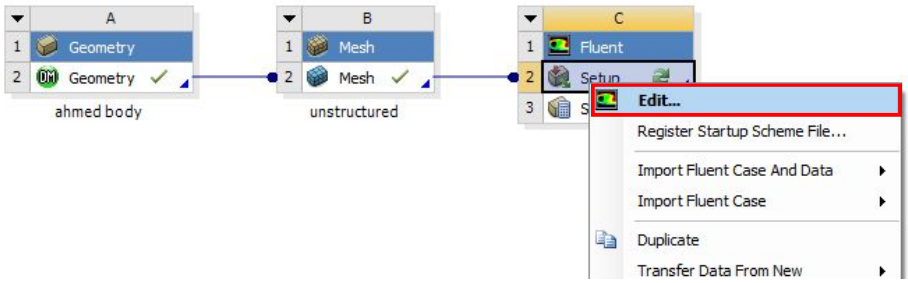
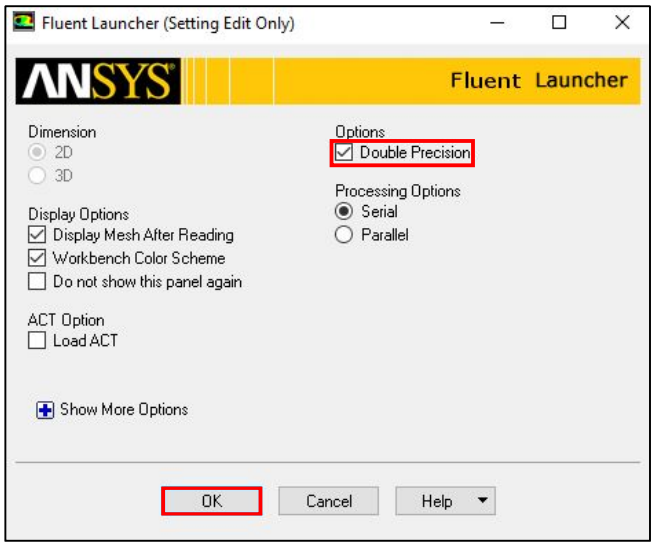
Numerical modeling of airflow over the Ahmed body

1. Setup

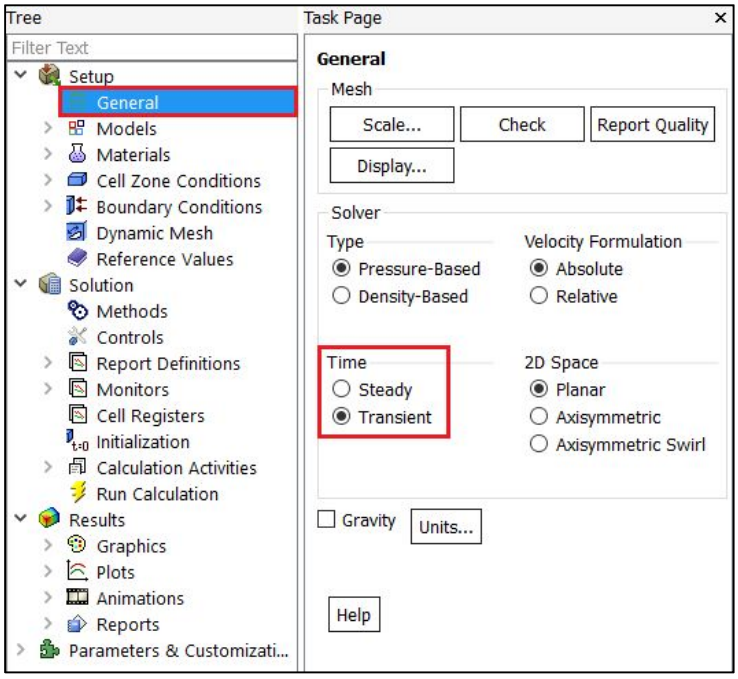
1.1. Right click **Setup** and select **Edit…**



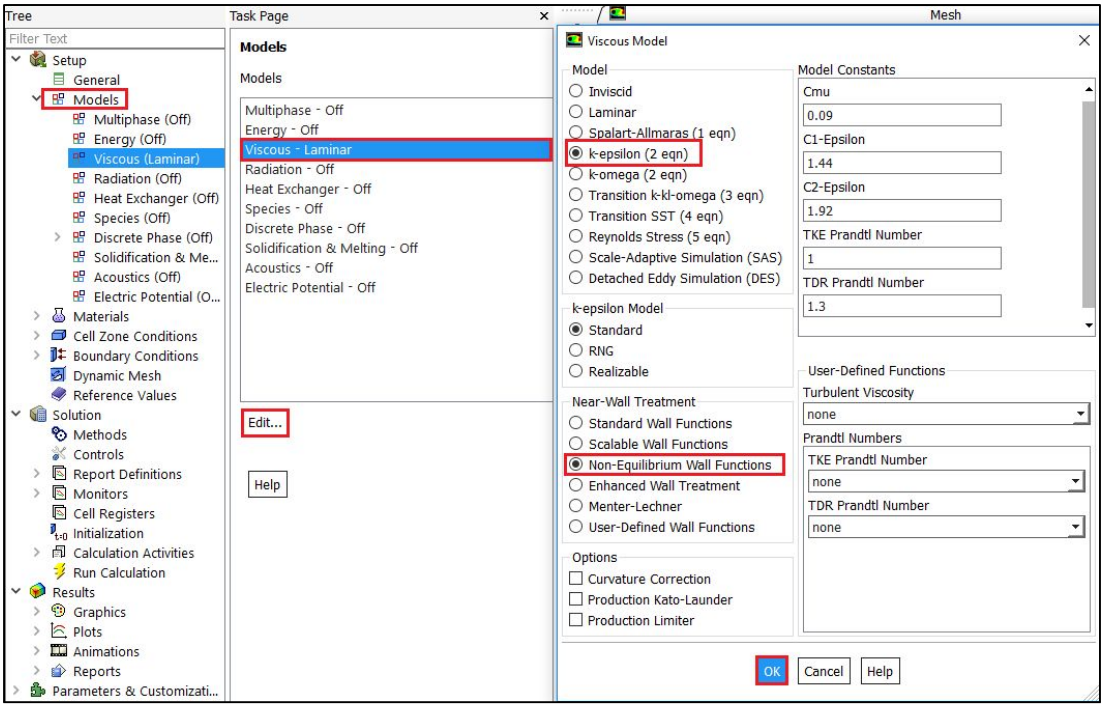
1.2. Select **Double Precision** and click **OK**.



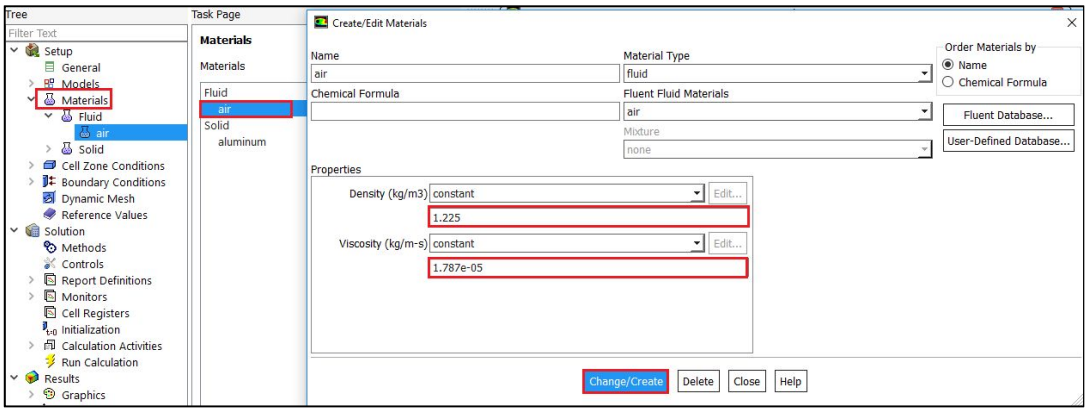
1.3. Setup > **General**. Change **Solver** to **Transient** as per below.



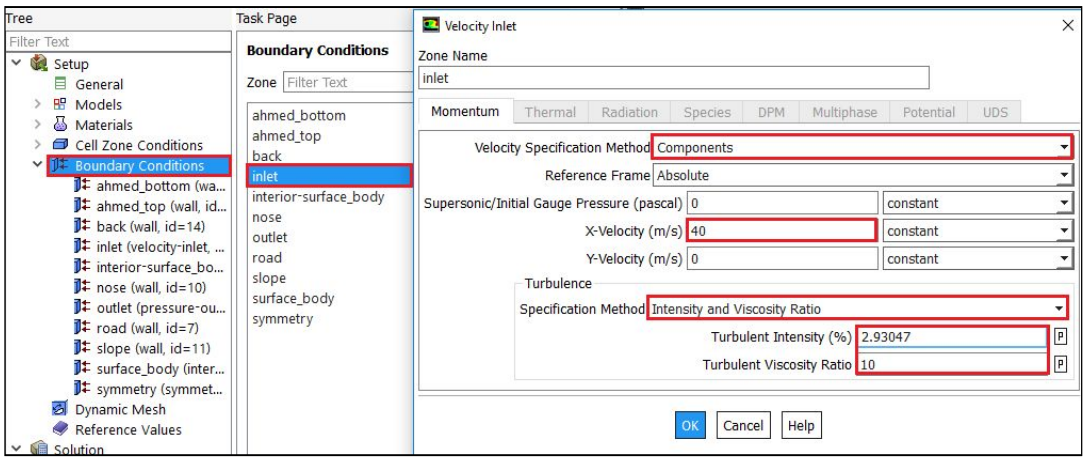
1.4. Setup > **Models** > **Viscous** > **Edit…** Change the turbulent model and near-wall treatment as per below.



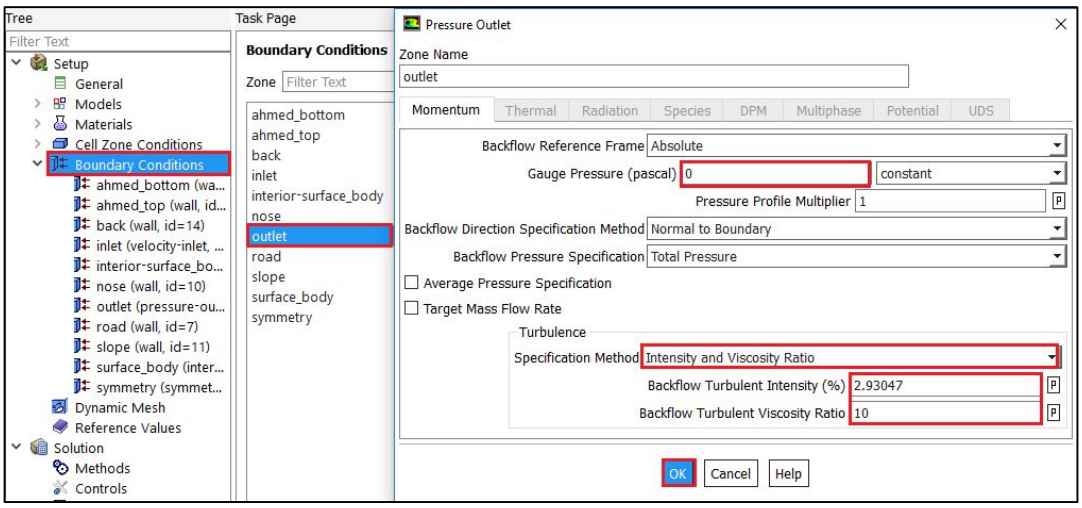
1.5. Setup > **Materials** > **Fluid** > **air** > **Create/Edit…** Change the air **Density** and **Viscosity** as per below and click **Change/Edit** then close the window.



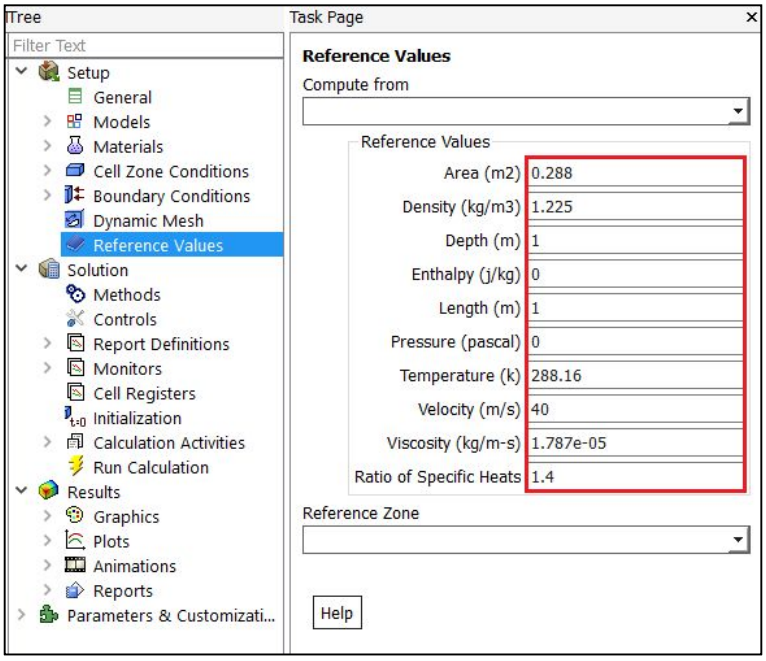
1.6. Setup > **Boundary Conditions** > **inlet** > **Edit…** Change the inlet boundary conditions as per below and click **OK**.



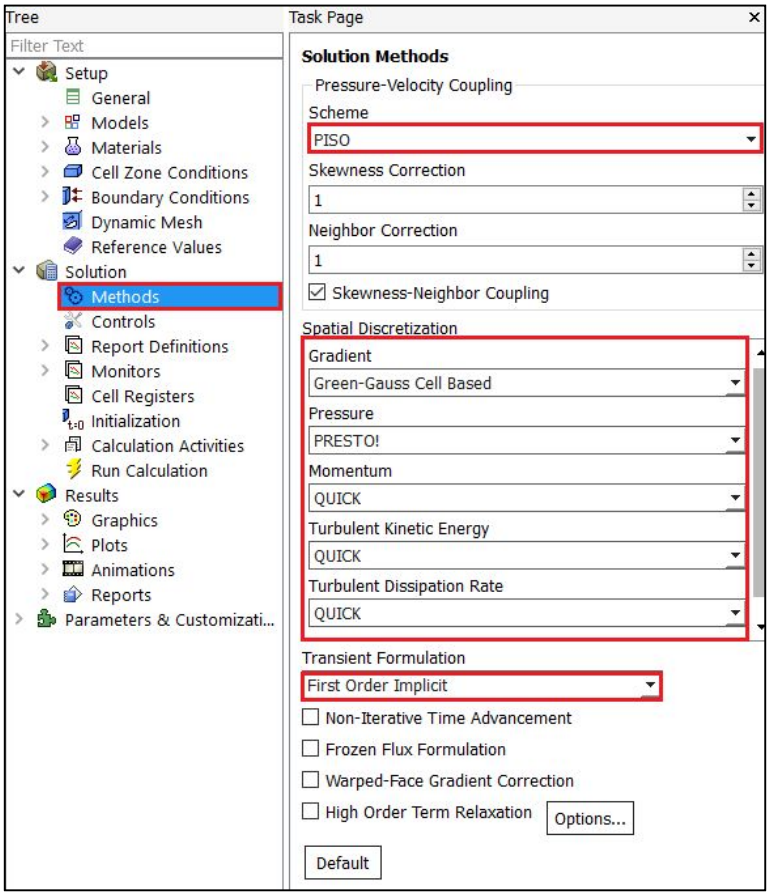
1.7. Setup > **Boundary Conditions** > **Zone** > **outlet** > **Edit…** Change the outlet boundary condition as per below and click **OK**.



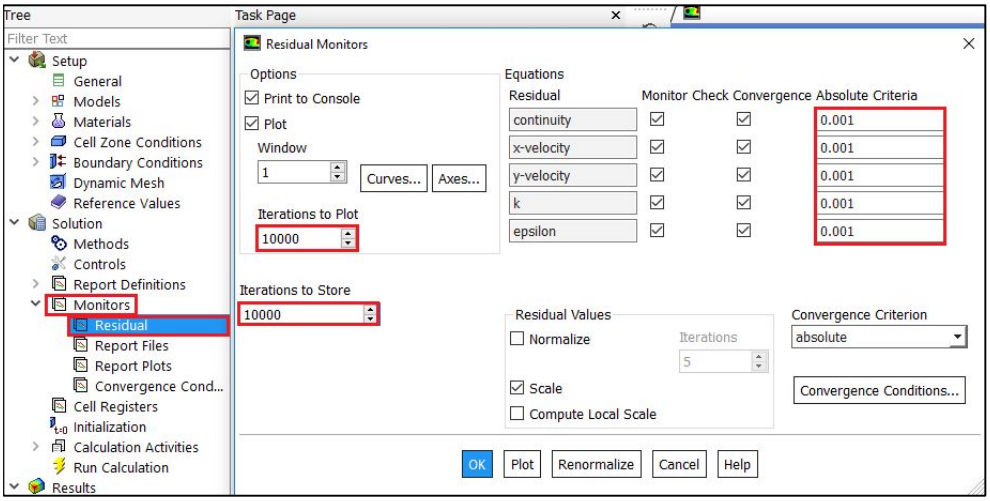
1.8. Setup > **Reference Values**. Change the reference values as per below.



1.9. Solution > **Methods**. Change solutions methods as per below.

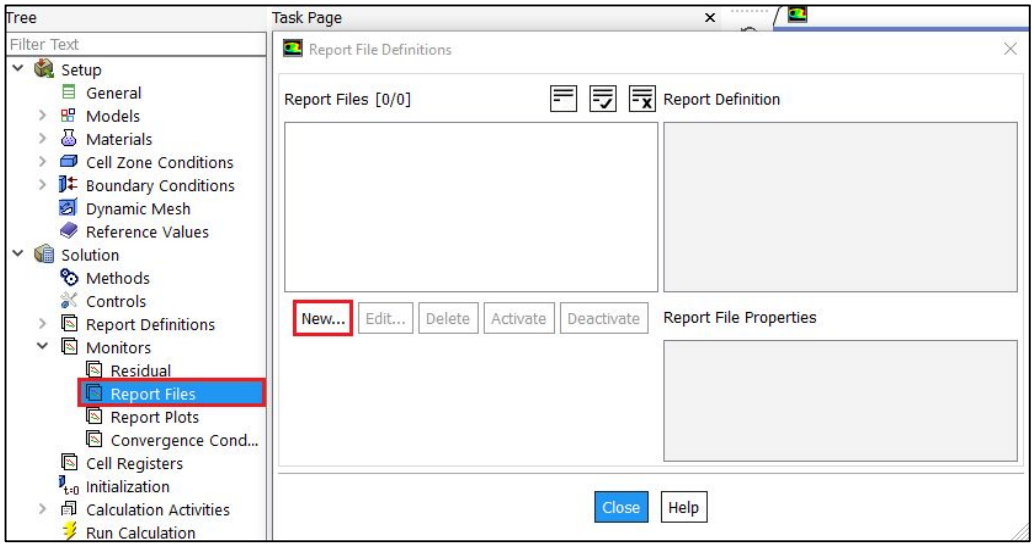


1.10. Solution > Monitors > Residuals. Change the parameters as per below and click ok.

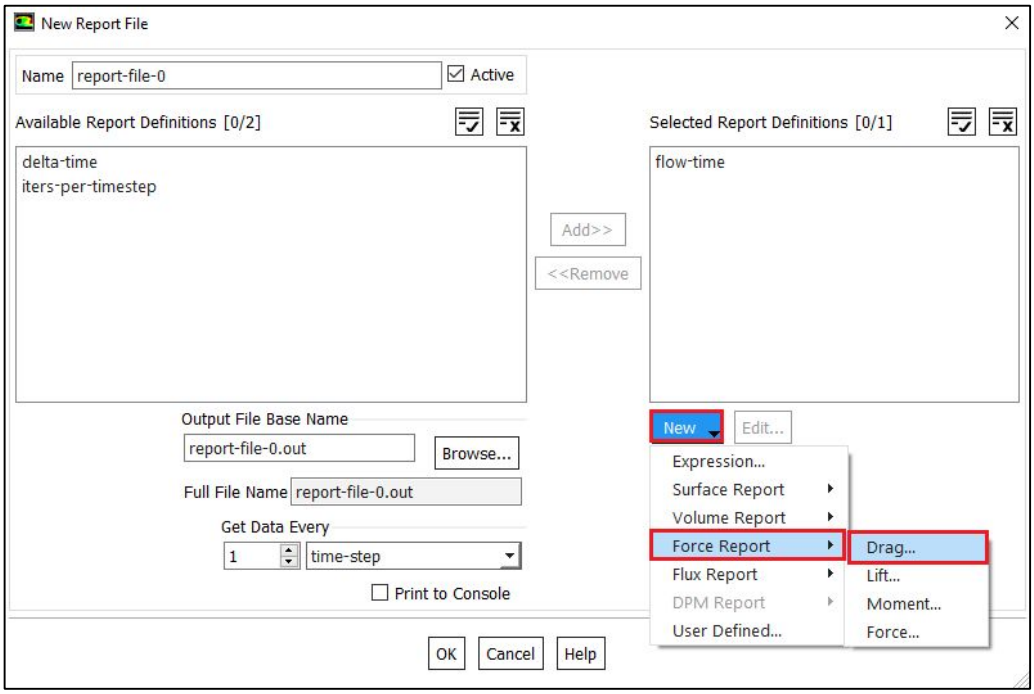


*Step 1.11~1.14 is for saving the time history file of the total drag coefficient.

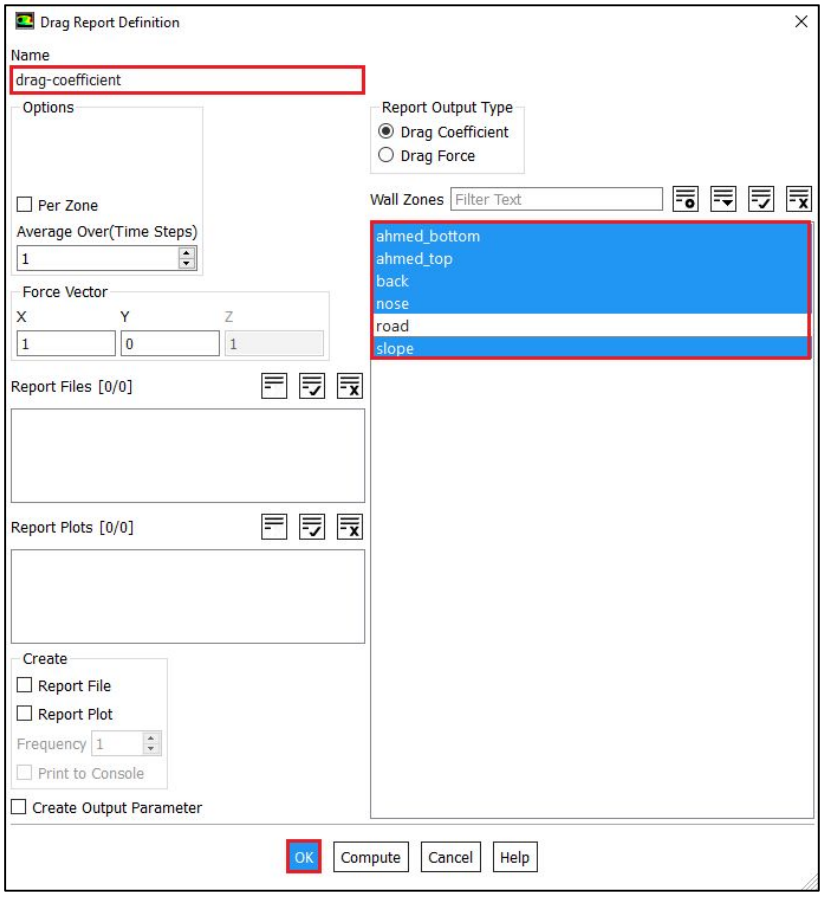
1.11. Solution > Monitors > Report Files > New…



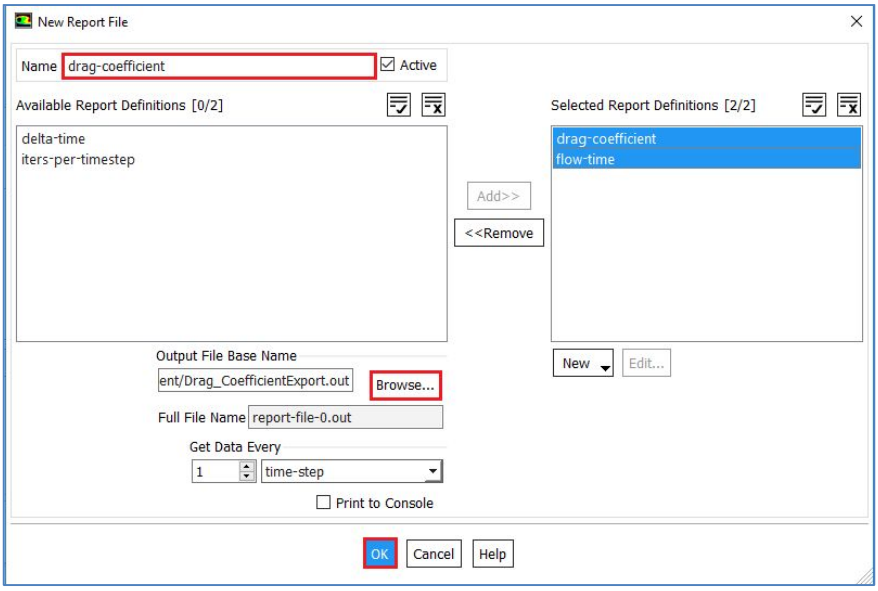
1.12. New > Force Report > Drag….



1.13. Change name, select wall zones as below and click **OK** to exit.

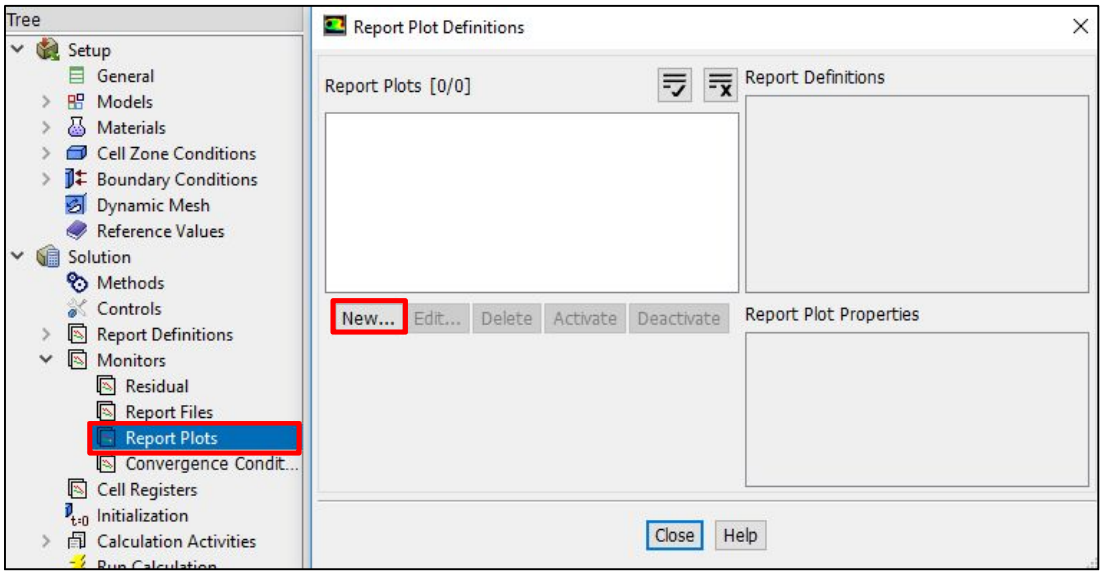


1.14. Change name and click **Browse** to locate the file. Click **OK** to exit. Exit the Report File Definition dialog as well.

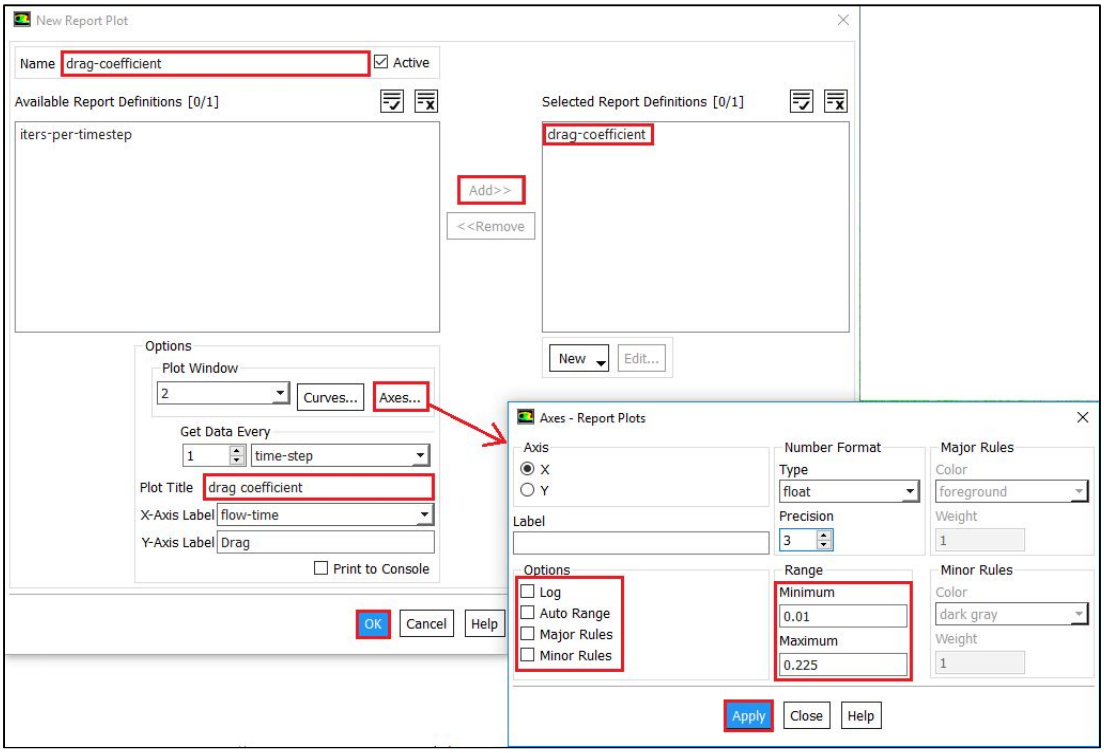


*Step 1.15~1.16 is for the plotting of the time history of the total drag coefficient during the computation.

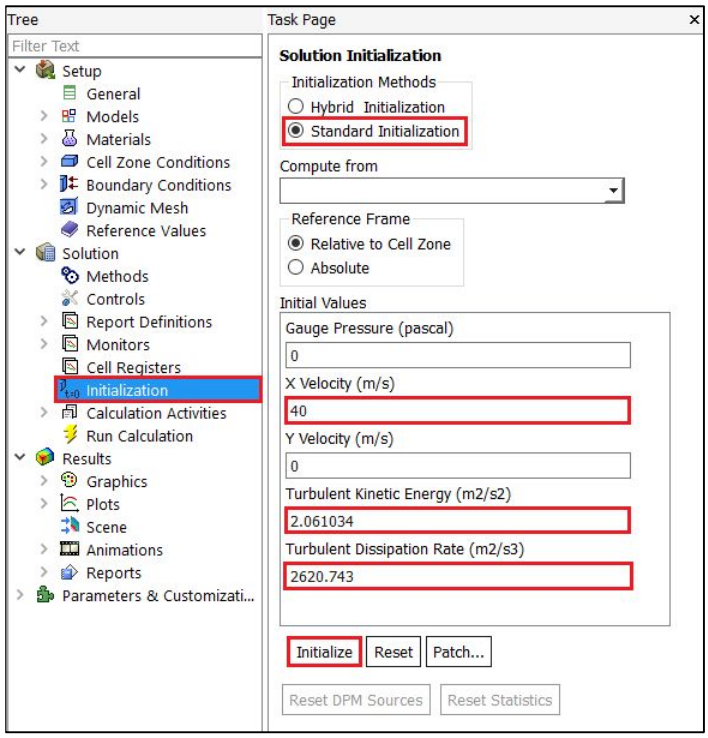
1.15. Solution > Monitors > Report Plots > New…



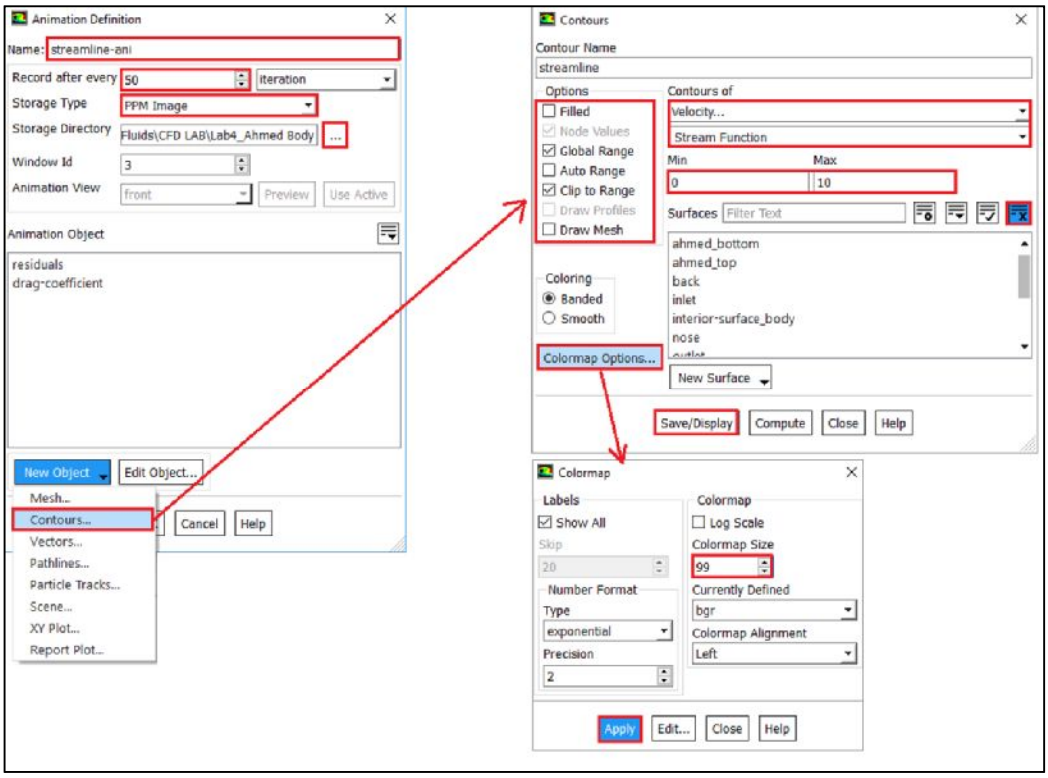
1.16. Select drag-coefficient generated from step 6.12~6.13 and **add** to right. Name the plot and change the x-axis condition by clicking **Axes…** as below. Exit from all dialogs.



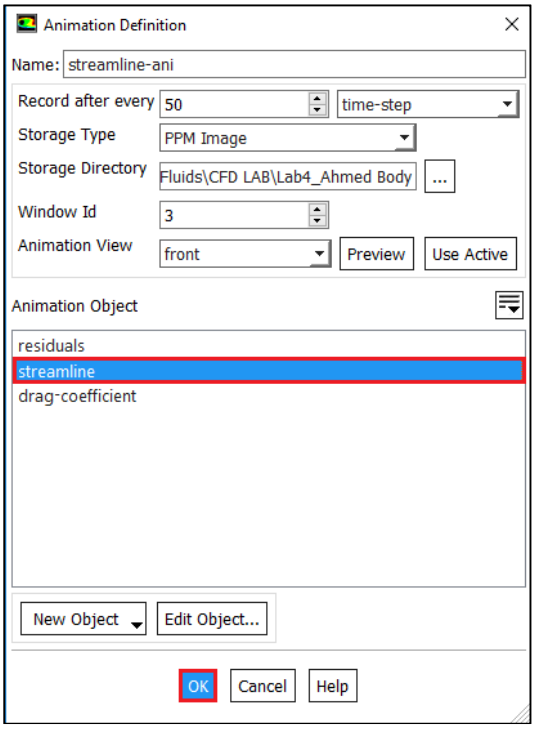
1.17. Solution > **Initialization**. Change **X-Velocity** and turbulent parameters as per below. Click **Initialize.**



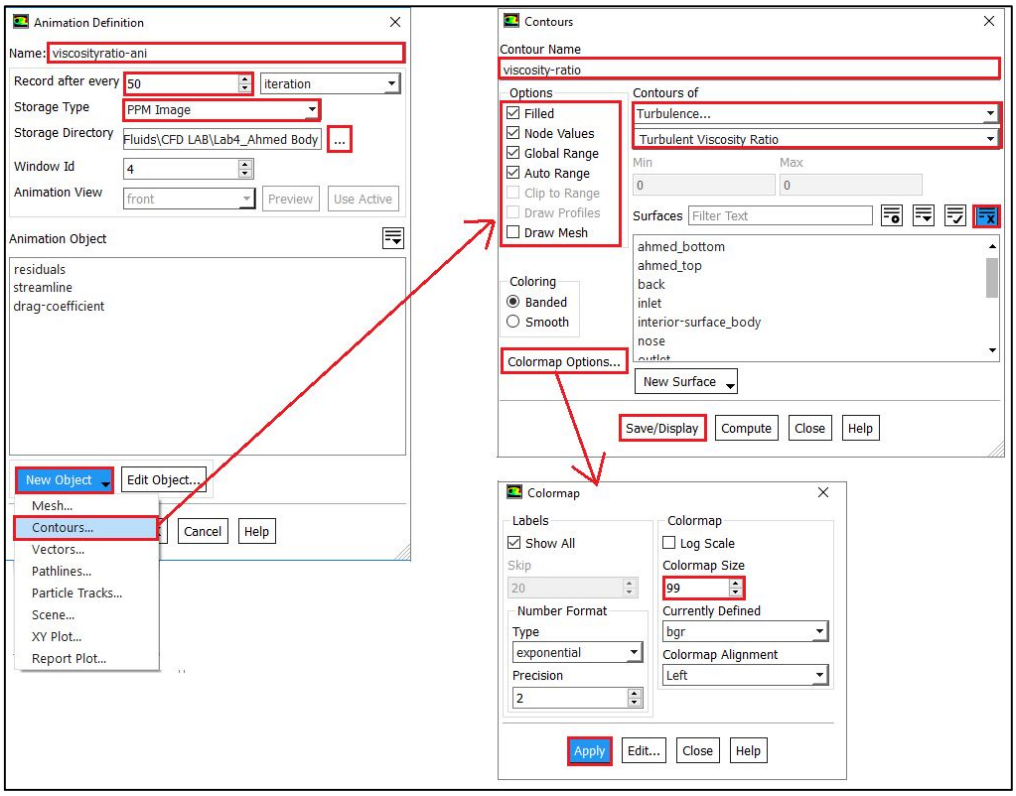
1.18. **Solution** > **Calculation Activities** > **Solution Animations** (right click) > **New...** Change the parameters as per below.



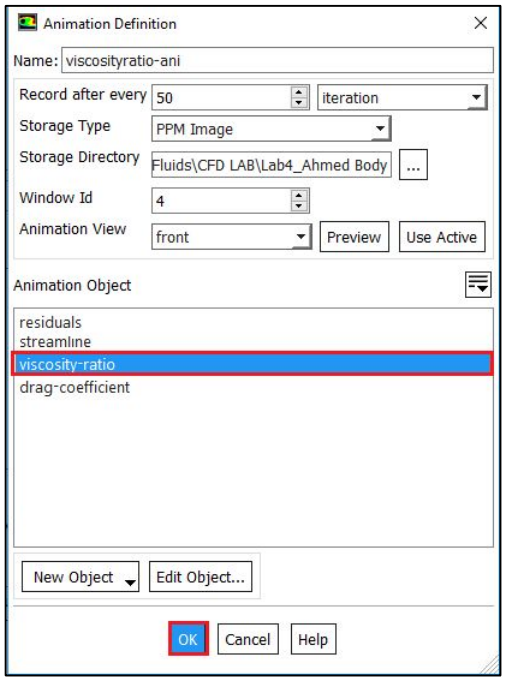
1.19. After 6.18, make sure to highlight **streamline** as an object and then close by clicking **OK**.



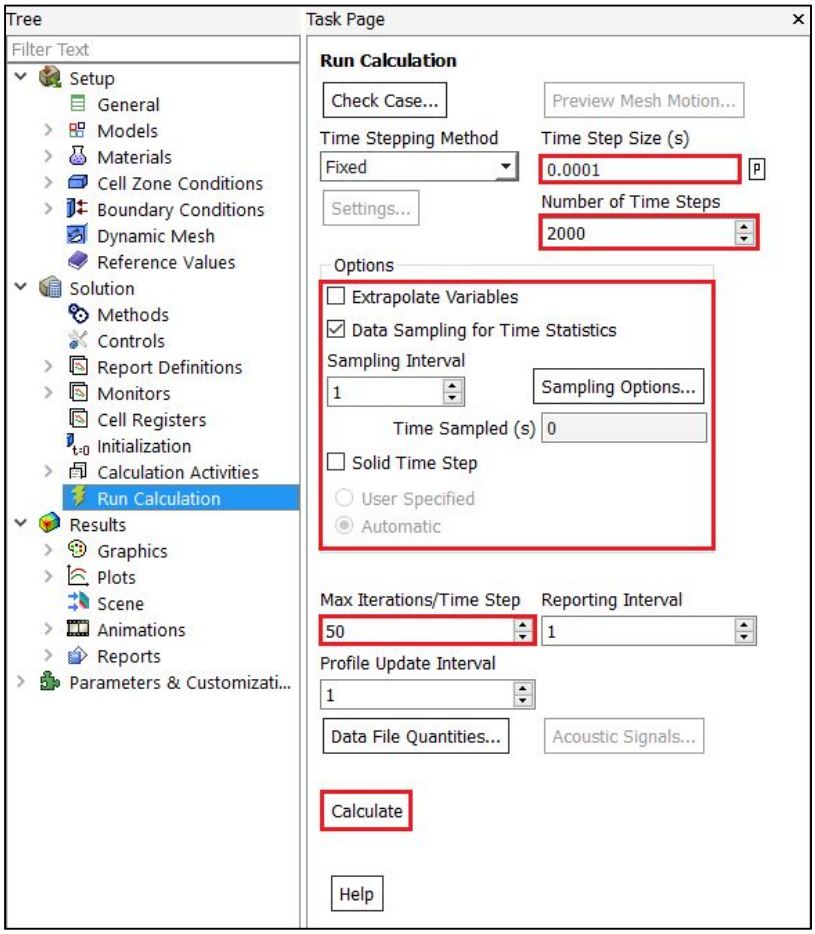
1.20. **Solution** > **Calculation Activities** > **Solution Animations** (right click) > **New...** Change the parameters as per below.



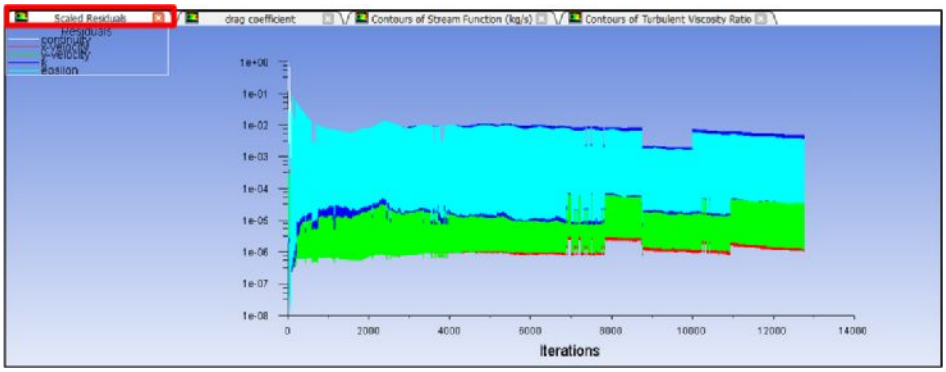
1.21. After 6.20, make sure to highlight **viscosity-ratio** as an object and then click **OK**.

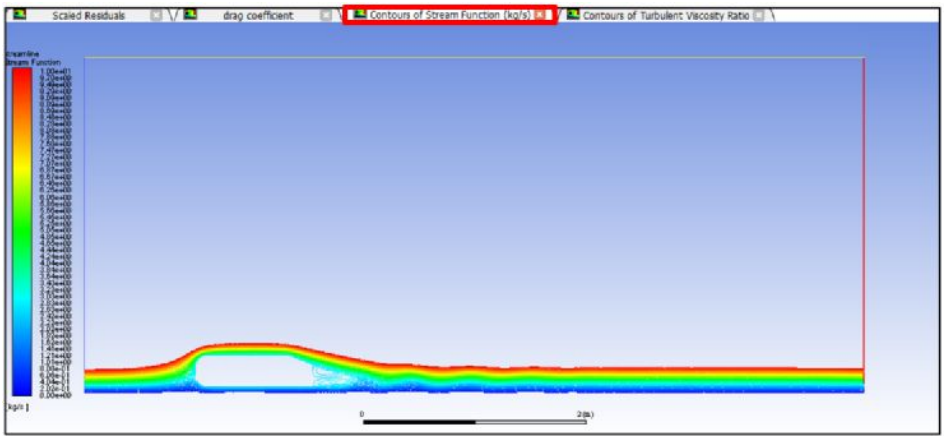


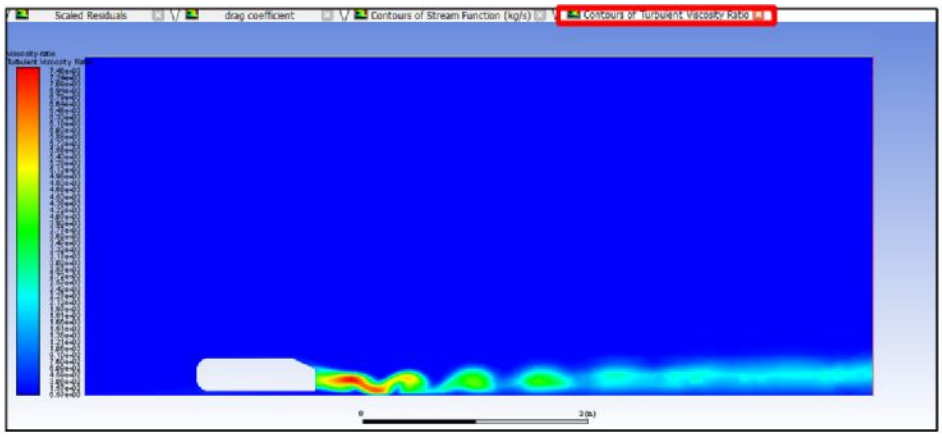
1.22. **Solution** > **Run Calculation**. Change parameters as per below and click **Calculate**. If you have the correct setup you should see four tabs on the upper sides of the display. You can change what the window shows by changing the tab. Tab 1-4 shows the residuals, streamlines, turbulent viscosity ratio and time-history of drag coefficient. After running for about 0.05 flow time you should see vortices at the back of the Ahmed car on tab 2 and 3. **NOTE: This simulation could take up to an hour depending on the computer performance. Please make sure your setup is correct before running the simulation! If you close Fluent window after running the simulation, the data for the post-process is not lost, but harder to access. If at all possible, finish post-processing after solving. Accessing the time-history of drag coefficient and post processing videos after the Fluent window is closed will be explained in later sections.**



After the computation, you should see the images below:



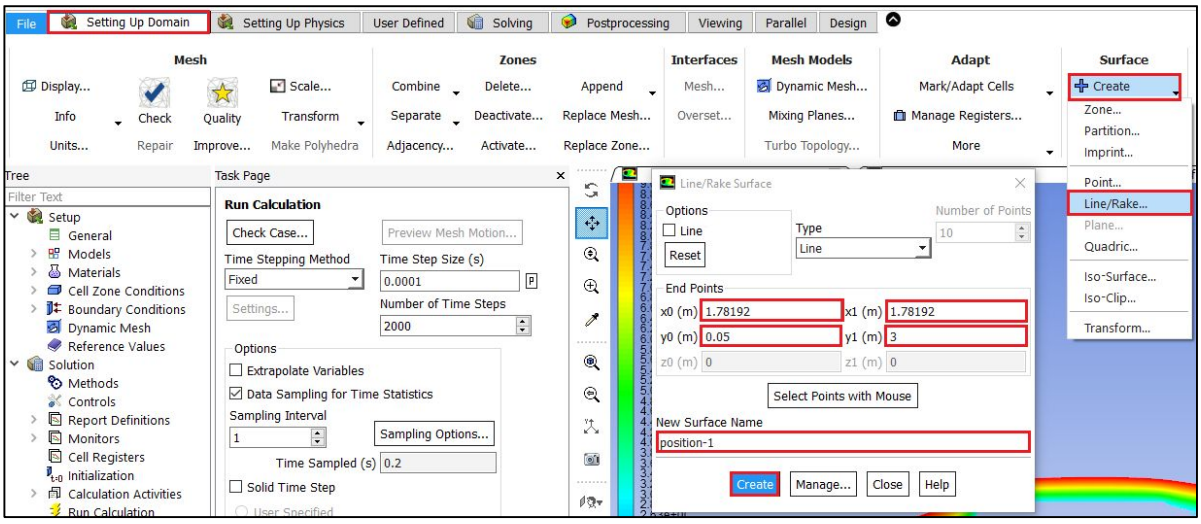




2. Results

2.1. Creating lines to plot modified TKE and modified U.

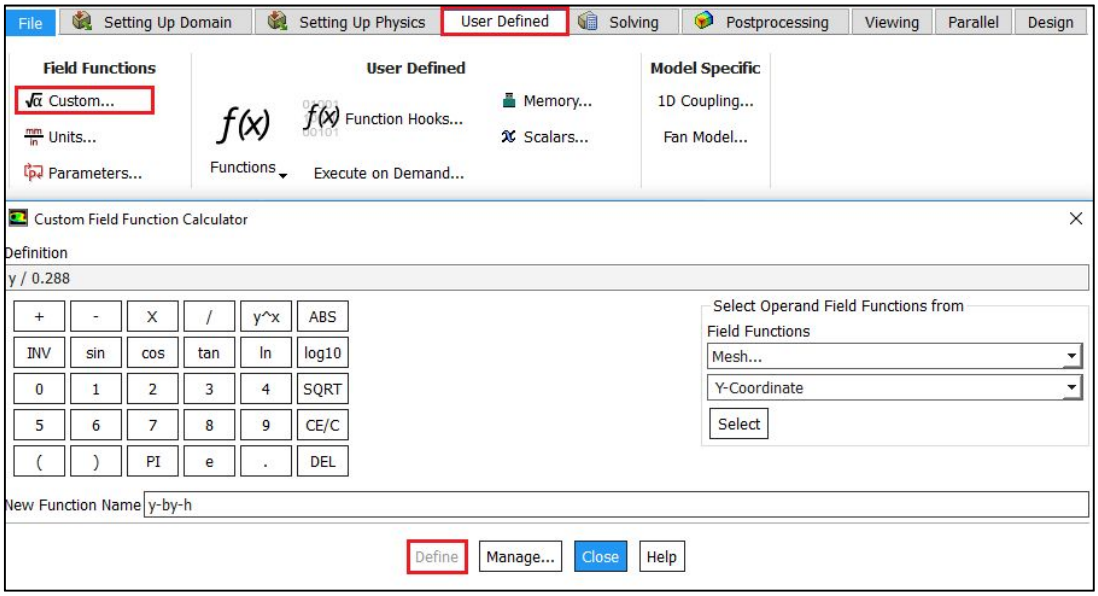
Setting Up Domain > Surface > Create > Line/Rake. Create 10 lines at the locations given at the table below.



Surface Name	x0	y0	x1	y1
position-1	1.78192	0.05	1.78192	3
position-2	1.932	0.05	1.932	3
position-3	1.98208	0.05	1.98208	3
position-4	2.03191	0.05	2.03191	3
position-5	2.08201	0	2.08201	3
position-6	2.13212	0	2.13212	3
position-7	2.23206	0	2.23206	3
position-8	2.332	0	2.332	3
position-9	2.482	0	2.482	3
position-10	2.6819	0	2.6819	3

2.2. Creating custom function

User-Defined > **Field Functions > Custom**. Create custom field functions and click **Define**. You will need to create three custom field functions shown in the table below.



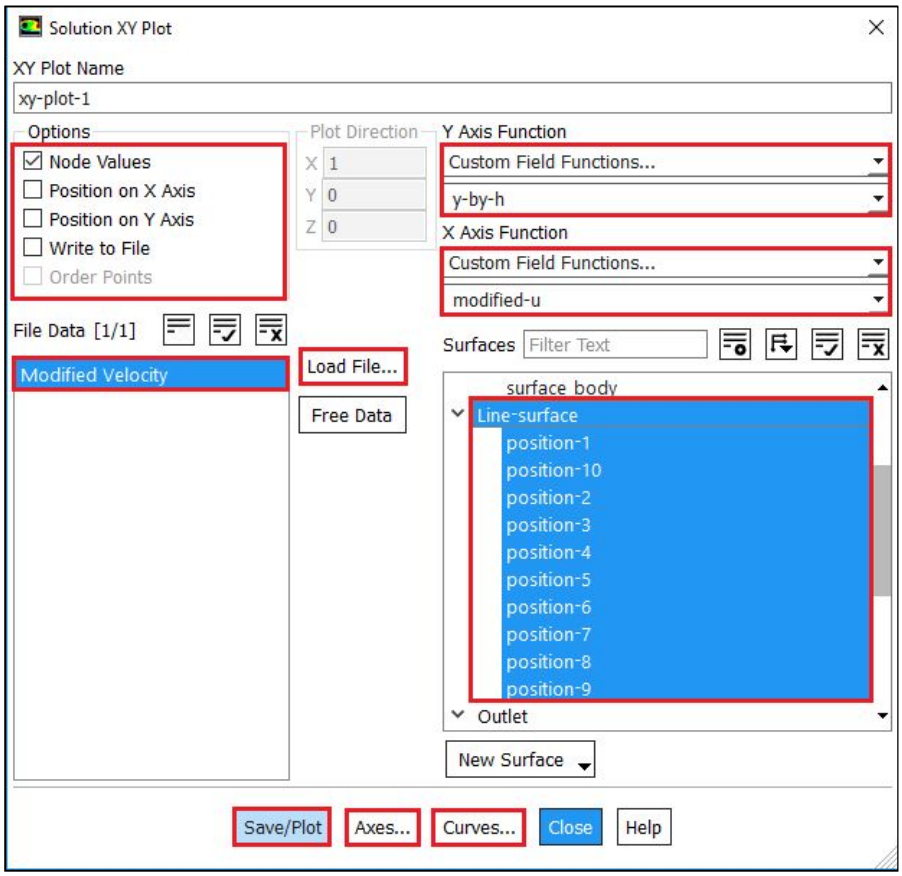
Function Name	Definition
y-by-h	y / 0.288
Modified-U	(mean-x-velocity / 120) + (x / 0.288)
Modified-TKE	(turb-kinetic-energy / 500) + (x / 0.288)

Operand field function including x and y position, mean-x-velocity and turb-kinetic-energy can be found in the following table:

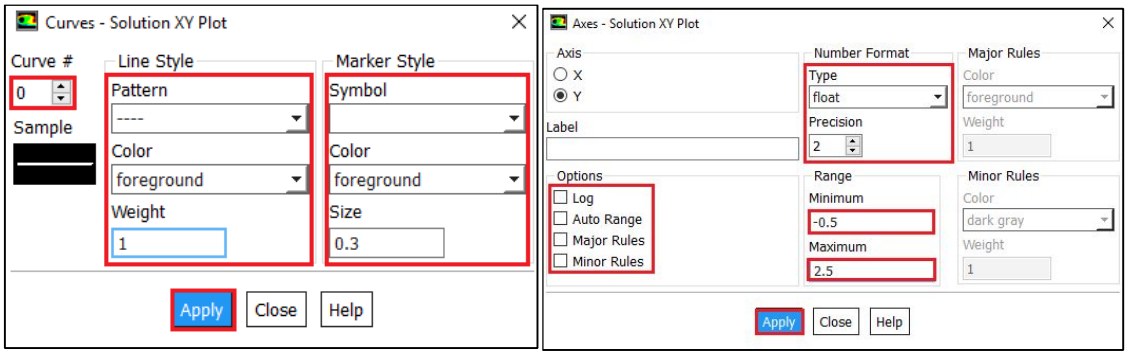
Operand field function	From field functions
x, y	Mesh
Mean-x-velocity	Unsteady statistics
Turb-kinetic-energy	Turbulence

2.3. Plotting values along the lines created

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: For *'Modified-TKE vs. y-by-h'* figure, please plot CFD values only)

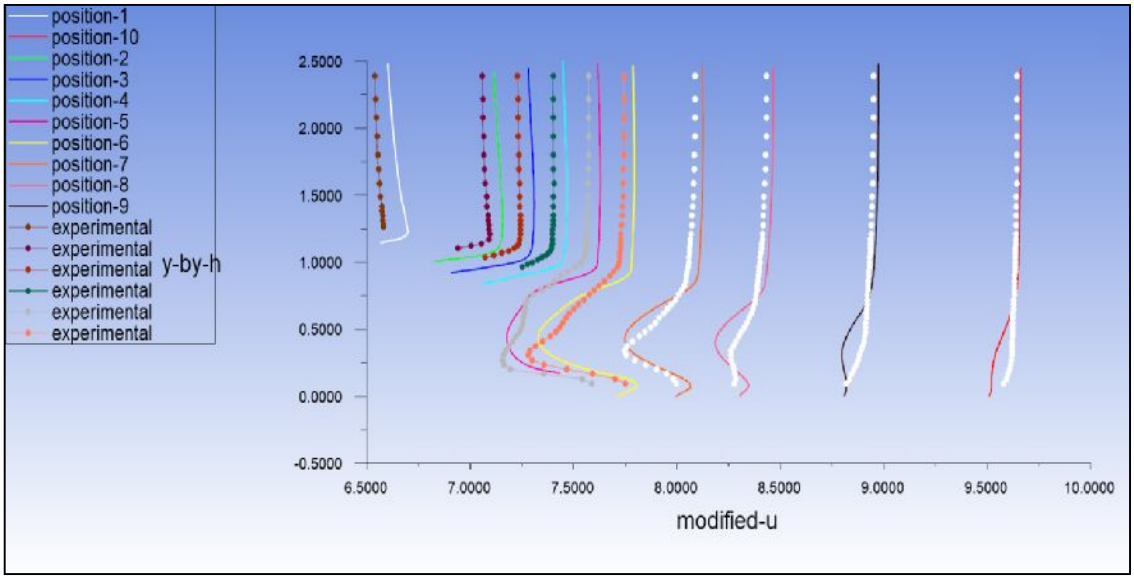


Note: You can change the style and color of the data by clicking **Curves** button and changing the parameters below then clicking **Apply**. Click **Axes…** and adjust the Y axis maximum to 2.5 and minimum to -0.5.


Curves - Solution XY Plot
Curve #
0
Sample
Line Style
Pattern

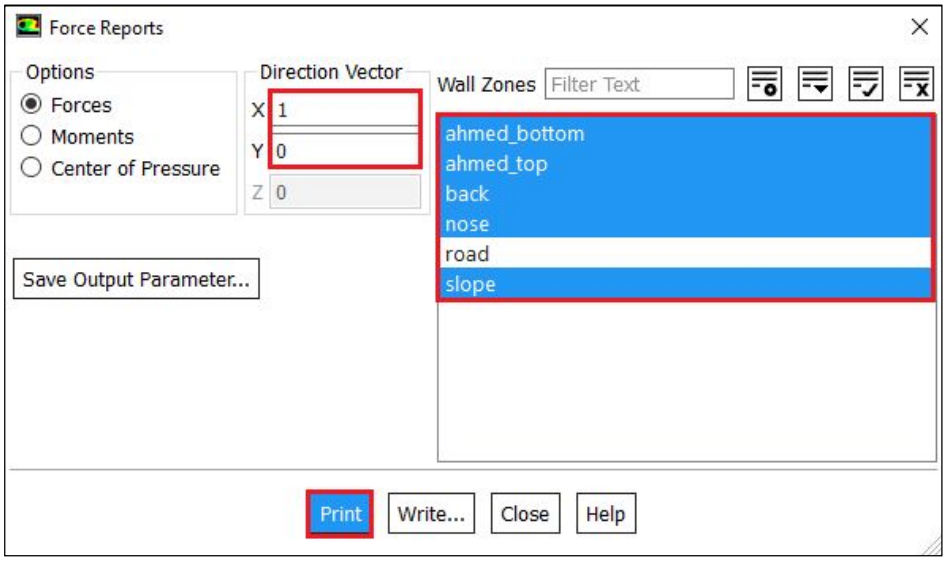
Color
foreground
Weight
1
Marker Style
Symbol
Color
foreground
Size
0.3
Apply
Close
Help
Axes - Solution XY Plot
Axis
X
Y
Label
Number Format
Type
float
Precision
2
Major Rules
Color
foreground
Weight
1
Options
Log
Auto Range
Major Rules
Minor Rules
Range
Minimum
-0.5
Maximum
2.5
Minor Rules
Color
dark gray
Weight
1
Apply
Close
Help

Result:


position-1
position-10
position-2
position-3
position-4
position-5
position-6
position-7
position-8
position-9
experimental
experimental
experimental
experimental
experimental
experimental
y-by-h
2.5000
2.0000
1.5000
1.0000
0.5000
0.0000
-0.5000
6.5000
7.0000
7.5000
8.0000
8.5000
9.0000
9.5000
10.0000
modified-u

2.4. Printing drag coefficient components

Results > **Reports** > **Forces**. Select the region where you want to calculate the drag coefficient under wall zone then click print.

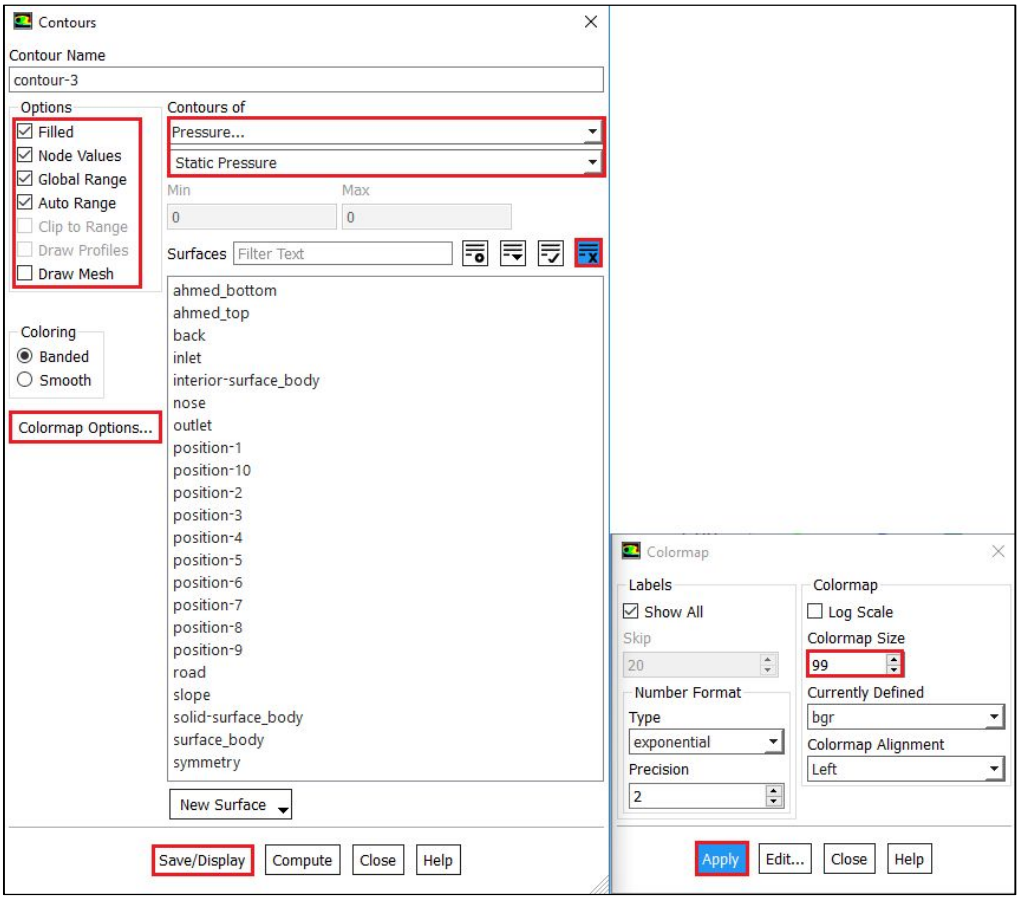


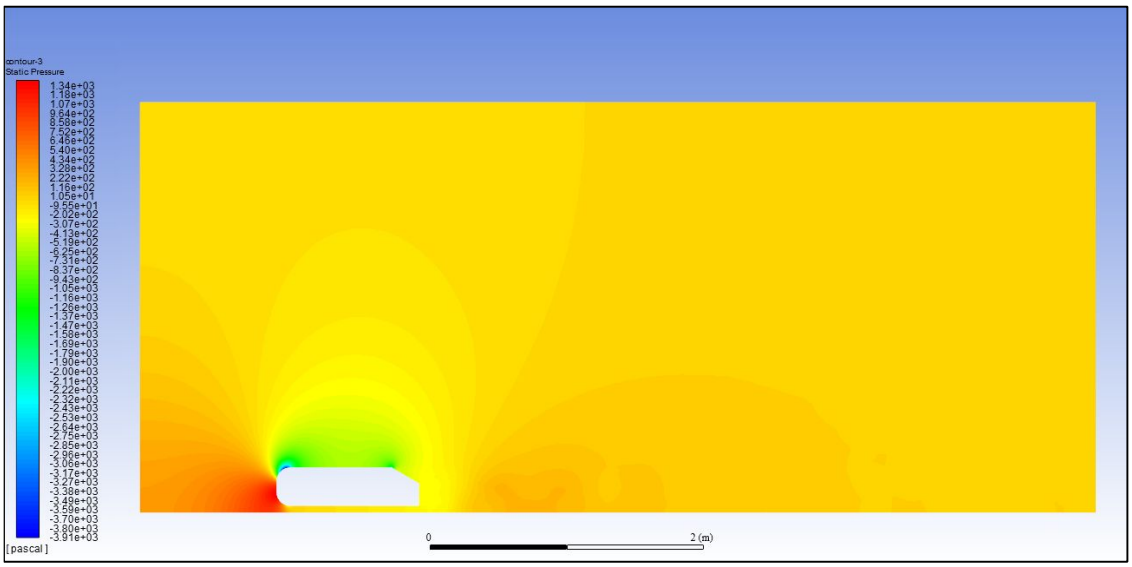
2.5. Plotting time-history of total drag coefficient

Note: If you closed Fluent without first plotting and saving drag coefficient time history, navigate to "\Lab 4 Project File_files\dp0\FLU\Fluent" and find an ascii file named as "cd-1-history". **You can choose Excel for plotting with this file:**

2.6. Plotting Pressure Contours

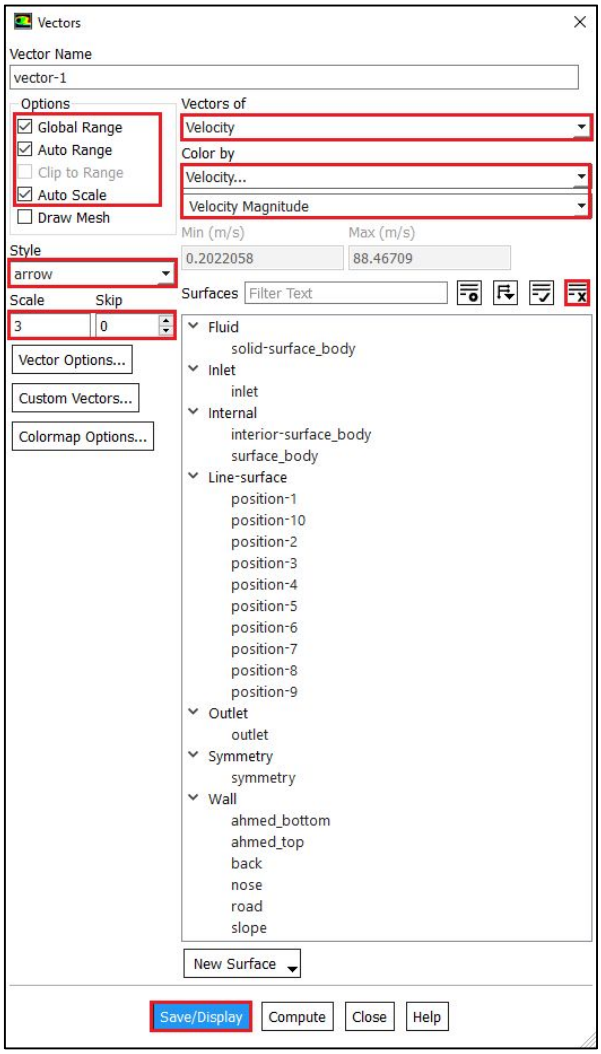
Results > **Graphics** > **Contours**. Change parameters as per below and click **Display**.

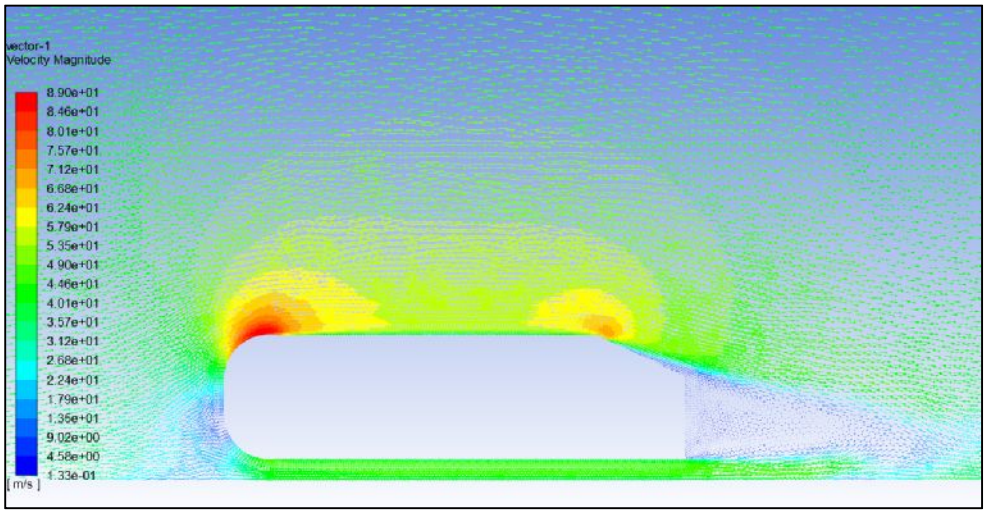



contour-3
Static Pressure
1.34e+03
1.18e+03
1.07e+03
9.54e+02
8.50e+02
7.52e+02
6.46e+02
5.40e+02
4.34e+02
3.28e+02
2.22e+02
1.16e+02
1.05e+01
-9.55e+01
-2.02e+02
-3.07e+02
-4.13e+02
-5.19e+02
-6.25e+02
-7.31e+02
-8.37e+02
-9.43e+02
-1.05e+03
-1.16e+03
-1.26e+03
-1.37e+03
-1.47e+03
-1.58e+03
-1.69e+03
-1.79e+03
-1.90e+03
-2.00e+03
-2.11e+03
-2.22e+03
-2.32e+03
-2.43e+03
-2.53e+03
-2.64e+03
-2.75e+03
-2.85e+03
-2.96e+03
-3.06e+03
-3.17e+03
-3.27e+03
-3.38e+03
-3.49e+03
-3.59e+03
-3.70e+03
-3.80e+03
-3.91e+03
[pascal]
0
2 (m)

2.7. Plotting Velocity Vectors

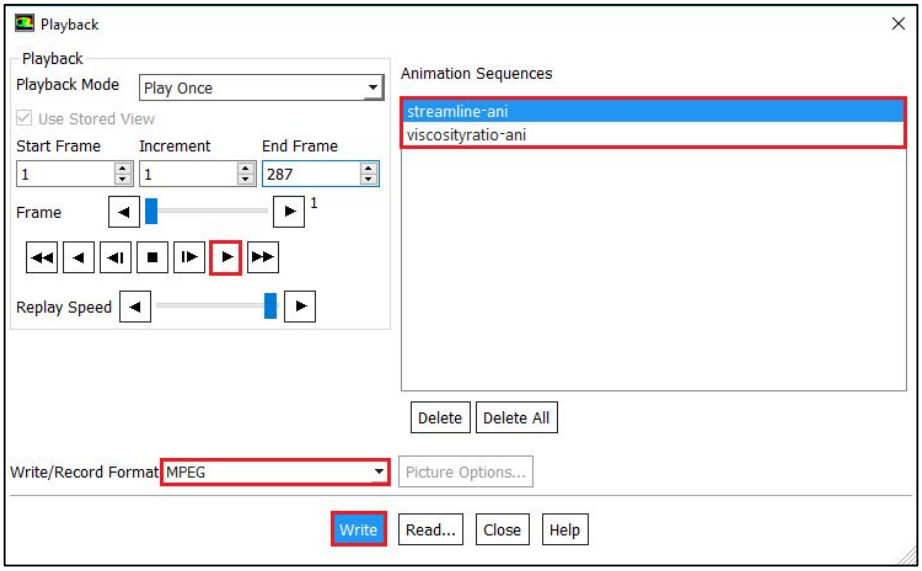
Results > **Graphics** > **Vectors** > **Set Up…** Change parameters as per below and click **Display**.

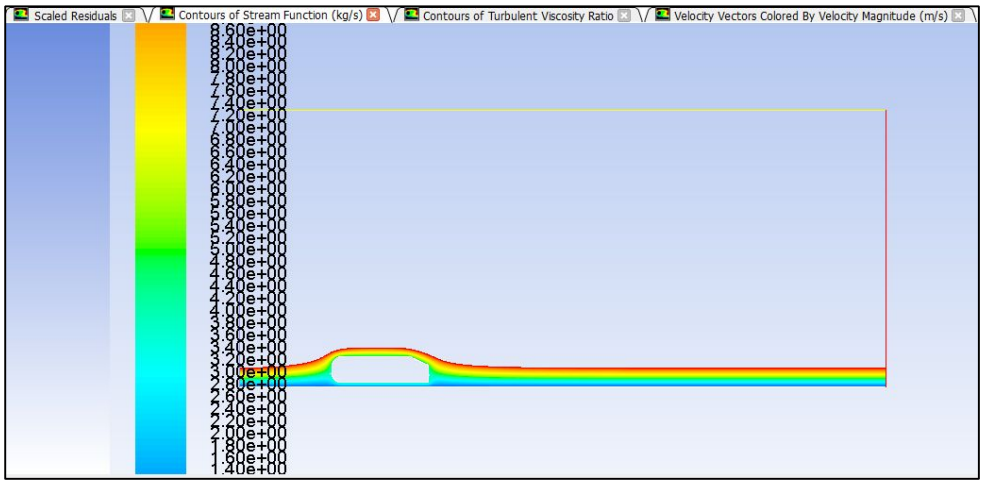




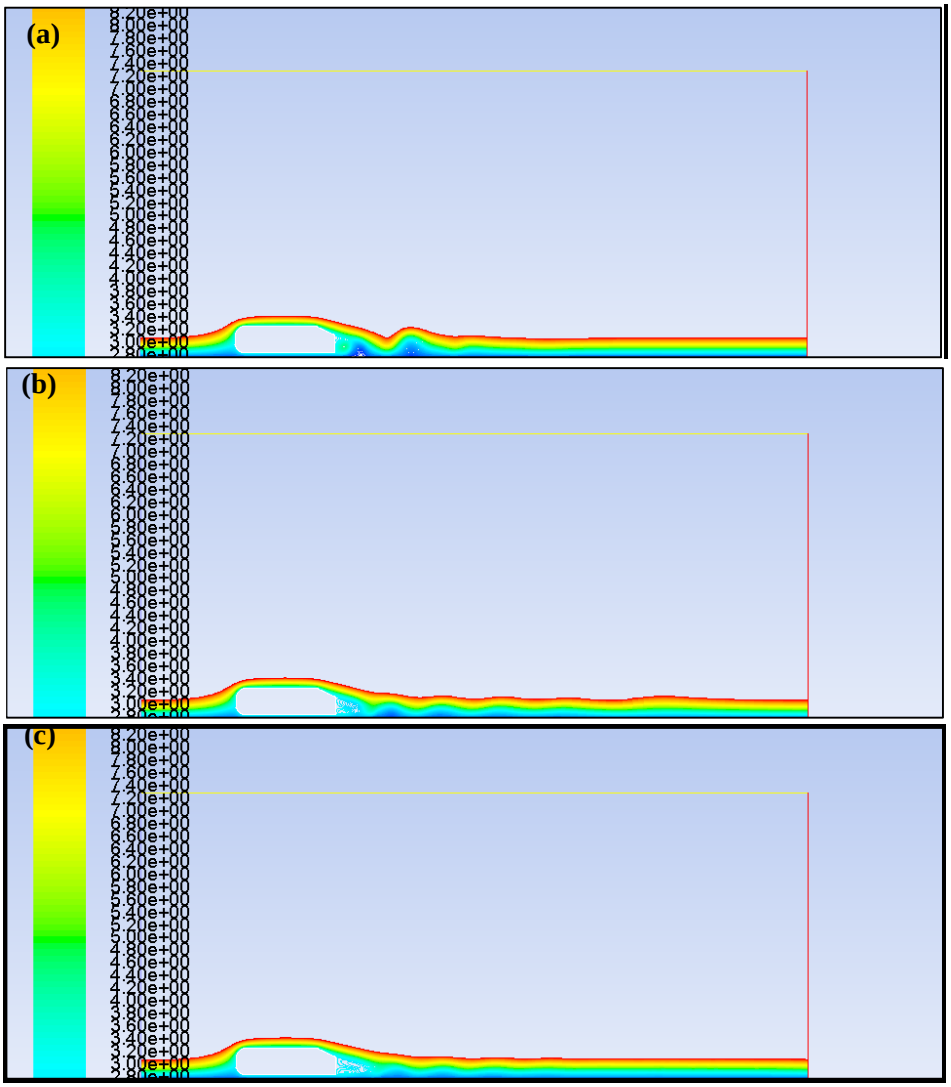
2.8. Creating videos

Results > **Animations** > **Solution Animation Playback**. Change the window to streams or viscous ratio then click play button to see the animation.

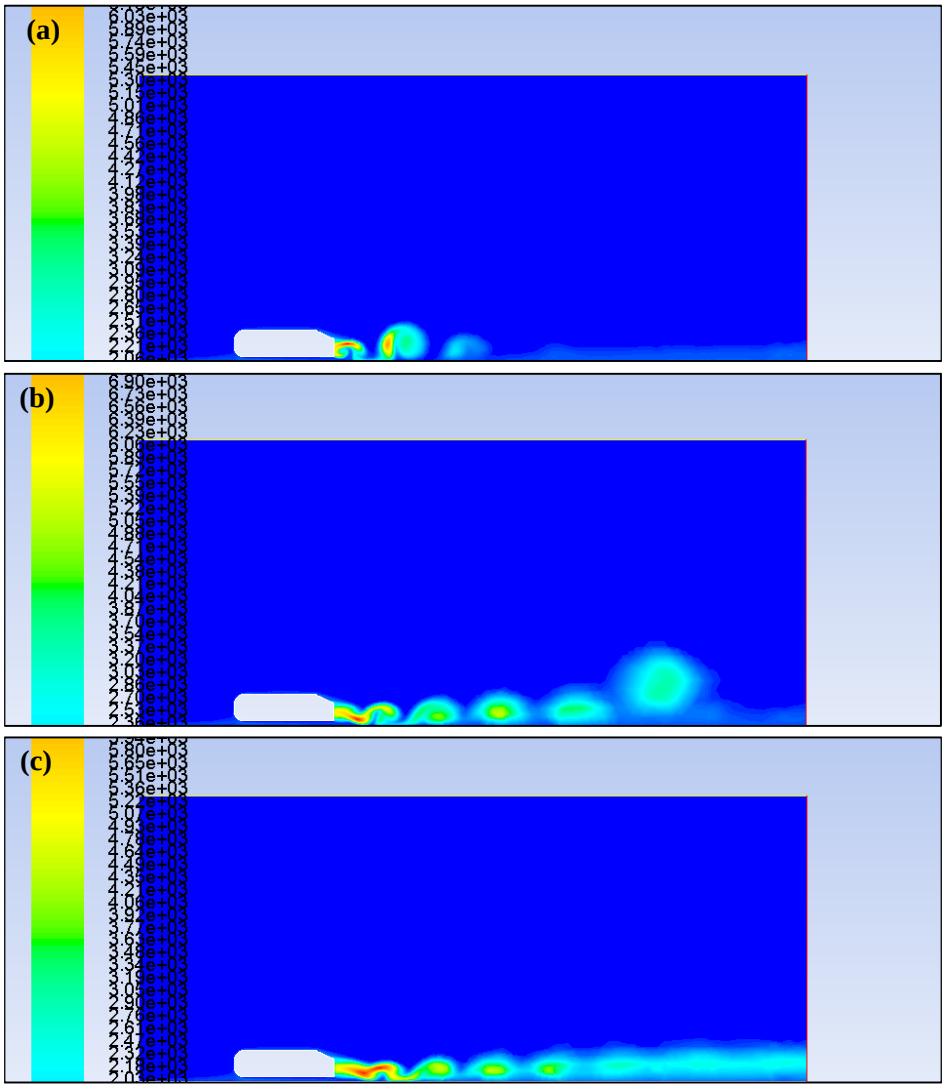




Once the video is playing correctly, return to the first time-step, change the **Write/Record Format** to **MPEG** and click **Write**. Generating the movie can take a few minutes. Please go through the same procedure for viscosity-ratios.

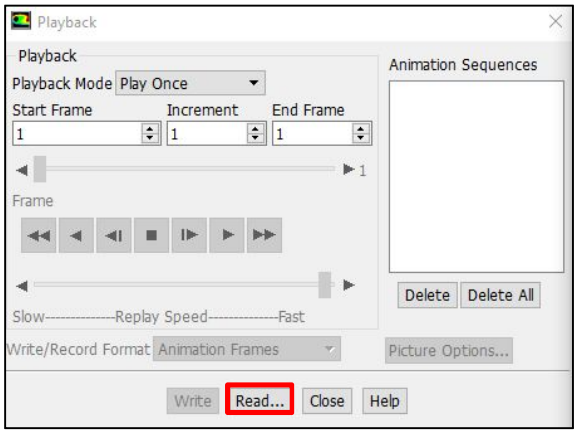


Pictures of streamline for ahmed car: (a) frame=100; (b) frame=200; (c) frame=300



Pictures of turbulent viscosity ratio for ahmed car: (a) frame=100; (b) frame=200; (c) frame=300

If you have closed the Fluent after the calculation, there could be no ‘streamline’ or ‘viscosity-ratio’ under the Animation Sequence in Animation Playback when you reopen it (select **solution** instead **setup** in the workbench when you reopen). You need to read the set-up file to bring back those options. If this is the case, click **Read** and read the ‘*.cxa’ file saved in the location you assigned at the animation part. Select both streamline and viscosity-ratio files (one at each time). Once the options appear under the Animation Sequence, save video files like the beginning of this section (section 2.8). Please note that the setup files (*.cxa) can be modified by opening it with notepad or any ascii readers. Please change the options according to your needs.




```
AnimationSequence1.0
NAME: .\\stream
WINID: 2
STORAGE: 4
FRAMES: 250
Frame 0 4 stream_0000.ppm 2
Frame 1 4 stream_0001.ppm 2
Frame 2 4 stream_0002.ppm 2
Frame 3 4 stream_0003.ppm 2
Frame 4 4 stream_0004.ppm 2
Frame 5 4 stream_0005.ppm 2
Frame 6 4 stream_0006.ppm 2
Frame 7 4 stream_0007.ppm 2
Frame 8 4 stream_0008.ppm 2
Frame 9 4 stream_0009.ppm 2
Frame 10 4 stream_0010.ppm 2
Frame 11 4 stream_0011.ppm 2
```

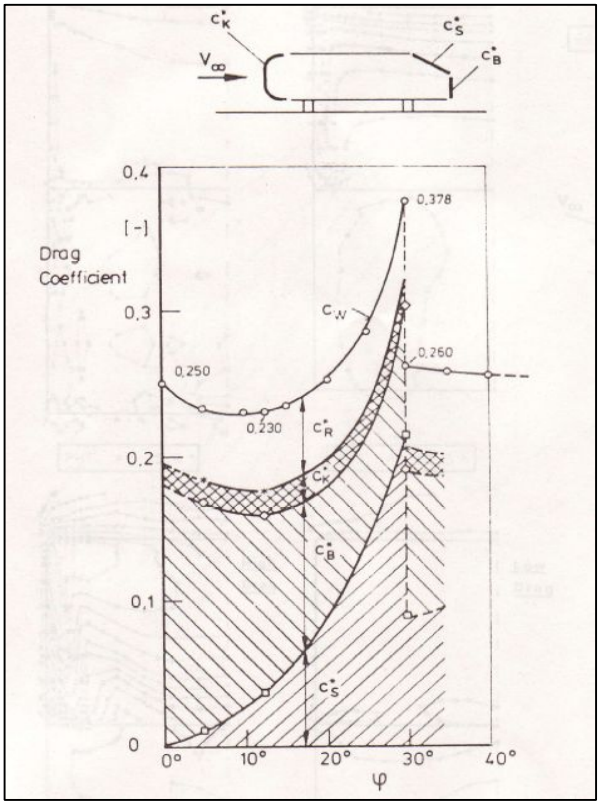
```
AnimationSequence1.0
NAME: .\\viscosity-ratio
WINID: 3
STORAGE: 4
FRAMES: 250
Frame 0 4 viscosity-ratio_0000.ppm 2
Frame 1 4 viscosity-ratio_0001.ppm 2
Frame 2 4 viscosity-ratio_0002.ppm 2
Frame 3 4 viscosity-ratio_0003.ppm 2
Frame 4 4 viscosity-ratio_0004.ppm 2
Frame 5 4 viscosity-ratio_0005.ppm 2
Frame 6 4 viscosity-ratio_0006.ppm 2
Frame 7 4 viscosity-ratio_0007.ppm 2
Frame 8 4 viscosity-ratio_0008.ppm 2
Frame 9 4 viscosity-ratio_0009.ppm 2
Frame 10 4 viscosity-ratio_0010.ppm 2
Frame 11 4 viscosity-ratio_0011.ppm 2
```

3. Data Analysis and Discussion

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

3.1. Simulation of turbulent flows over Ahmed body (slant angle=25 deg):

Fill in the table for the four drag coefficients and compute the relative error between CFD and experimental data (Ahmed data), experimantal 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 (%)				

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 need to be reported:** (1) XY plots for residual history, (2) modified U (with experimental data), (3) Modified-TKE, (4) time history of drag coefficient, (5) Contour of pressure, (6) contour of velocity magnitude, (7) velocity vectors, (8) 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 need to be reported:** the above table with values.