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.



N Unsaved Project - Workbench	and the second se	
File View Tools Units Extensions H	elp	
New 🚰 Open 🛃 Save 🔣 Save As	👔 Import 🏻 🍣 Reconnect 🛛 Refresh Project 🍼 Update Proje	ct 🕝 Project 🏠 Compact Mode
Toolbox 🗸 🕂 🗙	Project Schematic	- - + ×
Magnetostatic		
Modal		
Random Vibration	▼ A B	▼ C
Response Spectrum	1 🥪 Geometry 1 😻 Mesh	1 🖸 Fluent
nigid Dynamics	2 🔞 Geometry 🗸 🛛 2 🍘 Mesh 🥰	🗕
w Static Structural	CED Lab 2 Turbulant Flow Airfoil O-Mach	3 🗑 Solution 🏆
🕐 Steady-State Thermal		Kee
Thermal-Electric		k.e
Transient Structural		
U. Transient Thermal		
Component Systems		▼ D
🕰 Autodyn		1 Eluent
CFX CFX		
Sector Engineering Data		
External Connection		3 Mill Solution 😨 🔺
External Data		K-w
Finite Element Modeler		
Fluent (with TO iden action)		
Fluent (with IGrid meshing)		
Geometry		
Machanical APDI		
Machanical Model		
Mesh		
Microsoft OfficeExcel		
Polyflow - Blow Molding		
Polyflow - Extrusion		
💿 Results 🔻		
View All / Customize		
A Messages		
Ready		Show Progress

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

Sketching Toolboxes		
	Draw	
	Modify	
	Dimensions	
	Constraints	*
777 Fixed		
🚃 Horizontal		
Vertical		
✓ Perpendicular		
Coincident		
Midpoint		
ণ Symmetry		
🥢 Parallel		
Oncentric		
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.





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



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





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

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

cept Tools View Help	
Undo @Redo Select: 🌇 💱	k
🔸 Sketch1 👻 ಶ	
Share Topology 🐹 Parameters 📗 💽 Extrude	6
💊 Blend 👻 💊 Chamfer 🛭 👘 Slice 🔢 🛷 Poi	nt
	ą
Draw	
Modify	
Dimensions	
2	
-	
Constraints	1_
Settings	· ·
	_
	ų
Sketch1	
Show Sketch	
(NO	
12 m	
12 m	
12 m	
	ept Tools View Help Dudo @Redo Select To To To Select To To To Praw Chamfer Sice Poi Draw Modify Dimensions Constraints Settings g Sketch1 Show Sketch

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



4.11. **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.





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. 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 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.



- 4.16. Once you select both points hit **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. 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. 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.



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 Andrea 1233 mar 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"	ф.	
 Defaults 		
Physics Preference	CFD 🗸	
Solver Preference	Mechanical Electromagnetics	
+ Sizing	Explicit	
Inflation		
 Assembly Meshing 		
Method	None	
 Patch Conforming Optio 	ns	
Triangle Surface Mesher	Program Controlled	
Advanced		
Defeaturing	B Defeaturing B Statistics	
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				
G:K-omega Fluent [2d File Mesh Define So	, dp, pbns, ske] [ANSYS Academic Teaching Advanced] Ive Adapt Surface Display Report Parallel Vie の意味のないないないないないないないないないないないないないないないないないないない	ew Help		
Meshing Mesh Generation Solution Setup General Materials Phases Cell Zone Conditions Boundary 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 Energy - Off Vecous - Standard H-e, Enhanced Wall Fn Radiation - Off Heat Exchanger - Off Diarete Phase - Off Diarete Phase - Off Solidification & Mething - Off Acoustics - Off Edit	Viscous Model Model Invisoid Laminar Spalart-Almaras (1 eqn) & komega (2 eqn) ransition (24 eqn) Komega (2 eqn) Transition K4J omega (2 eqn) Transition K5T (4 eqn) Scale-Adaptive Simulation (SAS) kepsilon Model Standard RNG Realizable Near-Wall Treatment User-Defined Wall Functions Scalable Wall Functions Benhanced Wall Functions Enhanced Wall Functions Financed Wall Functions Finan	Model Constants Cmu 0.09 C1-Epsion 1.44 C2-Epsion 1.92 TXE Prandt Number 1 User-Defined Functions Turbulent Viscosity none Prandt Number none TDR Prandt Number none Cancel Help Cancel Help	

k-ω model

File Mesh Define Solve							
The mean benne bone	Adapt Surface Display Report Parallel View	File Mesh Define Solve Adapt Surface Display Report Parallel View Help					
i 📖 i 📂 🕶 🖬 🕶 🎯	∭G ❹ € € ↗ !! € 冼 !!! - □ -						
Meshing Ma	lodels	Viscous Model	<u> </u>				
Mesh Generation Solution Setup General Meth Generation Meth Generation Meth Generation Meth Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Initialization Calculation Activities Run Calculation Calculation Activities Run Calculation Ed Results Graphics and Animations Plots Reports	idels Aultiphase - Off free gy - Off isoate Suprated K-omega isoate	Model Invisid Laminar Spalart-Alimaras (1 eqn) K-expsilon (2 eqn) Transition k-4-omega (3 eqn) Transition k-4-omega (3 eqn) Transition SST (4 eqn) Cale-Adaptive Simulation (SAS) K-omega Model Satandard Satandard Satandard Vision k-4-corrections Vision k-4-corrections Options Curvature Correction Ok	Model Constants Alpha*_Inf I Alpha*_Inf O.52 Beta*_inf O.52 Beta*_inf O.09 Beta_j I 0.072 User-Defined Functions Trubulent Viscosity none Prandt Numbers Truck Prandt Number SDR Prandt Number SDR Prandt Number Conce 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, p	bns, ske] [ANSYS Academic Teaching Advanced]			a general designed of
File Mesh Define So	lve Adapt Surface Display Report Parallel Vi	iew Help		
i 💼 i 📂 🕶 🖬 🕶 💿	@ \$₽0€//!®潗∥+□+			
Meshing	Materials	1: Mesh	•	
Mesh Generation	Materials	Create/Edit Materials		
Solution Setup	Fluid	Name		Order Materials by
General	air Solid	air	Material Type	Name
Models	aluminum	Chamical Formula	India	Chemical Formula
Phases			Fluent Fluid Materials	
Cell Zone Conditions			air	Huent Database
Boundary Conditions			Mixture	User-Defined Database
Dynamic Mesh			none	
Reference Values		Properties		
Solution		Density (kg/m3)	ent 💌 Edit	<u>^</u>
Solution Methods		1 100		
Solution Controls Monitors		1.100	5	
Solution Initialization		Viscosity (kg/m-s)	nt 💌 Edit	
Calculation Activities		1.920	160 DE	
Run Calculation		1.035	62-03	=
Results				
Plots				
Reports				
	Create/Edit Delete			
	Help			
		, · · · · · · · · · · · · · · · · · · ·		
			Change/Create Delete Clo	se 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	
B:Copy of K-epsilon Fli File Mesh Define So	uent [2d, dp, pbns, ske] [ANSYS Academic Teaching Ad ilve Adapt Surface Display Report Parallel Vi @방송 도구하여 관 개발을 가 제목 ㅠㅠㅠ	dvanced] iew Help	
Meshing Mesh Generation Solution Setup General Models Materials	Boundary Conditions Zone arfol interior-flud outlet	1: Mesh Velocity Inlet Zone Name Inlet	X
Phases Cell Zone Conditions <u>Boundary Conditions</u> <u>Mesh Interfaces</u> Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors		Momentum Thermal Radiation Species DPM Multiphase LDS Velocity Specification Method Components	
Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Edt Coper Profiles Rearneters Operating Conditions Decide Mesh	Turbulence K and Epsion Turbulent Kinetic Energy (m2/s2) 0.08 constant Turbulent Dissipation Rate (m2/s3) 7.4 constant	
	Help	OK Cancel Help	,

	k-w model					
G:K-omega Fluent [2d,	dp, pbns, skw] [ANSYS Academic Teaching Advance	ed]			Summer of the	
File Mesh Define Sol	lve Adapt Surface Display Report Parallel	Viev	v Help			
1 🖬 1 📸 ד 🔚 ד 🚳	@∥\$\$₽€€≯∥®次⊪-□-					
Meshing	Boundary Conditions	Â	Velocity Inlet			X
Mesh Generation Solution Setup General Materials Phases Cell Zone Conditions Baundary Conditions Baundary Conditions Baundary Conditions Baundary Conditions Baundary Conditions Baundary Conditions Dynamic Mesh Mesh Inter Faces Dynamic Mesh Solution Methods Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots	Zone prifol interio-fuid outer putface_body Phase Type ID Edit Copy Profiles	Е	Zone Name Inlet Momentum Thermai Radiation Specie Velocity Specification Method Reference Frame Supersonic/Initial Gauge Pressure (pascal) X-Velocity (m/s) Y-Velocity (m/s) Turbulence Specification Method [Specification Method [Specification Method [Specific Dissipation Rate (1/s)]	s DPM Multip Components Absolute 0 15 0 x and Omega 0.08 9	hase UDS Constant Constant Constant Constant Constant	
Reports	Parameters Operating Conditions Display Mesh Periodic Conditions		OK	Cancel	Help	
	Lista	Ŧ	< III			F.

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

C:K-epsilon Fluent [2d	CK-epsilon Fluent [2d, dp, pbns, ske] [ANSYS Academic Teaching Advanced]				
File Mesh Define Sol	lve Adapt Surface Display Report Parallel Vi	ew Help			
i 💼 i 💕 🕶 🛃 🕶 🞯	@ \$₽€€/ ® 洗 開 - 🗆 -				
Meshing	Boundary Conditions	1: Mesh 👻			
Mesh Generation	Zone	Pressure Outlet			
Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions 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 Priet Prist P	Zone Name outlet Momentum Themail Radiation Species DPM Multiphase UDS Gauge Pressure (pascal) 0 constant • Backflow Direction Specification Torget Mass Flow Rate Turbulence Specification Method Intensity and Viscosity Ratio Backflow Turbulent Intensity (%) 3.25 Backflow Turbulent Intensity (%) 0.0035 OK Cancel			
	Periodic Conditions	4 III P			

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	Reference Values Area (m2) 0.3048 Density (kg/m3) 1.1885 Depth (m) 1 Enthaloy (j/kg) 0
Solution Solution Methods Solution Controls Monitors	Length (m) 1 Pressure (pascal) 0
Solution Initialization Calculation Activities Run Calculation	Temperature (k) 297.15
Results	Velocity (m/s) 15
Graphics and Animations Plots Reports	Viscosity (kg/m-s)
	Reference Zone
	Help

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	SIMPLE	
Materials	Spatial Discretization	
Phases	Gradient	
Cell Zone Conditions Boundary Conditions	Green-Gauss Cell Based	
Mesh Interfaces	Pressure	
Dynamic Mesh	Standard 🔹	
Solution	Momentum	
Solution Methods	Second Order Upwind	
Solution Controls	Turbulent Kinetic Energy	
Monitors	Second Order Upwind	
Calculation Activities	Second Order Upwind	
Run Calculation	Trapping Formulation	
Results		Mesh Aug 04, 2013
Graphics and Animations	Non-Iterative Time Advancement	ANSYS Fluent 14.5 (2d, dp, pbns, lam)
Reports	Frozen Flux Formulation	surface_body ^
	High Order Term Relayation	interior-fluidSetting fluid (mixtur
	Options	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	🗖 Chliccid Fluence (ka) (ANISYS Academic Sandhine Advanced)					
File Mesh Define So	File Mesh Define Solve Adapt Surface Display Report Parallel View Help					
Meshing	Solution Controls	1: Mesh				
Mesh Generation	Under-Relaxation Factors					
Solution Setup	Body Forces					
General	1					
Materials	Momentum					
Phases	0.5					
Cell Zone Conditions						
Mesh Interfaces	Turbulent Kinetic Energy					
Dynamic Mesh	0.8					
Reference Values	Turbulent Dissipation Rate					
Solution Methods	0.8					
Solution Controls	Turbulent Viscosity					
Monitors	1					
Calculation Activities	· · · · · · · · · · · · · · · · · · ·					
Run Calculation	Default					
Results	Equations Limits Advanced	Mesh Aug 04, 2013				
Graphics and Animations		ANSYS Fluent 14.5 (2d, dp, pbns, lam)				
Reports		surface_body				
		interior-fluidSetting fluid (mixtur				
		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 airfoil to 8.				
		Done.				
		Setting interior-fluid (mixture) Done.				
		Setting surface_body (mixture) Done.				
		Setting inlet (mixture) Done.				
		secting outlet (wixture) Dolle.				
		< III > 2				

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	Statistic - Off	V Plot
Materials		Window
Phases		1 Curves Axes y-velocity V 1e-05 =
Cell Zone Conditions		Iterations to Plot k V Ie-05
Mesh Interfaces	Create Cafere Marihan	1000
Dynamic Mesh	Surface Monitors	
Reference Values		Residual Values Convergence Criterion
Solution Methods		Iterations to Store
Solution Controls		
Monitors		Scale
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, puns, ram)
Reports		surface_body
	<u> </u>	interior-fluidSetting fluid (mixtur
	Create Edit Delete	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 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
i 💼 i 📂 🕶 🛃 🔻 🞯	Ø \$\$ € € €
Meshing Mesh Generation	Run Calculation
Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Number of Iterations 10000 Profile Update Interval 1 Data File Quantities Reporting Interval 1 Construction of the second sec
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Calculate

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
------------------------------	--------------------	---------------	----------

C:Fluent Fluent [2d, d File Mesh Define Sc	Ip, pbns, ske] [ANSYS Academic Teaching Advanced]	23
Meshing Mesh Generation Solution Setup	Image: Solution of Solution Solutions 1: Scaled Residuals Residuals, Statistic and Force Monitors Image: Solution Solutions Residuals solutions Here	ANSYS IBAS Access
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls	Statistic - Off	
Monitor: Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Equitable Equitable Image: Convergence Orterion Image: Convergence Orterion	15, 2013 ns, ske)
	1000	

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	Coloring © Color © Gray Scale © Monochrome	File Type Raster Vector	Resolution Width 960 Height 720	
TIFF PNG VRML Window Dump	Options Image: Contract of the second seco	entation und	low Dump Command ort -window %w	
Save	Apply Pr	review Clos	e 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*.

	t [2d dn nhns ska]	I MNSVS Academic Teaching Advanced	di kana kana						
El Ciriuent ruemi (zg. op. pors, seg [arvis 5 Adaemic resching Advances] El Market Defen Constantiation (Section 2014) El Market Defen Constantiation (Section 2014) El Market Defen Constantiation (Section 2014)									
The west beline solve adopt surface bisplay report Parallel view nep									
Meshing	Plots		1: Pressure Coefficient 👻						
Mesh Generation	Plots		experimental						ANSYS
Solution Setup	XY Plot			2000+00					
General	Histogram			1.00e+00	· · · ·				
Models	Profiles:					•••• • • • • •			
Phases	Interpola	ata - Unavailable ated Data		0.000+00					
Cell Zone Condit	ions FFT			-1.000+00					
Mesh Interfaces	0013		Pressure	-2.00++00	· · · · · ·	•••			
Dynamic Mesh			Coefficient						
Solution	s			-3.00+00					
Solution Method	s	-		-4.008+00					
Solution Controls	5	Solution XY Plot							
Monitors Solution Initializa	tion	Options Plot Direction	Y Axis Function						
Calculation Activ	ities	Node Values X 1	Pressure	-	0 0.05	0.1 0.15	5 0.2	0.25 0.3	0.36
Run Calculation	Set Up	Position on X Axis	Pressure Coefficient	•		Positio	n (m)		
Graphics and An	imations	Write to File	X Axis Function						
Plots	Help	Order Points 2 0	Direction Vector	-					
Reports		File Data 🔳 🚍	Surfaces				ANG	VS Eluent 14 5 (2d. d	Nov 15, 2013
		pressure coefficient	inlet				200	51011dent 14.5 (2d, d	p, pbila, ake)
			interior-fluid		.3736e-07	4.38888-06	1.0305e-05	0:05:44 7513	^
			surface_body		3697e-07	4.3768e-06	1.0271e-05	0:03:40 7511	
z					.3688e-07	4.3707e-06	1.0260e-05	0:02:56 7510	
		Load File			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	
					.3620e-07	4.3412e-06	1.0168e-05	0:11:13 7505	
		Plot Axes	Curves Close Hel	p	.3605e-07	4.3356e-06	1.0148e-05	0:08:58 7504	
		<u> </u>	iter continuity	x-velocity	y-velocity	4.32908-00 k	epsilon	time/iter	
			2498 2.5858e-06	1.9475e-07	1.3573e-07	4.3237e-06	1.0124e-05	0:05:44 7502	
			2499 2.56778-06	1.9430e-07	1.3548e-07	4.3179e-06 4.3124e-06	1.0081e-05	0:03:40 7500	
1			2501 2.6320e-06	1.9406e-07	1.3525e-07	4.3065e-06	1.0068e-05	0:02:56 7499	
			4						

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.

Ella Mark Def	ap, pons, skej (ANSYS Academ	ic reaching Advanced		A CONTRACTOR OF A CONTRACTOR OFTA CONT	
File Mesh Define	Bolve Adapt Surface Displ	ay Report Parallel View	w Help		
	। 🛛 🔁 🐨 🔍 प्र 🦯				
Meshing	Graphics and Animatio	ns	1: Velocity Vectors Colored B 👻		
Mesh Generation	Graphics		3/34e+01		ANSYS Internet Andreas Anternet Anternet Anterne
Solution Setup			3,138+01		
General Contours			284+01		
Models	Pathlines		2,658+01		
Phases	Particle Tracks		251e+01		
Cell Zone Conditions			2.17e+01		
Mesh Interfaces	Set I In		2016+01		
Dynamic Mesh	loccopiii		1.5%e+U1	and the second s	
Reference Values	Animations	-	1.510+01		
Solution	Sweep Surface	Vectors	and the second se		
Solution Methods Solution Controls	Scene Animation Solution Animation Playback	Options	Vectors of	AM STATISTICS	
Monitors		Global Range	Velocity		
Solution Initialization		Auto Range	Color by		
Run Calculation		Auto Scale	Velocity		
Results	Set Up	Draw Mesh	Velocity Magnitude		
Graphics and Animations		Style	Min (m/s) Max (m/s)		
Plots	Ontions Scene	arrow	 0.01478644 33.4482 		Nov 16, 2012
incepores	Lights Colorman	Scale Skip	Surfaces		ANSYS Fluent 14.5 (2d, dp, pbns, ske)
	Colornap	200 10	airfoil		05 0.05 11 3540
			inlet	.37360-07 4.38880-06 1.03050	-05 0:05:44 7513
	Help	vector Options	outlet	.3697e-07 4.3768e-06 1.0271e	-05 0:03:40 7511
		Custom Vectors	surface_body	.3688e-07 4.3707e-06 1.0260e	
				.3662e-07 4.3591e-06 1.0216e	-05 0:21:54 7508
		Surface Name Pattern	New Surface =	.3639e-07 4.3530e-06 1.0202e	-05 0:17:31 7507
		Match	INEW Surface	.3630e-07 4.34/1e-06 1.0192e 3620e-07 4.3412e-06 1.0168e	·05 0:14:01 /506 -05 0:11:13 7505
			Surface Types	.3605e-07 4.3356e-06 1.0148e	-05 0:08:58 7504
			dip-surf	.3581e-07 4.3296e-06 1.0134e	-05 0:07:10 7503
			exhaust-fan fan	3573e-07 4.3237e-06 1.0124e	-05 0:05:44 7502
				.3563e-07 4.3179e-06 1.0101e	-05 0:04:35 7501
				.3548e-07 4.3124e-06 1.0081e 3525e-07 4.3065e-06 1.0081e	-05 0:03:40 7500 -05 0:02:56 7400
		Display	Compute Close Help	25140 07 4.30050 00 1.00080	07 0.02.JU (477 0F 0.07.04 7500

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*.

File Mesh Define Solve Adapt Surface Display Report Parallel View Help						
Meshing Graphics and Animations 1: Contours of Stream Functi -	ANCVC					
Mesh Generation Graphics						
Solution Setup Mesh						
General Contours						
Moteriale Pathines						
Phases Particle Tracks						
Cel Zone Conditions						
Boundary Conditions						
Dynam Mesh						
Reference Values						
Solution Sweep Surface						
Solution Methods Scene Animation						
Monitors						
Solution Initialization Options Contours of						
Lacuation Activities International Inflied Velocity						
Results Set Linu Node Values Stream Function						
Grephics and Animetions						
Plots Contana Same 103 107.5						
ANSYS Fluent 14.5 (2d, d	o, pbns, ske)					
niet 1.3/306-0/ 4.38886-00 1.0305-0 9 01:05:44 /513	^					
Help Levels Setup interior-Huid 1.3697e-07 4.3768e-06 1.02571e-05 0.03340 7512						
1.3688e-67 4.3767e-66 1.6266e-65 6:62:56 7518						
1.306/8E-07 4.3304/E-00 1.02/30E-05 012/7123 /509						
Surface Name Pattern New Surface ▼ 1.3639e-07 4.3530e-06 1.0202e-05 0:17:31 7507						
Match Surface Types II = 1.36380=-07 4.34710=-06 1.0192e-05 0:14:01 7506						
axis 1.36059-07 4.39126-00 1.01149-05 0.08158 7504						
un-surfan 1.3581e-07 4.3296e-06 1.0134e-05 0:07:10 7503						
fan						
1.3563e-07 4.3179e-06 1.0101e-05 0.0144 55 7501						
Display Compute Close Help 1.35486-87 4.31246-86 1.808316-85 8:83:40 7568						
1.3525e-UV 4.3056e-U0 1.0006e-U5 U12255 /499	-					
۲۰۰۰ ۲۰۰۰ ۲۰۰۰ ۲۰۰۰ ۲۰۰۰ ۲۰۰۰ ۲۰۰۰ ۲۰۰						

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*.

1050107 1050107 1050107	Force Reports		
1:856782	Options Direction Vector Wall Zones		
Contours of Stream Function (kg/s)	Orces Moments Center of Pressure Y		Aug 05, 2013 ANSYS Fluent 14.5 (2d, dp, pbns, ske)
2234 2.1639e-06 9.4052e-08 7.7806e-08 2235 2.1624e-06 9.4014e-08 7.7784e-08 2236 2.1608e-06 9.3055e-08 7.7784e-08 2237 2.1592e-06 9.3917e-08 7.7784e-08 2238 2.1577e-06 9.3917e-08 7.7796e-08 2239 2.1572e-06 9.3826e-08 7.7796e-08 2239 2.1572e-06 9.3826e-08 7.7793e-08 2240 2.1542e-06 9.3771e-08 7.7669e-08	Z 0 Wal Name Pattern Match Save Output Parameter		
Forces	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		
Zone Pressure Vi	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	
< m			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.