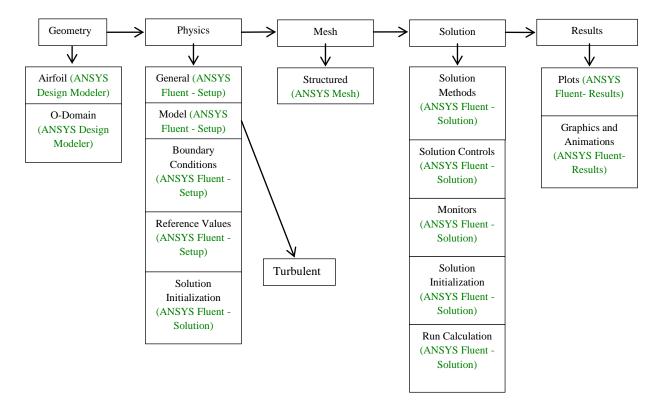
Simulation of Turbulent Flow around an Airfoil

57:020 Mechanics of Fluids and Transfer Processes CFD Pre-Lab 2

By Timur Dogan, Michael Conger, Maysam Mousaviraad, Tao Xing and Fred Stern IIHR-Hydroscience & Engineering The University of Iowa C. Maxwell Stanley Hydraulics Laboratory Iowa City, IA 52242-1585

1. Purpose

The Purpose of CFD PreLab 2 is to simulate **turbulent** flow around a Clarky airfoil following the "CFD process" with an interactive step-by-step approach. Students will have "hands-on" experiences using ANSYS to compute pressure, lift and drag coefficients using both **viscous and inviscid** models. Students will **validate** simulation results with EFD data measured at EFD Lab 3, analyze the differences and possible numerical errors, and present results in Lab report.



Flow chart for "CFD Process" for airfoil flow

2. Simulation Design

In EFD Lab 3, you have conducted experimental study for turbulent flow around a ClarkY airfoil (Re=300,000) for two angles of attack 0 and 16 degrees. The pressure on the foil surface you have measured will be used for CFD PreLab 2. In CFD PreLab 2, simulation will be conducted under the same conditions of EFD Lab 3 (geometry, Reynolds number, fluid properties) at angle of attack 0 degree using both viscous and inviscid models. Simulation results will be validated by your own EFD data.

The problem to be solved is turbulent flow around the ClarkY airfoil with angle of attack (α)

Table 1 - Geometry dimensions				
Parameters	Symbol	Unit	Value	
Chord Length	С	m	0.3048	
Domain radius	Rc	m	6	
Angle of attack	α	m	0	

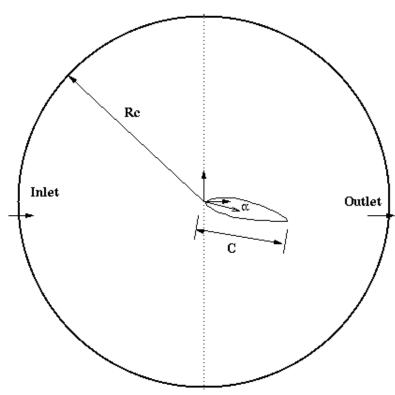
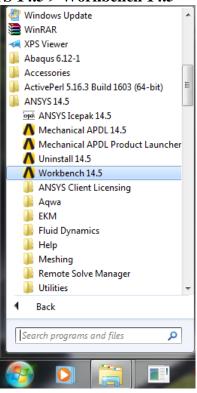


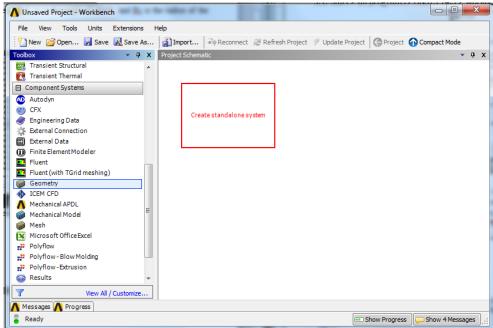
Figure 1 – Geometry

3. Open ANSYS Workbench

3.1. Start > All Programs > ANSYS 14.5 > Workbench 14.5



3.2. From the ANSYS Workbench home screen (**Project Schematic**), drag and drop a **Geometry, Mesh**, and **Fluent** component from the **Component Systems** drop down menu onto the **Project Schematic**. The **Project Schematic** should resemble the schematic below. Rename the components as per below.

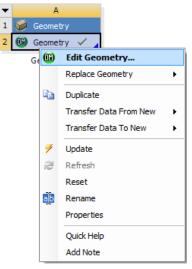


N Unsaved Project - Workbench		
File View Tools Units Extensions H	elp	
🎦 New 💕 Open 閕 Save 🔣 Save As	👔 Import 🏼 🖗 Reconnect 🛛 🥰 Refresh Project 🍼 Update Project	Compact Mode
Toolbox	Project Schematic	- д х
Transient Structural	*	
Transient Thermal		
Component Systems	▼ A B	▼ C
Autodyn	1 🥪 Geometry 1 👹 Mesh	1 Eluent
CFX CFX	2 🥪 Geometry 👕 🗕 🗕 2 📦 Mesh 😨 🚬	🗕 2 🍓 Setup 😨 🖌
Engineering Data		3 Solution ?
🔆 External Connection	CFD Pre-Lab 2 Turbulent Flow Airfoil O-Mesh	
🔁 External Data		Viscid
Finite Element Modeler		
Fluent		
Fluent (with TGrid meshing)		
Geometry		▼ D
Machanical APDI		1 Sluent
Mechanical Model		🗣 2 🍓 Setup 🔗 🧧
Mesh		3 🕥 Solution 😨 🛓
Microsoft Office Excel		Inviscid
20 Polyflow		
Polyflow - Blow Molding		
Polyflow - Extrusion		
💿 Results 💌		
View All / Customize	•	>
Messages A Progress		
Orag a Toolboxitem on top of a system to reu	e components and exchange data.	how Progress 🧔 Show 4 Messages 💠

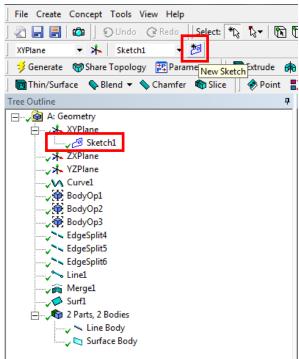
- 3.3. Create a Folder on the H: Drive called *CFD Pre-Lab 2*.
- 3.4. Save the project file by clicking **File** > **Save As...**
- 3.5. Save the project onto the H: Drive in the folder you just created and name it *CFD Pre-Lab 2 Turbulent Flow*.

4. Geometry

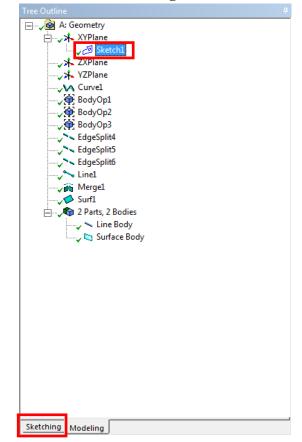
- 4.1. Right click **Geometry** then select **Import Geometry** > **Browse...** Select **intro_airfoil.igs** and click **OK**.
- 4.2. Right click Geometry and select Edit Geometry...



4.3. Select XYPlane and click New Sketch button.



4.4. Select the sketch you created and click **Sketching** button.



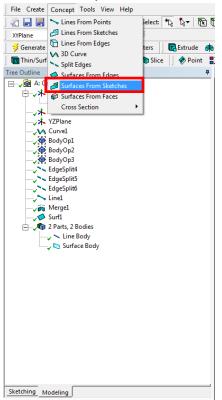
4.5. **Sketching Toolbox** > **Draw** > **Circle**. Click on the xy-plane origin and click behind the airfoil.

sketching Toolboxes	k	*	Maphics	
	Draw	~		
 ↓ Line ♦ Tangent Line ♦ Line by 2 Tangen ▲ Polytine ④ Polytine ④ Polytine □ Rectangle ☑ Rectangle by 3 F 				
Circle				
"Arc by 3 Peints Arc by 3 Peints Arc by Center ③ Elipse 3 Spline * Construction Po Ø Construction Po	int			
	Modify			
	Dimensions			
	Constraints			
	Settings			
				1
Sketching Modelin	19]			
Details View		4		1
- Details of Sketch1				
Sketch	Sketch1			1
Sketch Visibility Show Constraints	Show Sketch		N.	1
Edges: 1	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1			
Full Circle	C/7			1
run sincle	547			

4.6. **Sketching Toolboxes** > **Dimensions** > **General**. Click on the circle and change the diameter to 12m.

File Create Conce	ept Tools View Help
2 🔒 🛃 🧔	🕽 Undo @ Redo 🛛 Select: 🆎 🦙 🎦
	🛧 Sketch1 👻 ಶ
🧚 Generate 🛛 👘 Sl	hare Topology 😰 Parameters 📗 💽 Extrude 👩
Thin/Surface	💊 Blend 🔻 🔦 Chamfer 🏾 🏘 Slice 🔢 🛷 Point
ketching Toolboxes	
	Draw
	Modify
	Dimensions
General	
Horizontal	
1 Vertical	
Length/Distance	
Radius	
Diameter	
Angle	
Semi-Automatic	
Edit	
Move	
Animate	
B.BI Display	
M1	
	Constraints
	Settings
Sketching Modeling	g
Details View	-
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 1	1
D1	12 m
Edges: 1	
Full Circle	Cr7

4.7. Concept > Surface From Sketches. Select your sketch and click the Generate button.

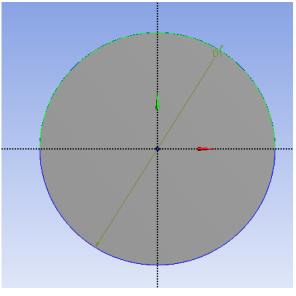


4.8. Create > Boolean. Change operation to Subtract then select the circle for Target Bodies then select the airfoil for Tool Bodies and click Generate. This will subtract the airfoil surface from the circle.

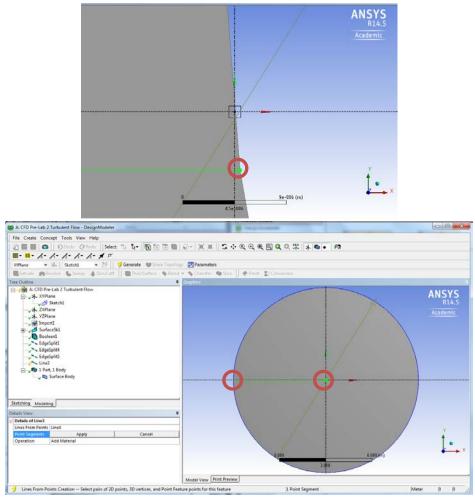


De	etails View	ą.
-	Details of Boolean1	
	Boolean	Boolean1
	Operation	Subtract
	Target Bodies	1 Body
	Tool Bodies	1 Body
	Preserve Tool Bodies?	No

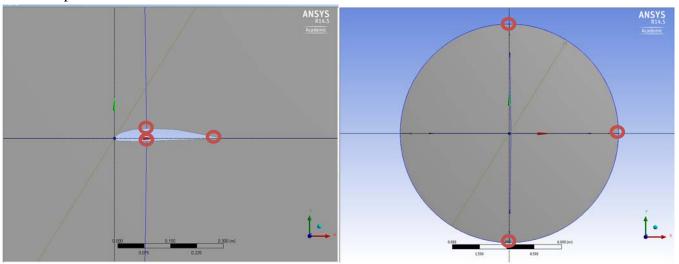
4.9. **Concept** > **Split Edges.** Select the perimeter of the circle and click **Apply.** Select **Generate.** This should split the circle into two semicircles. You can see the semicircles by selecting the perimeter above and below the x-axis.



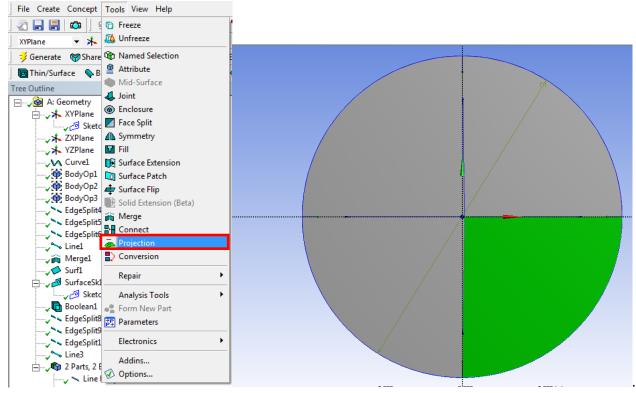
4.10. Repeat the process from 4.9 on the two semicircles. This should yield four circular quadrants.
4.11. Concept > Lines From Points. Draw a line from the point on the circle to the point on the airfoil making sure to hold Ctrl while doing so. (Note: The point on the airfoil is not exactly on the origin. Zoom in and find the point just below the origin and select that point. The images below show the locations of the points circled. When selecting points to generate lines with, be sure to select the point on circle and then the point on the airfoil as to avoid complications when sizing mesh).



- 4.12. Once you select both points hit **Apply.** Then click **Generate.**
- 4.13. Repeat this process creating lines from the edge of the circle to the airfoil starting from the circle and ending at the airfoil. The images below show the locations of the points on the airfoil and the points on the circle.



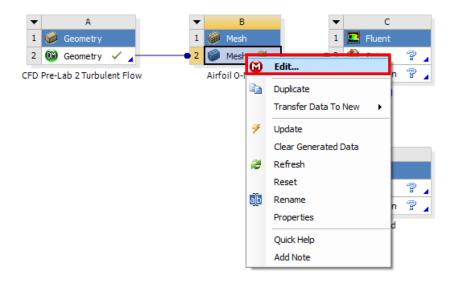
4.14. **Tools** > **Projections**. Select the four edges you created for **Edges** and click **Apply** and select the circle for **Target** and click **Apply**, then click **Generate**. This will split your geometry into four sections.



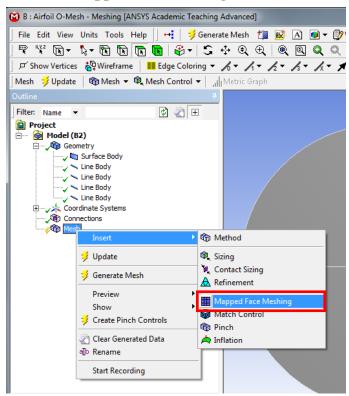
4.15. **File** > **Save project** and exit.

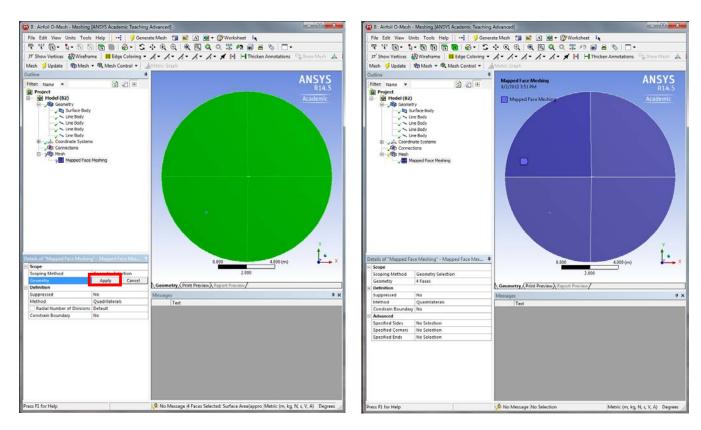
5. Mesh Generation

5.1. From Workbench home screen right click on Mesh and select Edit... from the dropdown menu.

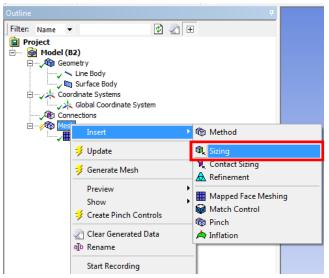


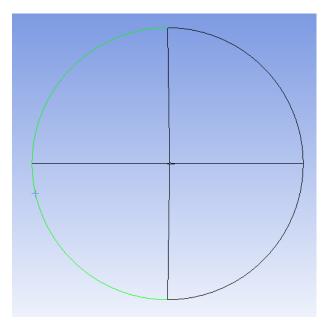
5.2. Right click **Mesh** then **Insert** > **Mapped Face Meshing**. Select the four surfaces then click **Apply**.





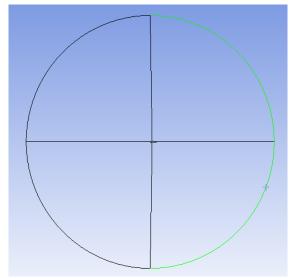
5.3. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.





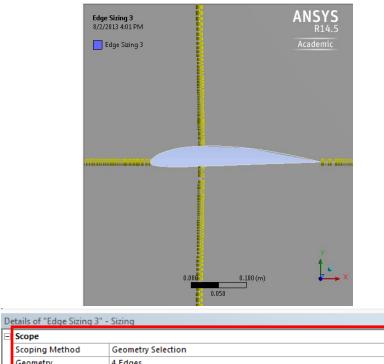
D	Details of "Edge Sizing" - Sizing 7				
E	Scope				
	Scoping Method	Geometry Selection			
	Geometry	2 Edges			
E	Definition	L			
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	50			
	Behavior	Hard 💌			
	Bias Type	No Bias			

5.4. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.



Details of "Edge Sizing 2" - Sizing 🛛 🗛					
-	Scope				
	Scoping Method	Geometry Selection			
	Geometry	Geometry 2 Edges			
-	Definition				
	Suppressed	No			
Type Number of Divisions		Number of Divisions			
	Number of Divisions	75			
	Behavior	Hard			
	Bias Type	No Bias			

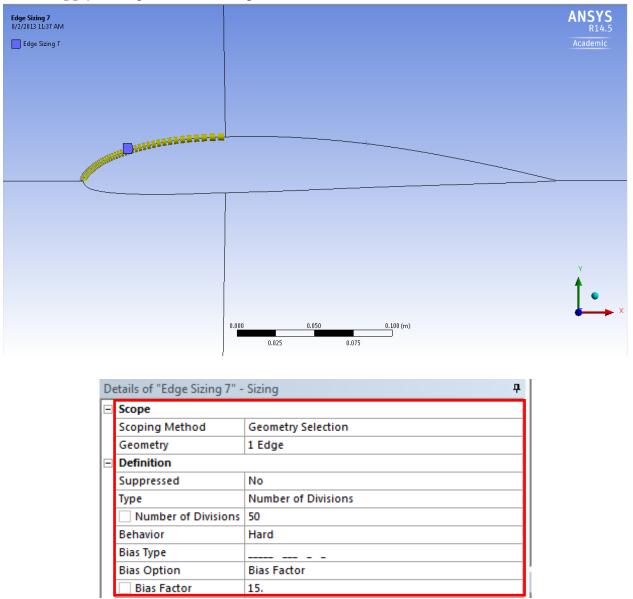
5.5. Right click **Mesh** and **Insert** > **Sizing**. Select all for lines leading from the circle to the airfoil surface, and click **Apply**. Change parameters as per below. (Note: If you did not create the lines starting from the outer circle and ending on the airfoil surface, you may have issues with biasing. If this is your case, size the lines individually making sure that the sizing is finest at the surface of the airfoil.)



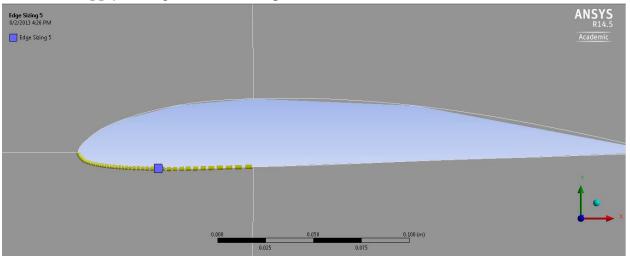
Ξ	Scope		
	Scoping Method	Geometry Selection	
Geometry 4 Edges			
Ξ	Definition		
	Suppressed No		
	Туре	Number of Divisions	
	Number of Divisions	200	
	Behavior	Hard	
	Bias Type		
	Bias Option	Bias Factor	
	Bias Factor	5000.	

џ

5.6. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the top of leading edge of the airfoil and click **Apply**. Change Parameters as per below.

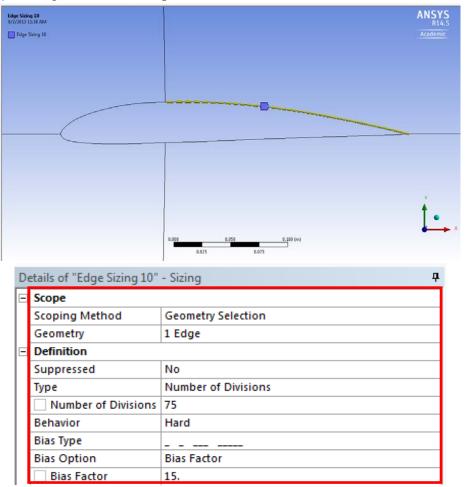


5.7. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the bottom of leading edge of the airfoil and click **Apply**. Change Parameters as per below.

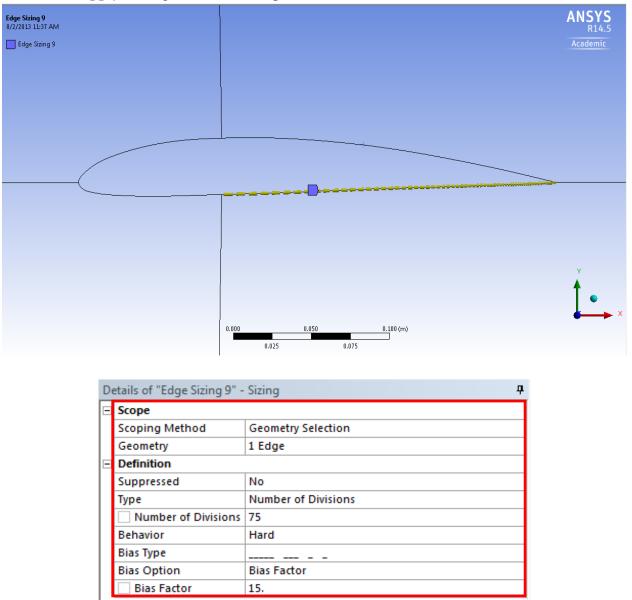


De	Details of "Edge Sizing 5" - Sizing 4				
Ξ	Scope				
	Scoping Method	Geometry Selection			
	Geometry	1 Edge			
Ξ	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	50			
	Behavior	Hard			
	Bias Type				
	Bias Option	Bias Factor			
	Bias Factor	15.			

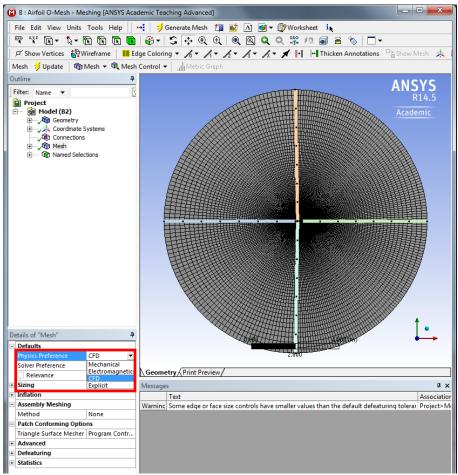
5.8. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the top of trailing edge of the airfoil and click **Apply**. Change Parameters as per below.



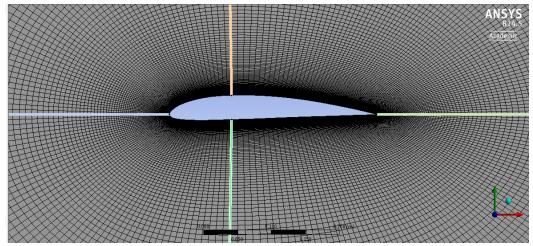
5.9. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the bottom of trailing edge of the airfoil and click **Apply**. Change Parameters as per below.



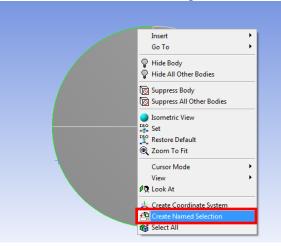
5.10. Click on **Physics Preference** under the **Details of "Mesh"**. Change the mesh type to **CFD** from **Mechanical**.



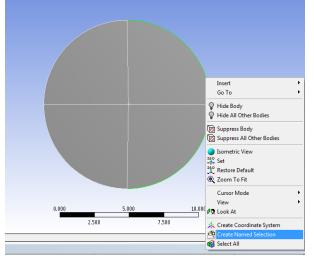
5.11. Click **Generate Mesh**. Click on the **Mesh** button under the **Outline** and make sure it resembles the mesh below.



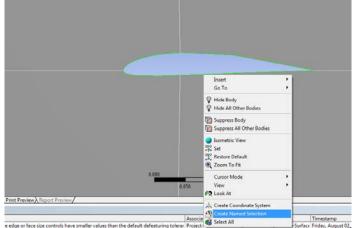
5.12. Hold **Ctrl** and select the two left most semicircle arcs, right click on them and select **Create Named Selection**, name the selection *inlet*. Use the edge select button from the toolbar.



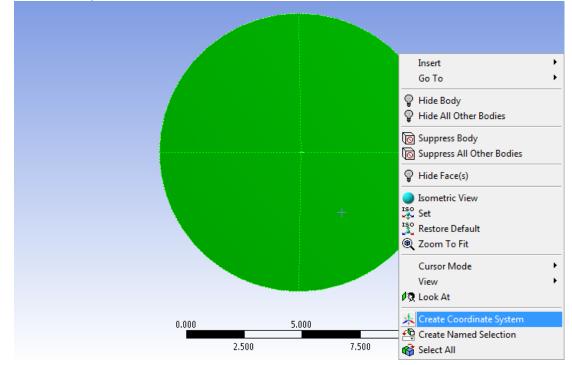
5.13. Do the same for the two right most semicircle arcs and name them *outlet*.



5.14. Select the four regions that make the airfoil, right click and **Create Named Selection** and name it *airfoil*.



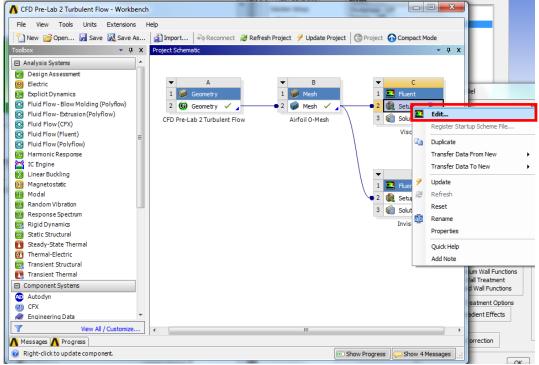
5.15. Now use the face button to select the four semicircle quadrants and **Create Named Selection** and name them *fluid*.



5.16. **File > Save Project.** Exit the window

6. Setup

6.1. From the Workbench home screen right click on Setup and select Edit...



6.2. Select **Double Precision** and click **OK**.

Fluent Launcher (Setting Edit Only)	
ANSYS	Fluent Launcher
Dimension 2D 3D Display Options ✓ Display Mesh After Reading ✓ Embed Graphics Windows ✓ Workbench Color Scheme ✓ Do not show this panel again 	Options Double Precision Processing Options Serial Parallel
	iancel <u>H</u> elp 🔻

6.3. Solution Setup > Models > Viscous –Laminar > Edit... Change the parameters as per below and click OK. (For the inviscid case, select Inviscid from the Viscous Model menu.)

C:Viscid Fluent [2d, dp, pbns, lam] [ANSYS Aca	demic Teaching Advanced]		
File Mesh Define Solve Adapt Surface	Display Report Parallel View Help		
in is 🕶 - 🖬 - 🗃 🎯 is 🔂 🤤 🗨	🥕 🔍 🏷 📑 🗖 🚺 🖬 Viscous Model		×
Meshing Mesh Generation Solution Setup Generation Materials Phases Cell Zone Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Setting flut Setting into	c1-Epsion ieqn) ieqn) (Seqn) imilation (SAS) TKE Prandtl Number 1 User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number Il Functions TKE Prandtl Number Indition (SAS) Vist Prandtl Number Inone TOR Prandtl Number none OK Cancel Help id (inixture) Done	

6.4. Solution Setup > Materials > air > Create/Edit... Change parameters as per experimental data and click Change/Create. (For inviscid model, you do not need to input Viscosity parameter.)

	, dp, pbns, ske] [ANSYS Academic Teaching Ac Solve Adapt Surface Display Report			A A A	
	Solve Adapt Sulface Display Report	· · · · · · · · · · · · · · · · · · ·		V	
Meshing	Materials	1: Mesh 🗸			
Mesh Generation	Materials	Create/Edit Materials			×
Solution Setup General Models	Ekid air Solid aluminum	Name air	Material Type fluid	•	Order Materials by
Materials Fhases Cell Zone Conditions	auminum	Chemical Formula	Fluent Fluid Materia	ls 🗸	Chemical Formula
Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values		Properties	Mixture none	~	User-Defined Database
Solution Methods Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animatio Plots Reports	ns Create/Edit Delete Help	Density (kg/m3) 1.225 Viscosity (kg/m-5) Constant 1.7894e-05		Edt	
		Change Setting interior-fluid (mi Setting surface_body (mixt Setting inlet (mixture) Setting outlet (mixture)	Done. Lixture) Done. Lure) Done.	close Help	y)

Use the air properties at the **room temperature** when you conducted EFD Lab3. **You can use the following** website to calculate air properties from the temperature:

http://www.mhtl.uwaterloo.ca/old/onlinetools/airprop/airprop.html

The values in the figure above are for 24° temperature.

NOTE: viscosity used in ANSYS is the **dynamic viscosity** $(kg/m \cdot s)$, **NOT kinematic viscosity** (m^2/s)

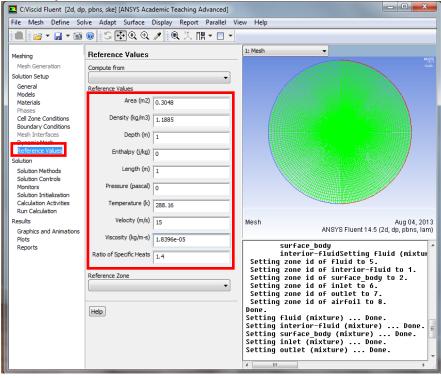
6.5. Solution Setup > Boundary Conditions > inlet > Edit... Change parameters as per experimental data and click OK. The value can be found from the EFD Lab 3 data reduction sheet. (For inviscid model you do not need to specify turbulence parameters.)

*	1 V	L /		
C:Viscid Fluent [2d, dp	o, pbns, ske] [ANSYS Academic Teaching Advanced]			
File Mesh Define So	lve Adapt Surface Display Report Parallel V	/iew Help		
i 📖 i 📂 🕶 🖬 🔻 🞯	❷ 5 ፼€ € ∥ € 芁 開 - □ -			COLOR MORE SHOP
Meshing	Boundary Conditions	Velocity Inlet		×
Mesh Generation	Zone	Zone Name		
Solution Setup	airfoil	inlet		
General	inlet		1 1 1	
Models Materials	outlet	Momentum Thermal Radiation Specie	es DPM Multiphase U	DS
Phases	surface_body	Velocity Specification Method	Components	•
Cell Zone Conditions		Reference Frame		
Boundary Conditions Mesh Interfaces				•
Dynamic Mesh		Supersonic/Initial Gauge Pressure (pascal)	0	constant 🔻
Reference Values		X-Velocity (m/s)	15	constant 👻
Solution				constant
Solution Methods		Y-Velocity (m/s)	0	constant 👻
Solution Controls Monitors		Turbulence		
Solution Initialization				
Calculation Activities	Phase Type ID	Specification Method		▼
Run Calculation Results	Phase Type ID mixture Velocity-inlet V	Turbulent Kinetic Energy (m2/s2)	0.08	constant 👻
Graphics and Animations		Turbulent Dissipation Rate (m2/s3)	7.4	
Plots	Edit Copy Profiles		7.4	constant 🔹
Reports	Parameters Operating Conditions			
	Display Mesh Periodic Conditions		K Cancel Help	
		Setting zone id of surface		
	Help	Setting zone id of inlet Setting zone id of outlet		
		Setting zone id of airfoi		
		Done.		
		Setting fluid (mixture) Setting interior-fluid (mix		
		Setting surface body (mixto		
		Setting inlet (mixture)	. Done.	
		Setting outlet (mixture) .	Done.	
		<	•	

6.6. Solution Setup > Boundary Conditions > Outlet > Edit... Change the parameters as per below and click OK. (For inviscid model you do not need to specify turbulence parameters.)

C:Viscid Fluent [2d, dp	, pbns, ske] [ANSYS Academic Teaching Advanced]	
File Mesh Define So	lve Adapt Surface Display Report Parallel Vie	w Help
i 📖 i 📂 🕶 🖬 🔻 🔟	@ 5⊕€€∕/ €Ҳ∥-□-	
Meshing	Boundary Conditions	Pressure Outlet
Mesh Generation	Zone	Zone Name
Solution Setup	airfoil	outlet
General Models	inlet outjet	Momentum Thermal Radiation Species DPM Multiphase UDS
Materials	surface_body	
Phases Cell Zone Conditions		Gauge Pressure (pascal) 0 constant
Boundary Conditions		Backflow Direction Specification Method Normal to Boundary
Dynamic Mesh		Average Pressure Specification
Reference Values		Target Mass Flow Rate
Solution		Turbulence
Solution Methods Solution Controls		Specification Method Intensity and Viscosity Ratio
Monitors		Backflow Turbulent Intensity (%) 3.25
Solution Initialization Calculation Activities		Backflow Turbulent Viscosity Ratio
Run Calculation	Phase Type ID	0.0035
Results	mixture v pressure-outlet v 7	
Graphics and Animations Plots	Edit Copy Profiles	OK Cancel Help
Reports	Parameters Operating Conditions	interior-fluidSetting fluid (mixtur
	Display Mesh Periodic Conditions	Setting zone id of fluid to 5.
		Setting zone id of interior-fluid to 1. Setting zone id of surface body to 2.
	Help	Setting zone id of inlet to 6. Setting zone id of outlet to 7

6.7. **Solution Setup** > **Reference Values**. Change parameters as per below. The velocity, temperature, density, and viscosity should be entered from EFD data.



7. Setup

7.1. **Solution** > **Solution** Methods. Change parameters as per below. (For inviscid case you do not need to input turbulence parameters.)

C:Viscid Fluent [2d, dp	, pbns, ske] [ANSYS Academic Teaching Advanced]	
File Mesh Define Sol	ve Adapt Surface Display Report Parallel Vie	w Help
i 💼 i 📂 🕶 🖬 🕶 🚳	@∥\$\$₽€€ ↗∥® 洙 ⊪ - □ -	
Meshing	Solution Methods	1: Mesh 🗸
Mesh Generation	Pressure-Velocity Coupling	
Solution Setup	Scheme	
General	SIMPLE	
Models Materials	Spatial Discretization	
Phases		
Cell Zone Conditions	Gradient	
Boundary Conditions Mesh Interfaces	Green-Gauss Cell Based	
Dynamic Mesh	Standard	
Reference Values	Momentum	
Solution	Second Order Upwind	
Solution Methods Solution Controls	Turbulent Kinetic Energy	
Monitors	Second Order Upwind 👻	
Solution Initialization	Turbulent Dissipation Rate	
Calculation Activities Run Calculation	Second Order Upwind 🔹 👻	
Results	Transient Formulation	Mesh Aug 04, 2013
Graphics and Animations	· · · · · · · · · · · · · · · · · · ·	ANSYS Fluent 14.5 (2d, dp, pbns, lam)
Plots	Non-Iterative Time Advancement	
Reports	Pseudo Transient	surface_body interior-fluidSetting fluid (mixtur
	High Order Term Relaxation Options	Setting zone id of fluid to 5.
	Default	Setting zone id of interior-fluid to 1.
	Delate	Setting zone id of surface_body to 2. Setting zone id of inlet to 6.
		Setting zone id of outlet to 7.
	Help	Setting zone id of airfoil to 8.
		Done. Setting fluid (mixture) Done.
		Setting interior-fluid (mixture) Done.
		Setting surface_body (mixture) Done. 📗
		Setting inlet (mixture) Done. Setting outlet (mixture) Done.

		×

7.2. Solution > Solution Controls. Change Parameters as per below. (If you have problems with the solution converging, you may decrease the Under –Relation Factors.) (For inviscid case you do not need to specify turbulence parameters.)

C:Viscid Fluent [2d, dp, pl	bns, ske] [ANSYS Academic Teaching Advanced]	
File Mesh Define Solve	Adapt Surface Display Report Parallel Vie	ew Help
i 📖 i 📂 🕶 🔛 🔻 🞯 🔞) S๋健�� 〃 � Հ 開 - □ -	
	colution Controls	1: Mesh 🔹
	Inder-Relaxation Factors	
Colution Sotup	Body Forces	
General Models	1	
	Momentum	
Phases Cell Zone Conditions	0.5	
Boundary Conditions	Turbulent Kinetic Energy	
Mesh Interfaces Dynamic Mesh	0.8	
Reference Values	Turbulent Dissipation Rate	
Solution	0.8	
Solution Methods Solution Controls	Turbulent Viscosity	
Monitors	1	
Solution Initialization Calculation Activities		
	Default	
	Equations Limits Advanced	Mesh Aug 04, 2013 ANSYS Fluent 14.5 (2d, dp, pbns, lam)
Graphics and Animations Plots		
Reports	Help	surface_body interior-fluidSetting fluid (mixtur
		Setting zone id of fluid to 5.
		Setting zone id of interior-fluid to 1. Setting zone id of surface body to 2.
		Setting zone id of inlet to 6.
		Setting zone id of outlet to 7. Setting zone id of airfoil to 8.
		Done. Setting fluid (mixture) Done.
		Setting interior-fluid (mixture) Done
		Setting surface_body (mixture) Done.
		Setting outlet (mixture) Done.

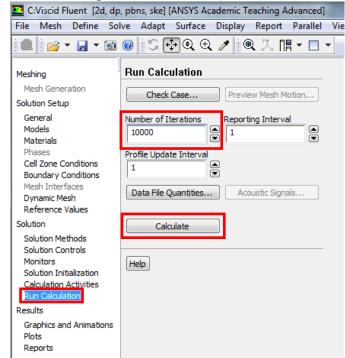
7.3. **Solution** > **Monitors** > **Residuals** –**Print**, **Plot**. Change the convergence limit to 1e-05 for all five equations and click OK. (For inviscid case you do not need to specify convergence criterion for turbulence parameters. You only need to specify criterion for three equations.)

C:Viscid Fluent [2d, dp	o, pbns, ske] [ANSYS Academic Teaching Advanced]			and a second second	PORTAGE IN COMPANY IN COMPANY
File Mesh Define So	lve Adapt Surface Display Report Parallel Vie	ew Help			2 Barrister
i 📖 i 📂 🕶 🔛 🕶 🞯	❷∥S ❹Q @ ↗ Q Ҳ I - □ -			Salar A	A CARLES
Meshing	Monitors	Residual Monitors		Constanting of the second	×
Mesh Generation	Residuals Statistic and Force Monitors	Options	Equations		
Solution Setup	Residuals - Print, Plot	Print to Console	continuity 📝		1e-05 🔺
General Models	Statistic - Off	V Plot	x-velocity	V	1e-05
Materials		Window	y-velocity		1e-05
Phases Cell Zone Conditions		1 Curves Axes	y-velocity	Laborat	
Boundary Conditions	Create - Edit Delete	Iterations to Plot	k 🔽		1e-05
Mesh Interfaces Dynamic Mesh	Surface Monitors	1000	epsilon		1e-05 👻
Reference Values			Residual Values		Convergence Criterion
Solution		Iterations to Store	Normalize	Iterations	absolute 👻
Solution Methods		1000		5	
Monitors			Scale		
Solution Initialization Calculation Activities	Create Edit Delete		Compute Local Scale		
Run Calculation	Volume Monitors		ot Renormalize	Cancel He	
Results				Cancer ne	P
Graphics and Animations Plots			4.3 (20, 0p, ppns, ram)		
Reports		surface_body interior-fluidSetting	a fluid (mistur		
	Create Edit Delete	Setting zone id of fluid to	5.	A DESCRIPTION OF	
	Create Edit Delete	Setting zone id of interior- Setting zone id of surface b			
		Setting zone id of inlet to	6.		
		Setting zone id of outlet to Setting zone id of airfoil t		And a state of the	
		Done.		-	
		Setting fluid (mixture) D Setting interior-fluid (mixtu			
	Convergence Manager	Setting surface_body (mixture	e) Done. 👘	and the second second	and the second second
		Setting inlet (mixture) D Setting outlet (mixture)			
	Help		•		
L		•	<u>t</u> ▲	Contraction of the local division of the loc	

7.4. Solution > Solution Initialization. Change the parameters as per below and click Initialize. (For inviscid case you do not need to initialize turbulence parameters.)

a han alwal (ANSVS A reductio Tranships Advance			
	-		
		r Help	
_❷ 5 🕀 € € 🥒 € 🏷 🖪 ▼ 🗉			
Solution Initialization	[1	1: Mesh 🗸	ANSYS
Initialization Methods Hybrid Initialization Standard Initialization Compute from Reference Frame Reference Frame Relative to Cell Zone Initial Values Gauge Pressure (pascal) X Velootry (m/s) 15			
Y Velocity (m/s)	=		Aug 04, 2013 tt 14.5 (2d, dp, pbns, lam)
Turbulent Kinetic Energy (m2/s2) 0.08 Turbulent Dissipation Rate (m2/s3) 7.4 Initialize Reset Reset DPM Sources Reset Statistics		Setting fluid (mixture) Setting interior-fluid (mix Setting surface_body (mixtu Setting inlet (mixture)	50 5. pr-fluid to 1. _body to 2. 50 6. to 7. to 8. bone. ture) Done. pone.
	Initialization Image: Construction of the second secon	Image: Section of the section of th	Interview Initialization Compute from Initialization Initialization

7.5. Solution > Run Calculation. Change the Number of Iterations to 10000 and click Calculate.



Iteration history should look similar to the one below.

	e, pbns, ske] [ANSYS Academic Teaching Advanced] Ive Adapt Surface Display Report Parallel	View Help
🧉 t 🔒 t 📾	@ S配QQ 〃 奧 久 隅・□・	
	Run Calculation	1: Scaled Residualiti •
ieneration Setup	Check Case Preview Mesh Motion	Confectuals ASS
is .	Number of Iterations Reporting Interval	Ensiton 1=+00
ne Conditions wy Conditions	Profile Update Interval	1e01 -
nterfaces ic Mesh nce Values	Data File Quantities Acoustic Signals	1602
n Methods	Calculate	1603
n Controis rs n Initialization	Help	1e04
tion Activities		1e05
cs and Animations		1607
5		1008
		0 250 500 750 1000 1250 1500 1750 2000 2250 Iterations
		Iteration's
		Scaled Residuals Aug 05, 20
		ANSYS Fluent 14.5 (2d, dp, pbm, sk) ANSYS Fluent 14.5 (2d, dp, pbm, sk)
		2221 2.2018e-06 0.4064e-08 7.800b-08 1.4057e-05 0.22122 7.774 2222 2.1018e-06 0.4005e-08 7.8007b-08 2.4025c-04 0.22127 7.774 2222 2.10182b-08 0.4054e-08 7.8007b-08 2.4025c-04 0.22127 7.774 2223 2.10182b-08 0.4554e-08 7.8007b-08 2.4063c-06 1.4077e-08 1.2212b-076 22252 2.1776e-06 0.4007b-08 7.4007b-08 0.4007b-08 1.0077e-08 1.2017c-08 1.2017c-08 22272 2.1774e-08 0.4007b-08 7.4007b-08 0.4007b-08 1.0077e-08 1.0077e-08
		1 4
1 1	台 🔞 💌 🔥 🖸	- P- 15 4 100 40

7.6. File > Save Project.

8. Results

8.1. Plotting and Saving Residual History Solution > Monitors > Residuals –Print, Plot > Edit... > Plot > Cancel.

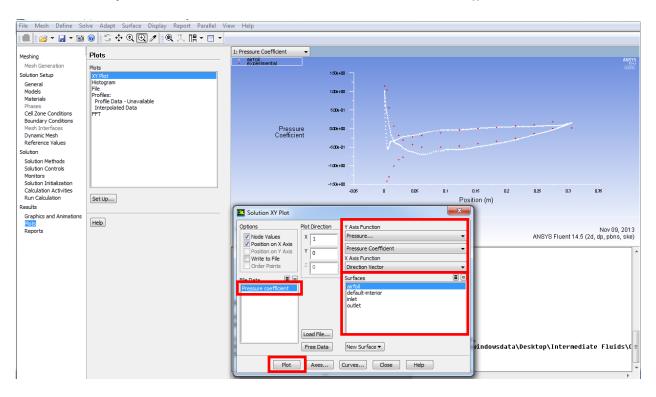
Meshing	Monitors				1: Scaled Residuals 🗸 🗸		
Mesh Generation	Residuals, Statistic and Force Monitors				Residuals continui x-velocit y-velocit		
Solution Setup	Residuals - Print, P	lot				1e+00 -	
General Models	Statistic - Off				- êpsilon	10.00	
Materials Phases						1e-01	
Cell Zone Conditions						1e-02 -	
Boundary Conditions Mesh Interfaces	Create 👻 Edit	Delete				Te-O2	Δ
Dynamic Mesh	Surface Monitors			-		1e-03 -	Λ_{a}
Reference Values						10.00	M.
Solution Solution Methods						1e-04 -	
Solution Controls							
Monitors				-		1e-05	
Solution Initialization Calculation Activities	Create Edit	. Delete					
Residual Monitors	part water.			-		100	×
Options		Equations					
Print to Console		Residual	Monitor Chec	k Conve	rgence Absolu	ite Criteria	
V Plot		continuity	V	V	- 1e-0	5	
Window		x-velocity			1e-0	F	-
	es Axes	x-velocity		V	IE-0	5	_
Iterations to Plot		y-velocity	V	V	1e-0	5	
1000		k		V	1e-0	5	•
		Residual Values			Con	vergence (Driterion
Iterations to Store		Normalize	Iter	rations	abs	solute	-
1000			5				
		Scale					
		Compute Loc	al Scale				
	OK Plot	Renormaliz	ze Cance		Help		

File > **Save Picture...** > **Save...** Make sure the parameters are as per below and click **Save...** Name the file *CFD Pre-Lab 2 Residual History* change the file directory to the CFD Pre-Lab 2 file you created on the H: drive and click **OK**.

Format	Coloring Fi	е Туре	Resolution
 EPS JPEG 	Color Gray Scale	Raster Vector	Width 960
PPM PostScript	Monochrome		Height 720
TIFF PNG	Options		
VRML Window Dump	 Landscape Orienta White Background 	auon	ow Dump Command ort -window %w

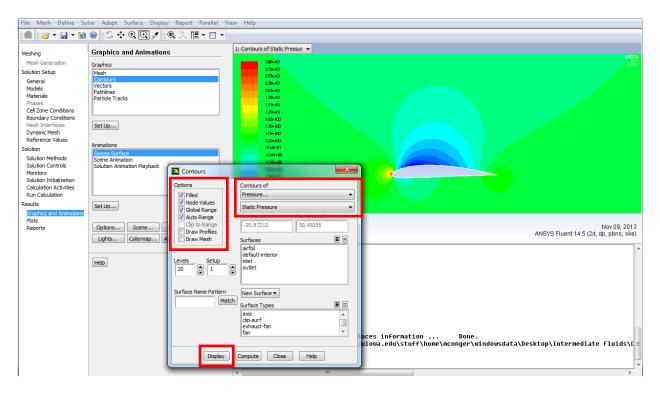
8.2. Plotting Pressure Coefficient Distribution with CFD and EFD Data

Results > **Plots** > **XY Plot** > **Set Up...** > **Load File...** Select Pressure-coef-attack0.xy. Change the parameters as per below and click **Plot**. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Pressure Coefficient Distribution*.



8.3. Plotting Contour of Pressure

Results > Graphics and Animations > Contours > Set Up... Change the parameters as per below and click **Display**. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Contour of Pressure*.



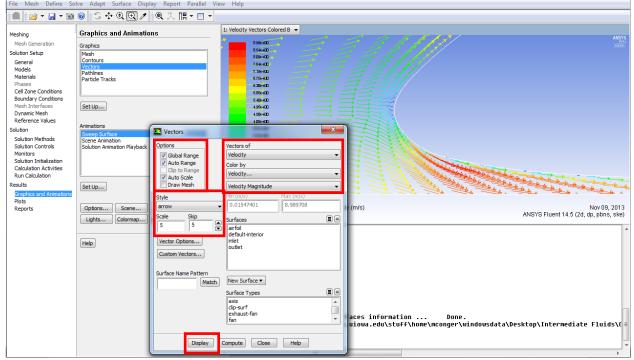
8.4. Plotting Contour of Velocity Magnitude

Results > **Graphics and Animations** > **Contours** > **Set Up...** Change the parameters as per below and click **Display**. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Contour of Velocity Magnitude*. Zoom in where you can see the airfoil clearly and the change in contour levels around the airfoil.

Meshing Graphics and Animations 1: Contours of Velocity Magn	
Mesh Generation Graphics 550x+00	
Solution Setup	
Control	
Madela Vectors	
Pathines	
Materials Particle Tracks 674400 Phases 9	
Cell Zone Conditions State Of Cell Conditions	
Boundary Conditions	
Mesh Interfaces Set Up.,	
Dynamic Mesh	
Reference Values	
Solution Animations 399+40	
Sweep Surface 31/2/00	
Selutian Centrale Division In the division 20040	
Solution Controls Solution Animation Playback	
Calculation Activities Options Contours of	
Run Calculation	
Results Set In Vode Values	
Craphics and Apimations	
Plata	
Percete Options Scene Clip to Range 0 8.984956 Not	09,2013
ANSYS Fluent 14.5 (2d. dn. r	ons, ske)
Lights Colomap A Draw Mesh Surfaces E =	
airfoil	· ·
default-interior Levels Setup injet	
Help Levels Setup intet	
Surface Name Pattern New Surface	
I we would be a set of the set of	
Surface Types	
axis	
dopauf exhaust fan	
aces information Done.	
piowa.edu\stuff\home\mconger\windowsdata\Desktop\Intermediate F1	ids\(≣
Display Compute Close Help	
Usphay Compute Cose Trep	

8.5. Plotting Velocity Vectors at Leading Edge

Results > **Graphics and Animations** > **Vectors** > **Set Up...** Change parameters as per below and click **Display**. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Vectors of Velocity at Leading Edge*. Zoom in on the leading edge.



8.6. Plotting Streamlines Close to Airfoil Surface

Results > **Graphics and Animations** > **Contours** > **Set Up...** Change parameters as per below and click display. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Streamlines Close to Surface*. You may need to adjust the **Min** and **Max** to adjust the flow over the airfoil.

💶 R:Fluent Fluent [2d, dp	o, pbns, ske] [ANSYS Academic	Teaching Advanced]			
File Mesh Define Sol	lve Adapt Surface Display	y Report Parallel V	iew Help		
i 🛋 i 📂 🕶 🖬 🕶 🚳	0 🕄 💠 Q 🕀 🖊	◉、洗 閒 ▾ □ ▾			
Meshing	Graphics and Animation	s	1: Contours of Stream Fu	uncti 👻	ANSYS
Mesh Generation	Graphics		しいの		
Solution Setup	Mesh				
General	Contours				
Models	Vectors		- 議報		
Materials	Pathlines Particle Tracks				
Phases	Particle fracks				
Cell Zone Conditions					
Boundary Conditions			- 一茶田		
Mesh Interfaces	Set Up		- ※1		
Dynamic Mesh			- 241		
Reference Values	Animations				
Solution	Sweep Surface				
Solution Methods	Scene Animation				
Solution Controls	Solution Animation Playback				
Monitors Solution Initialization		Contours		×	
Calculation Activities					
Run Calculation		Options	Contours of		
Results		Filled	Velocity	•	
	Set Up	✓ Node Values	Stream Function	*	
Graphics and Animations Plots		Global Range		Max (kg/s)	
Reports	Options Scene	Auto Range Clip to Range	Min (kg/s) 41	44	Nov 09, 2013
		Draw Profiles	41	**	ANSYS Fluent 14.5 (2d, dp, pbns, ske)
	Lights Colormap A	Draw Mesh	Surfaces		
			airfoil		
	Help		default-interior		
	(TEP)	Levels Setup	inlet outlet		
		100 🚔 1	outlet		
		Surface Name Pattern	New Surface 🔻		
		Mat	tch		
			Surface Types		
			axis	A	
			clip-surf exhaust-fan		
			fan	-	aces information Done.
			_		uiowa.edu\stuff\home\mconger\windowsdata\Desktop\Intermediate Fluids\[=
		Display	Compute Close	Help	•
					· · · · · · · · · · · · · · · · · · ·
					12:25 PM
🚱 🜔 🚺	📋 💽 🕘				▲ 😼 🗊 🜵 11/9/2013

8.7. Printing Lift and Drag Coefficients

Results > **Reports** > **Forces** > **Set Up...** Change parameters as per below and click **Print**. This prints out the drag force. If you change the X parameter to 0 and the Y parameter to 1, this prints out the lift force. If you change the X parameter to 1 and the Y parameter to 0, this prints out the drag force. Save these values on to the CFD Lab 2 report template.

195511/2 195522 195522 195622 195622	Force Reports Forces X X T Forces X T	
Contours of Stream Function (kg/s)	Moments Center of Pressure Z	Aug 05, 2013 ANSYS Fluent 14.5 (2d, dp, pbns, sko)
2234 2.1639e-06 9.4062e-08 7.7896e-08 2235 2.1624e-06 9.4014e-08 7.7784e-08 2236 2.1608e-06 9.3905e-08 7.7762e-08 2237 2.1593e-06 9.3917e-08 7.7739e-08 2238 2.1572e-06 9.3805e-08 7.7762e-08 2239 2.1572e-06 9.3820e-08 7.7769e-08 2239 2.1562e-06 9.3820e-08 7.7769e-08 2240 2.1547e-06 9.3771e-08 7.7669e-08 Forces Forces (n) 7.7669e-08 7.7762e-108 Forces Image: Forces (n) 9.3771e-08 7.7629e-108 Z010 2.1547e-06 9.3771e-08 7.7669e-08	2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2	Total 8) (0.88015666 13.115638 0)
Net (0.36176218 13.099	955 0) (0.51839448 0.015682267	0) (0.88015666 13.115638 0)
airfoil 0.36176218 0.	SCOUS Total Coefficients 51839448 0.88015666 0.0088768086 0.0127 51839448 0.88015666 0.0088768086 0.0127	720192 0.021596993
- III		

9. Exercises

Simulation of Turbulent Flow around an Airfoil

You must complete all the following assignments and present results in your CFD Lab 2 reports following the CFD Lab Report Instructions.

1. Validation using EFD Lab 3 data

Use the same flow conditions as those in your EFD Lab 3, including geometry (chord length, angle of attack 0) and Setup (Flow properties, inlet velocity). Use k-e model, 2nd order scheme, double precision with iteration number (10000) and convergent limit (1e-05). Run the simulation.
 Modify your EFD data of pressure coefficient in ANSYS format (sample EFD data format has been provided in CFD Lab 1) and import it into ANSYS Fluent for pressure coefficient distribution and conduct validation. Also compare the CFD lift coefficient value with EFD data.

- **Figures need to be saved:** 1. Time history of residuals (residual vs. iteration number); 2. Pressure coefficient distribution (CFD and EFD), 3. Contour of pressure, 4. Contour of velocity magnitude, 5. Velocity vectors (show the region that is interesting, such as the separation region when angle of attack is large or close to the leading edge of the airfoil), and 6. Streamlines close to airfoil surface.
- Data need to be saved: lift and drag coefficients

2. Inviscid flow simulation

Use the same conditions as those in exercise 1, except choose "**inviscid**" for "viscous model", set the iteration number to be (10000), and convergent limit to be 10^{-5} . Conduct the simulation and compare solutions with viscous flow results in exercise 1.

- **Figures need to be saved:** 1. Time history of residuals; 2. Pressure coefficient distribution (CFD only), 3. Contour of pressure, 4. Contour of velocity magnitude, 5. Velocity vectors (show the same region that your picked in exercise 1), and 6. Streamlines close to airfoil surface.
- Data need to be saved: lift and drag coefficients

3. Questions need to be answered in CFD Lab 2 report:

- 3.1. Does inviscid flow has boundary layer near the wall? Zoom in the near wall region and describe the differences of velocity vectors near the airfoil surface for inviscid and viscous flows.
- 3.2. What are the correct boundary conditions for velocity and pressure at "inlet" and "outlet".
- 3.3. What are the values for lift and drag coefficients for inviscid flow around the airfoil? Are they both zero?
- 3.4. Where are the highest and lowest locations for pressure and velocity magnitude? Why? Is pressure constant for inviscid flows around airfoil?
- 3.5. For turbulent flow around airfoil, try to qualitatively explain why there is a lift force (vertical up) on an airfoil using the contour plot of pressure or the XY plot of pressure coefficient distribution.