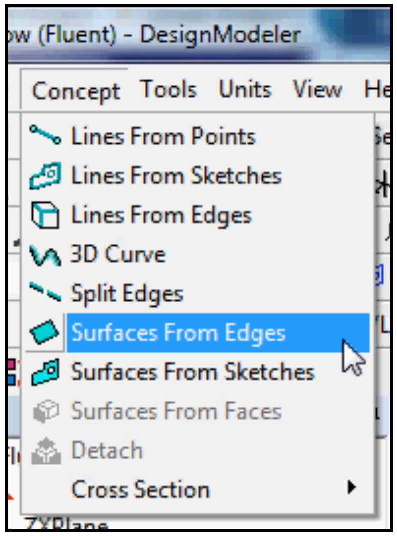


- Click the ***Look at Face/Plane/Sketch*** icon  in the top toolbar to see the sketch

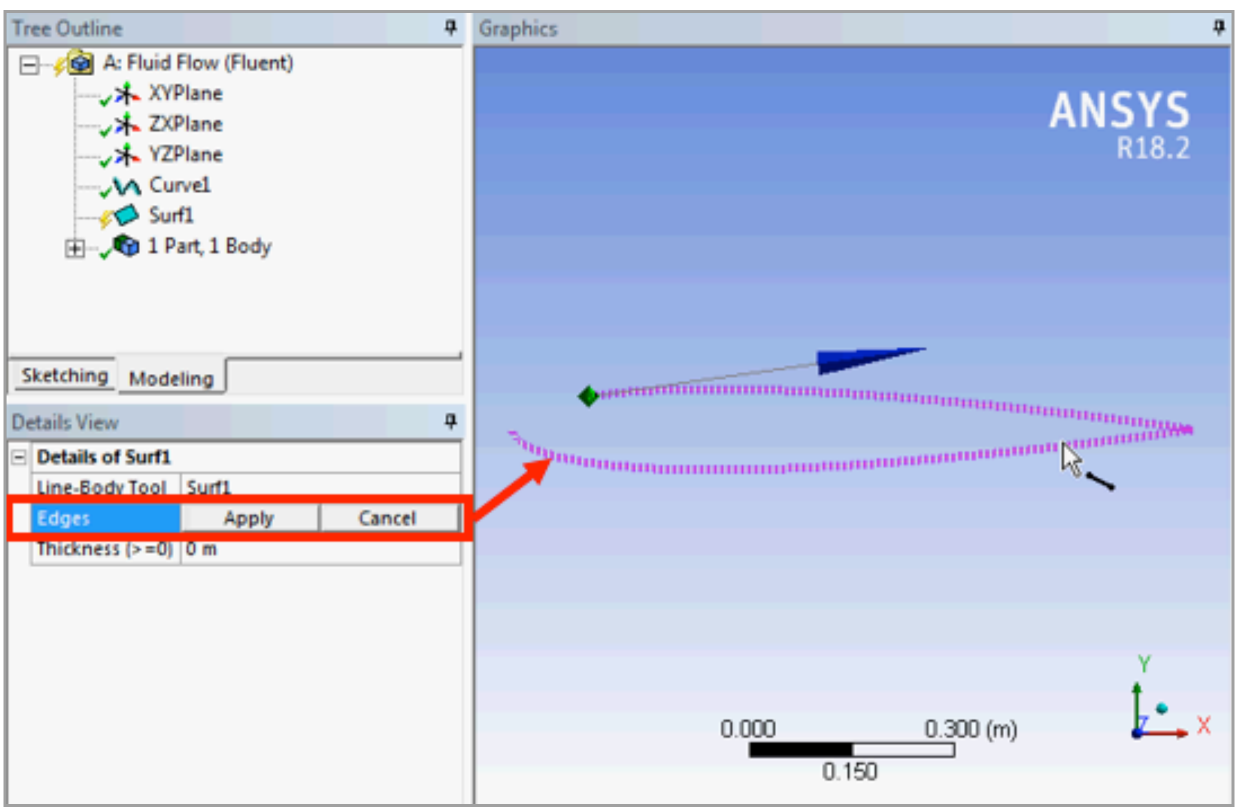


- Click ***Concept*** > ***Surfaces From Edges***

This uses the curve as a guideline to create a uniform surface which will represent the airfoil.



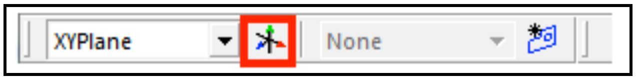
- Click on the airfoil edge
- Click next to ***Edges*** and click ***Apply***
- Click ***Generate***



Creating a New Coordinate System

In order for the C-mesh domain to completely cover the airfoil, the origin of the coordinate system should be moved so that it lies on the tail of the airfoil. This way, the tail can be designated as the center of the circular arc covering the airfoil to the left.

- Click the ***New Plane*** icon in the top toolbar to create a new coordinate system

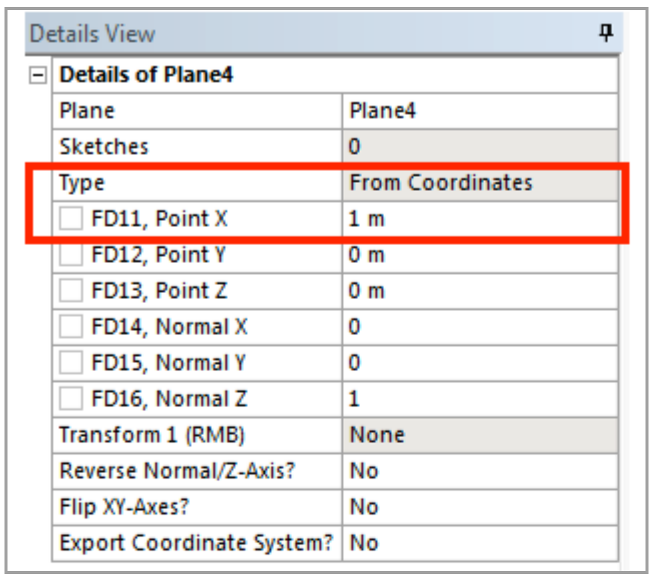


- In the *Details View*, for *Type*, select ***From Coordinates***

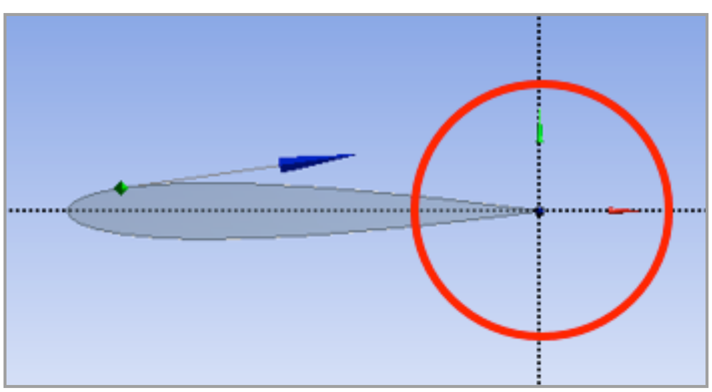
This will create a new coordinate system with the origin at a specified coordinate.

- For *FD11, Point X*, type “1” m

This will shift the origin of the coordinate system 1 m to the right, in the x-direction.

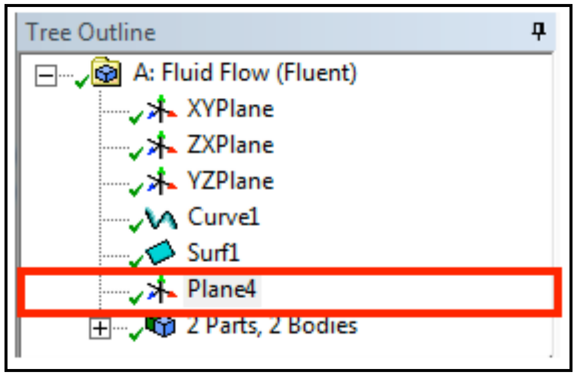


- Click ***Generate***



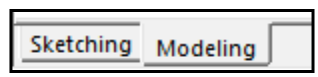
Creating the C-Mesh Domain

- Under the *Tree Outline*, click ***Plane4*** (the plane just created)



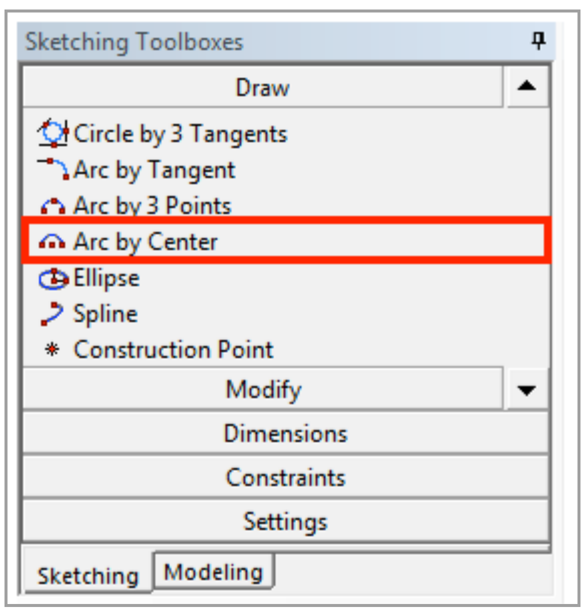
- Click ***New Sketch***  to create a new sketch

This creates a new sketch under the new plane. This sketch must be clicked and highlighted before working and reworking on the elements in this sketch. After clicking new sketch, this new sketch is already selected and can be worked on.

- Click on the ***Sketching*** tab, to the left of Modeling 

- Under ***Draw***, click ***Arc by Center*** 

In order to see the full list of options, click and hold on the down arrow (besides Modify, in this case).

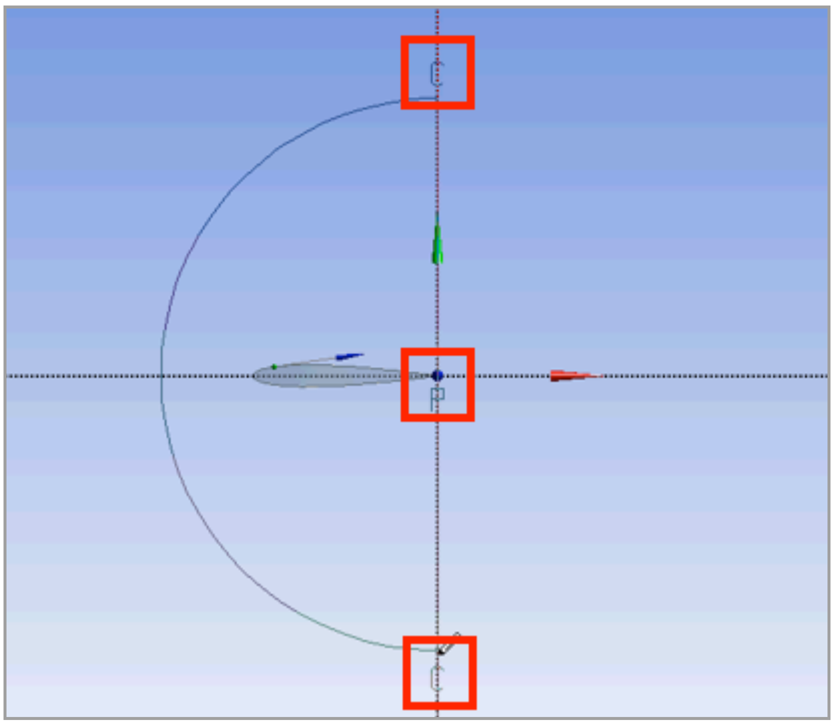


Arc by Center creates an arc by clicking on the center of the arc, then the two endpoints of the arc.

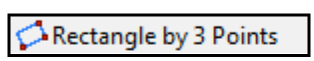
- Click on the origin of Plane4 at the tail to designate the center (A “P” will appear on the mouse arrow - see additional notes)

- Click above along the vertical y-axis so that the image of the circle fully encloses the airfoil (A “C” will appear on the mouse arrow)

- Click below along the vertical y-axis so that the image of the circle fully encloses the airfoil (A “C” will appear on the mouse arrow)



Creating the Right Side of the C-Mesh Domain

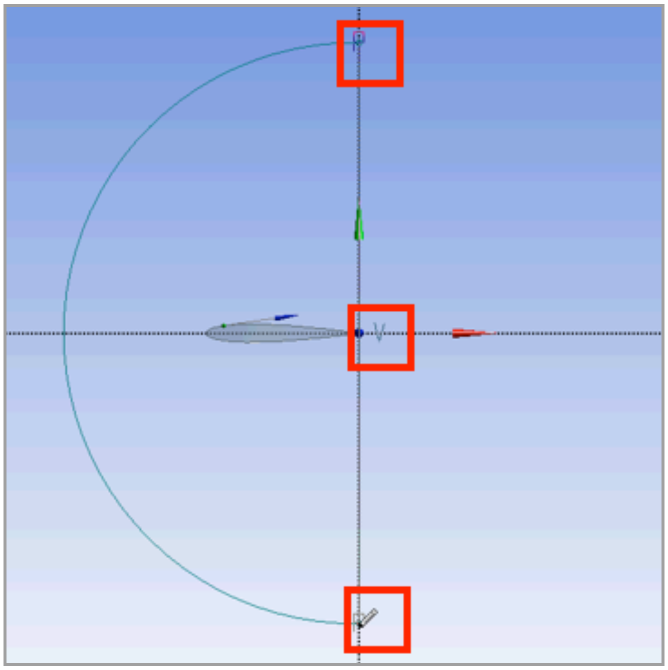
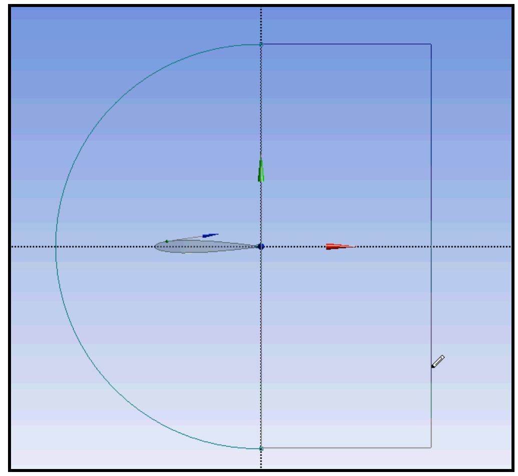
- Under ***Draw***, click ***Rectangle by 3 Points*** 

Rectangle by 3 Points creates a rectangle by clicking on two points to designate the length of one side of the rectangle, and then a third point to designate the width.

- Click on the point of intersection of the top of the C-Mesh and the vertical y-axis (A “P” will appear on the mouse arrow)

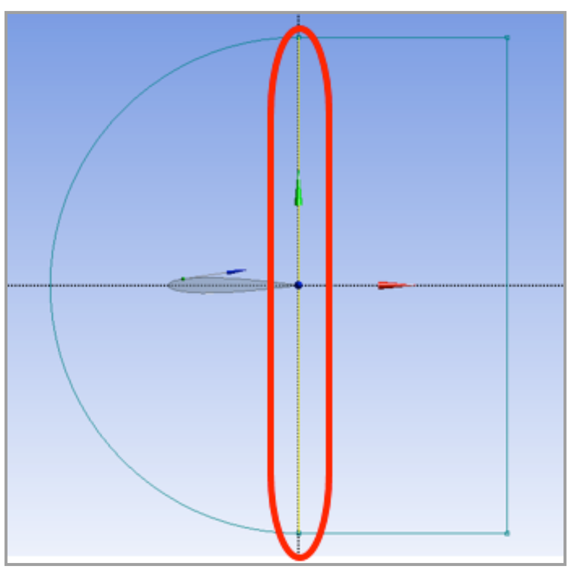
- Click on the point of intersection of the bottom of the C-Mesh and the vertical y-axis (A “P” will appear on the mouse arrow, and a “V” in the center of the line)

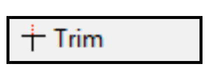
- Click to the right to create a rectangle

If the rectangle is not drawn with its corners coincident to the endpoints of the arc, manual constraints can be used to reposition the rectangle. Under ***Constraints***, click ***Coincident*** and choose the corner of the rectangle and the endpoint of the arc to make the two points coincide with each other.

The rectangle creates a vertical line along the y-axis. However, this line is not part of the domain surface, and therefore must be removed.

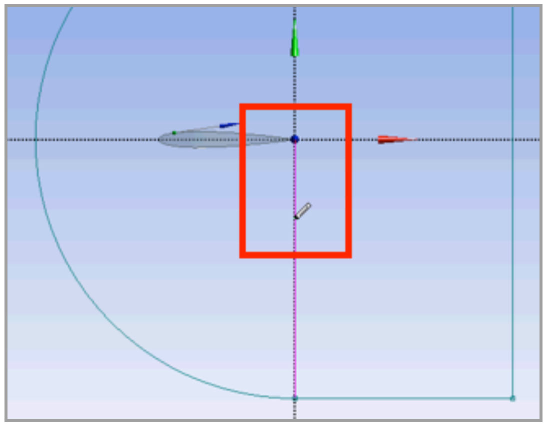


- Click on ***Modify***, and click ***Trim*** 

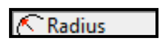
- Click on the top half of the vertical line

- Click on the bottom half of the vertical line

Trimming removes the excess line up to the nearest intersection of the line and previous sketches or points (point of origin of the axes, in this case).



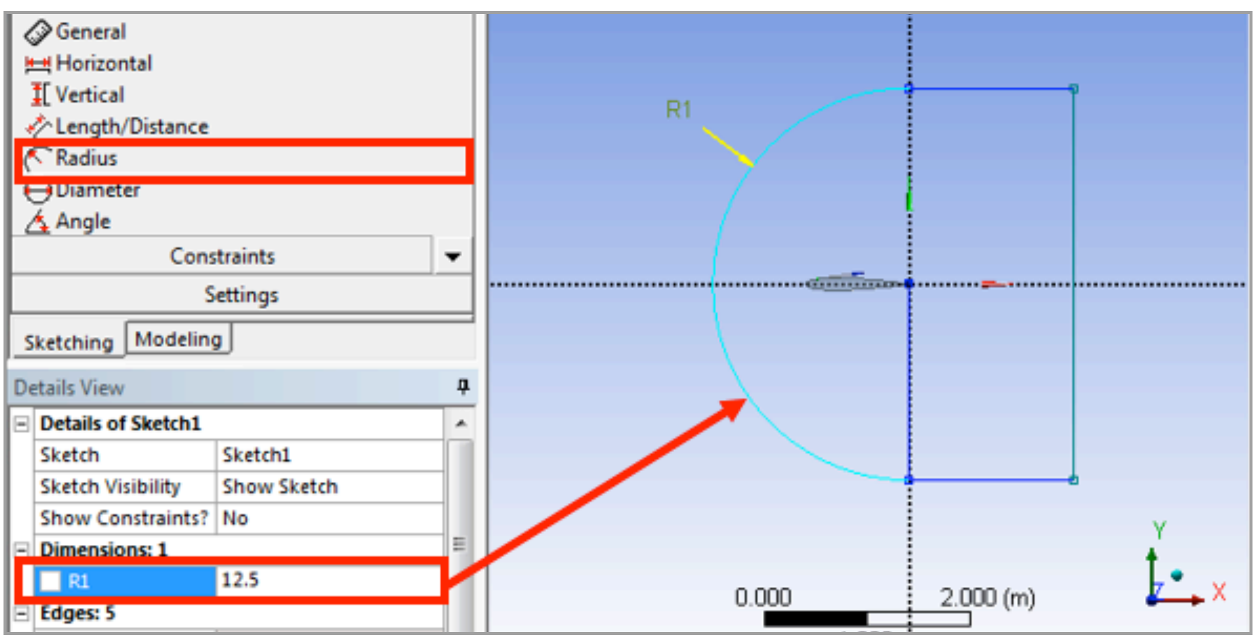
Dimensioning the Domain

- Click on ***Dimensions***, and choose ***Radius*** 

- Click the rim of the sketched arc to dimension the particular sketch, and then click outside the arc

This will bring up a new *Details View* on the bottom left area.

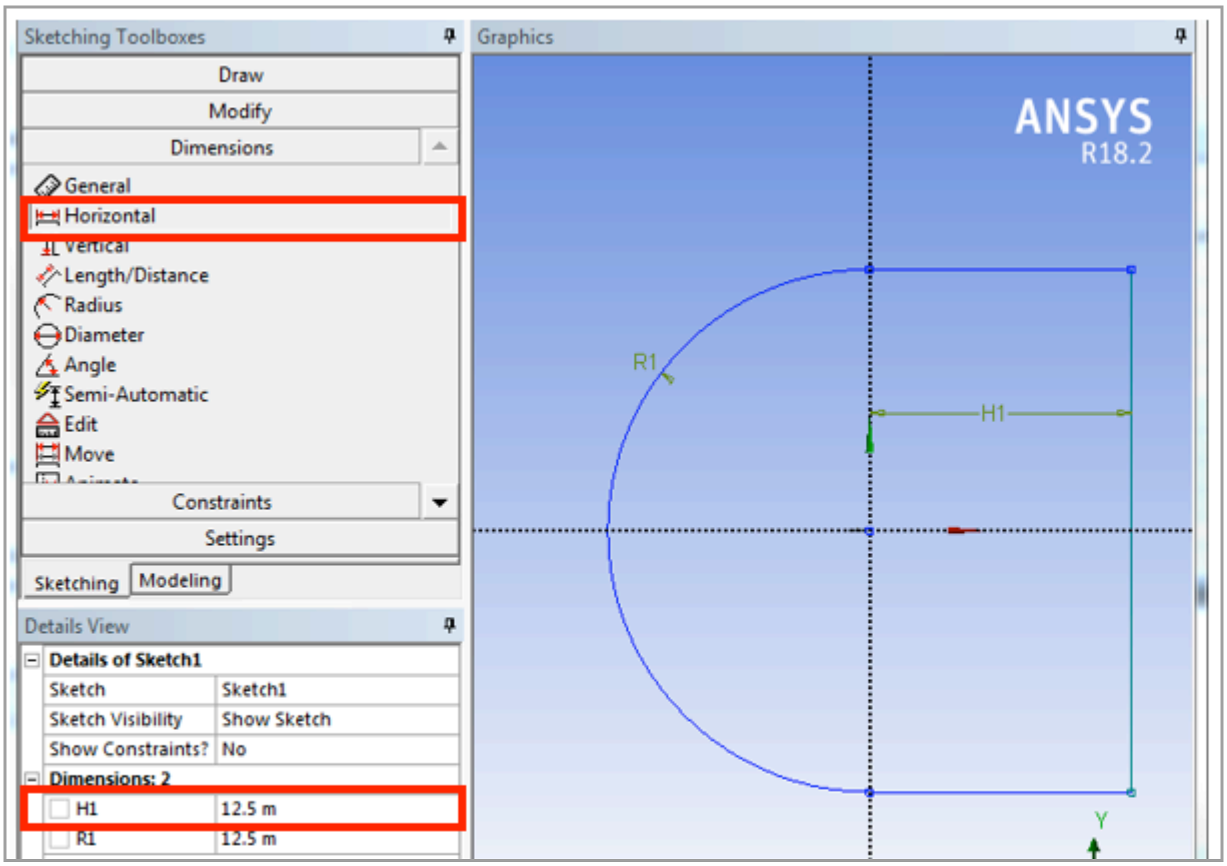
- In the *Details View*, under *Dimensions*, click next to *R1* and type in “12.5” (C-Mesh radius)



- Choose ***Horizontal*** 

- Click on the vertical y-axis and then click on the vertical line of the rectangle to the right to dimension the horizontal width between the two. Click again to dimension.

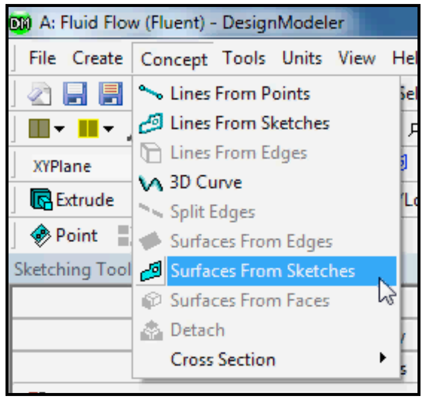
- In the *Details View*, under *Dimensions*, click next to *H1* and type in “12.5” (Rectangle width)



Creating the Flow Domain Surface

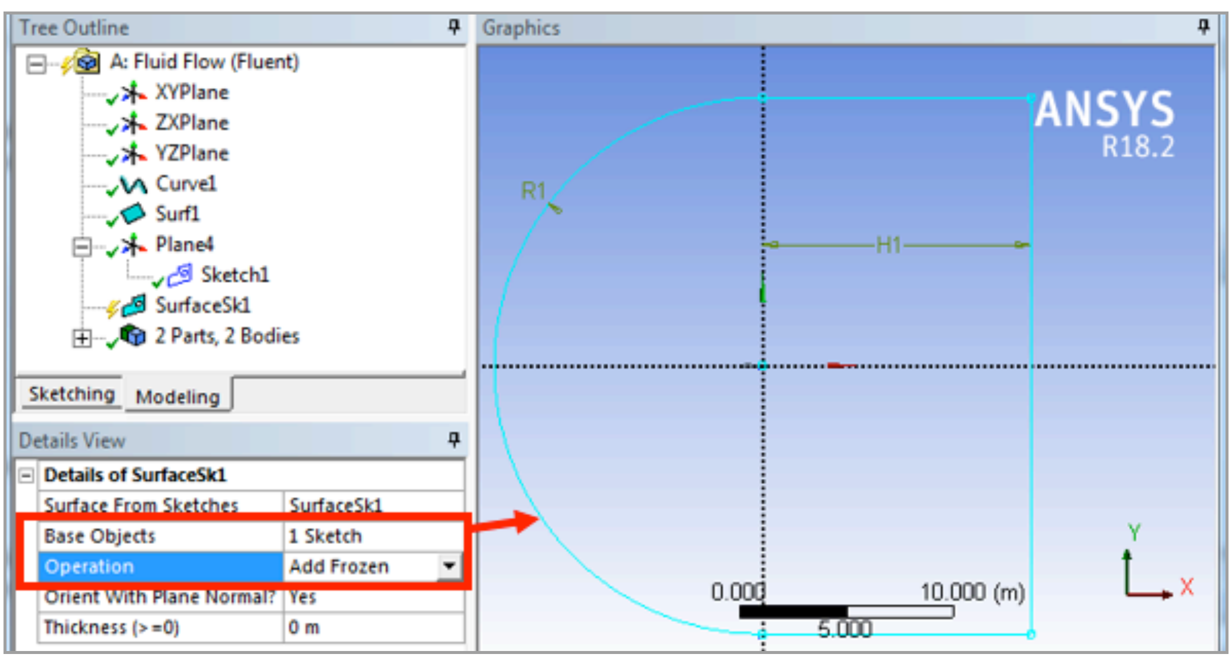
- Click ***Concept*** > ***Surfaces From Sketches***

This uses the sketches as a guideline to create a uniform surface, which in this case, will be filled with air.



- Set the *Base Objects* to ***Sketch 1*** (the sketch just made, under Plane4)
- Click ***Apply***
- For *Operation*, choose ***Add Frozen***

Add Frozen creates another surface, but does not merge the surface with a previously made surface. This is necessary to distinguish between the outer domain and the airfoil, which will eventually be removed.



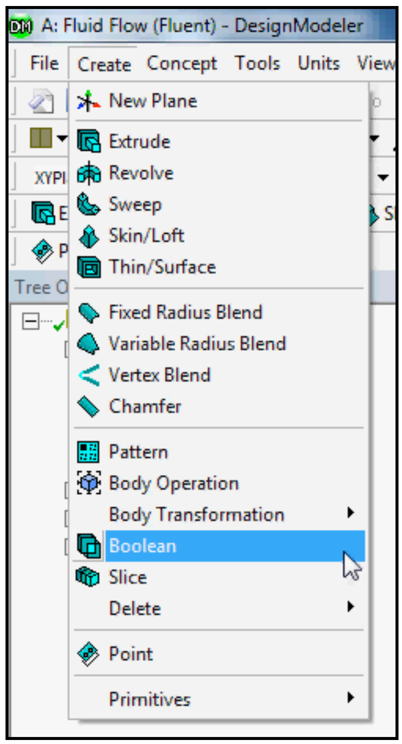
- Click ***Generate*** on the toolbar at the top of the window

Removing the Airfoil Surface

Since the air flow does not pass through the airfoil body, the surface of the airfoil can be removed from the flow domain.

- Click ***Create*** on the top toolbar > ***Boolean***

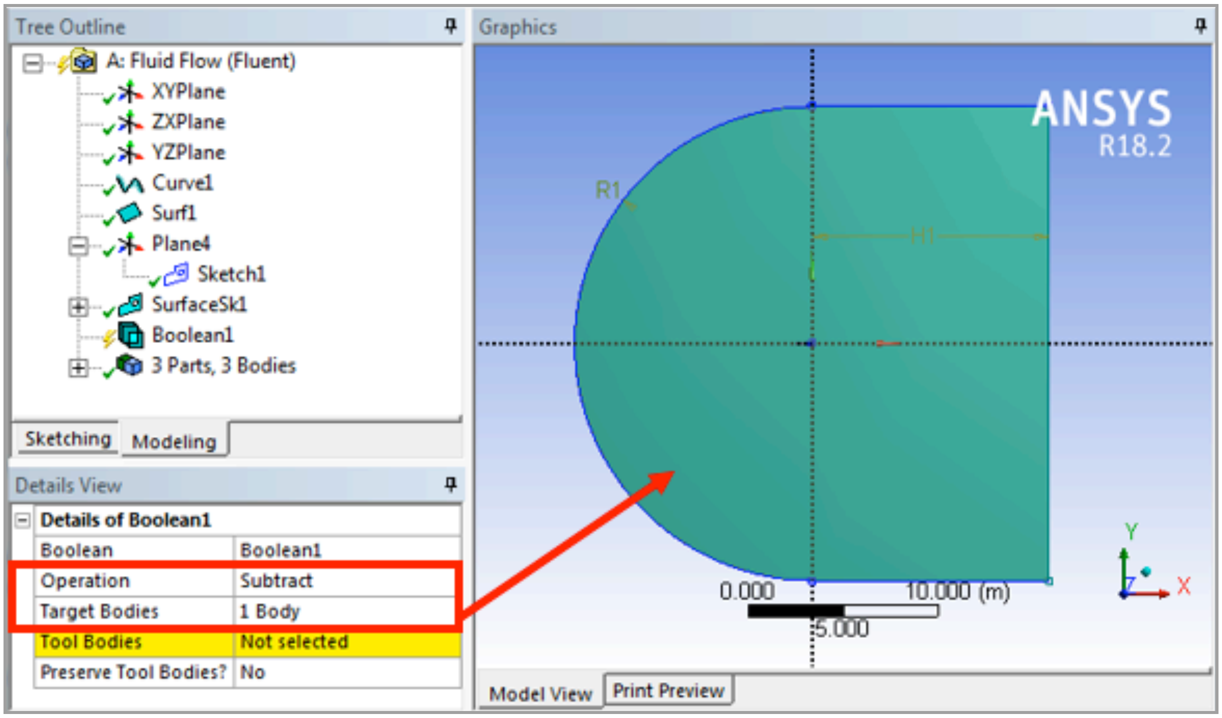
Boolean carries out operations of multiple bodies. This is useful in creating complex shapes from a combination of simpler shapes.



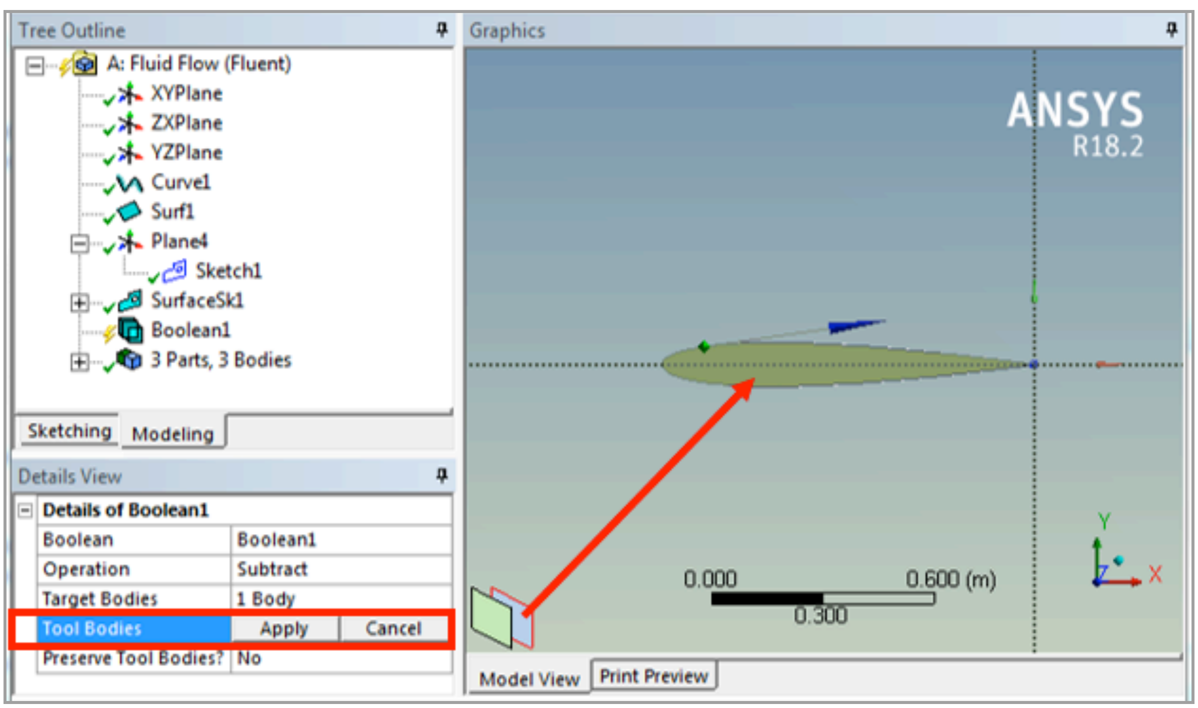
- For *Operation*, choose ***Subtract***

Subtract will be used to remove the airfoil surface from the C-Mesh domain surface.

- For the *Target Body*, click on the C-Mesh domain surface and click ***Apply***



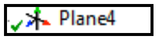
- For the *Tool Body*, zoom in for a clear view of the airfoil and click on the airfoil surface. An image of two layers will come up on the bottom left corner, showing that there are two layers, the C-Mesh domain and the airfoil, to choose from. Choose the layer that highlights the airfoil surface and click ***Apply***

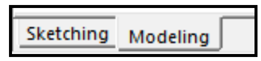


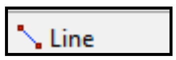
- Click ***Generate*** on the toolbar at the top of the window

Creating Quadrants

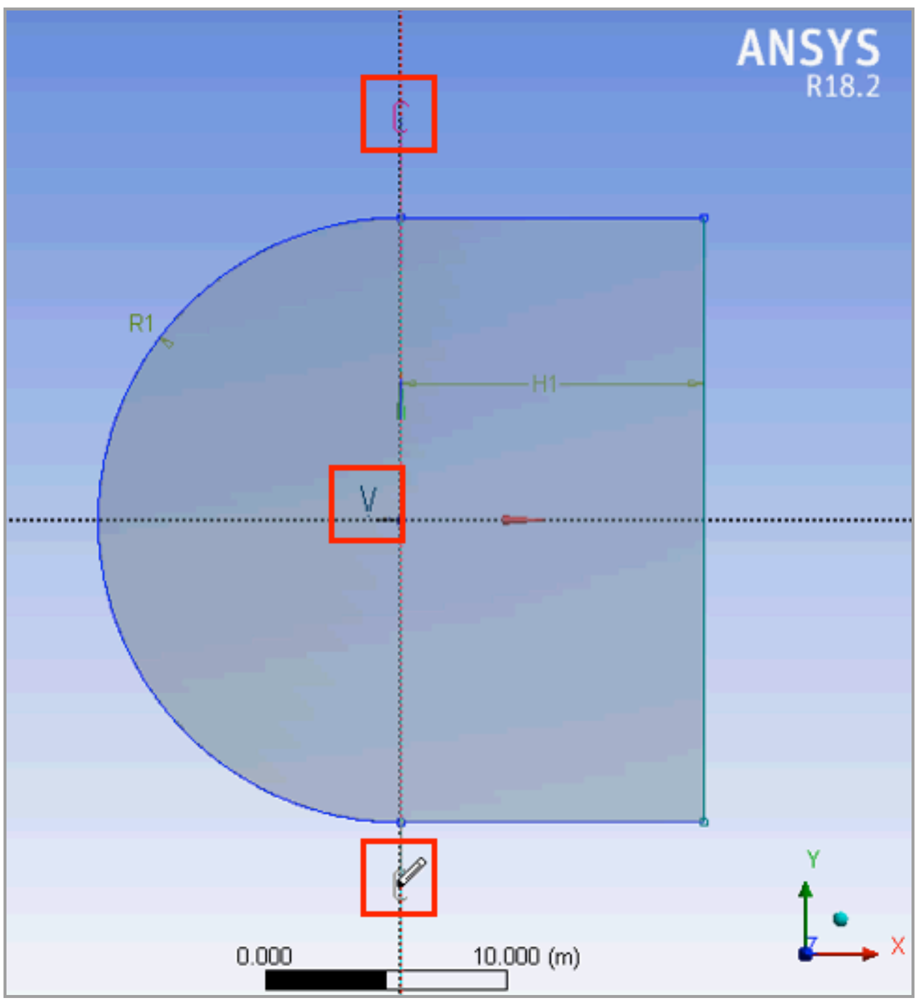
In order to create edge sizings in the meshing step (discussed later), a horizontal line and a vertical line breaking the surface into four quadrants must be imprinted onto the surface.

- Click ***Plane4*** 

- Click ***New Sketch*** 
- Click on the ***Sketching*** tab, to the left of Modeling 

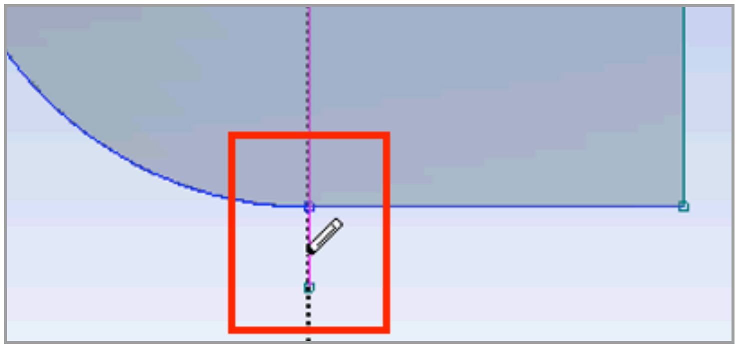
- Under ***Draw***, click ***Line*** 

- Click on a point along the y-axis above the C-Mesh domain (A "C" will appear on the mouse arrow), and drag the mouse to a point along the y-axis below the C-Mesh domain, creating a vertically straight line. Click to release



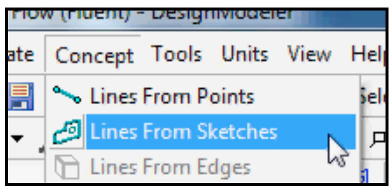
- Click on ***Modify***, and click ***Trim*** 

- Click on any point above the C-Mesh domain along the line just created
- Click on any point below the C-Mesh domain along the line just created



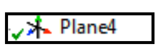
- Click ***Concept*** > ***Lines From Sketches***

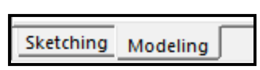
This uses the remaining portions of the vertical line sketch to create a physical line.

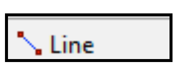


- Set the *Base Object* to ***Sketch 2*** (the line sketch just made, under Plane4)
- Click ***Apply***
- Click ***Generate***

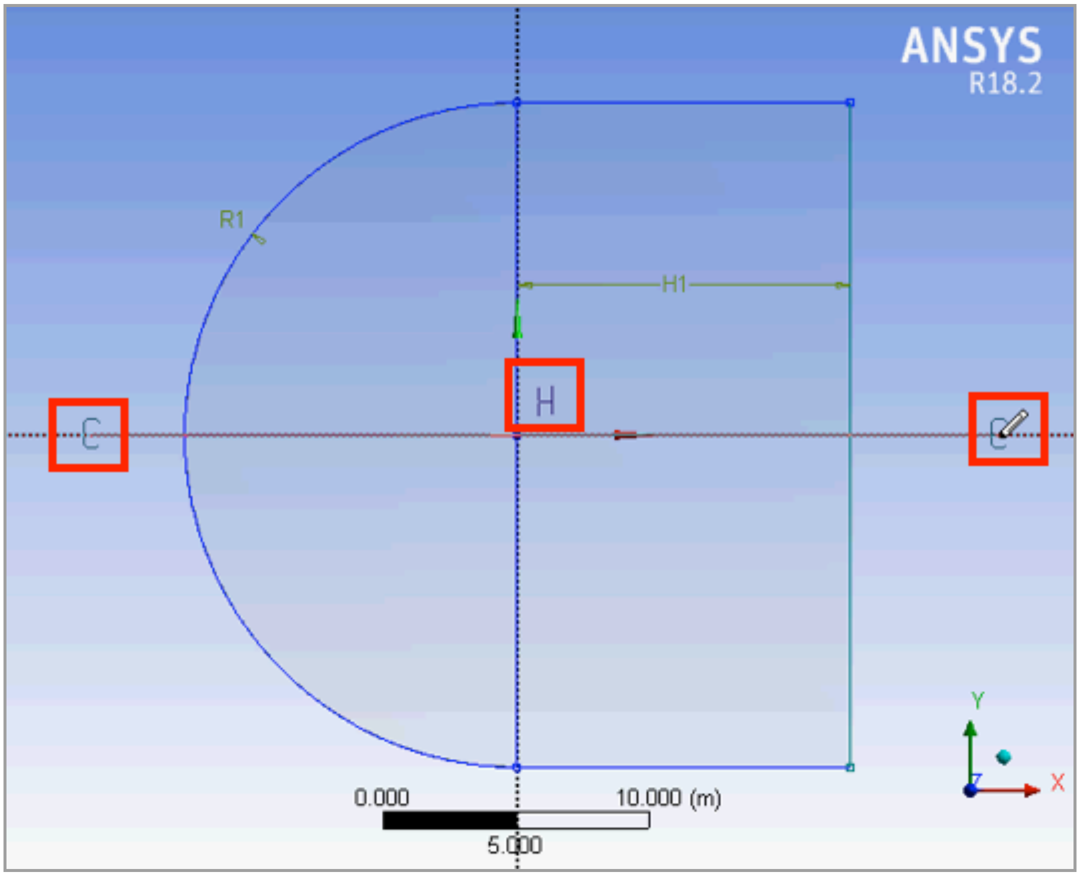
Now, the horizontal line will be created.

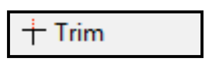
- Click ***Plane4*** 

- Click ***New Sketch*** 
- Click on the ***Sketching*** tab, to the left of Modeling 

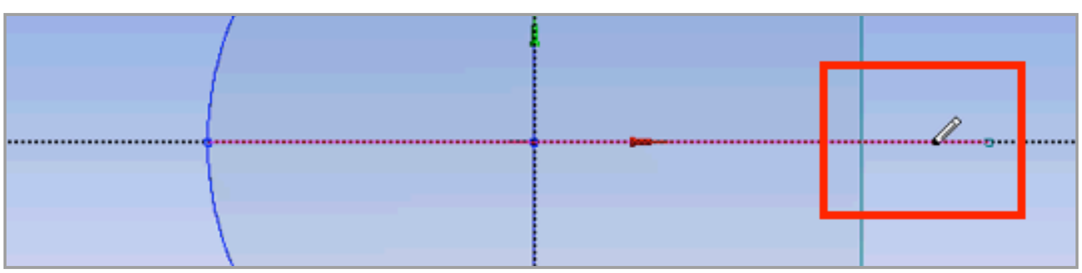
- Under ***Draw***, click ***Line*** 

- Click on a point along the x-axis to the left of the C-Mesh domain (A “C” will appear on the mouse arrow), and drag the mouse to a point along the x-axis to the right of the C-Mesh domain, creating a horizontally straight line. Click to release



- Click on ***Modify***, and click ***Trim*** 

- Click on any point to the left of the C-Mesh domain along the line just created
- Click on any point to the right of the C-Mesh domain along the line just created



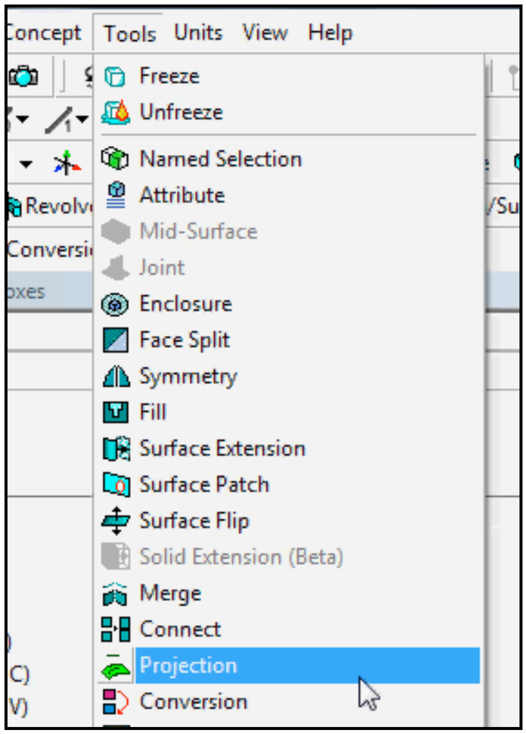
- Click ***Concept*** > ***Lines From Sketches***
- Set the *Base Object* to ***Sketch 3*** (the line sketch just made, under Plane4)
- Click ***Apply***

- Click ***Generate***

Projecting Quadrants

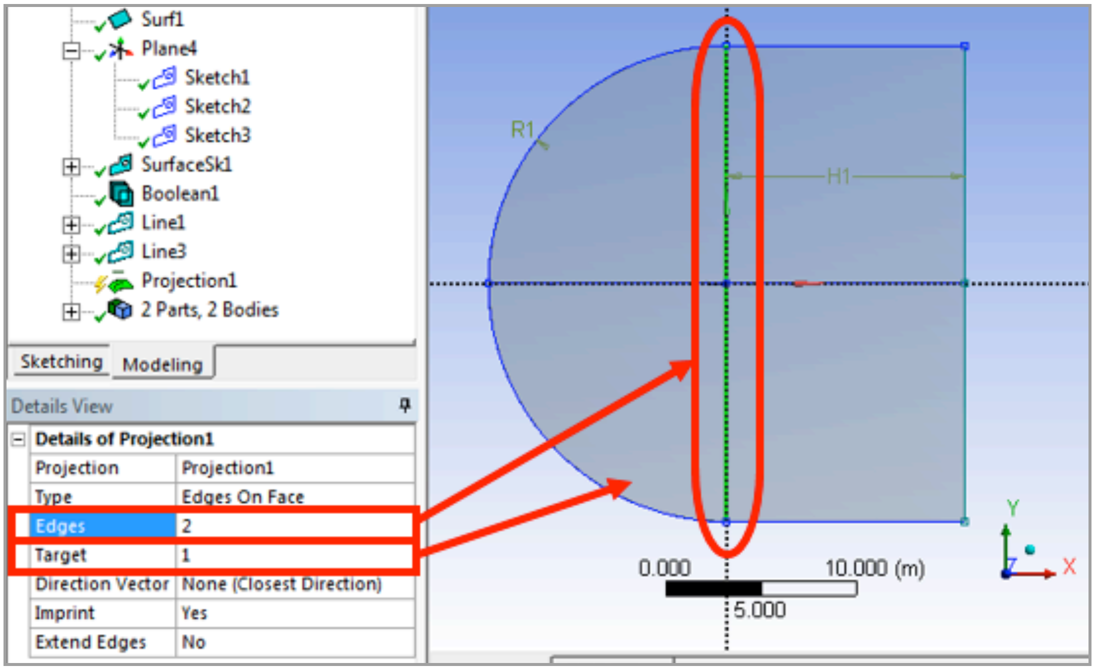
The line currently sits on a layer on top of the surface, and is not associated with the surface directly. In order to split the surface using the line, the line must be projected onto the surface.

- Click on ***Tools*** > ***Projection***



This will bring up a new set of specifications in the *Details View*.

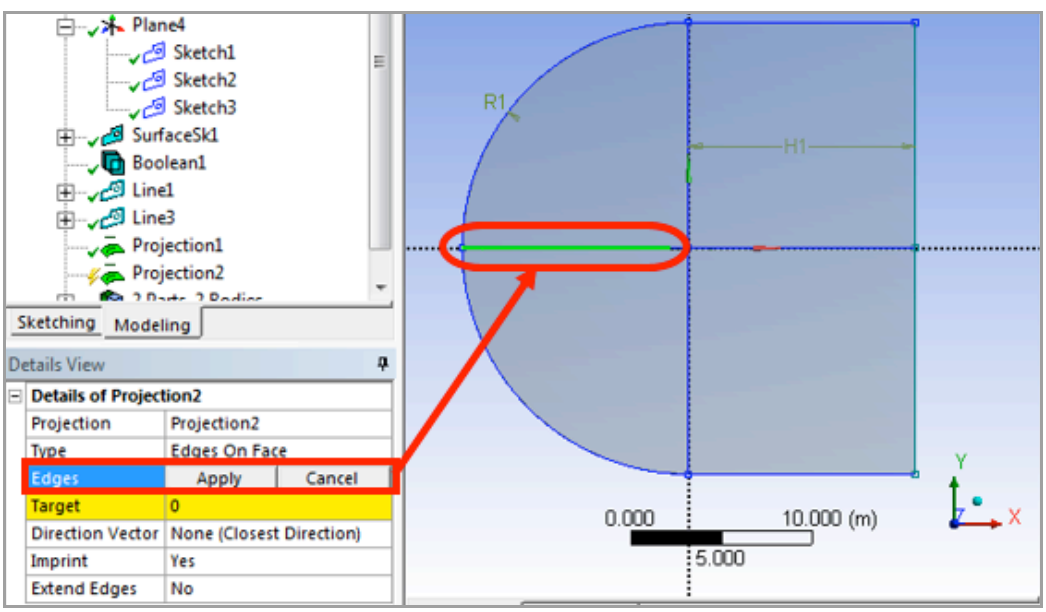
- Make sure the *Type* is ***Edges On Face***
- Set the *Edges* to the two vertical lines generated by clicking on one half of the vertical line, and while pressing the control key, clicking on the other half of the vertical line
- Click ***Apply***
- Set the *Target* to the entire surface body by clicking on any part of the surface
- Click ***Apply***



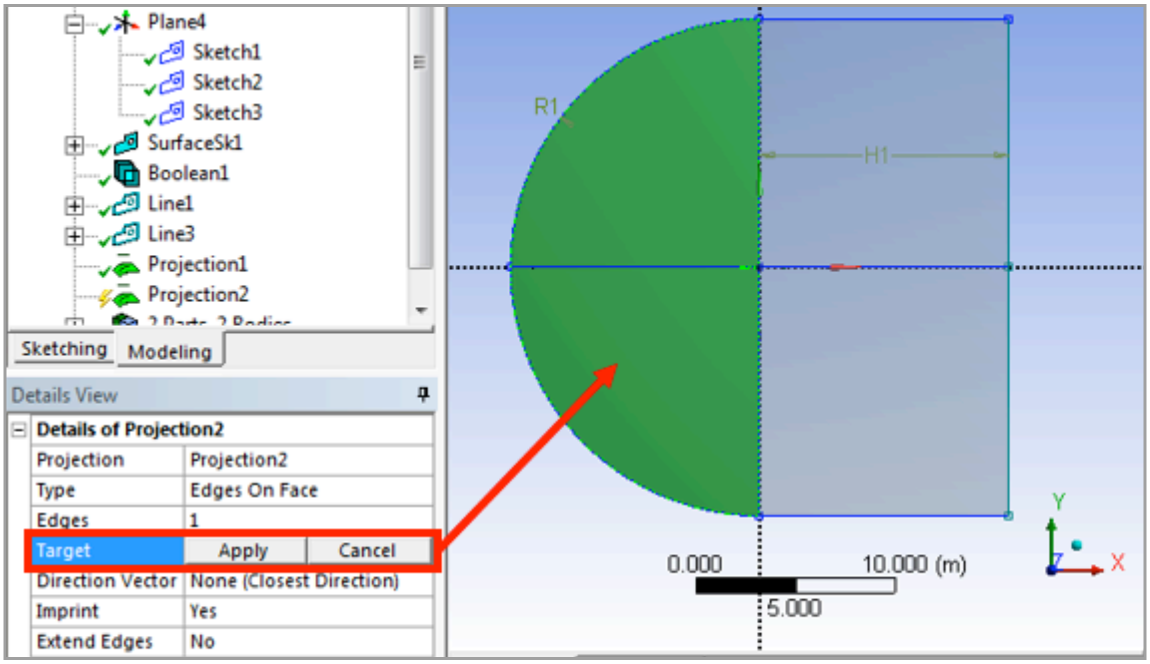
- Click ***Generate***

This separates the C-Mesh domain into two halves, a left half and a right half. The left half will be further split using the left horizontal line, and the right half will be split using the right horizontal line to create four total quadrants.

- Click on ***Tools*** > ***Projection***
- Make sure the *Type* is ***Edges On Face***
- Set the *Edges* to the left half of the horizontal line



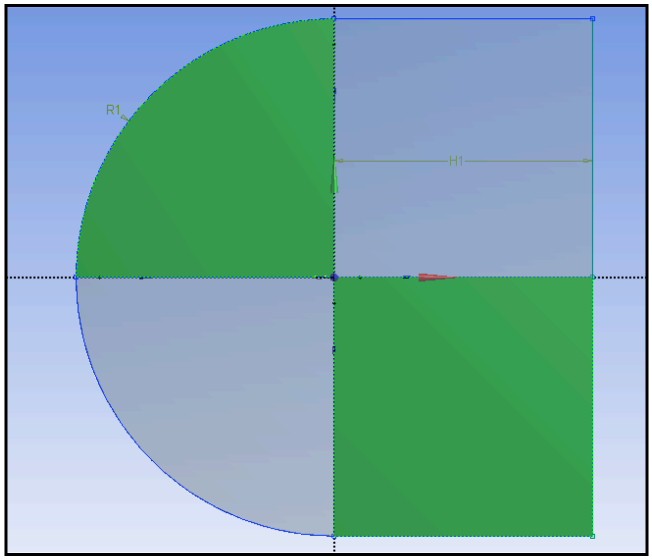
- Click ***Apply***
- Set the *Target* to the left side of the surface body by clicking on the left half of the surface
- Click ***Apply***



- Click ***Generate***

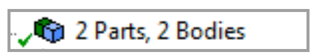
The process is repeated for the right half.

- Click on ***Tools*** > ***Projection***
- Make sure the *Type* is ***Edges On Face***
- Set the ***Edges*** to the right half of the horizontal line
- Click ***Apply***
- Set the ***Target*** to the right side of the surface body by clicking on the right half of the surface
- Click ***Apply***
- Click ***Generate***



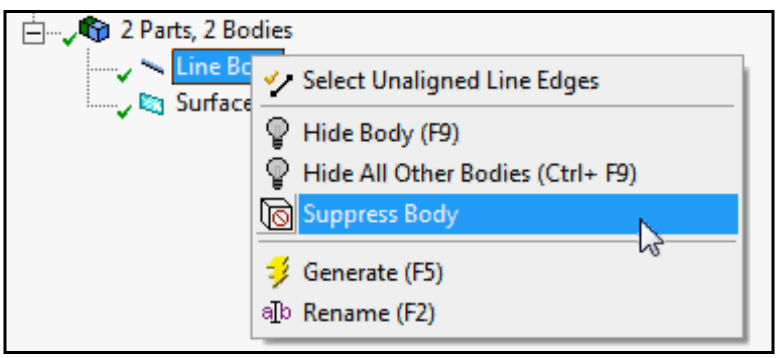
Suppressing Line Bodies

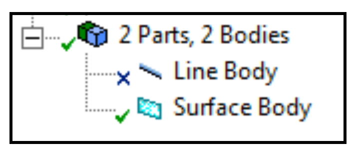
Since the vertical lines are not physically a part of the setup, the lines must be suppressed in order to prevent Fluent from treating them as physical boundaries.

- Click the + next to ***2 Parts, 2 Bodies*** 

- Right click ***Line Body*** 

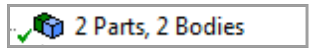
 > ***Suppress Body***



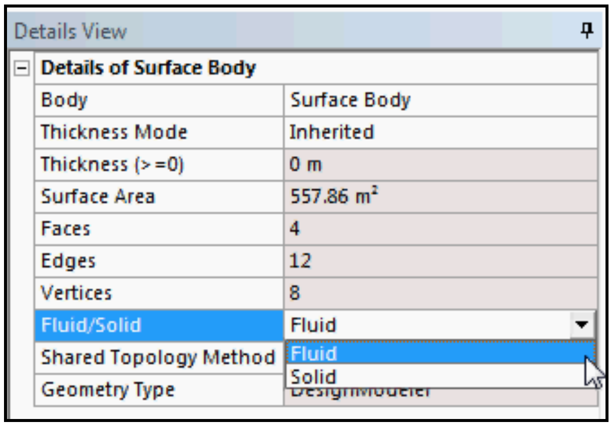


Changing the Surface Type to Fluid

DesignModeler assumes all surfaces to be solids. However, the surface for this case is a fluid. Therefore, the body type must be changed to fluid.

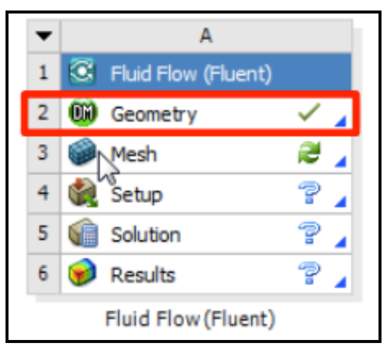
- Click the + next to ***2 Parts, 2 Bodies*** 

- Click the ***Surface Body***
- Under *Details View*, select ***Fluid/Solid*** > ***Fluid***



- Click ***File*** > ***Save Project*** and close DesignModeler. Return to the *WorkBench*.

At this point, *Geometry* should have a check mark.



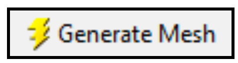
3. Mesh

In order to simplify the calculations of the flow for the software, a mesh is applied to the surface. This separates the surface into discrete sections where calculations for the flow will be applied to. The final calculation will use the data gathered at each mesh node to analyze the flow.

In order to obtain detailed results of the flow near the airfoil, the mesh will be sized to be concentrated near the airfoil.

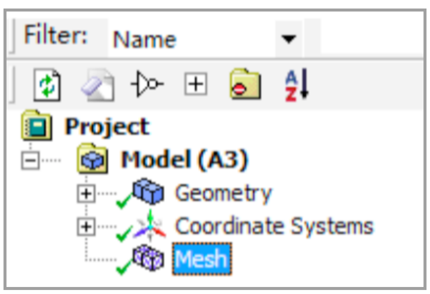
- Double click ***Mesh*** 

A new Meshing window will open. On the left side is the *Outline*.

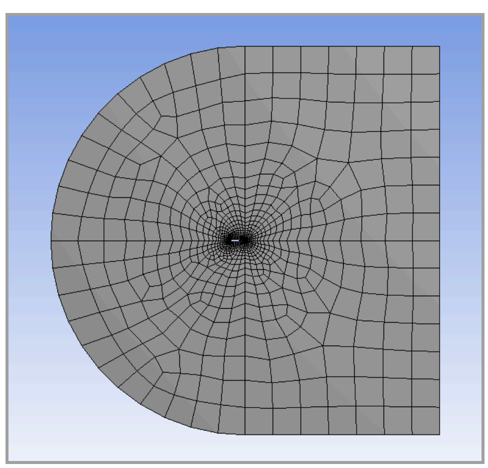
- Click ***Generate Mesh*** on the top toolbar 

- To view the mesh, click on ***Mesh*** 

Alternatively, click ***Show Mesh***



This creates a rough initial mesh for the surface similar to the one below:

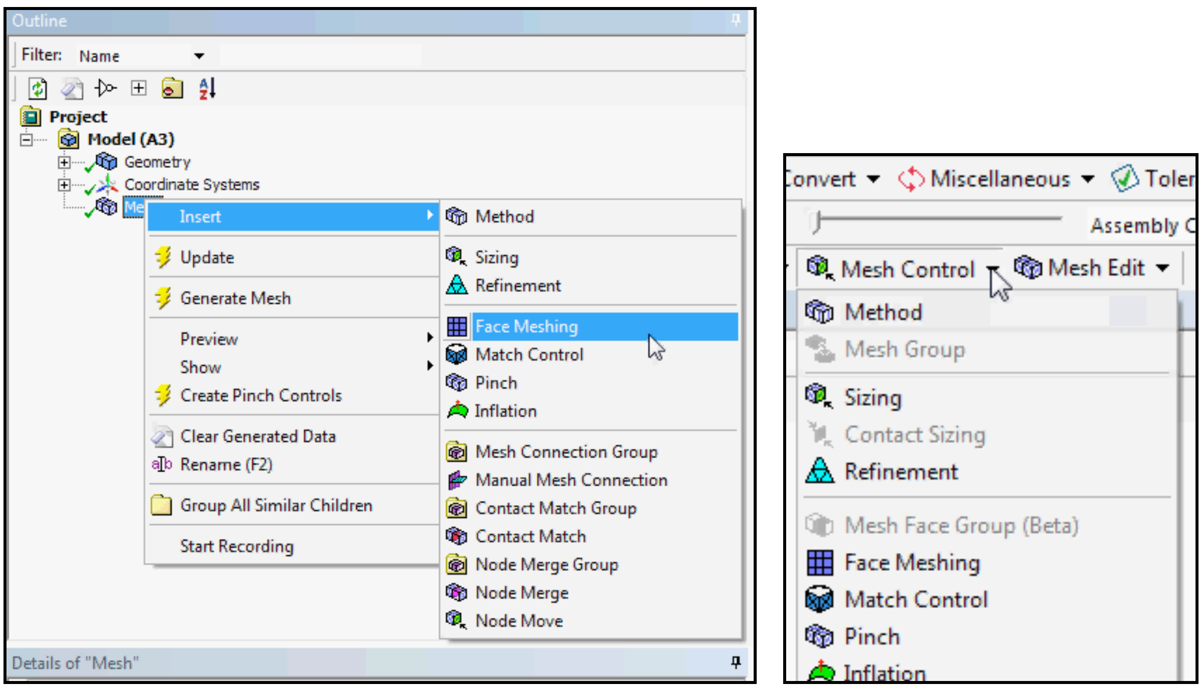


Mapped Face Meshing

In order to create a more structured and uniform mesh, mapped face meshing will be used. By inserting a face mesh, Fluent will automatically create a geometrically regular mesh on the applied face.

- Right click ***Mesh*** > ***Insert*** > ***Face Meshing***

Alternatively, ***Face Meshing*** and ***Sizing*** can be found under ***Mesh Control*** in the top toolbox.



These are two ways to access Face Meshing and Sizing

- In *Details View*, click next to ***Geometry***

- Click on the face selection icon on the top toolbar 

This selects a face.

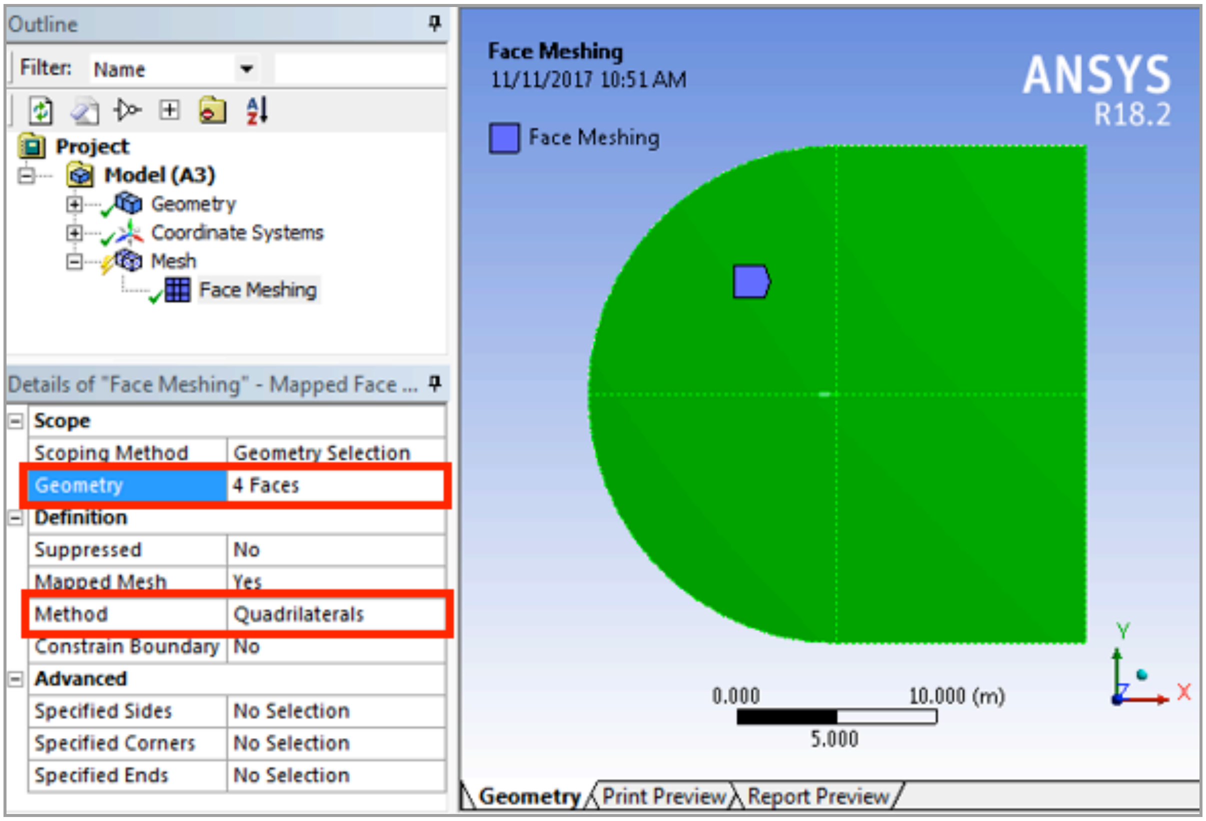
Other options are point selection , edge selection , and body selection .

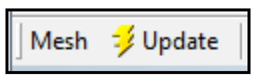
- Click on one quadrant of the surface, and while pressing the control key, click on the other quadrants of the surface to highlight four total quadrants

- Click ***Apply***

- Make sure the ***Method*** is ***Quadrilaterals***

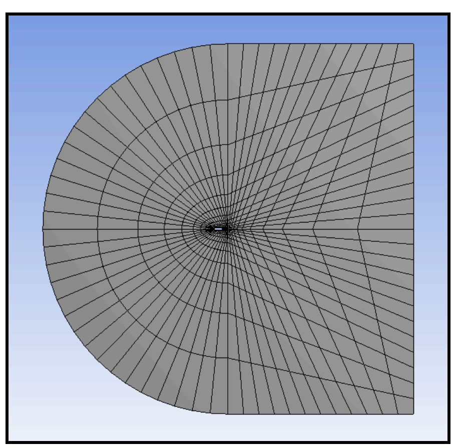
A quadrilateral mesh reduces the skew of the calculations.



- Click ***Update*** at the top toolbar 

Now, when you click ***Mesh*** 

in the *Outline*, the left side of the mesh will appear as concentric arcs with straight lines expanding radially outwards, and the right side will have straight lines concentrating towards the airfoil.

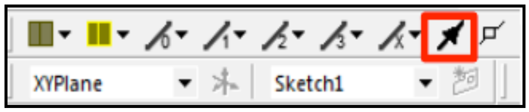


This mesh is already concentrated towards the airfoil, where the most detailed results are wanted. However, we will refine the mesh further to obtain more data points near the airfoil.

Edge Sizing 1
Edge sizing allows individual edges of the model to be meshed in a different format.

- Right click ***Mesh*** 

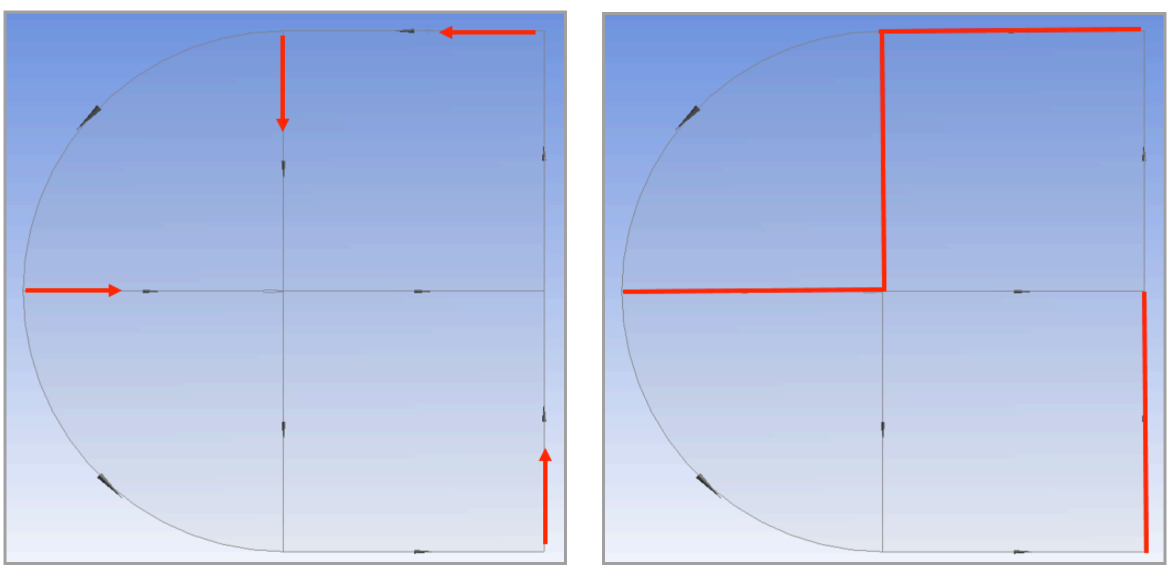
 > ***Insert*** > ***Sizing***
- In *Details View*, click next to *Geometry*
- Click on the Display Edge Direction icon  on the top toolbar



This shows the direction of the edges in order to set the proper bias type.

- Click on the edge selection icon  on the top toolbar
- While pressing the control key, select the top horizontal edge, the center vertical top half edge, the middle horizontal left half edge, and the right vertical bottom half edge (4 edges total, shown below)

These four edges have an edge direction pointing towards the airfoil. The edge sizing can be done for individual edges, however choosing 4 edges will save time.



- Click ***Apply***
- Set *Type* > ***Number of Divisions***
- For the *Number of Divisions*, type in “50”

This will divide each selected edge into 50 divisions. 50 divisions are used because the outer boundary arc has a radius of 12.5 m and the rectangle has a width of 12.5 m, so the number of divisions is easy to work with.

- Set *Behavior* > ***Hard***

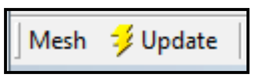
Setting the behavior to hard prevents the mesher from overwriting the user-inputted restrictions. Soft behavior allows the mesher to ignore restrictions based on the mesher's discretion. This is mainly used for highly complex models.

- Set *Bias Type* > **----- --- - -** (first option)

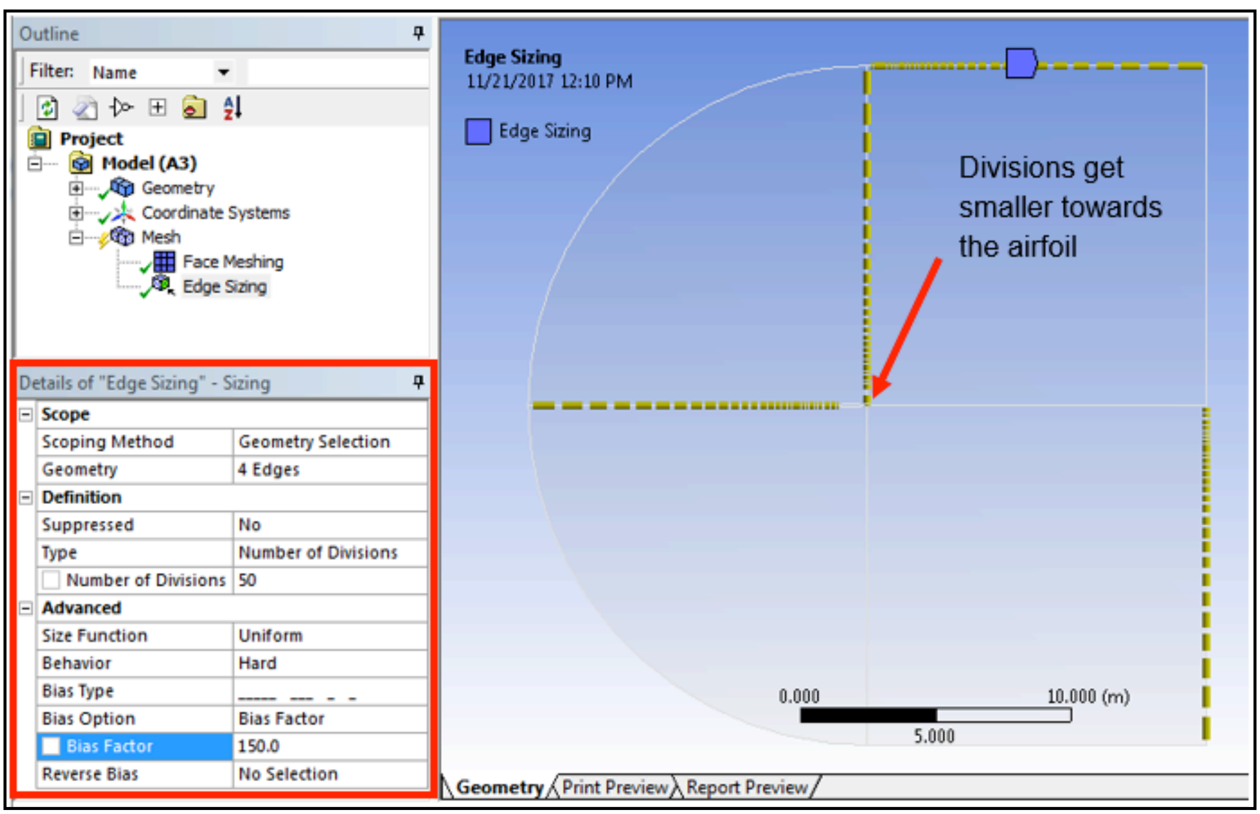
Bias changes the distances between individual divisions by a growth factor, specified in the next step. The first bias type starts off with larger divisions and becomes smaller in the direction of the edge.

- For the *Bias Factor*, type in "150"

The bias factor is calculated as the ratio of the longest division and the shortest division.

- Click ***Update*** 

on the top toolbar to update the mesh with the new sizing

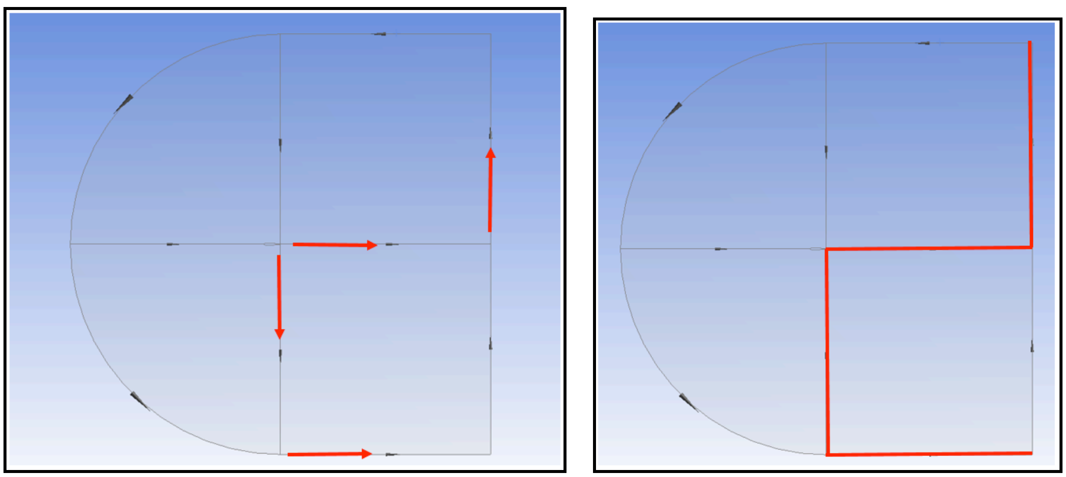


Edge Sizing 2

- Right click ***Mesh*** 

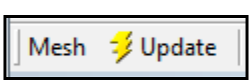
 > ***Insert*** > ***Sizing***
- In *Details View*, click next to *Geometry*
- Click on the edge selection icon  on the top toolbar
- While pressing the control key, select the bottom horizontal edge, the center vertical bottom half edge, the middle horizontal right half edge, and the right vertical top half edge (4 edges total, shown below)

These four edges have an edge direction pointing away from the airfoil.



- Click ***Apply***
- Set *Type* > ***Number of Divisions***
- For the *Number of Divisions*, type in “50”
- Set *Behavior* > ***Hard***
- Set *Bias Type* > **- - --- -----** (second option)

The second bias type starts off with smaller divisions and becomes larger in the direction of the edge.

- For the *Bias Factor*, type in “150”
- Click ***Update*** 

 on the top toolbar to update the mesh with the new sizing

Edge Sizing 3

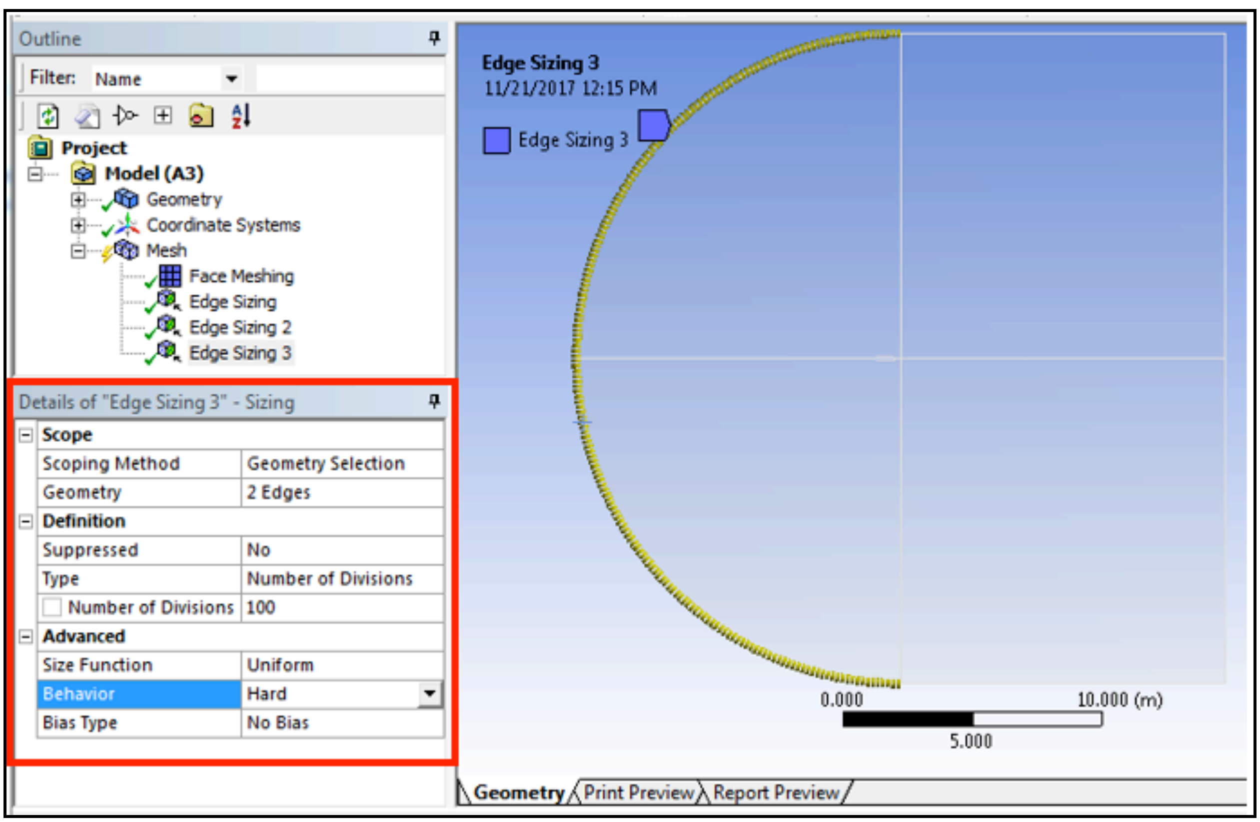
The C-Mesh arcs will be sized. No bias is necessary because the previous edge sizings will already take care of the mesh concentration near the airfoil.

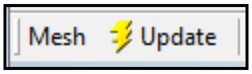
- Right click ***Mesh*** 

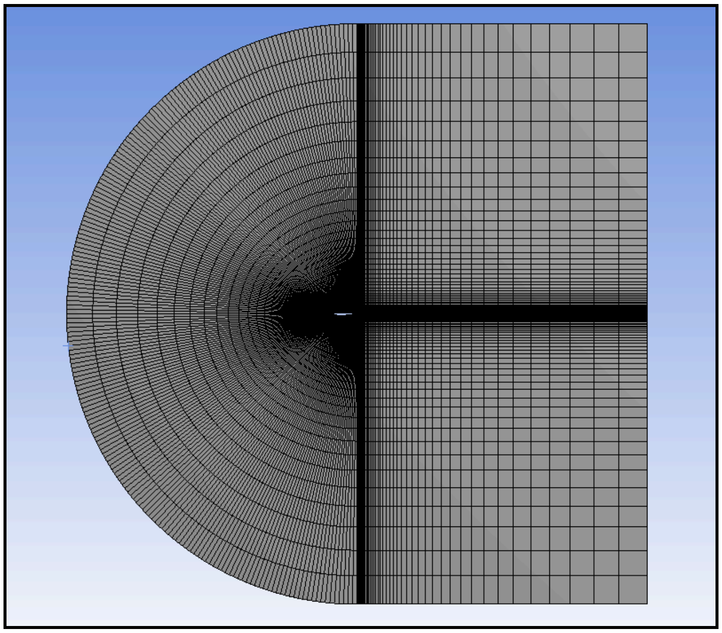
 > ***Insert*** > ***Sizing***
- In *Details View*, click on *Geometry*
- Click on the edge selection icon  on the top toolbar
- While pressing the control key, select two arcs of the C-Mesh boundary, the top half and the bottom half
- Click ***Apply***
- Set *Type* > ***Number of Divisions***
- For the *Number of Divisions*, type in “100”

This will divide each arc into 100 equal divisions.

- Set *Behavior* > ***Hard***



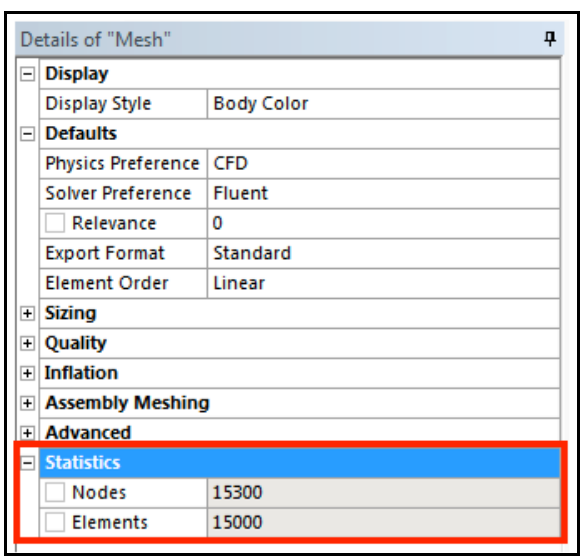
- Click ***Update***  on the top toolbar to update the mesh with the new sizing



Verifying the Mesh Size

- Click ***Mesh*** 
- Click the + next to ***Statistics***

The mesh should have 15300 Nodes and 15000 Elements.



Creating Named Selections

Named selections create a specific name for selected points, edges, faces or bodies in order to make them easier to identify in the setup stage. In this case, the velocity inlet, pressure outlet, and airfoil wall will be named.

- Click on the edge selection icon 
- While pressing the control key, select the two arcs of the C-Mesh domain to the left and the top and bottom edges of the rectangle to the right
- Right click > ***Create Named Selection***
- In the *Details View*, name the edges “inlet”

This will be the velocity inlet of the flow.

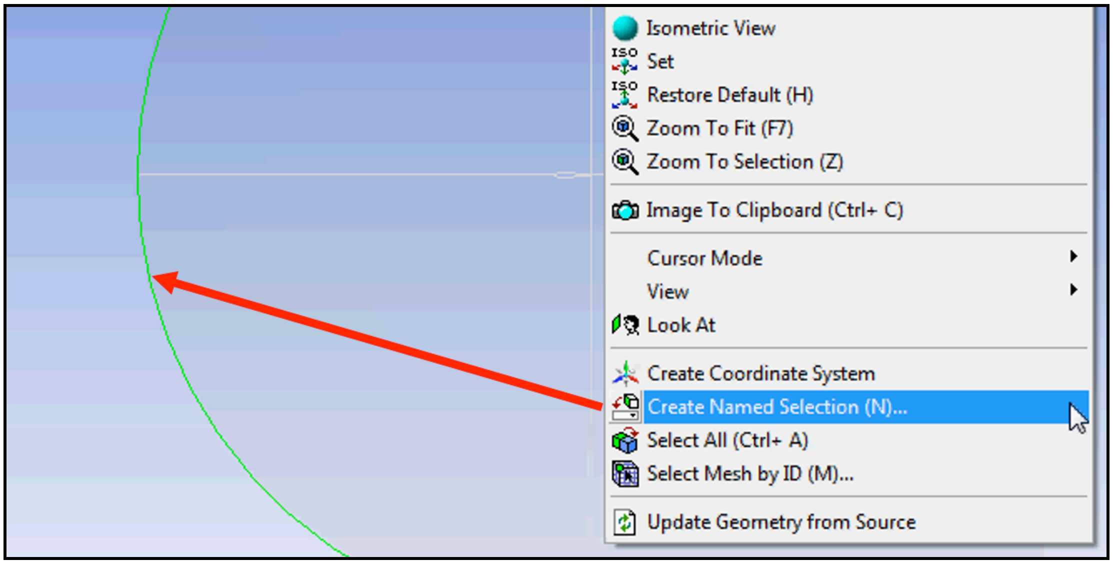
- While pressing the control key, select the two vertical lines of the rectangle to the right
- Right click > ***Create Named Selection***
- In the *Details View*, name the right arc edge “outlet”

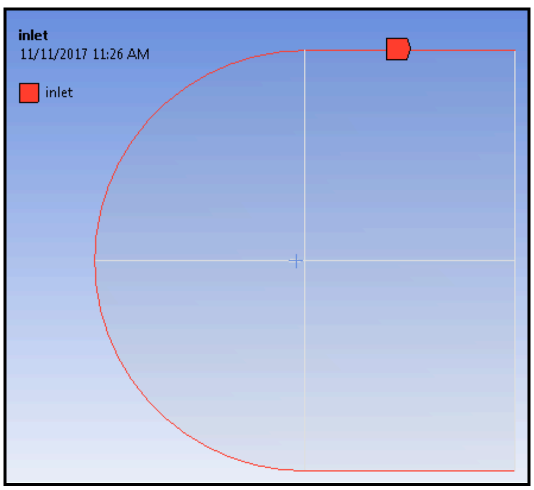
This will be the pressure outlet of the flow.

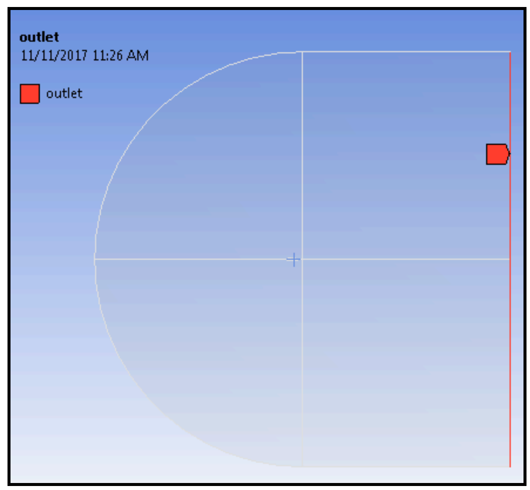
- While pressing the control key, select both the top and bottom edge of the airfoil
- Right click > ***Create Named Selection***
- In the *Details View*, name the airfoil wall edges “airfoil”

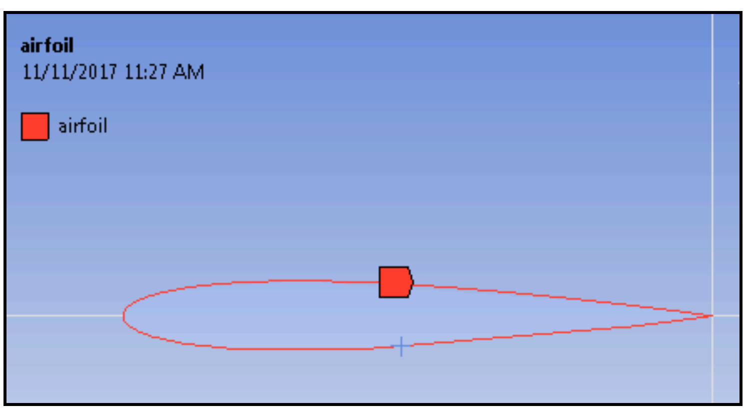
This will be the airfoil wall.

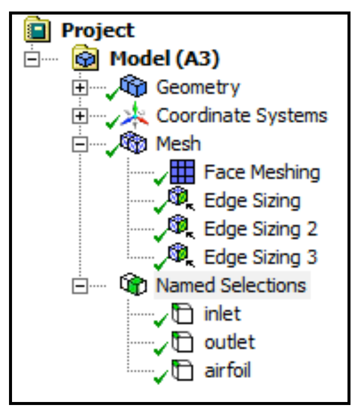
The named selections can be viewed in the *Outline*.


Isometric View
Set
Restore Default (H)
Zoom To Fit (F7)
Zoom To Selection (Z)
Image To Clipboard (Ctrl+ C)
Cursor Mode
View
Look At
Create Coordinate System
Create Named Selection (N)...
Select All (Ctrl+ A)
Select Mesh by ID (M)...
Update Geometry from Source


inlet
11/11/2017 11:26 AM
inlet

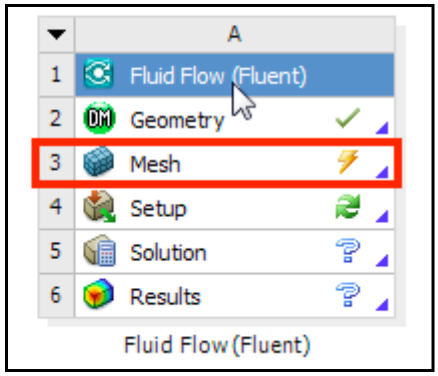

outlet
11/11/2017 11:26 AM
outlet


airfoil
11/11/2017 11:27 AM
airfoil

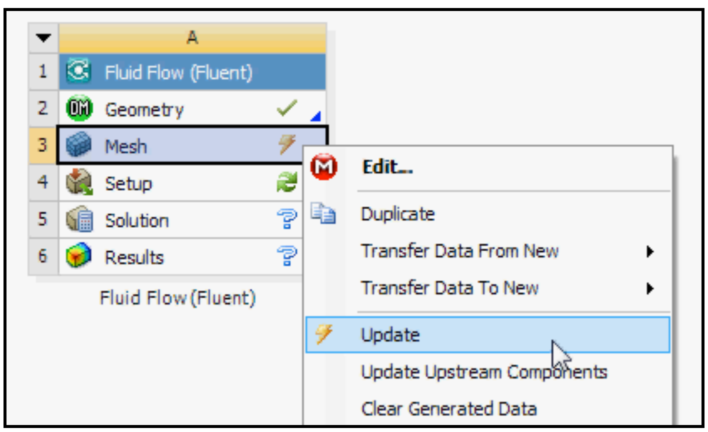

Project
Model (A3)
Geometry
Coordinate Systems
Mesh
Face Meshing
Edge Sizing
Edge Sizing 2
Edge Sizing 3
Named Selections
inlet
outlet
airfoil

- ***File*** > ***Save Project*** and close the Mesher
- Return to the *Workbench Project Schematic*

At this point, *Mesh* should have a lightning mark which signals that the mesh needs to be updated.



- Right Click ***Mesh*** > ***Update***



4. Setup

Now the physics of the fluid flow will be set up.

- Double Click ***Setup*** 

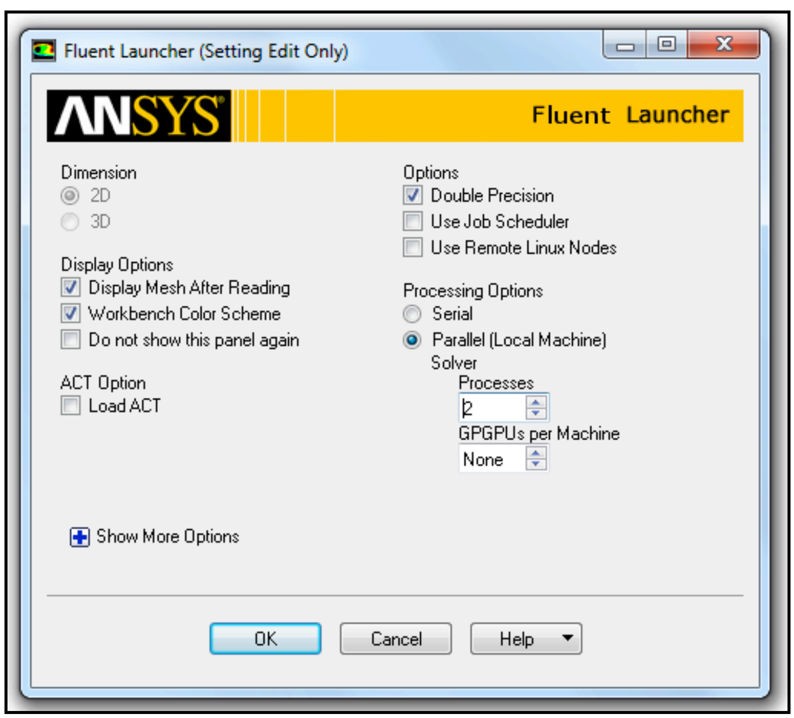
The Fluent Launcher will pop up.

- Check ***Double Precision***

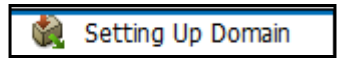
Double precision will produce more accurate results, although it will take the solver longer to calculate the result.

Series / Parallel Processing

- If your computer has more than one core, parallel processing will help Fluent run the calculations faster by splitting the work between the two cores. For a two-core computer, click on ***Parallel***, and under *Processes*, type in 2. The limit of the number of processes is 4.
- Otherwise, choose ***Serial***.
- Click ***OK***

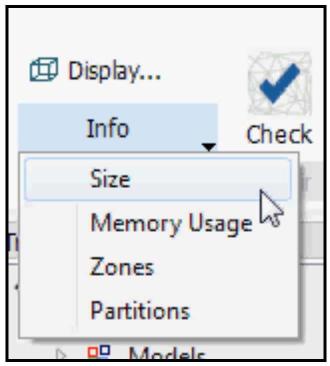


Checking the Mesh
In order to start the setup, the mesh must be verified to make sure that it functions properly.

- In the top toolbox, click ***Setting Up Domain*** 

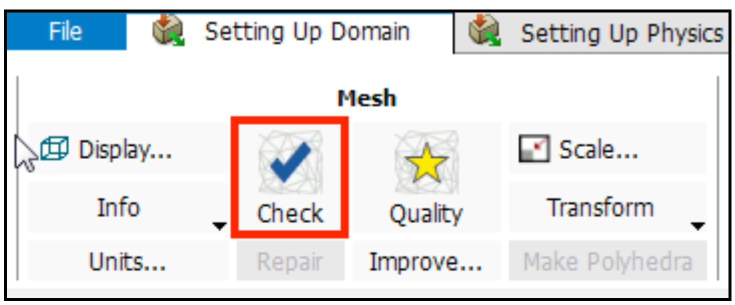
 > ***Mesh*** > ***Info*** > ***Size***

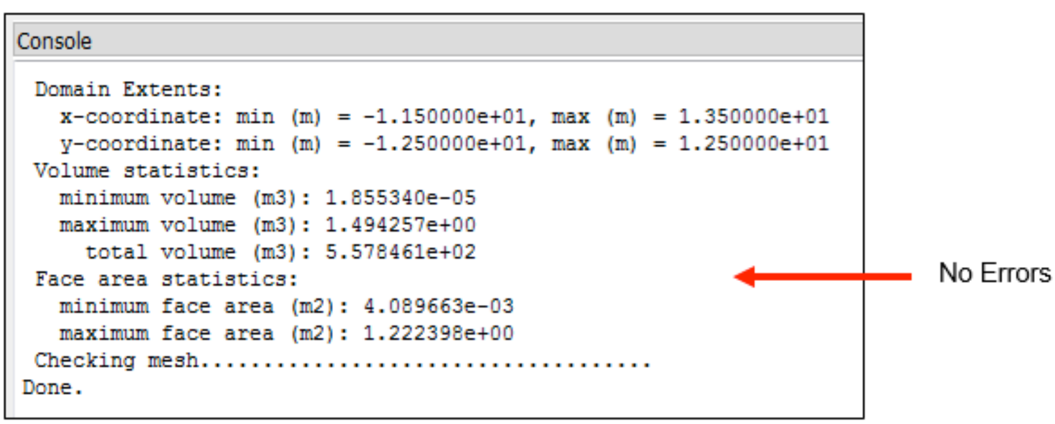
The Console pane should output that there are 15,000 cells in the mesh.



- Under *Mesh*, click ***Check***

If there are no errors in the Console, the mesh is properly functioning.



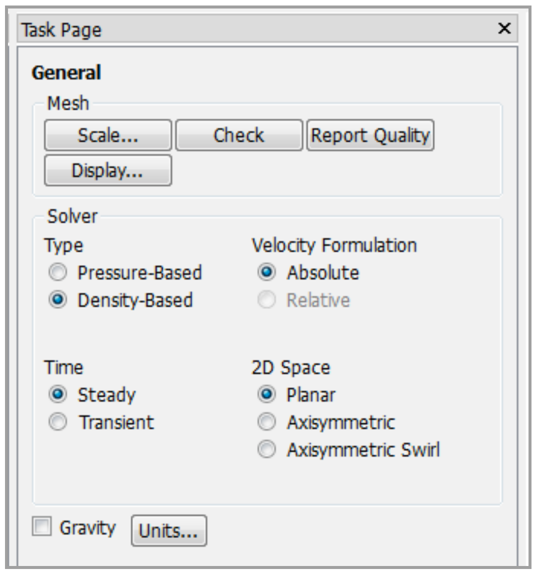


General Setup
To the left is the *Tree* where the settings are located.

- Under the *Tree*, under ***Setup***, double click ***General***
- Under *Solver*, for the *Type*, select ***Density-Based***

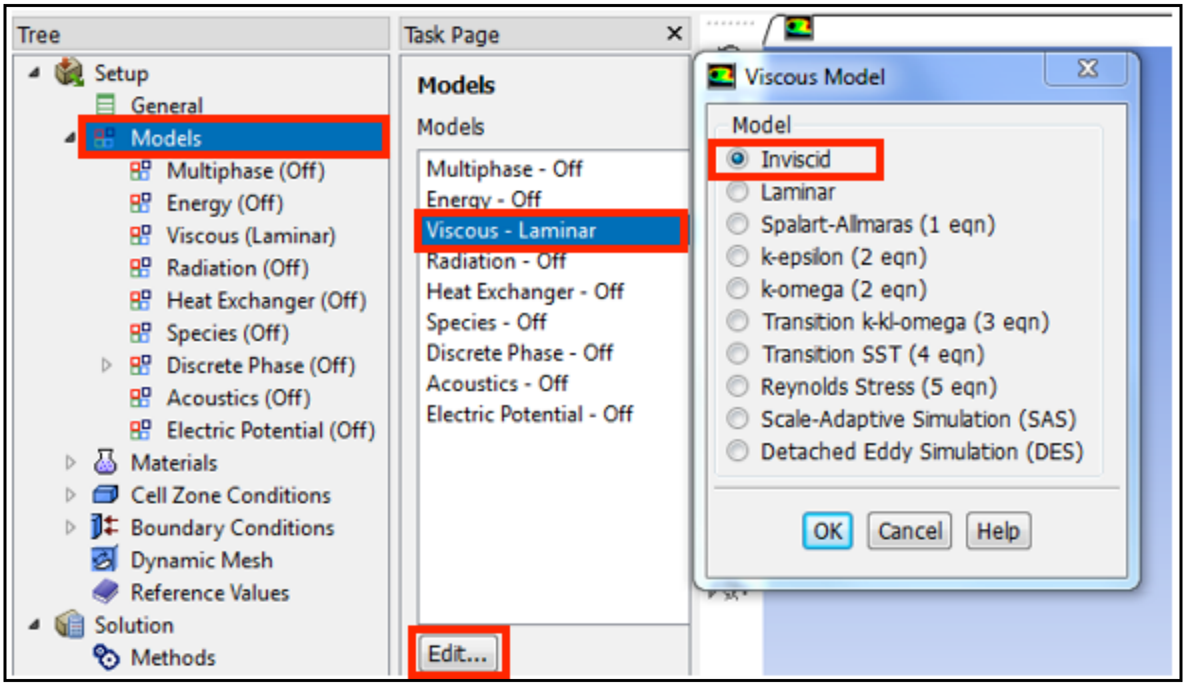
Because the airfoil will undergo inviscid flow, the air is considered incompressible and the density is fixed; therefore, Density-Based type will be used.

Make sure the *Time* is ***Steady*** (default). Transient time is used for unsteady flows.



Models
Different types of flow can be modeled. In this tutorial, the air flow will have no viscosity Therefore, the flow is inviscid.

- Under the *Tree*, under ***Setup***, double click ***Models***
- Select ***Viscous – Laminar*** > ***Edit***
- Under *Model*, select ***Inviscid***
- Click ***OK***



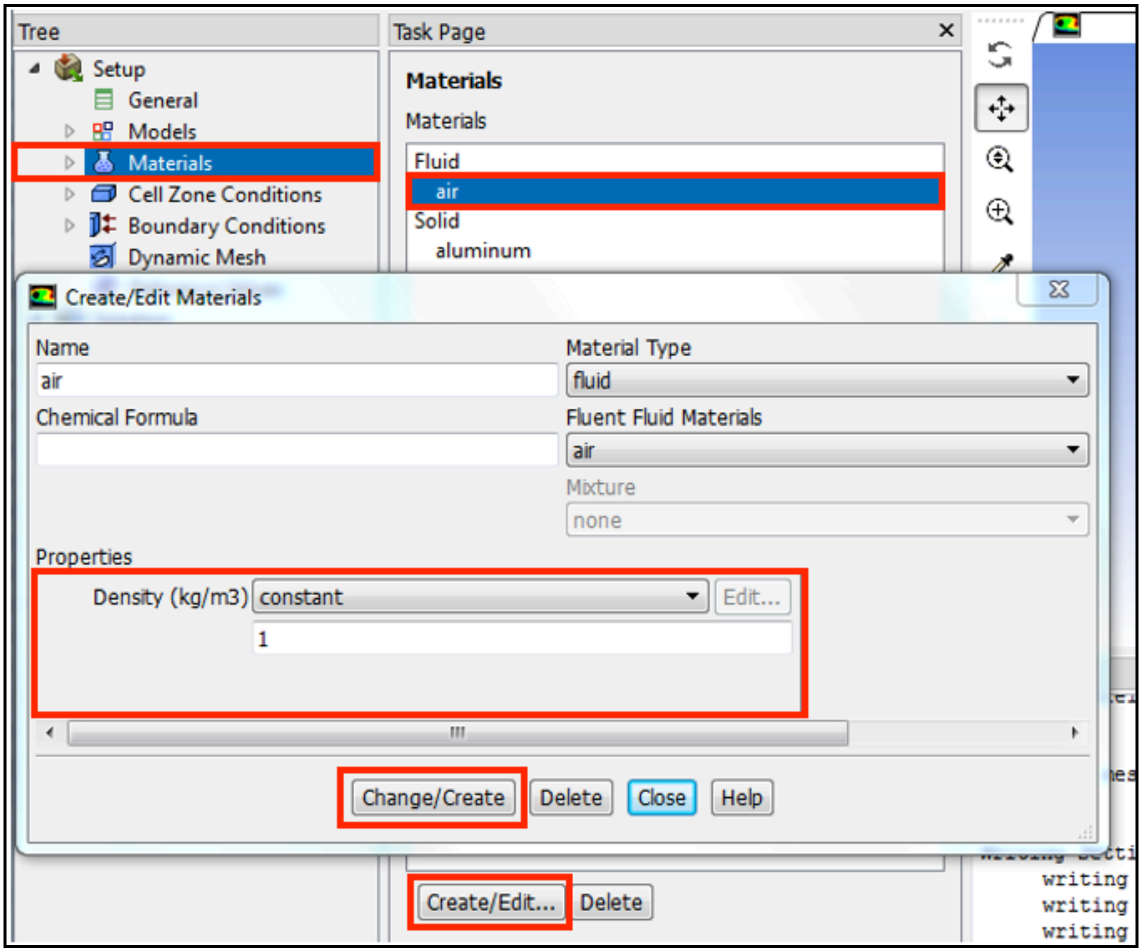
Specifying Material Properties

The fluid and its properties will be specified.

- Under the *Tree*, under ***Setup***, double click ***Materials***
- Select ***Fluid*** > ***Create/Edit***

This brings up a window. The fluid name should be defaulted to air.

- For the *Density*, make sure it is set to ***constant***
- Under *Density*, type in “1” kg/m^3
- Click ***Change/Create*** and ***Close***



Boundary Conditions
Inlet:

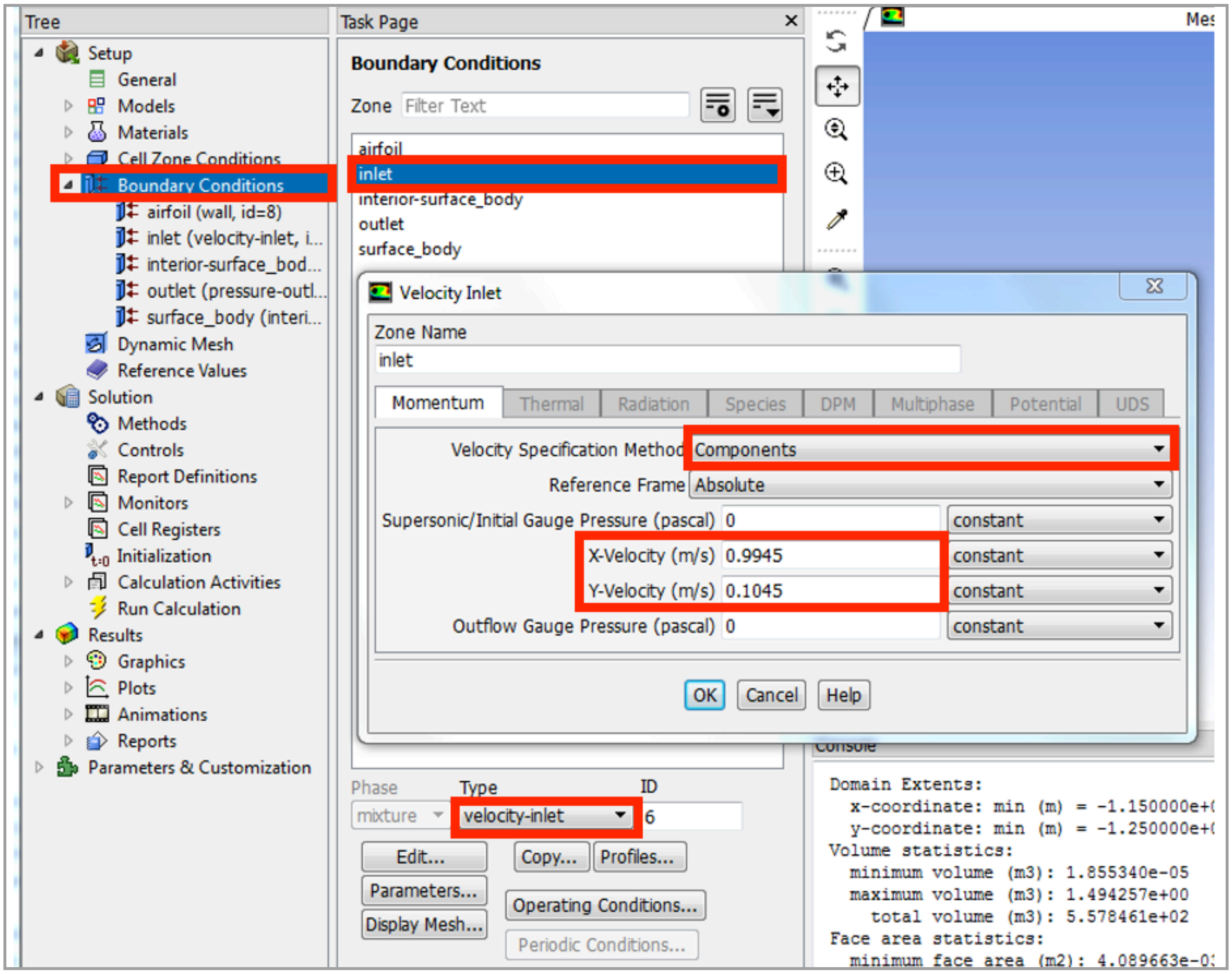
- Under the *Tree*, under ***Setup***, double click ***Boundary Conditions***
- Under *Zone*, select ***inlet***

This inlet name is based on the named selection created during the meshing stage.

- Set *Type* > ***velocity-inlet*** (should already be set)
- Click ***Edit...***
- Set *Velocity Specification Method* to ***Components***
- For the *X-Velocity*, type in “0.9945” m/s
- For the *Y-Velocity*, type in “0.1045” m/s

These values are chosen because the air velocity is 1 m/s at a 6 degree angle of attack, and therefore the x component is cos (6°) and the y component is sin (6°).

- Click ***OK***



Outlet:

- Under the *Tree*, under ***Setup***, double click ***Boundary Conditions*** (already done)
- Under *Zone*, select ***outlet***

This outlet name is based on the named selection created during the meshing stage.

- Set *Type* > ***pressure-outlet*** (should already be set)
- Click ***Edit...***
- Make sure the *Gauge Pressure (pascal)* is 0 Pa
- Click ***OK***

Airfoil:

- Under the *Tree*, under ***Setup***, double click ***Boundary Conditions*** (already done)
- Under *Zone*, select ***airfoil***

This airfoil name is based on the named selection created during the meshing stage.

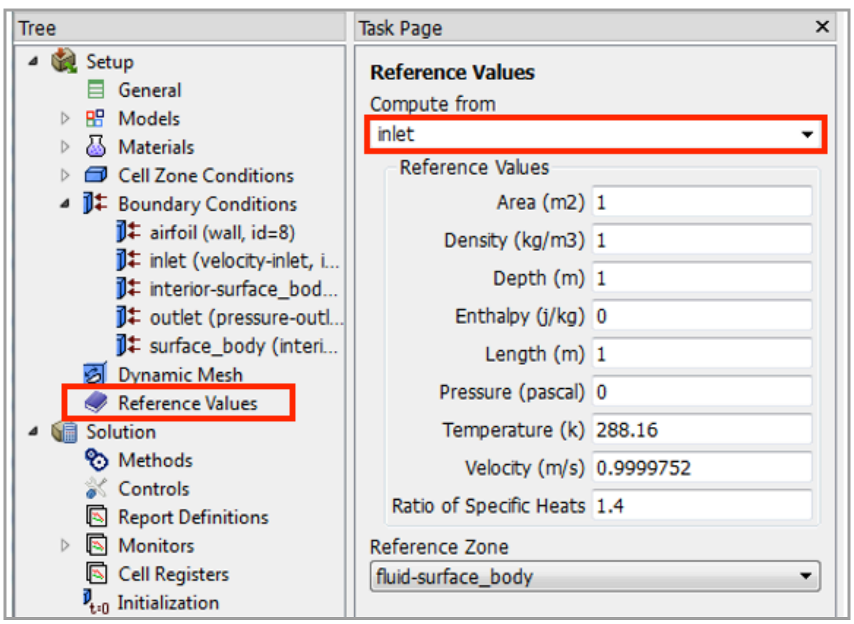
- Set *Type* > ***wall*** (default)

Reference Values
Here, the reference values that will be used to calculate the solution will be defined.

- Under the *Tree*, under ***Setup***, double click ***Reference Values***
- Under *Compute from*, select ***inlet***

This will set the density to 1 and velocity to 0.9999752 (calculated from cos (6°) and sin (6°))

- Make sure the *Reference Zone* is ***fluid-surface_body***

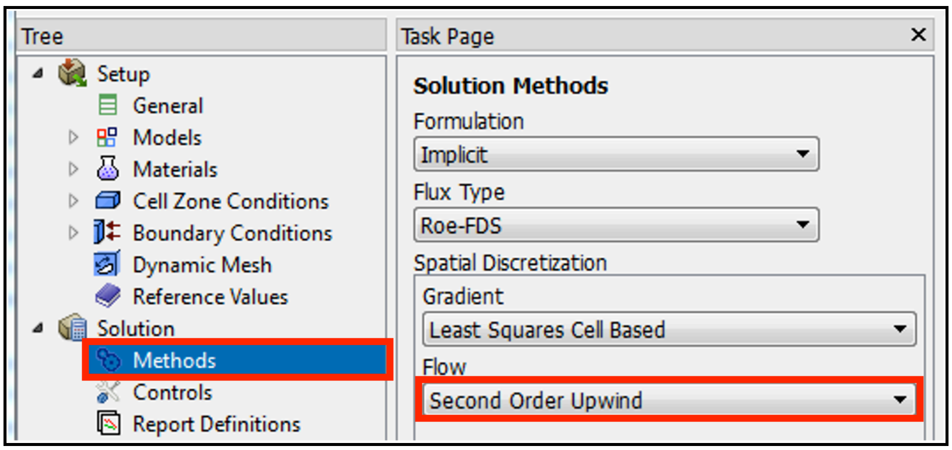


5. Solution

Preparations for calculation will be made.

- Under the *Tree*, under ***Solution***, double click ***Methods***
- Under *Spatial Discretization*, set *Flow* to ***Second Order Upwind*** (default)

The Upwind scheme uses values upstream to calculate values at the center of cells. Compared to First Order, Second Order requires more time to converge but produces more accurate results.



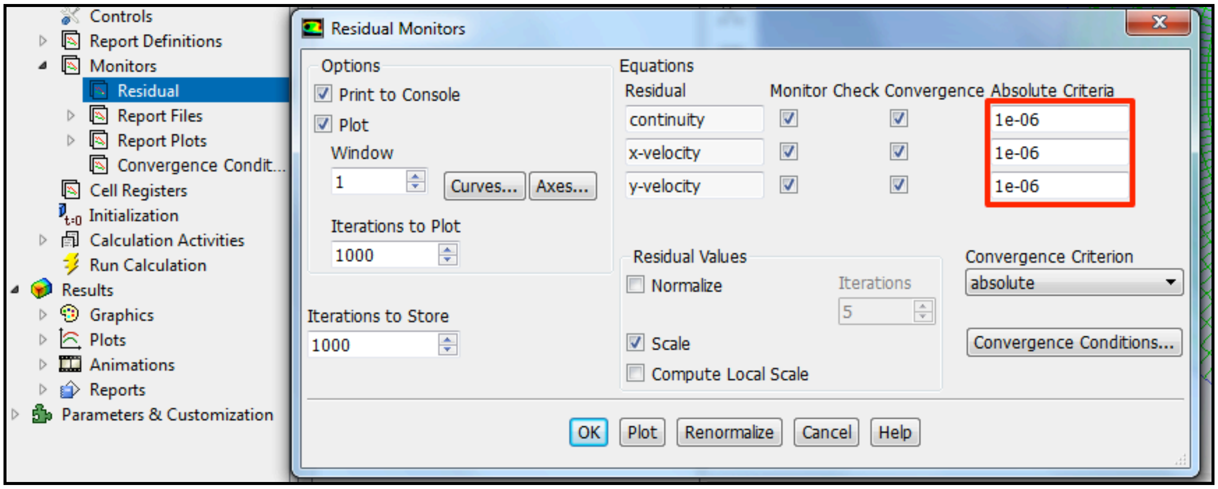
Convergence Criterion

The convergence criterion establishes how small the difference of the values produced by two iterations must be in order for the calculation to be considered converged.

- Under the *Tree*, under ***Solution*** > ***Monitors***, double click ***Residual***

This should automatically pop up a window.

- For *Absolute Criteria* for *continuity*, *x-velocity*, and *y-velocity*, type in “1e-6”
- Click ***OK***

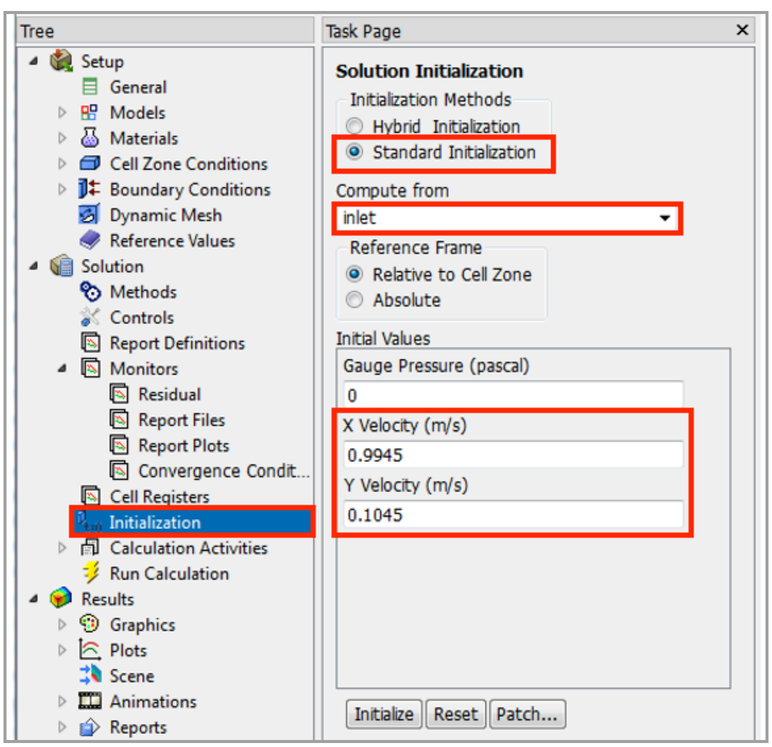


Initialization

- Under the *Tree*, under ***Solution***, double click ***Initialization***
- Under *Initialization Methods*, select ***Standard Initialization***
- Under *Compute From*, select ***inlet***

Alternatively, you can simply set X Velocity to “0.9945” m/s and Y Velocity to “0.1045” m/s

- Click ***Initialize***

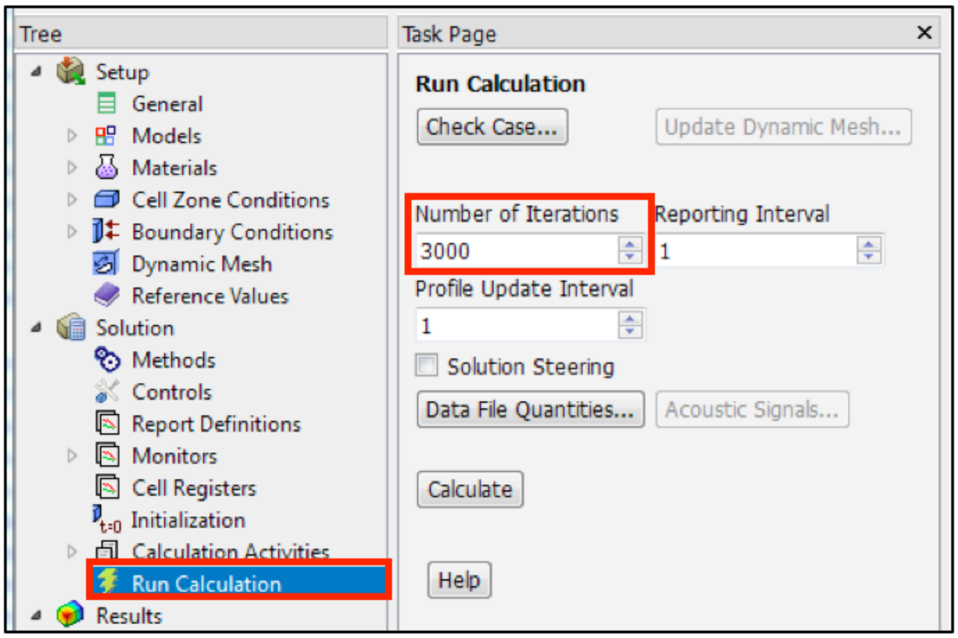


Iterating Until Convergence

- Under the *Tree*, under ***Solution***, double click ***Run Calculation***
- For the *Number of Iterations*, type in “3000”

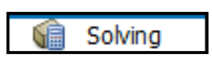
NOTE: If you want to create an animation, move onto the next section before clicking Calculate

- Click ***Calculate***

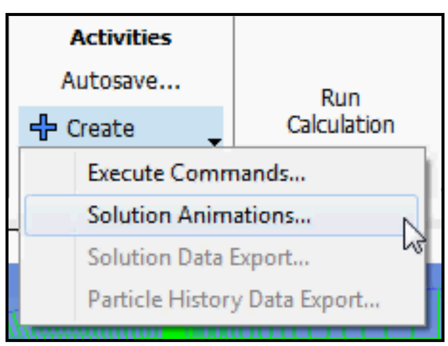


Video Animation

This creates an animation of various results using a static image of the results taken every set number of iterations. The animation setup can only be done after initialization.

- In the top toolbox, click ***Solving***  > ***Activities*** > ***Create*** > ***Solution Animations…***

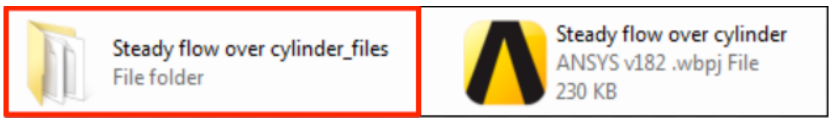
This will pop up an *Animation Definition* window



- For *Record after every iteration*, type in “25”

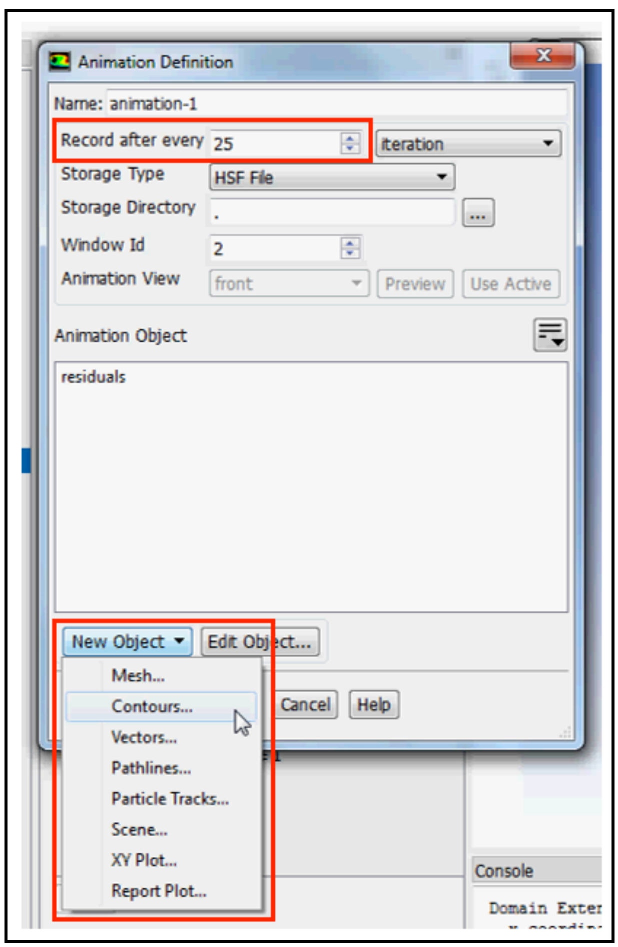
The software will take a static image of the animation object after every 25 iterations. This number can be any number, however because the total number of iterations is 3000, it is inadvisable to record every 1 iteration.

To save the animation records on a location outside the solver, click on the three dots  besides the *Storage Directory*. If no location is selected, the images will be stored by default inside the project ***files folder*** > ***dp0*** > ***FFF*** > ***Fluent***



- Click ***New Object*** > ***Contours***

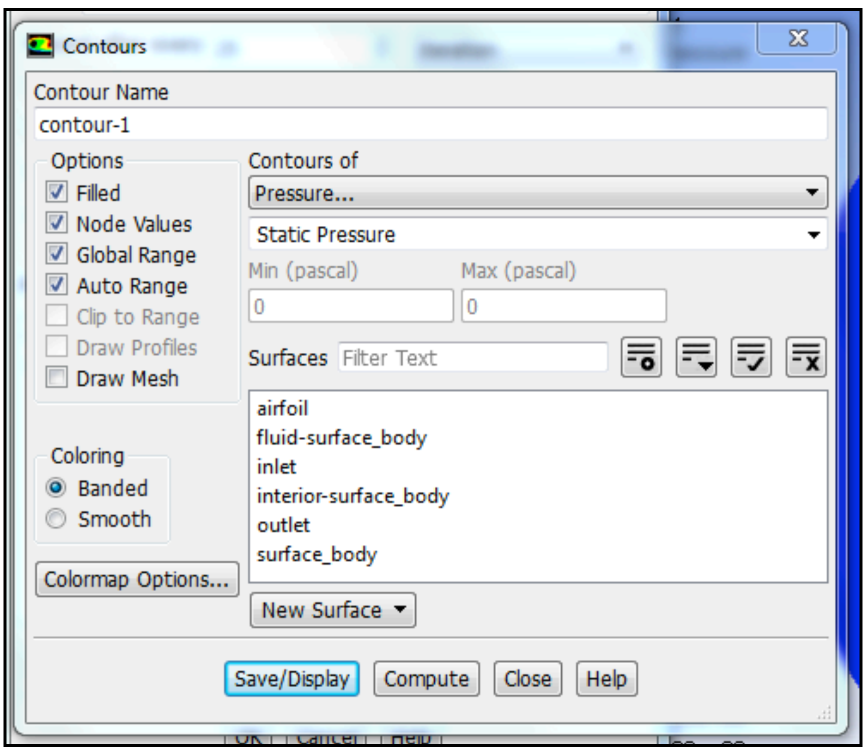
This creates a new contour which will be animated.



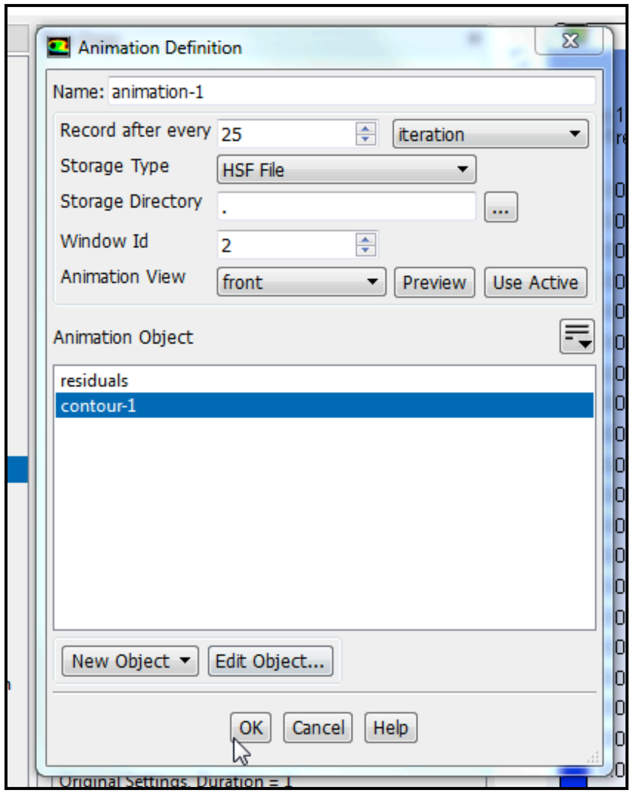
- In order to animate a contour of static pressure, under Contours of, choose ***Pressure***
- Underneath, choose ***Static Pressure***
- Under *Options*, check ***Filled***

This will fill in areas where the pressure is approximately the same by color.
Make sure under *Surfaces*, no surface is highlighted to show all surfaces.

- To obtain a more detailed contour, click ***Colormap Options***
- For *Colormap Size*, type in "100" and click ***Apply*** and then ***Close***
- On the *Contours* window, click ***Save/Display***
- Click ***Save/Display***
- Click ***Close***



- In the *Animation Definition* window, under *Animation Object*, select the contour just created



- Click ***OK***
- Under the *Tree*, under ***Solution***, double click ***Run Calculation***
- Click ***Calculate***

The results of the simulation can be viewed within the Solution.

Animation

Once the calculations are done, the animation can be played back.

- Under the *Tree*, under ***Results*** > ***Animations***, double click ***Solution Animation Playback***
- Under *Animation Sequences*, select the animation corresponding to the contour
- Click the ***Play Button***  to view the animation in the window

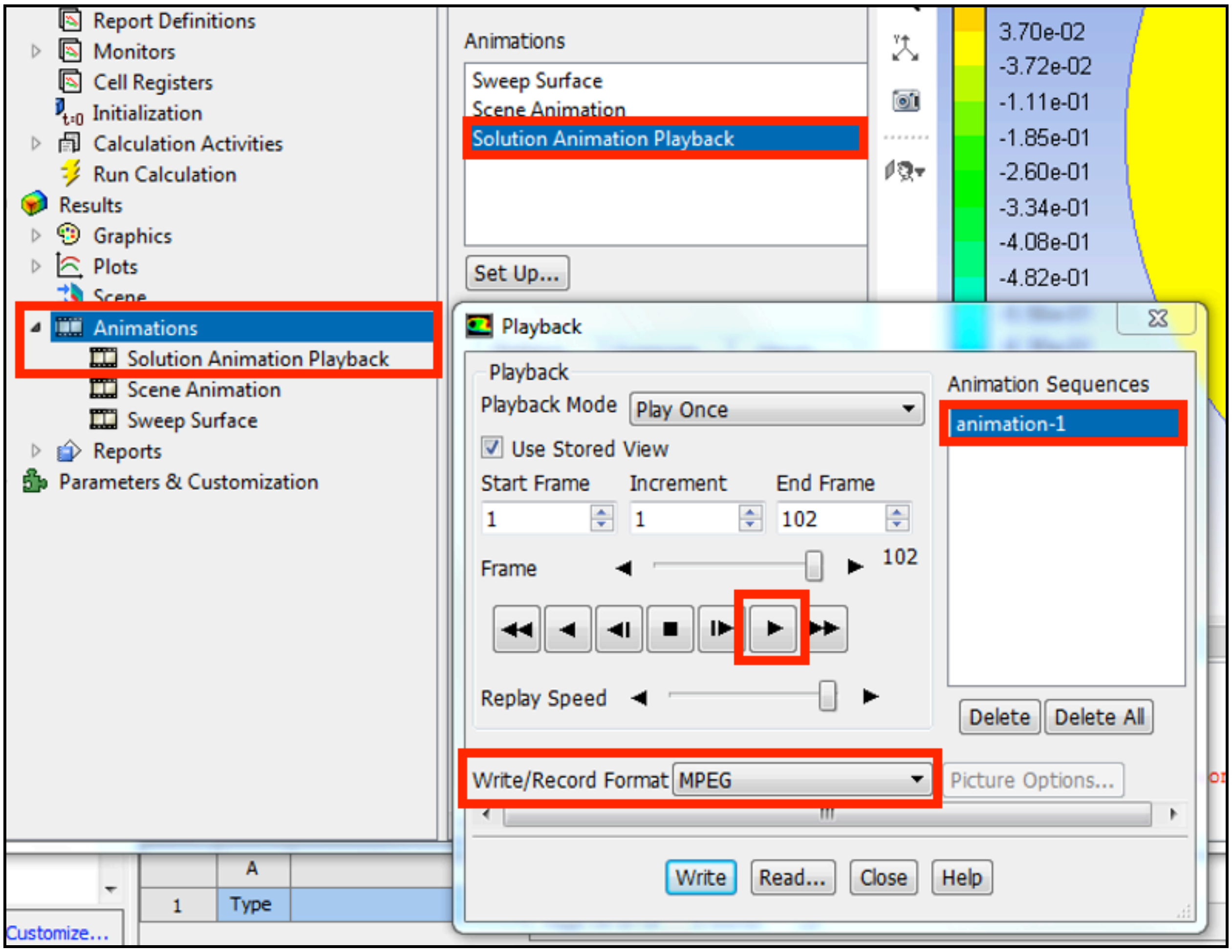
You may have to select a different tab within the window to find contour.

Different tabs

In order to save the animation, each frame can be saved individually as Animation Frames or Picture Files. Alternatively, the animation can be saved as an animation by choosing MPEG.

- For *Write/Record Format*, choose ***MPEG***
- Click ***Write*** and ***Close*** after the file is written
- On your desktop, go to project ***files folder*** > ***dp0*** > ***FFF*** > ***Fluent***

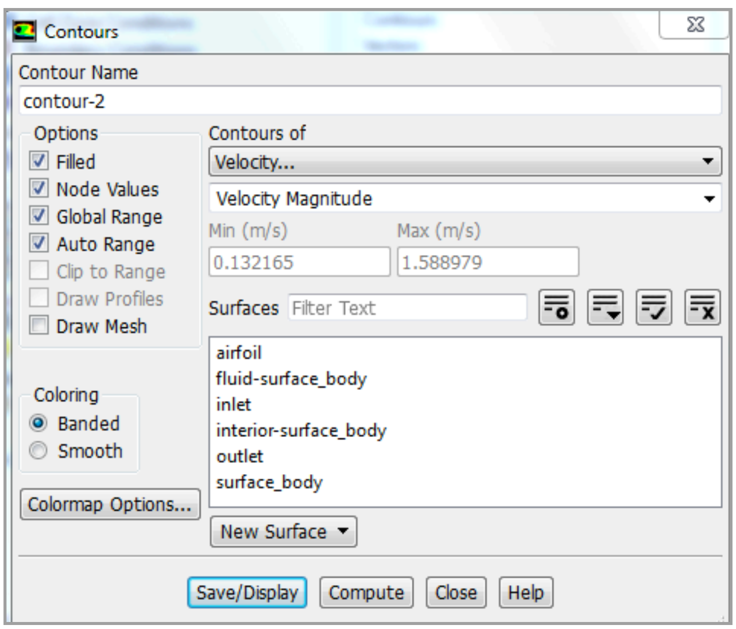
Inside will be an MPEG file containing the animation.

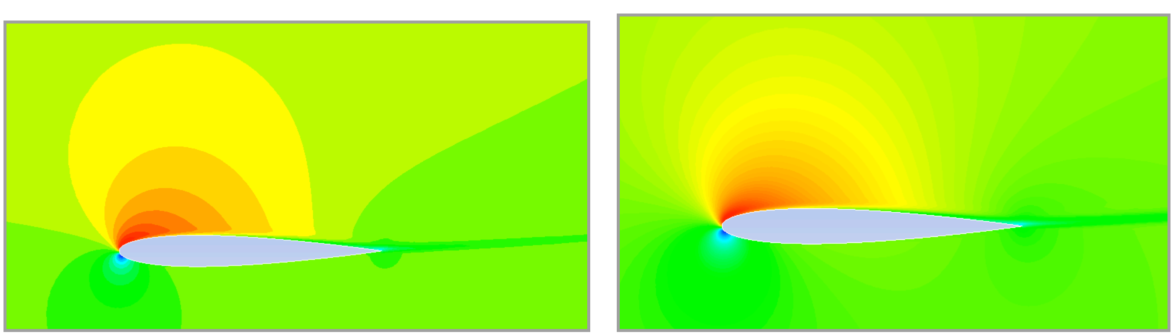


Contours

You can view the contours of the velocity.

- Under the *Tree*, under *Results* > *Graphics*, right click ***Contours*** and select ***New***
- Under *Contours of*, choose ***Velocity…***
- Underneath, choose ***Velocity Magnitude***
- Check ***Filled***
- Make sure none of the *Surfaces* is selected, and click ***Save/Display***
- To obtain a more detailed contour, click ***Colormap Options***
- For *Colormap Size*, type in 100 and click ***Apply*** and then ***Close***
- On the *Contours* window, click ***Save/Display*** and ***Close***

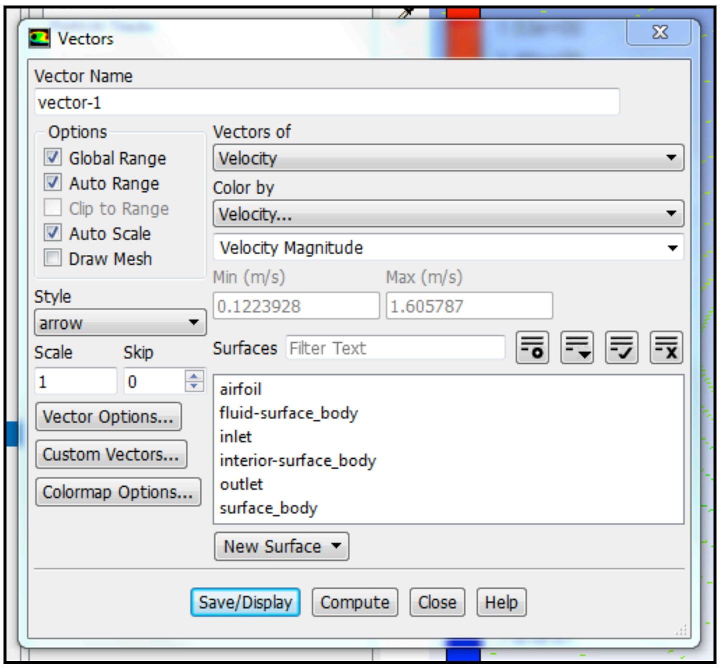


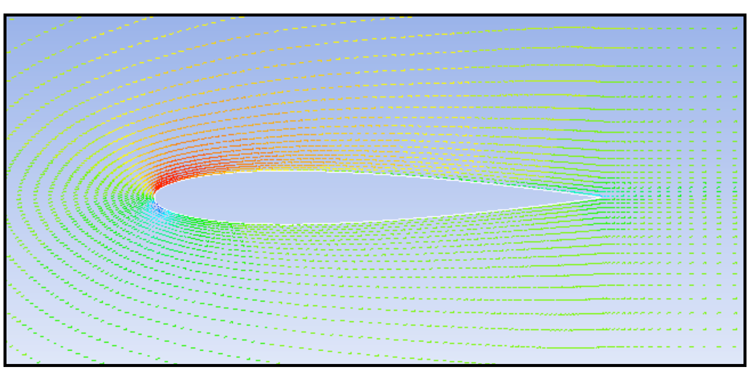


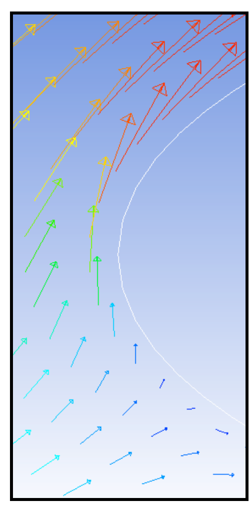
Colormap Size 20 (left) vs. 100 (right)

Vectors
The vectors of velocity can be viewed.

- Under the *Tree*, under *Results* > *Graphics*, right click ***Vectors*** and select ***New***
- Under *Vectors of*, choose ***Velocity***
- Under *Color by*, choose ***Velocity…***
- Underneath, choose ***Velocity Magnitude***
- Make sure none of the *Surfaces* is selected, and click ***Save/Display*** and ***Close***







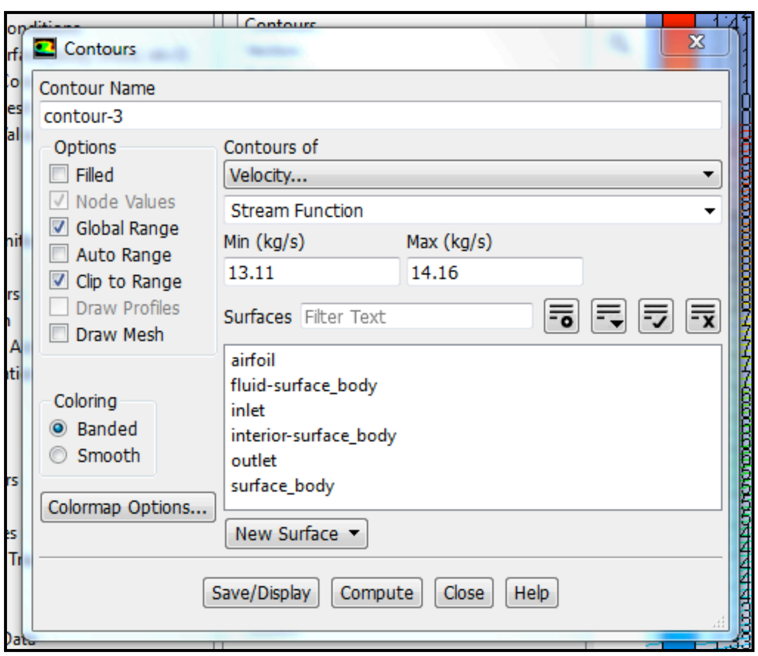
Stream Function

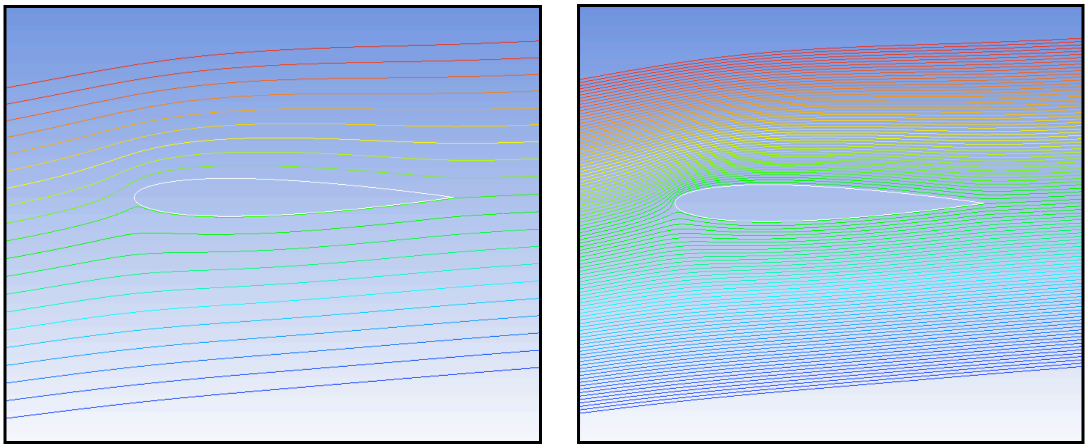
Streamlines can be viewed using contours.

- Under the *Tree*, under *Results* > *Graphics*, right click ***Contours*** and select ***New***
- Under *Contours of*, choose ***Velocity…***
- Underneath, choose ***Stream Function***
- Uncheck ***Filled*** (default)
- Uncheck ***Auto Range***
- Under *Min (kg/s)*, type in “13.11”
- Under *Max (kg/s)*, type in “14.16”

This defines the range of the stream function using mass flow rate. By changing the minimum and maximum values, the stream function will only be displayed for areas where the mass flow rate is between 13.11 kg/s and 14.16 kg/s, which is near the airfoil.

- Make sure none of the *Surfaces* is selected, and click ***Save/Display***
- To obtain a more detailed contour, click ***Colormap Options***
- For *Colormap Size*, type in 100 and click ***Apply*** and then ***Close***
- On the *Contours* window, click ***Save/Display*** and ***Close***



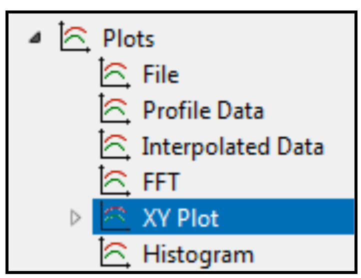


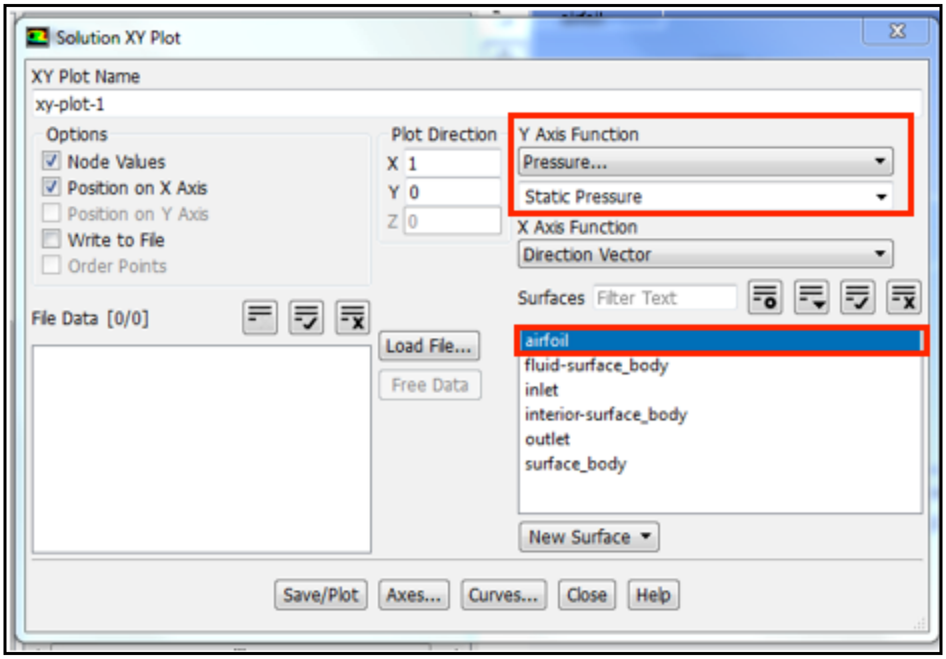
Colormap Size 20 (left) vs. 100 (right)

Pressure Coefficient vs. Position Graph

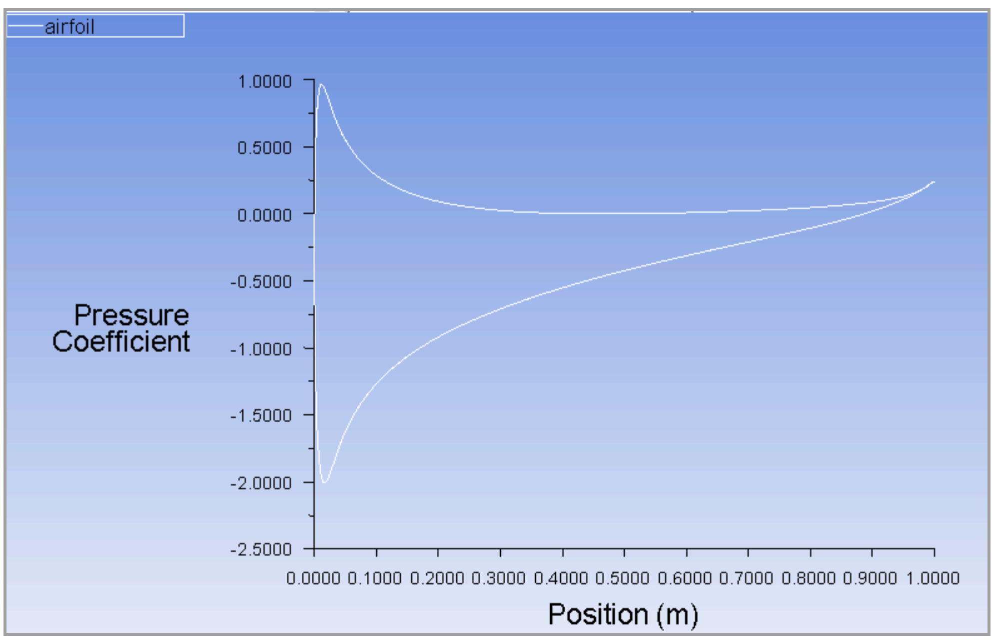
The plot of the pressure coefficient can be displayed.

- Under the *Tree*, under *Results*, under *Plots*, double click ***XY Plot***
- Under the *Y Axis Function*, choose ***Pressure…***
- Underneath, choose ***Pressure Coefficient***
- Under *Surfaces*, select and highlight the ***airfoil***





- Click ***Save/Plot***



The data can be saved in a text file.

- Under *Options*, check ***Write to File***

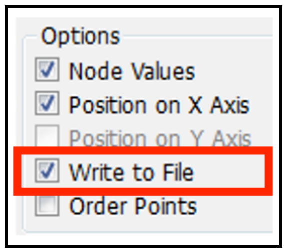
This changes the Save/Plot button to Write…

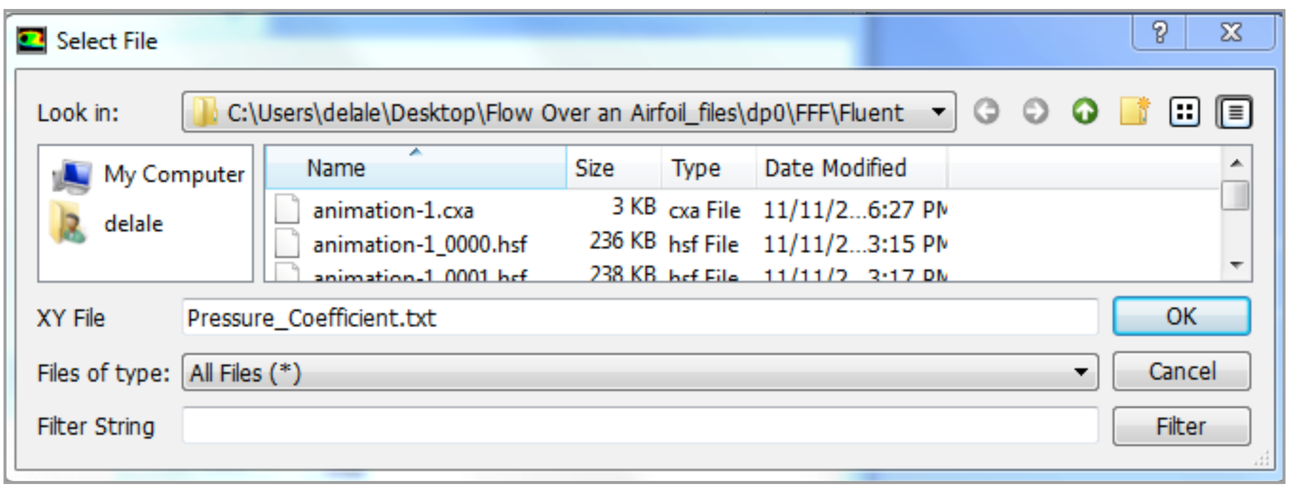
- Click ***Write…***

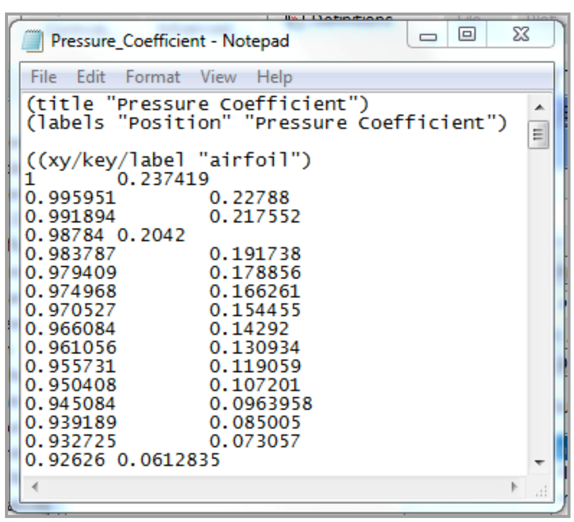
- Select a location to save the file

If no different location is selected, the images will be stored by default inside the project ***files folder*** > ***dp0*** > ***FFF*** > ***Fluent***

- Change *Files of Type:* to ***All Files (*)***
- Change the file name by *XY File* to ***Pressure_Coefficient.txt***
- Click ***OK***





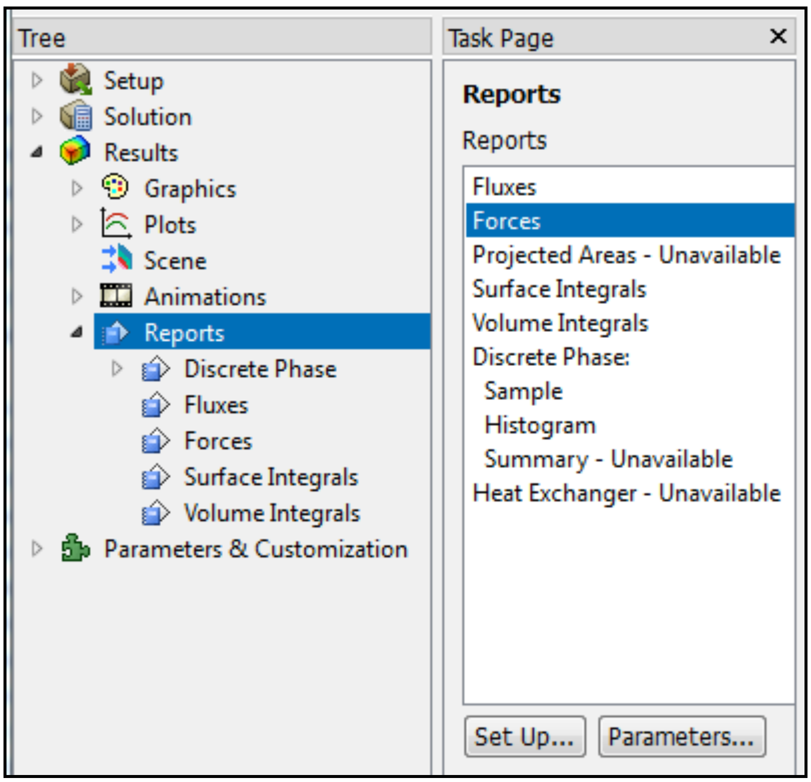


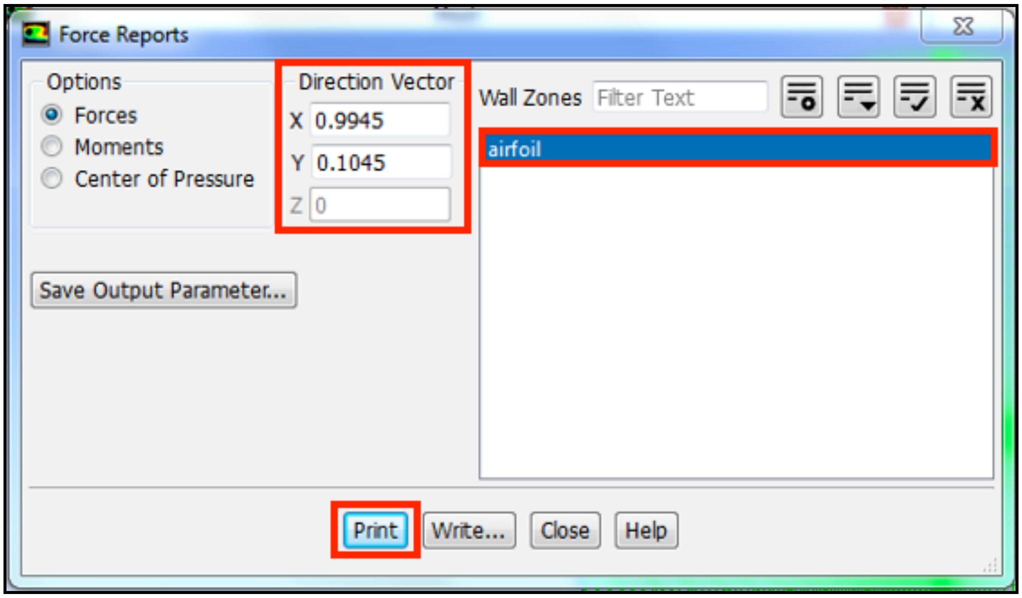
Coefficient of Drag

The drag is the force that opposes the relative motion of the object. Therefore, to calculate drag, the force along the direction of movement is used.

- Under the *Tree*, under *Results*, double click *Reports* and choose ***Forces***
- Click ***Set Up…***
- Under the *Direction Vector*, for X, type in “0.9945”
- Under the *Direction Vector*, for Y, type in “0.1045”

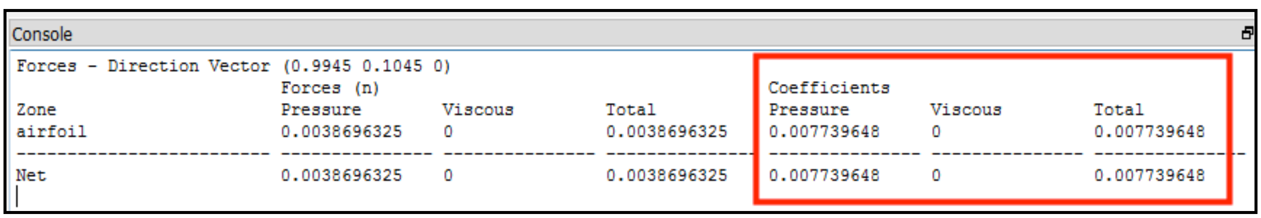
This will output the forces in the direction of the angle of attack of 6°.





- Click ***Print***

This will output the force reports in the *Console*.



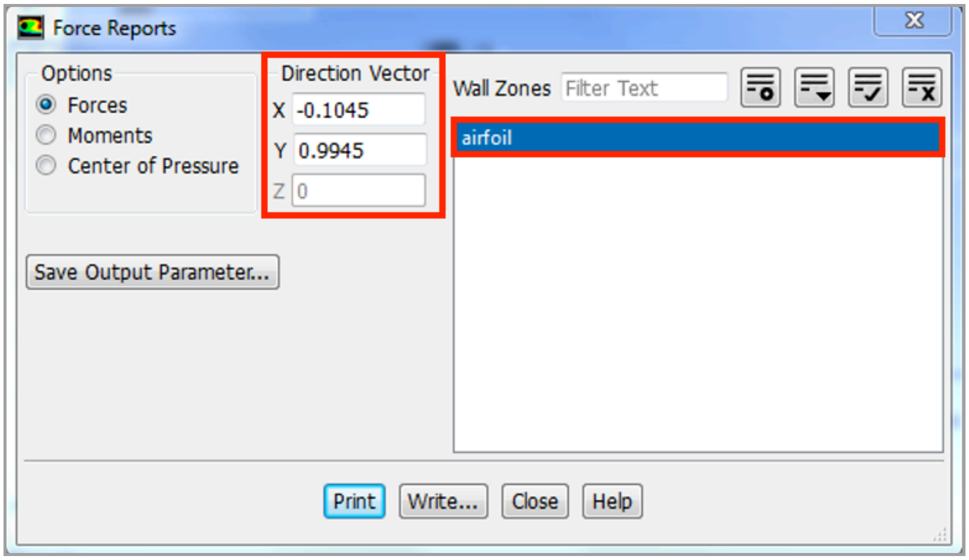
The coefficient of drag for this particular calculation is 0.007739648

Coefficient of Lift

The lift uses the same function; however, the lift force is perpendicular to the direction of movement. Therefore, the direction must be adjusted to face perpendicular to the angle of attack.

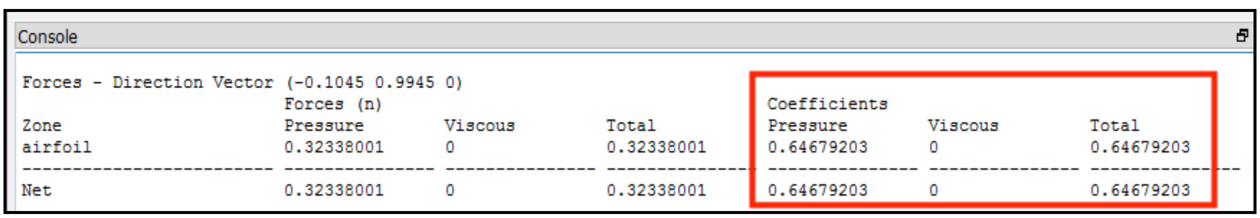
- Under the *Direction Vector*, for X, type in “-0.1045”
- Under the *Direction Vector*, for Y, type in “0.9945”

This will output the forces in the direction perpendicular to the angle of attack of 6°.



- Click ***Print***

This will output the force reports in the *Console*.



Console

Forces - Direction Vector (-0.1045 0.9945 0)

Zone	Forces (n) Pressure	Viscous	Total	Coefficients Pressure	Viscous	Total
airfoil	0.32338001	0	0.32338001	0.64679203	0	0.64679203
Net	0.32338001	0	0.32338001	0.64679203	0	0.64679203

The coefficient of lift for this particular calculation is 0.64679203

6. Results

The results can be formally viewed in the Results section.

- Double click ***Results*** 

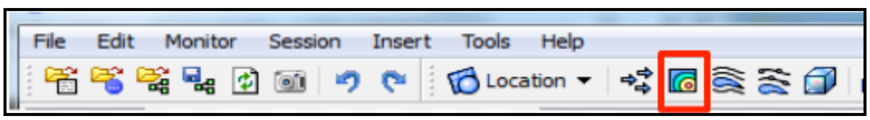
This opens up CFD-Post with the data from the solution already imported.

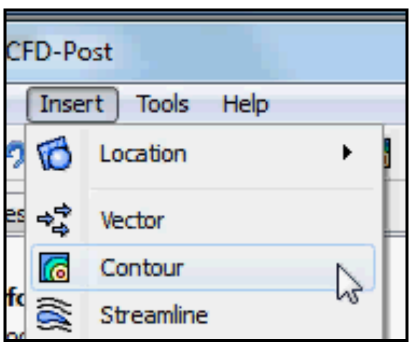
Pressure Contour
We will view a pressure contour.

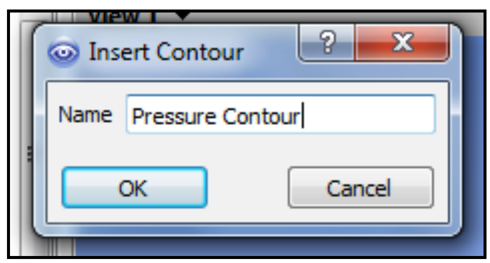
- Click ***Insert*** > ***Contour***, name the contour “Pressure Contour” in the pop-up window, and click ***OK***

Do not name the contour “contour” as it has the same name as the function, which cannot be done.

Alternatively, click on the Contour icon







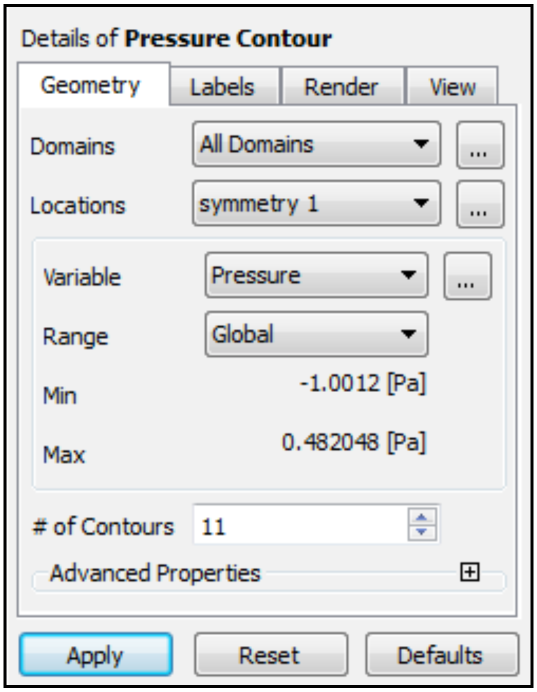
On the bottom left side, a *Details* Window will come up.

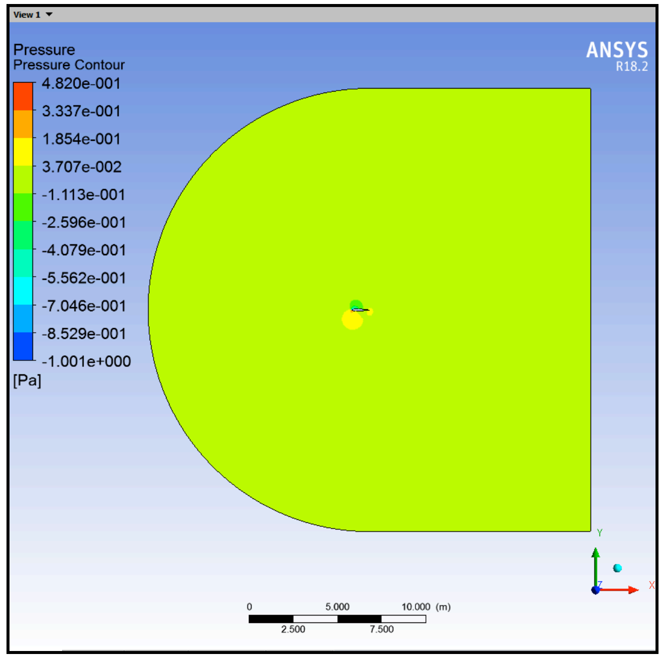
- Set *Locations* to ***symmetry 1***
- Set *Variable* to ***Pressure***

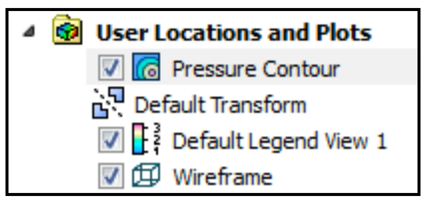
The number of contours is defaulted at 11; this can be increased for a more detailed contour.

- Click ***Apply***

This will create a new contour that can be viewed in the *Outline*.







Velocity Contour

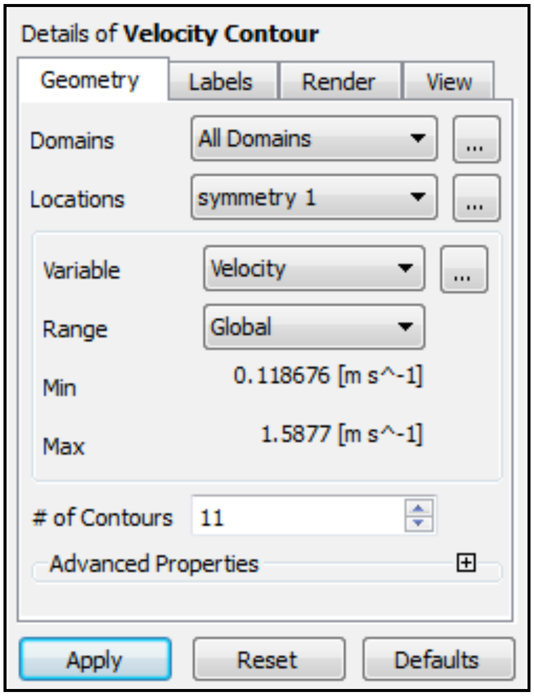
We will view a velocity contour.

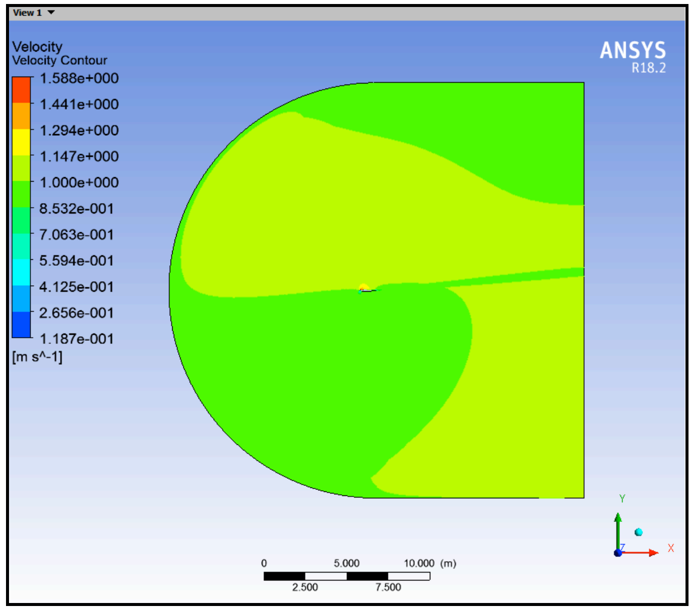
- Click ***Insert*** > ***Contour***, name the contour "Velocity Contour" in the pop-up window, and click ***OK***
- In *Details*, set *Locations* to ***symmetry 1***
- Set *Variable* to ***Velocity***

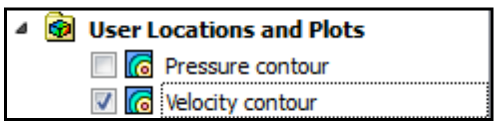
The number of contours is defaulted at 11; this can be increased for a more detailed contour.

- Click ***Apply***
- Uncheck the ***Pressure Contour*** found under the *Outline* and check the ***Velocity Contour***

This will allow only the velocity contour to be displayed.







Comparing Contours

To compare the pressure and velocity contours, the *View* window can be separated into two parts.

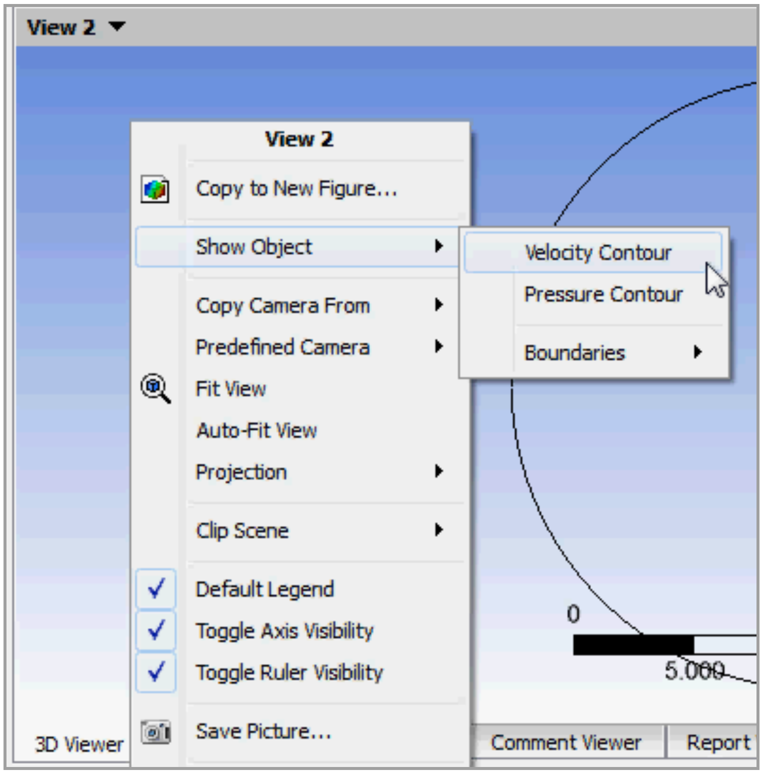
- Click on the ***View*** icon and select the image of a horizontal double view 
- Select the ***Synchronize camera in displayed view*** icon 

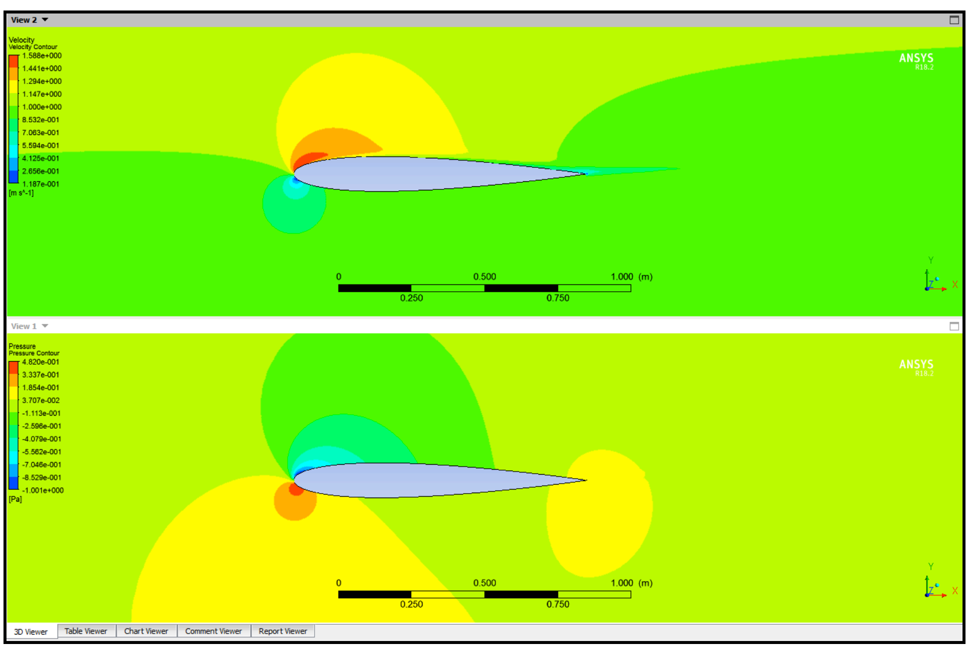
When the camera is synchronized, if the image in one view is zoomed in or shifted, the image in the other view will also zoom in or shift, making comparisons easier.

- Deselect the ***Synchronize visibility in displayed view*** icon 

When the visibility is synchronized, both the top and bottom views will only be view of one contour, preventing both contours from being shown.

- Right click in each view, select ***Show Object*** and select the contour to display





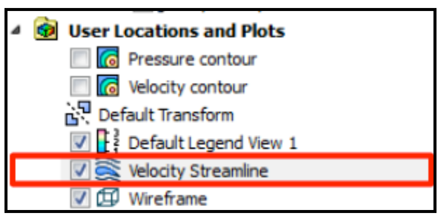
Velocity Contour on top, Pressure Contour on bottom

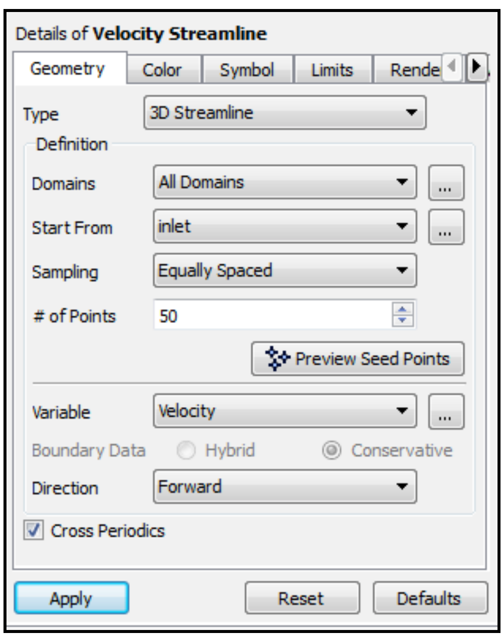
Streamlines

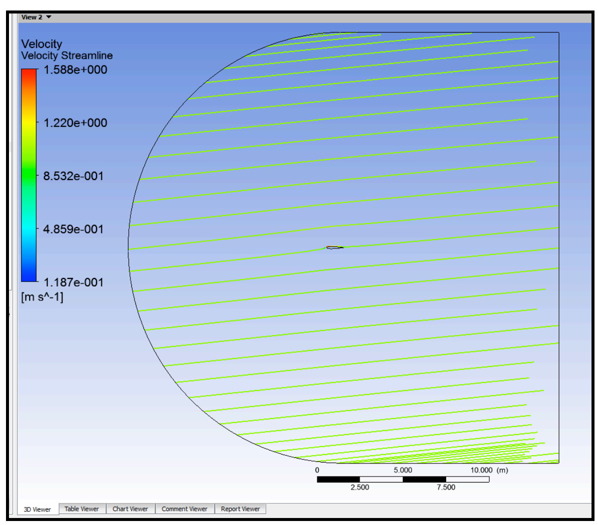
- Click ***Insert*** > ***Streamline***, name the streamline “Velocity Streamline” in the pop-up window, and click ***OK***
- In *Details*, set *Start From* to ***inlet***
- Set *Sampling* to ***Equally Spaced***
- For *# of Points*, type in “50”

This will create 50 lines.

- Set *Variable* to ***Velocity***
- Click ***Apply***
- Uncheck the pressure contour and velocity contour found under the outline and check the Velocity Streamline

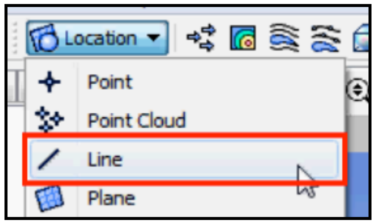






In order to make the streamlines originate from a point closer to the airfoil, a seedline can be placed. The seedline will restrict the area where the streamlines are, and therefore the 50 points can be more concentrated near the cylinder.

- Click on ***Location*** > ***Line***, name the line in the pop-up window “Seedline”, and click ***OK***



- Under *Details*, under *Method*, select ***Two Points*** (default)

This will create a line based on two coordinate points.

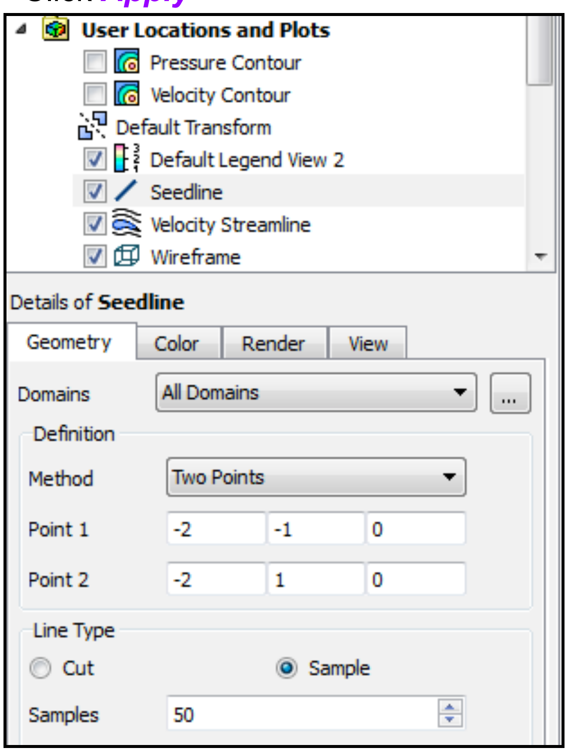
- For *Point 1*, type in “-2”, “-1”, and “0” (x, y, z values in meters)
- For *Point 2*, type in “-2”, “1”, and “0”

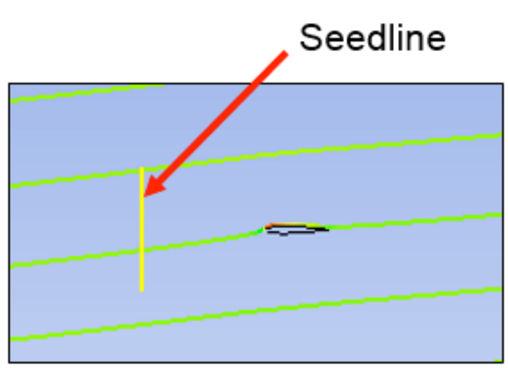
This will create a line 2 meters to the left of the tail of the airfoil, where the origin of the coordinate axis is, with a length spanning from 1 meter above the tail to 1 meter below.

- For the *Samples*, type in “50”

This sample number determines the number of particles released.

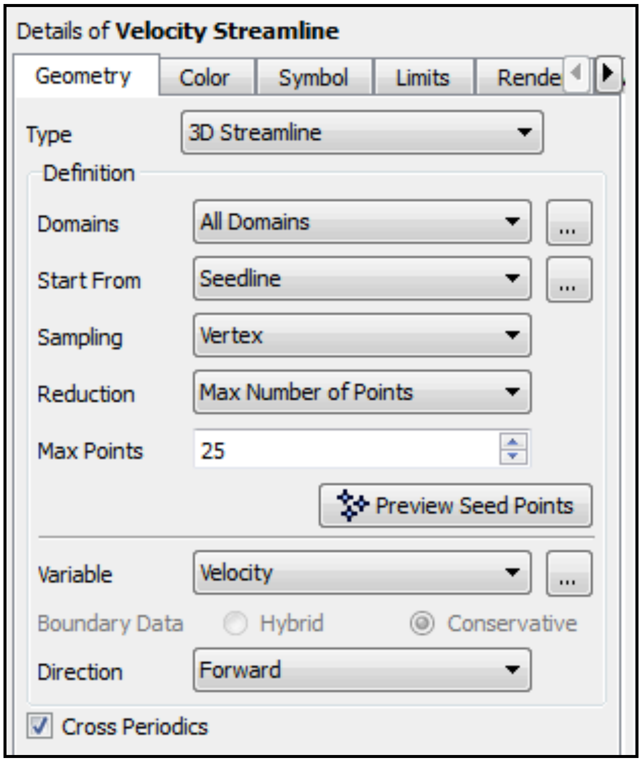
- Click ***Apply***

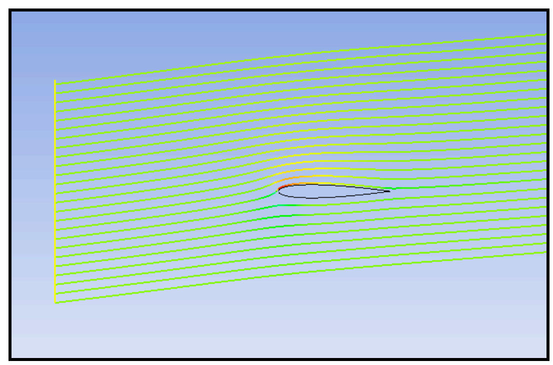




The velocity streamline must be adjusted to concentrate on the seedline.

- Under *Outline*, double click ***Velocity Streamline***
- Set *Start From* to ***Seedline*** (this name is from the user-inputted line name)
- Click ***Apply***



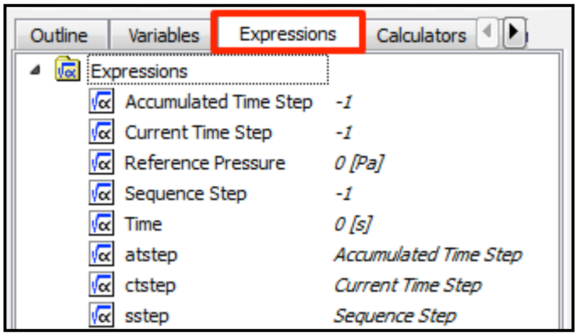


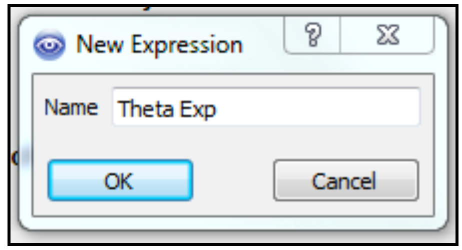
Max Points can be increased to make the streamlines more concentrated.

Pressure vs. Theta Graph

- Click on ***Expressions*** to the right of the Outline tab
- Right click in the Expressions window and click New 

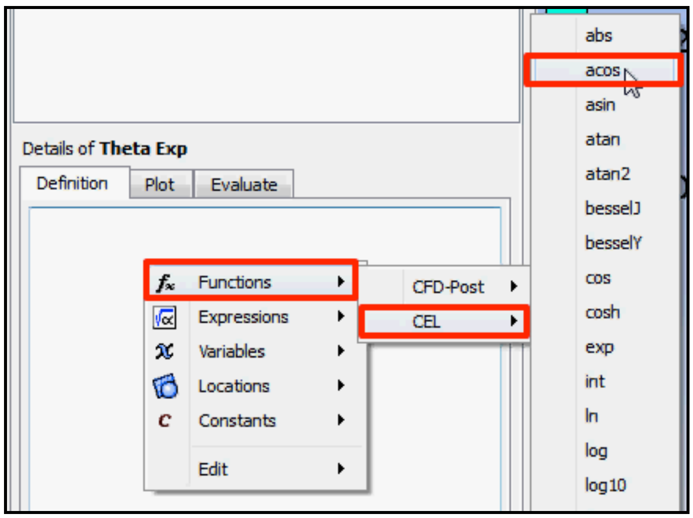
- Name the new expression “Theta Exp”
- Click ***OK***





- In *Details*, right click in *Definition* and select ***Functions*** > ***CEL*** > ***acos***

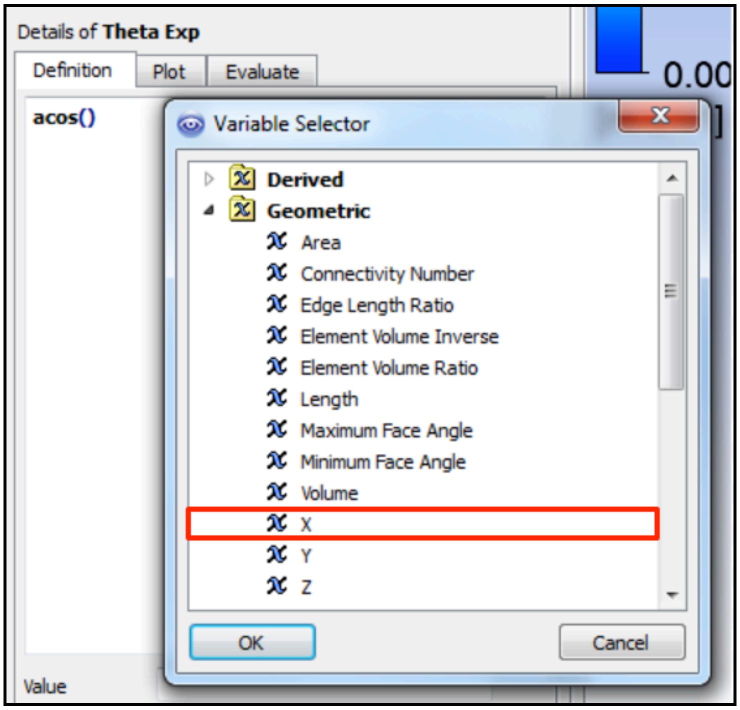
This will create a cosine function in the *Definition*.



- Inside the parenthesis of ***acos( )***, right click and select ***Variables*** > ***Other***

This will pop up a variables window.

- Select ***Geometric*** > ***X***

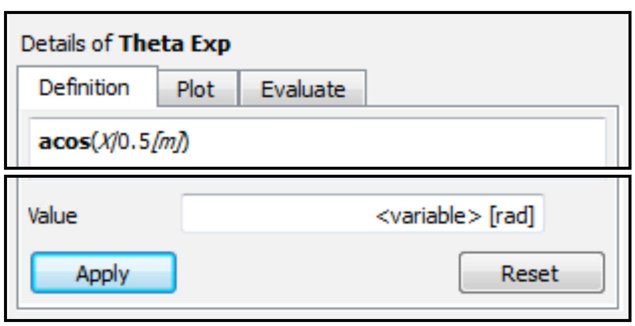


- After the *X*, type in “/0.5[m]”

This divides X by 0.5 and sets the units in meters for the radius of the cylinder.

- Click ***Apply***

The *Value* should now read <variable>[rad] showing that the variable is in radians.



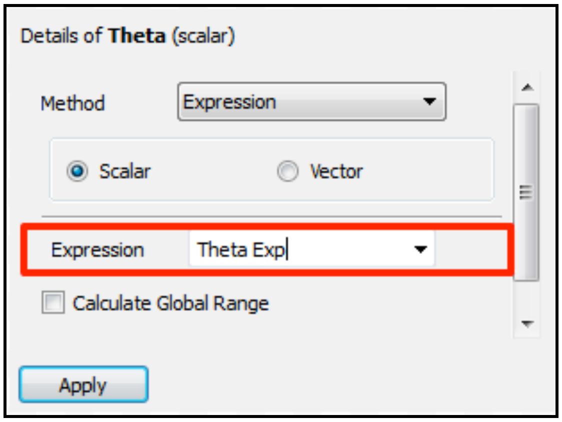
Now the theta variable will be created.

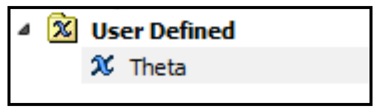
- Click on ***Variables*** to the right of the Outline tab
- Right click in the Variables window and click New 

- Name the new variable “Theta”

- Click ***OK***
- In *Details*, for *Expression*, choose ***Theta Exp***
- Click ***Apply***

This will create a new variable Theta under *User Defined*.

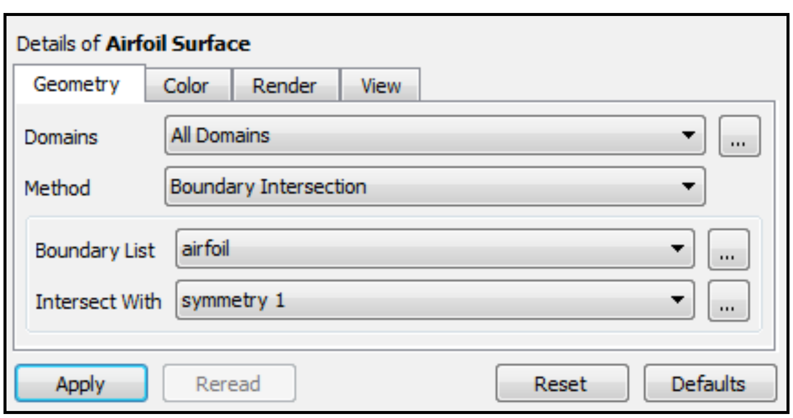


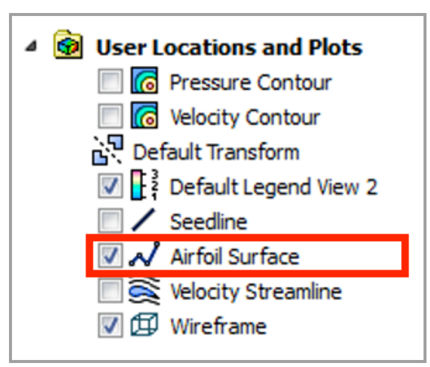


A polygraph will be created to help graph.

- Click on ***Location*** > ***Polyline***, name the line in the pop-up window “Airfoil Surface”, and click ***OK***
- Under *Details*, for *Method*, select ***Boundary Intersection***
- For *Boundary List*, select ***airfoil***
- For *Intersect With*, select ***symmetry 1***
- Click ***Apply***

This will create a new object of the intersection between the airfoil and the mesh under *Outline*, under *User Locations and Plots*.

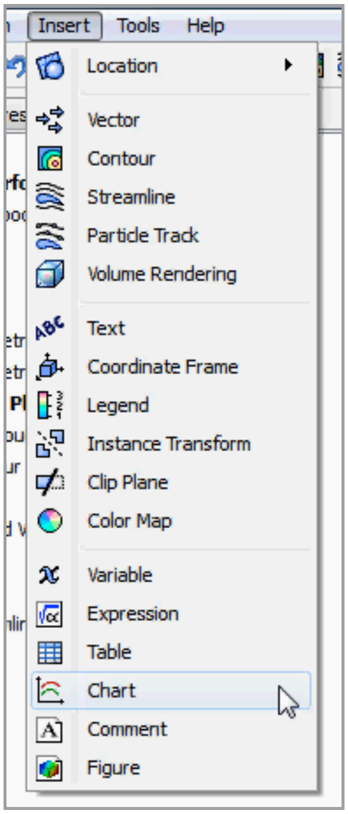


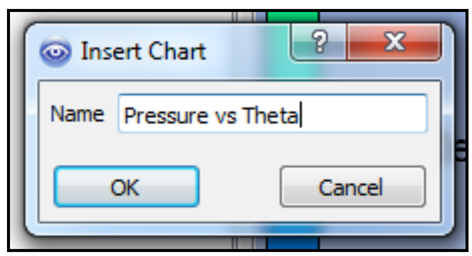


Next, the chart will be inserted.

- Click ***Insert*** > ***Chart***, name the chart "Pressure vs Theta" in the pop-up window, and click ***OK***

- Alternatively, select the ***Chart*** icon in the toolbar. 

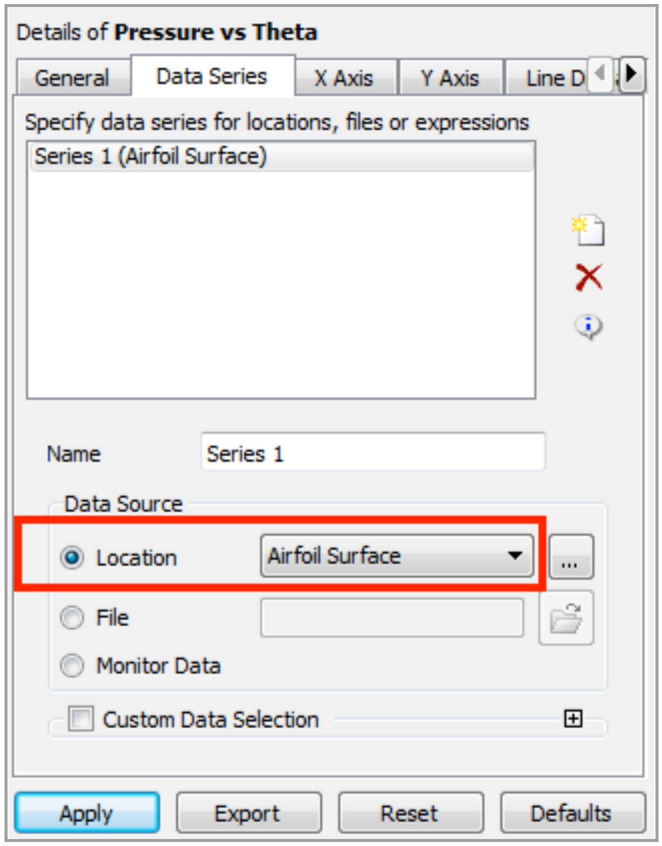




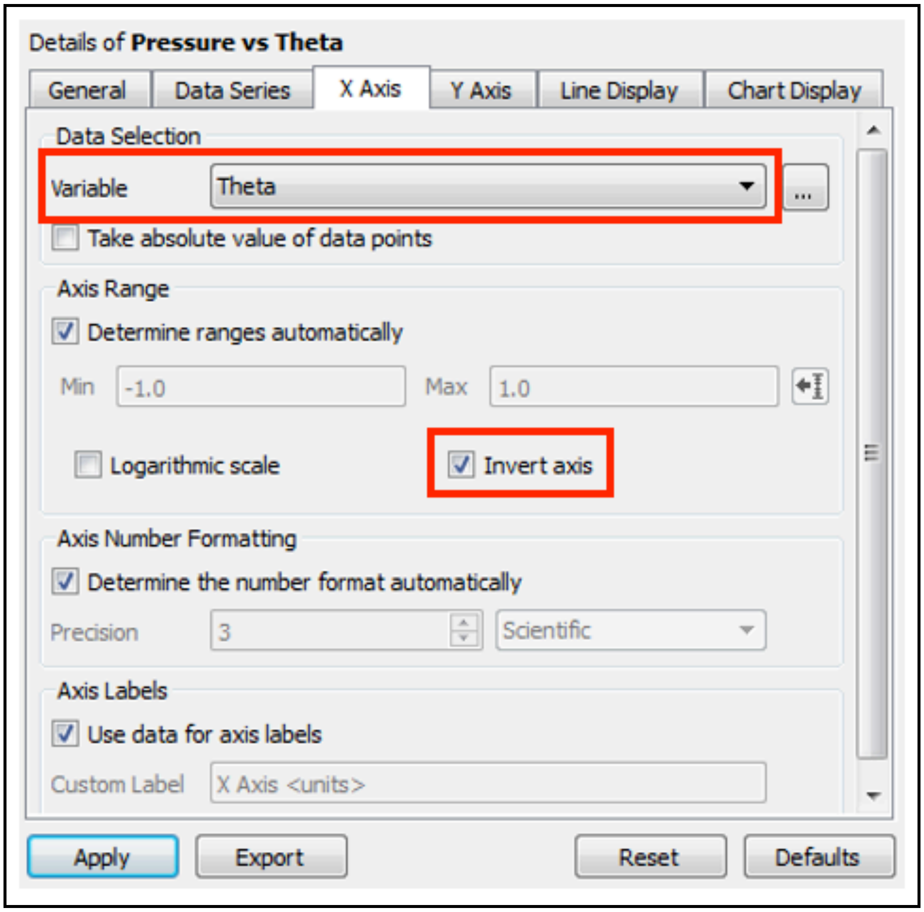
- In *Details*, select the ***Data Series*** tab

- Under *Data Source*, for *Location*, choose ***Airfoil Surface*** (this name is from the user-inputted polyline name)

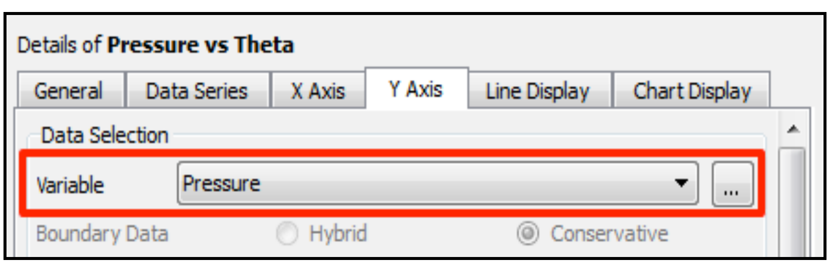
- Click ***Apply***



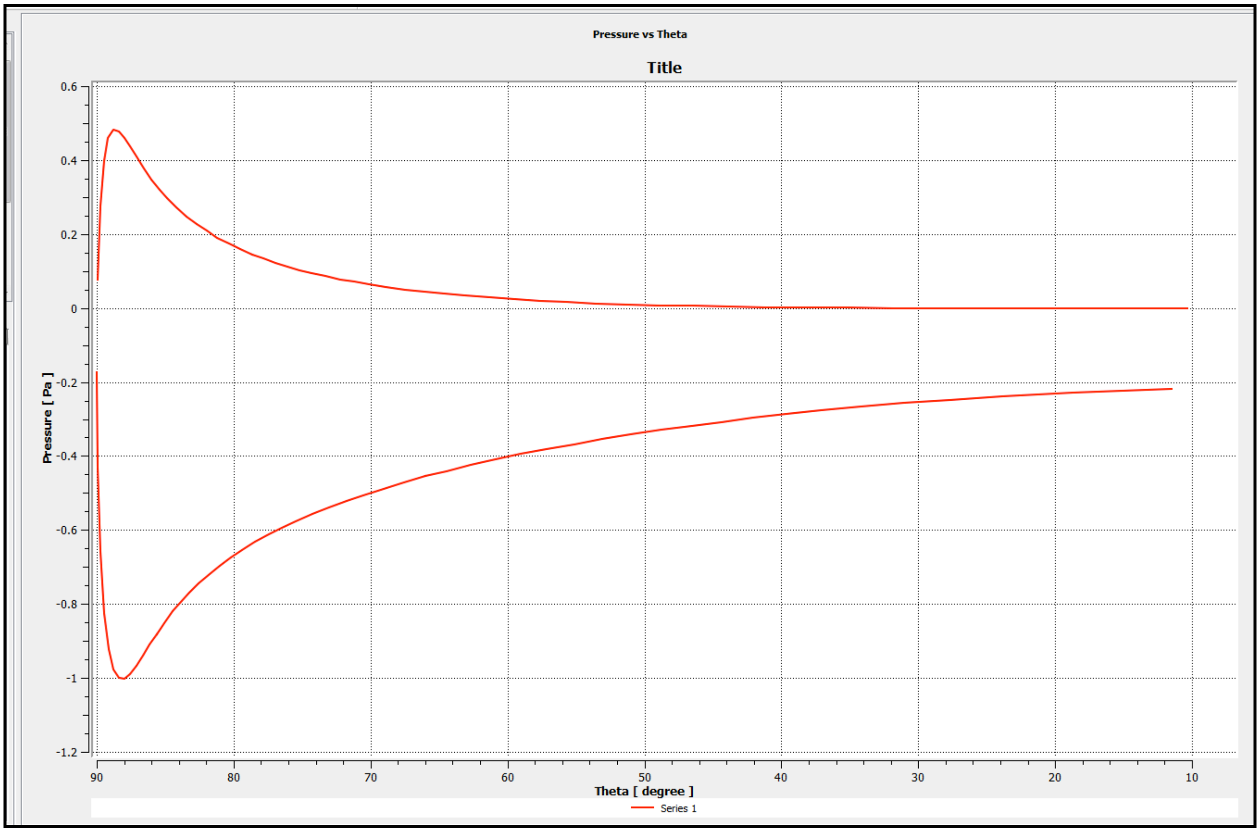
- In *Details*, select the ***X Axis*** tab
- Under *Data Selection*, for *Variable*, choose ***Theta***
- Check ***Invert Axis***
- Click ***Apply***



- In *Details*, select the ***Y Axis*** tab
- Under *Data Selection*, for *Variable*, choose ***Pressure*** (default)
- Click ***Apply***



The final graph should look as follows:



Pictures of the graph can be taken using the camera icon.

- Click ***File*** > ***Save Project***

After finishing the tutorial, do not move the files in the project folders around.

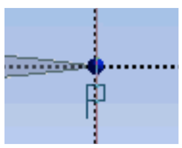
Any missing or misplaced file may corrupt the entire project.

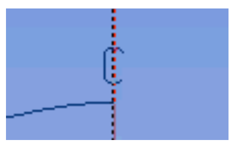
- End of Fluent 18.2 Flow Over an Airfoil Tutorial -

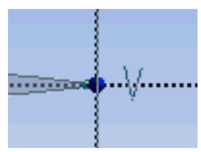
Additional Notes

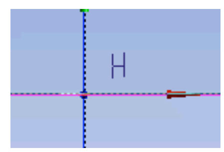
Geometry:
SpaceClaim vs. DesignModeler
The default *Geometry* platform is SpaceClaim, which will open if *Geometry* is double clicked initially. SpaceClaim is used mainly for 3D cases. In this tutorial, DesignModeler is used as it is a 2D case and is an overall more familiar program.

Automatic Constraints
When the mouse is hovered over a point (such as the point of intersection of x and y axes), a “P” will appear over the mouse arrow to denote that it is coincident with a point. When the mouse is hovered along a line, a “C” will appear to denote that it is coincident with a line. When a line is vertical, a “V” will appear by the line. When a line is horizontal, a “H” will appear by the line. This is because DesignModeler is, by default, in auto-constraint mode.

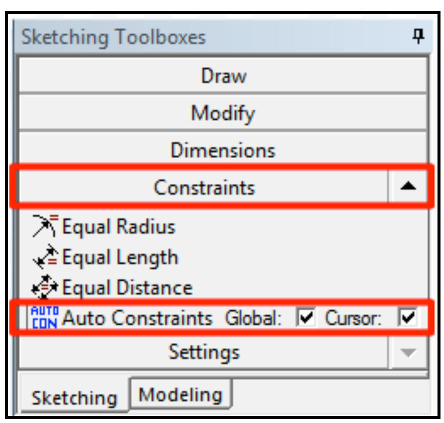








Auto constraints can be turned on and off in the ***Sketching*** tab, under ***Constraints***. To activate, click on ***Auto Constraints*** and check ***Global*** and ***Cursor***.



Manual Constraints
Relations between sketches can be established manually in the ***Sketching*** tab, under ***Constraints***.

Constraints for a particular sketch can be viewed by clicking on the sketch, and under *Details View*, for *Show Constraints?*, choose ***Yes***. Clicking on each constraint highlights the constraint in the *Graphics* window.

Mesh:

Named Selections

If a named selection has “wall”, “(velocity) inlet” or “(pressure) outlet” in its name, ANSYS Fluent automatically assigns these boundary conditions to the corresponding named selection.

Solution:

Verification and Validation

A refined mesh will collect more data near the airfoil.

- Double click ***Mesh*** 

- Click Edge Sizing (sizing of first set of 4 edges)
- For the ***Number of Divisions,*** type in “100”
- Click Edge Sizing 2 (sizing of second set of 4 edges)
- For the ***Number of Divisions,*** type in “100”
- Click ***Mesh*** 

- Click the + next to ***Statistics***

The mesh should now have 40400 Nodes and 40000 Elements.

- Exit the Mesher
- Right click ***Results***, ***Solution***, ***Setup*** and select ***Reset*** for each
- Continue the project starting with the Setup (all information will need to be re-entered)
- For the *Number of Iterations*, type in “10000”, and continue with the project

The new data from the refined mesh will be compared to the original mesh and experimental data.

	Unrefined Mesh	**Refined Mesh**	**Experimental Data***
Drag Coefficient	0.0077	0.0037	0.0090
Lift Coefficient	0.6468	0.6815	0.6630

*Gregory & O’Reilly, NASA R&M 3726, Jan 1970

While the lift coefficient is more accurate in this particular calculation, the drag coefficient is less accurate. These values will vary by calculation.

Therefore, a denser mesh does not imply a more accurate result.

Other:
To zoom in any window, roll the middle mouse button.

To pan in any window, press down the control key, press down the middle mouse button, and drag the mouse.

If any step does not seem to be functioning properly, you may want to go back to the Workbench project schematic, right click the step, and click reset. This will erase the data from the particular step, allowing you to redo it from the beginning.

References

FLUENT Learning Modules. Retrieved November 11, 2017, from https://confluence.cornell.edu/display/SIMULATION/FLUENT+Learning+Modules

ANSYS FLUENT 12.0/12.1 Documentation. Retrieved November 11, 2017, from http://www.afs.enea.it/project/neptunius/docs/fluent/index.htm