

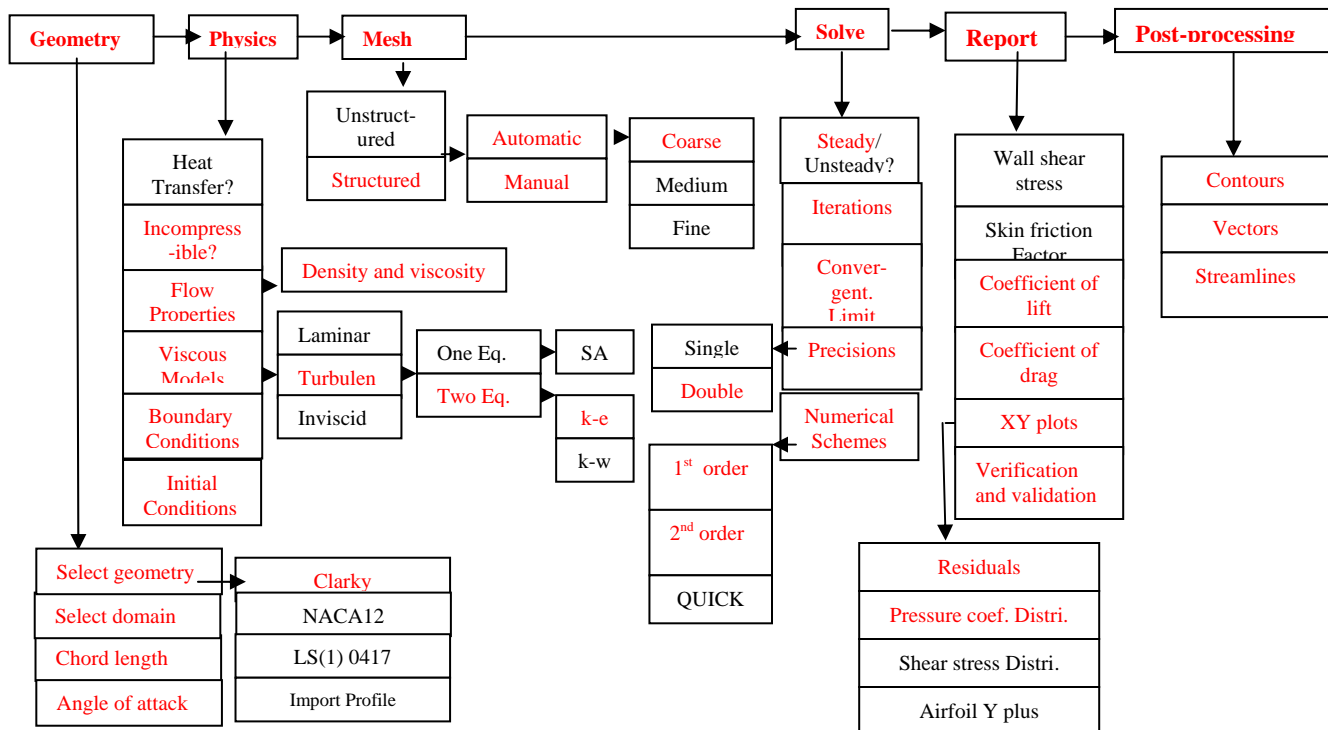
Verification and Validation of Turbulent Flow around a Clark-Y Airfoil

58:160 Intermediate Mechanics of Fluids CFD LAB 2

By Tao Xing and Fred Stern
 IIHR-Hydrosience & Engineering
 The University of Iowa
 C. Maxwell Stanley Hydraulics Laboratory
 Iowa City, IA 52242-1585

1. Purpose

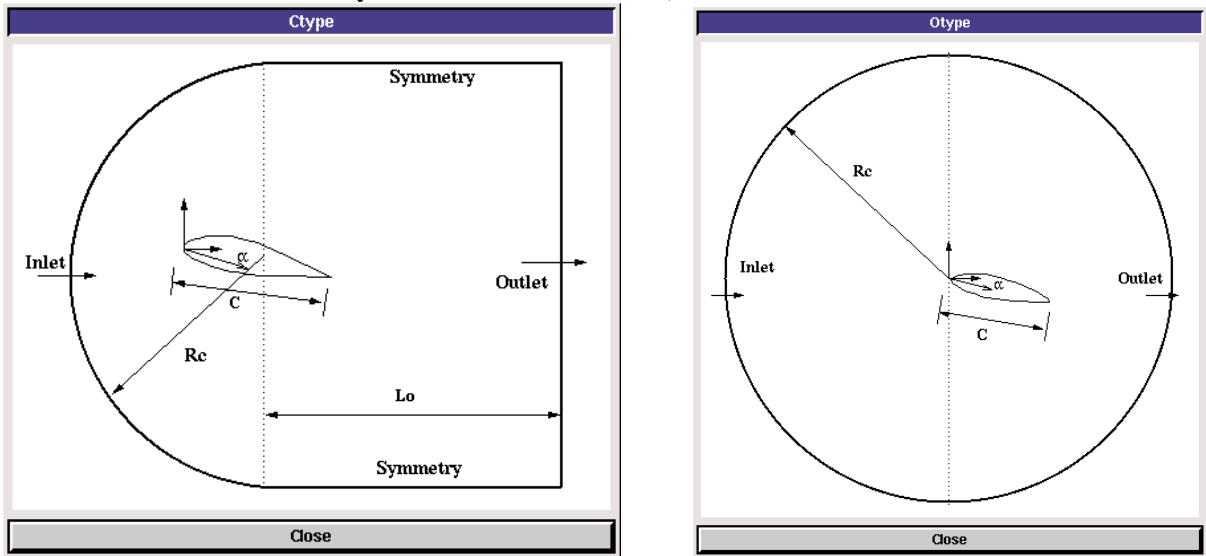
The Purpose of CFD Lab 2 is to simulate **turbulent** airfoil flows following “CFD process” by an interactive step-by-step approach and conduct verifications using CFD Educational Interface (FlowLab 1.2). Students will have “hands-on” experiences using FlowLab to conduct **verification and validation for lift coefficient and pressure coefficient distributions**, including **effect of numerical scheme**. Students will manually generate the “O” type and “C” type meshes and investigate the **effect of domain size** and **effect of angle of attack** on simulation results. Students will analyze the differences between CFD and EFD, analyze possible source of errors, and present results in the CFD 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

The problem to be solved is that of turbulent flows around a Clark-Y airfoil. Reynolds number is 143,000 based on the inlet velocity and airfoil chord length. The following figures show the illustrations for “C” type and “O” type meshes. (Note: the figures are not in the exact scale as the true size of the domain and airfoil, as you will see in FlowLab).



In CFD Lab2, **Boundary conditions for “C” type of meshes will be “inlet”, “outlet”, “symmetry”, and “airfoil”,** as described later. **Boundary conditions for “O” type of meshes will be “inlet”, “outlet”, and “airfoil”.** Uniform flow was specified at inlet. For outlet, zero gradients are fixed for all velocities and pressure is constant. No-slip boundary condition will be used on the “airfoil”. Symmetric boundary condition will be applied on the “symmetry”.

All EFD data for turbulent airfoil flow in this Lab can be downloaded from class website.

3. CFD Process

Step 1: (Geometry)

Geometry		
Select Geometry	ClarkY	
Select Domain	O type	
Chord Length (C)	0.3048 m	
Angle of attack	0	
Mesh Type	Map	
Circle Radius	5 m	
Reset	Create	Next >

Geometry		
Select Geometry	ClarkY	
Select Domain	C type	
Chord Length (C)	0.3048 m	
Down stream length (Lo)	12 m	
Angle of attack	0	
Radius (Rc)	5 m	
Reset	Create	Next >

1. **Select Geometry:** Clark-Y
2. **Chord length** (0.3048 m)
3. **Angle of attack** (0 or 6, refer to exercises for details)
4. **Circle Radius** (5 meters except for exercise 1, where effect of domain size is studied)
5. **Radius (Rc)** for “C” mesh (5 meters)
6. **Down stream length (Lo) for “C” mesh** (12 meters).

Click <<Create>>, after you see the airfoil geometry created in the graphic window, click <<Next>>.

Step 2: (Physics)

(1). With or without Heat Transfer?

No thermal effects are considered in this lab, switch the <<Heat Transfer >> button OFF, which is the default setup.

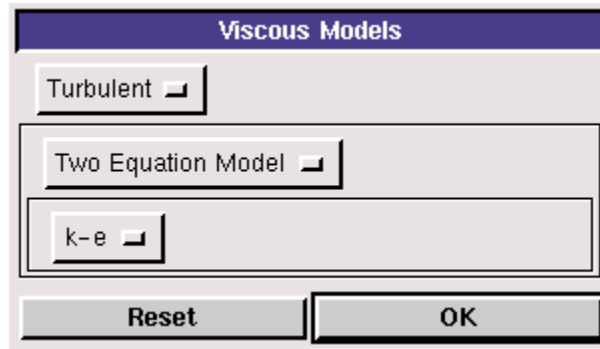
(2). Incompressible or compressible

Choose “Incompressible”, which is the default setup.

(3). Flow Properties

Use the values shown in the above figure. Input the values and click <<OK>>.

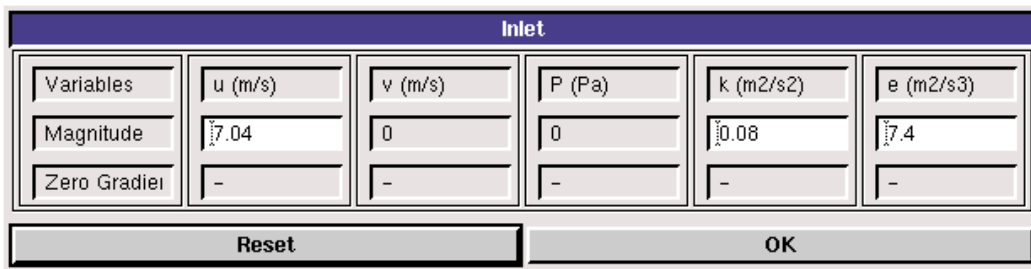
(4). Viscous Model



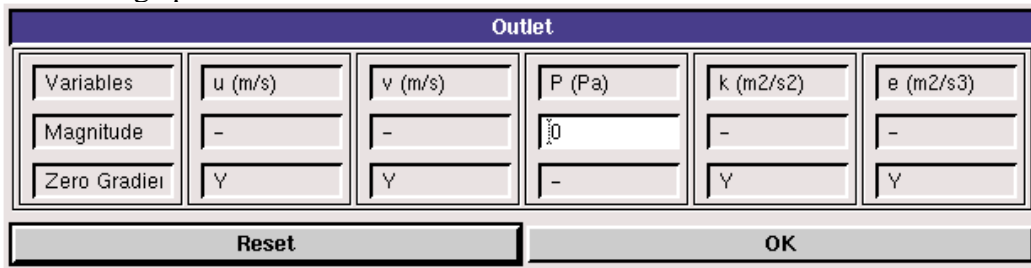
Choose turbulent model (k-e).

(5). Boundary Conditions

At **“Inlet”**, we use constant pressure and fix the velocity to 7.04m/s for turbulent airfoil flow. Use default values for “k” and “e”.



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



On **“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. 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		

For “C” type mesh, there is one additional boundary condition, i.e., “**Symmetry**”.

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

(6). Initial Conditions

Use the default setup for initial conditions.

Initial Condition					
Variables	P (Pa)	u (m/s)	v (m/s)	k (m2/s2)	e (m2/s3)
Magnitude	0	7.04	0	0.08	7.4
Reset			OK		

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

Step 3: (Mesh)

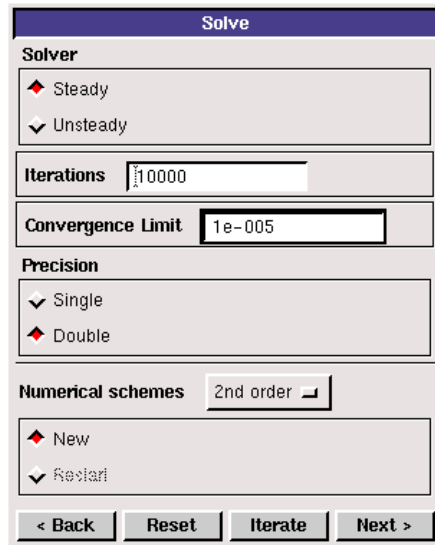
For CFD Lab 2, “**Structured**” meshes will be generated using either “**Automatic**” or “**Manual**” generations (see exercises at the end of this document for details). For “**Automatic**” generation, just choose the mesh density: “coarse”, “medium” or “fine”, and click <<**Create**>>, FlowLab will automatically create the mesh you required. For manually generation, you should first choose the “Manual” button, and then the following panel will be shown:

1. **“O” mesh generation:** to generate an “O” type mesh, you need to specify the number of intervals and the first length in grid spacing for NA1, NA2, NA3 and NA4, which are the four curves that form the airfoil geometry. Then choose the number of intervals and the grid spacing near airfoil for the edge NC1, which lays from the airfoil surface to the far-field. Finally choose **“Uniform”** distribution for NF1, NF2, NF3, and NF4, which are the four curves that form the far field circumference. Click **<<Create>>** after you input the parameters for each edge and then click **<<Create>>** in the mesh step window to generate the whole mesh.
2. **“C” mesh generation:** to generate a “C” type mesh, you need to specify “NA1”, “NA2”, “NA3”, “RC”, “Symmetry”, and “inlet”. For details on setting up those parameters, see exercises on “C” mesh generation.

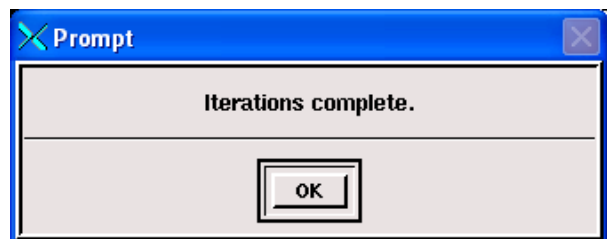
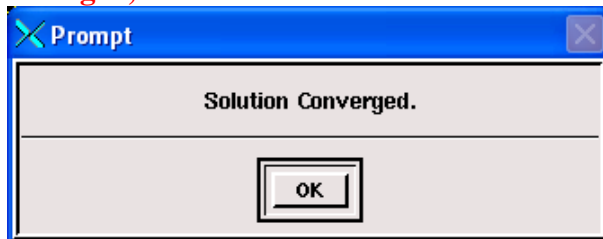
NOTE: When specifying parameters for NA1, NA2, NA3, NA4, you have to use the zoom in order to visualize the mesh distribution near the airfoil.

Step 4: (Solve)

In this Lab, both 1st order and 2nd order numerical schemes will be used (read exercises for details).

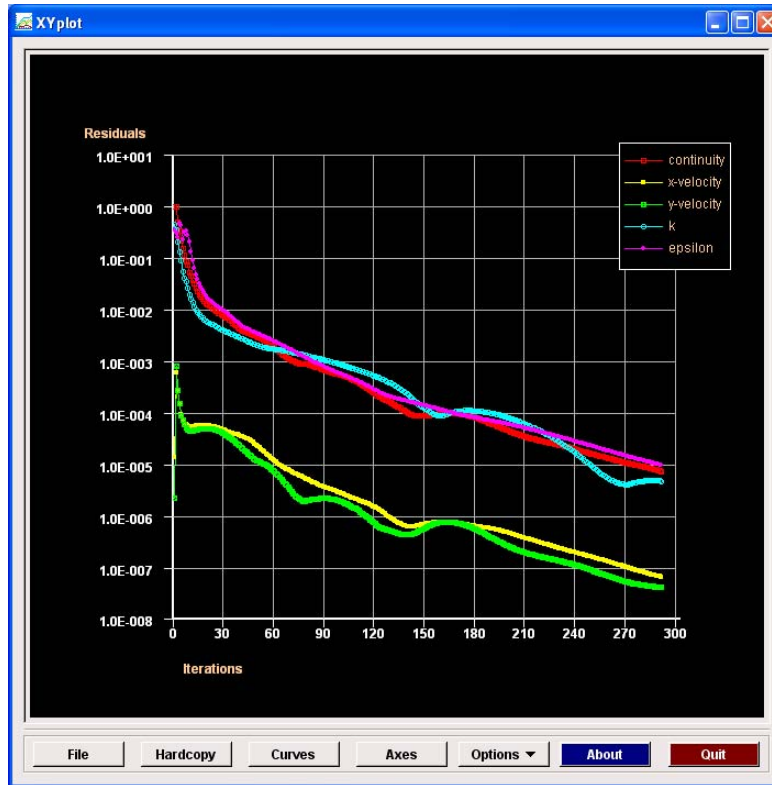


The flow is steady, so turn ON the “**Steady**” option. Specify the iteration number and convergence limit to be **10000** and **10^{-5}** , respectively. Choose “**Double precision**” with “**1st or 2nd order scheme,**” based on which exercise you are working on. Use “**New**” calculation for this Lab. Then click <<**Iterate**>> and FlowLab will begin the calculation. Whenever you see the window, “**Solution Converged,**” click <<**OK**>>.



NOTE: FlowLab stops a calculation when the maximum number of iterations or the convergence limit is reached, whatever happens first. In the first case, the “Iterations complete” window will be shown. In the second case, the “Solution Converged” window will be displayed.

The iterative history of residuals for continuity equation, x and y momentum equations, and k and epsilon equations (for turbulent flow) will be shown automatically.

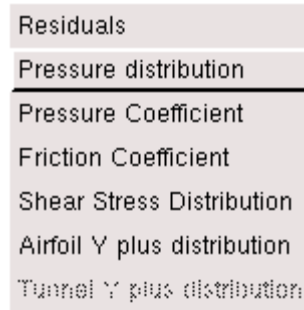


Step 5: (Reports)

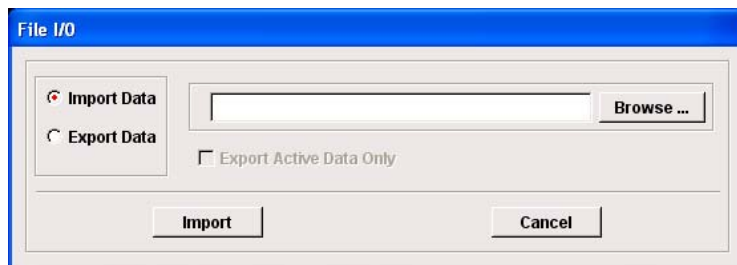
“**Reports**” first provides you the information on “wall shear stress”, “skin friction factor”, “coefficient of lift”, and “coefficient of drag”. “XY plots” provide plots for “residuals”, “pressure distribution”, “pressure coefficient”, and “Shear Stress Distribution”, etc. **In this Lab, only XY plots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

Reports	
Wall Shear Stress	0.233419 Pa
Skin Friction Factor	0.00768927
Coefficient of Lift	0.333423
Coefficient of Drag	0.0218821
Verification and Validation	Open
XY Plots	
Airfoil Y plus distribution	Plot
Data	Browse...
< Back	
Close	

“XY Plots” provides the following options:



To import the EFD data and plot it 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 Lab 1 for pipe flow.

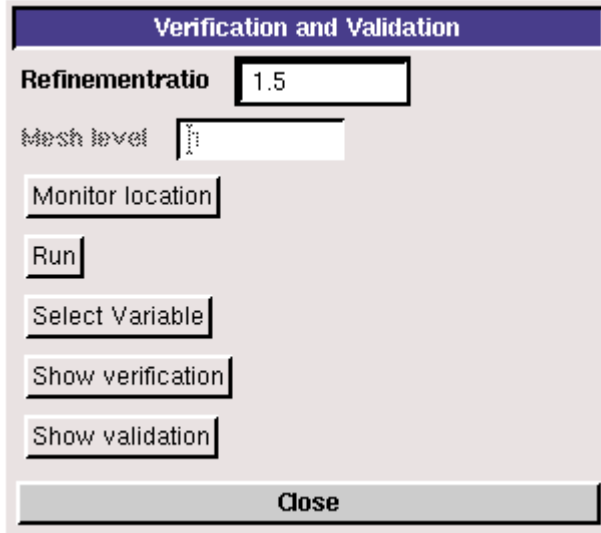


In this Lab:

1. EFD data for pressure coefficient distribution for angle of attack of zero and six degrees can be downloaded from class website: http://css.engineering.uiowa.edu/~me_160, in the names of **Pressure-coef-attack0.xy** and **Pressure-coef-attack06.xy**, respectively.

Verification and Validation

In CFD Lab 2, you will conduct Verification and Validation studies using the manually mesh generation options (see exercise notes for details). The V&V panel is shown below. Whenever you manually create a mesh, that mesh will be automatically used as the default “fine” mesh for verification.



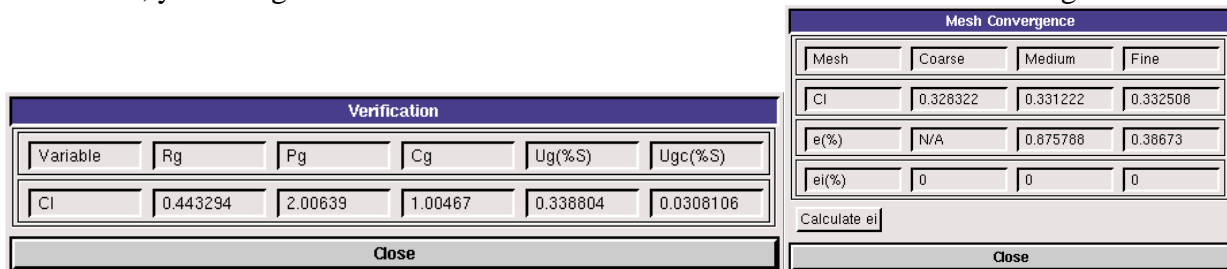
First input the refinement ratio used to create the “coarse” and “medium” meshes from the “fine” mesh.

<<**Monitor location**>> is used to specify the locations for line monitors (for axial velocity profile only). In this Lab, we will do validation for pressure coefficient only, so, leave “Monitor location” as it was. The locations for the pressure coefficient have been hard coded in the software so you don’t need to specify them by yourself.

Click <<**Run**>> and FlowLab will conduct the simulation in the order of “fine mesh” → “medium mesh” → “coarse mesh”. The information on which mesh is being solved now will be displayed in the left bottom window.

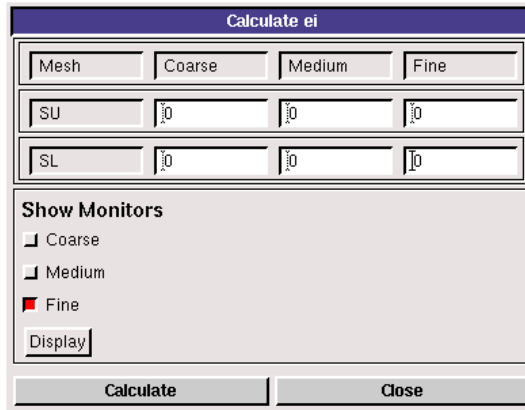
Whenever the V&V completed, you can select which variables you want to be shown for verification results. In this Lab, you will choose either “lift coefficient” or “pressure coefficient”.

For lift coefficient, choose “lift coef.” under button <<**Select Variables**>>, then click “Show Verification”, you will get two tables: verification results table and the “mesh convergence” table.

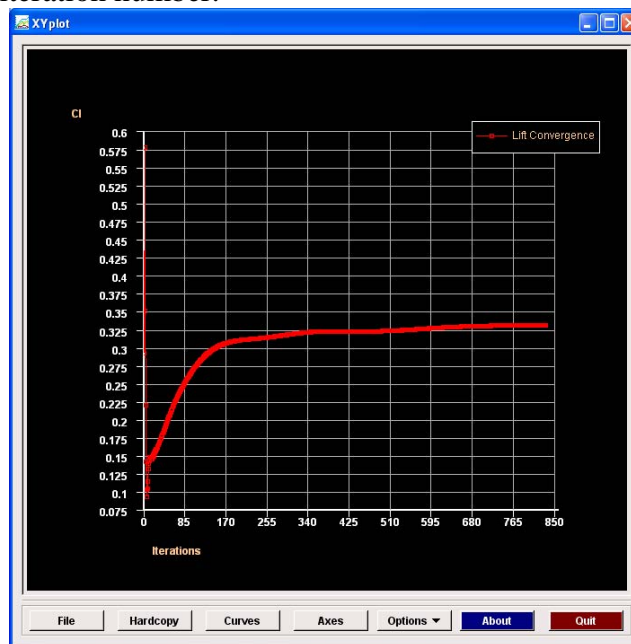


You need to save both. You can use <<**Alt+printScreen**>> to copy the active window and then paste it into a WORD document.

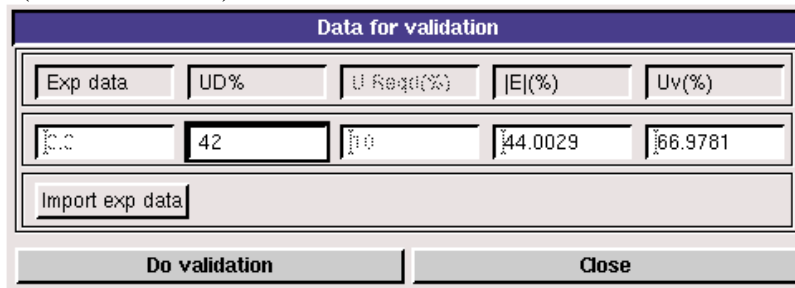
To estimate how many iterations will be enough, you can click <<**Calculate ei**>> button and will see the following panel:



Choose <<**Fine**>> mesh and click <<**Display**>>, you will see the XY plot showing the lift coefficient as a function of iteration number.



For “pressure coefficient monitors”, choose “pressure coef. Monitor” under <<**Select Variables**>>, and click <<**Show Validation**>>, then the following panel is shown, specify the experimental data uncertainty to be 42% (from EFD data).

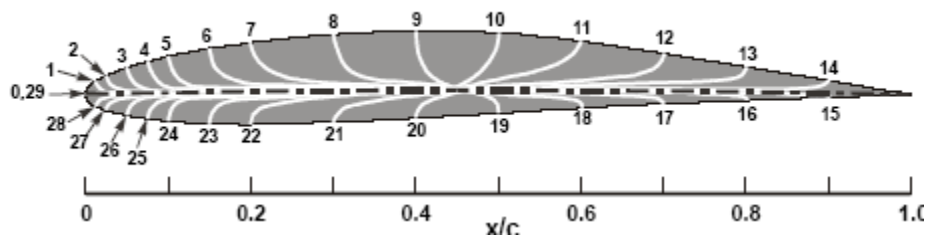


Click <<**Import exp data**>>, then 29 pressure coefficients will be shown. To make it simple, we have made the default values in the software same as the experimental data, so you do not need change them.

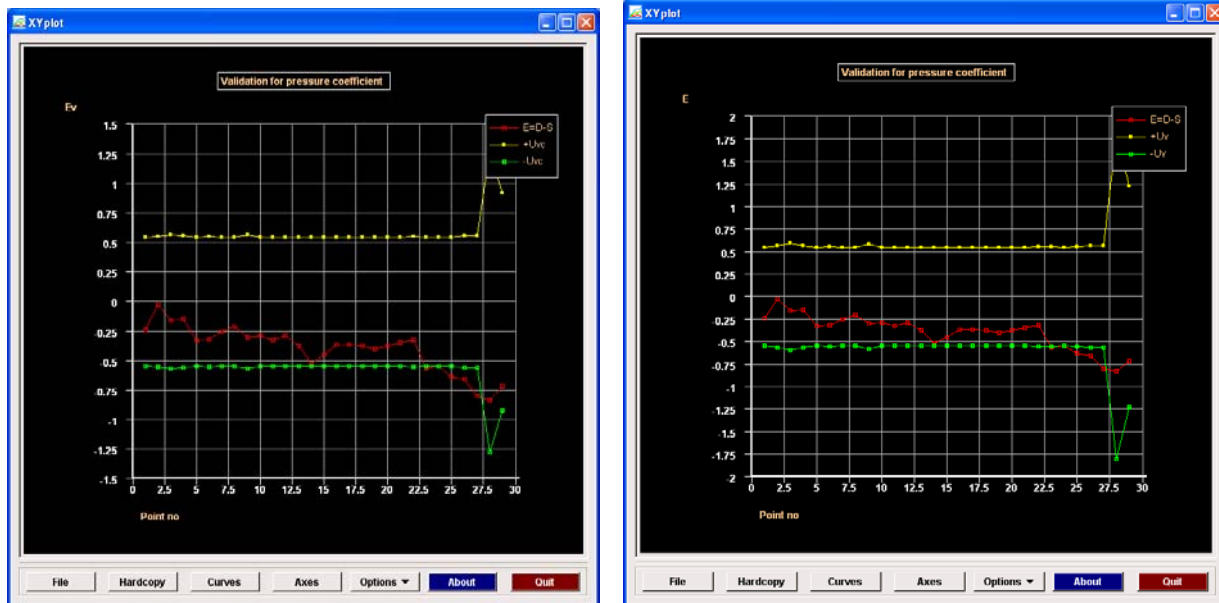
Exp data for pres coefficient monitors	
Point 1	0.7705
Point 2	0.40556
Point 3	-0.06083
Point 4	-0.4596
Point 5	-0.6759
Point 6	-0.7638
Point 7	-0.8314
Point 8	-0.8584
Point 9	-0.8381
Point 10	-0.784

Close

The locations of all 29 pressure transducers are shown below:



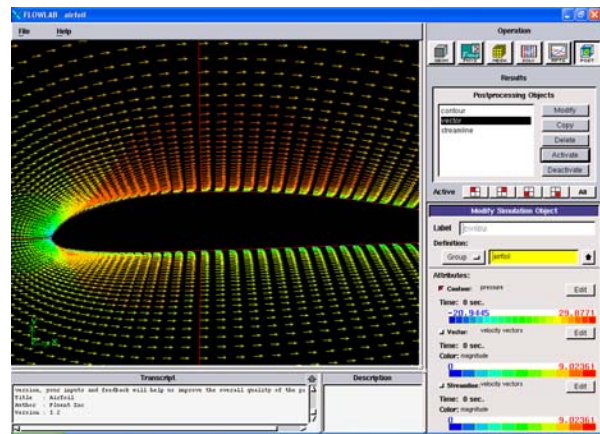
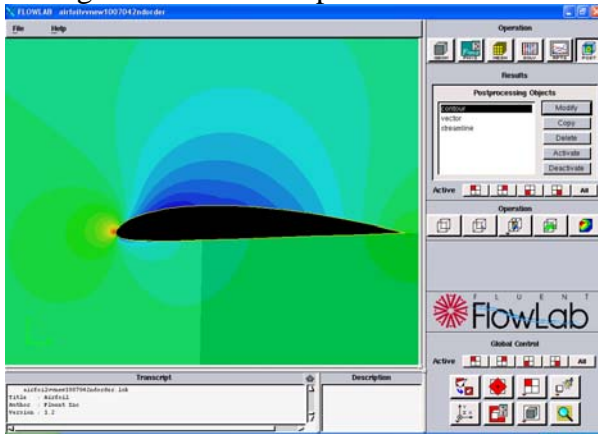
Click <<Do Validation>>, the figures similar to the following will be shown:



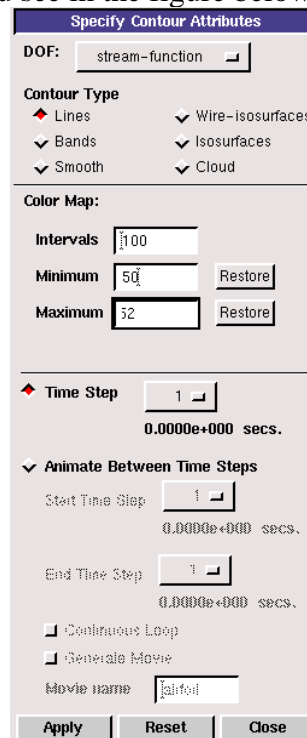
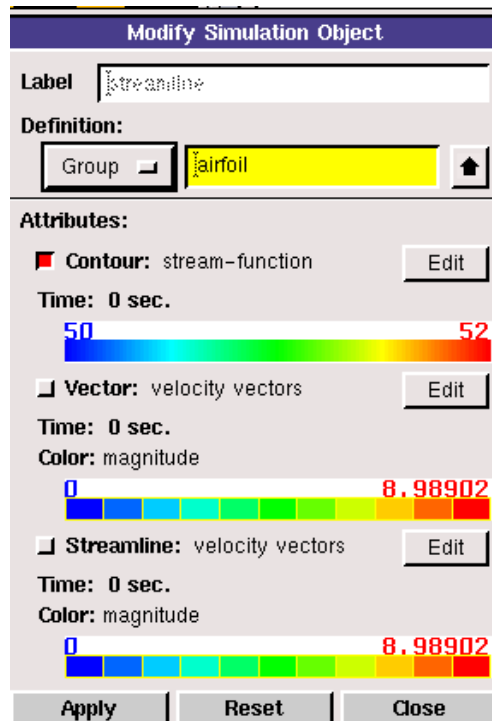
You can use the concepts of validation presented in the CFD lecture to explain which points are validated and which points are not and explain why.

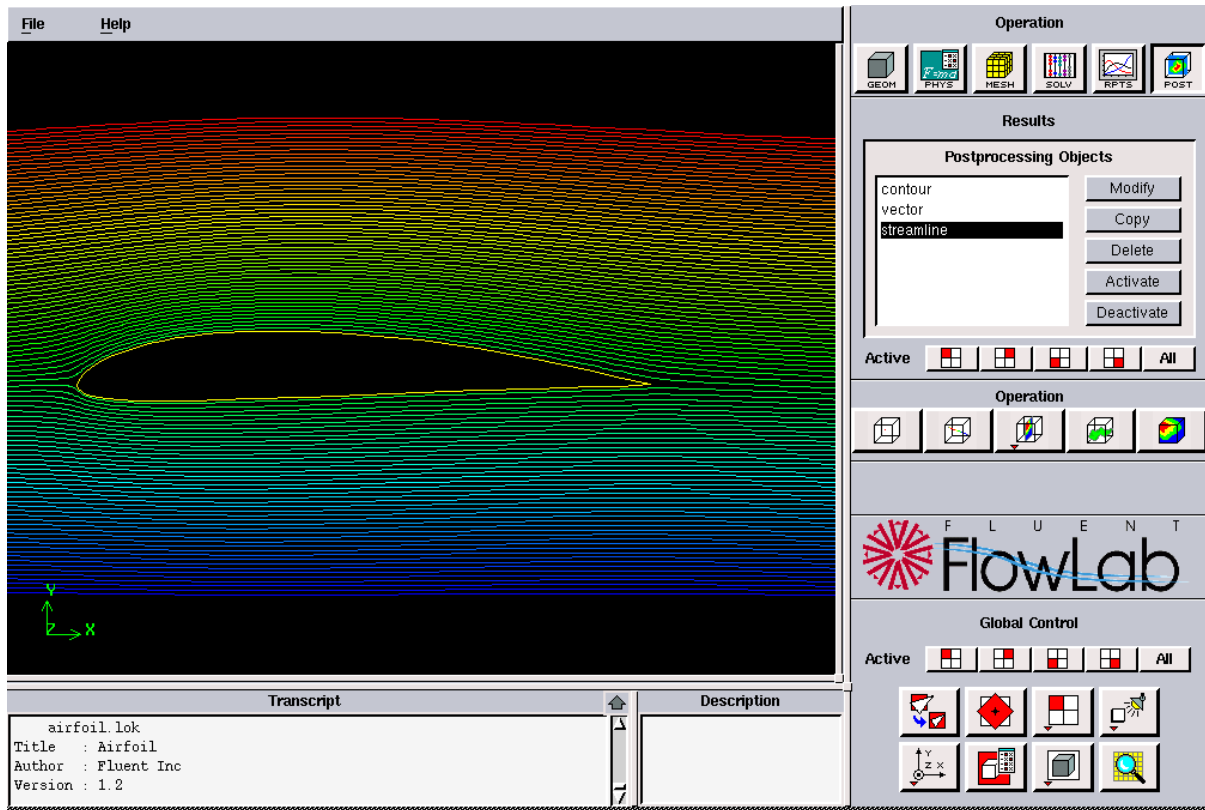
Step 6: (Post-processing)

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 CFD Lab 1 and the workshop. The following shows some sample results.



To plot “streamlines”, first <<**Deactivate**>> “contour” and “velocity vector”, then click “Streamline” and <<**Activate**>>. Click <<**Modify**>> button and the contour variable will be “stream-function”. Usually, the default stream-function values range from the minimum to maximum through the whole domain. To illustrate the streamlines close to the airfoil surface, we need re-specify the range of stream-function values. Click <<**Edit**>>, using <<**Lines**>> instead of <<**Bands**>>. Specify appropriate contour “intervals” and “minimum” and “maximum” stream function values, then you can zoom in the region close to airfoil surface to show the streamlines, as you see in the figure below.





4. Exercises

You need to complete the following assignments and present results in your lab report following the lab report instructions

Verification and Validation of Turbulent Flow around a Clark-Y Airfoil

- You can save each case file for each exercise using “file”→ “save as”
- Except for exercise 5, all the cases must run with the turbulent model.
- Otherwise stated, use the parameters shown in the instruction.
- Otherwise stated (exercise 2), use 2nd order scheme.

1. Effect of domain size: using the default values shown in the instruction, and choose “automatic coarse” mesh, run 5 simulations using five different domain sizes (as listed in the following table) for “O” meshes. Fill the table with lift coefficient with their relative difference between two successive meshes. If the relative change between two successive domain sizes should be less than 1%, then which domain sizes will be enough large to make the CFD simulation results to be independent of the domain size?

Circle radius (m)	1	2	3	4	5
Lift Coefficient					
Relative change	N/A	()%	()%	()%	()%

- **Figures to be saved:** None.
- **Data to be saved:** the above table with values.

2. Effect of numerical scheme on Verification study for lift coefficient and validation of pressure coefficient:

Use “O” type geometry with 0 degree angle of attack. Use the parameters shown in the following table to manually generate the “O” mesh.

For this exercise only, find one partner in the class to form a group, one student will run V&V using first order upwind scheme, the other will use 2nd order upwind scheme. Then, you must borrow the figures/data from the other student and present in your lab report.

	NA1	NA2		NA3		NA4	NC1	NF1	NF2	NF3	NF4
Intervals	67	135		135		67	168	Uniform			
First length	3.35e-4	1	2.2237e-3	1	2.2237e-3	3.35e-4	1.19e-4				
		2	2.2237e-4	2	2.2237e-4						

Based on verification results for lift coefficient, which numerical scheme is closer to the asymptotic range? Which numerical scheme has a lower grid uncertainty? Discuss the validation figure. For which locations of 29 points the CFD simulation has been validated? For which locations the CFD simulation has not been validated? For iterative history of lift coefficient, what is the minimum iteration number for you to determine the lift coefficient has been converged to a “constant” value?

NOTE: (1). You must create the mesh manually. The calculation time for a complete V&V could take around 40 minutes, depending on the computer you are using.

(2). For V&V for lift coefficient and pressure coefficient, you don’t need to specify line monitor locations. FlowLab has hard coded the pressure tap locations.

- **Figures to be saved** (only for the numerical scheme you used, but you must also present the figures for the calculations from your partner): 1. The “O” mesh you generated. 2. FlowLab “Mesh Convergence” panel and “Verification” panel for lift coefficient. 3. Validation figures for pressure coefficient. 4. Iterative history for lift coefficients on fine mesh.
- **Data to be saved:** None.

3. “C” mesh generation: Use “C” type domain and zero degree angle of attack for geometry and use the following parameters for mesh generation. Other parameters are the same as the values in the instruction part.

	NA1	NA2		NA3		Symmetry	RC	Inlet
Intervals	40	30		30		40	75	Uniform
First length	1	0.001	1	0.01	1	0.01	1.19e-4	
	2	0.001	2	0.006	2	0.006		

- **Figures to be saved:** “C” mesh generated by yourself.
- **Data to be saved:** converged lift coefficient.

4. Effect of angle of attack on airfoil flow: Using “O”, automatic “coarse” meshes, run two simulations using angle of attack 0 degrees and 6 degrees, respectively. Analyze the difference of flow fields. Which case has a higher lift coefficient, which has a higher drag coefficient?

- **Figures to be saved** (for both attack angles): 1. pressure contours, 2. comparisons with EFD on pressure coefficient distribution, 3. velocity vectors near airfoil surface, 4. streamlines near the airfoil surface.
- **Data need to be saved** (for both attack angles): lift and drag coefficients.

5. Questions need to be answered when writing CFD report:

- 5.1. Answer all the questions in exercises 1 to 4
- 5.2. Analyze the difference between CFD/EFD and possible error sources.