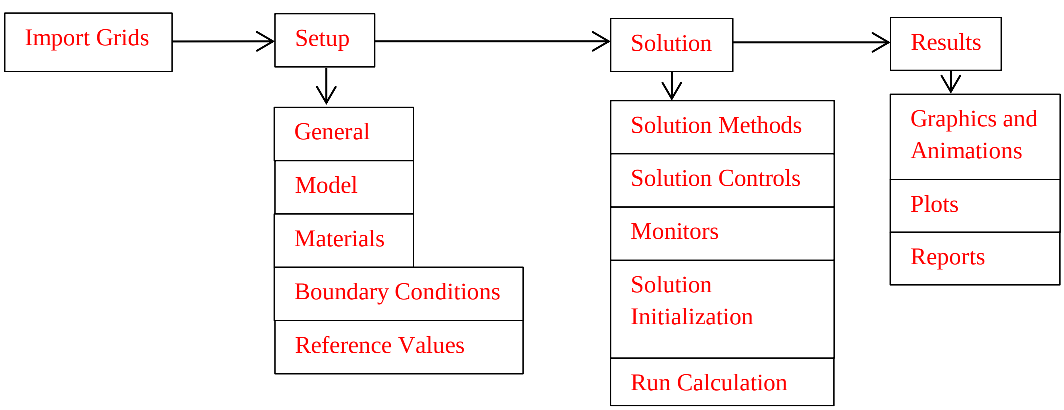
Simulation of Turbulent Flow over the Ahmed Body

58:160 Intermediate Mechanics of Fluids
CFD LAB 4

By Timur K. Dogan, Michael Conger, Maysam Mousaviraad, and Fred Stern
IIHR-Hydroscience & Engineering
The University of Iowa
C. Maxwell Stanley Hydraulics Laboratory
Iowa City, IA 52242-1585

1. Purpose

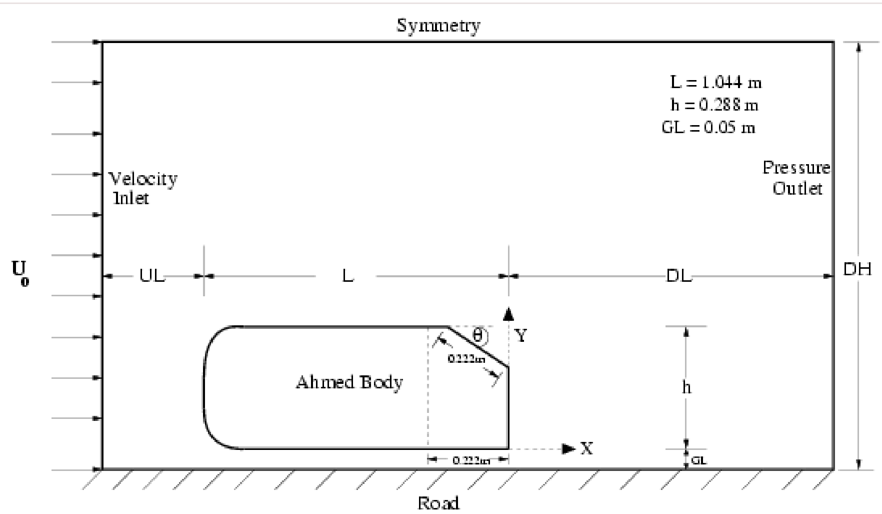
The Purpose of CFD Lab 4 is to simulate **unsteady turbulent** flows over the Ahmed body following the "CFD process" by an interactive step-by-step approach and conduct verifications using CFD Educational Interface (FlowLab 1.2). Students will have "hands-on" experiences using FlowLab to **predict drag 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 ANSYS

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.

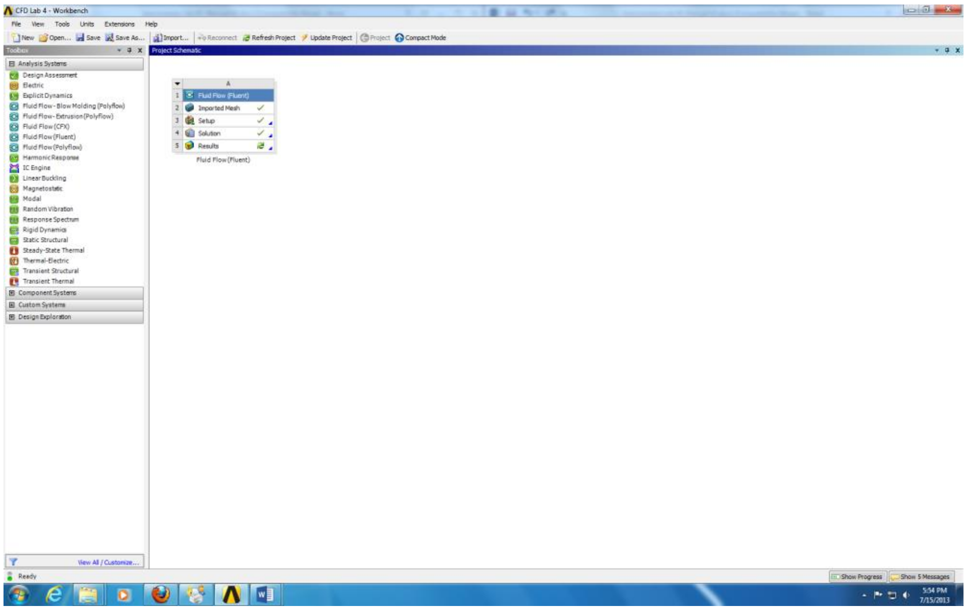


In CFD Lab4, all EFD data for turbulent airfoil flow in this Lab will be provided by the TA and saved on the Fluids Lab computers.

3. Open ANSYS Workbench Template

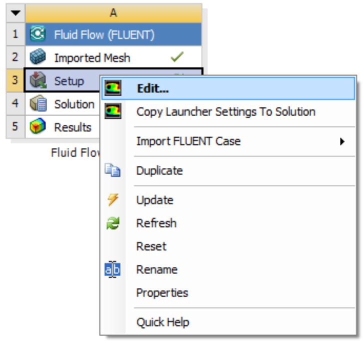
3.1. Download CFD Lab 4 Template from class website.

3.2. Open Workbench Project Zip file simply by double clicking file. This file contains all the systems that must be solved for CFD Lab 4.

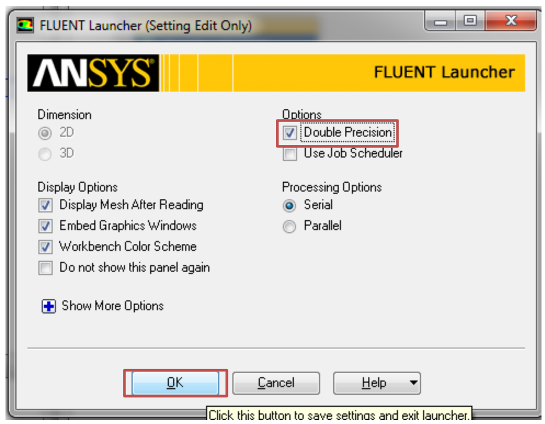


4. Setup

4.1. Right click Setup and select Edit

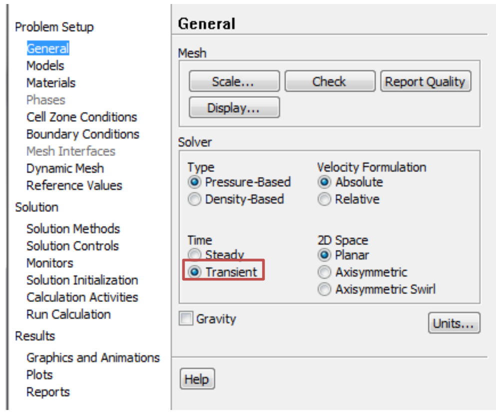


4.2. Select double precision and click Ok

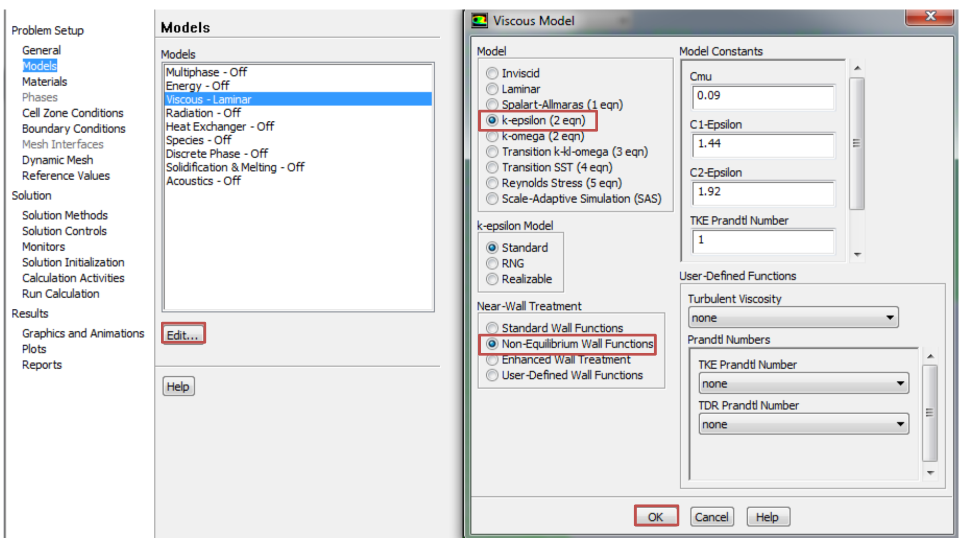


5. Problem Setup

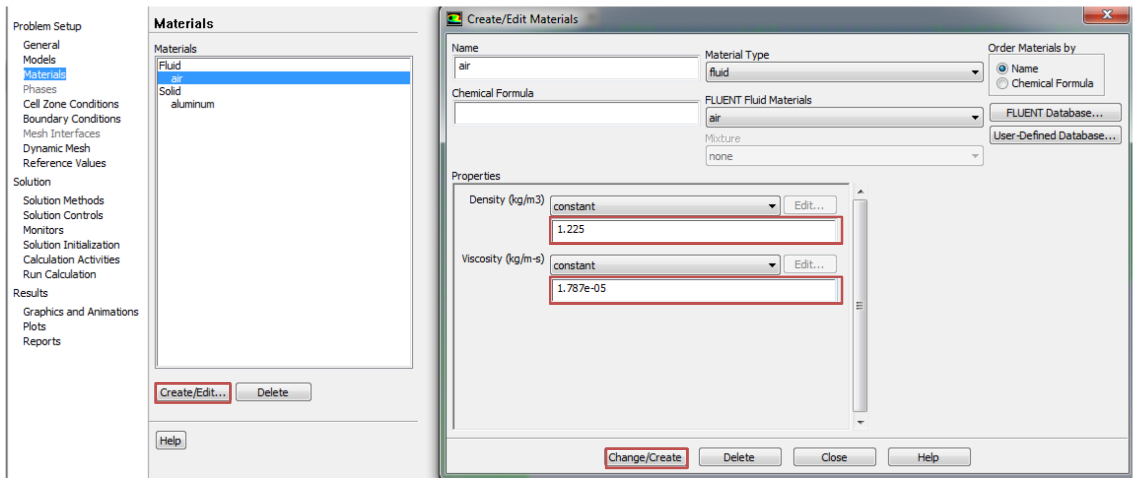
5.1. Problem Setup > General. Change solver to transient as per below.



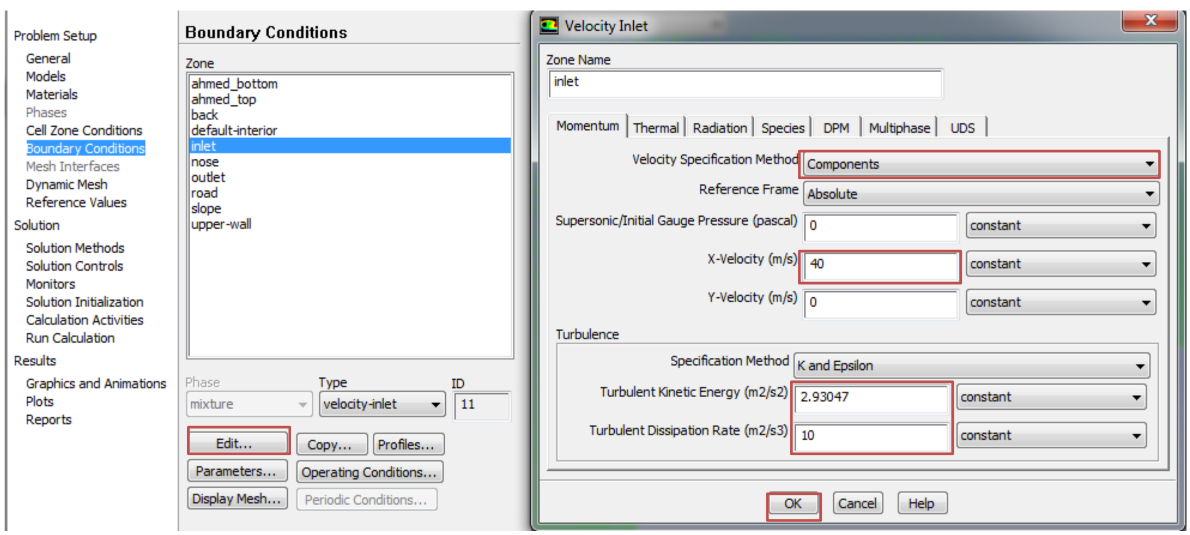
5.2. Problem Setup > Models > Viscous > Edit. Change the turbulent model and near-wall treatment as per below.



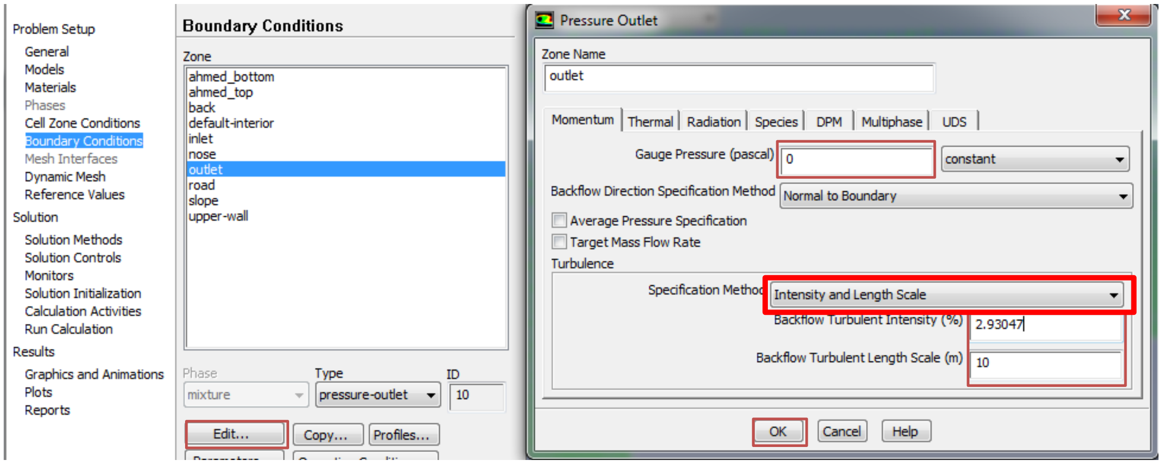
5.3. Problem > Materials > Fluid > air > Create/Edit. Change the air density and viscosity as per below and click Change/Edit then close the window.



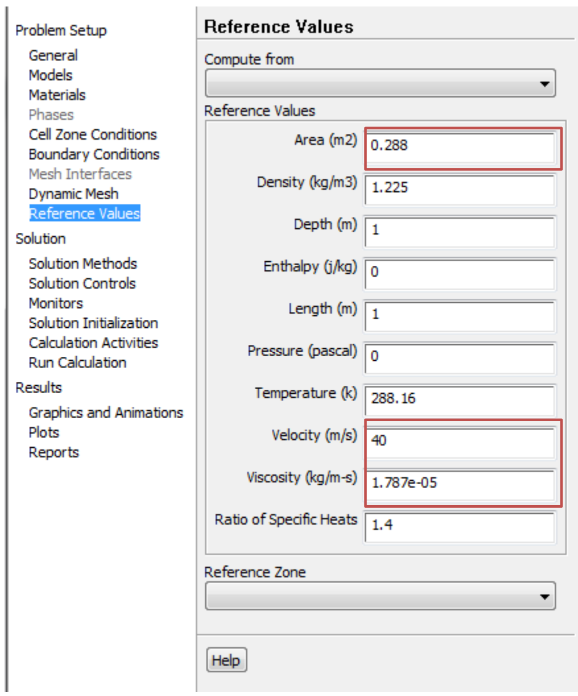
5.4. Problem Setup > Boundary Conditions > inlet > Edit. Change the inlet boundary conditions as per below and click OK.



5.5. Problem Setup > Boundary Conditions > Zone > outlet > Edit. Change the outlet boundary condition as per below and click OK.

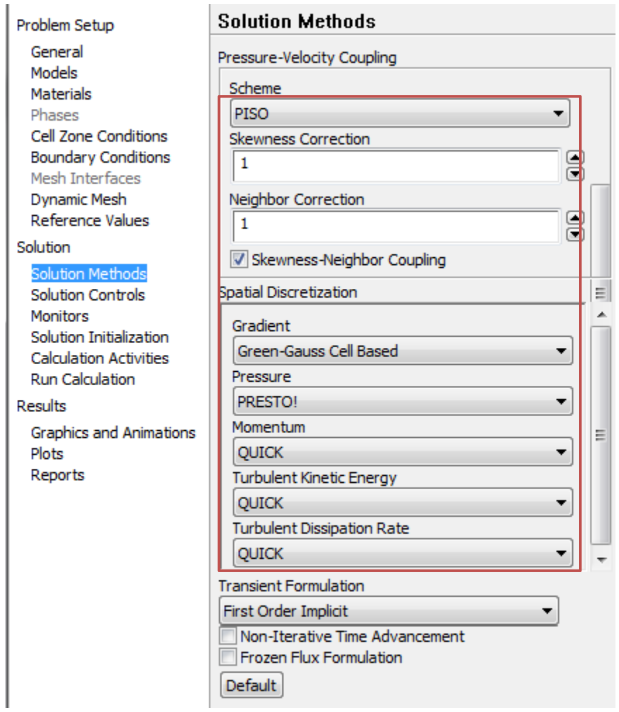


5.6. Problem Setup > Reference Values. Change the reference values as per below.

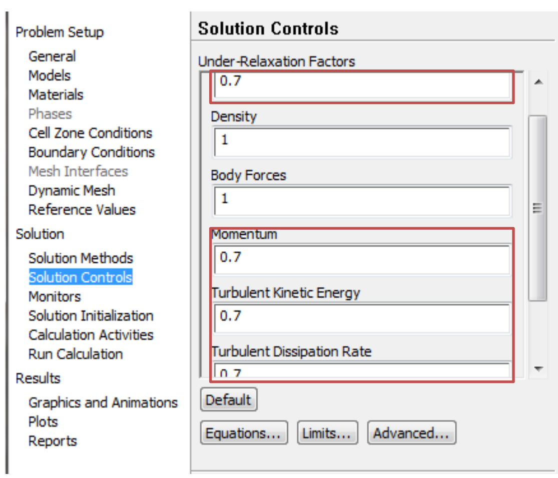


6. Solution

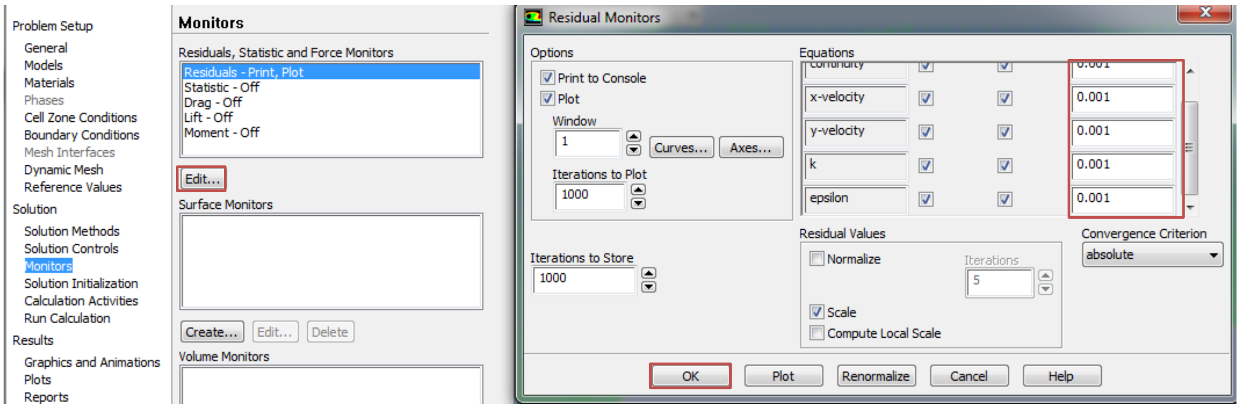
6.1. Solution > Solution Methods. Change solutions methods as per below.



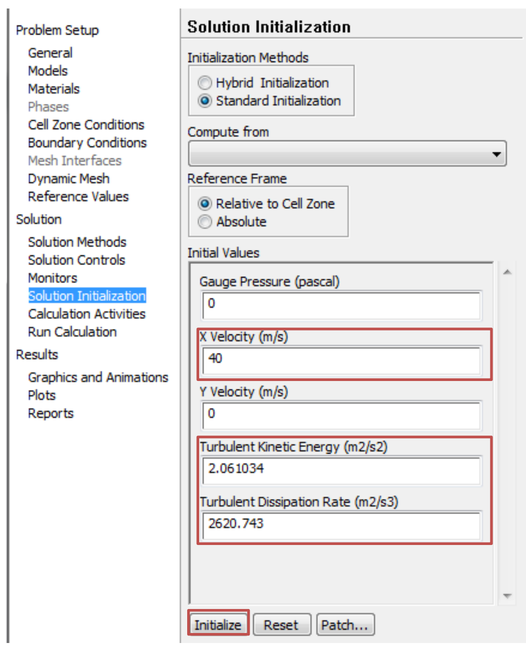
6.2. Solution > Solution Controls. Change under-relax factors for pressure, momentum, turbulent kinetic energy, and turbulent dissipation rate to 0.7.



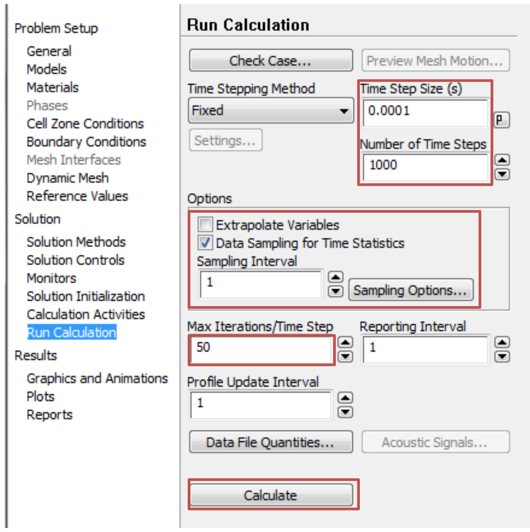
6.3. Solution > Monitors > Edit. Change the convergence criterion and click OK.

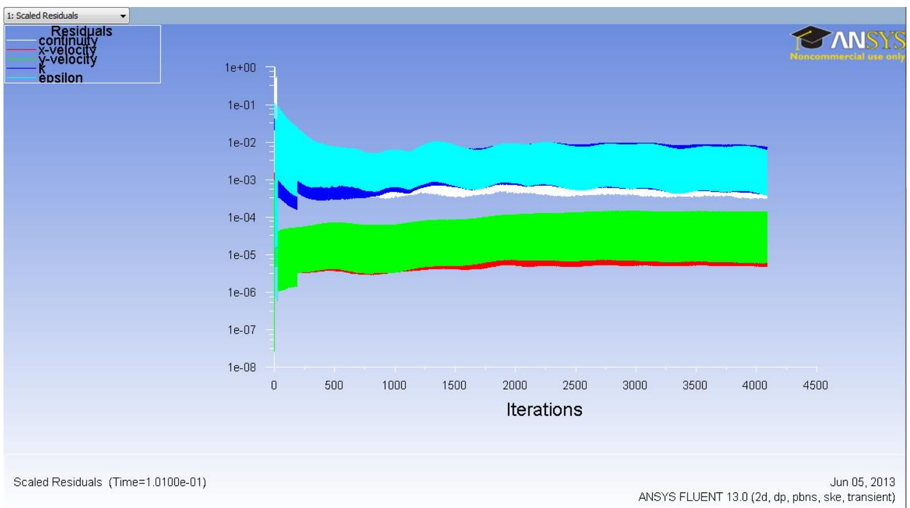


6.4. Solutions > Solution Initialization. Change x-velocity and turbulent parameters as per below.



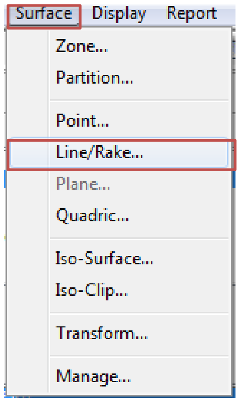
6.5. Solution > Run Calculation. Change parameters as per below and click Calculate.

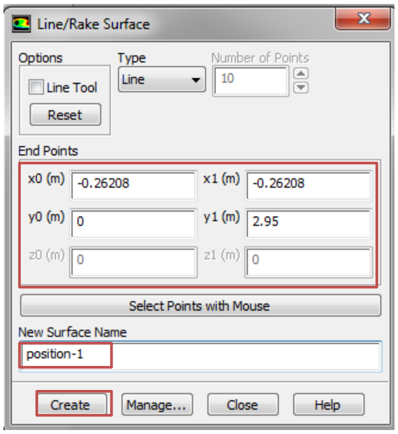

Problem Setup
General
Models
Materials
Phases
Cell Zone Conditions
Boundary Conditions
Mesh Interfaces
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
Solution Initialization
Calculation Activities
Run Calculation
Results
Graphics and Animations
Plots
Reports
Run Calculation
Check Case...
Preview Mesh Motion...
Time Stepping Method
Fixed
Settings...
Time Step Size (s)
0.0001
Number of Time Steps
1000
Options
Extrapolate Variables
Data Sampling for Time Statistics
Sampling Interval
1
Sampling Options...
Max Iterations/Time Step
50
Reporting Interval
1
Profile Update Interval
1
Data File Quantities...
Acoustic Signals...
Calculate


1: Scaled Residuals
Residuals
continuity
x-velocity
y-velocity
k
epsilon
1e+00
1e-01
1e-02
1e-03
1e-04
1e-05
1e-06
1e-07
1e-08
0
500
1000
1500
2000
2500
3000
3500
4000
4500
Iterations
Scaled Residuals (Time=1.0100e-01)
Jun 05, 2013
ANSYS FLUENT 13.0 (2d, dp, pbns, ske, transient)

7. Results

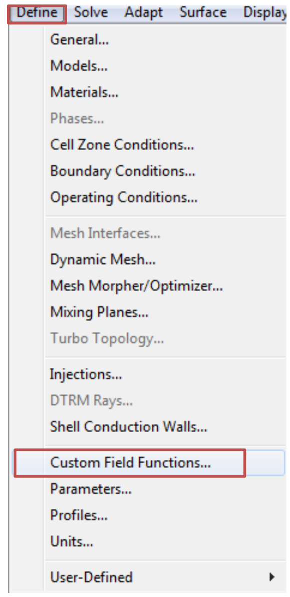
7.1. Surface > Line/Rake. Create 10 lines at the locations given at the table below.

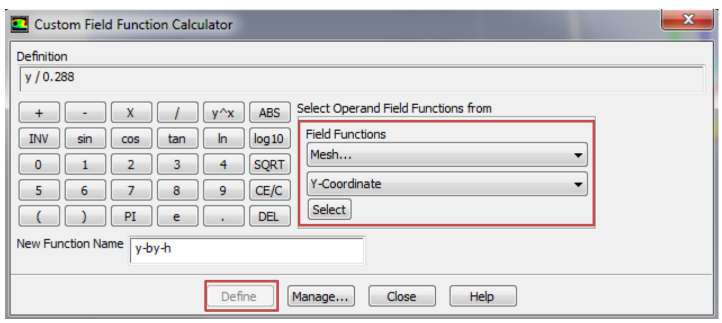




Surface Name	x0	y0	x1	y1
Position-1	-0.26208	0.00	-0.26208	2.95
Position-2	-0.11200	0.00	-0.11200	2.95
Position-3	-0.06192	0.00	-0.06192	2.95
Position-4	-0.01209	0.00	-0.01209	2.95
Position-5	0.03801	-0.05	0.03801	2.95
Position-6	0.08812	-0.05	0.08812	2.95
Position-7	0.18806	-0.05	0.18806	2.95
Position-8	0.28800	-0.05	0.28800	2.95
Position-9	0.43800	-0.05	0.43800	2.95
Position-10	0.63790	-0.05	0.63790	2.95

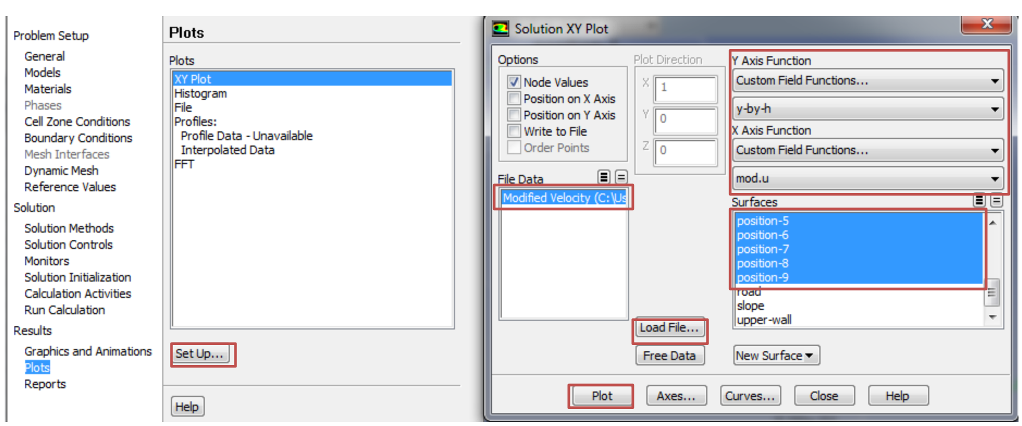
7.2. Define > Custom Field Functions. Create custom field functions and click Define. You will need to create two custom field functions shown in the table below.

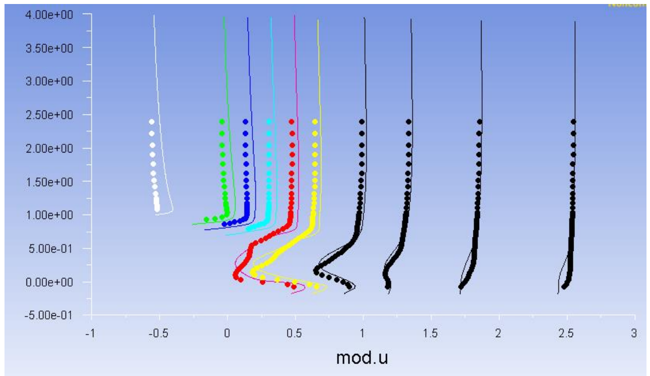




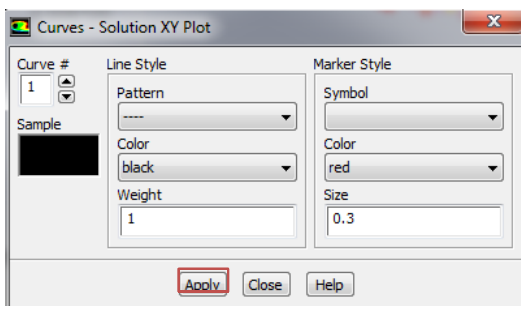
Function Name	Definition
y-by-h	y/0.288
Mod. U	(mean-x-velocity/120)+(x/0.288)

7.3. Results > Plots > XY Plot > Set Up. Click load file and load the experimental data. Select the lines you created (position-1 through position-10) and experimental data then click Plot.

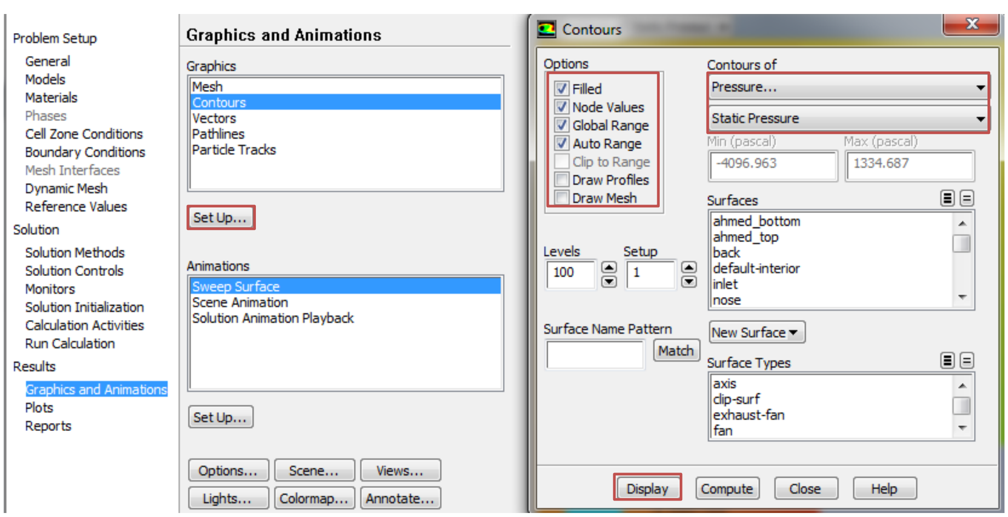


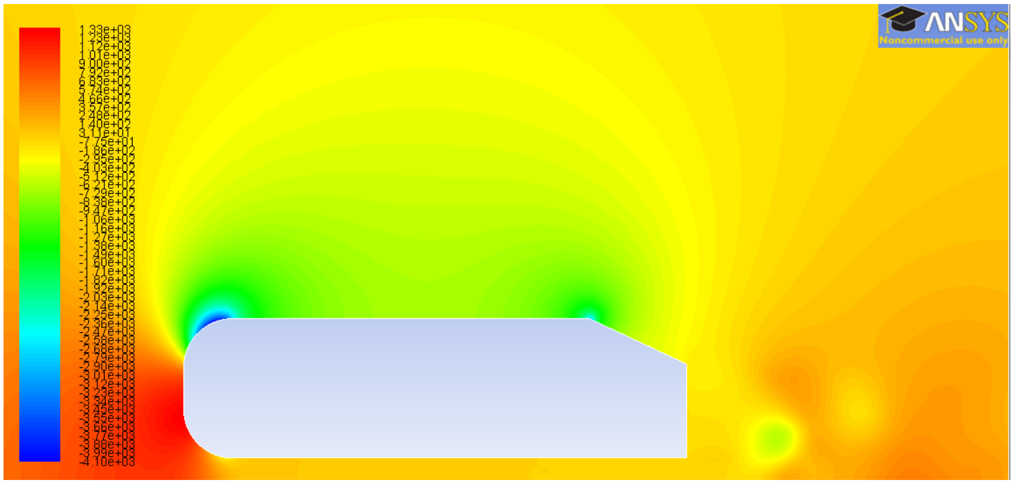


Note: You change the style and color of the data by clicking Curves button and changing the parameters below then clicking apply. Adjust Y axis maximum to 2.5 and minimum to -0.5.

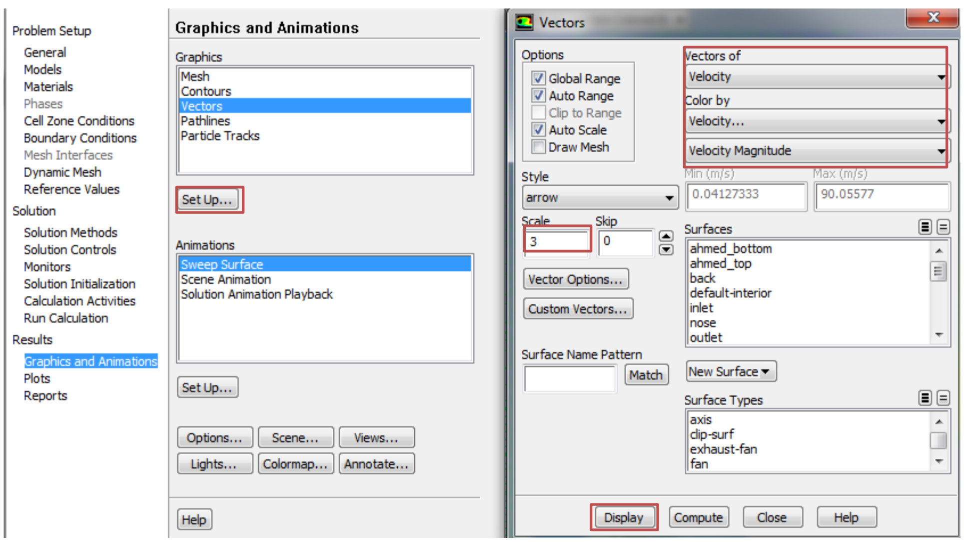


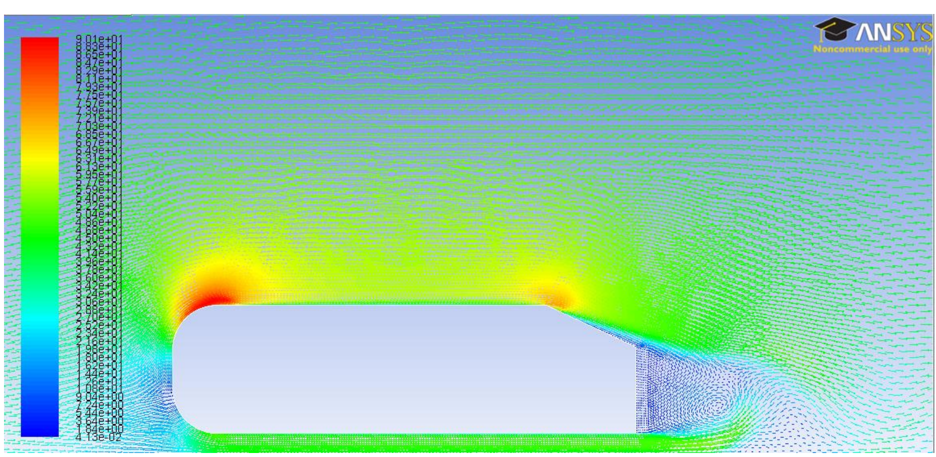
7.4. Results > Graphics and Animations > Contours. Change parameters as per below and click display.



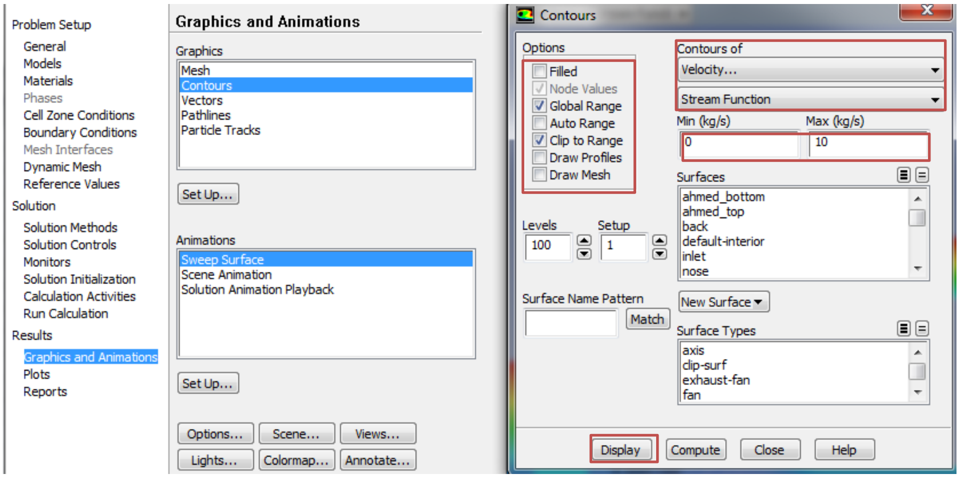


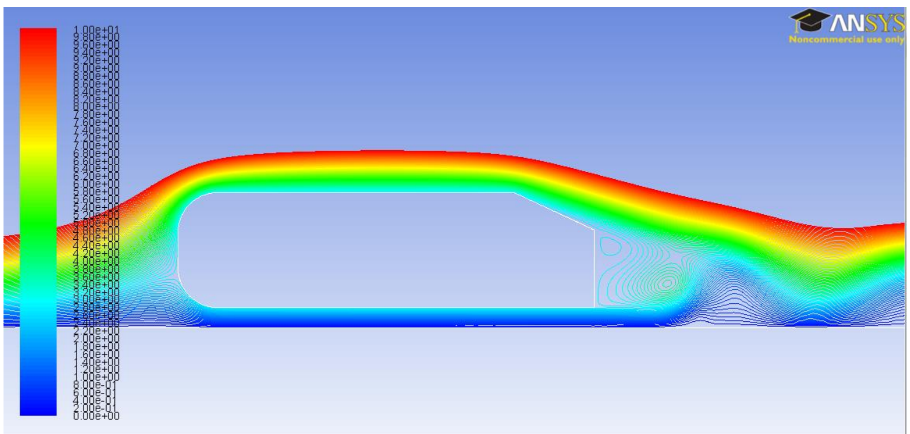
7.5. Results > Graphics and Animations > Vectors > Set Up. Change parameters as per below and click display.



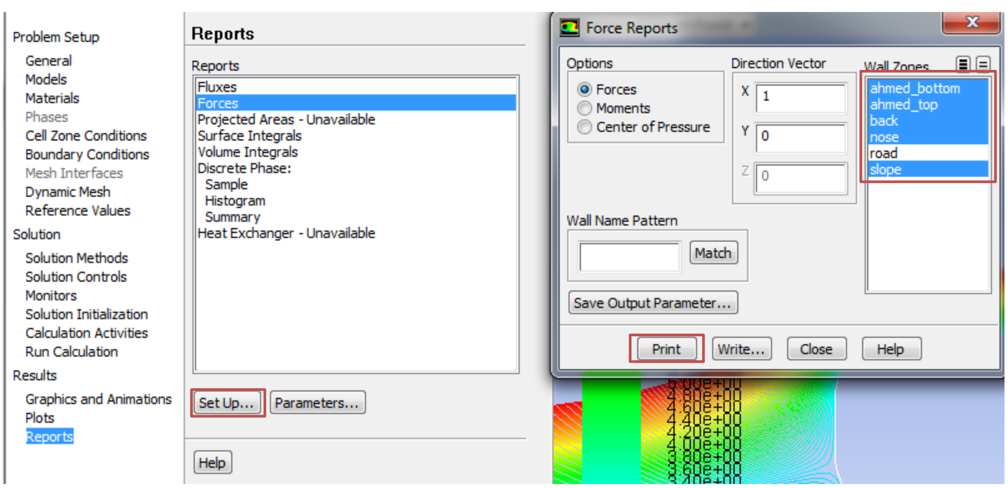


7.6. Results > Graphics and Animations > Contours > Set Up. Change parameters as per below and click Display.





7.7. Results > Reports > Forces > Setup. Change parameters as per below and click print.



```
>
Forces
                          Forces (n)
Zone                      Pressure                                     Viscous                                      Total
ahmed_bottom              (0 -7.1236193 0)                             (4.2638587 0 0)                              (4.2638587 -7.1236193 0)
ahmed_top                 (0 724.71536 0)                              (4.6776174 0 0)                              (4.6776174 724.71536 0)
back                      (27.573758 0 0)                              (0 0.021909745 0)                            (27.573758 0.021909745 (
nose                      (-16.074916 363.49982 0)                     (2.4533813 1.7017059 0)                      (-13.621535 365.20153 0)
slope                     (74.575871 159.92847 0)                      (0.11512858 -0.05368534 0)                   (74.691 159.87479 0)
------------------------- ---------------------------------------------  ---------------------------------------------  ------------------------
Net                       (86.074712 1241.02 0)                        (11.509986 1.6699303 0)                      (97.584698 1242.69 0)

Forces - Direction Vector (1 0 0)
                          Forces (n)                                     Coefficients
Zone                      Pressure        Viscous         Total          Pressure        Viscous         Total
ahmed_bottom              0               4.2638587       4.2638587      0               0.01510721      0.01510721
ahmed_top                 0               4.6776174       4.6776174      0               0.016573192     0.016573192
back                      27.573758       0               27.573758      0.097696141     0               0.097696141
nose                      -16.074916      2.4533813       -13.621535     -0.056954779    0.008692536     -0.048262243
slope                     74.575871       0.11512858      74.691         0.26422858      0.00040791025   0.26463649
------------------------- --------------- --------------- -------------- --------------- --------------- ---------------
Net                       86.074712       11.509986       97.584698      0.30496994      0.040780848     0.34575079
```

8. Exercises

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

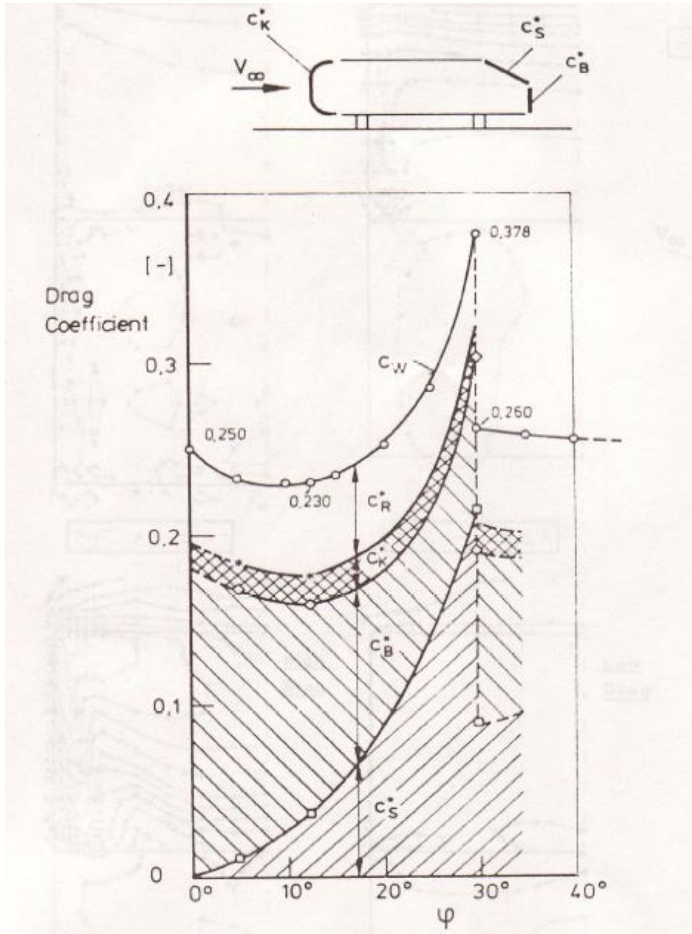
Simulation of Turbulent Flow over the Ahmed Body

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

8.1. 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.**

a. 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 (%)				

b. **Questions**:

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