

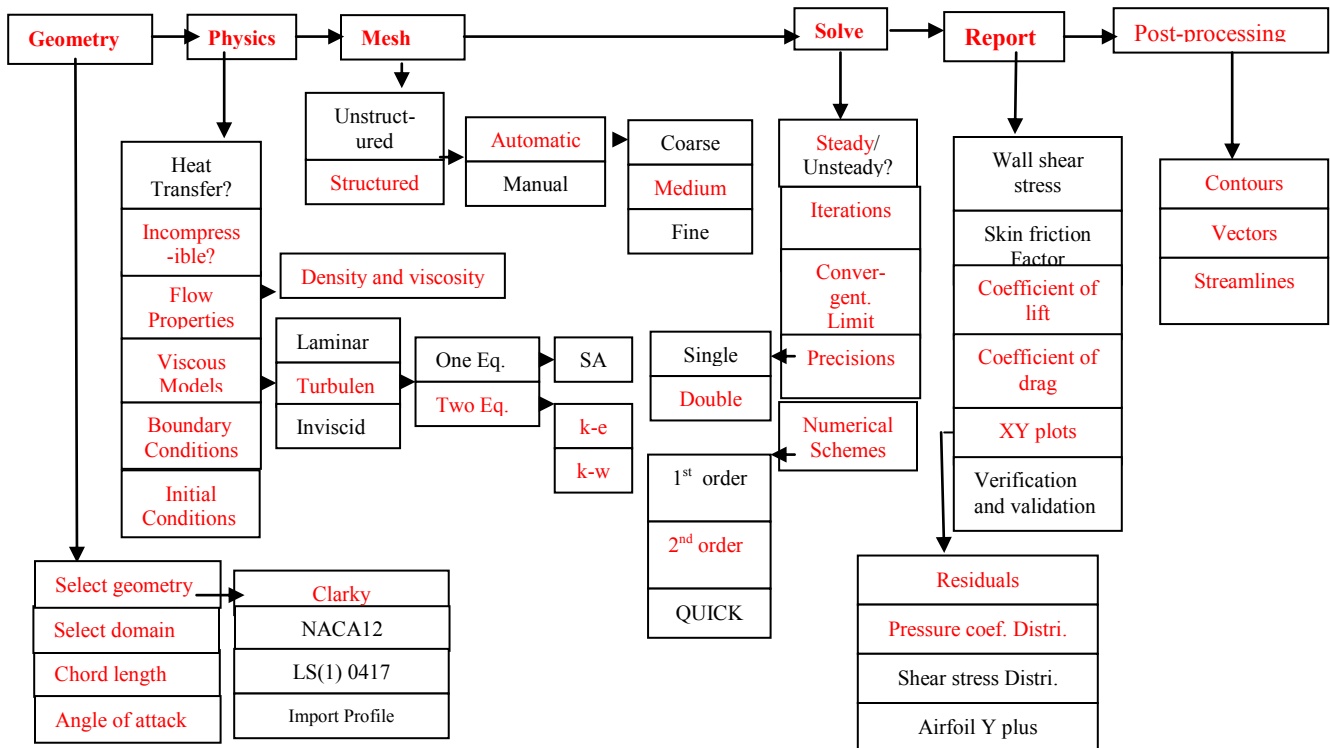
# Parametric Studies of Turbulent Flow around an Airfoil

57:020 Mechanics of Fluids and Transport Processes  
CFD LAB 2

By Tao Xing and Fred Stern  
IHR-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 FlowLab 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.

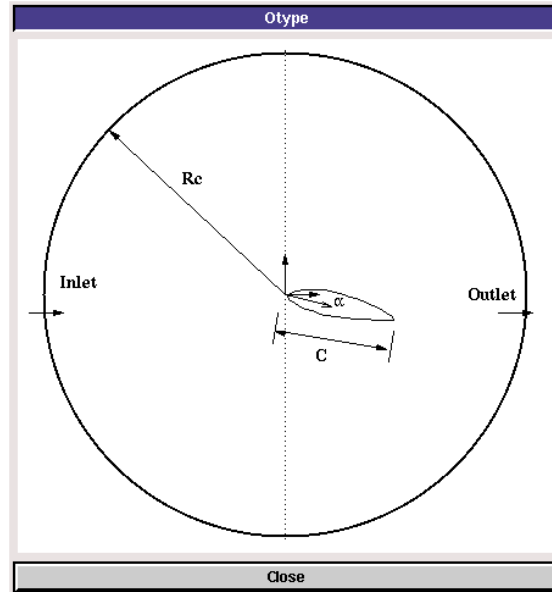


Flow Chart for ISTUE Teaching Module for Airfoil flow (red color illustrates the options you will use in CFD Lab 2)

## 2. Simulation Design

In EFD Lab 3, you have conducted experimental study for turbulent airfoil flow around a ClarkY 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.

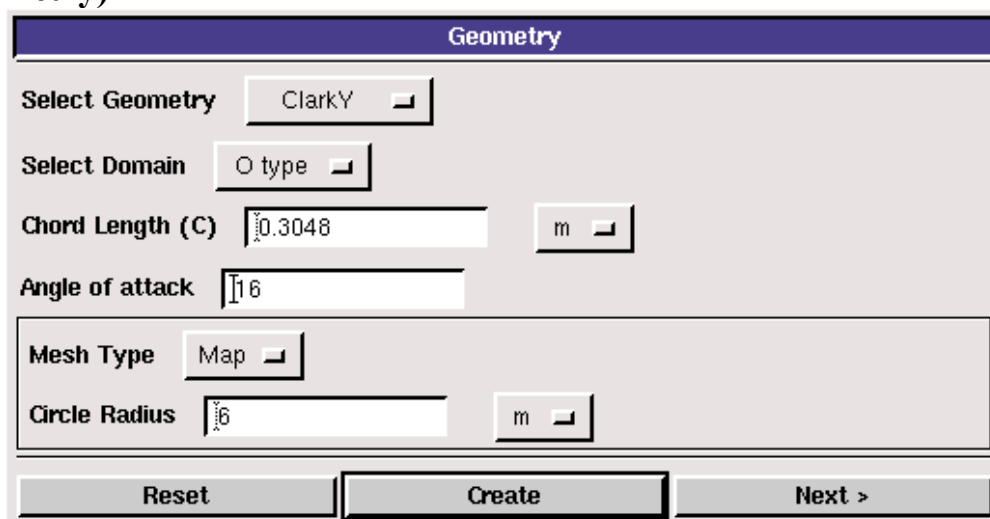
The problem to be solved is the turbulent flow around the Clarky airfoil with angle of attack ( $\alpha$ )



In the figure above,  $C$  is the chord length of the airfoil,  $\alpha$  is the angle of attack, and  $R_c$  is the radius of the "O" domain.

## 3. CFD Process

### Step 1: (Geometry)



The screenshot shows the "Geometry" configuration window. It includes the following fields and controls:

- Select Geometry:** ClarkY
- Select Domain:** O type
- Chord Length (C):** 0.3048 m
- Angle of attack:** 16
- Mesh Type:** Map
- Circle Radius:** 6 m

At the bottom, there are three buttons: "Reset", "Create", and "Next >".

Choose “ClarkY” airfoil and “O type” domain. Input the following conditions:

1. **Select Geometry** (Clarky)
2. **Select domain** (O type)
3. **Chord length** (0.3048 m)
4. **Angle of attack** (**16 degrees**)
5. **Mesh type** (Map)
6. **Circle Radius  $R_c$**  (6m)

**You need to push “create” button after you enter the values for the changes to take effect.**

There are two angles of attack available in EFD Lab3 for all the groups, i.e., **0 and 16 degrees**. In CFD PreLab2, you have conducted the simulation for 0 degree. In CFD Lab2, to investigate the “**effect of angle attack**”(exercise 1), you will simulate angle of attack **16 degrees** and compare with CFD Prelab2 results. To investigate the “**effect of turbulence models**” (exercise 2), you will use the same angle of attack, i.e., **16 degree**, and compare with results from exercise 1.

## Step 2: (Physics)

The screenshot shows the 'Physics' settings panel. The 'Heat Transfer' section has a checkbox that is currently unchecked. The 'Incompressible' section has a dropdown menu showing 'Incompressible'. The 'Flow Properties', 'Viscous Models', 'Boundary Condition', and 'Initial Condition' sections are currently empty. At the bottom, the 'Re #' field contains the value '294968' and there is a 'Compute' button. Below these are three navigation buttons: '<Back', 'Reset', and 'Next>'.

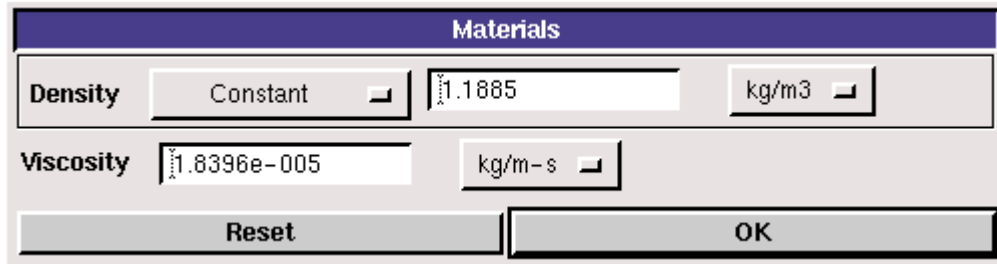
### (1). **With or without Heat Transfer?**

Since we are not dealing with the thermal aspects of the flow, like heat transfer, etc., switch the <<**Heat Transfer**>> button OFF, which is the default setup.

### (2). **Incompressible**

Choose “Incompressible”, which is the default setup.

### (3). **Flow Properties**



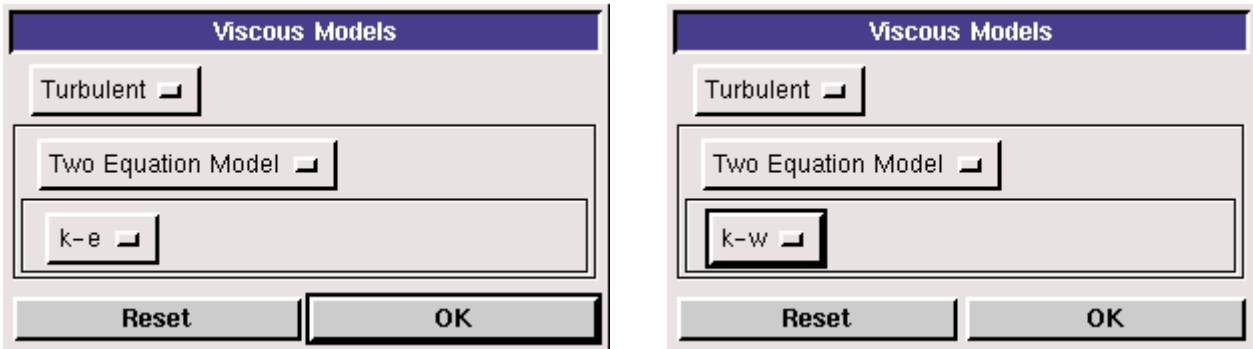
Use the air properties at the operating temperature of your EFD Lab3 and click <<OK>>. You can use the following website to calculate air properties if you know the temperature:

<http://www.mhfl.uwaterloo.ca/old/onlinetools/airprop/airprop.html>

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

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

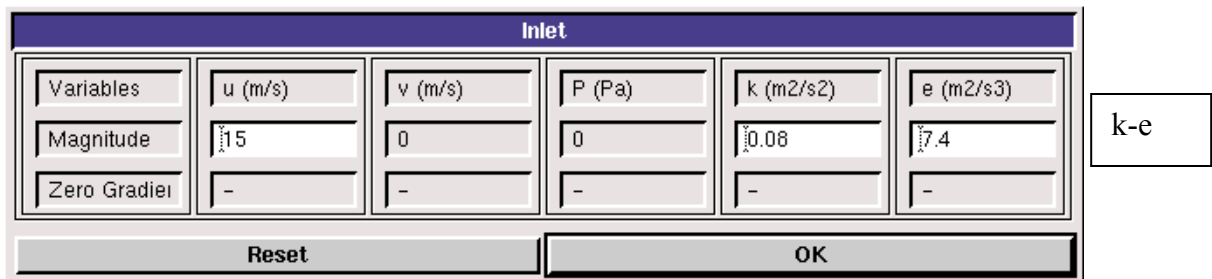
**(4). Viscous Model**



For exercise 1 (effect of angle of attack), use “k-e” model. For exercise 2 (effect of turbulence models), using “k-w” model.

**(5). Boundary Conditions**

At “Inlet”, we use constant pressure. Inlet velocity is 15 m/s. Use default values for “k”, “epsilon” and “omega(w).”



Inlet					
Variables	u (m/s)	v (m/s)	P (Pa)	k (m2/s2)	w (1/s)
Magnitude	15	0	0	0.08	9
Zero Gradient	-	-	-	-	-
Reset			OK		

k-w

At “**Outlet**”, FlowLab uses magnitude for pressure and zero gradients for axial and radial velocities. Input “0” for gauge pressure and click <<OK>>.

Outlet					
Variables	u (m/s)	v (m/s)	P (Pa)	k (m2/s2)	e (m2/s3)
Magnitude	-	-	0	-	-
Zero Gradient	Y	Y	-	Y	Y
Reset			OK		

k-e

Outlet					
Variables	u (m/s)	v (m/s)	P (Pa)	k (m2/s2)	w (1/s)
Magnitude	-	-	0	-	-
Zero Gradient	Y	Y	-	Y	Y
Reset			OK		

k-w

At “**Airfoil**”, if flow is turbulent, FlowLab uses no-slip boundary conditions for velocities and zero-pressure gradient. Turbulent quantities k and e are also specified to be zero, except for omega (w), on which zero-pressure gradient applied. Read all the values and click <<OK>>.

Airfoil					
Variables	u (m/s)	v (m/s)	P (Pa)	k (m2/s2)	e (m2/s3)
Magnitude	0	0	-	0	0
Zero Gradient	-	-	Y	-	-
Reset			OK		

k-e

Airfoil					
Variables	u (m/s)	v (m/s)	P (Pa)	k (m2/s2)	w (1/s)
Magnitude	0	0	-	0	-
Zero Gradient	-	-	Y	-	Y
Reset			OK		

k-w

## (6). Initial Conditions

Use the default setup for initial conditions.

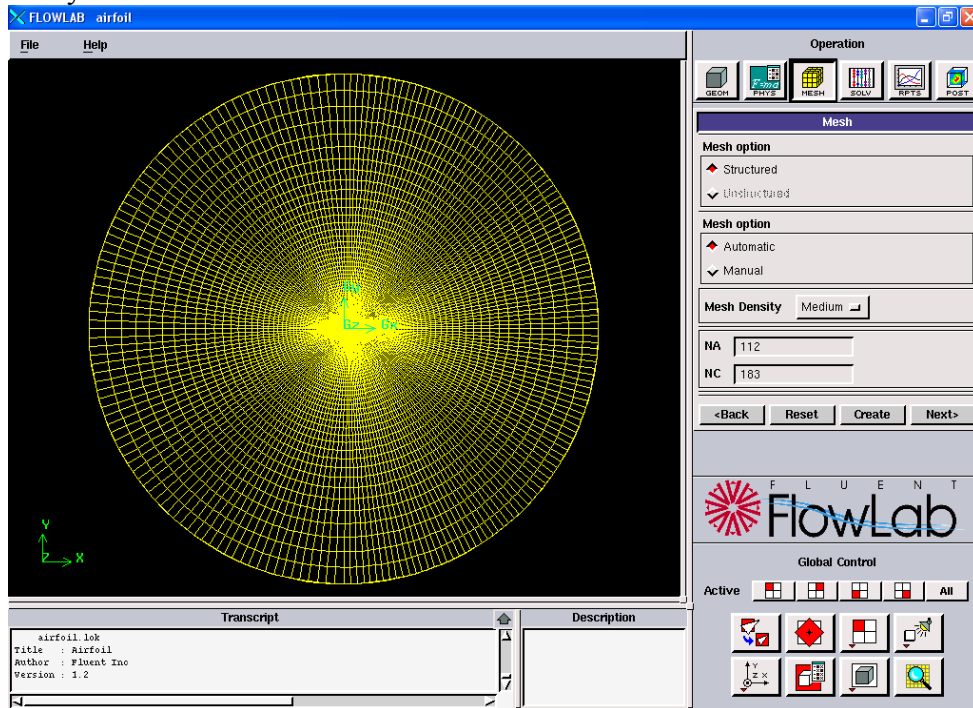
The image shows two screenshots of the 'Initial Condition' dialog box. The top screenshot is for the k-e model, with the 'e (m2/s3)' variable set to 7.4. The bottom screenshot is for the k-w model, with the 'w (1/s)' variable set to 9. Both screenshots show the same values for 'P (Pa)' (0), 'u (m/s)' (15), 'v (m/s)' (0), and 'k (m2/s2)' (0.08). The 'Reset' and 'OK' buttons are visible at the bottom of each dialog.

After specifying all the above parameters, click <<Compute>> button and FlowLab will automatically calculate the Reynolds number based on the inlet velocity, airfoil chord length, and flow properties. Click <<Next>>, it takes you to the next step, “Mesh”.

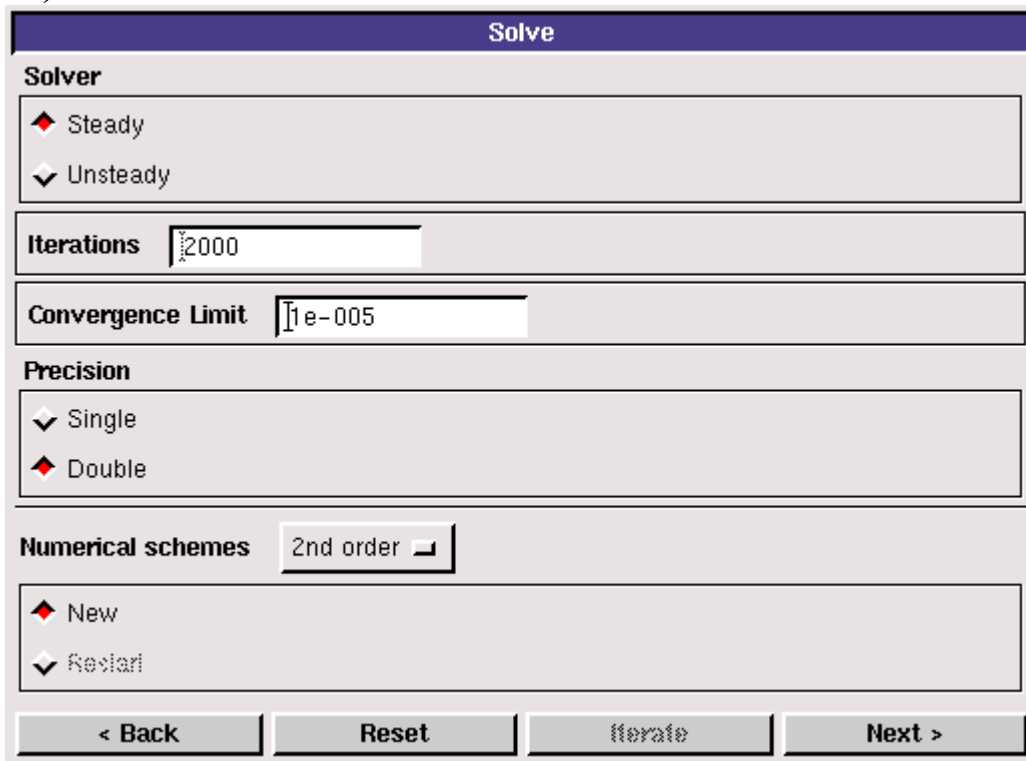
### Step 3: (Mesh)

The image shows the 'Mesh' dialog box. Under the first 'Mesh option' section, 'Structured' is selected. Under the second 'Mesh option' section, 'Automatic' is selected. The 'Mesh Density' is set to 'Medium'. The 'NA' value is 112 and the 'NC' value is 183. The 'Create' button is highlighted.

For CFD Lab 2, use “**Automatic**” meshing and “**Medium**” mesh. After clicking <<Create>>, mesh information will be displayed. NA and NC are the numbers of grid points on the airfoil surface and radial direction, respectively.



#### Step 4: (Solve)

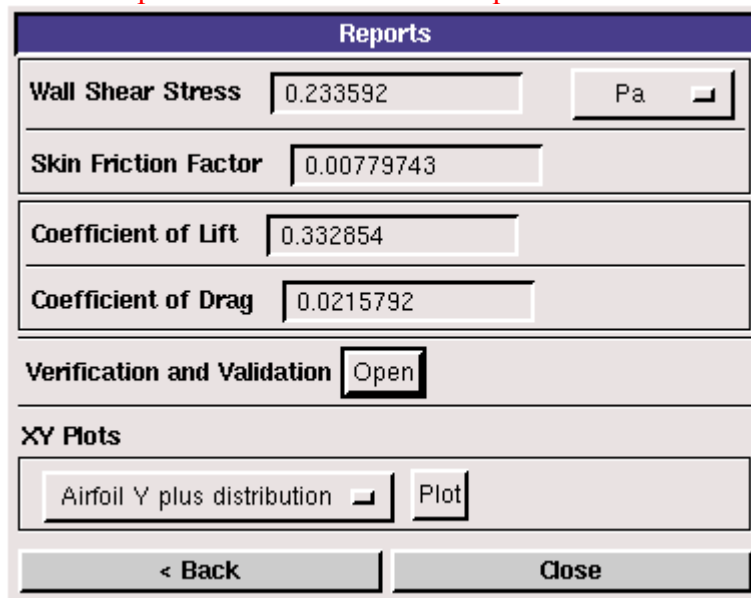


Specify the iteration number and convergence limit to be **2000** and  $10^{-5}$ , respectively. Choose “**Double Precision**”, “**2<sup>nd</sup> order**” for numerical schemes, and “**New**” calculation. Click <<**Iterate**>>, FlowLab will fire the XY plot for residuals that is dynamically updated during the calculation. Whenever you see the window, “**Solution Converged**”, click <<**OK**>>.

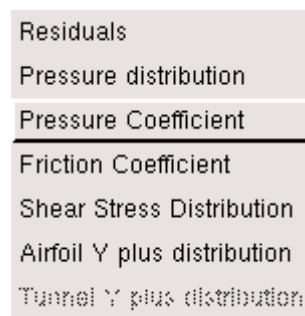


### Step 5: (Reports)

“**Reports**” first provides you the information on “wall shear stress”, “skin friction factor”, “coefficient of lift”, and “coefficient of drag”. XY plot provide plots for “residuals”, “pressure distribution”, “pressure coefficient”, “friction coefficient”, “shear stress distribution”, and “Airfoil Y Plus distribution”. **In this Lab, only XY plots for “residuals” and “pressure coefficient” are required.**

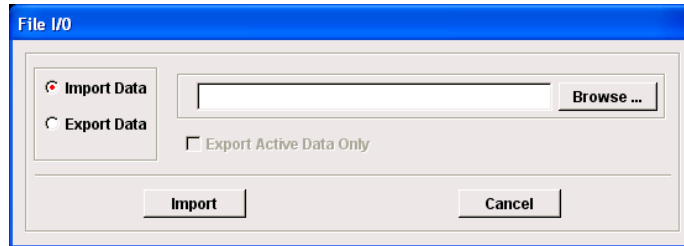


“XY Plots” provides the following options:





To import the EFD data on top of the above CFD results, just click <<File>> button and use the browse button to locate the file you want and click <<Import>>. Details have been described in previous CFD PreLab 1 for laminar pipe flow.

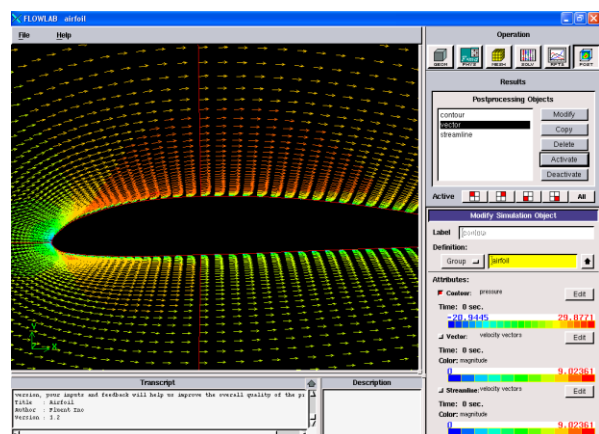
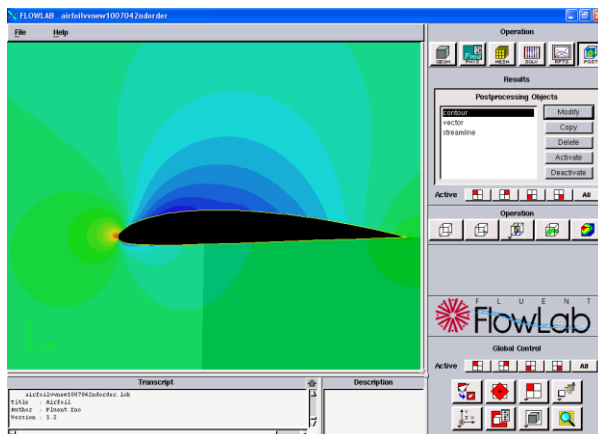


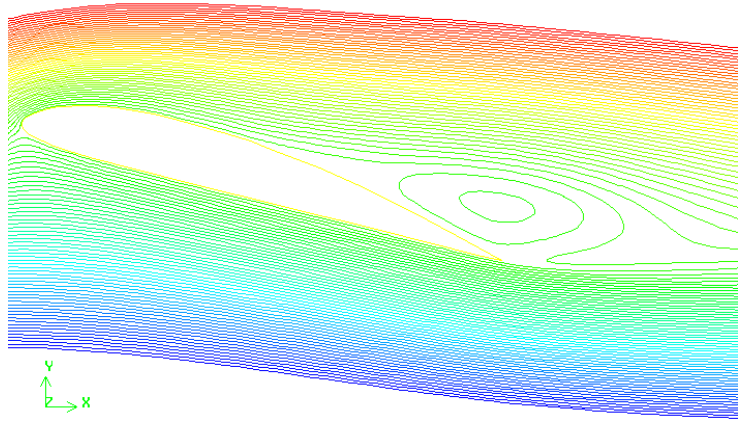
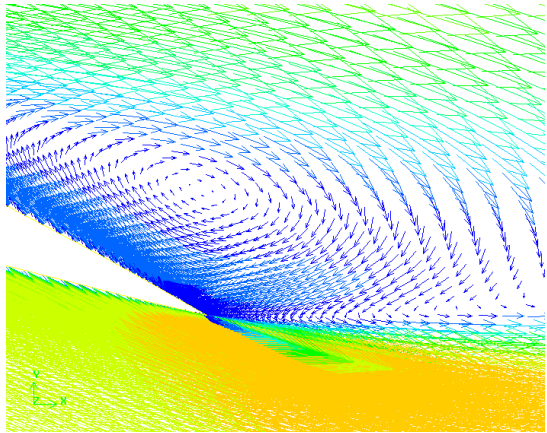
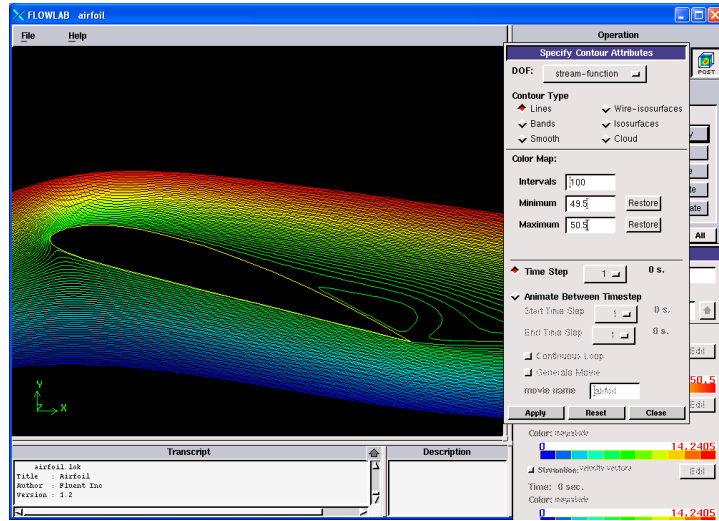
In this Lab:

1. Since you already created your own EFD Lab 3 data in CFD PreLab 2, it is suggested that you use the same computer as you used in CFD PreLab2 unless if you have saved your work in your network drive, which is recommended.
2. For exercise 1, you will compare the pressure coefficient distribution with EFD for an angle of attack that may be different from your own data in EFD Lab 3. **Benchmark data of the pressure coefficients** for different angles of attack can be downloaded from class website under “CFD Lab2”. The angle of attack can be told from the name of the file, such as “Pressure-coef-attack06.xy” is the EFD data of pressure coefficient for angle of attack equals 6 degrees.
3. **Benchmark data for lift and drag coefficients** also can be found from the same place, or: <http://www.engineering.uiowa.edu/~fluids/Lab-documents/CFD/CFD%20Lab2/Lift.xls>

## Step 6: (Post-processing)

Use the “contour”, “vector” buttons to show pressure contours and velocity vectors. All the details on how to plot those figures have been explained in details in previous lab instructions. The following shows some sample results only, your simulations in this lab will all be angle of attack 16.





## 4. 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 “CFDLab2-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 **physics** (Flow properties, Reynolds number, inlet velocity), **EXCEPT use angle of attack 16 Degrees regardless of AOA in you EFD Lab 3**. Use “**O**” mesh with **domain size 6 meters**, **k-e model**, **automatic medium mesh**, **2<sup>nd</sup> 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-w**” 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-e**” 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 “CFDlab2-Template” where you will type answers.

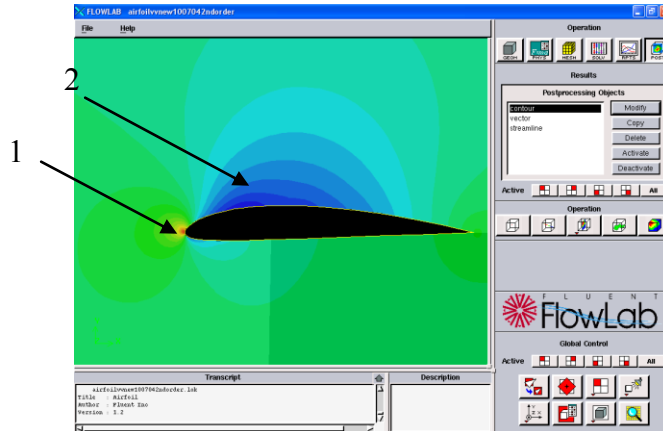
##### 3.1. For Exercise 1 (effect of angle of attack):

- (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?

**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.**