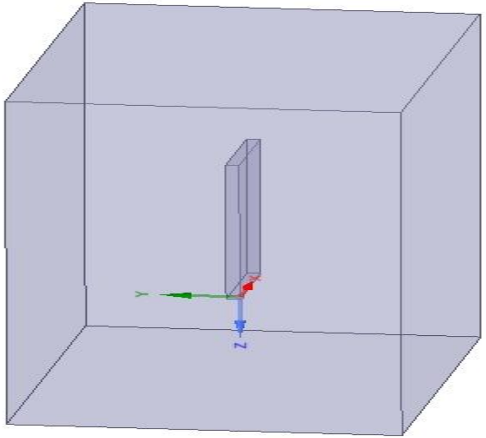
ASSIGNMENT - 3
ME615

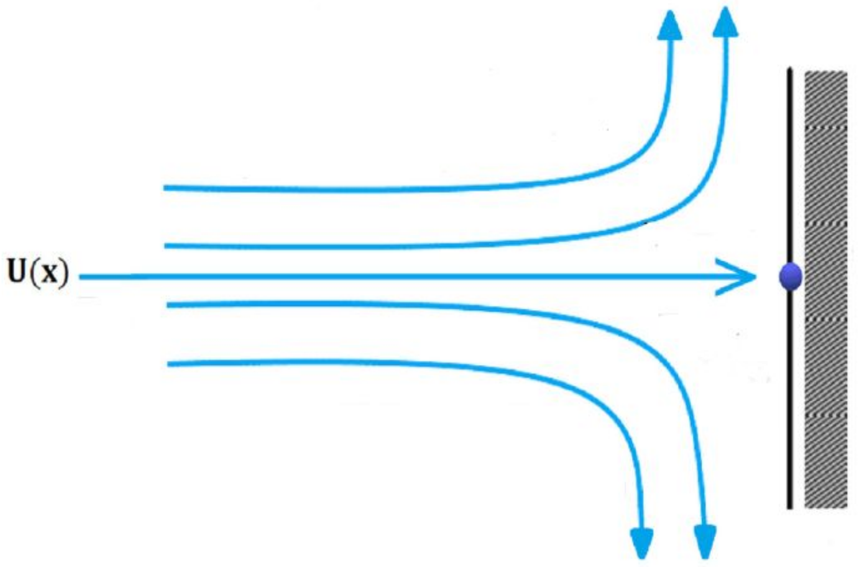
Solving the External Flow Problem on a Thin Plate(rectangular plate) Oriented at 90° to the Flow Using ANSYS

Team Members
S. Ananya - B21323
Sonam Shrivastava - B21324



Topics of Discussion

- Model description with the governing differential equations and boundary conditions.
- How to do analysis on Ansys which includes the following:
  - Geometry
  - Meshing
  - Setup and Solver Options
  - Results
    - Velocity analysis
    - Pressure analysis
    - Vortex Shedding analysis
- Graphs and Animations.
  - Lift and drag coefficients
  - Residuals



Model Description

We analyse the flow past a thin rectangular plate when it is oriented perpendicularly to the flow.

We consider a flat plate perpendicular to a stream of fluid flowing with constant velocity U. We are interested in the steady state solution.

At the walls there is no flow across it, which implies that

$$v(\text{at walls}) = 0$$

Due to the viscosity we have the no slip condition at the plate. In other words,

$$u(\text{at walls}) = 0$$

Governing Differential Equation:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0,$$

Boundary Conditions:

- At adjacent walls of plate, **‘No slip Condition’**. So, velocity components at the walls are zero.
- The plate is at rest. No relative motion between plate and the fluid.
- Velocity at the inlet is constant.

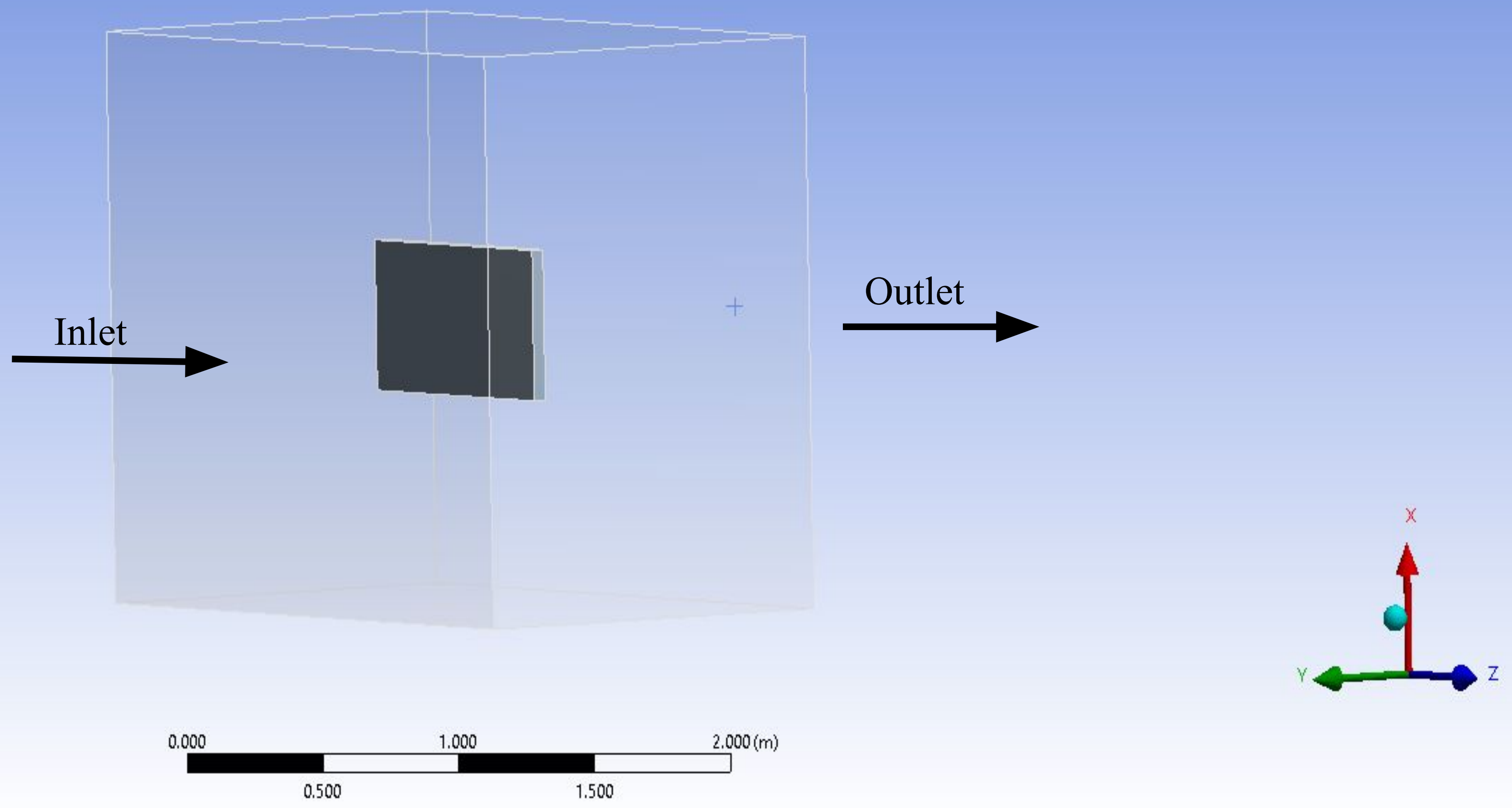
Geometry

We made Geometry using SpaceClaim software.

Dimensions of the plate: 100 cm x 50 cm x 5 cm.

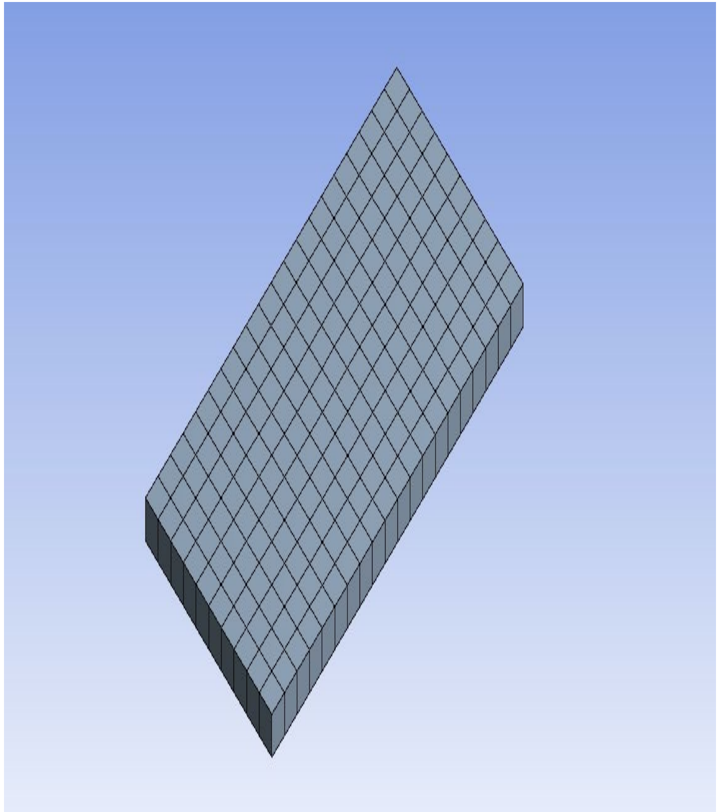
Steps:

1. Draw a rectangle with dimensions 100cm length and 50 cm width. Extrude the rectangle by 5cm using ‘Pull’ command in design bar.
2. To analyse the flow we need to make an enclosure around the rectangular plate.
3. To make enclosure, go to the ‘Prepare’ option on the screen and then click on the ‘enclosure’ option. Then, give the dimensions.
4. We have taken the dimensions of enclosure as 145 cm x 240 cm x 190 cm(l x b x h).


Inlet
Outlet
X
Y
Z
0.000
1.000
2.000 (m)
0.500
1.500

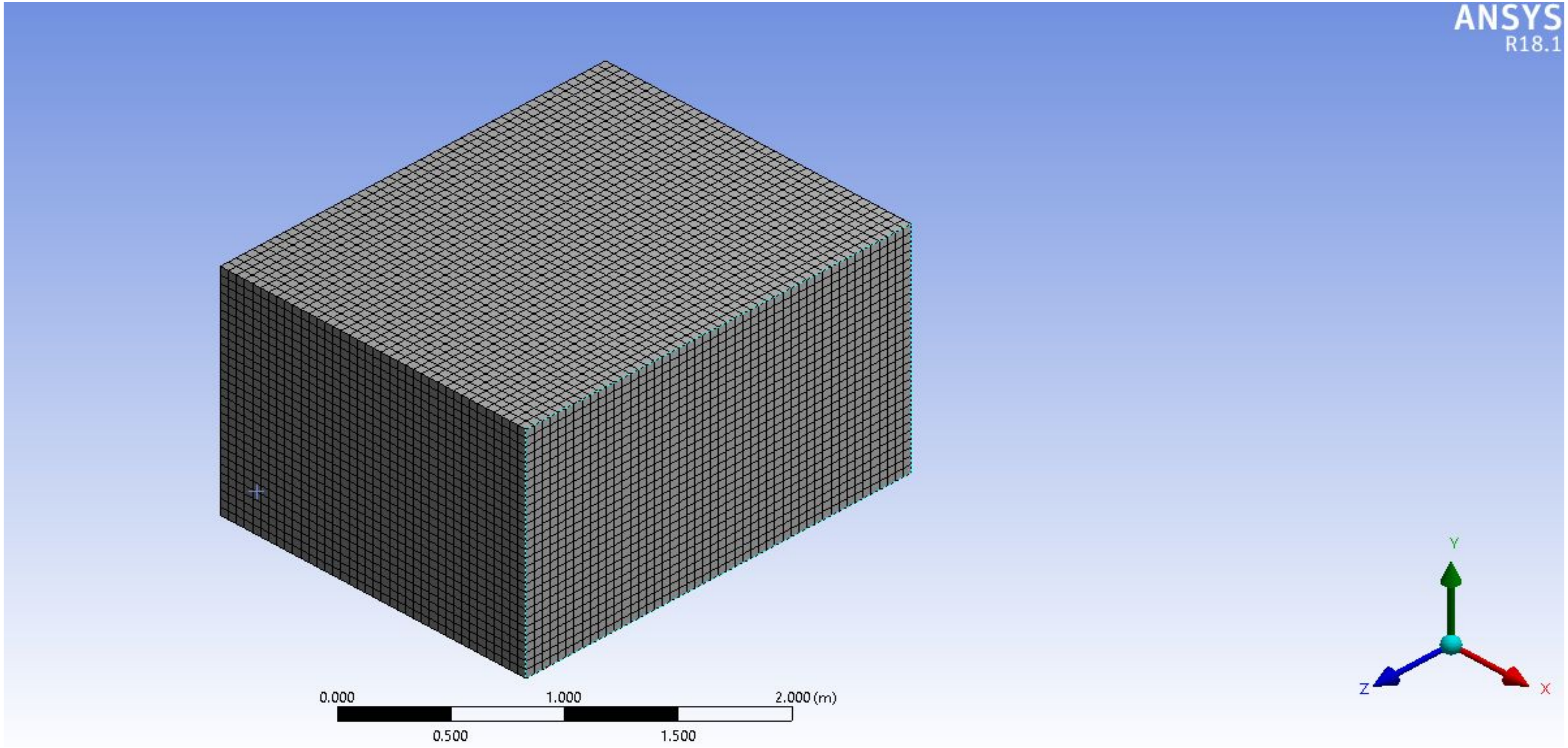
Meshing

1. Meshing can be done by using ‘Meshing' software in Ansys.
2. Mesh size used in this analysis is 50mm.
3. Apply face meshing on all the surfaces of plate and enclosure.
4. Fine meshing can be used to make the results more accurate but this comes with a cost of increased computational time.



Mesh grid on plate

Mesh Grid on Enclosure



Setup and Solver Options

- We assumed the flow to be in ‘Steady state’ for calculating velocity and pressures.
- We can use SIMPLE algorithm to solve for velocities and pressures (False Transient Analysis).
- To calculate lift and drag coefficients, we should use a turbulent flow model.
- K-epsilon turbulence model is used in this case as it is easy to implement, gives reasonable results and leads to stable calculations.
- K-epsilon is not advisable to be used with swirling and rotating flows and fully developed flows in non-circular ducts.

- We know that SIMPLE is an iterative algorithm. We need to check for convergence after each iteration.Total number of iterations used is n = 1000.
- Value used for convergence check for all variables (k, velocities and epsilon) is 0.000001.
- Fluid properties: Density= 1 kg/m^3 and ν = 0.001kg/ms
- As we are solving for a steady state, we can take initial values to be arbitrary. So, in solution initialisation option, choose ‘Hybrid initialisation’.
- In hybrid initialisation, computer guesses the initial values.

To calculate drag coefficient

Report Definitions -> New -> Force Report ->Drag -> Drag Coefficient and check the boxes report file and report plots

To calculate lift coefficient

Report Definitions -> New -> Force Report ->Lift -> Lift Coefficient and check the boxes report file and report plots

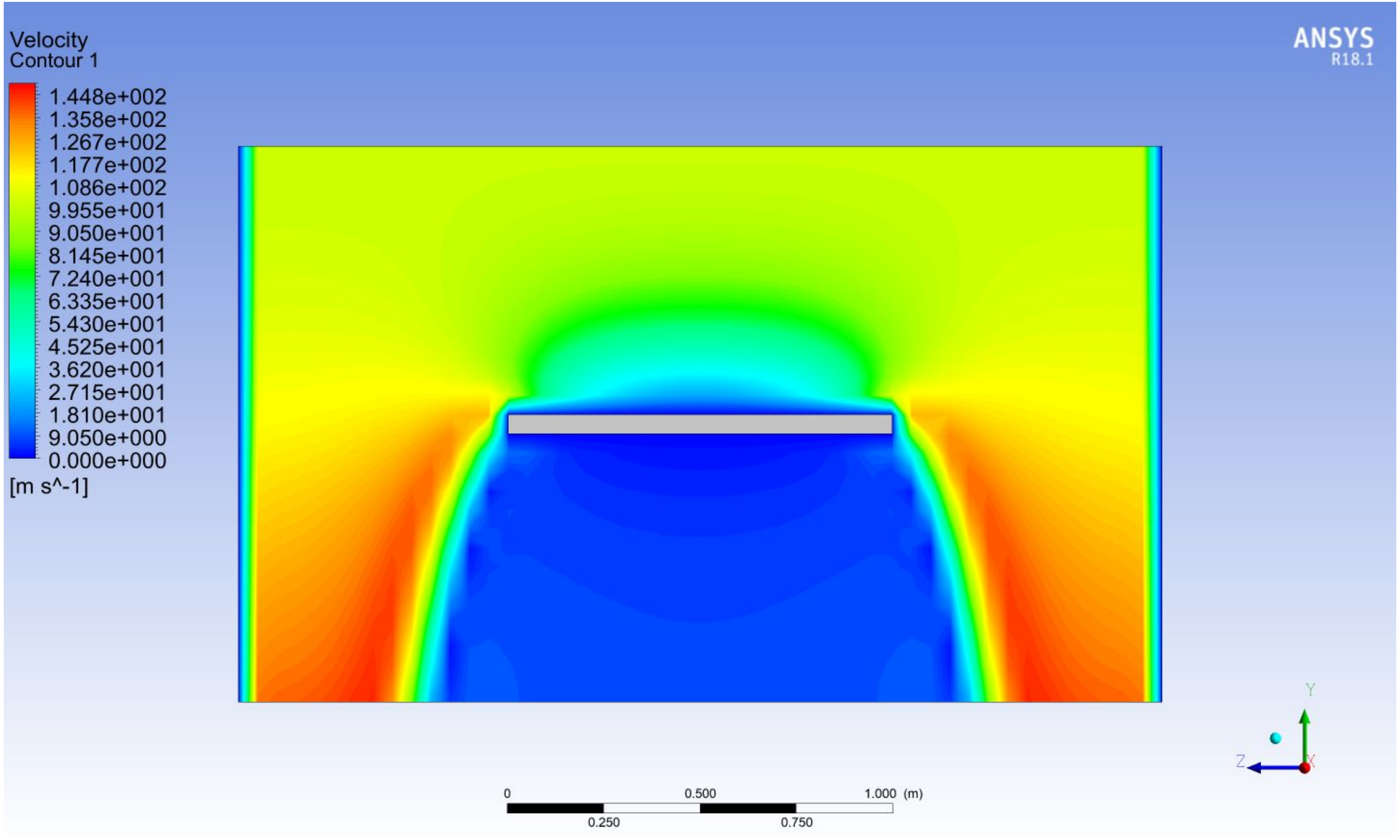
Results

The main results includes simulation of k-epsilon flow over a flat plate held perpendicular to the flow and visualize the velocity profile close to the solid boundary.

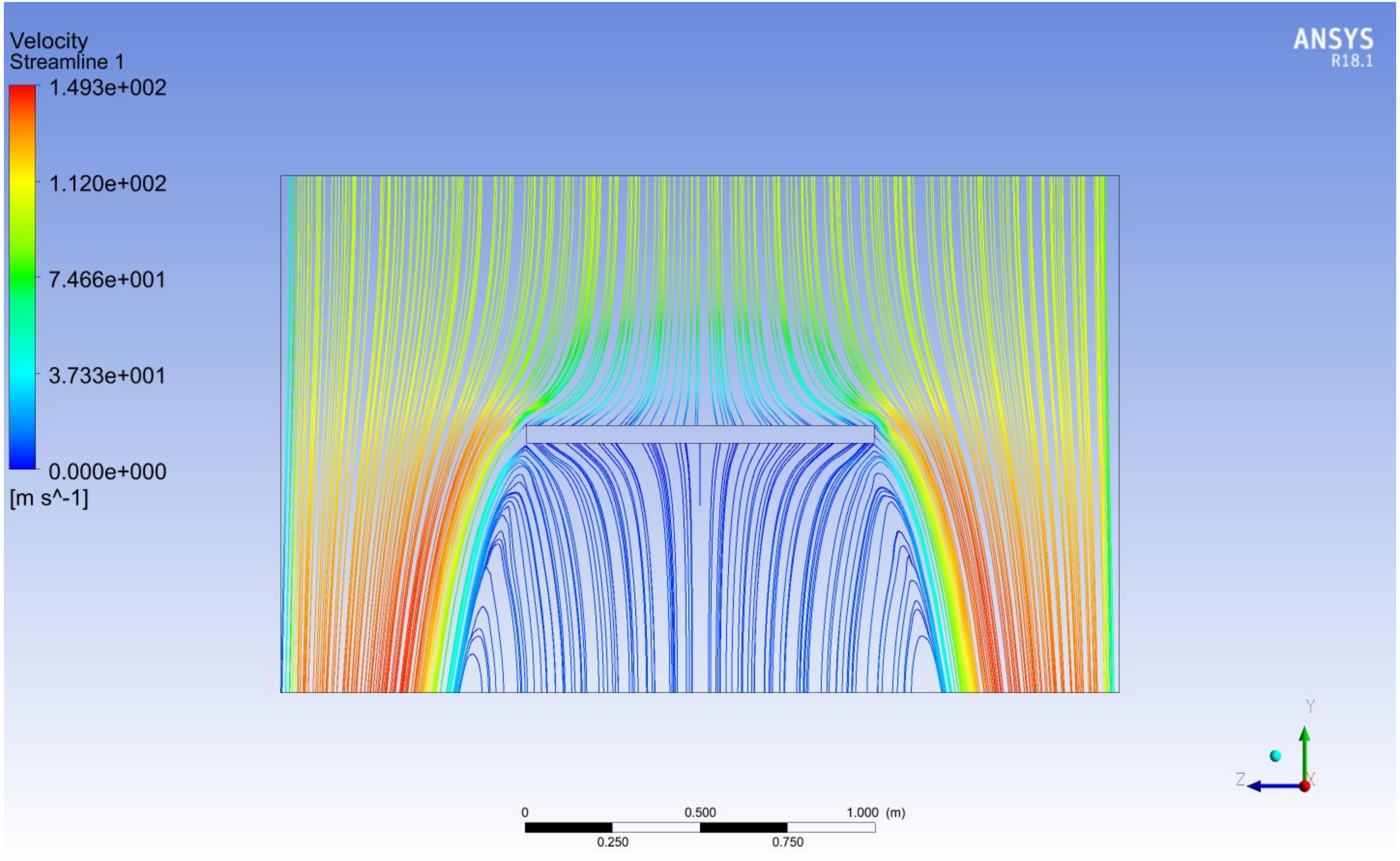
We can observe the different plots for different velocities with same convergence conditions and other parameters.

After certain iterations we observe that the velocity profiles, pressure profiles and the coefficients of drag and left and all other residual parameters remains constant and met the convergence criteria.

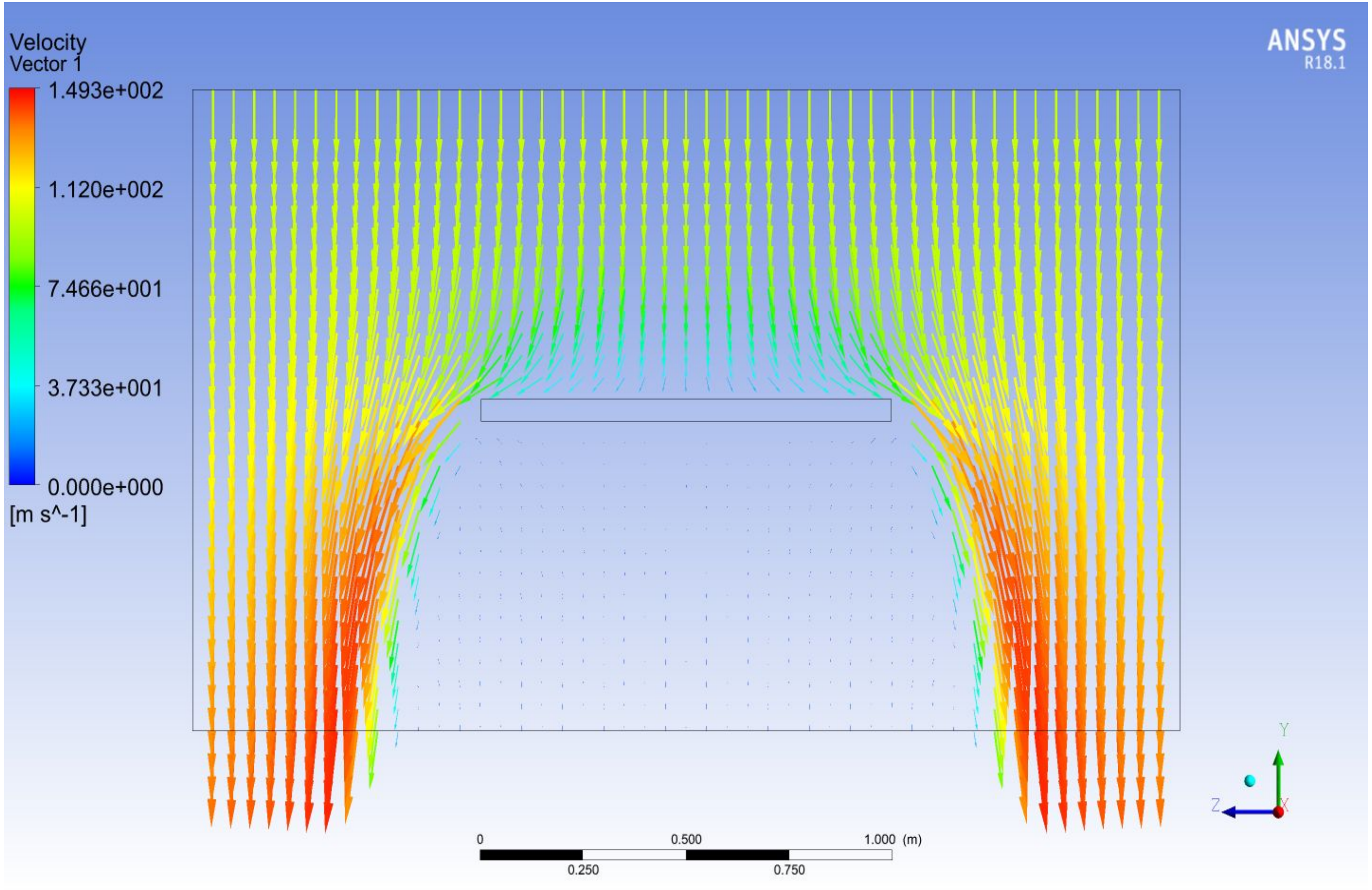
Velocity Contours



Velocity Streamlines



Velocity Vectors

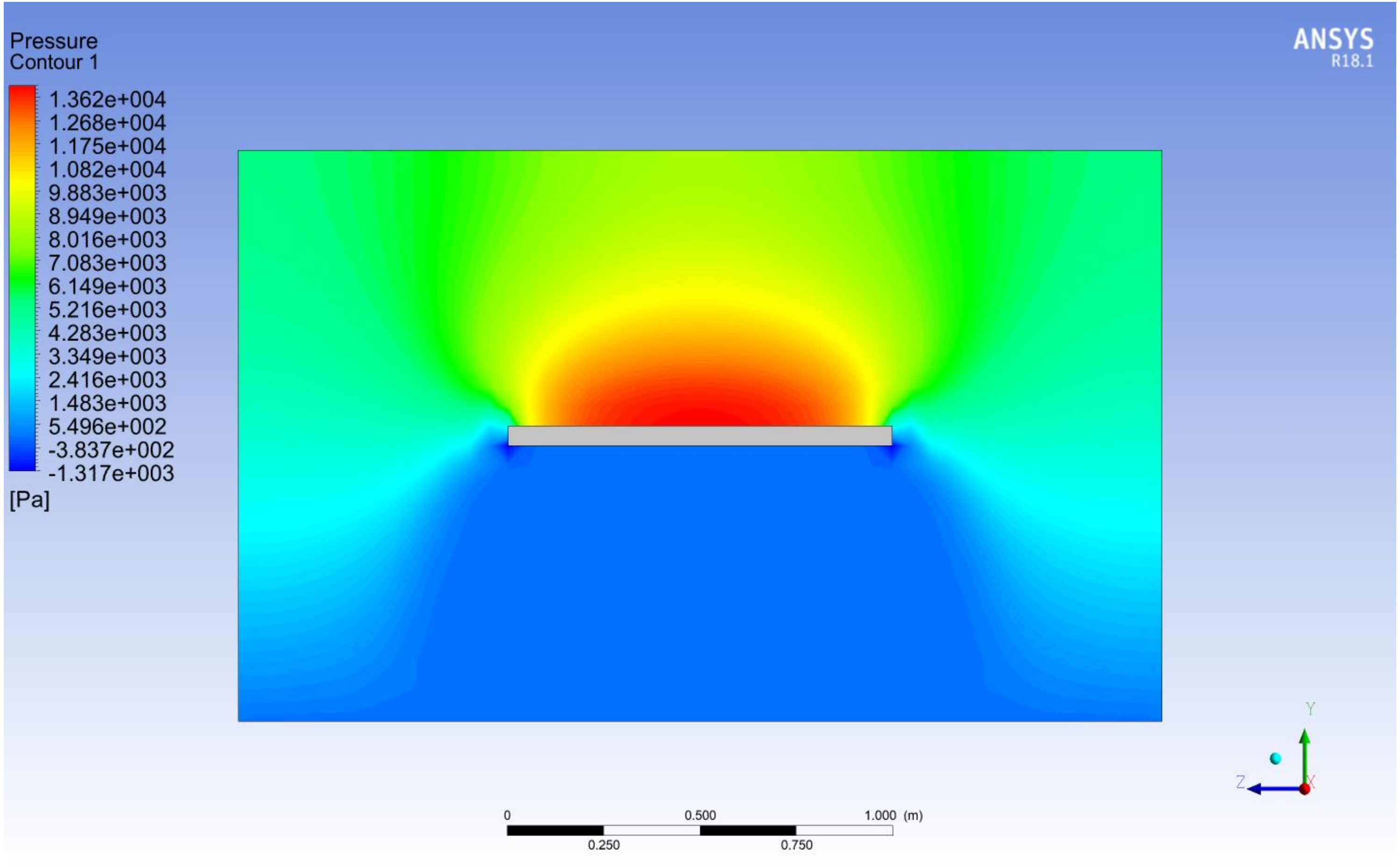


Observation from above images

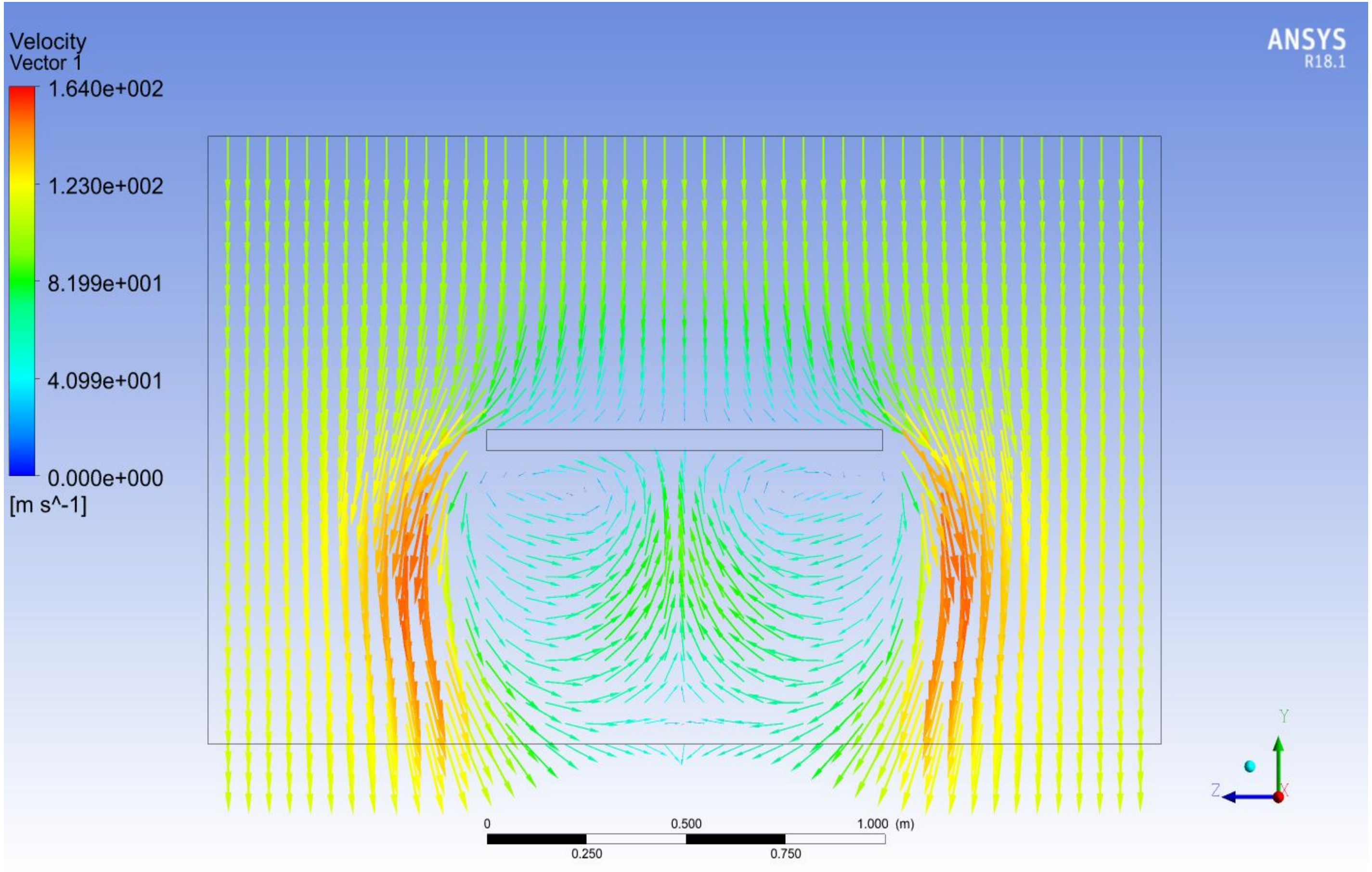
The above pictures represent the velocity contours, flow streamlines and velocity vectors when the inlet velocity is 100m/s.

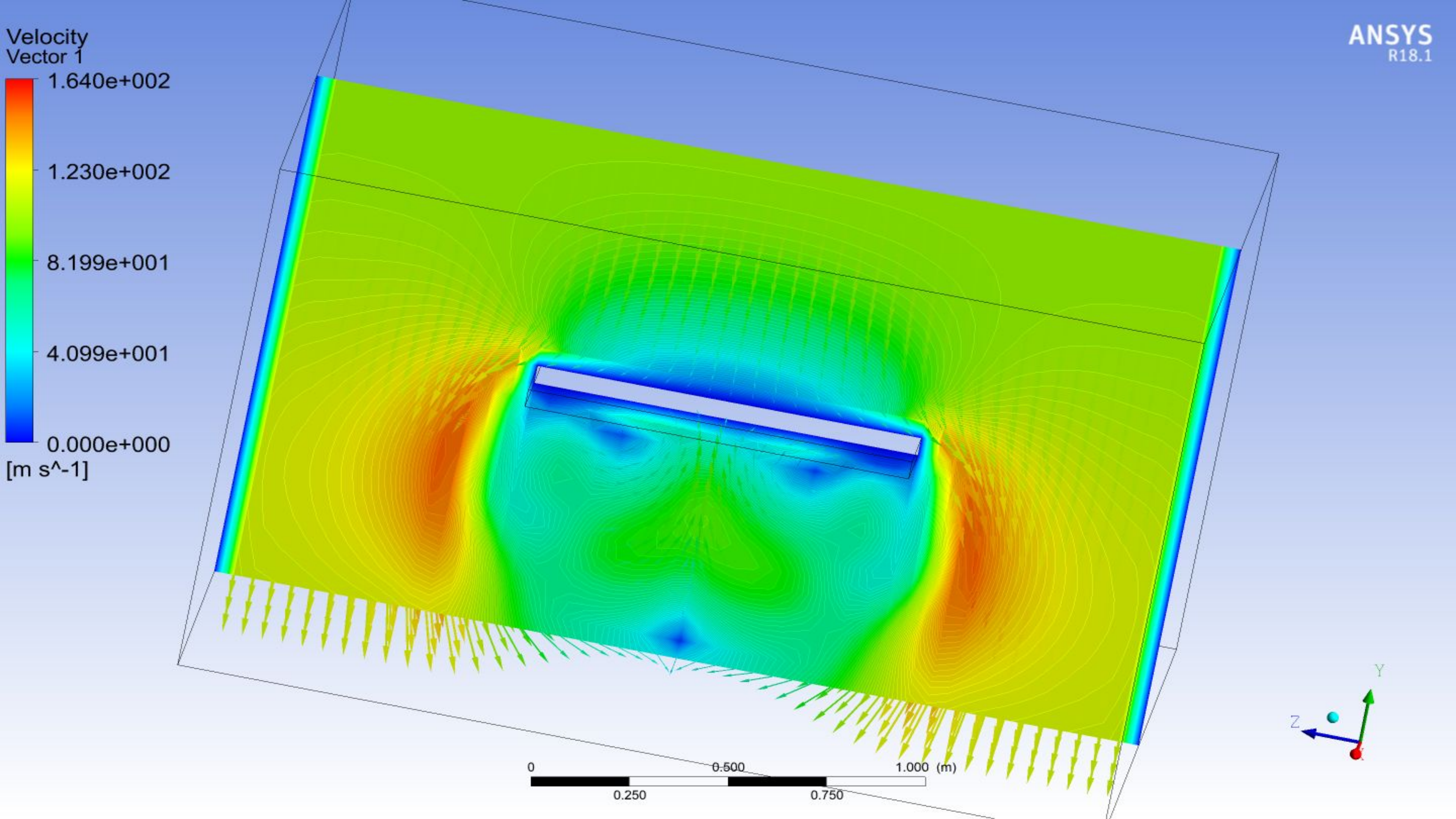
1. We can observe that flow is maximum at the inlet and near either sides of the plate.
2. Velocity is minimum right behind the plate.
3. Because of no slip condition, velocity of fluid layer in contact with the walls is zero.
4. It then increases as we move far from the walls of plates.

Pressure Contours



Vortex Shedding




Velocity
Vector 1
1.640e+002
1.230e+002
8.199e+001
4.099e+001
0.000e+000
[m s^-1]
ANSYS
R18.1
0
0.500
1.000 (m)
0.250
0.750
Y
Z
X

Effect of Reynold’s number on Vortex Shedding

In the Reynolds-number range $43 < Re < 220$, the vortex shedding disappeared for sufficiently large shear parameters.

Moreover, in the Reynolds-number range $100 < Re < 1000$, the Strouhal number increased as the shear parameter increased beyond about $0{\cdot}1$.

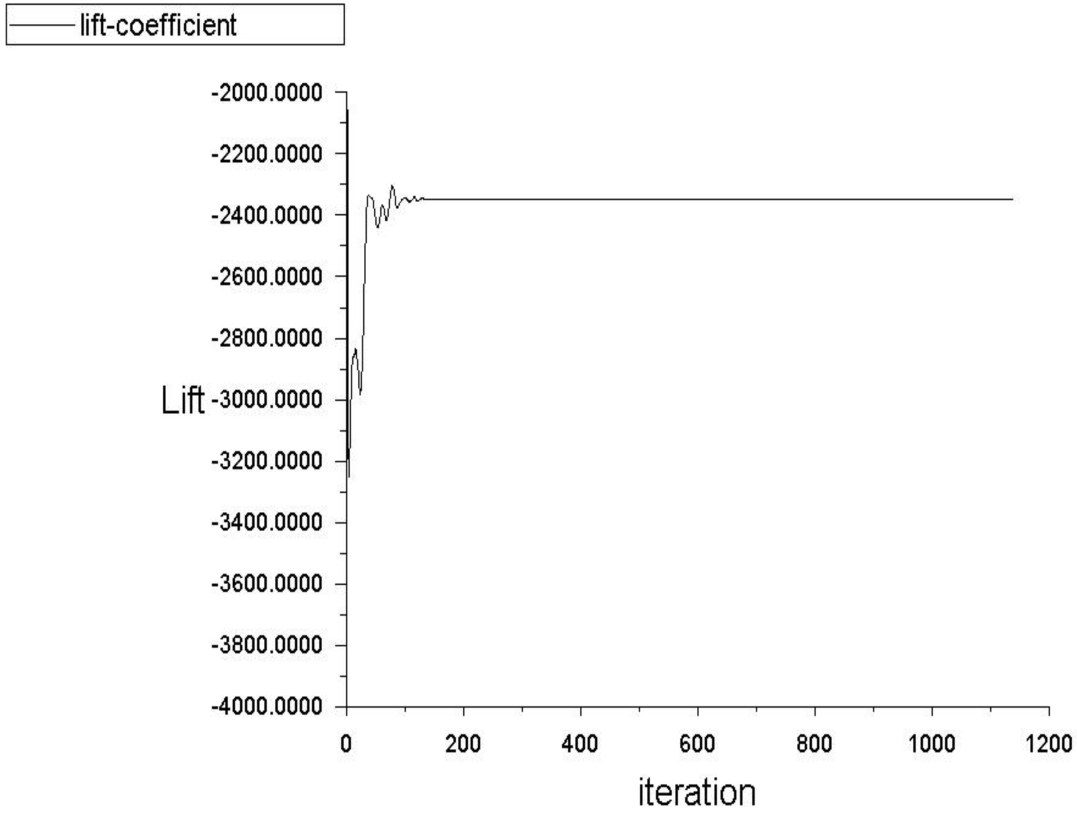
At higher Reynolds number, the shed vorticity becomes irregular and the flow becomes so turbulent that the pattern of shed velocity is no longer discernible.

Effect of Reynold’s number on lift and drag Coefficients

1. To analyse the effect of reynold’s number on the lift and drag coefficients, we have take three different inlet velocities 50m/s, 75m/s and 100m/s.
2. More the velocity, more the Reynold’s number.
3. We calculated the coefficients at these three velocities.
4. We have compared the resultant values in the following slides.

Variation in lift Coefficient

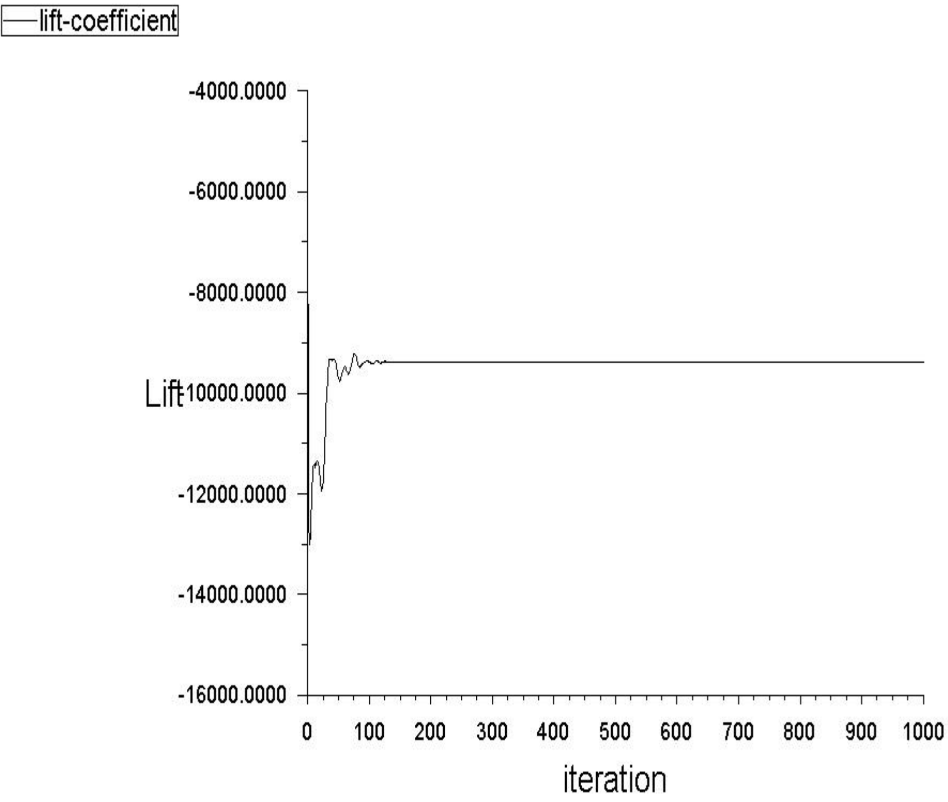
U = 50 m/s



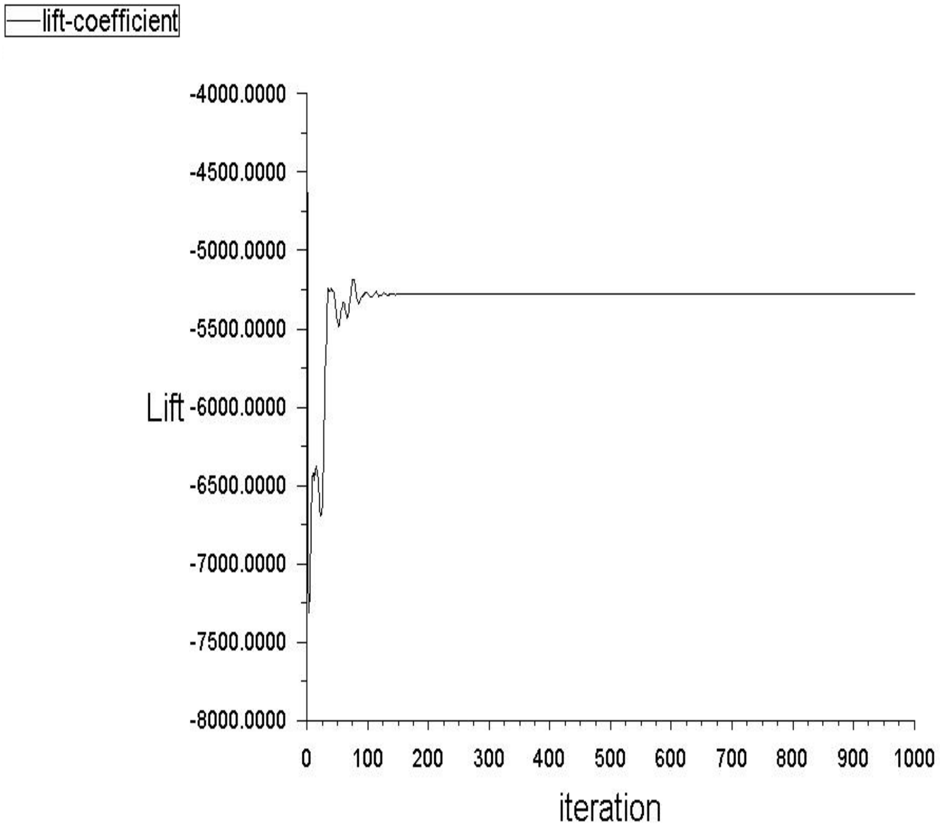
We can observe that when we increase the velocity, Reynolds number increases, so does the lift coefficient.

Lift Coefficient is directly proportional to Reynolds number.

U = 75 m/s

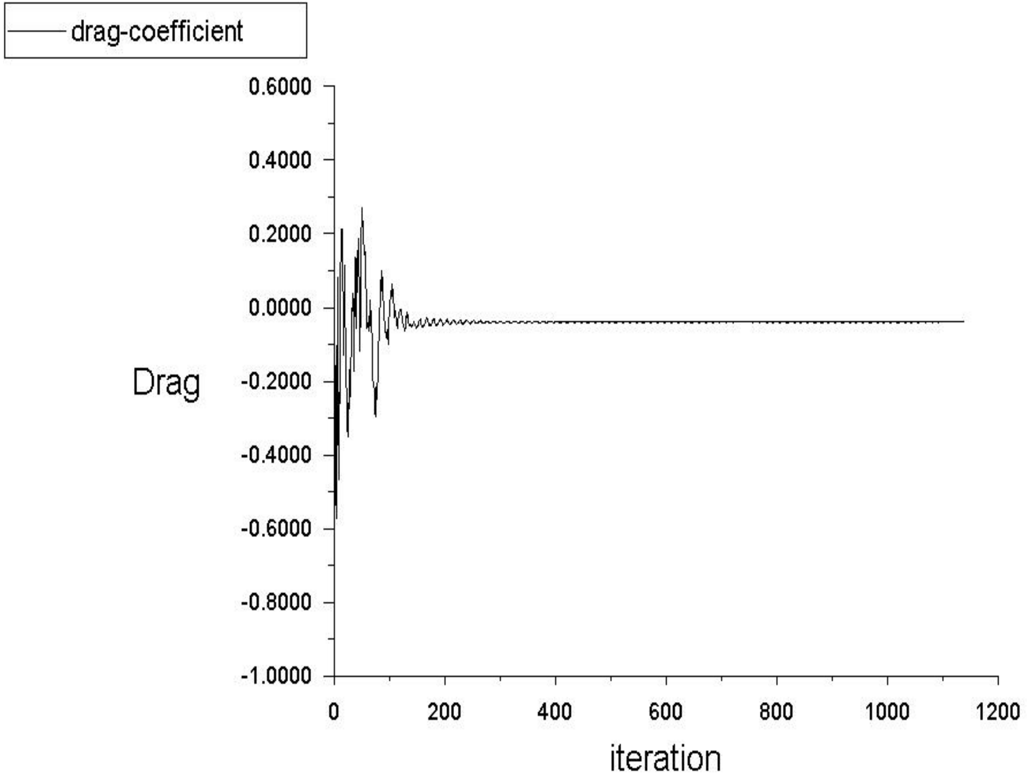


U = 100 m/s



Variation in drag Coefficient

U = 50 m/s

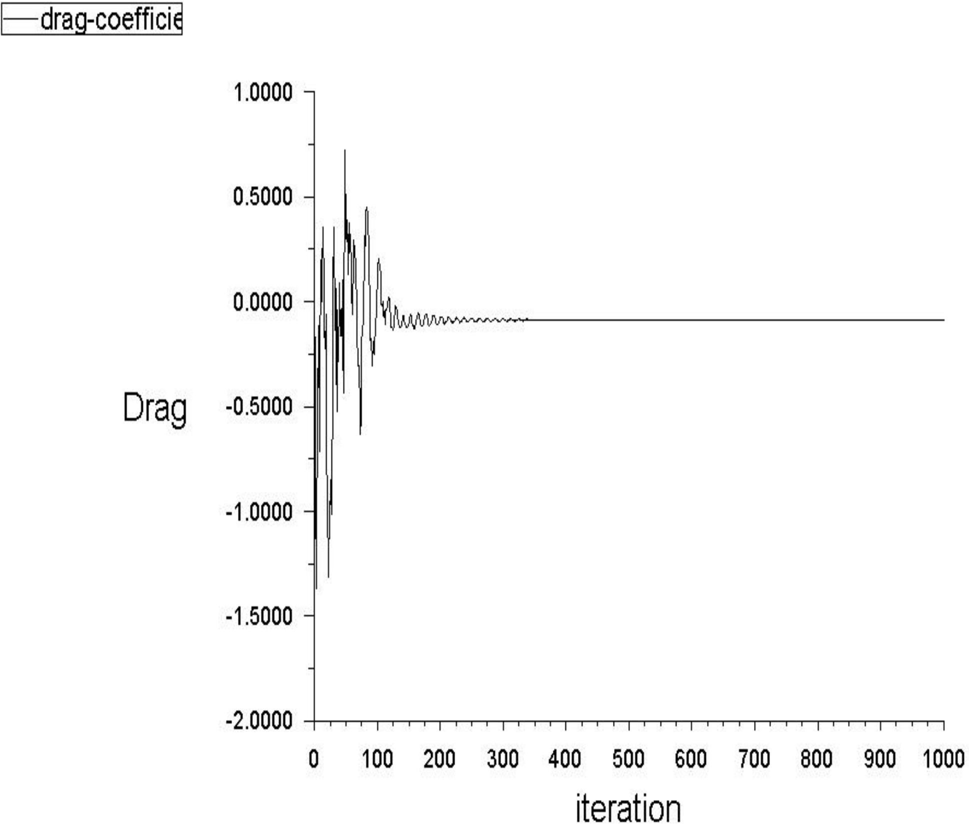


When we increase the velocity from 75 m/s to 100 m/s, we can observe that drag force coefficient decreases. The change if very small though.

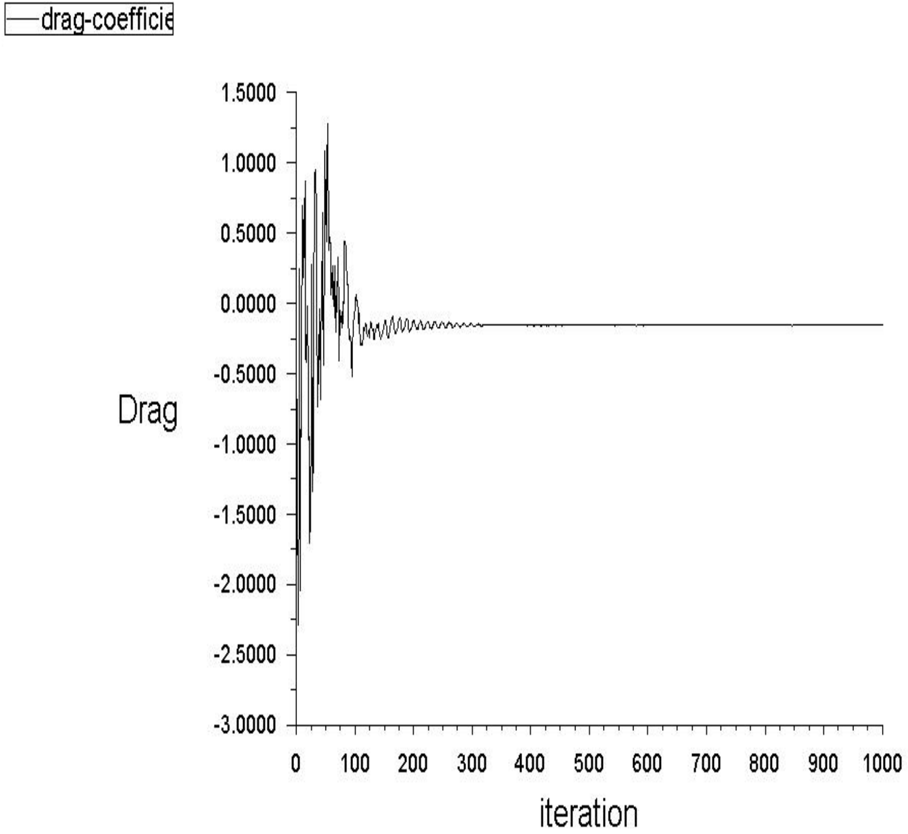
Hence, with increase in the Reynolds number, drag coefficient decreases.

They are inversely proportional.

U = 75 m/s

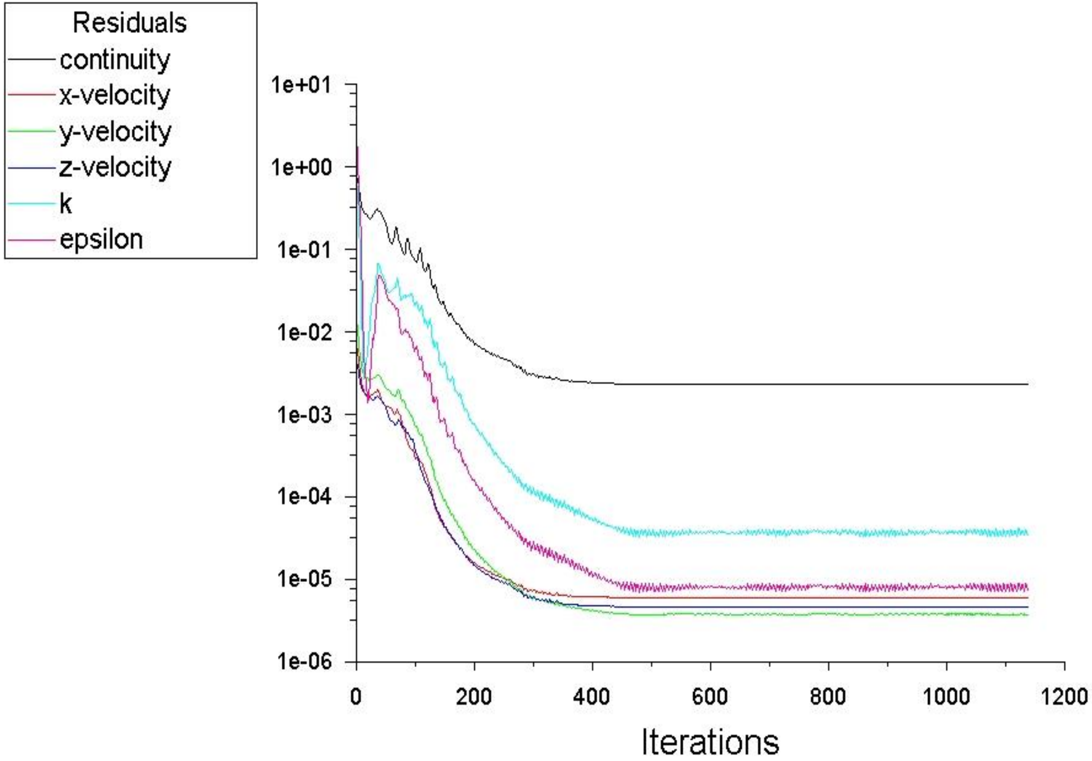


U = 100 m/s



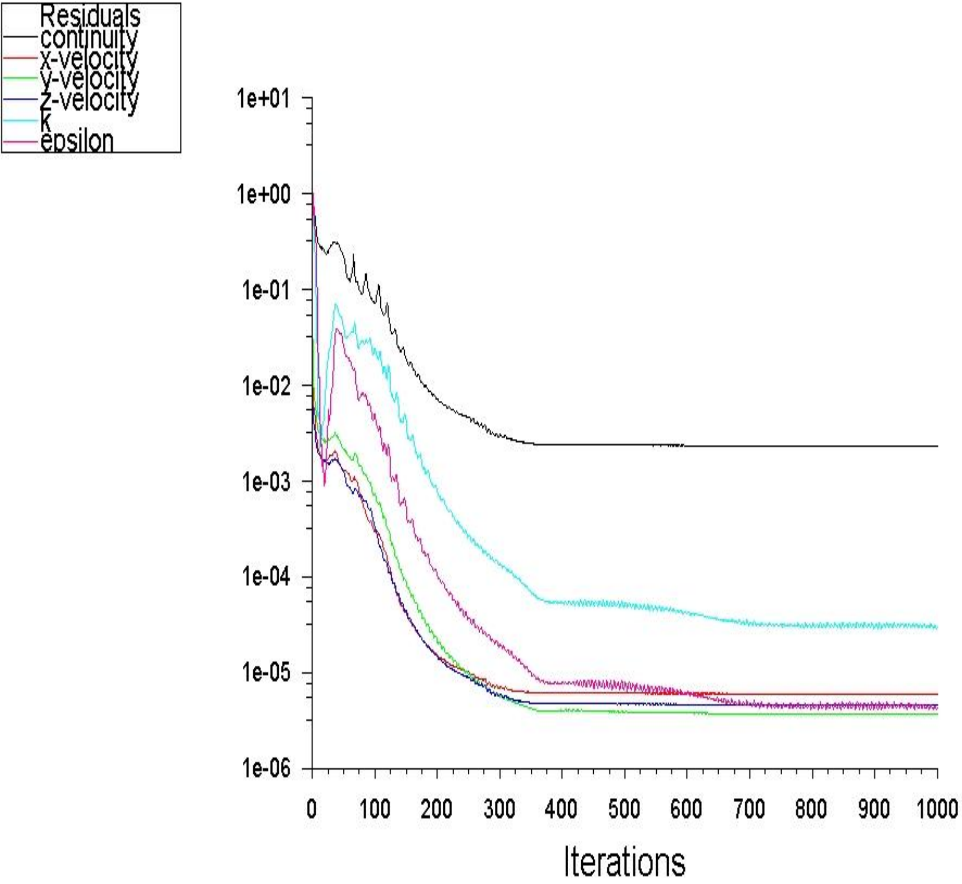
Variation in Residuals

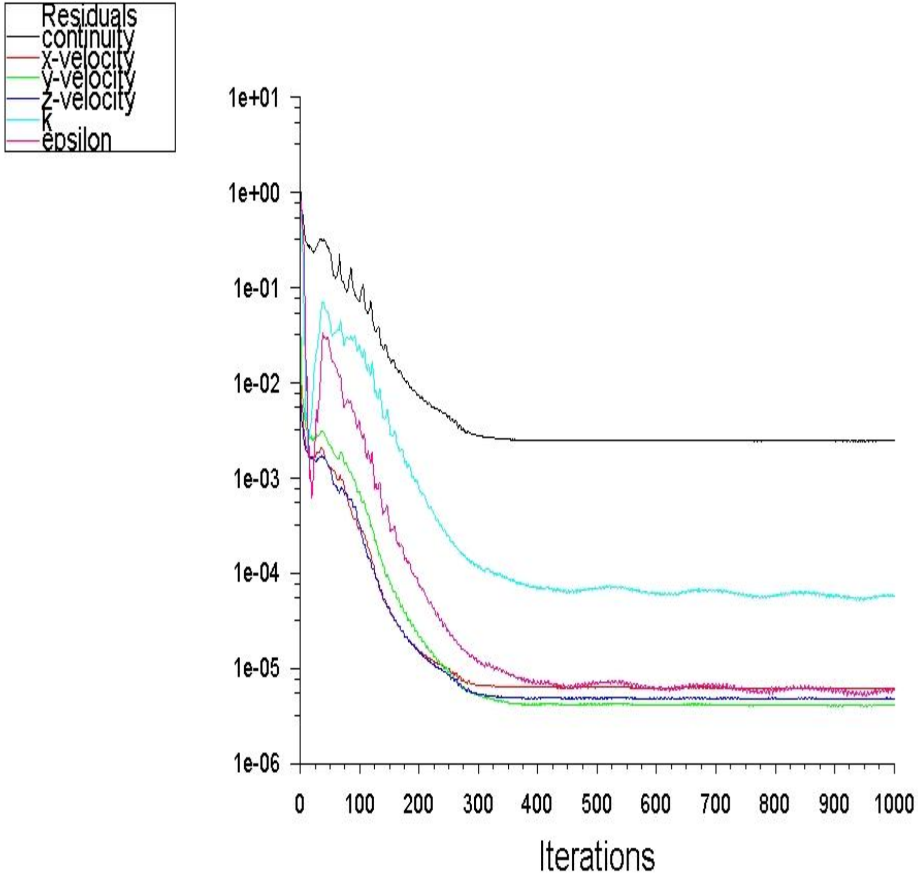
U = 50 m/s



U = 75 m/s

U = 100 m/s





Animations for visualization

Shared drive folder contains the required animations for better visualization.

All the data is taken at U = 100m/s

https://drive.google.com/drive/folders/1hv1SqKy6EefLCDJS2ABwqcKVkhS7wEIu?usp=sharing

THANK YOU