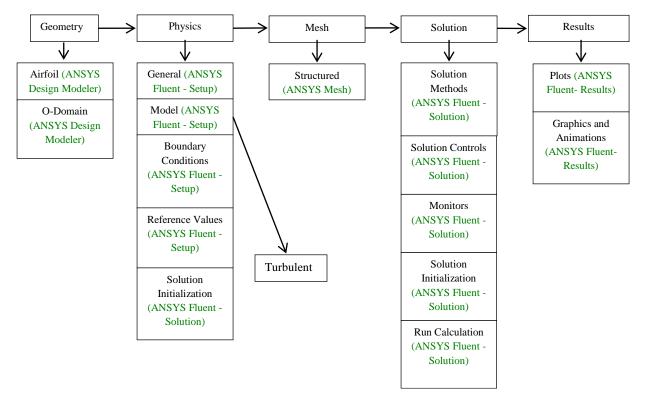
Simulation of Turbulent Flow around an Airfoil

57:020 Mechanics of Fluids and Transfer Processes CFD 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 Lab 2 is to conduct **parametric studies** for **turbulent** flow around Clarky airfoil following the "CFD process" by an interactive step-by-step approach. Students will have "hands-on" experiences using ANSYS to investigate the **effect of angle of attack** and **effect of different turbulence models** on the simulations results. These effects will be studied by comparing simulation results with EFD data. Students will 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 airfoil flow around a ClakY airfoil (Re \approx 300,000). The data you have measured were used for CFD PreLab 2. In CFD Lab 2, simulation will be conducted under the same conditions of EFD Lab 3, except angle of attack and turbulent models that will be changed in this lab. 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	12		
Angle of attack	α	m	16		

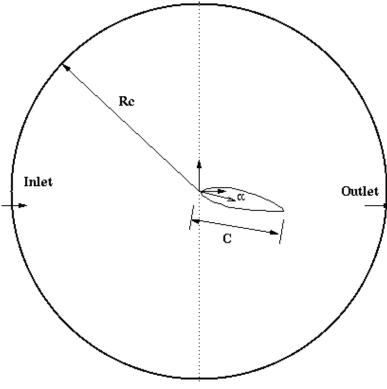
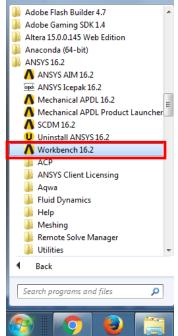


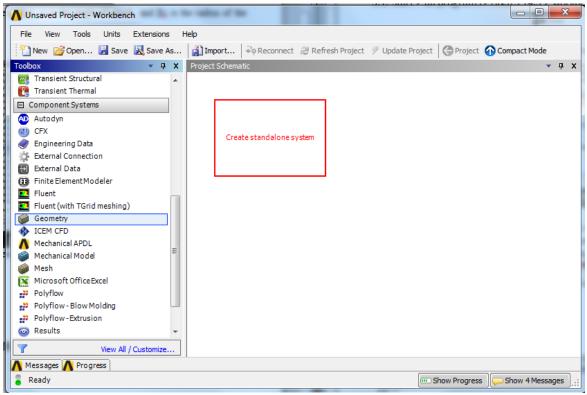
Figure 1 – Geometry

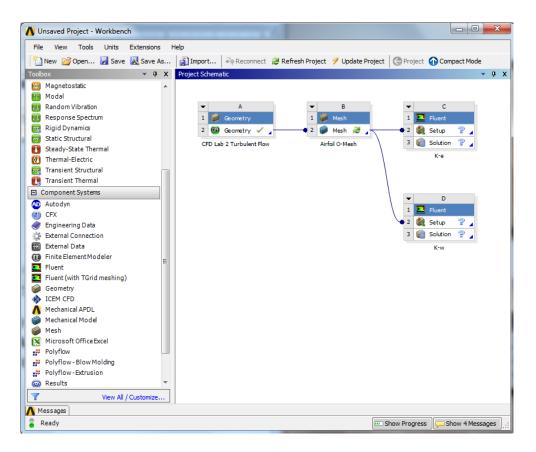
3. Open ANSYS Workbench

3.1. Start > All Programs > ANSYS 16.2 > Workbench 16.2



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

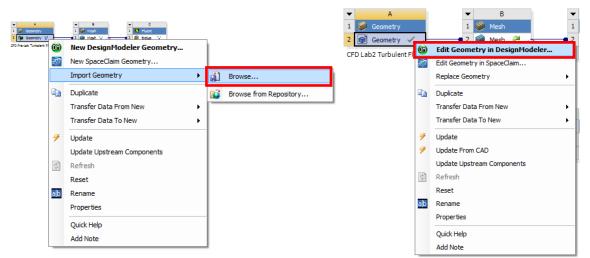




- 3.3. Create a Folder on the H: Drive called *CFD Lab 2*.
- 3.4. Save the project file by clicking **File** > **Save As...**
- 3.5. Save the project onto the H: Drive in the folder you just created and name it *CFD Lab 2 Turbulent Flow*.

4. Geometry

- 4.1. Right click **Geometry** then select **Import Geometry** > **Browse...** Select **airfoil.igs** and click **OK**. Note: The airfoil geometry is found on the class website <u>http://user.engineering.uiowa.edu/~fluids/</u> under the heading CFD PreLab2.
- 4.2. Right click Geometry and select Edit Geometry in DesignModeler...

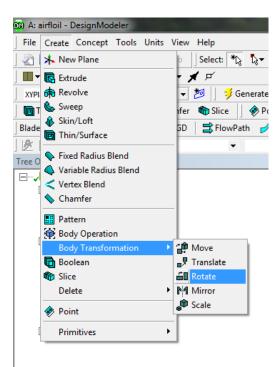


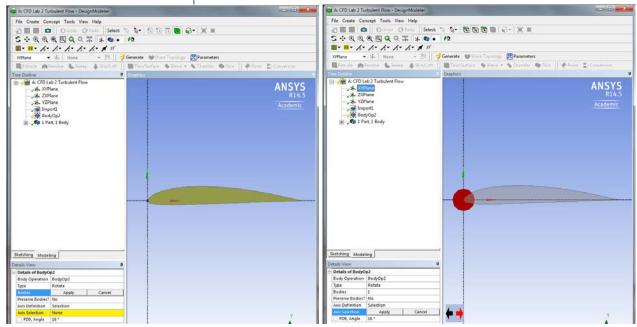
4.3. Click Generate.

4.4. Select **Sketching** > **Constraints** > **Auto Constraints**. Enable the auto constraints option to pick the exact point as below

Sketching Toolboxes	
Draw	
Modify	
Dimensions	
Constraints	A
777 Fixed	
🚃 Horizontal	
Vertical	
✓ Perpendicular	
Coincident	
Midpoint	
ণ Symmetry	
N Parallel	
Oncentric	
🛪 Equal Radius	
📌 Equal Length	
★ Equal Distance	
CON Auto Constraints	Global: 🗖 Cursor: 🔽
Settings	
Sketching	

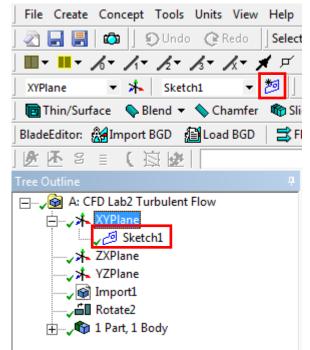
4.5. Create > Body Transformation > Rotate. Select the airfoil and click Apply. Click the yellow box labeled Axis Selection then click the XYPlane in the Tree Outline, then click Apply. Change the Angle to 16° and click Generate.





630 A: CFD Lab 2 Turbulent Flow - DesignModeler		(2) A: CFD Lab 2 Turbulent Flow - DesignModeler	
File Create Concept Tools View Help		File Create Concept Tools View Help	
🛛 🖉 📰 📰 🔹 🗍 Đưnđo 🛛 Piedo 🗍 Select 🐩			
5 · Q Q Q Q Q Q X + 0 ·	12	S + Q Q Q Q Q + + • +	PR
······································		h- h- h- h- h- h- # #	
XiPlane • 水 None • 천 👂	Senerate 🛯 🕸 Shine Topology 😨 Parameters	XYPlane - 🛧 None - 🏄 🛛	
Etaude Revolve & Sweep & Sinfort	Thin/Surface Stend - Schander Slice & Point D Conversion	Extrude 💏 Revolve 🗞 Sweep 🎄 Skin/Loft	Thin/Surface Selend - Schamfer Slice Point Conversion
	Graphics		🕈 Graphics 🗣
A CFD Lik 2 Fullward Flow 	ANSYS R14.5 Academic	See A C fO Lab 2 Totubert Flow - ↓ XVPlane - ↓ XVPlane - ↓ YVPlane - ↓ YVPlane - ↓ Seo(pg) Boo(pg) Boo(pg) Boo(pg) Boo(pg)	ANSYS R14.5 Academic
Setting Modeling Cetals for BodyOp2 Cetals for BodyOp2 Body Operation Body Operation Preserve Bodes: Preserve Bodes: Preserv		Details of BodyOp2 BodyOp2 BodyOp2 BodyOp2 Type Rolate Bodes 1 Preserve Bodies No Asis Definition Selection	
Axis Selection Plane Normal	V V	Axis Selection Plane Normal	Y
FD9, Angle 16 *		ED9, Angle 16.*	-

4.6. Select **XYPlane** and click New Sketch button.



4.7. Select the sketch you created and click sketching button.

Tree Outline 4	
⊢	
XYPlane	
・・・」で到 <u>Sketch1</u> ・・・・・・・・・・・・・・・・・・・・・・・・・・・・・・・・・・・・	
Import1	
Rotate2	
📺 🖓 1 Part, 1 Body	
Sketching Modeling	1

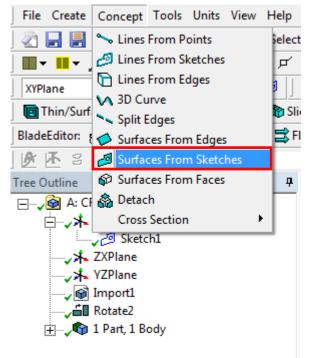
4.8. **Sketching Toolbox** > **Draw** > **Circle**. Click on the xy-plane origin and click behind the airfoil. (Click z-arrow at right bottom to set the view as perpendicular to xy-plane)

(m) A: CFD Lab 2 Turbulent Flow - DesignModeler		and the second se	- • • ×
File Create Concept Tools View Help	Contract Contract of Contract Contract		
🖉 🔜 🛃 📫 🕽 Undo @ Redo Select: 🆎 💱 🛐 🛐	R 🖩 🖓 🗉 🕱 🕽 🛠 @ 🕀 @ 🔍 @	Q 🞇 🗼 📦 🔸 🕅	
■ - I - K - K - K - K - K - K			
XYPlane • 🖈 Sketch1 • 🙋 💈 Generate 🐨 Share			
Sketchi S			
	12.1	rersion	
Sketching Toolboxes	7 Graphics		
Draw			ANSYS
Line			R14.5
C Tangent Line C Line by 2 Tangents			Academic
A Polyline			Academic
Polygon			
Rectangle			
Rectangle by 3 Points			
O Var		• \	
S Circle			
Circle by 3 Tangents			
Modify	▼		
Dimensions			
Constraints			
Settings	——————————————————————————————————————		
Sketching Modeling		/	
Details View	7	/	
Details of Sketch1			
Sketch Sketch1			Y
Sketch Visibility Show Sketch Show Constraints? No			+
Edges: 1			•
Full Circle Cr7		0.300 (m)	↓ ×
	0.0		
		0.150	
	Model View Print Preview	:	
Circle Click, or Press and Hold, for center of circle		No Selection	Meter 0 0

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

File Create Conce	ept Tools View Help
2 🔒 📕 🖚	OUndo @Redo Select: *b by the test
	Sketch1 👻 ಶ
Generate 🕅 Si	hare Topology 📴 Parameters 🛛 💽 Extrude 🏘
	Blend 🔻 💊 Chamfer 🏟 Slice 🛛 🚸 Point
Sketching Toolboxes	4 4 4 1 % 1
Sketening roonboxes	Draw
	Modify
	Dimensions
	Limensions
General 🖉	
Horizontal	
Length/Distance	
Radius	
Diameter	
Angle	
✓ Angre	
Edit	
Move	
Animate	
따라 Display	
Hd Disbidy	
	Constraints 🔍
	Settings
Sketching Modeling	
Details View	
 Details of Sketch1 	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
-	
D1	12 m
- Edges: 1	
Full Circle	Cr7

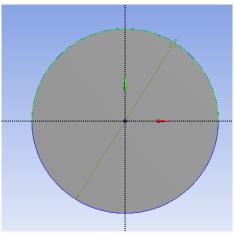
4.10. **Concept** > **Surface From Sketches**. Select **Sketch1** from the Tree Outline, click **Apply**, then click the **Generate** button.



4.11. Create > Boolean. Change operation to Subtract then select the circle for Target Bodies and click Apply then select airfoil (click on the airfoil until only the airfoil's color turns yellow) for Tool Bodies and click Apply then click Generate. This will subtract the airfoil surface from the circle.

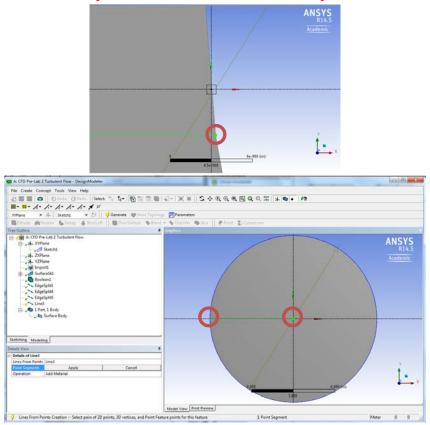
File Cre	ate Concept Tools Units View Help
1 🖍 🖈	New Plane 💿 🗍 Selev
	Extrude 🛛 🖌 📈
XYPI 6	
Т 🌭	Sweep Ifer 👘 S
Blade 🌢	Skin/Loft GD 式
	Thin/Surface
	Fixed Radius Blend
Tree O	Variable Radius Blend
· · · · / · · ·	Vertex Blend
•	Chamfer
1	Pattern
*	Body Operation
	Body Transformation
G	Boolean
4	Slice
	Delete •
٠	Point
	Primitives •
Details View	φ.
Details of Boolean1	
Boolean	Boolean1
Operation	Subtract
Target Bodies	1 Body
Tool Bodies	1 Body
Preserve Tool Bodies?	No
1	

4.12. **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.



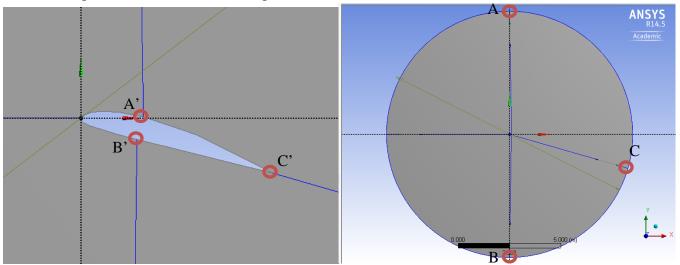
- 4.13. Repeat the process from 4.11 on the two semicircles. This should yield four circular quadrants.
- 4.14. Repeat this process for the arc in quadrant IV (lower right). Change the Fraction to 0.822222. This splits the arc into a 16° and a 74° arc.

4.15. 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 front part of the airfoil is not exactly on the origin, you need to make sure to select the point on the airfoil and not 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).

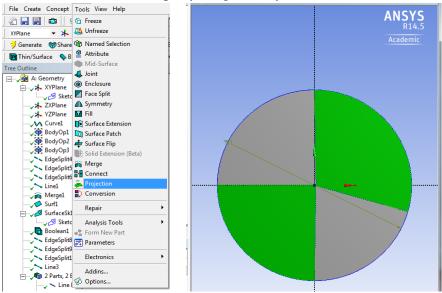


4.16. Once you select both points click **Apply**, then click **Generate**.

4.17. Repeat this process creating lines from the edge of the circle to the airfoil starting from the circle and ending at the airfoil (A to A', B to B' and C to C'). The images below show the locations of the points on the airfoil and the points on the circle.



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



- 4.19. Tools > Merge. Change the Merge Type to Edges. Select the 16° arc and the arc in quadrant I and select Apply. Click Generate. This merges the lines into one line which can be sized for meshing easier.
- 4.20. **File** > **Save project** and exit.

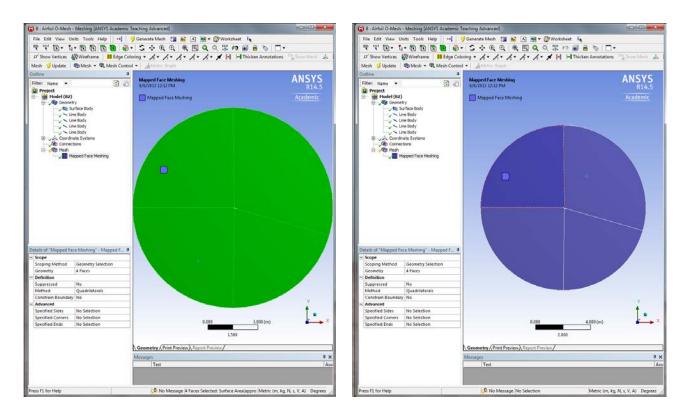
5. Mesh Generation

5.1. From the Project Schematic right click on **Mesh** and select **Edit...** from the dropdown menu.

A Geometry		B sh	C 1 E Fluent
2 颐 Geometry 🗸 🗕 2	🎯 Mes	6	Edit
CFD Pre-Lab 2 Turbulent Flow	Airfoil O	-	n ? _
			Duplicate
			Transfer Data To New
		7	Update
			Clear Generated Data
		2	Refresh
			Reset
		ab	Rename n 😨
			Properties
			Quick Help
			Add Note

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

B : Airfoil O-Mesh - Meshing [ANSYS Academic Teaching Advanced]				
] File Edit View Units Tools Help] ↔	🗄 🕴 🏓 Generate Mesh 🛛 🏙 👪 🙆			
	୫+ 5.⊕.@.@.Q			
📙 🕂 Show Vertices 🛱 Wireframe 🛛 🛱 Sho	w Mesh 🛛 🙏 🔡 Random Colors 🛛 🐼 Ann			
Edge Coloring ▼ 1/0 ▼ 1/1 ▼ 1/2 ▼ 1/2	🖇 🔹 🗶 🛪 🗱 🙌 🛛 Heil Thicken Annotation			
Mesh 💈 Update 🛛 🎯 Mesh 🔻 🔍 Mesh C	Control ▾ 🍘 Mesh Edit ▾ 🗐 📶 Metric Grap			
Outline 🕂				
Filter: Name 🔻				
Project Genetry Geometry				
Surface Body				
Line Body				
Line Body				
Line Body				
Coordinate Systems				
Connections				
Insert	🕑 🕼 Method			
🧚 Update	🔍 Sizing			
🔰 Generate Mesh	🙀 Contact Sizing			
	📥 Refinement			
Preview	Face Meshing			
Show	Match Control			
Create Pinch Controls	Pinch			
2 Clear Generated Data	A Inflation			
allo Rename (F2)	🙍 Mesh Connection Group			
Start Recording	Manual Mesh Connection			
	Node Merge Group			
	Prode Merge			
Details of "Mesh"	Node Move			
·				

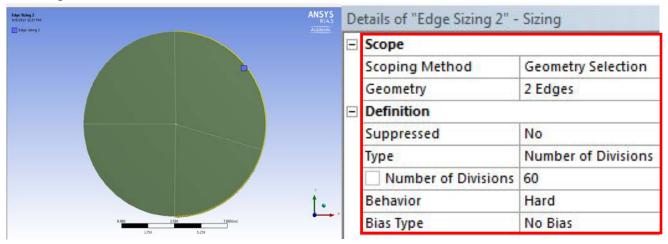


5.3. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below. You might need to change the cursor to "Edge selector" at this moment.

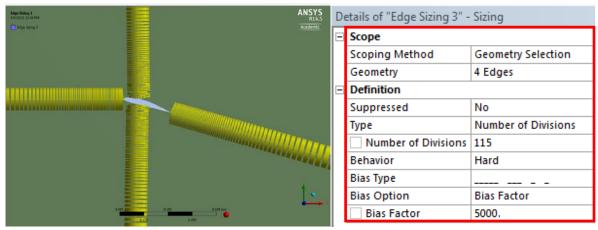
Outline		4
Filter: Name 🔻	· 🚺 🧔	2 E
⊡,∦ Coo	metry Line Body Surface Body rdinate Systems Global Coordinate System mections	
⊡		🕨 🍘 Method
	誟 Update	🔍 Sizing
	芽 Update ኝ Generate Mesh	 Sizing X Contact Sizing ▲ Refinement
		🖗 Contact Sizing

MNSYS IV/20 029 PM try/20 029 PM try/storage R14.5 R14.5 R14.5 R14.5	Details of "Edge Sizing" - Sizing		
2	🖃 Scope		
		Scoping Method	Geometry Selection
		Geometry	2 Edges
	Ξ	Definition	
		Suppressed	No
		Туре	Number of Divisions
		Number of Divisions	45
t.		Behavior	Hard
6.888 1.558 7.589 pe) 1.759 5.250		Bias Type	No Bias

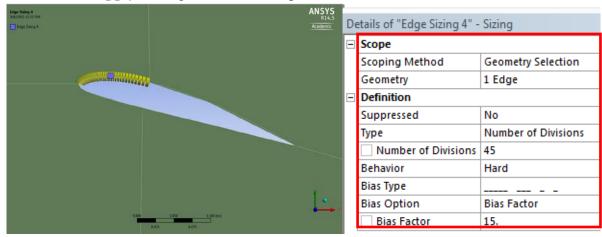
5.4. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.



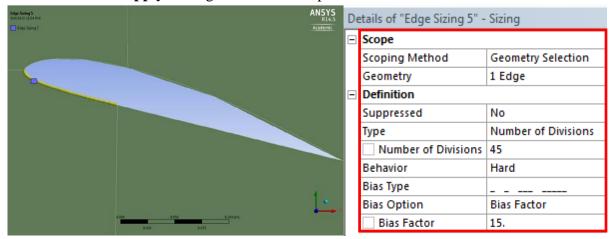
5.5. Right click **Mesh** and **Insert** > **Sizing**. Select all for lines leading from the circle to the airfoil surface, and click **Apply**. Change parameters as per below. Note: If you did not create the lines starting from the outer circle and ending on the airfoil surface, you may have issues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



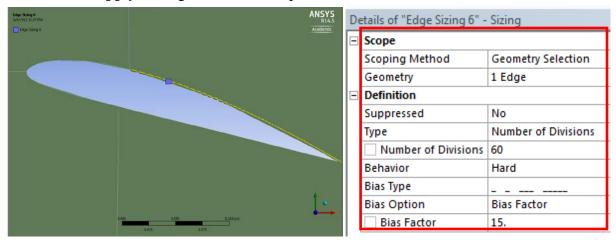
5.6. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the top of leading edge of the airfoil and click **Apply**. Change Parameters as per below.



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.



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.



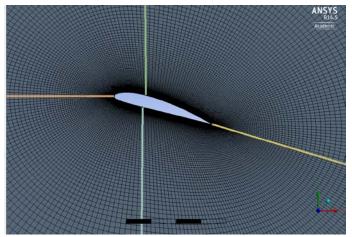
5.9. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the bottom of trailing edge of the airfoil and click **Apply**. Change Parameters as per below.

Tagestions 7 ENCIDED ALL IN A	De	tails of "Edge Sizing 7" -	Sizing
Lisp Gray 7 Azademic	Ξ	Scope	
		Scoping Method	Geometry Selection
		Geometry	1 Edge
		Definition	
		Suppressed	No
		Туре	Number of Divisions
		Number of Divisions	60
		Behavior	Hard
		Bias Type	
<u>1</u>		Bias Option	Bias Factor
100 10 ⁰ 10 ¹		Bias Factor	15.

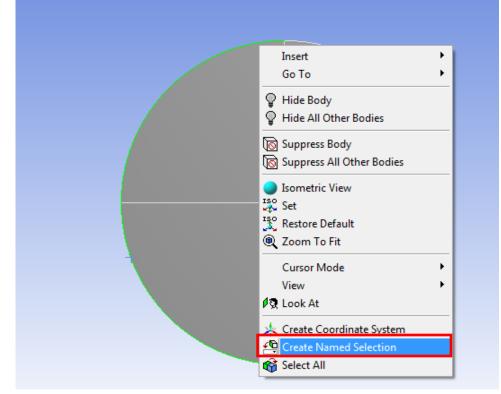
5.10. Click on Mesh under the Outline, under the Details of "Mesh", change the PhysicsPreference from Mechanical to CFD. This changes the grid solver to a fluids style solver rather than a FEA style solver.

D	etails of "Mesh"	д				
	Defaults					
	Physics Preference	CFD 🗸				
	Solver Preference	Mechanical Electromagnetics				
÷	Sizing	CFD Explicit				
÷	Inflation					
	Assembly Meshing					
	Method	None				
	Patch Conforming Options					
	Triangle Surface Mesher Program Controlled					
Ŧ						
Ð						
÷	E Statistics					

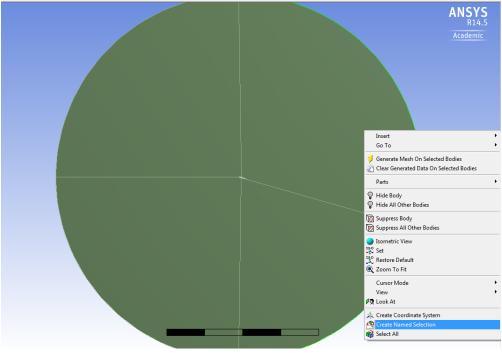
5.11. Click **Generate Mesh**. Click on the **Mesh** button under the **Outline** and make sure it resembles the mesh below.



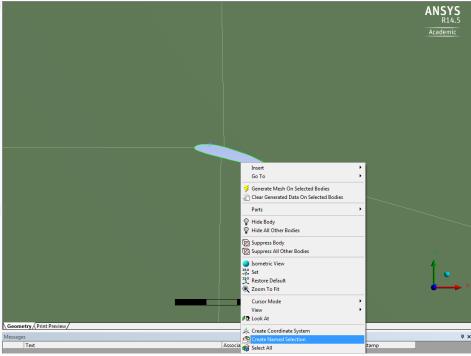
5.12. Hold Ctrl and select the two left most semicircle arcs, right click on them and select **Create Named Selection**, name the selection *inlet*. Use the edge select button from the toolbar.



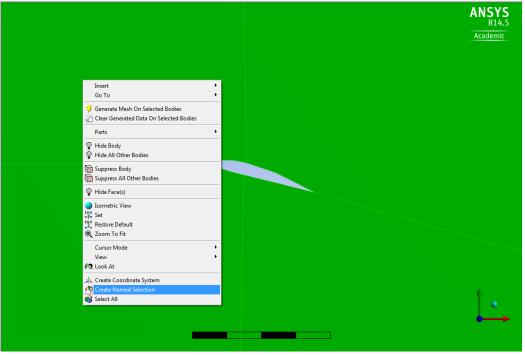
5.13. Do the same for the two right most semicircle arcs and name them *outlet*.



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



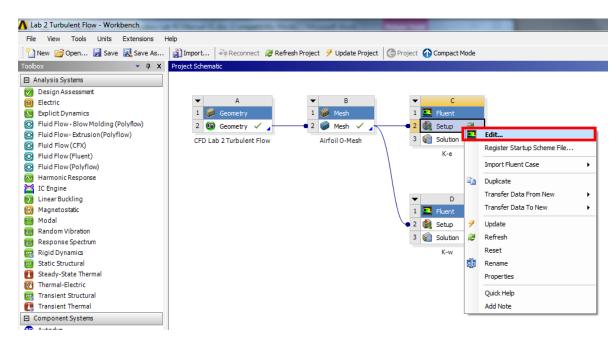
5.15. Now use the face button to select the four semicircle quadrants and **Create Named Selection** and name them *fluid*.



- 5.16. **File > Save Project.** Exit the window.
- 5.17. Right click **Mesh** and select **Update** from the dropdown menu.

6. Setup (Physics)

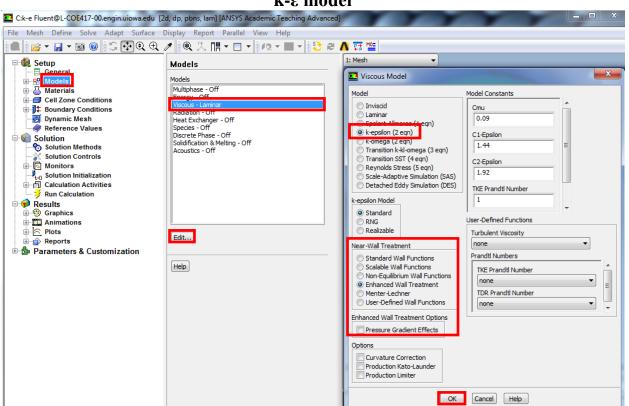
6.1. From the Project Schematic right click on Setup and select Edit...



6.2. Select Double Precision and click OK.

Fluent Launcher (Setting Edit Only)	
ANSYS"	Fluent Launcher
Dimension	Options Double Precision Processing Options Serial Parallel
	ancel <u>H</u> elp 🔻

6.3. Solution Setup > Models > Viscous –Laminar > Edit... Change the parameters as per below and click OK. (For the k-ω case, you will select k-omega (2 eqn).



k-ω model

C:k-e Fluent@L-COE417-00.engin.uiowa.edu [2	2d, dp, pbns, Iam] [ANSYS Academic Teaching Advance	ed]
File Mesh Define Solve Adapt Surface		
🚛 🛙 🚘 र 🚂 र 🞯 🥝 🗄 🤁 🕀 ।	🥒 🔍 洗 唱 - 🗆 - 🕴 🕸 🖉 🖉	* A 韓 些
🖃 🍓 Setup	Models	1: Mesh 🔹
General ⊕ - <mark>P:</mark> Models	Models	Viscous Model
	Multiphase - Off	Model Model Constants
Cell Zone Conditions Cell Zone Conditions Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Solution Controls Solution Initialization Solution Initialization Calculation Activities Run Calculation Run Calculation See Sults Solution Solution Page Soluti	Viscois - Laminar Viscois - Laminar Readation - Off Species - Off Species - Off Solidification & Melting - Off Acoustics - Off Edit	Invisid Laminar Spalart-Almaras (1 eqn) Spalart-Almaras (1 eqn) Spalart-Almaras (1 eqn) Fransition K31 (4 eqn) Transition K31 (4 eqn) Spalart-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) K-omega Model Standard BSL SST Low-Re Corrections V Shear Flow Corrections Options TrE Prandtl Numbers Tree Prandtl Number Inone SDR Prandtl Number
		Production Limiter

k-ε model

6.4. Solution Setup > Materials > air > Create/Edit... Change parameters as per experimental data and click Change/Create.

Ck-e Fluent@L-COE417-00.engin.uiowa.edu [2d, dp, pbns, lam] [ANSYS Academic Teaching Advanced]						
Ele Mesh Define Solve Adapt Surface Display Report Parallel View Help						
	Materials Materials	1: Mesh Create/Edit Materials				
B-2 Materials C-B-Zone Conditions B-3 C-B-Zone Conditions B-3 C-B-Z	air Sold sold aluminum	Name air Chemical Formula	Material Type fluid Fluent Fluid Materials air	Order Materials by Image: State of the state of		
Solution Solution Methods Solution Controls Solution Initialization		Properties	Mixture none	User-Defined Database		
B - ∱ Calculation Activities → Run Calculation ⊕ P Results ⊕ Graphics ⊕ I Animations		Density (kg/m3) constant I.1885 Viscosity (kg/m-s)	Edit			
⊕- ि Piots ⊕- ŵ Reports ⊕- ∰ Parameters & Customization		1.8396e-05				
	Create/Edit Delete		Ţ			
		, Change/Creat	e Delete Close Help			

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 (blue box) and default values (red boxes) and click **OK**. The experimental value can be found from the EFD Lab 3 data reduction sheet.

K-E model						
	rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> iew <u>H</u> elp					
	Q 🕀 🥒 🔍 🏷 🖪 🕶 🖬 🖉 🖉 🖉 🕄 🍋	a V 🖆 📠				
🖃 🍓 Setup	Boundary Conditions	Velocity Inlet	X			
General Genera	Zone sirfol Interior-nuic outlet surface_body	Zone Name inlet Momentum Thermal Radiation Species DPM Multiphase UDS Velocity Specification Method Components Reference Frame Absolute Supersonic/Initial Gauge Pressure (pascal) 0 constant X-Velocity (m/s) 15 constant				
	Phase Type ID mixture velocity-inlet 6 Edit Copy Profiles Parameters Operating Conditions Display Mesh Periodic Conditions	V-Velocity (m/s) 0 constant Turbulence Specification Method Turbulent Kinetic Energy (m2/s2) 0.08 constant Turbulent Dissipation Rate (m2/s3) 7.4 constant OK Cancel Help	•			

k-ω model

Cik-e Fluent@L-COE417-00.engin.uiowa.edu [2d, dp, pbns, skw] [ANSYS Academic Teaching Advanced]							
Eile <u>M</u> esh D <u>e</u> fine <u>S</u> olve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> iew <u>H</u> elp							
■∥☞▼ 🖬 ▼ 📾 ❷∥公 🔁 ❷ ⊕ 🥕 🔍 弐 開▼ 🗆 ▼ 🕸 ▼ 🔳 ▼ 💱 🤻 🗛 इ ≌							
■ Setup General	Boundary Conditions	Velocity Inlet					
Models	Zone	Zone Name					
	inlet	inlet					
Cell Zone Collutions Boundary Conditions Dynamic Mesh	interior-nuid outlet surface body	Momentum Thermal Radiation Species DPM Multiphase UDS					
Reference Values		Velocity Specification Method Components					
Solution		Reference Frame Absolute					
Solution Controls		Supersonic/Initial Gauge Pressure (pascal) 0 constant					
■ Solution Initialization 		X-Velocity (m/s) 15 constant					
₩ 🤣 Run Calculation		Y-Velocity (m/s) 0 constant					
Graphics		Turbulence					
		Specification Method K and Omega					
eners ⊕ • • • • • • • • • • • • • • • • • • •	Phase Type ID mixture velocity-inlet velocity-inlet 6	Turbulent Kinetic Energy (m2/s2)					
🗄 🎰 Parameters & Customizat	mixture velocity-inlet -						
	Edit Copy Profiles	Specific Dissipation Rate (1/s) 9 constant					
	Parameters Operating Conditions						
	Display Mesh Periodic Conditions	OK Cancel Help					

6.6. Solution Setup > Boundary Conditions > Outlet > Edit... Change the parameters as per below and click OK. (Use outlet B.C. for both k-ε and k-ω models)

🖳 C:k-e Fluent@L-COE417-00.engin.uiowa.edu [2d, dp, pbns, ske] [ANSYS Academic Teaching Advanced]					
<u>File M</u> esh D <u>e</u> fine <u>S</u> olve <u>A</u> dapt S <u>u</u>	Irface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> iew <u>H</u> elp				
i 🔳 i 📂 🗸 🖌 🖬 🐨 🎯 i 🕾 🔂	३ 🕀 🥒 🔍 🔍 🔚 र 🗖 र 🛛 🖉 🖗 🖉				
🖃 🍓 Setup	Boundary Conditions	1: Mesh			
General	Zone	Pressure Outlet			
🗈 🐻 Materials	airfoil	Zone Name			
Cell Zone Conditions	inlet interior fluid	outlet			
Boundary Conditions Dynamic Mesh	outlet				
Reference Values	Surface_body	Momentum Thermal Radiation Species DPM Multiphase UDS			
Solution		Gauge Pressure (pascal)			
Solution Methods		Parkfare Direction Constitution Marked			
		Backflow Direction Specification Method Normal to Boundary			
Solution Initialization					
		Target Mass Flow Rate			
Results					
Graphics		Specification Method Intensity and Viscosity Ratio			
		Backflow Turbulent Intensity (%) 3,25			
Reports	Phase Type ID	Backflow Turbulent Viscosity Ratio 0.0035			
Parameters & Customizatic	mixture v pressure-outlet v 7				
	Edit Copy Profiles				
	Parameters Operating Conditions	OK Cancel Help			
1					

6.7. Solution Setup > Reference Values. Change parameters as per below. The Velocity, Temperature, Density, and Viscosity should be entered from EFD data.

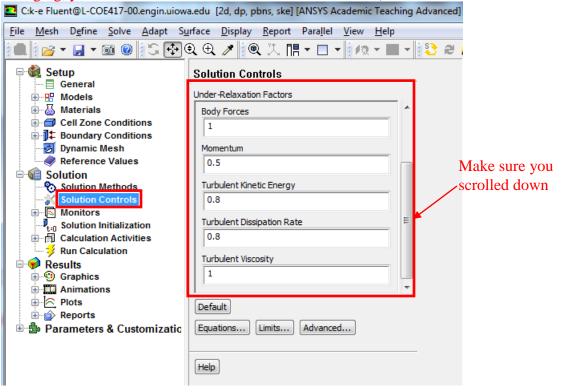
C:k-e Fluent@L-COE417-00.engin.uiow	a.edu [2d, dp, pbns, ske] [ANSYS Academic Teaching Adv	anced]		
<u>File Mesh Define Solve A</u> dapt Su	Irface <u>D</u> isplay <u>R</u> eport	t Para <u>l</u> lel <u>V</u> iew <u>H</u> elp			
i 💼 i 📂 🕶 🖬 🔻 🗃 🞯 👘 🤆	2 Q 🗡 🔍 🏌 🛛	18 • 🔳 • 🚺 🕫 • 🔳 • 👫) æ		
E Setup	Reference Values				
General	Compute from	Compute from			
		•			
Cell Zone Conditions	Reference Values				
Boundary Conditions	Area (m2)	0.3048			
Dynamic Mesh Reference Values		0.3048			
Solution	Density (kg/m3)	1.1885			
Solution Methods	Death (m)				
Solution Controls	Depth (m)	1			
Monitors ↓ Solution Initialization	Enthalpy (j/kg)	0			
Run Calculation	Length (m)	1			
E 🗭 Results	Pressure (pascal)	0			
Graphics Graphics Animations	Tressere (pascal)	0			
er for the second seco	Temperature (k)	297.15			
	Vala site (as (a)				
🗄 🎰 Parameters & Customizatic	Velocity (m/s)	15			
	Viscosity (kg/m-s)	1.8396e-05			
	Ratio of Specific Heats	1.4			
	Reference Zone				
		•			
	Help				

7. Solution

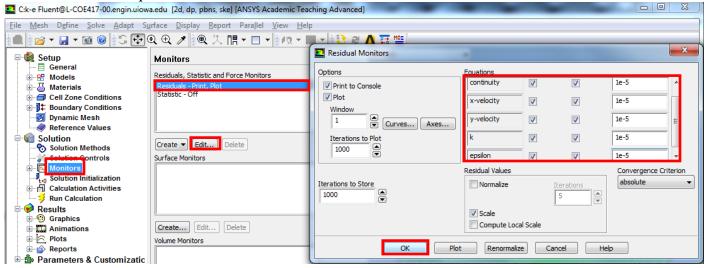
7.1. Solution > Solution Methods. Change parameters as per below.

	k-e	k-	Ø
Ck-e Fluent@L-COE417-00.engin.uiowa File Mesh Define Solve Adapt Sur Setup General Hereinals H	K-E ra.edu [2d, dp, pbns, ske] [ANSYS Academic Teaching A urface Display Beport Parallel View Help C P Parallel View Help Solution Methods Pressure-Velocity Coupling Scheme Submute Solution Methods Pressure-Velocity Coupling Scheme Submute Scheme Submute Standard Versure Standard Versure Standard	D:K-w Fluent@L-COE095-02.enginuiowa.edu [2/ File Mesh Define Solve Adapt Surface Di Image: Solve Adapt Surface Di Di	isplay Report Parallel View Help
 B Monitors P₁₀ Solution Initialization Calculation Activities ✓ Run Calculation ✓ Results ⊕ Graphics ⊕ Taimations ⊕ Piots ⊕ Parameters & Customizatic 			Standard • Momentum Second Order Upwind Second Order Upwind • Second Order Upwind • Second Order Upwind • Second Order Upwind • Tarsient Disspation Rate • Second Order Upwind • Transient Pormulation • Prozen Flux Formulation • Prozen Flux Formulation • Bedult •

7.2. Solution > Solution Controls. Change Parameters as per below. (If you have problems with the solution converging, you can decrease the Under –Relation Factors.)



7.3. Solution > Monitors > Residuals – Print, Plot > Edit.... Change the convergence limit to 1e-05 for all five equations.



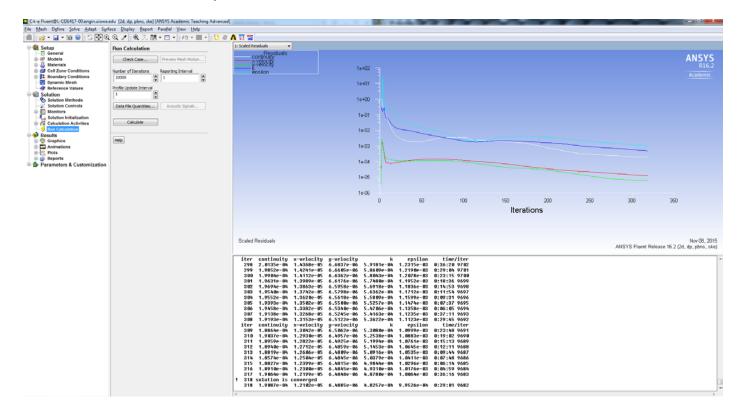
7.4. Solution > Solution Initialization. Change the parameters as per below and click Initialize.

k-ε model		k-ω model		
File Mesh Dgfine Solve Adapt Su Image: Setup Image: Setup <t< th=""><th>a.edu [2d, dp, pbns, ske] [ANSYS Academic Teaching Advanced] ufface Display Report Parallel View Help Q Q A A Q A R A A A A A A A A A A A A A</th><th>Ck-e Fluent@L-COE417-00.engin.uic File Mesh Dgfine Solve Adapt Methods Setup General B B Models Cell Zone Conditions D Cell Zone Conditions Cell Zone Conditions Solution Solution</th><th>Solution Initialization Initialization Standard Initialization Compute from Reference Frame</th></t<>	a.edu [2d, dp, pbns, ske] [ANSYS Academic Teaching Advanced] ufface Display Report Parallel View Help Q Q A A Q A R A A A A A A A A A A A A A	Ck-e Fluent@L-COE417-00.engin.uic File Mesh Dgfine Solve Adapt Methods Setup General B B Models Cell Zone Conditions D Cell Zone Conditions Cell Zone Conditions Solution Solution	Solution Initialization Initialization Standard Initialization Compute from Reference Frame	
Solution Methods Solution Controls Solution Controls Solution Initialization Calculation Activities Run Calculation Results Solution Solution Plots Plots Parameters & Customizatic	Reset DPM Sources	Solution Methods Solution Controls Solution Initialization Image: Calculation Activities <	Relative to Cell Zone Reset DPM Sources Initial Values Gauge Pressure (pascal) 0 X Velocity (m/s) 15 Y Velocity (m/s) 0 Turbulent Kinetic Energy (m2/s2) 0.08 Specific Dissipation Rate (1/s) 9 Initialize Reset Patch	

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

C:k-e Fident@E-COE417-00.engin.diowa	a.edu (zu, up, pons, skej (ANSTS Academic Teachin
<u>File Mesh Define Solve Adapt Su</u>	rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> iew <u>H</u> elp
💼 🛙 📂 🖌 🖬 🐨 🞯 👘 🔂	२ ⊕ ↗ 🛛 @ ≒ 🖪 ▾ 🗖 ▾ 📕 ¹
Setup General General Models Materials	Run Calculation Check Case Preview Mesh Motion
Cell Zone Conditions Conditions Dynamic Mesh Reference Values	Number of Iterations 10000 Profile Update Interval
Solution Solution Methods Solution Controls Monitors	Image: Second
 ↓ Solution Initialization ⊕ ⊕ ⊕ Calculation Activities ✓ Run Calculation ♥ Results 	Calculate
 Graphics ∰ Graphics ∰ ∰ Animations ∰ ∯ Plots ∰ ∯ Reports ∯ Parameters & Customizatic 	Help

Iteration history should look similar to the one below.



7.6. File > Save Project.

8. Results

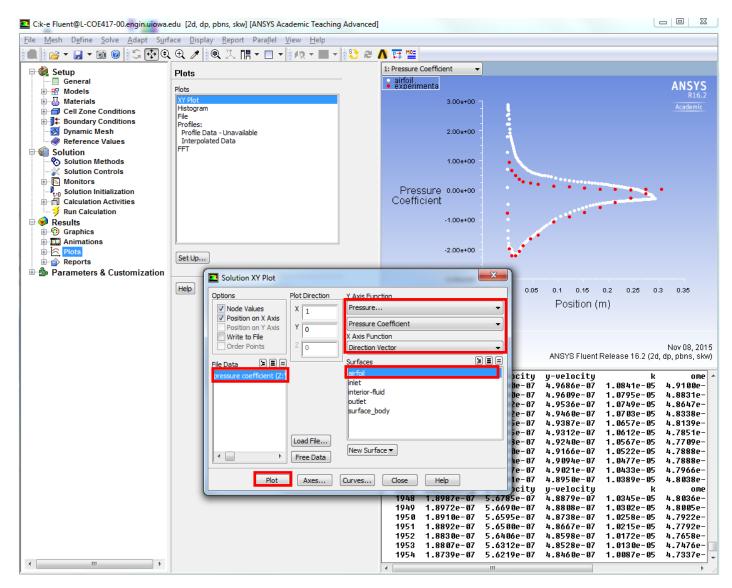
8.1. Plotting and Saving Residual History Solution > Monitors > Residuals –Print, Plot > Edit... > Plot > Ok.

C:k-e Fluent@L-COE417-00.engin.uiowa	edu [2d,	dp, pbns, skw] [ANSYS Academic Teaching A	ivanced]	1.0			
Eile Mesh Dgfine Solve Adapt Syrface Display Report Parallel View Help							
i 💼 i 📸 🔻 🔙 🕶 🚳 🔞 i 🛱 🔂	(🕀 🧪	@ 洗 開 - □ - /Ջ - ■ -	ີ ສ	∧ 🖬 🔤			
⊫-∰ Setup	Monito	prs		1: Scaled Residual			
- General	-	s. Statistic and Force Monitors		continuity x-velocity y-velocity	µals		
		als - Print, Plot					
Cell Zone Conditions	Stausu			òmeαa		1e+00	
						1e-01 -	
- 🛃 Dynamic Mesh						1e-01	
Reference Values						1e-02 -	
Solution Methods	Create	Edit Delete				1e-02	\mathbf{N}
-X Solution Controls	Surface	Monitors				1e-03	
I - 🔁 Monitors						1e-U3	
						1e-04	
						16:04	
Results						1e-05	
Graphics	Create	Edit Delete				18.00	
Plots	Volume 1					1e-06	
Reports						reioo	
Barameters & Customization						1e-07	
						0	0 200 400 500 800 1000 1200 1400 1500 1800 2000
		Residual Monitors					Iterations
						_	
	Create	Options	Equation				
	Converg	I Fint to console	continu	uity 🔽	\checkmark	1e-05	
		V Plot	x-velo	city 🗸	V	1e-05	ANSYS Fluent Release 16.2 (2d
		Window	y-veloo	city 🗸	V	1e-05 =	
		1 Curves Axes	y-veloc	uty V	v	16-05	y k omega time/iter 7 1.0841e-05 4.9100e-06 0:05:12 8063
		Iterations to Plot	k		V	1e-05	7 1.0795e-05 4.8831e-06 0:31:02 8062
	Conver	1000	omega			1e-05 -	7 1.0749e-05 4.8647e-06 0:24:50 8061 7 1.0703e-05 4.8338e-06 0:19:52 8060
	-		Residual	Values		Convergence Criterio	
	Help	Iterations to Store	Nor	malize	Iterations	absolute	▼ 7 1.0612e-05 4.7851e-06 0:12:42 8058
		1000	-		5		7 1.0567e-05 4.7709e-06 0:10:10 8057 7 1.0522e-05 4.7888e-06 0:08:08 8056
			✓ Scale	le			7 1.0477e-05 4.7888e-06 0:06:30 8055
				npute Local Scale			7 1.0433e-05 4.7966e-06 0:05:12 8054 7 1.0389e-05 4.8038e-06 0:31:00 8053
							y k omega time/iter
		OK Plot	F	Renormalize C	ancel H	elp	7 1.0345e-05 4.8036e-06 0:24:48 8052
I					_		7 1.0302e-05 4.8005e-06 0:19:50 8051

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

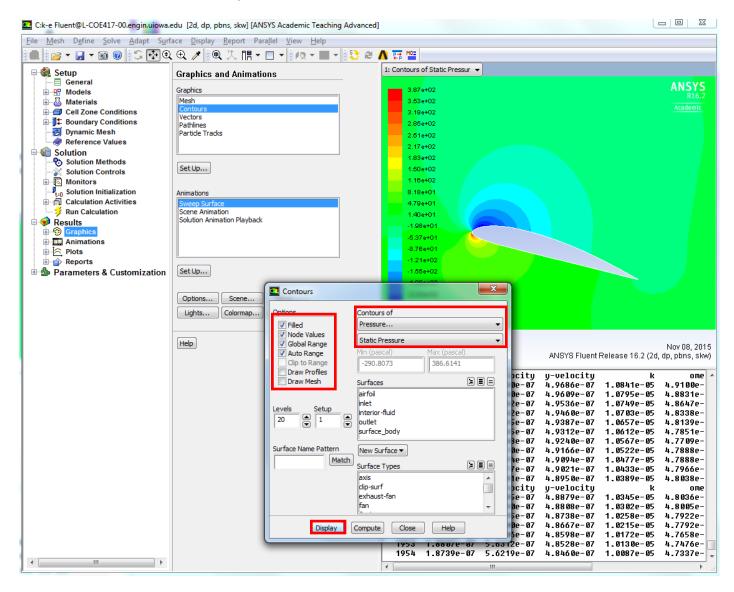
Save Picture				×
Format	Coloring	File Type	Resolution	
© EPS JPEG	 Color Gray Scale 	 Raster Vector 	Width 960	
 PPM PostScript 	Monochrome		Height 720	
TIFF PNG	Options	LUI:		
© VRML O Window Dump	 Landscape Orie White Backgrou 	mauon	dow Dump Command oort -window %w	
Save Apply Preview Close Help				

8.2. Plotting Pressure Coefficient Distribution with CFD and EFD Data
Results > Plots > XY Plot > Set Up... > Load File... Select Pressure-coef-attack16.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 Lab 2 Pressure Coefficient Distribution*.



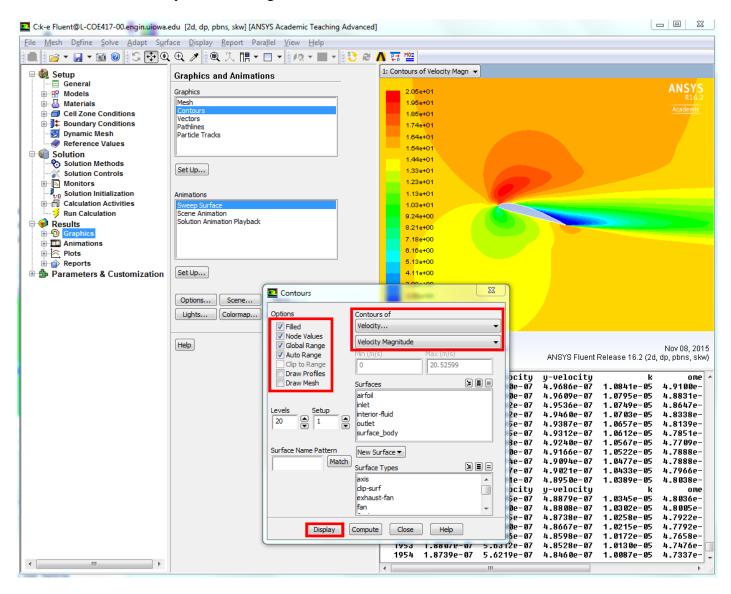
8.3. Plotting Contour of Pressure

Results > Graphics > 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 Lab 2 Contour of Pressure*.



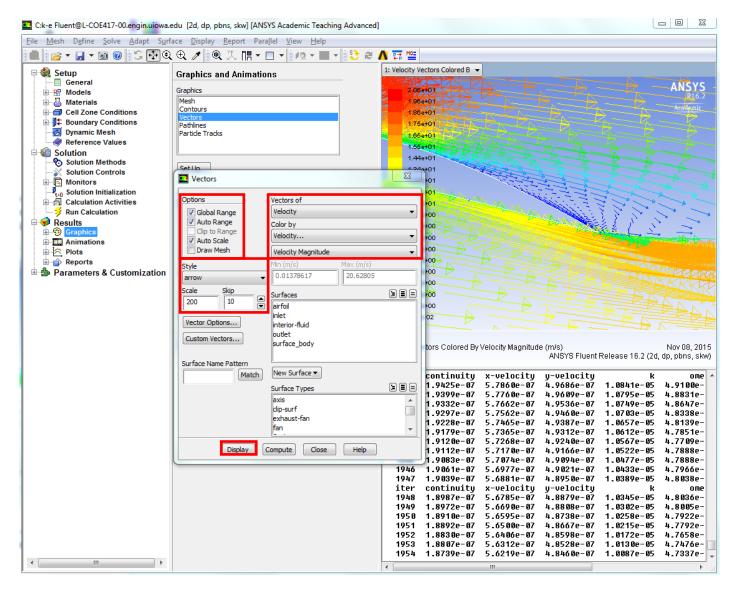
8.4. Plotting Contour of Velocity Magnitude

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 Lab 2 Contour of Velocity Magnitude*. Zoom in where you can see the airfoil clearly and the change in contour levels around the airfoil.



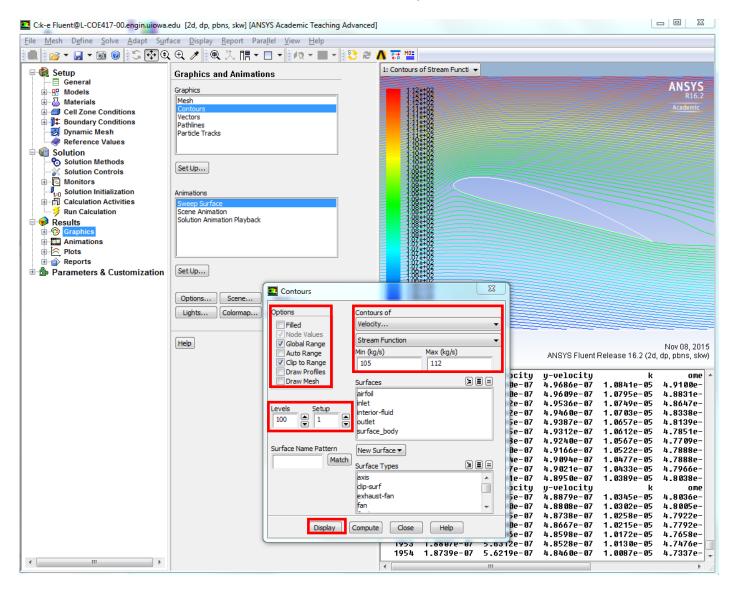
8.5. Plotting Velocity Vectors at Trailing Edge

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 Lab 2 Vectors of Velocity at Trailing Edge*. Zoom in on the trailing edge.



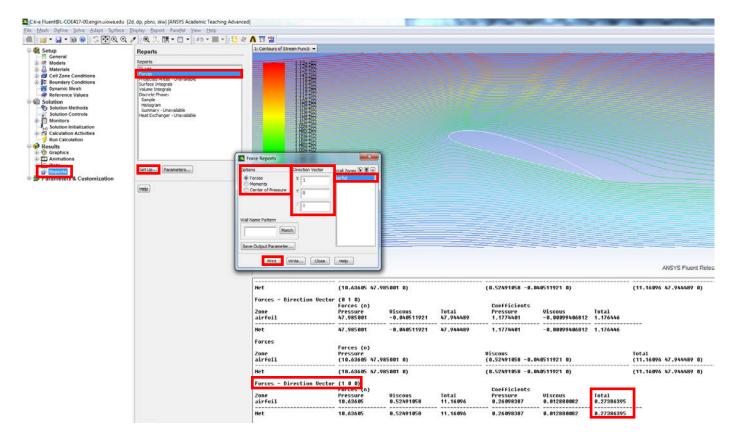
8.6. Plotting Streamlines Close to Airfoil Surface

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 Lab 2 Streamlines Close to Surface*.



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 zero and the Y parameter to 1, this prints out the lift force. Save the coefficients by clicking **Write**. This creates a text file of what was printed on the screen. Name the file *Drag Coefficient* or *Lift Coefficient*.



9. Exercises

Parametric Studies 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.
- Use "CFD Lab2 Report Template.doc" to save the figures and data for each exercise below.
- Use the benchmark EFD data from the class website if your EFD Lab 3 data are with a different angle of attack than 16 degrees.

1. Effect of angle of attack.

1.1. Use the same flow conditions as those in your EFD Lab 3, including geometry (chord length) and setup (Flow properties, Reynolds number, inlet velocity), EXCEPT use angle of attack 16
 Degrees regardless of AOA in you EFD Lab 3. Use k-ε model, 2nd order upwind scheme, double precision with iteration number (2000) and convergent limit (10⁻⁵).

• **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.Velocity vectors, and 5. Streamlines

• Data need to be saved: lift and drag coefficients

2. Effect of turbulence models

2.1. Use the same conditions as those in exercise 1, **EXCEPT** using the "k- ω " for "viscous models". Set up the boundary conditions following instructions part, set the iteration number to be (2000), and convergent limit to be 10⁻⁵. Perform the simulation and compare solutions with the simulation results using "k- ε " model (you have finished in CFD PreLab2)

- 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. Velocity vectors, and 5. Streamlines
- Data need to be saved: lift and drag coefficients

3. Questions need to be answered in CFD Lab2 report:

Using the figures obtained in exercises 1 and 2 in this Lab and those figures you created in CFD PreLab2 to answer the following questions and present your answers in your CFD Lab 2 report.

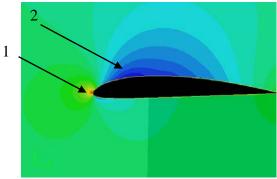
Note: These questions are also available in the "CFD Lab2 Report Template" where you will type answers.

3.1. For Exercise 1 (effect of angle of attack):

- (1). Which angle of attack simulation requires more iterations to converge?
- (2). Which angle of attack produces higher lift/drag coefficients? Why?
- (3). What is the effect of angle of attack on lift and drag coefficients?
- (4). Describe the differences of streamline distributions near the trailing edge of airfoil surface for these two different angles of attack. Do you observe separations for both? If so, does the separation occur at the same location?

3.2. For Exercise 2 (different turbulence models):

- (1). Do the two different turbulence models have the same convergence path? If not, which one requires more iterations to converge.
- (2). Do the two different turbulence models predict the same results? If not, which model predicts more accurately by comparing with EFD data?
- **3.3.** For the following contour plot, qualitatively compare the values of pressure and velocity magnitude at point 1 and 2, if the flow is from left to right. Which location has higher pressure and which location has higher velocity magnitude? Why?



3.4. Questions in CFD PreLab2.