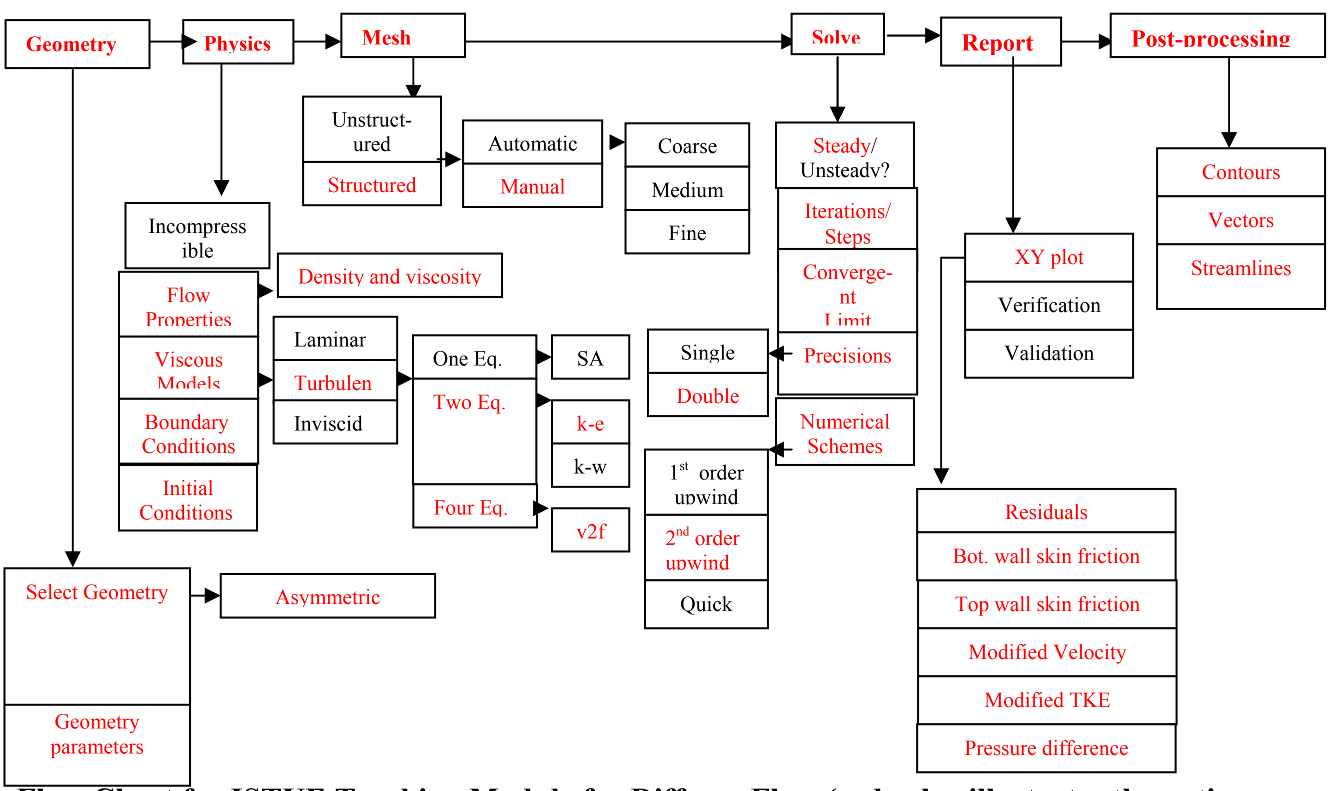
Simulation of Turbulent Flow in an Asymmetric Diffuser

58:160 Intermediate Mechanics of Fluids
CFD LAB 3

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

1. Purpose

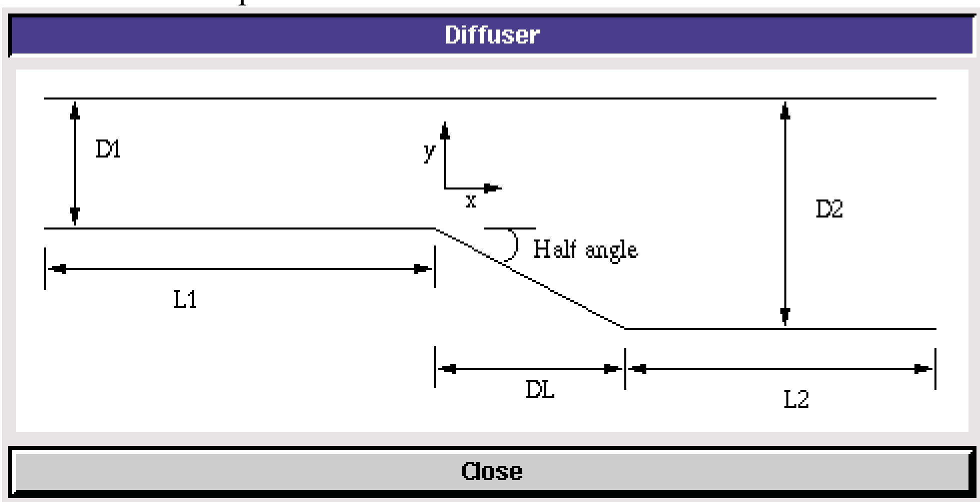
The Purpose of CFD Lab 3 is to simulate **turbulent** flows inside a diffuser following the "CFD process" by an interactive step-by-step approach and conduct verifications using CFD Educational Interface (FlowLab 1.2). Students will have "hands-on" experiences using FlowLab to conduct **validation of velocity, turbulent kinetic energy, and skin friction factor. Effect of turbulent models will be investigated, with/without separations**. Students will manually generate meshes, solve the problem and use post-processing tools (contours, velocity vectors, and streamlines) to visualize the flow field. Students will analyze the differences between CFD and EFD and present results in a CFD Lab report.



Flow Chart for ISTUE Teaching Module for Diffuser Flow (red color illustrates the options you will use in CFD Lab 3)

2. Simulation Design

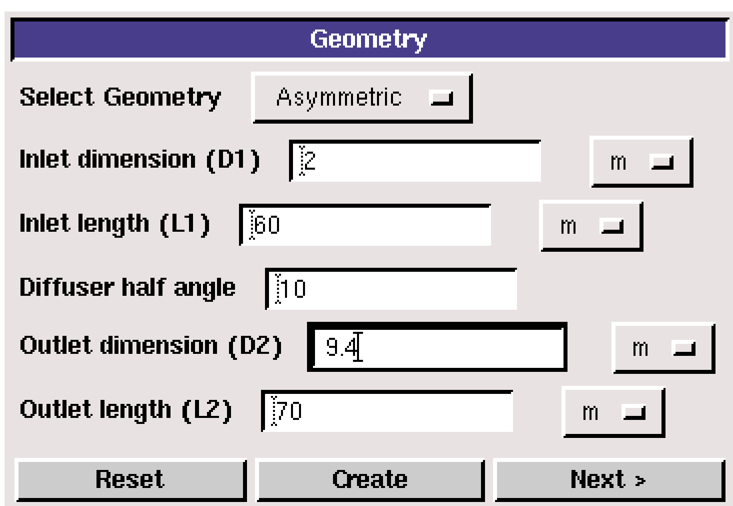
The problem to be solved is that of turbulent flows inside an asymmetric diffuser (2D). Reynolds number is 17,000 based on inlet velocity and inlet dimension (D1). The following figure shows the sketch window you will see in FlowLab with definitions for all geometry parameters. Before the diffuser, a straight channel was used for generating fully developed channel flow at the diffuser inlet. The origin of the coordinates is placed at the inlet of the channel before diffuser.



In CFD Lab3, all EFD data for turbulent airfoil flow in this Lab will be provided by the TA and saved on the Fluids lab computers.

3. CFD Process

Step 1: (Geometry)

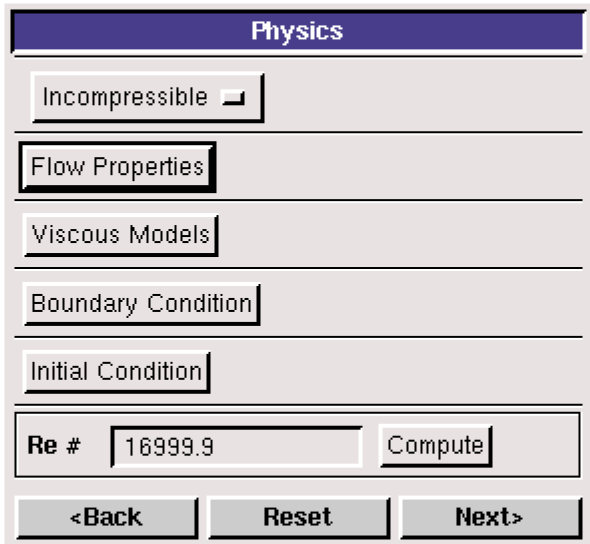


1. Select Geometry: Asymmetric
2. Inlet dimension (D1) (2 m)
3. Inlet length L1 (60 m)

4. Diffuser half angle (10 or 4, read exercises at the end)
5. Outlet dimension (D2) (9.4 m)
6. Outlet length (L2) (70 meters).

Click <<**Create**>>, after you see the airfoil geometry created, click <<**Next**>>.

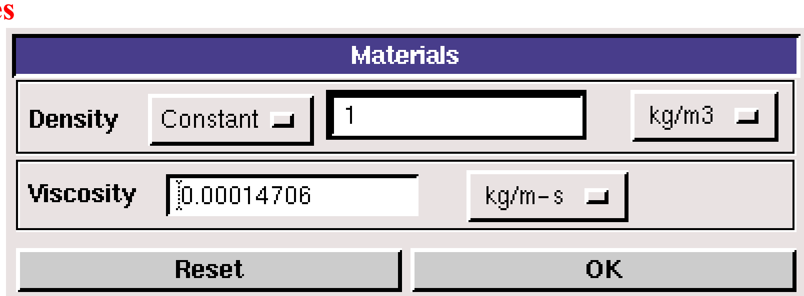
Step 2: (Physics)



(1). Incompressible

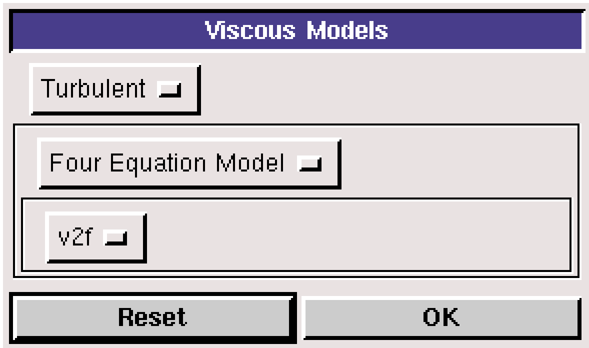
"Incompressible", which is the default setup.

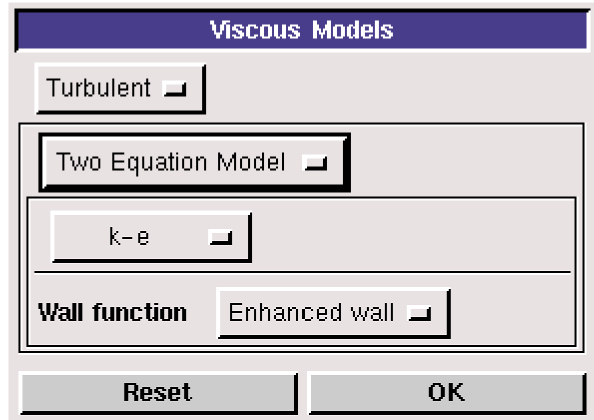
(2). Flow Properties



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

(3). Viscous Model



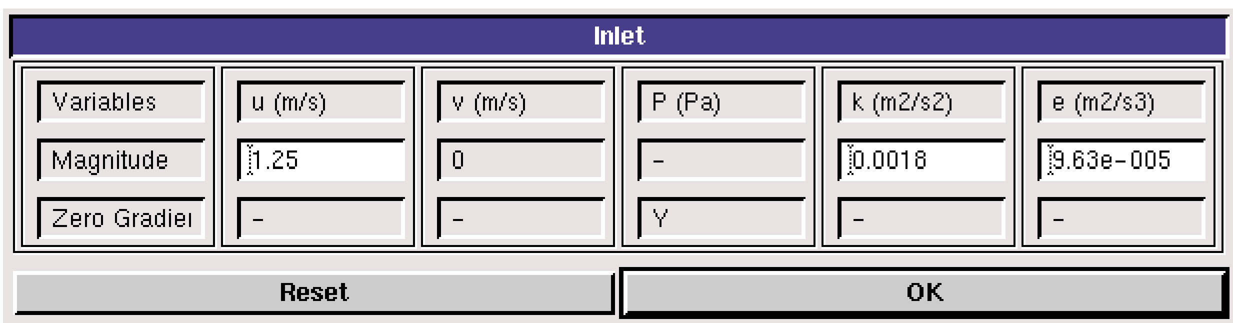


In this lab, the two equation (k-ε) model and the four equation v2f model will be used. "Wall function" with "Enhanced Wall" means that "near wall" models will be used for k-ε.

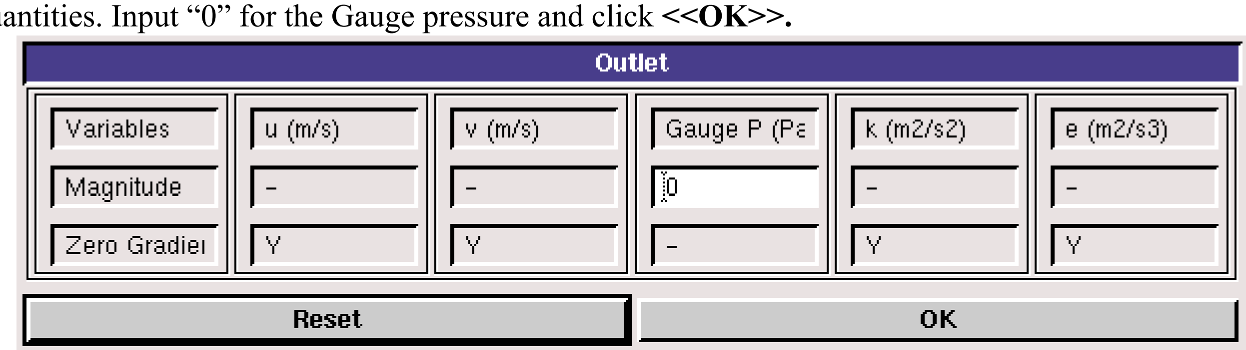
(4). Boundary Conditions

NOTE: for k-e and v2f models, boundary conditions are the same.

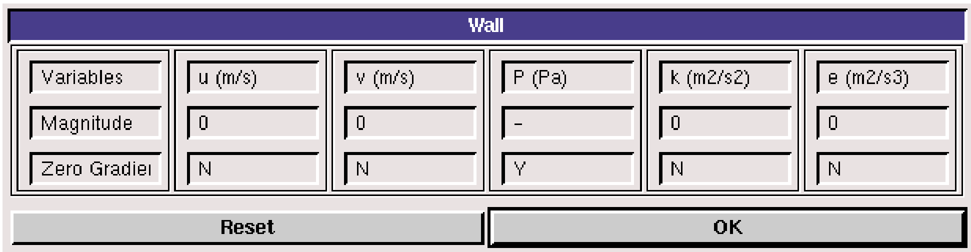
At **"Inlet",** we use constant pressure and fix the velocity to 1.25 m/s. Use default values for "k" and "e".

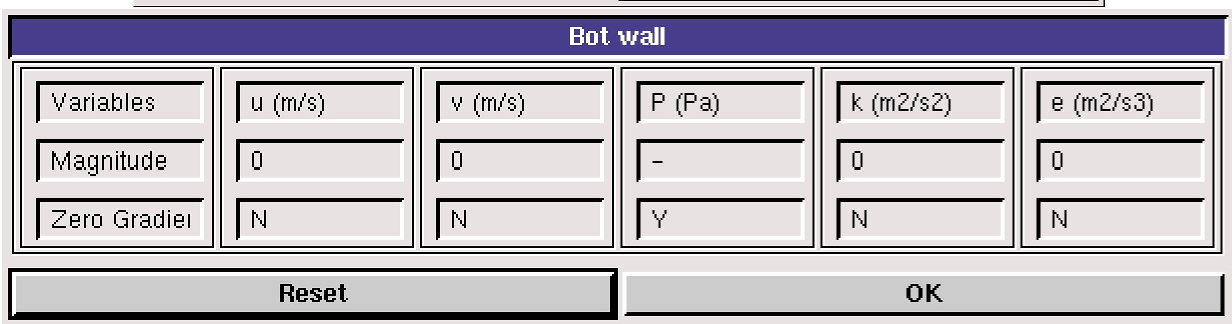


At "**Outlet**", FlowLab uses magnitude for pressure and zero gradients for velocities and turbulent quantities. Input "0" for the Gauge pressure and click **<<OK>>.**



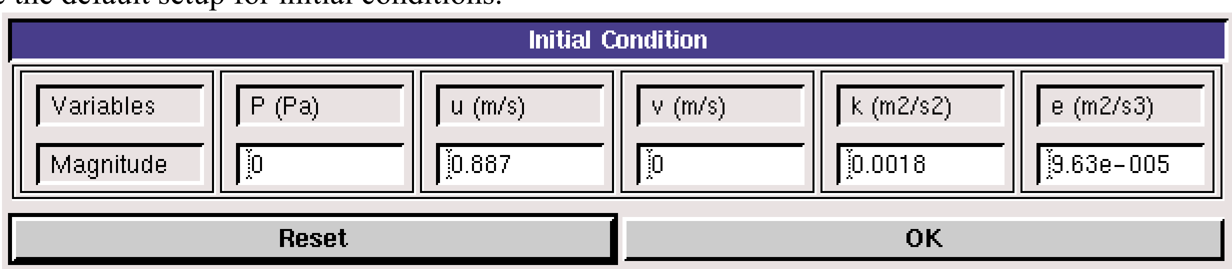
At "**Wall**" or "**Bottom wall**", FlowLab uses no-slip boundary conditions for velocities and zero-gradient for pressure. Turbulent quantities k and ε on the wall are also specified to be zero. Read all the values and click **<<OK>>.**





(5). Initial Condition

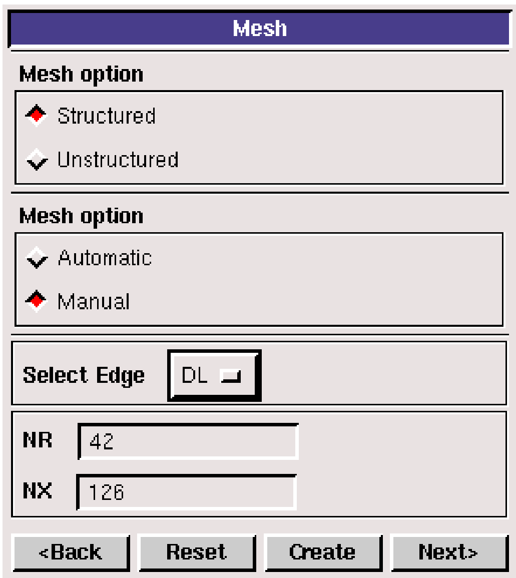
Use the default setup for initial conditions.

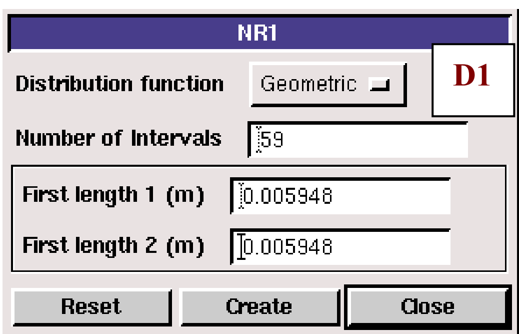


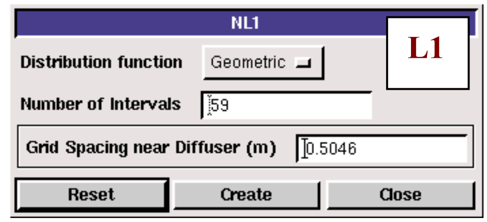
After specifying all the above parameters, click the <<**Compute**>> button and FlowLab will automatically calculate the Reynolds number based on the inlet velocity and inlet dimension you specified. Click <<**Next**>>. This takes you to the next step, "**Mesh**".

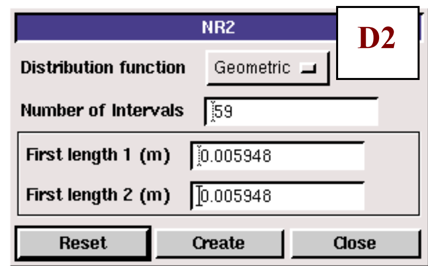
Step 3: (Mesh)

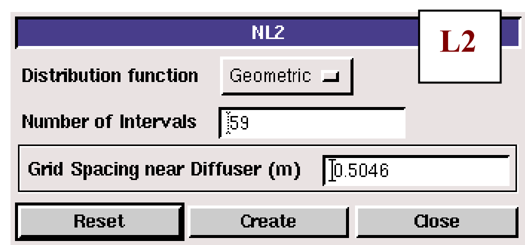
In CFD Lab 3, "**Structured**" meshes will be generated using "**Manual**" option

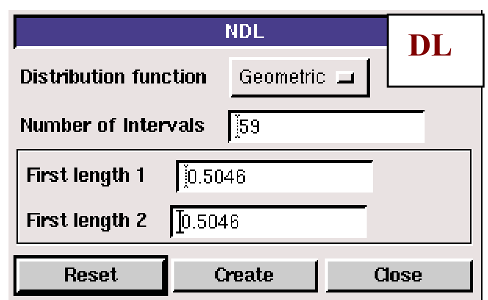








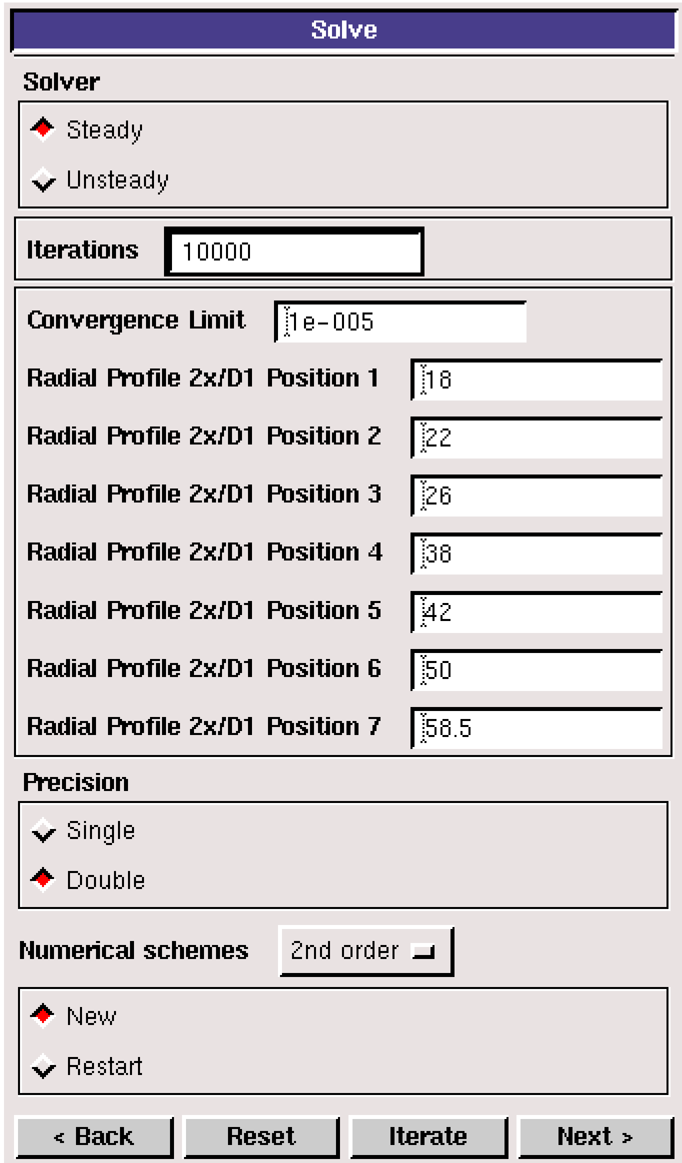




Use the above setup to generate the mesh in this lab. For verification study, this mesh will be used as the "fine" mesh. After you create the mesh, you should zoom in the inlet and outlet of the diffuser, and think about where the mesh was refined and why. Click <<**Create**>> after you input the parameters for each edge and then click <<**Create**>> in the mesh step window to generate the whole mesh.

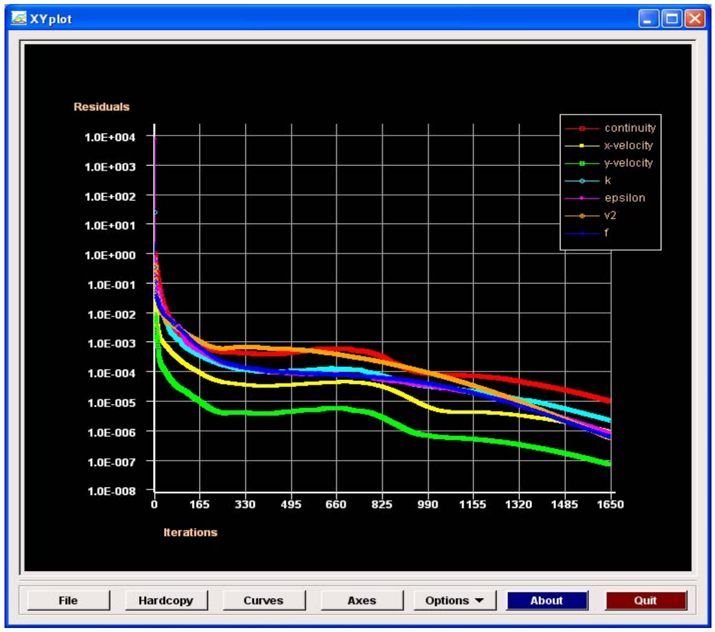
Step 4: (Solve)

In this Lab, ONLY 2nd order numerical schemes will be used.



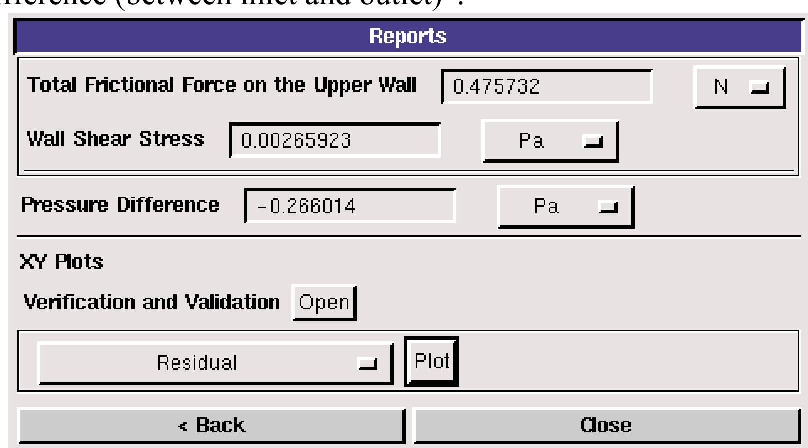
The flow is steady, so turn ON the <<**Steady>>** option and the <<**Unsteady>>** button will automatically be turned OFF. Specify the iteration number and convergence limit to be **10000** and **10^{-5},** respectively. 7 axial positions can be specified (use values shown in the above figure). These positions will be used to plot modified velocity and modified TKE. Choose "**Double precision**" with "**2nd order scheme**". Use "**New**" calculation for this Lab. Then click <<**Iterate**>> and FlowLab will start calculation, whenever you see the window, "**Solution Converged**". Click <<**OK**>>.

The following is an example of XY plot for residuals.

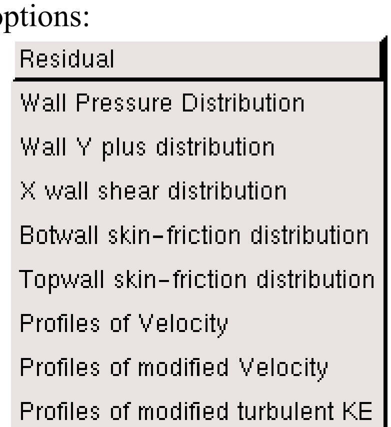


Step 5: (Reports)

"**Reports**" first provide you the information on "Total frictional force on the upper wall", "wall share stress", "pressure difference (between inlet and outlet)".



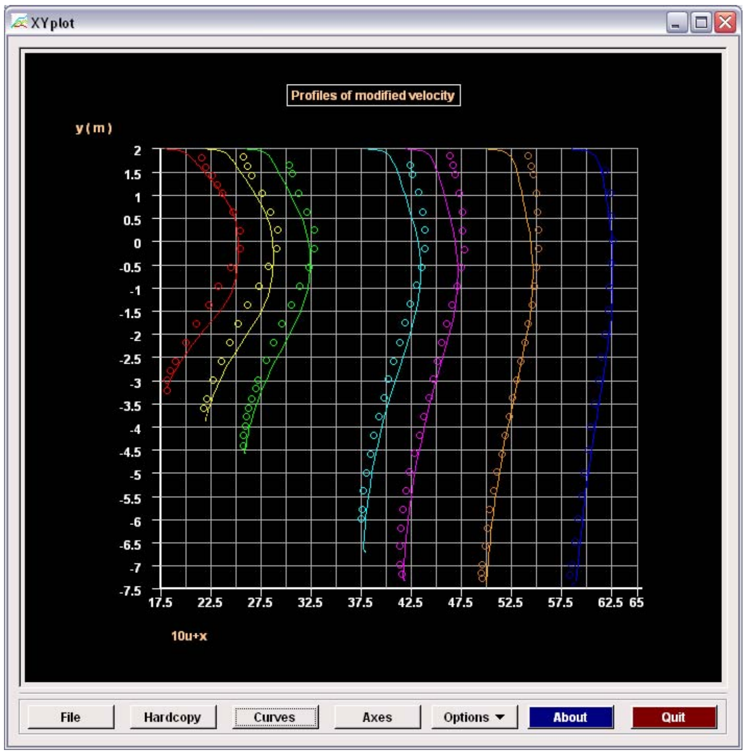
"XY Plots" provides the following options:



In this Lab, all the four EFD data files can be downloaded from the class website, with the following names:

1. EFD data for modified velocity (10u+x) is: Modified_u-10degree.xy
2. EFD data for modified Turbulent Kinetic Energy (500k+x) is: Modified_TKE-10degree.xy
3. EFD data for bottom wall friction factor distribution: Skin_friction_bot_wall.xy
4. EFD data for upper wall friction factor distribution: Skin_friction_top_wall.xy

The following figure shows an example for modified velocity, for both EFD and CFD using v2f model. It is possible to modify the style of the curves by clicking “curves”, selecting a curve with the right button and the clicking on “change style”. For this lab, it is recommended to use lines (without symbols) for CFD and symbols (solid circles) for EFD data. Also, same color for same abscissa of CFD and EFD.



NOTE: 1. For modified velocity and modified TKE, use the default setup for x and y ranges.

2. For bottom wall friction factor distribution, use the following range: x (0, 120), y(-0.001, 0.004). You need click “Axes” in XY plot and turn off “Auto Range”.

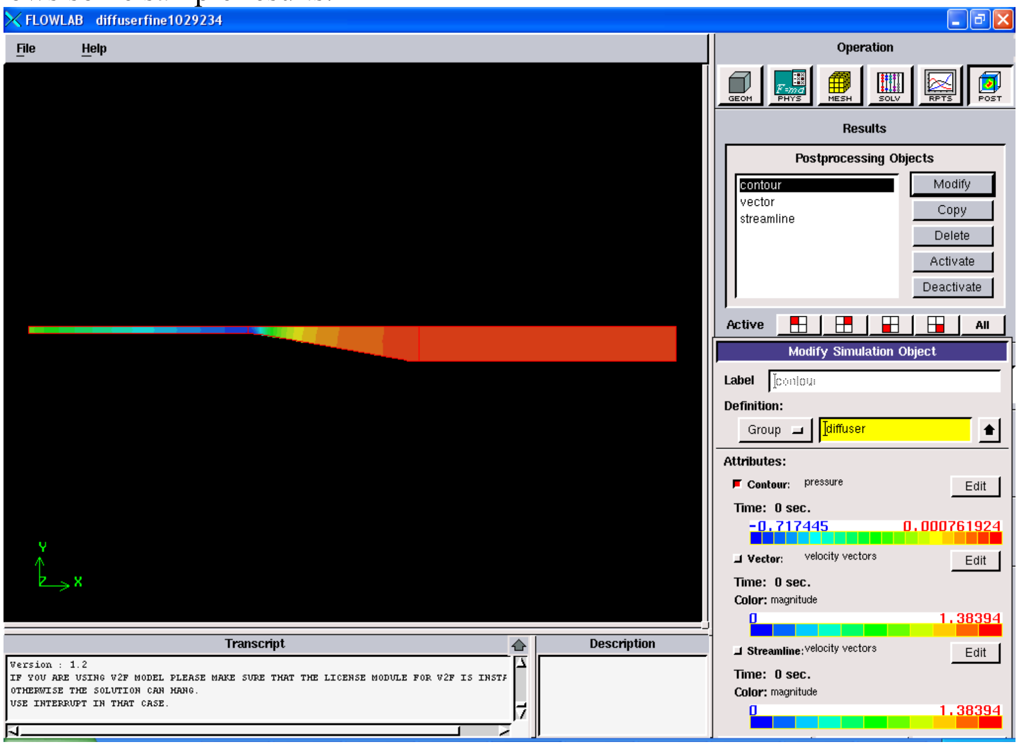
3. For upper wall friction factor distribution, use the following range: x (0, 120), y(0, 0.004)

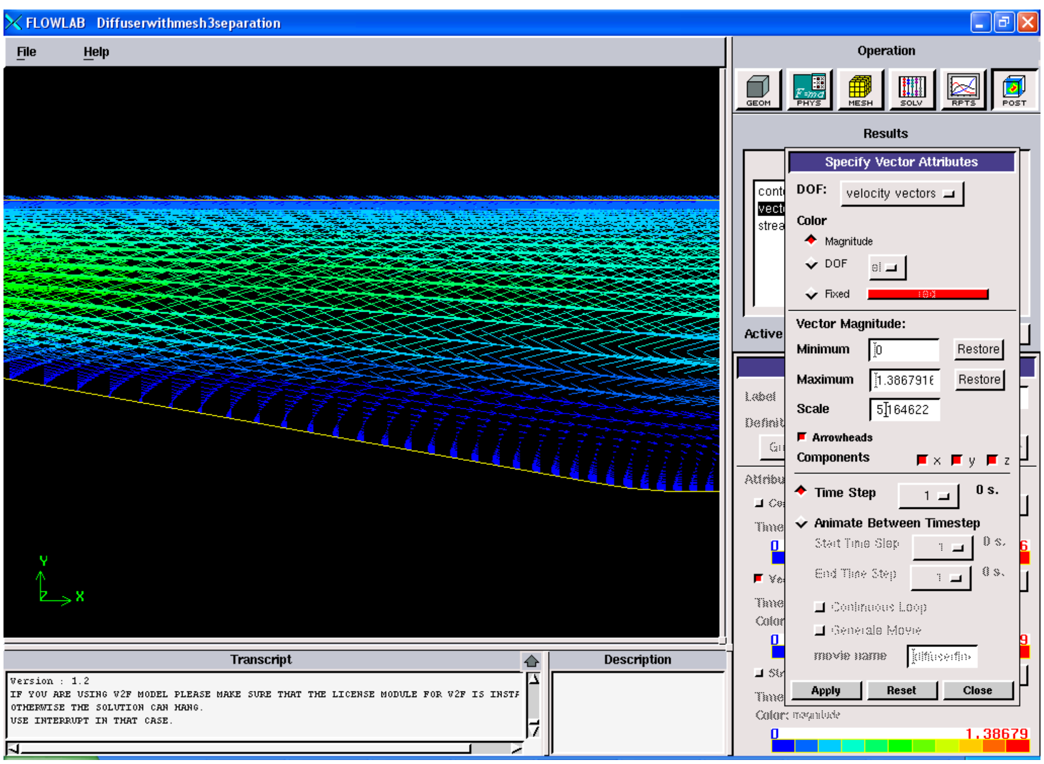
4.For more detailed information on modified velocity, TKE and skin friction factor, please read reference 1 for diffuser on class website:

http://css.engineering.uiowa.edu/~me_160/Lab/CFDdiffuser.pdf

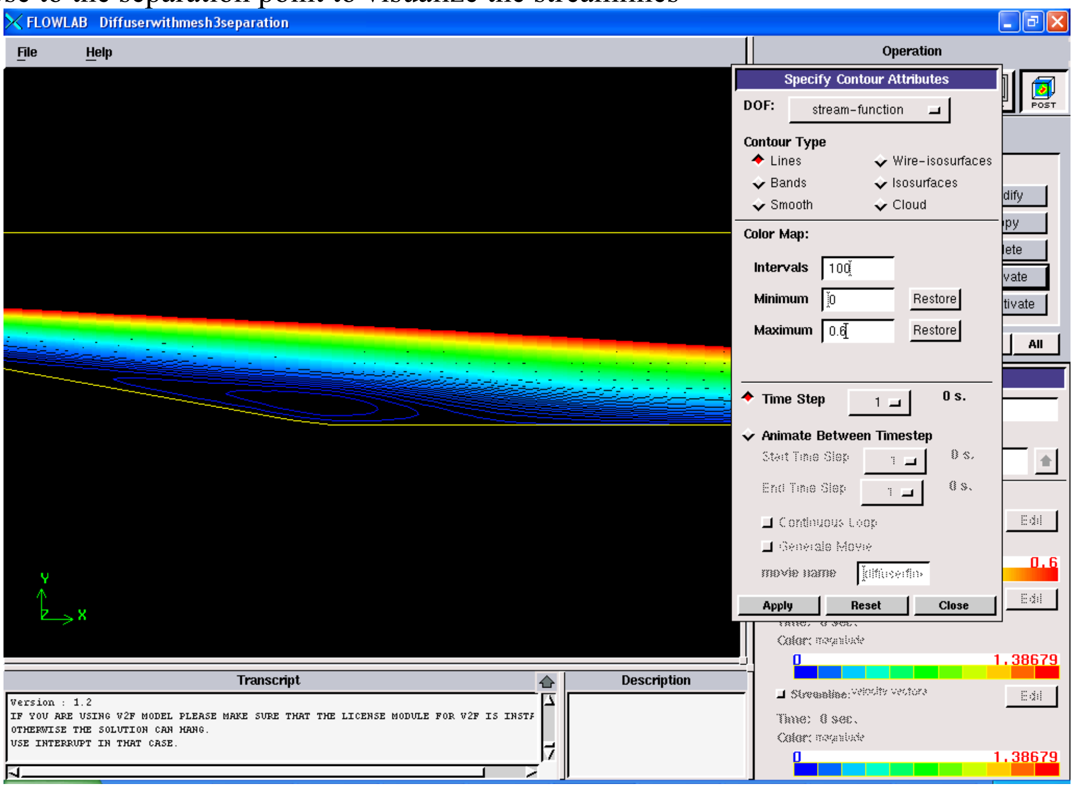
Step 6: (Post-processing)

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” or “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 values 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 the separation point to visualize the streamlines



4. Exercises

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

Simulation of Turbulent Flow in an Asymmetric Diffuser

- **You can save each case file for each exercise using “file”→ “save as”**
- **Otherwise stated, use the parameters shown in the instruction.**

1. Simulation of turbulent diffuser flows without separation

1.1. Iterate until the solution converges using the default values in the instructions, EXCEPT for the following parameters:

1.1.1. Diffuser half angle: 4 degree

1.1.2. Viscous model: v2f.

1.2. Repeat 1.1 but use the k-ε model.

1.3. Questions:

1.3.1. Do you observe separations in 1.1 or 1.2? (use streamlines)

1.3.2. What are the differences between 1.1 and 1.2 regarding modified u, modified TKE, and the variables in the following table?

Turbulent model	Total pressure difference (Pa)	Total friction force on the upper wall (N)
V2f		
k-e		
Relative error (%)		

- Figures to be saved (for both 1.1 and 1.2): 1. XY plots for residual history, modified u vs. x and modified TKE vs. x, 2. Contours of pressure and contours of axial velocity.
- Data to be saved: the above table with values.

2. Simulation of turbulent diffuser flows with separation:

2.1. Iterate until the solution converged using the default values in the instructions, except the following parameters:

2.1.1. Diffuser half angle: 10 degrees

2.1.2. Viscous model: v2f

2.2. Repeat 2.1 but use the k-ε model.

2.3. Questions:

2.3.1. Do you observe separations in 2.1 or 2.2? (using streamlines)

2.3.2. Comparing with EFD data, what are the differences between 2.1 and 2.2 on the following aspects: (1). Modified velocity, (2). Modified TKE, (3). Skin friction factor on top and bottom walls, (4). Variables in the following table.

Turbulent models	Total pressure difference	Total friction force on the upper wall
V2f		
k-e		
Relative error (%)		

2.3.3. If any separation shown, where is the separation point on the diffuser bottom wall (x=?) and where does the flow reattach to the diffuser bottom wall again (x=?) (hint: use wall friction factor)

2.3.4. Do you find any separation on the top wall?

- Figures to be saved (for both 2.1 and 2.2): 1. Residual history, 2. Modified u vs. x with EFD data, 3. Modified TKE vs. x with EFD data, 4. Skin friction factor distributions on top and bottom walls with EFD data, 5. Contour of pressure, 6. Contour of axial velocity, 7. Velocity vectors and streamlines with appropriate scales showing the separation region if the simulation shows separated flows.
- Data to be saved: The above table with values.

3. Questions need to be answered in CFD Lab3 report:

3.1. Questions in exercises 1-2.

3.2. By analyzing the results from exercise 1 and exercise 2, what can be concluded about the capability of k- ε and v2f models to simulate turbulent flows inside a diffuser with and without separations?