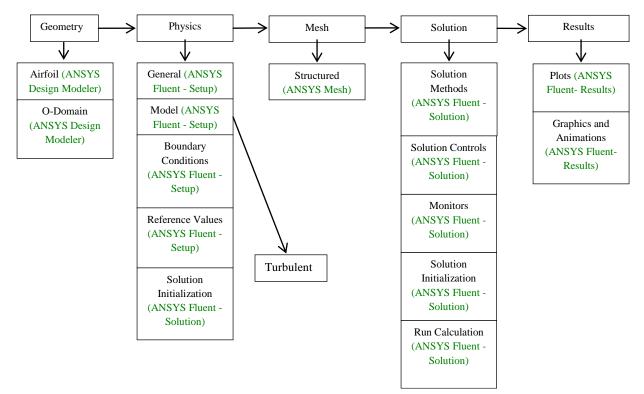
Simulation of Turbulent Flow around an Airfoil

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	

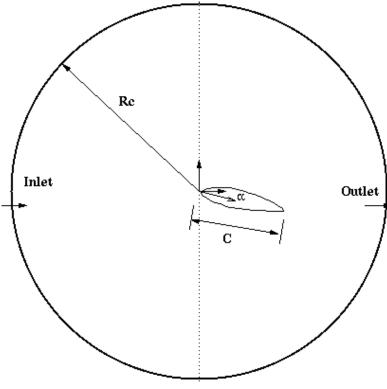
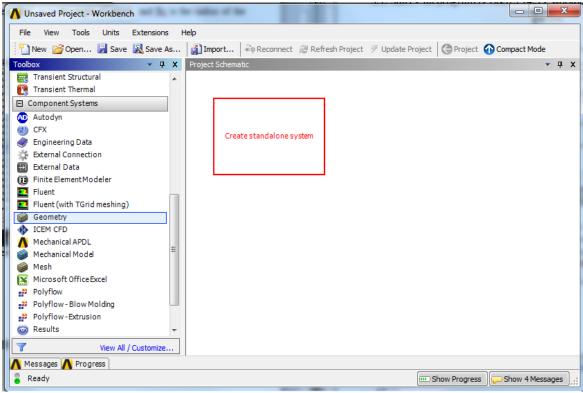


Figure 1 – Geometry

3. Open ANSYS Workbench

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

- 🚰 Windows Update 📜 WinRAR 🛤 XPS Viewe Abaqus 6.12-1 Accessories ActivePerl 5.16.3 Build 1603 (64-bit) ANSYS 14.5 MANSYS Icepak 14.5 Mechanical APDL 14.5 \Lambda Mechanical APDL Product Launcher \Lambda Uninstall 14.5 A Workbench 14.5 ANSYS Client Licensing Agwa 📗 EKM Fluid Dynamics Help Meshing Remote Solve Manager Utilities Back Search programs and files Q
- 3.2. From the ANSYS Workbench home screen (**Project Schematic**), drag and drop a **Geometry, Mesh**, and a **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.

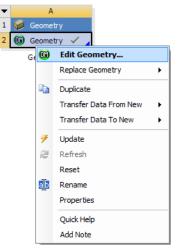


Vunsaved Project - Workbench		x
	Heb	_
New Copen Save Save As		
Toolbox 🔻 🕂 🗙	Project Schematic	џ ;
Magnetostatic		
Modal		
Random Vibration	▼ A ▼ B ▼ C	
Response Spectrum	1 🥥 Geometry 1 💭 Mesh 1 🔽 Fluent	
Rigid Dynamics Static Structural	2 🕼 Geometry 🗸 🚽 4 2 💓 Mesh 🥔 🚽 🔶 2 🎉 Setup 🍞 🖌	
	CFD Lab 2 Turbulent Flow Airfoil O-Mesh 3 🍿 Solution 💡 🖌	
Steady-State Thermal Thermal-Electric	K-e	
Transient Structural Transient Thermal		
Component Systems		
	▼ D	
	1 💶 Fluent	
CFX Engineering Data	🗣 2 🌒 Setup 💡	
Engineering Data External Connection	3 🙀 Solution 💡	
External Data		
Finite Element Modeler	K-w	
Fluent		
Fluent (with TGrid meshing)		
Geometry		
ICEM CFD		
Mechanical APDL		
Mechanical Model		
🍯 Mesh		
Microsoft Office Excel		
Polyflow		
Polyflow - Blow Molding		
Polyflow - Extrusion		
💿 Results 💌		
View All / Customize		
Messages		
Ready		es
• •		

- 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*.

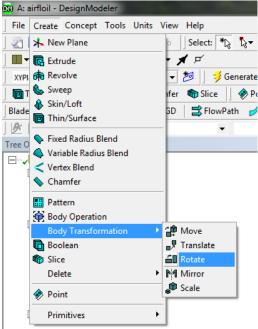
4. Geometry

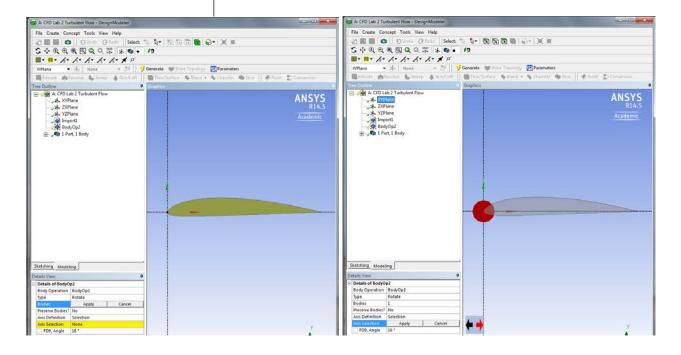
- 4.1. Right click **Geometry** then select **Import Geometry** > **Browse...** Select **airfoil.igs** and click **OK**.
- 4.2. Right click Geometry and select Edit Geometry...



4.3. Click Generate.

4.4. 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.



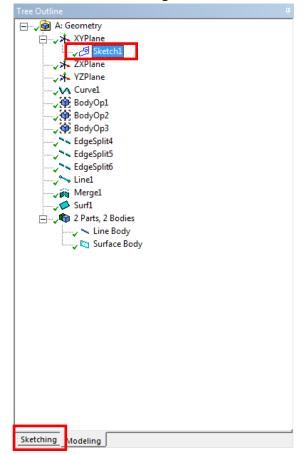


(g) A: CFD Lab 2 Turbulent Flow - DesignModeler	- G X.	600 A: CFD Lab 2 Turbulent Flow - DesignModeler		X
File Create Concept Tools View Help		File Create Concept Tools View Help		
2 - 0 Dundo @Redo Select 12 2- 10 10 10 10	- 1 W W	🖉 🛃 📓 🚳 🛛 Dillindo 📿 Rindo 🛛 Select	14 4 10 10 10 0 0 0 X X	
5 ·	120	5 · @ @ @ @ Q Q % * .	· #2	
h. h. h. h. h. h. #		k- k- k- k- k- # #		
XTPlane ・ 水 None ・ 刮 Generate Withour Topology	Disconstant	XIPlane - 🖈 None - 🏄	Generate 100 Share Topology	
Estrude ARevolve Some Standart Brin/Surface Siltend •			Thin/Surface & Blend • & Chamfer & Slice & Point	
	Countrie Sales Sevent E. Conversion		Graphics	
Tree Outline Graphics Graphics Graphics	*	A: CFD Lab 2 Turbulent Flow		
→ Star Column row → X 20 June → X 20 June → Y 20 June → Y 20 June → Y 20 June → Star Star Star Star Star Star Star Star	ANSYS R14.5 <i>Academic</i>	XXPine XZPine Niport RoyAQ2 RoyAQ2 NorNor		ANSYS R14.5 Academic
Skrtching Maddeling Decide of objects 0 Bedrig of berding 0 Decide of objects 0 Person (Endotic) 0 Preserver Societti No 1 Preserver Societti No 1 Preserver Societti No 1 Aus Definition Sietching		Skrithing Modeling Debils View Chails of BodyDg2 Body Operation (BodyOp2 Type Robet Boddet 1 Preserve Bodiet 7 Aus Definition Selection		
Axis Selection Plane Normal		Axis Selection Plane Normal		v
F09, Angle 16 *	Y	ED9, Angle 16.*		

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

File Create Concept Tools View Help
] 🖉 🔚 📕 📫 🗍 💬 Undo 📿 Redo 🔢 Select: 🌇 🎝 🗸 🖌 🕅
J XYPlane ▼ 🛧 Sketch1 ▼ 💆
🚽 🗦 Generate 🛯 Share Topology 🔀 Parame New Sketch Extrude 🏟
📙 🛅 Thin/Surface 🛛 💊 Blend 🔻 🥎 Chamfer 🏾 🍿 Slice 🔢 🛷 Point 📲
Tree Outline 4
A: Geometry

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



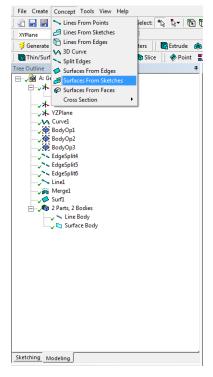
4.7. Sketching Toolbox > Draw > Circle. Click on the xy-plane origin and click behind the airfoil.

File Create Concept Tools View Help Image: I	A: CFD Lab 2 Turbulent Flow - DesignModeler			
Image: International State Internat	File Create Concept Tools View Help			
Image: International State Internat	2 🗐 🗐 👩 🖸 OUndo @ Redo Select: 🏠 🖙 🦻		£ @ Q Q X * 6 • 12	
Whene * Statude * Revolve Statude * Statude * Revolve Statude Totalous * Chamfer Statude * Point Conversion Statude Totalous				
Extrude © Revolve © Skin/Left © Thin/Surface © Blend © Lonersion © Conversion © Stecting Toolboxes © Draw Oraphics © ANSYSS R14.5 Conversion © Steching legistration © Conversion © ANSYSS R14.5 Conversion © Conversion Conversion <td< td=""><td></td><td>Share Topology 😰 Parameters</td><td></td><td></td></td<>		Share Topology 😰 Parameters		
Draw A Vine Stargentiae Constraints Academic Sectangle by 3 Points Academic Oplying Retangle Sectangle by 3 Points Academic Oblight Constraints Secting Modify Dimensions Constraints Settings Setting Stetch Visbility Show Setch Show Constraints? No E deges: Full Circle Full Circle C7			Point Deconversion	
Line Line Chargent Line Chargent	Sketching Toolboxes	Graphics		9
Line R14.5 C Tangents Academic A Polyline Setsings C Rectangle by 3 Points Academic Ø Oxal Owner © Circle Setsings Stetching Modeling Settings Details View P © Details View P E betas of Secthal Settings Stetch Visibility Show Constraints Show Constraints? No B E beges: Fail Circle Full Circle Constraints Model View Print Preview	Draw			ANCVC
G Inte by 2 Tangents A Polyline C Polygon Rectangle by 3 Points G Oval G Cricle Q Cricle by 3 Tangents Constraints Constraints Setching Stetching Stetching <td>Line</td> <td></td> <td></td> <td></td>	Line			
A cademic A Polyline G Polygon Catectangle Catectangle of SP onto Ø Oval C Circle 4 Circle by 3 Tangents Modify Dimensions Constraints Stetching	🖌 Tangent Line			R14.5
Pelogon Rectangle by 3 Points Ø Oval Ø Circle Q Circle by 3 Tangents Otimerations Constraints Settings Stetch				Academic
Rectangle by 3 Points Q Oval Q Oval Q Cricle by 3 Tangents	∧ Polyline			
Retargle by 3 Points © Dval © Circle Circle by 3 Tangents Dimensions Constraints Settings Sketching Sketchi				
20 Oval © Circle y3 Tangents Modify Dimensions Constraints Stetching Stetch				
Sciele				
Circle by 3 Tangents Dimensions Constraints Settings Sketching Modeling Details of Sketchi Sketch		/		
Modify Image: Constraints Constraints Settings Settings Settings Stetching Settinits Stetching Settinits Stetching Settinits Stetching Show Sketchin Show Constraints? No Settinits? Edges: Pull Circle Full Circle Cr7 Model View Print Preview				
Dimensions Constraints Settings Sketching Modelling Details View Patients of Sketchi Sketch is	Circle by 3 Tangents			
Constraints Settings Settings Details View P Details of Sketch1 Sketch Sketch1 Sketch Sketch1 Sketch Sketch1 Show Constraints1 No E Edges: 1 Full Circle C/7 Model View Print Preview	Modify	▼		
Settings Settings Details of Sketch Sketch Sketch Show Constants? No 0.000 E Edges: 1 Full Circle Cr7 Model View Print Preview	Dimensions			
Sketching Modeling Details View P © Details of Sketchi Sketchi Sketching Show Constraint? Show Constraint? No Edges: 1 0.000 Full Circle C/7 Model View Print Preview	Constraints			
Details View Details View Details View Details View Details View Details View Details View Details View	Settings			
□ Details of Sketch1 Sketch Sketch1 Shetch Visibility Show Sketch Show Constraints? No □ Edges: 1 Full Circle Cr7 0.000 0.300 (m) Model View Print Preview	Sketching Modeling		/	
Sketch Sketch1 Sketch Visibility Show Sketch Show Constants No E Edges: 1 Full Circle Cr7 Model View Print Preview 0.000 0.300 (m)	Details View			
Sketch Visibility Show Sketch Show Constaints? No Edges: 1 0.000 Puil Circle Cr7 Model View Print Preview	Details of Sketch1			
Show Constraints? No Edges: 1 Full Circle Cr7 Model View Print Preview				×
Edges: 1 0.080 0.300 (m) Full Circle C/7 0.150 Model View Print Preview				· · · · · · · · · · · · · · · · · · ·
Full Circle C/7 0.000 0.300 (m) 0.150 0.150 0.150				
Model View Print Preview				
Model View Print Preview	Full Circle Cr7		0.000 0.300 (m)	• •
			0.150	
Circle Click, or Press and Hold, for center of circle No Selection Meter 0 0		Model View Print Preview		
	Circle Click, or Press and Hold, for center of circle	,	No Selection	Meter 0 0

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



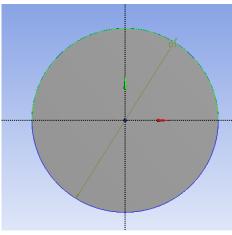
4.9. Concept > Surface From Sketches. Select your sketch, click Apply, then click the Generate button.



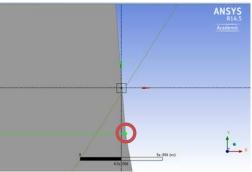
4.10. **Create** > **Boolean**. Change operation to **Subtract** then select the circle for **Target Bodies** then select airfoil for **Tool Bodies** and click **Generate**. This will subtract the airfoil surface from the circle.

File Create Concer	ot Tools View He
🛛 🖉 🛛 🛧 New Plane	
XYPI 💽 Extrude	
🗧 🥩 🤆 💏 Revolve	
🕞 T 🌭 Sweep	
Tree O	
Im Thin/Surfac	e ·
Fixed Radius	Blend
Straible Rac	
< Vertex Blend	
♦ Chamfer	
· · · · · · · · · · · · · · · · · · ·	
Pattern	
👰 Body Opera	tion
Boolean	
Slice	
S Face Delete	
Ne Edge Delete	
Point	
Primitives	•
Details View	4
Details of Boolean1	
Boolean Boolean1	
Operation Subtract	
Target Bodies 1 Body	
Tool Bodies 1 Body	
Preserve Tool Bodies? No	

4.11. **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.

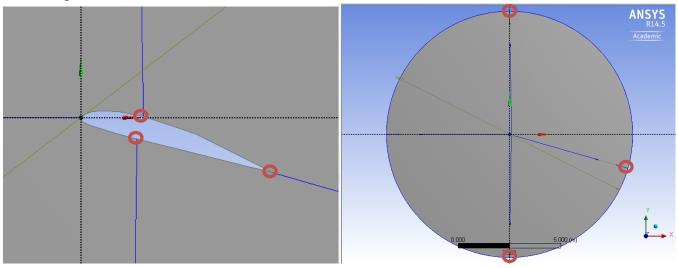


- 4.12. Repeat the process from 4.11 on the two semicircles. This should yield four circular quadrants.
- 4.13. Repeat this process for the arc in quadrant IV. Change the Fraction to 0.822222. This splits the arc into a 16° and a 74° arc.
- 4.14. **Concept** > **Lines From Points**. Draw a line from 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 use that point. The images below show the locations of the points circled.

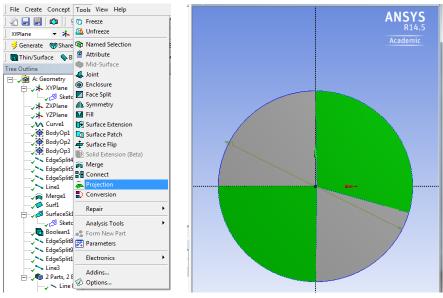


A: CFD Pre-Lab 2 Turbulent Flow - DesignModeler		0 B - X
File Create Concept Tools View Help		
2		
II- II- h- h- h- h- h- # #		
XMane + A- Sketchs + 21 Generate @filmer?	malany Parameters	
Bonule Alexabre Savera Shoulant BTher/Surface		
Tree Outline	Graphics	
Constant Constan		ANSYS REAS Academic
Sketching Modeling		
Details View	•	
Details of Line3		
Lines From Points Line3 Point Secondaria Apoly Cancel		
Operation Add Material		1.
	1.80	sama dirij
Unes From Points Creation Select pairs of 2D points, 3D vertices, and Points Points Creation Select pairs of 2D points.	Model View Print Preview int Feature 2 Point Segment	Meter 0 0

- 4.15. Once you select both points hit **Apply.** Then click **Generate.**
- 4.16. 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.



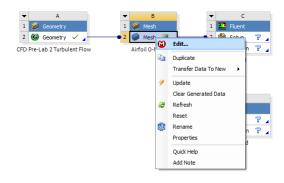
- 4.17. **Tools** > **Projections**. Select the four lines you created for **Edges** and select the circle for **Target** then click **Generate**. This will split your geometry into four sections
- 4.18. **Tools > Merge.** Select the 16° arc and the arc in quadrant I and select **Apply.** Click **Generate.** This merges the lines into one line which can be sized for meshing easier.



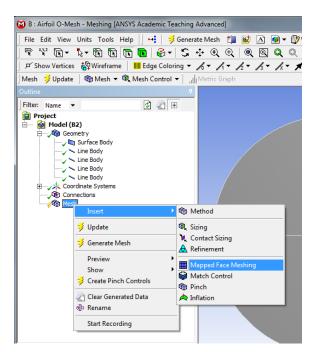
4.19. **File** > **Save project** and exit.

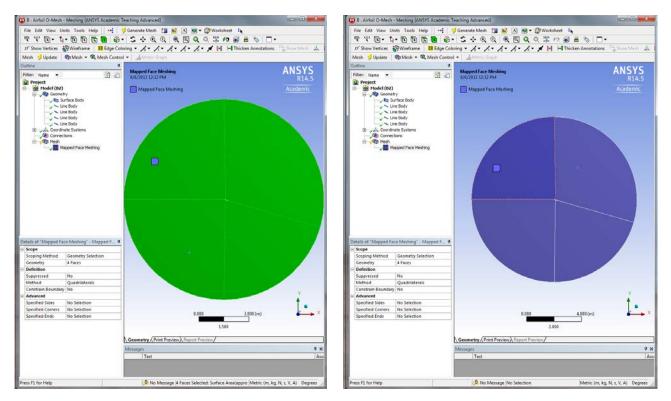
5. Mesh Generation

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

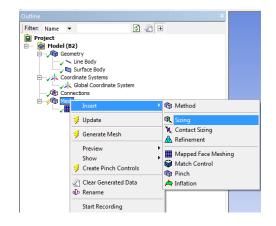


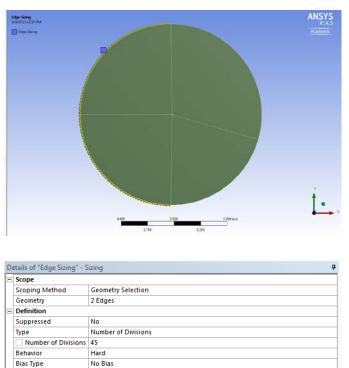
5.2. Right click **Mesh** then **Insert** > **Mapped Face Meshing**. Select the four surfaces then click **Apply**.



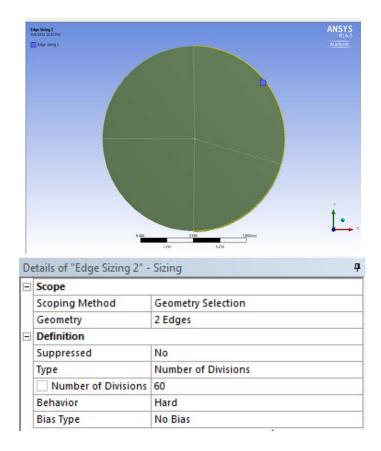


5.3. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.

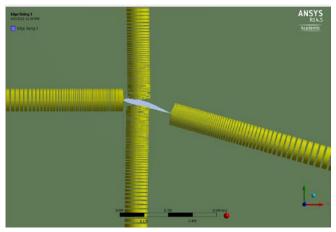




5.4. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.



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.

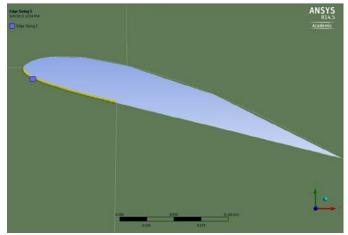


De	etails of "Edge Sizing 3" - Sizing 🕂 🖓				
-	Scope				
	Scoping Method	Geometry Selection			
	Geometry	4 Edges			
-	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	115			
	Behavior	Hard			
	Bias Type				
	Bias Option	Bias Factor			
	Bias Factor	5000.			

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.

Kange Salag 4 And Set (2020 Hard) Frige Sang 4	ANSYS RELS Aradimic
	8486 (2.83) X,200 (m) A.825 K.875
Details of "Edge Sizing 4" -	Sizing 4
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
 Definition 	
Suppressed	No
Туре	Number of Divisions
Number of Divisions	45
Behavior	Hard
Bias Type	
Bias Option	Bias Factor
Bias Factor	15.

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.

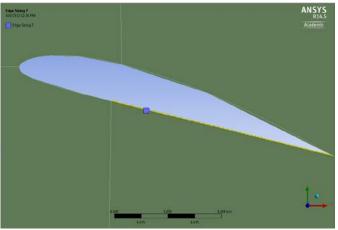


D	etails of "Edge Sizing 5" - Sizing 🕈 🕈			
-	Scope		_	
	Scoping Method	Geometry Selection		
	Geometry	1 Edge		
Ξ	Definition			
	Suppressed	No		
	Туре	Number of Divisions		
	Number of Divisions	45		
	Behavior	Hard		
	Bias Type			
	Bias Option	Bias Factor		
	Bias Factor	15.		

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.

	Principal Constructions Lidge Some 6	ANSYS
		170 <u>1</u> 00 (v)
De	tails of "Edge Sizing 6" -	Sizing 7
	Scope	
	Scoping Method	Geometry Selection
	Geometry	1 Edge
	Definition	
	Suppressed	No
	Туре	Number of Divisions
	Number of Divisions	60
	Behavior	Hard
	Bias Type	
	Bias Option	Bias Factor
	Bias Factor	15.

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.

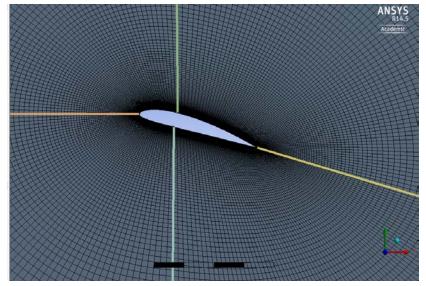


De	etails of "Edge Sizing 7" - Sizing 🗸 🖓 🖓					
-	Scope		_			
	Scoping Method	Geometry Selection				
	Geometry	1 Edge				
-	Definition					
	Suppressed	No				
	Туре	Number of Divisions				
	Number of Divisions	60				
	Behavior	Hard				
	Bias Type					
	Bias Option	Bias Factor				
	Bias Factor	15.				

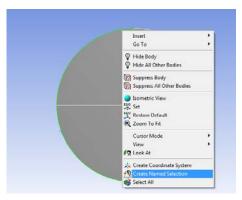
5.10. Click on Mesh under the Outline, under the Details of "Mesh", change the PhysicsPreference from Mechanical to CFD. This changes the grid solver to a fluids style solver rather than a FEA style solver.

Details of "Mesh"			
 Defaults 			
Physics Preference	CFD 🔻		
	Mechanical Electromagnetics CFD		
	Explicit		
+ Inflation			
 Assembly Meshing 			
Method	Method None		
Patch Conforming Option	ns		
Triangle Surface Mesher	Triangle Surface Mesher Program Controlled		
+ Advanced			
Defeaturing	Defeaturing		
+ Statistics			

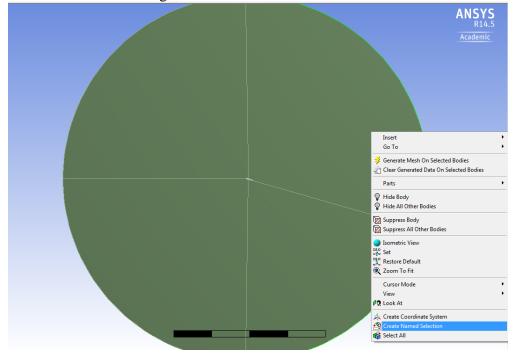
5.11. Click **Generate Mesh**. Click on the **Mesh** button under the **Outline** and make sure it resembles the mesh below.



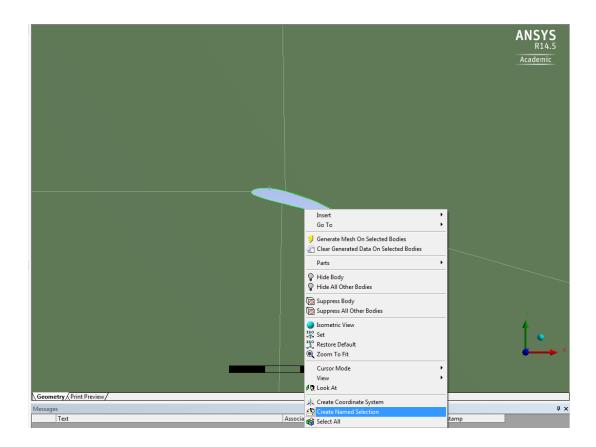
5.12. Hold Ctrl and select the two left most semicircle arcs, right click on them and select **Create Named Selection**, name the selection *inlet*. Use the edge select button from the toolbar.



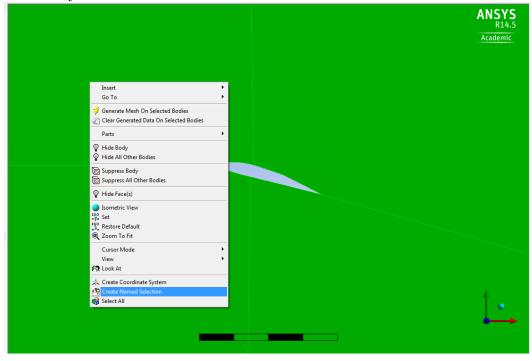
5.13. Do the same for the two right most semicircle arcs and name them *outlet*.



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



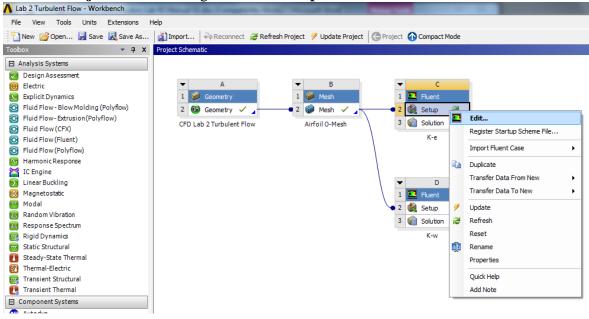
5.15. Now use the face button to select the four semicircle quadrants and **Create Named Selection** and name them *fluid*.



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

6. Physics

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



6.2. Select Double Precision and click OK.



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-ɛ model			
File Mesh Define So	, dp, pbns, ske] [ANSYS Academic Teaching Advanced] Ive Adapt Surface Display Report Parallel Vie @ [] ♡ ⑦ ⑦ ① ① 》 [] ◎ 八 [] • □ •]		
Meshing Mesh Generation Solution Setup General Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Solution Initialization Calculation Results Graphics and Animations Plots Reports	Models Models Models Multiphase - Off Encrys - Off Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Solidification & Melling - Off Acoustics - Off Edit	Viscous Model Model Inviscid Laminar Spalart-Almaras (1 eqn) kepsilon (2 eqn) Transition k43 omega (2 eqn) Transition k43 omega (2 eqn) Transition k51 (eqn) Scale-Adaptive Simulation (SAS) kepsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Enhanced Wall Freatment User-Defined Wall Functions Enhanced Wall Treatment Options Pressure Gradient Effects Options Curvature Correction OK	Model Constants Omu O.09 C1:Epsilon I.44 C2:Epsilon I.92 TVE Prandtl Number User-Defined Functions Turbulent Viscosity none TRE Prandtl Number TRE Prandtl Number Inone TRE Prandtl Number Cancel Help

k-ω model

🖸 G:K-omega Fluent [2d, dp, pbns, skw] [ANSYS Academic Teaching Advanced]				
File Mesh Define Sol	File Mesh Define Solve Adapt Surface Display Report Parallel View Help			
i 📖 i 📂 🕶 🖬 🕶 🚳	@∥5⊕€€ ∥∥® % ⊪ - □ -			
Meshing	Models	Viscous Model	<u> </u>	
Mesh Generation Solution Setup General Motified Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Models Multiphase - Off Function - Off Versional-Standard K-senege Reduction - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Soldriction & Mething - Off Acoustics - Off Edit	Model Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-expsilon (2 eqn) Transition K4-lonega (3 eqn) Transition SST (4 eqn) Scale-Adaptive Simulation (SAS) k-omega Model Standard SST komega Options Low-Re Corrections Options Curvature Correction Octions Curvature Correction Octions Options Curvature Correction Octions Options Curvature Correction Octions Options Curvature Correction Octions Octions Options Curvature Correction Octions Oction Octio	Model Constants Alpha*_inf Alpha*_inf O.52 Beta*_inf 0.69 Beta_i 0.072 User-Defined Functions Turbulent Viscosity Prandil Numbers TRE Prandil Number Inone SDR Prandtl Number Cancel Help Cancel Help	

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

🛂 C:K-e Fluent [2d, dp, pbns, ske] [ANSYS Academic Teaching Advanced]				
File Mesh Define So	lve Adapt Surface Display Report Parallel Vi	iew Help		
i 💼 i 📂 🕶 🖬 🕶 💿	@ \$₽0€//!®┴⊪+□+			
Meshing	Materials		•	
Mesh Generation	Materials	Create/Edit Materials		×
Solution Setup	Fluid	Name		Order Materials by
General	air Solid	air	Material Type	Name
Models Materials	aluminum	Chemical Formula	India	Chemical Formula
Phases			Fluent Fluid Materials	Fluent Database
Cell Zone Conditions			air	· · · · · · · · · · · · · · · · · · ·
Boundary Conditions Mesh Interfaces			Mixture	User-Defined Database
Dynamic Mesh			none	
Reference Values		Properties		
Solution		Density (kg/m3) consta	ent Edit	<u>^</u>
Solution Methods		1.188		
Solution Controls Monitors		1.100	5	
Solution Initialization		Viscosity (kg/m-s) consta	nt Edit	
Calculation Activities			i6e-05	
Run Calculation Results		1.035	62-03	=
Graphics and Animations				
Plots				
Reports				
	Create/Edit Delete			
	Help			
		, · · · · · · · · · · · · · · · · · · ·		
			Change/Create Delete Clo	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 and click OK. The value can be found from the EFD Lab 3 data reduction sheet.

k-ε model				
File Mesh Define So	uent [2d, dp, pbns, ske] [ANSYS Academic Teaching Ad Ive Adapt Surface Display Report Parallel Vi @ [15] (中) () (中) () () () () () () () () () () () () ()			
Meshing Mesh Generation Solution Setup General Models	Boundary Conditions Zone December 201 Decemb	1: Mesh Velocity Inlet Zone Name inlet		
Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors		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 Y-Velocity (m/s) 0 constant		
Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Edt Copy Profies Parameters Operating Conditions Display Meth	Turbulence Specification Method K and Epsilon Turbulent Kinetic Energy (m2/s2) 0.08 constant Turbulent Dissipation Rate (m2/s3) 7.4 constant		
	Help	OK Cancel Help		

k-ω model						
G:K-omega Fluent [2d	, dp, pbns, skw] [ANSYS Academic Teaching Advance	ed]				- 0 X
File Mesh Define So	lve Adapt Surface Display Report Parallel	Viev	v Help			
1 🛋 1 📸 ד 🖬 ד 🚳	@∥\$₽€€⊁∥®Ҳ⊪+□+					
Meshing	Boundary Conditions	Â	Velocity Inlet			X
Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Barndsay Conditions Barndsay Conditions Dynamic Mesh Mesh Inter Faces Dynamic Mesh Reference Values Solution Solution Methods Solution Intellazation Calculation Activities Run Calculation Result	Zone partol pret interior-fuid outlet surface_body Phase Type to yelocity-interior yelocity-interior f 6	E	Zone Name Inlet Momentum Thermai Radiation Specier Velocity Specification Method Reference Frame Supersonic/Initial Gauge Pressure (pascal) X-Velocity (m/s) Y-Velocity (m/s) Turbulence Specification Method [Turbulent Kinetic Energy (m2/s2)	Components Absolute 0 15 0 Cand Omega 0.08	hase UDS (constant (constant (constant (constant	• • • • • • • • • • • • • • • • • • •
Graphics and Animations Plots Reports	Edit Copy Profiles Parameters Operating Conditions Periodic Conditions		Specific Dissipation Rate (1/s)		constant	•
		+	< III			4

6.6. Solution Setup > Boundary Conditions > Outlet > Edit... Change the parameters as per below and click OK.

	, dp, pbns, ske] [ANSYS Academic Teaching Advanced]	
	lve Adapt Surface Display Report Parallel Vi	iew Help
💼 i 📸 🕶 🛃 🕶 🎯	❷∥\$\$ €€ € ↗∥® % ⊪ - □ -	
Meshing	Boundary Conditions	1: Mesh 🗸
Mesh Generation	Zone	Pressure Outlet
Solution Setup General Models Materials Phases Cell Zone Conditions Ecoundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Prist Phase Type mixture V pressure-outlet V Faster Profiles	Zone Name outlet Momentum Thermal Radation Species DPM Multiphase UDS Gauge Pressure (pascal) 0 constant Backflow Direction Specification Method [Normal to Boundary Average Pressure Specification Target Mass Flow Rate Turbulence Specification Method [Intensity and Viscosity Ratio Backflow Turbulent Intensity (%) 3.25 B Backflow Turbulent Viscosity Ratio 0.0035 B OK Cancel Help
	Display Mesh Periodic Conditions	r K III

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

Meshing	Reference Values
Mesh Generation	Compute from
Solution Setup	· · · · · · · · · · · · · · · · · · ·
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Reference Values Area (m2) 0.3048 Density (kg/m3) 1.1885 Depth (m) 1 Enthalpy (j/kg) 0
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Length (m) 1 Pressure (pascal) 0 Temperature (k) 297.15
Results	Velocity (m/s)
Graphics and Animations Plots Reports	Viscosity (kg/m-s) 1.8396e-05
	Ratio of Specific Heats 1.4
	Reference Zone

7. Solution

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

C:Viscid Fluent [2d, dp	, pbns, ske] [ANSYS Academic Teaching Advanced]	
File Mesh Define So	lve Adapt Surface Display Report Parallel Vie	w Help
:	@∥\$\$₽€€≯∥®% № ⊪ - □ -	
Meshing	Solution Methods	1: Mesh
Mesh Generation	Pressure-Velocity Coupling	
Solution Setup	Scheme	
General Models	SIMPLE	
Materials	Spatial Discretization	
Phases	Gradient	
Cell Zone Conditions Boundary Conditions	Green-Gauss Cell Based	
Mesh Interfaces	Pressure	
Dynamic Mesh Reference Values	Standard 🔹	
Solution	Momentum	
Solution Methods	Second Order Upwind	
Solution Controls	Turbulent Kinetic Energy	
Monitors	Second Order Upwind	
Solution Initialization Calculation Activities	Second Order Upwind	
Run Calculation	Transient Formulation	
Results		Mesh Aug 04, 2013
Graphics and Animations Plots	Non-Iterative Time Advancement	ANSYS Fluent 14.5 (2d, dp, pbns, lam)
Reports	Frozen Flux Formulation	surface_body ^
		interior-fluidSetting fluid (mixtur
		Setting zone id of fluid to 5. Setting zone id of interior-fluid to 1.
	Default	Setting zone id of surface_body to 2.
		Setting zone id of inlet to 6.
	Help	Setting zone id of outlet to 7. Setting zone id of airfoil to 8.
		Done.
		Setting fluid (mixture) Done.
		Setting surface_body (mixture) Done.
		Setting inlet (mixture) Done.
		Setting outlet (mixture) Done.
		• • • • •

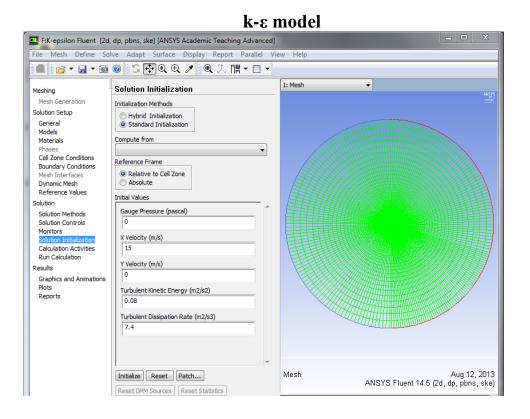
7.2. Solution > Solution Controls. Change Parameters as per below. (If you have problems with the solution converging, you can decrease the Under –Relation Factors.)

CViscid Eluent 12d. dr	p, pbns, ske] [ANSYS Academic Teaching Advanced]	
	lve Adapt Surface Display Report Parallel Vi	ew Help
	⑧ ⑤ ❹ € € ↗ ◎ 久 閙 - □ -	
Meshing	Solution Controls	1: Mesh
Mesh Generation	Under-Relaxation Factors	
Solution Setup	Body Forces	
General Models	1	
Materials	Momentum	
Phases	0.5	
Cell Zone Conditions Boundary Conditions		
Mesh Interfaces	Turbulent Kinetic Energy 0.8	
Dynamic Mesh	0.8	
Reference Values	Turbulent Dissipation Rate	
Solution	0.8	
Solution Methods	Turbulent Viscosity	
Monitors	1	
Solution Initialization Calculation Activities		
Run Calculation	Default	
Results	Equations Limits Advanced	Mesh Aug 04, 2013
Graphics and Animations	Equatoristic Emiliant Advancediti	ANSYS Fluent 14.5 (2d, dp, pbns, lam)
Plots		surface body
Reports	Help	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 inlet (mixture) Done.
		Setting outlet (mixture) Done.

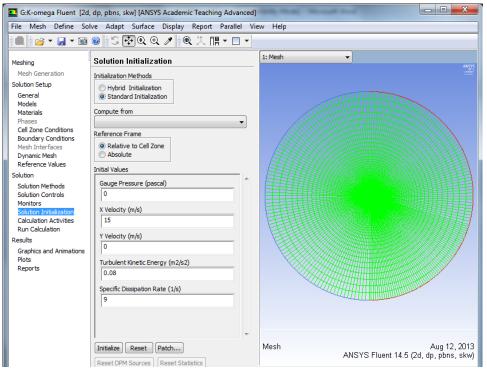
7.3. **Solution** > **Monitors** > **Residuals** –**Print**, **Plot**. Change the convergence limit to 1e-05 for all five equations.

C:Viscid Fluent [2d, dp	o, pbns, ske] [ANSYS Academic Teaching Advanced	
File Mesh Define So	lve Adapt Surface Display Report Parallel	I View Help
i 📖 i 📂 - 🛃 - 🕥	@ \$₽€€ /∥€ % - □	
Meshing	Monitors	Residual Monitors
Mesh Generation	Residuals, Statistic and Force Monitors	Options Equations
Solution Setup	Residuals - Print, Plot	Print to Console Continuity V 1e-05
General Models	Statistic - Off	V Plot Ie-05
Materials		Window
Phases		1 ▲ Curves Axes v-velocity V 1e-05 Ξ
Cell Zone Conditions Boundary Conditions	Create V Edit Delete	Iterations to Plot k V Ie-05
Mesh Interfaces	Surface Monitors	1000 epsilon le-05 v
Dynamic Mesh	Surface Monitors	
Reference Values Solution		Residual Values Convergence Criterion
Solution Methods		Teratoris to otore
Solution Controls		
Monitors		Scale
Solution Initialization Calculation Activities	Create Edit Delete	Compute Local Scale
Run Calculation	Volume Monitors	
Results		OK Plot Renormalize Cancel Help
Graphics and Animations		Area to tradem 14.5 (20, up, poins, iam)
Plots Reports		surface_body
	<u> </u>	interior-fluidSetting fluid (mixtur
	Create Edit Delete	Setting zone id of fluid to 5. Setting zone id of interior-fluid to 1.
	Convergence Monitors	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.
	Convergence Manager	Setting surface_body (mixture) Done.
		Setting inlet (mixture) Done.
		secting outlet (Mixture) pone.
	Help	■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■

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



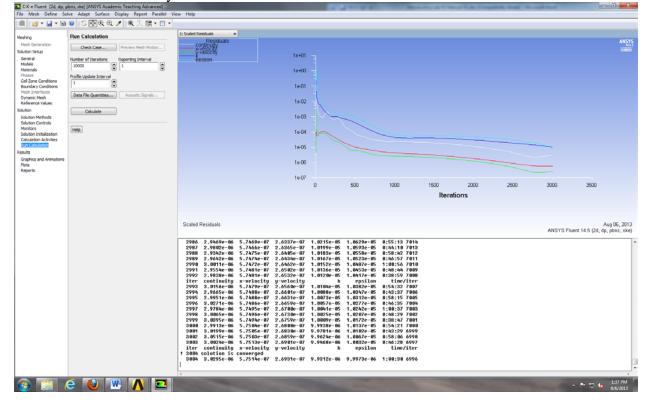
k-ω model



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

C:Viscid Fluent [2d, dp	, pbns, ske] [ANSYS Academic Teaching Advanced]
File Mesh Define So	lve Adapt Surface Display Report Parallel Vi
💷 🛛 🗃 🕶 🔛 🕶 🚳	Ø \$\$ € € €
Meshing	Run Calculation
Mesh Generation Solution Setup	Check Case Preview Mesh Motion
General Models Materials Phases	Number of Iterations 10000 Reporting Interval
Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Profile Update Interval 1 Image: Construction of the second
Solution Solution Methods Solution Controls Monitors Solution Initialization	Calculate
Calculation Activities Run Calculation Results Graphics and Animations Plots	
Reports	

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 > Residu	als –Print, Plot >	> Edit > Plot	: > Cancel
------------------------------	--------------------	---------------	------------

File Mesh Define Sol	o, pbns, skej [ANSYS Academic Teaching Advanced]	23
Meshing Mesh Generation Solution Setup General Models Materials Cell Zone Conditions	Image: Statistic and Force Monitors 1: Scaled Residuals Statistic and Force Monitors Image: Statistic and Force Monitors Statistic - Off Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors Image: Statistic and Force Monitors I	ANSYS Ista Acteri
Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitore Solution Initialization Calculation Activities	Create Edituri Delete Surface Monitors It-05 It-	
Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Vou Options Equations V Print to Console V Pri	
	Con I Convergence Criterion Iterations to Flot Residual Values Convergence Criterion Residual Values Convergence Criterion absolute	-
	1000 Image: Compute Local Scale Image: Compute Local	
	Unit OK Plot Renormalize Cancel Heip A. 3237e-86 1.8124e-85 0:165:44 7502 2500 2.5628e-86 1.9436e-87 1.3538e-87 4.3124e-95 0:105:44 7502 2500 2.5628e-86 1.9436e-87 1.3538e-87 4.3124e-95 0:83:40 7501 2501 2.6328e-86 1.9486e-87 1.3525e-87 4.33965e-86 1.8086e-85 0:82:56 7499	C

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				x		
Format © EPS © JPEG © PPM © PostScript	ColoringFile TypeImage: ColorRasterImage: Gray ScaleVectorImage: MonochromeVector		Resolution Width 960 Height 720			
TIFF PNG VRML Window Dump	Options Image: Contract of the second seco		low Dump Command ort -window %w			
Save Apply Preview Close Help						

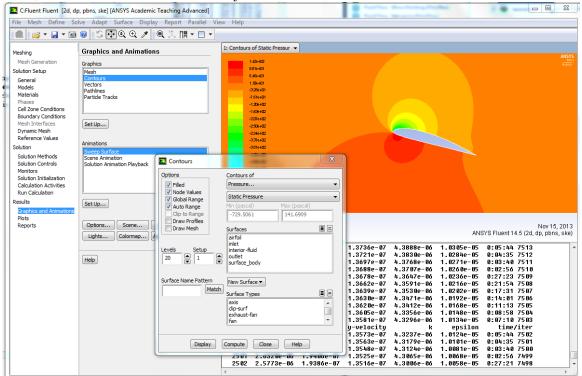
8.2. Plotting Pressure Coefficient Distribution with CFD and EFD Data

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*.

CiFluent Fluent (2d dn nhns	, ske] [ANSYS Academic Teaching Advanced]	A restore later	
	dapt Surface Display Report Parallel Vie		
		w Help	
,	5 🕀 Q 원 🗡 🔍 🎘 📗 🗸 📗 🗸		
Meshing Plots	S	1: Pressure Coefficient 🔹	
Mesh Generation Plots		airfoil experimental	ANSYS RADA TATATAT
Solution Setup		2008+00	
General Histog	ogram	1.008+00	\sim
Profile			
Phases Inter	file Data - Unavailable erpolated Data	0.000++00	
Cell Zone Conditions FFT Boundary Conditions		-1.000+400	
Mesh Interfaces		Pressure -2000++000	August and a second
Dynamic Mesh		Coefficient	· · · · /
Reference Values Solution		-3.006+60	
Solution Methods		-4 D0:+D	
- Solution Controls	Solution XY Plot	×	
Monitors Solution Initialization	Options Plot Direction	Y Axis Function	
Calculation Activities	Node Values X 1	Pressure	
Run Calculation Set Up		Pressure Coefficient 👻	Position (m)
Results Graphics and Animations	Write to File	X Axis Function	
Plots Help	Order Points Z 0	Direction Vector 🗸	
Reports	File Data	Surfaces 🔳 🗏	
	pressure coefficient	airfoil inlet	ANSYS Fluent 14.5 (2d, dp, pbns, ske)
		interior-fluid outlet	.3736e-07 4.3888e-06 1.0305e-05 0:05:44 7513 ^
-		surface_body	.3721e-07 4.3830e-06 1.0284e-05 0:04:35 7512 .3697e-07 4.3768e-06 1.0271e-05 0:03:40 7511
2			.3688e-07 4.3707e-06 1.0260e-05 0:02:56 7510
	Load File		.3678e-07 4.3647e-06 1.0236e-05 0:27:23 7509 .3662e-07 4.3591e-06 1.0216e-05 0:21:54 7508
	Free Data	New Surface 💌	.3639e-07 4.3530e-06 1.0202e-05 0:17:31 7507
		incursurace .	
	Plot Axes	Curves Close Help	.3605e-07 4.3356e-06 1.0148e-05 0:08:58 7504
4		iter continuity x-velocity	
			1.3573e-07 4.3237e-06 1.0124e-05 0:05:44 7502
			1.3563e-07 4.3179e-06 1.0101e-05 0:04:35 7501
			1.3548e-07 4.3124e-06 1.0081e-05 0:03:40 7500 1.3525e-07 4.3065e-06 1.0068e-05 0:02:56 7499
			•
		<	۹

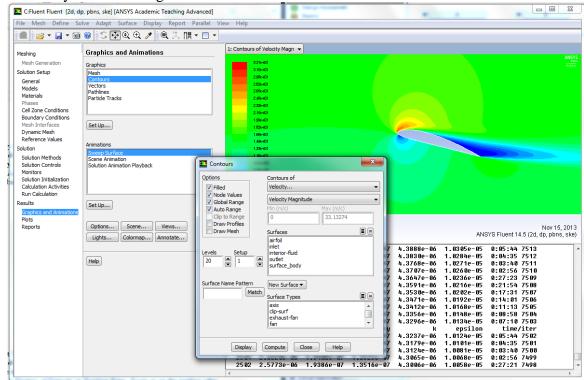
8.3. Plotting Contour of Pressure

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 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.

	iolve Adapt Surface Displ		Help					
hina	Graphics and Animatio		: Velocity Vectors Colored B 👻					
esh Generation	Graphics		3.34+01					AN
tion Setup	Mesh		3.18e+01					
eneral	Contours		301e+01					
odels	Vectors		2,5%+01					
aterials	Pathlines Particle Tracks		25310+01	+ + -				
lases			234+01					
ell Zone Conditions			2.17e+01					
undary Conditions			2.016+01	Sec. 2 L				
esh Interfaces /namic Mesh	Set Up		1.84e+01	Carl.	1	~ ~ ~	+ +	
eference Values			1.67e+01		1 · ·		A	Ł.
tion	Animations		1516401/2	×	Maria and	• • 、*		
lution Methods	Sweep Surface	Vectors	and the second se			$()) \rightarrow $		
lution Methods	Scene Animation Solution Animation Playback	Options	Vectors of	A.A.	a the second	$(I = 1) = (N_{ij})$		
onitors	Solution Animation Playback	Global Range	Velocity		ALA ST	$\mathcal{A}_{\mathcal{A}}(\mathcal{A}_{\mathcal{A}}) = \mathcal{A}_{\mathcal{A}}$	N	
lution Initialization		Auto Range	Color by		A CONTRACTOR OF THE OWNER	Cit Carlos		
alculation Activities		Clip to Range				and the second s		-
In Calculation	I	Auto Scale	Velocity		North H		A A A	N.
ults	Set Up	Draw Mesh	Velocity Magnitude		Real K	A PROPERTY AND		
aphics and Animations		Style	Min (m/s) Max (m/s)		A - K			
ots		arrow -	0.01478644 33.4482					
eports	Options Scene			m/s)			Nov 1	
	Lights Colormap	A Scale Skip	Surfaces			ANSYS FI	uent 14.5 (2d, dp, pb	uns, s
		200 10		97	36e-07 4.3888e-06	1.0305e-05 0:0	5:44 7513	
			inlet interior-fluid		21e-07 4.3830e-06		4:35 7512	
	Help	Vector Options	outet		97e-07 4.3768e-06		3:40 7511	
		Custom Vectors	surface_body	.36	88e-07 4.3707e-06		2:56 7510	
					78e-07 4.3647e-06		7:23 7509	
		Surface Name Pattern			62e-07 4.3591e-06		1:54 7508	
		Match	New Surface -		39e-07 4.3530e-06 30e-07 4.3471e-06		7:31 7507 4:01 7506	
		Match			20e-07 4.3412e-06		4:01 7500	
			Surface Types		05e-07 4.3356e-06		8:58 7504	
			axis dip-surf		81e-07 4.3296e-06		7:10 7503	
			exhaust-fan		locity k		time/iter	
			fan		73e-07 4.3237e-06		5:44 7502	
					63e-07 4.3179e-06		4:35 7501	
					48e-07 4.3124e-06 25e-07 4.3065e-06		3:40 7500	
		Display	Compute Close Help	1.35	25e-07 4.3065e-06	1.0068e-05 0:0	2:56 7499	

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*.

C:Fluent Fluent [2	d, dp, pbns, ske] [ANSYS Academ	nic Teaching Advanced]	-	A residence landers						
File Mesh Define Solve Adapt Surface Display Report Parallel View Help										
:■ :::::::::::::::::::::::::::::::::::										
Meshing	Graphics and Animatic	ons	1: Contours of	Stream Functi 👻						11101/0
Mesh Generation	Graphics			1185						ANSYS
Solution Setup	Mesh									
General	Contours Vectors									
Models Materials	Pathlines									
Phases	Particle Tracks									
Cell Zone Conditions										
Boundary Conditions Mesh Interfaces	Set Up									
Dynamic Mesh										
Reference Values	Animations									
Solution Solution Methods	Sweep Surface									
Solution Controls	Scene Animation Solution Animation Playback	—		TR	×					
Monitors		Contours								
Solution Initialization Calculation Activities		Options	Contours of							
Run Calculation		Filled	Velocity		•					
Results	Set Up	Vode Values	Stream Functi	on	•					
Graphics and Animati Plots	ons	Auto Range	Min (kg/s)	Max (kg/s)						
Reports	Options Scene	Clip to Range	103	107.5						Nov 15, 2013
	Lights Colormap	Draw Mesh	Surfaces					ANS	SYS Fluent 14.5 (2d,	
			airfoil inlet			1.3736e-07	4.3888e-0ó	1.0305e-05	0:05:44 7513	~
	Help	Levels Setup	interior-fluid			1.3721e-07 1.3697e-07	4.3830e-06 4.3768e-06	1.0284e-05 1.0271e-05	0:04:35 7512 0:03:40 7511	
		100 🔺 1 🗭	outlet surface body			1.3688e-07		1.0260e-05	0:02:56 7510	
						1.3678e-07		1.0236e-05	0:27:23 7509	
		Surface Name Pattern	New Surface	•		1.3662e-07 1.3639e-07		1.0216e-05 1.0202e-05	0:21:54 7508 0:17:31 7507	
		Match	Surface Types			1.3630e-07	4.3471e-06	1.0192e-05	0:14:01 7506	
			axis		<u> </u>	1.3620e-07 1.3605e-07		1.0168e-05 1.0148e-05	0:11:13 7505 0:08:58 7504	
			dip-surf exhaust-fan			1.3581e-07		1.0134e-05	0:07:10 7503	
			fan		-	y-velocity 1.3573e-07	k	epsilon 1.0124e-05	time/iter 0:05:44 7502	
						1.3573e-07		1.0124e-05	0:05:44 7502	
		Display	Compute	Close Help		1.3548e-07	4.3124e-06	1.0081e-05	0:03:40 7500	_
					20008-87	1.3525e-07 1.3516e-07		1.0068e-05 1.0058e-05	0:02:56 7499 0:27:21 7498	-
			<		10000 01				5.21.21 1470	

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*.

1050102 1050102 1050102	Force Reports		
1:056+02	Options Direction Vector Wall Zones		
Contours of Stream Function (kg/s)	Moments Center of Pressure Y		Aug 05, 2013 ANSYS Fluent 14.5 (2d, dp, pbns, ske)
2234 2.1639e-06 9.4062e-08 7.7806e-08 2235 2.1624e-06 9.4014e-08 7.7784e-08 2236 2.1689e-06 9.3056e-08 7.7784e-08 2237 2.1592e-06 9.3057e-08 7.7784e-08 2238 2.1597e-06 9.3017e-08 7.7792e-08 2239 2.1572e-06 9.3806e-08 7.7764e-08 2239 2.1572e-06 9.3826e-08 7.7769a-08 2240 2.1542e-06 9.38720e-08 7.7693e-08 2240 2.1547e-06 9.3771e-08 7.7669e-08	2 Val Name Pattern 2 Match 2 Save Output Parameter		
Forces Forces (n)	Print Write Close Help		
Zone Pressure	Viscous		Total
airfoil (0.36176218 13.099	955 0) (0.51839448 0.015	682267 0)	(0.88015666 13.115638 0)
Net (0.36176218 13.099	955 0) (0.51839448 0.015	682267 0)	(0.88015666 13.115638 0)
Forces - Direction Vector (1 0 0) Forces (n)	Coefficients		
	scous Total Pressure	Viscous Total	
airfoil 0.36176218 0.	51839448 0.88015666 0.0088768006	0.012720192 0.021596993	
Net 0.36176218 0.	51839448 0.88015666 0.0088768006	0.012720192 0.021596993	
< III			4

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.

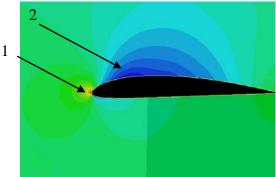
Note: These questions are also available in the "CFD Lab2 Report Template" where you will type answers.

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.