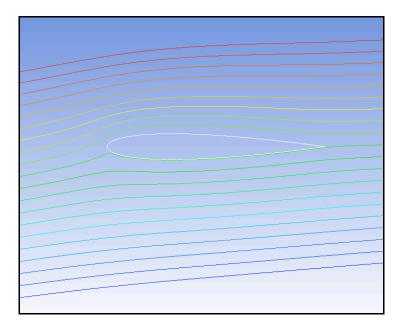
# Fluent 18.2 Tutorial

Case Study: Flow Over an Airfoil



Presented by Aerodynamics Laboratory, Department of Mechanical Engineering, CCNY

The City College of New York

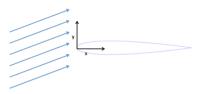
Introduction Problem Specification Solution Domain Boundary Conditions	3 3
1. Start-Up	4 ~ 5
2. Geometry	6 ~ 26
Analysis Type	6
Downloading the Airfoil Sketch	7~9
Creating a New Coordinate System	10
Creating the C-Mesh Domain	11 ~ 12
Creating the Right Side of the C-Mesh Domain	12 ~ 14
Dimensioning the Domain	14 ~ 15
Creating the Flow Domain Surface	16
Removing the Airfoil Surface	17 ~ 18
Creating Quadrants	
Projecting Quadrants	22 ~ 25
Suppressing line Bodies	25
Changing the Surface Type to Fluid	
3. Mesh	27 ~ 37
Mapped Face Meshing	
Edge Sizing 1	
Edge Sizing 2	
Edge Sizing 3	
Verifying the Mesh Size	
Creating Named Selections	
4 Sotup	28 ~ <i>1</i> 1
4. Setup Series / Parallel Processing	
Checking the Mesh	
General Setup	
Models	
Specifying Material Properties	
Boundary Conditions	
Doundary Conductors	42~44

## **Table of Contents**

Inlet	
Outlet	
Airfoil	
Reference Values	44
5. Solution	45 ~ 60
Convergence Criterion	45~46
Initialization	46
Iterating Until Convergence	47
Video Animation	47 ~ 50
Animation	50 ~ 51
Contours	52
Vectors	53
Stream Function	54 ~ 55
Pressure Coefficient vs. Position Graph	55 ~ 57
Coefficient of Drag	
Coefficient of Lift	
6. Results	61 ~ 74
Pressure Contour	
Velocity Contour	
Comparing Contours	
Streamlines	
Pressure vs. Theta Graph	
·	
Additional Notes	75 ~ 77
Geometry	75
SpaceClaim vs. DesignModeler	75
Automatic Constraints	75
Manual Constraints	75
Mesh	76
Named Selections	76
Solution	
Verification and Validation	
Other	77
References	78

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.

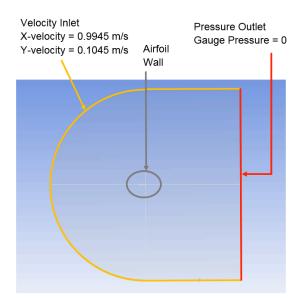
The airfoil coordinates are provided, with the distance from the leading edge (front of airfoil) to trailing edge (back of airfoil) spanning 1 m. The density of the fluid will be 1 kg/m<sup>3</sup> in order to simplify the computation. The air will be assumed to have no viscosity.

#### **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.



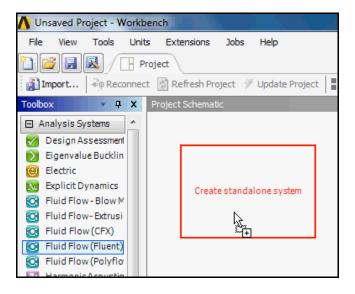
## 1. Start-Up

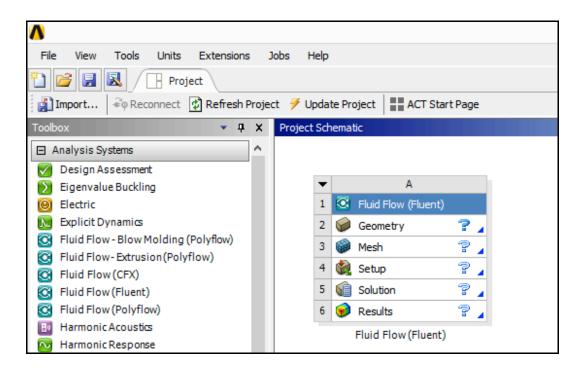
• Open Ansys WorkBench 18.2



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.

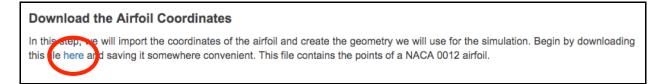
The project name can be changed by clicking below the box, where the default name is "Fluid Flow (Fluent)".

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



Right click here > Save Link/Target as... to save the file

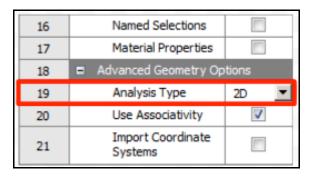
The file contains a text document with a list of all of the coordinates used to create an airfoil shape.

#### 2. Geometry

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

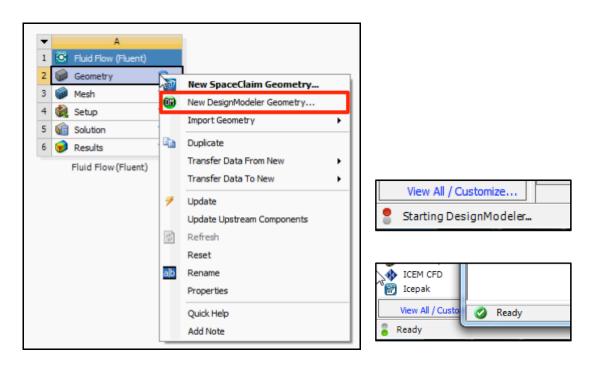
#### **Analysis Type**

- 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.

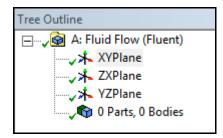


#### **Downloading the Airfoil Sketch**

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

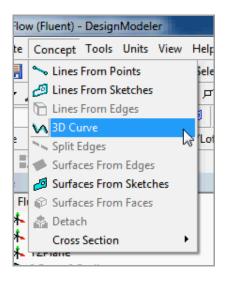
• In the toolbar at the top, click on *Units* > *Meter* 

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

De	Details View 🕈		
	Details of Curve1		
	Curve	Curve1	
	Definition	From Coordinates File	
	Coordinates File	Not selected	
	Coordinates Unit	Meter	
	Base Plane	XYPlane	
	Operation	Add Material	
	Refresh	No	
	Merge Topology?	No	

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

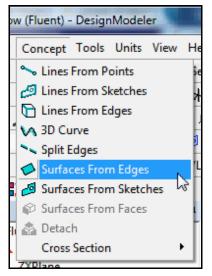


• Click the *Look at Face/Plane/Sketch* icon in the top toolbar to see the sketch

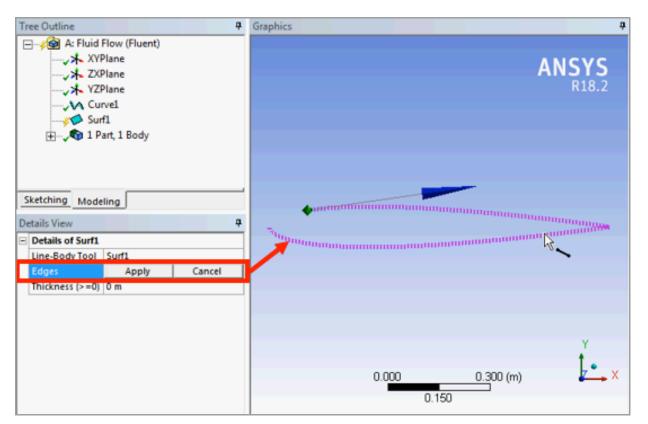
T	ree Outline	₽	Graphics <b>P</b>
	A: Fluid Fla → XYPla XYPLA	ane ane e1 e1 t, 1 Body	ANSYS R18.2
	etails View	<b>4</b>	
	Details of Curve1		
L	Curve	Curve1	
	Definition	From Coordinates File	
	Coordinates File	C:\User\naca0012coords.txt	
	Coordinates Unit	Meter	
	Base Plane	XYPlane	
	Operation	Add Material	
	Refresh	No	Y
	Merge Topology?	No	1
			0.000 0.300 (m) 0.150

• 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

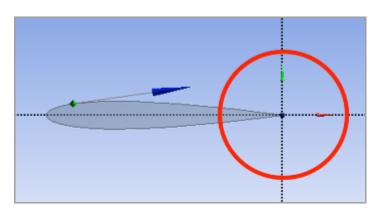
This will create a new coordinate system with the origin at a specified coordinate.

• 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.

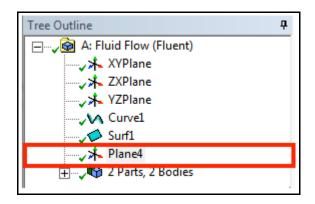
Details View	ф	
Details of Plane4		
Plane	Plane4	
Sketches	0	
Туре	From Coordinates	
FD11, Point X	1 m	
FD12, Point Y	0 m	
FD13, Point Z	0 m	
FD14, Normal X	0	
FD15, Normal Y	0	
FD16, Normal Z	1	
Transform 1 (RMB)	None	
Reverse Normal/Z-Axis?	No	
Flip XY-Axes?	No	
Export Coordinate System?	No	

• Click Generate



## **Creating the C-Mesh Domain**

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



Click New Sketch <sup>1</sup>/<sub>2</sub> 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

Sketching Modeling

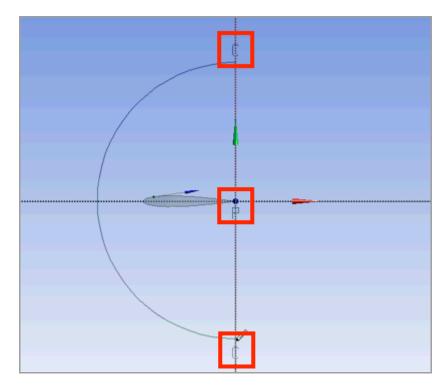
Under Draw, click Arc by Center
 Arc by Center

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

Sketching Toolboxes 🛛 🕈		
Draw 🔺		
🙀 Circle by 3 Tangents		
- Arc by Tangent		
🗥 Arc by 3 Points		
🙃 Arc by Center		
🔁 Ellipse		
Spline		
* Construction Point		
Modify 👻		
Dimensions		
Constraints		
Settings		
Sketching Modeling		

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)

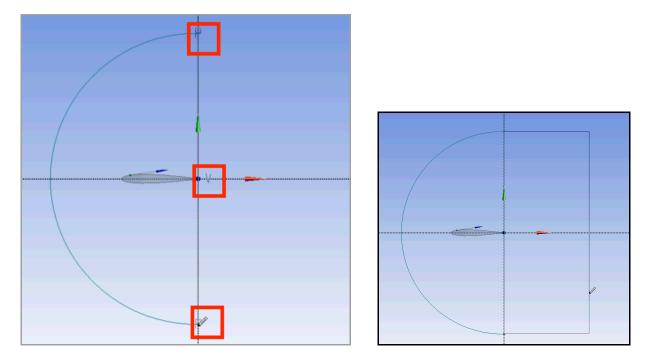


## Creating the Right Side of the C-Mesh Domain

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

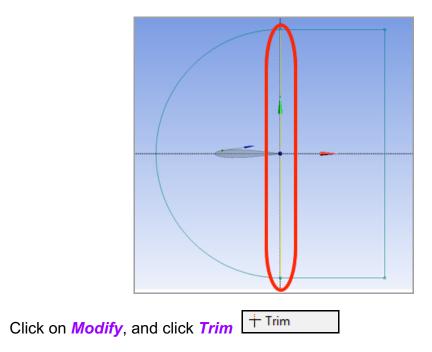
Rectangle by 3 Points creates a rectangle by clicking on two points to designate the length of one side of the rectangle, and then a third point to designate the width.

- 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 yaxis (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.

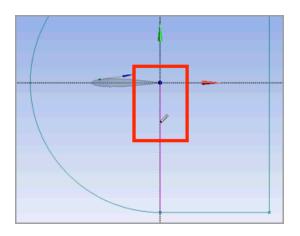
The rectangle creates a vertical line along the y-axis. However, this line is not part of the domain surface, and therefore must be removed.



• Click on 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).

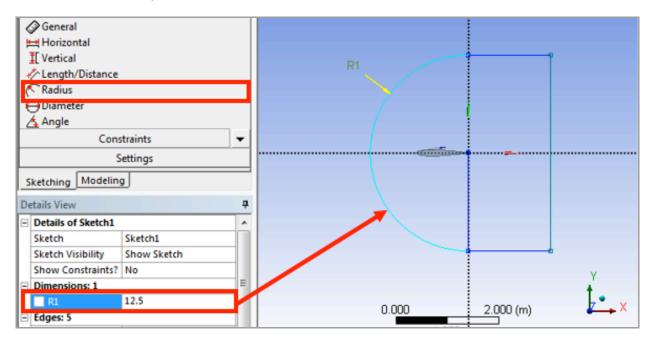


#### **Dimensioning the Domain**

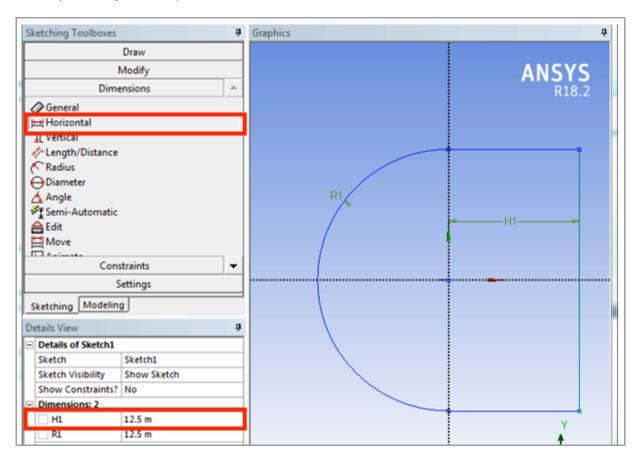
- Click on *Dimensions*, and choose *Radius* Kradius
- Click the rim of the sketched arc to dimension the particular sketch, and then click outside the arc

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)



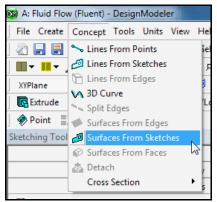
- Choose Horizontal Horizontal
- Click on the vertical y-axis and then click on the vertical line of the rectangle to the right to dimension the horizontal width between the two. Click again to dimension.
- In the *Details View*, under *Dimensions*, click next to *H1* and type in "12.5" (Rectangle width)



#### **Creating the Flow Domain Surface**

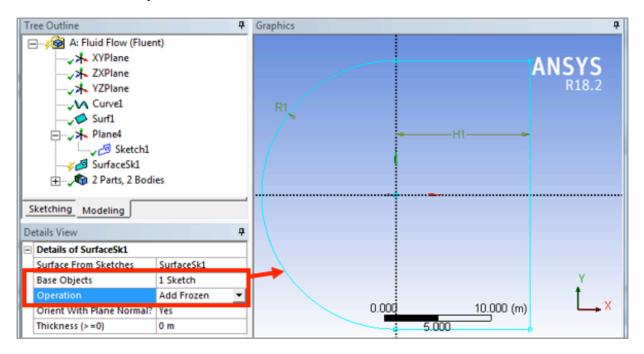
• Click Concept > Surfaces From Sketches

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.



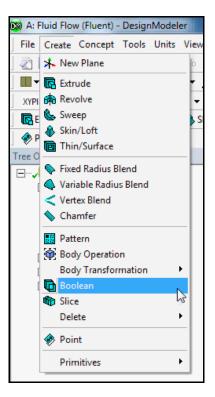
• Click Generate on the toolbar at the top of the window

#### **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

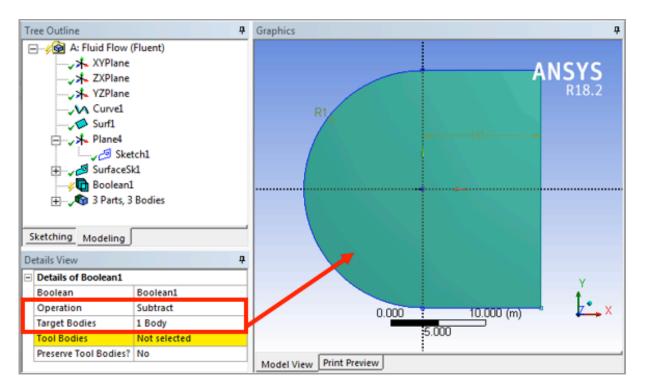
Boolean carries out operations of multiple bodies. This is useful in creating complex shapes from a combination of simpler shapes.



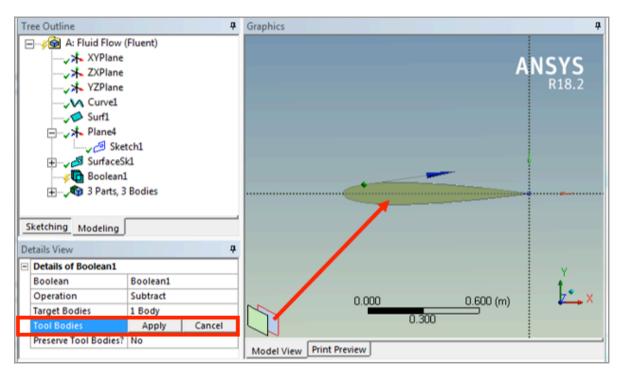
• For Operation, choose Subtract

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* 

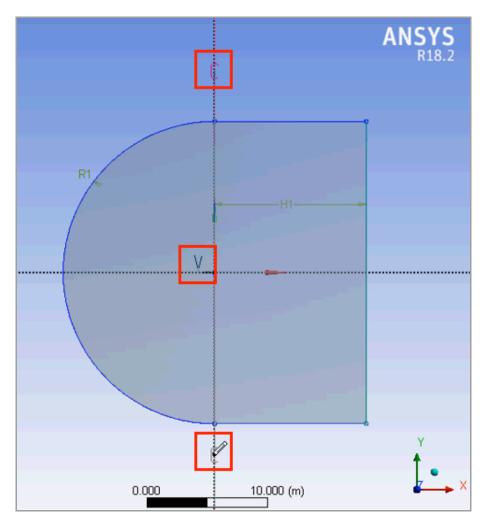


• Click Generate on the toolbar at the top of the window

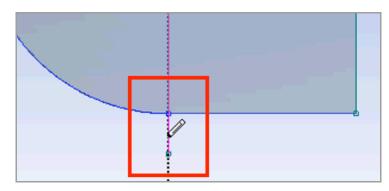
## **Creating Quadrants**

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 ↓ Plane4
- Click New Sketch
- Click on the **Sketching** tab, to the left of Modeling <u>Sketching Modeling</u>
- Under *Draw*, click *Line* 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.

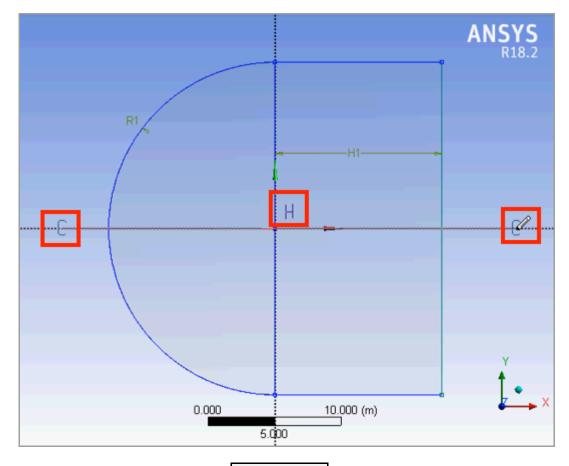
HO	v (Huent) -	- Desigr	iwoqei	er	
ate	Concept	Tools	Units	View	Help
	🛰 Lines	From P	oints		Sele
<b>•</b> .	🖉 Lines	From S	ketches	N	Р
	Eines	From E	dges	Ь	5 1

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

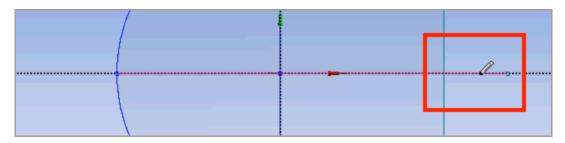
Now, the horizontal line will be created.

- Click Plane4
- Click New Sketch
- Click on the Sketching tab, to the left of Modeling Sketching 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



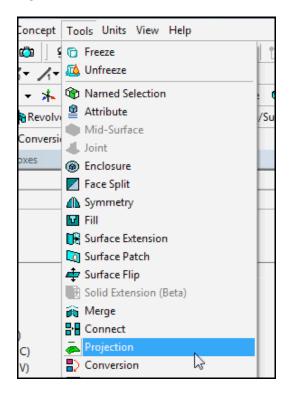
- Click Concept > Lines From Sketches
- Set the *Base Object* to **Sketch 3** (the line sketch just made, under Plane4)
- Click Apply

• Click Generate

## **Projecting Quadrants**

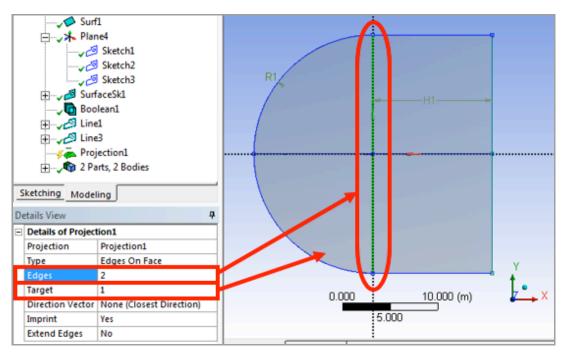
The line currently sits on a layer on top of the surface, and is not associated with the surface directly. In order to split the surface using the line, the line must be projected onto the surface.

• Click on Tools > Projection



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

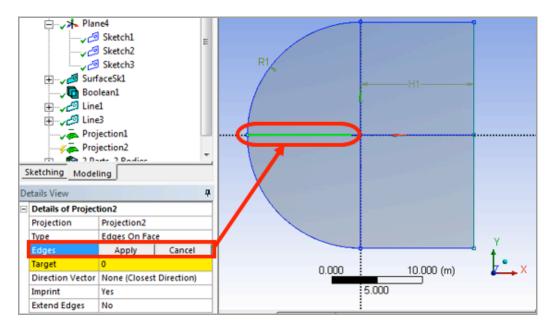
- Make sure the *Type* is *Edges On Face*
- Set the *Edges* to the two vertical lines generated by clicking on one half of the vertical line, and while pressing the control key, clicking on the other half of the vertical line
- Click Apply
- Set the *Target* to the entire surface body by clicking on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



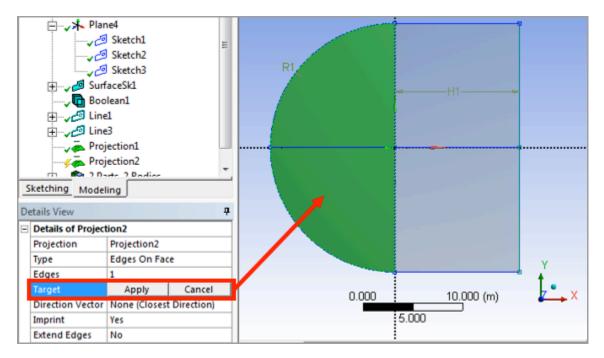
• Click Generate

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



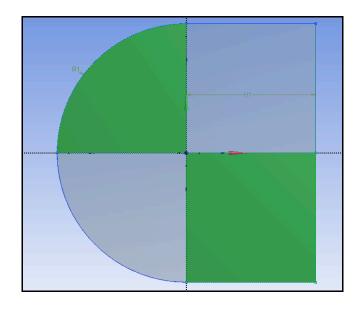
- Click Apply
- Set the *Target* to the left side of the surface body by clicking on the left half of the surface
- Click Apply



• Click Generate

The process is repeated for the right half.

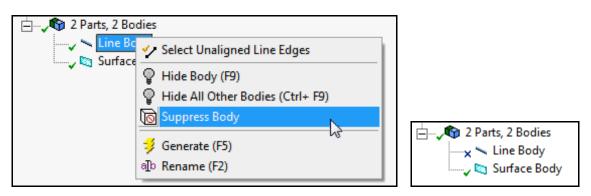
- Click on Tools > Projection
- Make sure the *Type* is **Edges On Face**
- Set the *Edges* to the right half of the horizontal line
- Click Apply
- Set the *Target* to the right side of the surface body by clicking on the right half of the surface
- Click Apply
- Click Generate



#### 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
   Suppress Body



#### Changing the Surface Type to Fluid

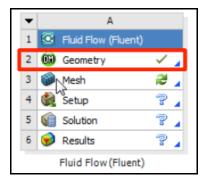
DesignModeler assumes all surfaces to be solids. However, the surface for this case is a fluid. Therefore, the body type must be changed to fluid.

- Click the + next to 2 Parts, 2 Bodies
- Click the *Surface Body*
- Under Details View, select Fluid/Solid > Fluid

Details View	д
<ul> <li>Details of Surface Body</li> </ul>	
Body	Surface Body
Thickness Mode	Inherited
Thickness (>=0)	0 m
Surface Area	557.86 m <sup>2</sup>
Faces	4
Edges	12
Vertices	8
Fluid/Solid	Fluid 💌
Shared Topology Method	Fluid
Geometry Type	Solid

• 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.

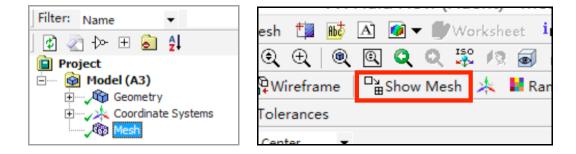
📁 Generate Mesh

Double click Mesh

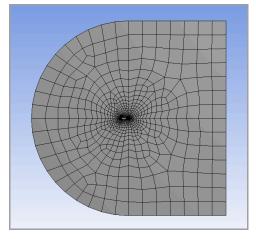
A new Meshing window will open. On the left side is the Outline.

- 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:

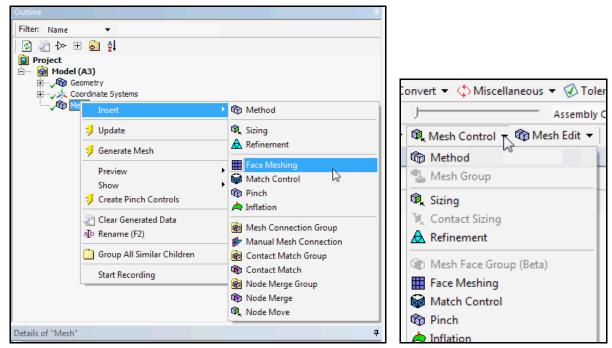


#### Mapped Face Meshing

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 .

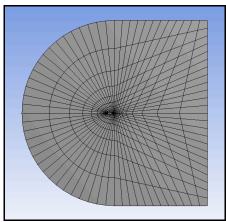
- Click on one quadrant of the surface, and while pressing the control key, click on the other quadrants of the surface to highlight four total quadrants
- Click Apply
- Make sure the *Method* is *Quadrilaterals*

A quadrilateral mesh reduces the skew of the calculations.

0	utline	ф.				
F	ilter: Name	*	Face Meshing 11/11/2017 10:51	лм	Δ	NSYS
1	🖸 🔄 🗠 🗄 🗟	<u>4</u>	11/11/2017 10.51		· · · · · · · · · · · · · · · · · · ·	R18.2
6	Project	<b>2</b> *	Face Meshing	9		110.2
Ē	🗁 🗑 Model (A3)					
	🗄 🗸 🖓 Geometr	у				
	E Coordina	ite Systems				
	⊡ <b>/</b> ੴ Mesh	e Mechina				
		te mestining				
De	tails of "Face Meshin	g" - Mapped Face 🕈				
	Scope					
	Scoping Method	Geometry Selection				
	Geometry	4 Faces				
Ξ	Definition					
	Suppressed	No				
	Mapped Mesh	Yes				
	Method	Quadrilaterals				v
1	Constrain Boundary	No				·····
=	Advanced			0.000	10 000 ()	
	Specified Sides	No Selection			10.000 (m)	••• ^
	Specified Corners	No Selection		5.0	00	
	Specified Ends	No Selection	Geometry	Draview Denast De	niou /	
			[] Geometry APrint	Preview A report Pro	eview/	

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.

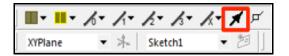


This mesh is already concentrated towards the airfoil, where the most detailed results are wanted. However, we will refine the mesh further to obtain more data points near the airfoil.

## Edge Sizing 1

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

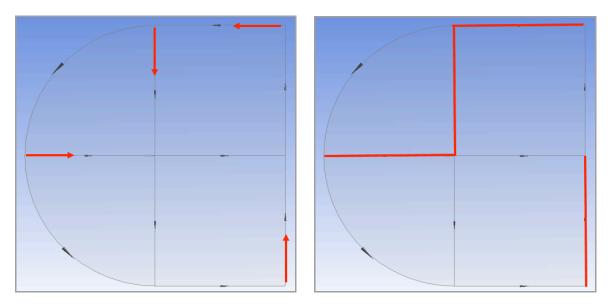
- Right click Mesh Mesh > Insert > Sizing
- In Details View, click next to Geometry
- Click on the Display Edge Direction icon I on the top toolbar



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

- Click on the edge selection icon **b** 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.

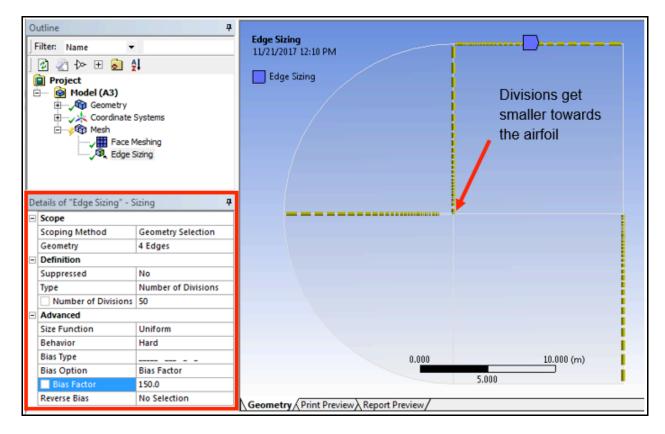
• Set *Bias Type* > ----- - - (first option)

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 Mesh > Insert > Sizing
- In Details View, click next to Geometry
- Click on the edge selection icon **b** 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

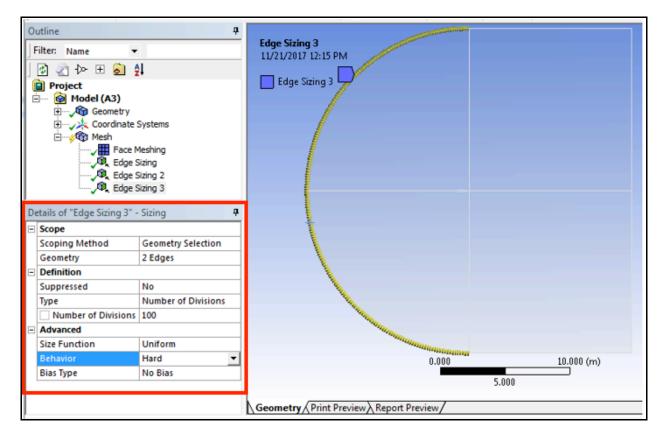
## **Edge Sizing 3**

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.

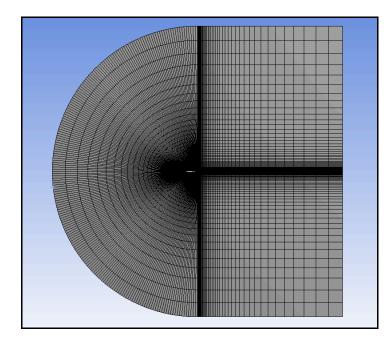
- Right click Mesh Mesh > Insert > Sizing
- In Details View, click on Geometry
- Click on the edge selection icon
   on the top toolbar
- While pressing the control key, select two arcs of the C-Mesh boundary, the top half and the bottom half
- Click Apply
- Set Type > Number of Divisions
- For the Number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

De	etails of "Mesh"	Ŧ	
-	Display		
	Display Style	Body Color	
-	Defaults		
	Physics Preference	CFD	
	Solver Preference	Fluent	
	Relevance	0	
	Export Format	Standard	
	Element Order	Linear	
+ Sizing			
+	+ Quality		
+	Inflation		
+	Assembly Meshing		
+	Advanced     Statistics		
Ξ			
	Nodes	15300	
	Elements	15000	

### **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.

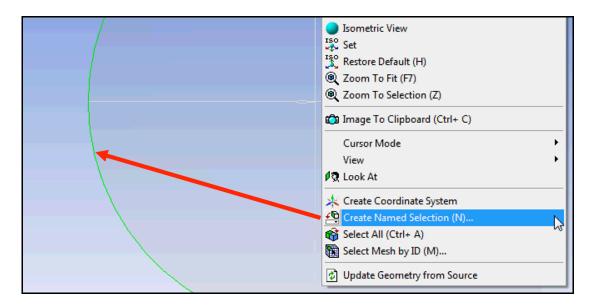
- While pressing the control key, select the two vertical lines of the rectangle to the right
- Right click > Create Named Selection
- In the Details View, name the right arc edge "outlet"

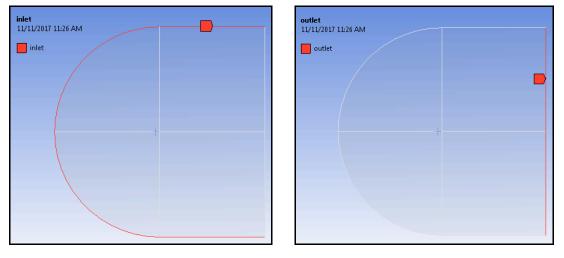
This will be the pressure outlet of the flow.

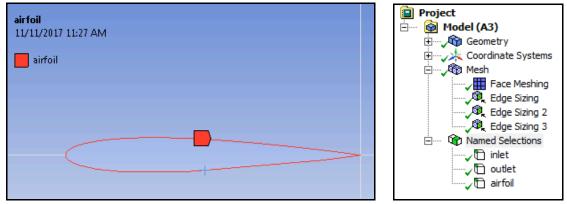
- While pressing the control key, select both the top and bottom edge of the airfoil
- Right click > Create Named Selection
- In the Details View, name the airfoil wall edges "airfoil"

This will be the airfoil wall.

The named selections can be viewed in the Outline.

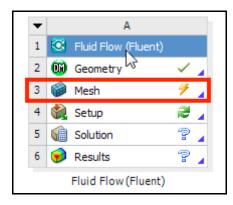






- File > Save Project and close the Mesher
- Return to the Workbench Project Schematic

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



• Right Click *Mesh* > *Update* 

•		A				
1		Fluid Flow (Fluent)				
2	07	Geometry	$\checkmark$	4		
3	۲	Mesh	1	~	Edit	
4		Setup	2	Ŵ	EOIC	
5	6	Solution	?		Duplicate	
6	1	Results	?		Transfer Data From New	-
		Fluid Flow (Fluent)			Transfer Data To New	•
				<b>9</b>	Update N	
					Update Upstream Components	
					Clear Generated Data	

#### 4. Setup

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

• Double Click Setup 4 🙀 Setup 2

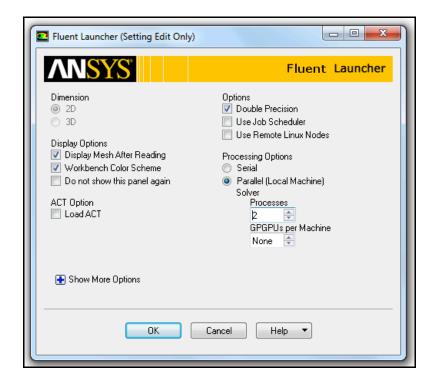
The Fluent Launcher will pop up.

• 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 twocore computer, click on *Parallel*, and under *Processes*, type in 2. The limit of the number of processes is 4.
- Otherwise, choose Serial.
- Click OK

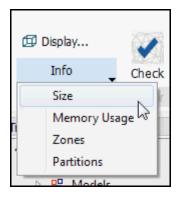


#### **Checking the Mesh**

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

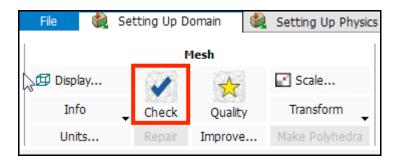
In the top toolbox, click Setting Up Domain
 Mesh > Info > Size

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.



Console	
<pre>Domain Extents: x-coordinate: min (m) = -1.150000e+01, max (m) = 1.350000e+01 y-coordinate: min (m) = -1.250000e+01, max (m) = 1.250000e+01 Volume statistics: minimum volume (m3): 1.855340e-05 maximum volume (m3): 1.494257e+00 total volume (m3): 5.578461e+02</pre>	
Face area statistics: minimum face area (m2): 4.089663e-03 maximum face area (m2): 1.222398e+00 Checking mesh Done.	No Error

## **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.

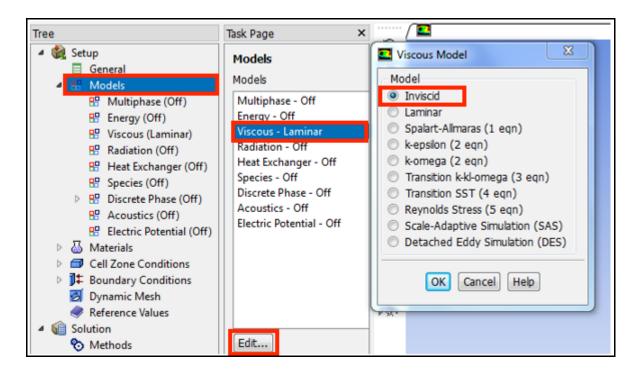
Make sure the *Time* is **Steady** (default). Transient time is used for unsteady flows.

Task Page	×
General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Туре	Velocity Formulation
Pressure-Based	Absolute
Density-Based	<ul> <li>Relative</li> </ul>
Time	2D Space
Steady	Planar
Transient	Axisymmetric
	Axisymmetric Swirl
Gravity Units	

#### 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<sup>3</sup>
- Click Change/Create and Close

Tree	Task Page		×		2
<ul> <li>✓ 🌺 Setup</li> <li></li></ul>	Materials Materials		(	S ₽	
🕨 👗 Materials	Fluid			€.	
<ul> <li>Cell Zone Conditions</li> <li>I Boundary Conditions</li> <li>Dynamic Mesh</li> </ul>	air Solid aluminum			⊕ <b>.</b>	
Create/Edit Materials				23	
Name		Material Type			
air		fluid			•
Chemical Formula		Fluent Fluid Materials			
		air			•
		Mixture			
		none			-
Properties					
Density (kg/m3) constant		▼ [Edit]			
•					F
	Change/Create	Delete Close Help			
	Create/Edit.	Delete		Wr	ritin citin citin

# **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  $\cos(6^\circ)$  and the y component is  $\sin(6^\circ)$ .

• Click OK

Tree	Task Page ×	Mes
	Boundary Conditions Zone Filter Text airfoil inlet interior-surface_body outlet surface_body Velocity Inlet Zone Name	
	inlet Momentum Thermal Radiation Species Velocity Specification Method Components Reference Frame Absolute Supersonic/Initial Gauge Pressure (pascal) 0 X-Velocity (m/s) 0.9945 Y-Velocity (m/s) 0.1045 Outflow Gauge Pressure (pascal) 0	DPM Multiphase Potential UDS
<ul> <li>Image: Provide the second seco</li></ul>	OK Cancel Phase Type ID mixture velocity-inlet 6 Edit Parameters Display Mesh, Periodic Conditions Periodic Conditions	Help Console Domain Extents: x-coordinate: min (m) = -1.150000e+( y-coordinate: min (m) = -1.250000e+( Volume statistics: minimum volume (m3): 1.855340e-05 maximum volume (m3): 1.494257e+00 total volume (m3): 5.578461e+02 Face area statistics: minimum face area (m2): 4.089663e-0;

#### 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  $\cos(6^{\circ})$  and  $\sin(6^{\circ})$ )

• Make sure the Reference Zone is *fluid-surface\_body* 

Tree	Task Page	×
▲ 🧶 Setup     ☐ General     ▷ 🖽 Models	Reference Values Compute from	
Models     Materials	inlet	•
Cell Zone Conditions	Reference Values	
▲ J Boundary Conditions	Area (m2)	1
]‡ airfoil (wall, id=8) ]‡ inlet (velocity-inlet, i	Density (kg/m3)	1
↓ interior-surface_bod	Depth (m)	1
1 outlet (pressure-outl	Enthalpy (j/kg)	0
Jt surface_body (interi	Length (m)	1
Dynamic Mesh Reference Values	Pressure (pascal)	0
▲ Solution	Temperature (k)	288.16
S Methods	Velocity (m/s)	0.9999752
💞 Controls 🛐 Report Definitions	Ratio of Specific Heats	1.4
Monitors	Reference Zone	
Cell Registers	fluid-surface_body	•
₽ <sub>t=0</sub> Initialization		

## 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.

Tree	Task Page ×
<ul> <li>Setup         <ul> <li>General</li> <li>General</li> <li>Models</li> <li>Materials</li> <li>Cell Zone Conditions</li> <li>It Boundary Conditions</li> <li>It Boundary Conditions</li> <li>It Boundary Conditions</li> <li>Reference Values</li> <li>Solution</li> <li>Methods</li> <li>Controls</li> <li>Report Definitions</li> </ul> </li> </ul>	Solution Methods Formulation Implicit   Flux Type Roe-FDS   Spatial Discretization Gradient Least Squares Cell Based   Flow Second Order Upwind

#### **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

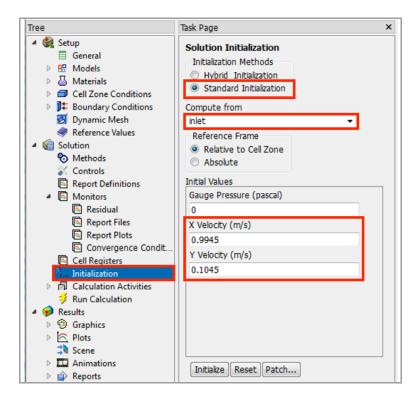
Residual Monitors			
Options	Equations		
Print to Console	Residual	Monitor Check Converge	nce Absolute Criteria
V Plot	continuity	✓	1e-06
Window	x-velocity	✓	1e-06
	v-velocity	✓	1e-06
	7 7		
1000 🚖			Convergence Criterion
	Normalize		absolute
Iterations to Store		5 💼	
1000 ≑	Scale		Convergence Conditions
	Compute Local	Scale	
l	<u></u>		
ОК	Plot Renormalize	e Cancel Help	
	Options ✓ Print to Console ✓ Plot Window 1	Options       Equations         Image: Plot       Residual         Continuity       x-velocity         Image: Plot       Y-velocity         Image: Plot       Y-velocity         Iterations to Plot       Residual Values         Iterations to Store       Image: Plot         Image: Plot       Image: Plot         Iterations to Store       Image: Plot         Image: Plot       Ima	Options       Equations         Image: Print to Console       Residual       Monitor Check Converges         Image: Plot       Continuity       Image: Plot         Window       Image: Plot       Image: Plot         Image: Plot       Image: Plot       Image: Plot         Interations to Plot       Plot       Plot         Iterations to Store       Image: Plot       Image: Plot         Iterations to Store       Image: Plot       Image: Plot         Image: Plot       Image: Plot       Image: Plot         Image: Plot

#### 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

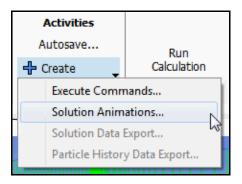
Tree	Task Page ×
<ul> <li>✓ Setup</li> <li></li></ul>	Run Calculation       Check Case       Update Dynamic Mesh
<ul> <li>Cell Zone Conditions</li> <li>J‡ Boundary Conditions</li> <li>Dynamic Mesh</li> <li>Reference Values</li> <li>Solution</li> <li>Methods</li> </ul>	Number of Iterations 3000 Profile Update Interval 1 Colution Standows
<ul> <li>Controls</li> <li>Report Definitions</li> <li>Monitors</li> <li>Cell Registers</li> <li>Initialization</li> </ul>	Solution Steering Data File Quantities Acoustic Signals Calculate
Calculation Activities     Run Calculation     Generation     Results	Help

## **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 Solving > Activities > Create > Solution Animations...

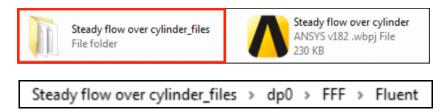
This will pop up an Animation Definition window



• 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.

Name: animation-1				
Record after every	25	-	iteration	•
Storage Type	HSF File		•	]
Storage Directory				
Window Id	2	*		
Animation View	front	7	Preview	Use Active
Animation Object				=
residuals				
New Object 🔻	Edit Object			
New Object	Edit Obj sct			
	Can		qls	
Mesh			gle	
Mesh Contours	Can		elp	
Mesh Contours Vectors	Can F I		elp.)	
Mesh Contours Vectors Pathlines	Can F I		elp]	

- 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

Contours	
Contour Name contour-1	
Options Filled	Contours of Pressure
<ul> <li>Node Values</li> <li>Global Range</li> <li>Auto Range</li> <li>Clip to Range</li> <li>Draw Profiles</li> </ul>	Static Pressure       Min (pascal)       0
Draw Mesh	Surfaces Filter Text
Coloring Banded Smooth Colormap Options	inlet interior-surface_body outlet surface_body
	New Surface  Save/Display Compute Close Help

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

Name: animation-1					
Record after every	25	÷ it	eration		•
Storage Type	HSF File		•	]	
Storage Directory					
Window Id	2	* *			
Animation View	front		review	Use /	Active
					_
Animation Object					-
residuals					
residuals contour-1					
contour-1	Edit Object				
contour-1	Edit Object)				

- 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.

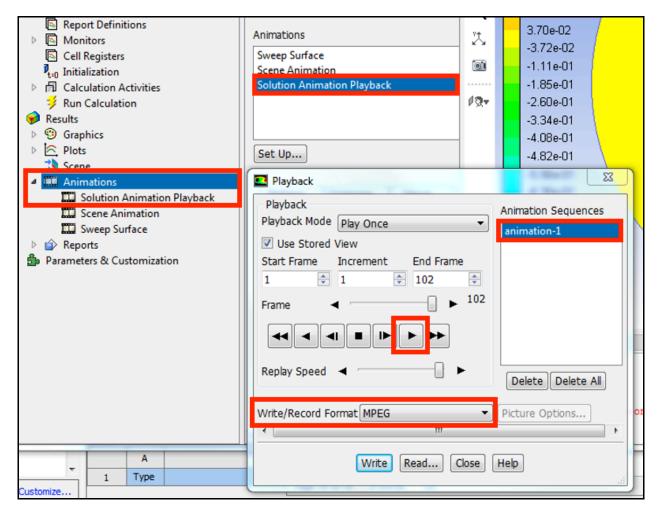


Different tabs

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.

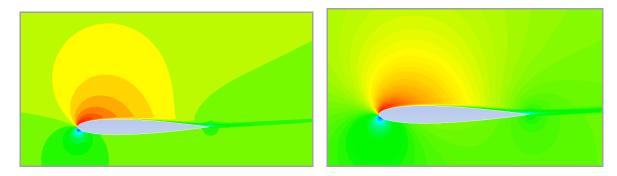


#### Contours

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

Contours		23
Contour Name contour-2		
Options Filled	Contours of Velocity	•
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Velocity Magnitude	•
Auto Range Clip to Range	Min (m/s) Max (m/s) 0.132165 1.588979	
Draw Profiles	Surfaces Filter Text	] <b>-</b>
Coloring Banded Smooth Colormap Options	airfoil fluid-surface_body inlet interior-surface_body outlet surface_body New Surface	
	Save/Display Compute Close Help	



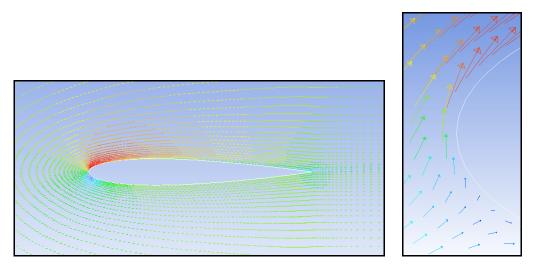
Colormap Size 20 (left) vs. 100 (right)

#### 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

Vectors	
vector-1	
Options	Vectors of
Global Range	Velocity
Auto Range	Color by
Clip to Range	Velocity
Auto Scale Draw Mesh	Velocity Magnitude 🔹
	Min (m/s) Max (m/s)
Style	0.1223928 1.605787
arrow	
Scale Skip	Surfaces Filter Text
1 0 🖨	airfoil
Vector Options	fluid-surface_body
Custom Vectors	inlet
Custom vectors	interior-surface_body
Colormap Options	outlet surface body
	New Surface
S	ave/Display Compute Close Help



#### **Stream Function**

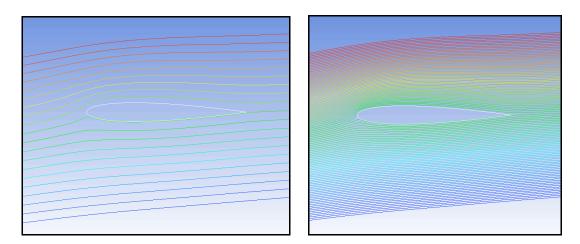
Streamlines can be viewed using contours.

- 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

0p-lisi	Contours				-17
f. Contours	-			X	3
Contour Name contour-3					
Options Filled Vode Values Global Range	Contours of Velocity Stream Function				•
Auto Range     Olip to Range     Draw Profiles     Draw Mesh	Min (kg/s) 13.11 Surfaces Filter Text	Max (kg/s) 14.16			x
A Coloring	airfoil fluid-surface_body inlet interior-surface_body outlet surface_body				
	Save/Display Comput	te Close H	elp		

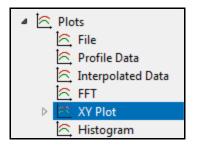


Colormap Size 20 (left) vs. 100 (right)

# 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



XY Plot Name		
xy-plot-1		
Options	Plot Direction	Y Axis Function
Node Values	X 1	Pressure •
Position on X Axis	Y 0	Static Pressure 👻
Position on Y Axis     Write to File	Z 0	X Axis Function
Order Points		Direction Vector
	Free Data	airfoil fluid-surface_body inlet interior-surface_body outlet surface_body New Surface
S	ave/Plot Axes Curve	

• Click Save/Plot

airfoil	
	1.0000 ]/
	0.5000 -
	0.0000
Pressure	-0.5000 -
Coefficient	-1.0000 -
	-1.5000 -
	-2.0000 -
	Position (m)

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

Options	
Node Values	
Position on X Axis	
Position on Y Axis	
Write to File	
Order Points	

Select File					8 🛛
Look in:	C:\Users\delale\Desktop\Flow	w Over an Airfoil_fi	es\dp0\FFF\Fluent	000	📑 🗉 🔳
📃 My Com	outer Name	Size Typ	Date Modified		
🗽 delale	animation-1.cxa animation-1_0000.hs	f 236 KB hsf F	ile 11/11/26:27 Pl ile 11/11/23:15 Pl il <u>e 11/11/2 3:17 Pl</u>	v	-
XY File F	Pressure_Coefficient.txt				ОК
Files of type:	All Files (*)				
Filter String					Filter

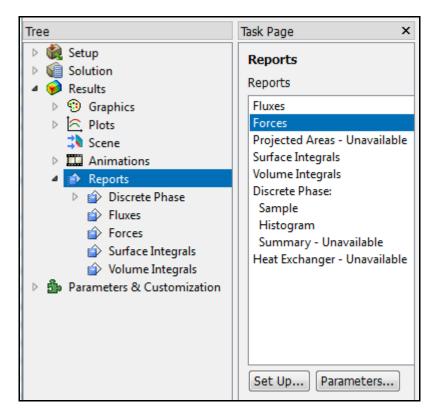
Pressure_Coefficient	t - Notepad	23
File Edit Format	View Help	
(title "Pressure (labels "Positio	e Coefficient") on" "Pressure Coefficient")	
((xy/key/label 1 0.237419		
0.995951	0.22788 0.217552	
0.98784 0.2042	0.217552	
0.983787 0.979409	0.191738 0.178856	
	0.166261	
	0.154455	
	0.14292 0.130934	
0.955731	0.119059	
	0.107201 0.0963958	
	0.085005	
	0.073057	
0.92626 0.06128	35	<b>T</b>

#### **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°.



E Force Reports			X				
Options <ul> <li>Options</li> <li>Forces</li> <li>Moments</li> <li>Center of Pressure</li> </ul> Save Output Parameter.	Direction Vector X 0.9945 Y 0.1045 Z 0	Wall Zones Filter Text					
Print Write Close Help							

• Click Print

This will output the force reports in the Console.

Console						
Forces - Direction Vec Zone airfoil	ctor (0.9945 0.1045 Forces (n) Pressure 0.0038696325	0) Viscous 0	Total 0.0038696325	Coefficients Pressure 0.007739648	Viscous 0	Total 0.007739648
 Net	0.0038696325	0	0.0038696325	0.007739648	0	0.007739648

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°.

Force Reports			X					
Options Forces Moments Center of Pressure Save Output Parameter	Direction Vector X -0.1045 Y 0.9945 Z 0	Wall Zones Filter Text						
Print Write Close Help								

• Click Print

This will output the force reports in the Console.

Console							8
Forces - Direction Vect	or (-0.1045 0.99 Forces (n)	945 0)		Coefficients			
Zone	Pressure	Viscous	Total	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
 G Results
 C 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

	File	Edit Monitor S		Insert	Tools Help	¢¢		
_	-Pos							
2 4	-	Location	•		Insert Cor	ntour	?	×
fr 1	3	Vector			Name Pressu	ure Conto	our	_1
fc 📻	6	Contour	6		ОК		Cance	1
		Streamline	10					

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

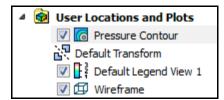
- Set Locations to symmetry 1
- Set Variable to Pressure

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*.

Details of <b>Press</b>	ure Contour		View 1 🔻	
Geometry	Labels Render	View	Pressure Pressure Contour	ANSYS R18.2
Domains Locations	All Domains symmetry 1	<ul><li></li><li></li></ul>	4.820e-001 3.337e-001 1.854e-001 3.707e-002 -1.113e-001	
Variable Range	Pressure Global	•	-2.596e-001 -4.079e-001 -5.562e-001 -7.046e-001	
Min Max	-1.0012 [P 0.482048 [P		-8.529e-001 -1.001e+000 [Pa]	
# of Contours Advanced Pro		÷ •		, , ,
Apply	Reset [	efaults	0 5.000 (m) 2.500 7.500	<mark>v →</mark> ×



## **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.

Details of Veloc	city Contour					
Geometry	Labels Render Vi	iew	View 1 🔻			
Domains Locations	All Domains		Velocity Velocity Contour 1.588e+000 1.441e+000 1.294e+000			ANSYS <sub>R18.2</sub>
Variable	Velocity 🔻		- 1.147e+000 - 1.000e+000			
Range Min	Global   0.118676 [m s^-1]		8.532e-001 7.063e-001 5.594e-001 4.125e-001 2.656e-001 1.187e-001			_
Max	1.5877 [m s^-1]		[m s^-1]			
# of Contours	11					
Advanced Pro	operties	- E				×
Apply	Reset Defa	ults		0 2.50	5.000 10.000 (m) 10 7.500	×
	4 🚱	User L	ocations and Pl	ots		

#### **Comparing Contours**

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

Pressure contour Velocity contour

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.

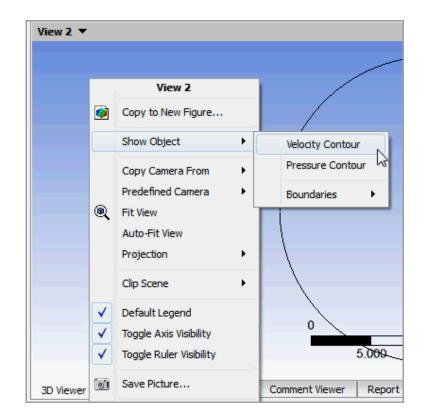
• Deselect the Synchronize visibility in displayed view icon

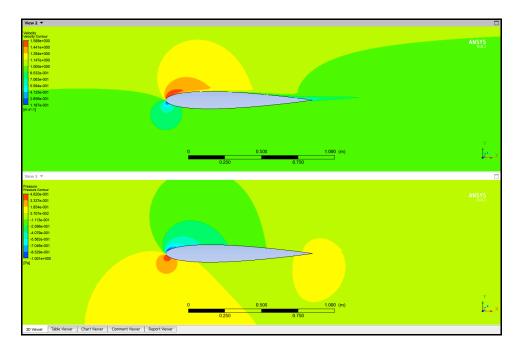
When the visibility is synchronized, both the top and bottom views will only be view of one contour, preventing both contours from being shown.



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

8





Velocity Contour on top, Pressure Contour on bottom

#### **Streamlines**

- Click Insert > Streamline, name the streamline "Velocity Streamline" in the popup 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 •

Variable

Direction Cross Periodics

Apply

Boundary Data

Velocity

Forward

O Hybrid

• Uncheck the pressure contour and velocity contour found under the outline and check the Velocity Streamline

	<ul> <li>User Locations and Plots</li> <li>Pressure contour</li> <li>Velocity contour</li> <li>Default Transform</li> <li>Default Legend View 1</li> <li>Velocity Streamline</li> <li>Wireframe</li> </ul>
	ity Streamline Color Symbol Limits Rende
Type Definition Domains Start From	3D Streamline       All Domains       inlet
Sampling # of Points	Equally Spaced

Preview Seed Points

Reset

• ....

Ŧ

Defaults

Onservative

4.859e-001

1.187e-001 [m s^-1]

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*

22222	10 L	ocation 🔹 🚓 🛣	6
T	+	Point	
	30	Point Cloud	_
	/	Line	
		Plane 45	

• 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 Ap	ply				
4 😨 User l	ocations	and Plots			
iii (iii)	Pressure (	Contour			
	Velocity Co				
B <sup>D</sup> e	fault Transf	orm			
	Default Le	gend View	2		
<b>V</b>	Seedline				
V 🔍	Velocity St	reamline			
V 🗇	Wireframe			-	
Details of See	dline				
Geometry	Color	Render	View		
Domains	All Domai	ns		▼	Seedline
Definition					Seedime
Method	Two Poi	nts		•	
Point 1	-2	-1	0		
Point 2	-2	1	0		×
Line Type					 
Cut		Sar	nple		
-					
Samples	50			-	

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

Details of Veloc	ity Streamline	
Geometry	Color Symbol Limits Rende	
Type Definition	3D Streamline 🔻	
Domains	All Domains	
Start From	Seedline 🔹 🛄	
Sampling	Vertex 🔻	
Reduction	Max Number of Points 🔹	
Max Points	25	
	Preview Seed Points	
Variable	Velocity 🔹	
Boundary Data	a 🔿 Hybrid 💿 Conservative	
Direction	Forward 🔻	
Cross Period	lics	

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

New New

- Name the new expression "Theta Exp"
- Click OK

Outline	Variables	Expressions	Calculators	_
🔺 底 Exp	pressions			
<u>v</u>	Accumulate	ed Time Step - J	1	
v	Current Tin	me Step - J	1	💿 New Expression 🛛 😵 🕺
<u>\</u>	Reference	Pressure 0	[Pa]	
<u>{</u>	Sequence S	Step -1		Name Theta Exp
<u>\</u>	Time	0	[s]	
v	atstep	A	coumulated Time Step	OK Cancel
<u>\</u>	ctstep	a	urrent Time Step	
<mark>√</mark> α	sstep	Se	equence Step	

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

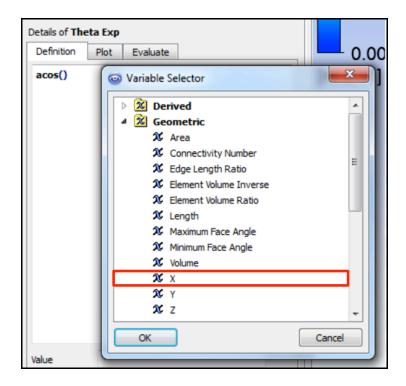
This will create a cosine function in the *Definition*.

		abs acos <sub>N</sub>
Details of <b>Theta</b>	e Exp Plot Evaluate	asin <sup>NS</sup> atan atan2 besselJ
	<i>f</i> <sub>∞</sub> Functions <i>Expressions W</i> Variables <i>Locations C</i> Constants <i>Edit</i>	besselY cos cosh exp int In log log10

• 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.

Details of <b>Theta Exp</b>						
Definition	Plot	Evaluate				
acos(X/0.5[m])						
Value			<variable> [rad]</variable>			
Apply			Reset			

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"

New

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

This will create a new variable Theta under User Defined.

Details of <b>Theta</b> (scalar)		
Method Expression 💌		
Scalar	E	
Expression Theta Exp 🔹		
Calculate Global Range	Ţ	
	÷	4 🕺 User Defined
Apply		🎗 Theta

A polygraph will be created to help graph.

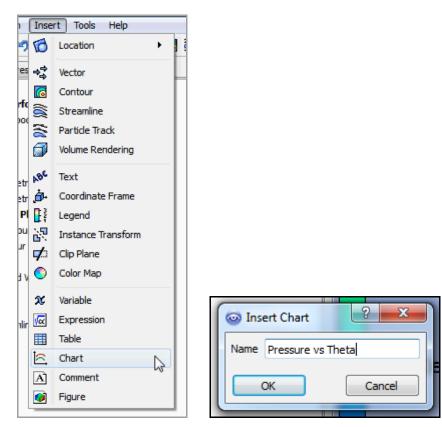
- Click on *Location* > *Polyline*, name the line in the pop-up window "Airfoil Surface", and click *OK*
- Under Details, for Method, select Boundary Intersection
- For Boundary List, select airfoil
- For Intersect With, select symmetry 1
- Click Apply

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

Details of Airfoil Surface	
Geometry Color Render View	Iser Locations and Plots
Domains All Domains	Pressure Contour
Method Boundary Intersection	Contour
	Default Transform
Boundary List airfoil	
Intersect With symmetry 1	📝 📈 Airfoil Surface
	Velocity Streamline
Apply Reread Reset Defaults	🔽 🗇 Wireframe

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

Details of <b>Pressure vs Theta</b>					
General	Data Series	X Axis	Y Axis	Line D 🔹 🕨	
	a series for loca Airfoil Surface)	tions, files o	or expressio	ons *`` *`` ;;	
Name ⊂Data So	Series 1				
Local	ation Ai	rfoil Surface	2		
File     Monitor Data					
Custom Data Selection					
Apply	Export	R	eset	Defaults	

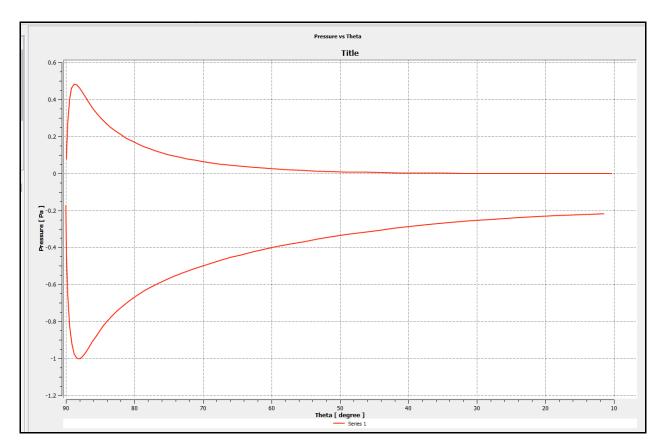
- In Details, select the X Axis tab
- Under Data Selection, for Variable, choose Theta
- Check Invert Axis
- Click Apply

etails of <b>Pressure vs Th</b>	eta	_				
General Data Series	X Axis	Y Axis	Line Display	Chart Displ	ay	
Data Selection					-	
Variable Theta				•		
Take absolute value of	f data point	ts				
Axis Range						
Determine ranges auto	omatically					
Min -1.0 Max 1.0						
Logarithmic scale		V Invert	t axis		Ε	
Axis Number Formatting						
Determine the number format automatically						
Precision 3		Scie	ntific	-		
Axis Labels						
Use data for axis labe	s					
Custom Label X Axis <u< td=""><td>inits&gt;</td><td></td><td></td><td></td><td>-</td></u<>	inits>				-	
Apply Export			Reset	Defau	lts	

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

Details of <b>Pressure vs Theta</b>								
	General Da	ata Series	X Axis	Y Axis	Line Display	Chart Display		
	Data Selection						-	
	Variable Pressure 🔹		•					
ľ	Boundary Data	3	O Hybrid		Onser	vative		

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*.

Sketching Toolboxes	ą				
Draw					
Modify					
Dimensions					
Constraints 🔺					
r Equal Radius ∢ È Equal Length ∢ È Equal Distance					
📶 Auto Constraints Global: 🔽 Cursor: 🔽					
Settings 👻					
Sketching Modeling					

#### 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 2 .
- 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 reentered)
- 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