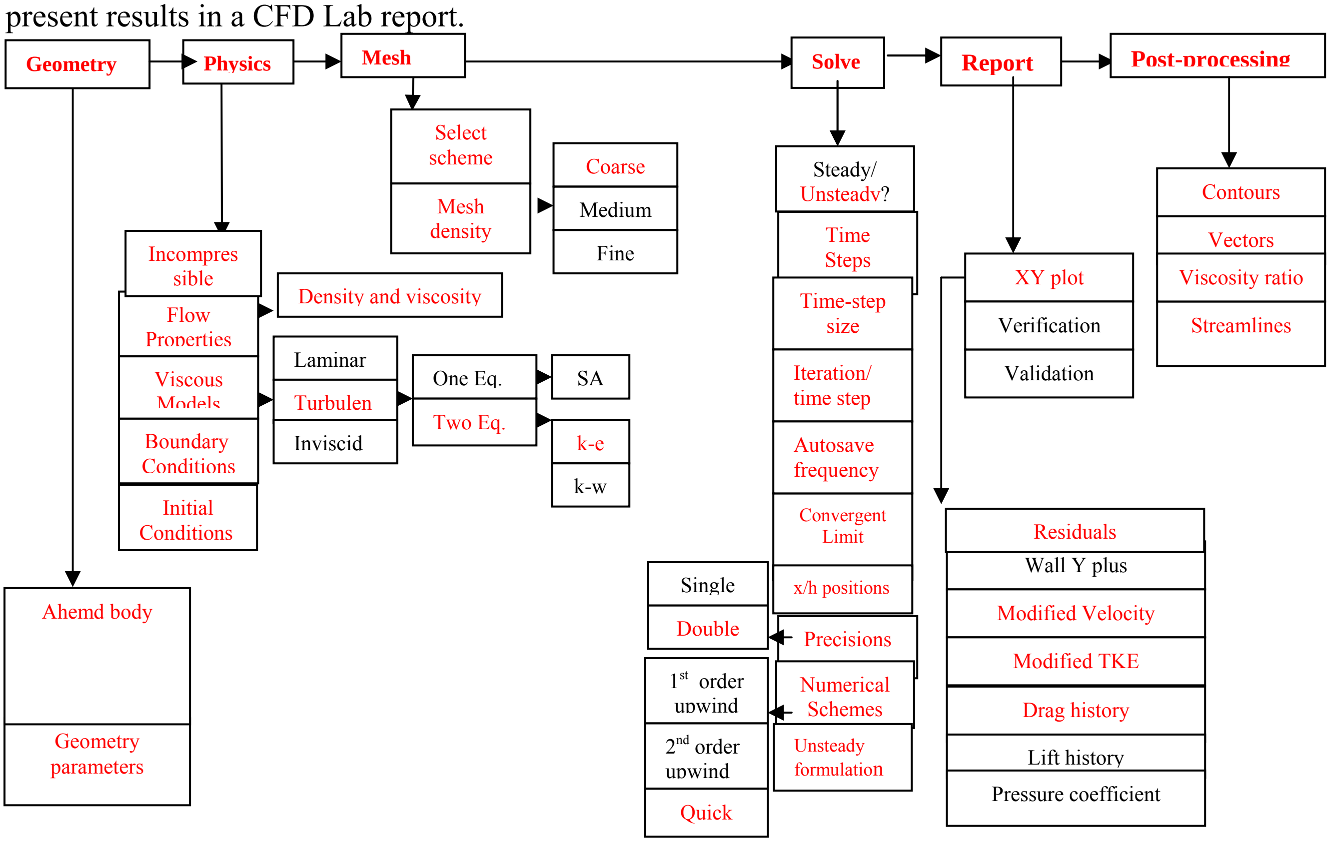
Simulation of Turbulent Flow over the Ahmed Body

58:160 Intermediate Mechanics of Fluids
CFD LAB 4

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

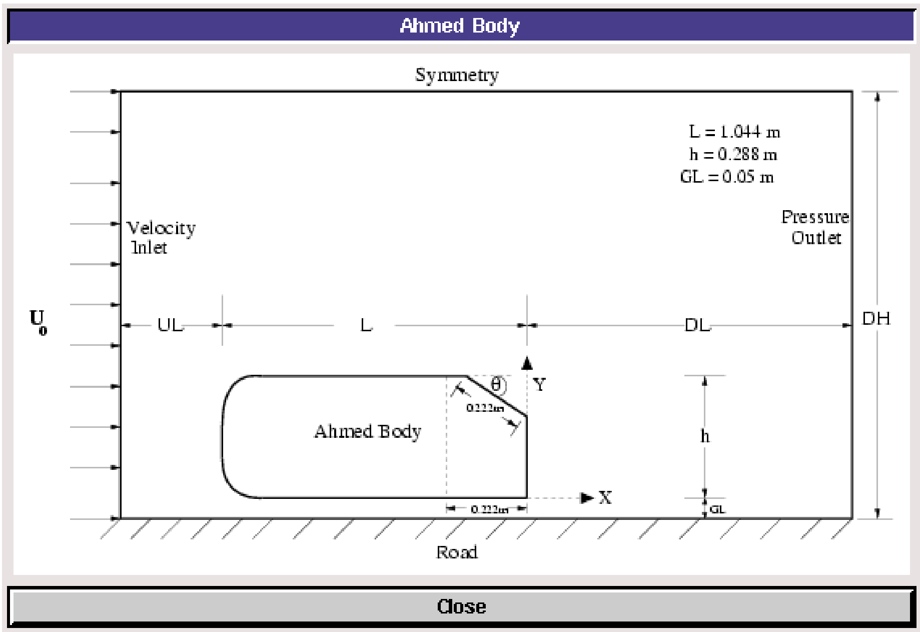
The Purpose of CFD Lab 4 is to simulate **unsteady turbulent** flows over the Ahmed body following the “CFD process” by an interactive step-by-step approach and conduct verifications using CFD Educational Interface (FlowLab 1.2). Students will have “hands-on” experiences using FlowLab to **predict drag coefficients and axial velocity for slant angle 25 degrees and compare them with EFD data.** Students will use post-processing tools (streamlines, velocity vectors, contours, animations) to **visualize the mean and instantaneous flow fields and compute the non-dimensional shedding frequency (Strouhal number)**. Students will analyze the differences between CFD and EFD and present results in a CFD Lab report.



Flow Chart for ISTUE Teaching Module for External flow (red color illustrates the options you will use in CFD Lab 4)

2. Simulation Design

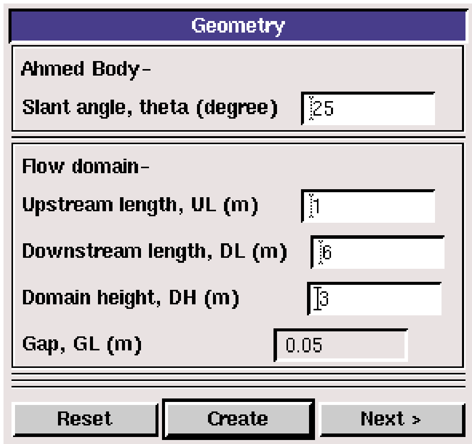
The problem to be solved is unsteady turbulent flows over the Ahmed body (2D). Reynolds number is around 768,000 based on inlet velocity and vehicle height (h). The following figure shows the sketch window you will see in FlowLab with definitions for all geometry parameters. The origin of the simulation is located at the rear of the body. θ is the slant angle. L is the length of the body and h is the height of the body. Uniform velocity specified at inlet and constant pressure specified at outlet. The top boundary of the simulation domain is regarded as "Symmetry" and there is a distance between the car body and road, GL.

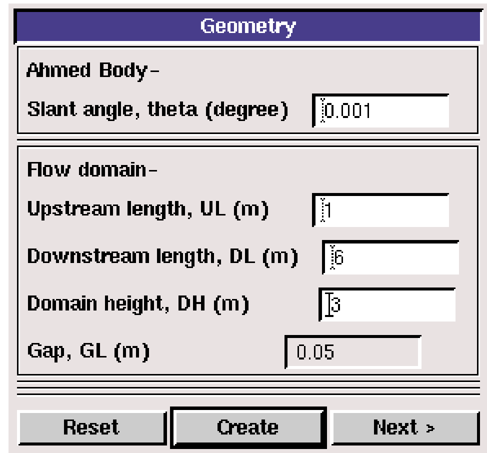


In CFD Lab4, all EFD data for turbulent airfoil flow in this Lab will be provided by the TA and saved on the Fluids Lab computers.

3. CFD Process

Step 1: (Geometry)

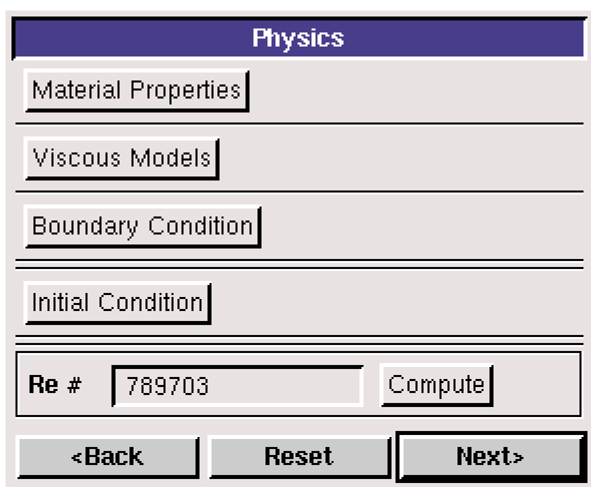




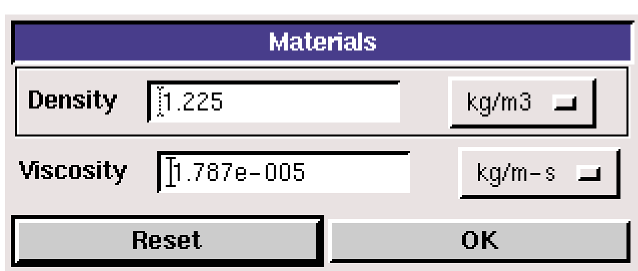
1. Ahmed body, slant angle: (25 degrees)
2. Flow domain, upstream length, UL (1 m)
3. Down stream length, DL (6 m)
4. Domain height, DH (3 m)
5. **5.** Gap, GL (0.05 m, hard coded)

Click <<**Create**>>, after you see the Ahmed car body with domain created, click <<Next>>.

Step 2: (Physics)

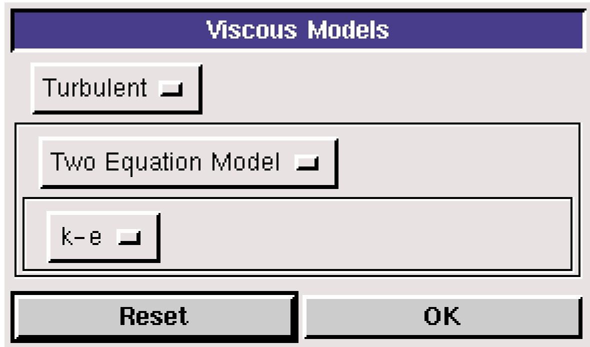


(1). Material Properties



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

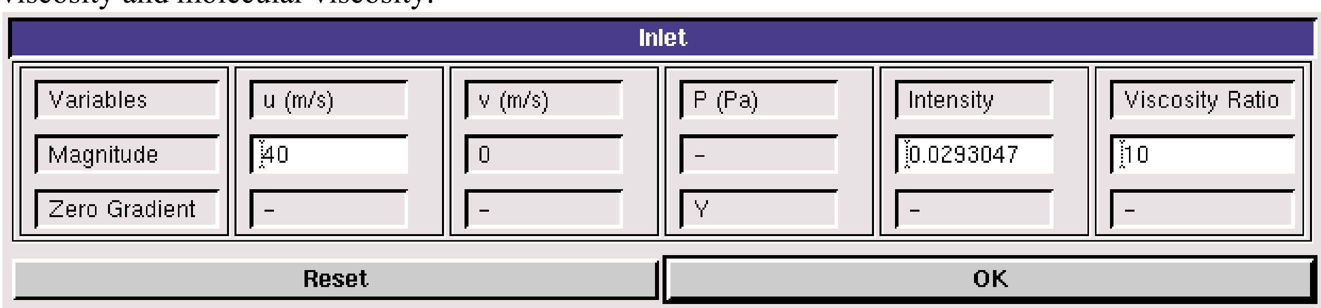
(2). Viscous Model



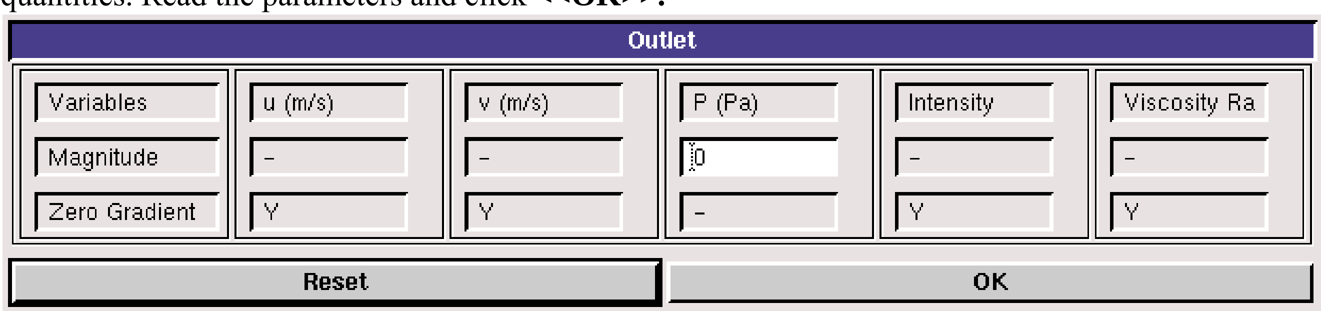
In this lab, the two equation (k-ε) model will be used.

(3). Boundary Conditions

At **"Inlet",** we use uniform velocity (40 m/s) and zero gradient for pressure. Use default values for turbulence intensity and viscosity ratio. Here, viscosity ratio is defined as the ratio of turbulence viscosity and molecular viscosity.



At "**Outlet**", pressure is fixed to be atmosphere pressure. Zero gradients are applied for all other quantities. Read the parameters and click <<**OK**>>**.**



For "**Ahmed body**" or "**Road**", FlowLab uses no-slip boundary conditions for velocities and zero-gradient for pressure. Turbulent quantities (intensity and viscosity ratio) on the Ahmed body and Road are also specified to be zero. Read all the values and click <<**OK**>>**.**

Ahmed Body

Variables	u (m/s)	v (m/s)	P (Pa)	Intensity	Viscosity Ra
Magnitude	0	0	-	0	0
Zero Gradiei	-	-	Y	-	-

Reset | OK

Road

Variables	u (m/s)	v (m/s)	P (Pa)	Intensity	Viscosity Ra
Magnitude	0	0	-	0	0
Zero Gradiei	-	-	Y	-	-

Reset | OK

For "**Symmetry**" boundary, vertical component of velocity is fixed zero and 1 atm specified for pressure. Zero gradients are applied for all other quantities. Read the parameters and click **<<OK>>.**

Symmetry

Variables	u (m/s)	v (m/s)	P (pa)	Intensity	Viscosity Ra
Magnitude	-	0	0	-	-
Zero Gradiei	Y	-	-	Y	Y

Reset | OK

(4). Initial Conditions

Use the default setup for initial conditions.

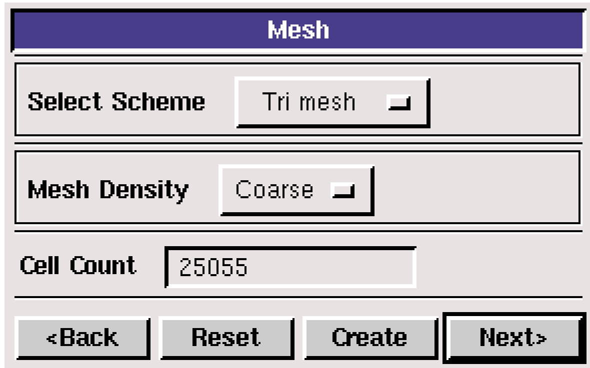
Initial Condition

Variables	P (Pa)	u (m/s)	v (m/s)	k (m2/s2)	e (m2/s3)
Magnitude	0	40	0	2.06103	2620.74

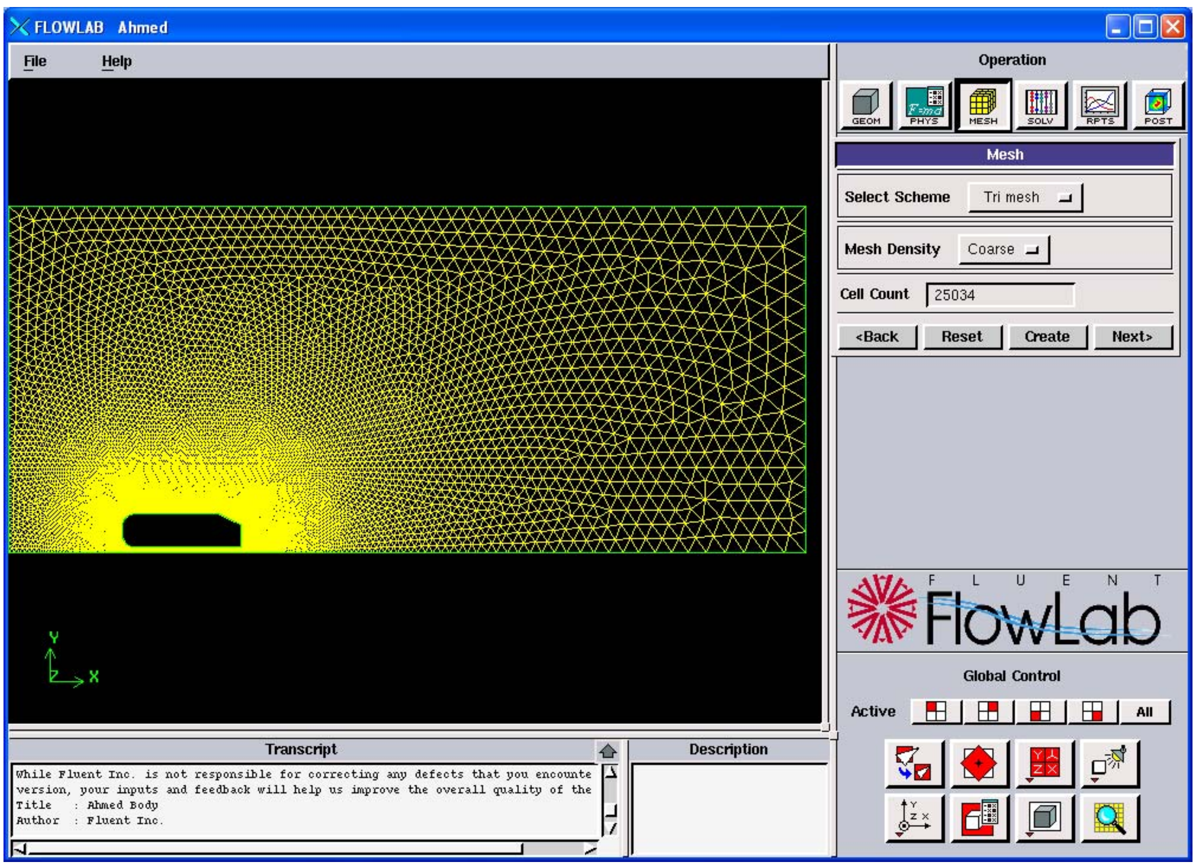
Reset | OK

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

Step 3: (Mesh)

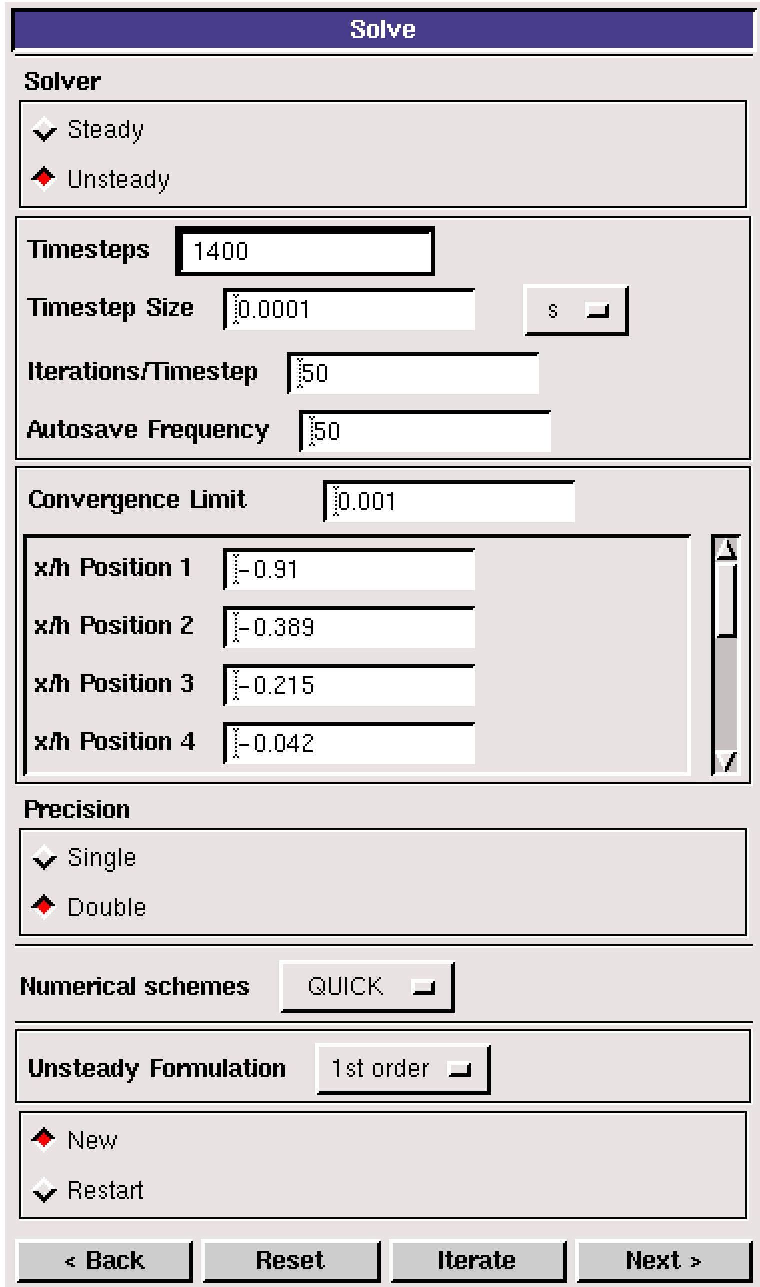


Unstructured Tri. Coarse Meshes will be used in this lab. No manual meshing is available for this lab due to the complexity of the geometries. After you create the mesh, you should zoom in regions close to the Ahmed car body and in the wake of the Ahmed car body and think about where the mesh is refined and why. Click <<**Create**>> to generate the whole mesh, as per below an example:



Step 4: (Solve)

In this Lab, only QUICK scheme (2^{nd} order upwind biased) will be used.

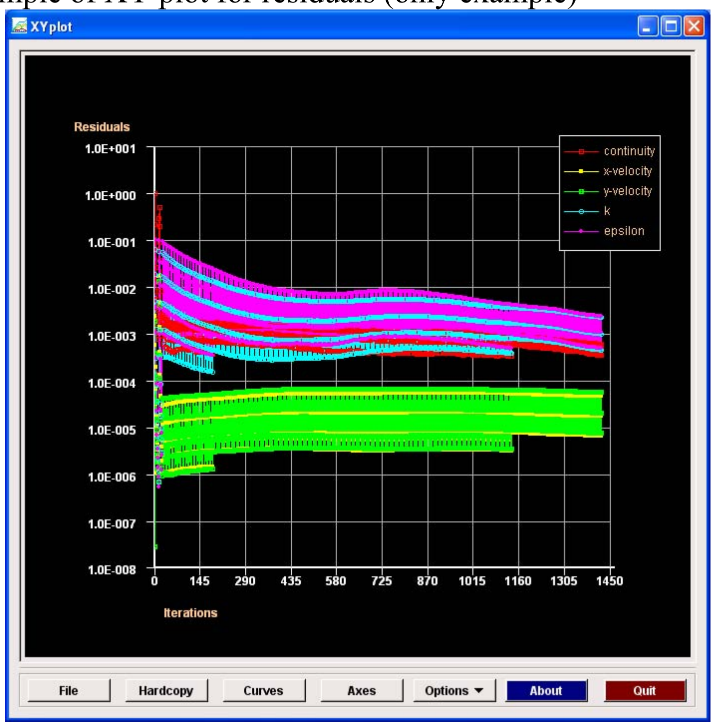


The flow is unsteady, so turn on the "**Unsteady**" option (default setup). Specify the time steps to be **1400.** Time step size is $\mathbf{10^{-4}}$. Iterations/timestep is the maximum iterations number for each time step. Specify that to be 50 (note: iterations for each time step will be stopped when either the maximum iterations number or the convergent limit is reached). "**Autosave frequency**" is to save solutions by skipping certain time steps so finally the "saved" solutions at different time steps can be used to generating animations to visualize the development flow field. "**Convergent limit**" is set to be 0.001, considering a lower value could cause tremendous increase of computational costs.

10 axial positions (x/h) can be specified (use the following values:-0.910, -0.389, -0.215, -0.042, 0.132, 0.306, 0.653, 1.000, 1.521, 2.215). These positions will be used to plot time-averaged axial

velocity and Turbulent Kinetic Energy and be compared with EFD data. Choose "**Double precision**", "**Quick scheme**" for spatial derivatives and 1st order for unsteady time integration. Here, "Quick scheme" is a 2nd order upwind biased scheme. Use <<**New**>> calculation for this Lab. Then click <<**Iterate**>> and FlowLab will begin the calculation, whenever you see the window, "**Solution Converged**". Click <<**OK**>>.

The following is an example of XY plot for residuals (only example)



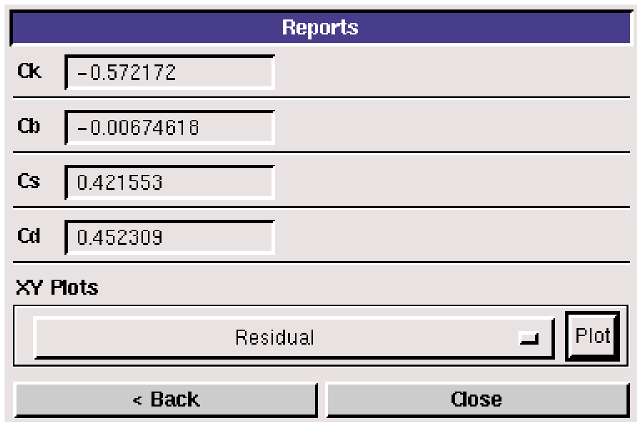
Step 5: (Reports)

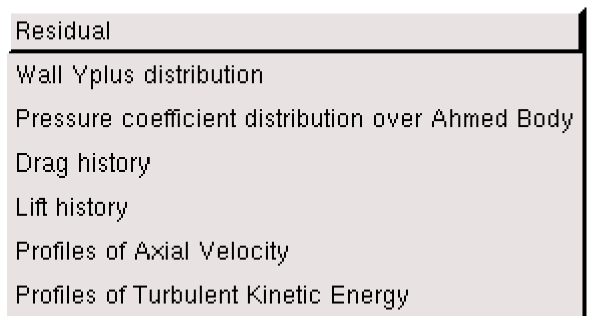
Time averaged drag force F_D is found by integrating surface pressure and the shear stress, the corresponding drag coefficient is computed by:

$$C_D = \frac{F_D}{\frac{1}{2}\rho U^2 A_x}$$

Where ρ is the fluid (air) density, U is the upstream velocity, A_x is the projected area of the Ahmed body in x direction. C_k, C_B, C_S, and C_D represent the drag coefficient at the nose, back, the rear slop and the total, respectively (read reference 1 for Ahmed car on class website for more details).

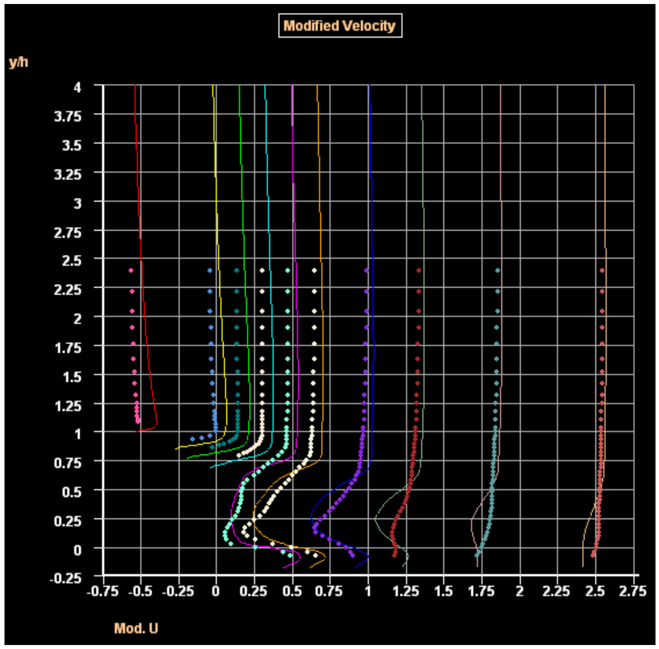
XY plots provide you the ability to plot the time-averaged quantities (note: time-averaged quantities are computed automatically by FlowLab after each simulation), including “residuals”, “Wall Yplus distribution”, “Pressure coefficient distribution”, “Drag history”, “Lift history”, “Profiles of axial velocity”, and “Profiles of TKE”.





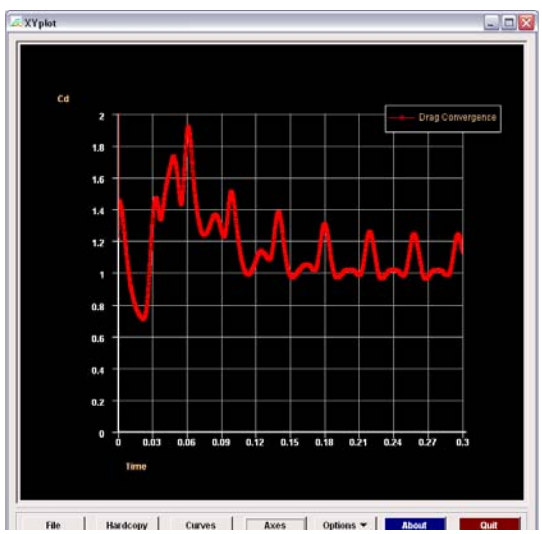
In this Lab, CFD results will be compared with EFD data for slant angle equals 25 degrees. EFD data for axial velocity [(mean-x-velocity/120)+ (x/0.288)], Modified_u_slant25.xy, can be downloaded from class website.

The following figure shows an example for axial velocity, for both EFD and CFD using k-ε model (your results could be different from it).



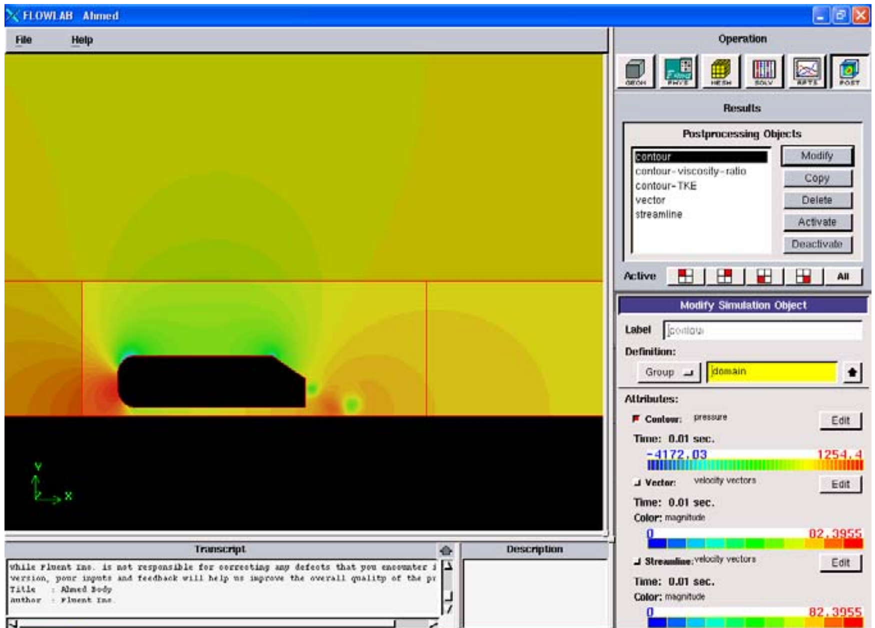
It is possible to modify the style of the curves by clicking "curves", selecting a curve with the right button and the clicking on "change style". For this lab, it is recommended to use lines (without symbols) for CFD and symbols (solid circles) for EFD data.

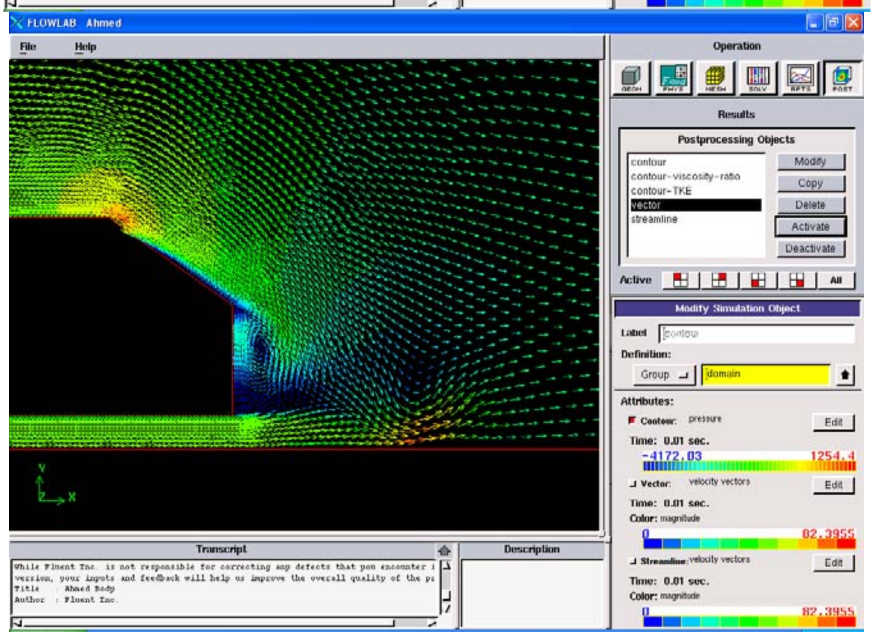
NOTE: 1. For time history of the total drag coefficient, you need change the range of x and y coordinates to appropriate values. One example is shown below.

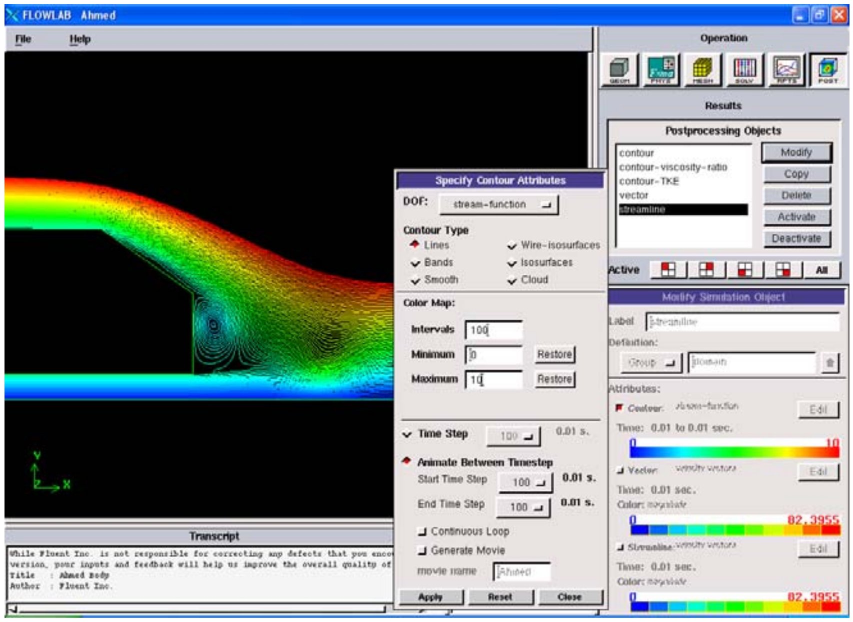


Step 6: (Post-processing)

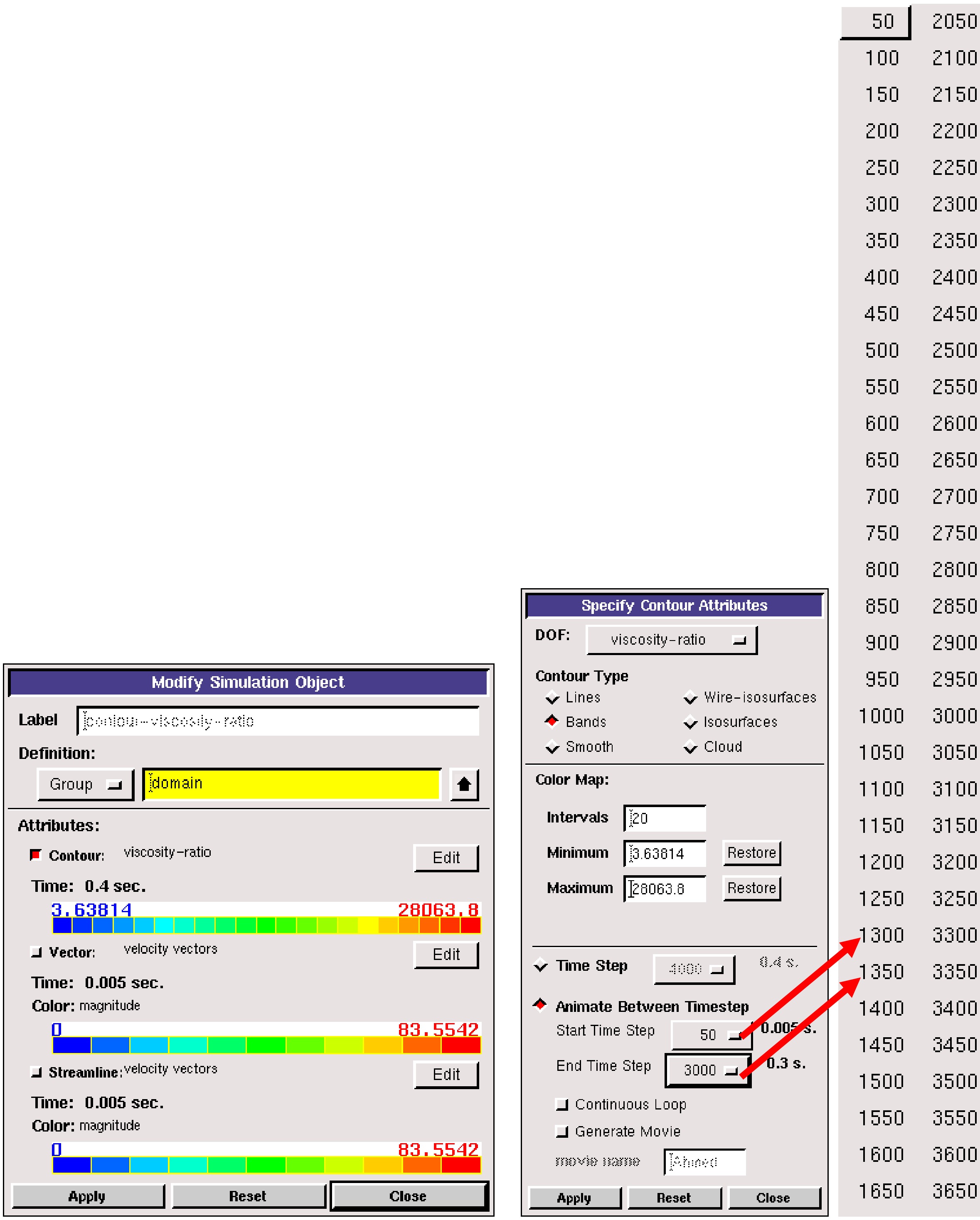
Use the "contour", "vector" buttons to show pressure contours and velocity vectors. The following figures are some sample results (contour of pressure, velocity vectors, and streamlines). To plot streamlines, you can use "lines" for "stream function" under either "contour" or "streamlines" with appropriate ranges of maximum and minimum values.


FLOWLAB Ahmed
File
Help
Operation
Results
Postprocessing Objects
contour
contour-viscosity-ratio
contour-TKE
vector
streamline
Modify
Copy
Delete
Activate
Deactivate
Active
All
Modify Simulation Object
Label
Definition:
Group
domain
Attributes:
Contour: pressure
Edit
Time: 0.01 sec.
-4172.03
1254.4
Vector: velocity vectors
Edit
Time: 0.01 sec.
Color: magnitude
0
82.3955
Streamline: velocity vectors
Edit
Time: 0.01 sec.
Color: magnitude
0
82.3955
Transcript
Description
While Fluent Inc. is not responsible for correcting any defects that you encounter i
version, your inputs and feedback will help us improve the overall quality of the p
Title : Ahmed Body
Author : Fluent Inc.


FLOWLAB Ahmed
File
Help
Operation
Results
Postprocessing Objects
contour
contour-viscosity-ratio
contour-TKE
vector
streamline
Modify
Copy
Delete
Activate
Deactivate
Active
All
Modify Simulation Object
Label
Definition:
Group
domain
Attributes:
Contour: pressure
Edit
Time: 0.01 sec.
-4172.03
1254.4
Vector: velocity vectors
Edit
Time: 0.01 sec.
Color: magnitude
0
82.3955
Streamline: velocity vectors
Edit
Time: 0.01 sec.
Color: magnitude
0
82.3955
Transcript
Description
While Fluent Inc. is not responsible for correcting any defects that you encounter i
version, your inputs and feedback will help us improve the overall quality of the p
Title : Ahmed Body
Author : Fluent Inc.



To create animations for contour of viscosity ratio, activate the contour plot for viscosity ratio, and click <<**Edit**>>, then turn on "animate between time step" and specify the "start time step" and "end time step". Click <<**Apply**>>, the animation will be shown in the window, you can use <<**Alt+print screen**>> to save several frames and post it into word. By now, FlowLab can not automatically create "*avi" file, so do NOT click the <<**Generate movie**>> button. You can use similar approach to create animations for streamlines.



Modify Simulation Object
Label
contour-viscosity-ratio
Definition:
Group
domain
Attributes:
Contour: viscosity-ratio
Edit
Time: 0.4 sec.
3.63814
28063.8
Vector: velocity vectors
Edit
Time: 0.005 sec.
Color: magnitude
0
83.5542
Streamline: velocity vectors
Edit
Time: 0.005 sec.
Color: magnitude
0
83.5542
Apply
Reset
Close
Specify Contour Attributes
DOF: viscosity-ratio
Contour Type
Lines
Wire-isosurfaces
Bands
Isosurfaces
Smooth
Cloud
Color Map:
Intervals 20
Minimum 3.63814 Restore
Maximum 28063.8 Restore
Time Step 4000 0.4 s.
Animate Between Timestep
Start Time Step 50 0.005 s.
End Time Step 3000 0.3 s.
Continuous Loop
Generate Movie
movie name Ahmed
Apply
Reset
Close
50 2050
100 2100
150 2150
200 2200
250 2250
300 2300
350 2350
400 2400
450 2450
500 2500
550 2550
600 2600
650 2650
700 2700
750 2750
800 2800
850 2850
900 2900
950 2950
1000 3000
1050 3050
1100 3100
1150 3150
1200 3200
1250 3250
1300 3300
1350 3350
1400 3400
1450 3450
1500 3500
1550 3550
1600 3600
1650 3650

4. Exercises

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

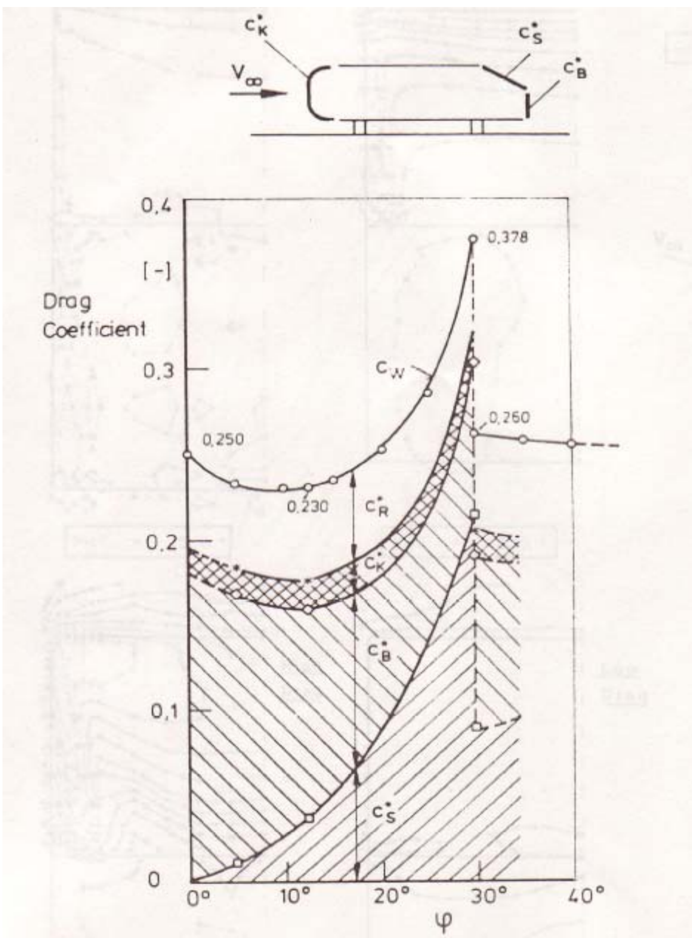
Simulation of Turbulent Flow over the Ahmed Body

- **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 flows over Ahmed body (slant angle=25 degree):

Use 25 degrees for slant angle to create the geometry, create "**Tri Coarse**" mesh, and run the simulation with time steps 1400. **NOTE: This simulation could take up to 3 hours.**

1.1. Fill in the table for the four drag coefficients and compute the relative error between CFD and EFD (Ahmed data), EFD data for C_k, C_B, and C_s can be found from the figure below. Where $C_k = C_k^*$, $C_B = C_B^*$, and $C_s = C_s^*$. The definitions of the drag coefficients are: C_k is the forebody pressure drag coefficient, C_B is the vertical based pressure drag coefficient, C_R is the friction drag coefficient, C_s is the slant surface pressure drag coefficient, and $C_w = C_D$ is the total drag coefficient. So, $C_w = C_D = C_S + C_B + C_k + C_R$



	C_k	C_B	C_S	C_D
Ahmed (EFD)				0.289
k-e				
Error (%)				

1.2. **Questions:**

1.2.1. Do you observe separations in the wake region (use streamlines)? If yes, where is the location of separation point?

1.2.2. What is the Strouhal number based on the shedding frequency (C_D vs. time), the height of the Ahmed body and the inlet velocity? Note: the shedding frequency f=1/T where T is the typical period of the oscillation of C_D that can be evaluated using the peaks between $0.1<time<0.14$.

- **Figures to be saved:** 1. XY plots for residual history, axial velocity vs. x/h (with EFD), TKE vs. x/h and time history of drag coefficient, 2. Contour of pressure, contour of axial velocity and velocity vectors, 3. 3 or 4 snapshots of animations for turbulent-viscosity-ratio and streamlines (hints: you can use <<**Alt+print Screen**>> during the play of the animations).
- **Data to be saved:** the above table with values.