Numerical Fluid Mechanics II

Summer Semester 2018

DELIVERABLE TASK I:

Laminar flow over a square cylinder

Erlangen

Submitted By:

Kudeti Siddhartha (22433020)

Jishnu Jayaraj (22448952)

Mohammad Moataz (22455424)

Vasu Kolli (22290271)

Contents

INTRODUCTION

We modeled the incompressible viscous flow around a square cylinder using the OpenFoam toolbox. We develop a mesh with varying resolutions (500*80, 1000*160, 2000*320) as well as the bonus 3D case (500*80*20). We run our simulations for varying Reynolds number cases (50,150,310) and see results which corresponds to increase wake shedding for increasing Reynolds number.

The domain is divided into 8 blocks arranged as follows (block numbers are written in bold in the middle and vertices numbers on the corners)

9/25 8/24 **6**	7/23 **5**	6/22 **4**
10/26 15/31 **7** 11/27 12/28	14/30 **0** 13/29	5/21 **3** 4/20
8 0/16 1/17	**1** 2/18	**2** 3/19

The boundary conditions are:

- No-slip boundary at walls (u=v=0)
- Uniform Stream wise velocity profile at the inlet
- Neumann boundary condition at the outlet

Since we are interested in studying variations happening around the cylinder, it was convenient to use a non-uniform grid where the grid is finer near the cylinder and coarser away from it. This will ensure more accurate results for flow variables (P, u).

Grid Convergence Index Study

We created three sets of grid varying the grid density by a factor of 2.

Mesh Density	Refinement Factor
Fine	1
Medium	2
Coarse	4

From our simulations we have generated data for pressure along the center line and recirculation length along the centerline (at t = 2seconds)

Pressure values for each were as follows

	Grid 1 (2000*320)	Grid 2(1000*120)	Grid 3(500*80)
Pressure	2.7506e-5	2.9612e-5	2.9756e-5

By calculating GCI we got (GCI21=-0.1027 and GCI32=-0.0065) and we were able to verify that the equations are being solved correctly by a confidence interval of 0.9289.

Even after refining the mesh several times and comparing the solutions the error from discretization was negligible compared to computational cost. With Grid convergence study we were able to quantify the improvement and provide insight to actual quality of fine grid.

Recirculation length

	Grid 1 (2000*320)	Grid 2 (1000*120)	Grid 3 (500*80)
Recirculation Length	0.00814	0.00805	0.00675
Error compared to Grid 1	0%	1.1%	17.07%

The results show that just by doubling the grid the error reduced drastically from 17% to 1% which is acceptable and that further refining of the grid will not add more value to the accuracy.

Pimple-Foam and Relaxation Factor

The PIMPLE Algorithm is a combination of PISO (Pressure Implicit with Splitting of Operator) and SIMPLE (Semi Implicit Method for Pressure Linked Equations). PIMPLE algorithm is SIMPLE algorithm for every time step, where outer correctors are the iterations, and once converged will move on to the next time step until the solution is complete. If we don't give the value for n-Corrector its by default 1 (in PISO mode).

Relaxation factor improves the stability of computation. By under relaxation we limit the change in particular variable in each iteration. Low relaxation factor means the solution will be stable but gives a slow convergence. If the relaxation factor is high then solution will be instable but provide high convergence.

Since the given problem was found to be stable for given flow parameters we have updated the relaxation factor to maximum so we get maximum convergence

Vortex shedding

Vortex shedding occurs when fluid flows past a bluff (un-streamlined) body, causes the flow to separate from the surface of the structure rather than follow the body contour. Beyond a critical Reynolds number the flow gets separated and this causes a low pressure region close to the body making the velocity opposite to flow direction, this causes wake formation. And if the Reynolds number is further increased the wakes become unstable and a transverse oscillation starts near the end of the wake. If the Re is increased further, the vortices are shed alternately from the upper and lower cylinder surface at a definite frequency depending on Reynolds number.

The shed vortices produce a force that acts on the body in cross stream direction. At relatively low Reynolds number the spiral vortices are created periodically and symmetrically from both sides of the body. Since these vortices are symmetrical they cancel each other out and effect of vortex shredding can be ignored. However, at higher Reynolds number, the vortices are shed alternatively. As a result, alternating low pressure zones are formed on downstream of the body and a fluctuating force is created.

Results and Interpretation

Stream wise u and Cross-Stream wise v along the centerline (y=0) behind the cylinder

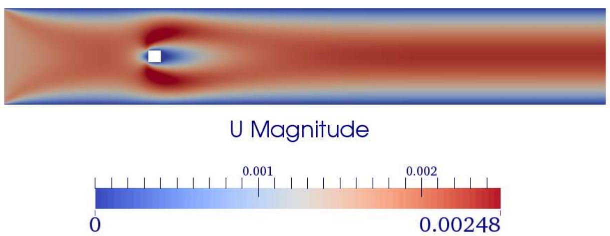


Fig.1. Velocity contour at Re=50

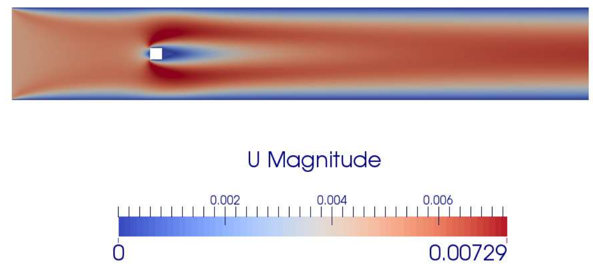
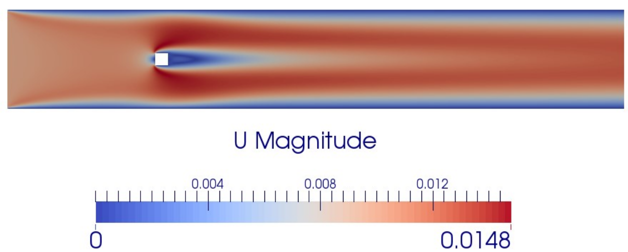


Fig.2. Velocity contour at Re=150



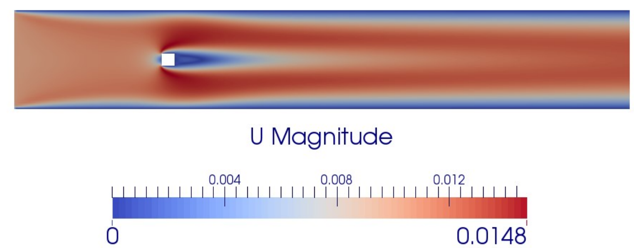


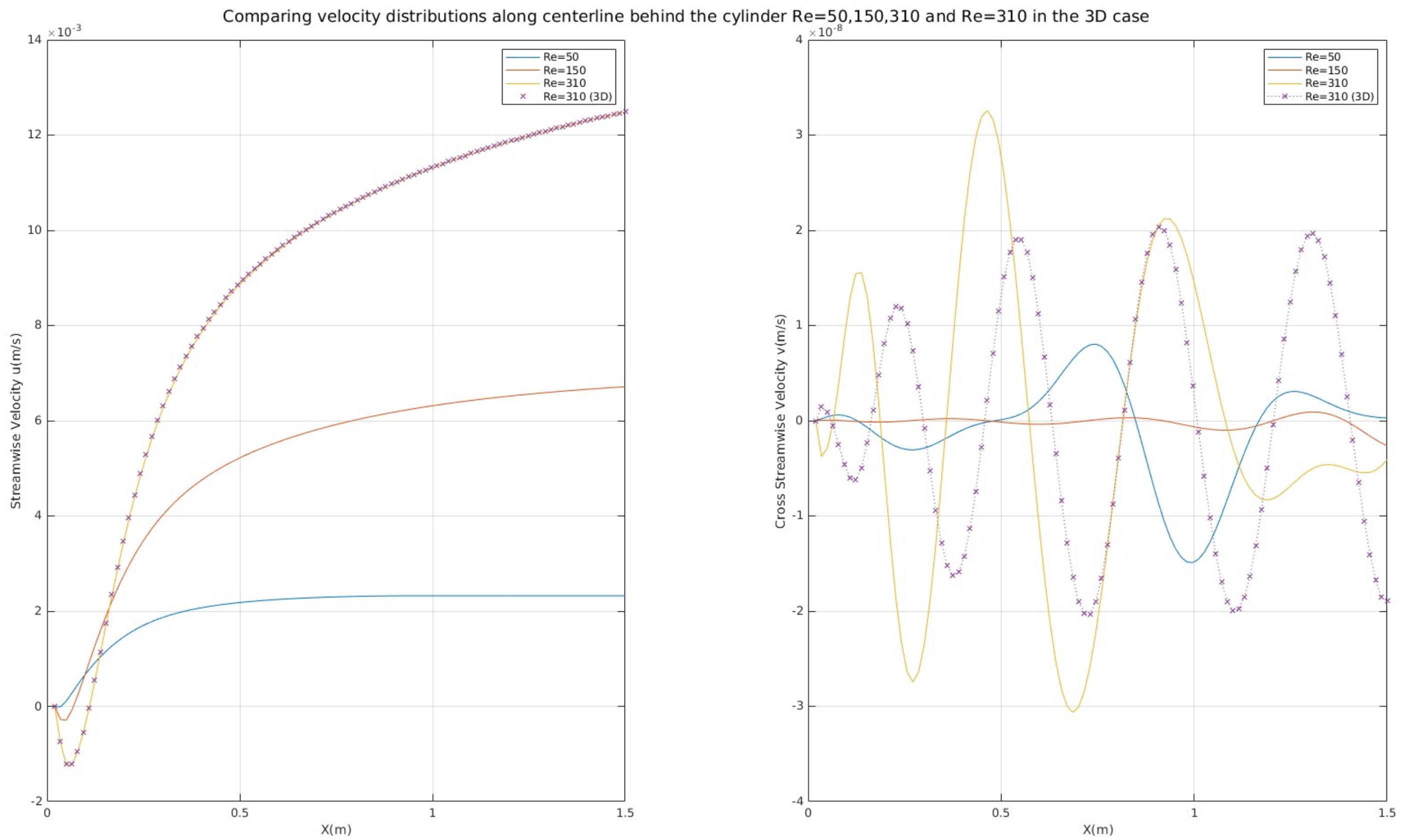
Fig.3. Velocity contour at Re=310

Fig.4. Velocity contour at Re=310 (3-D)

General observation:

- Velocities = 0, Along the surface of square block and on the domain wall because of the no slip condition.
- Maximum velocities are located at the top and bottom of the cylinder due to the reduction in the cross section area.
- The maximum velocities values are increases as Re increases.
- The re-circulation length increases as Re increases.

Velocity Distributions along the center line behind the cylinder



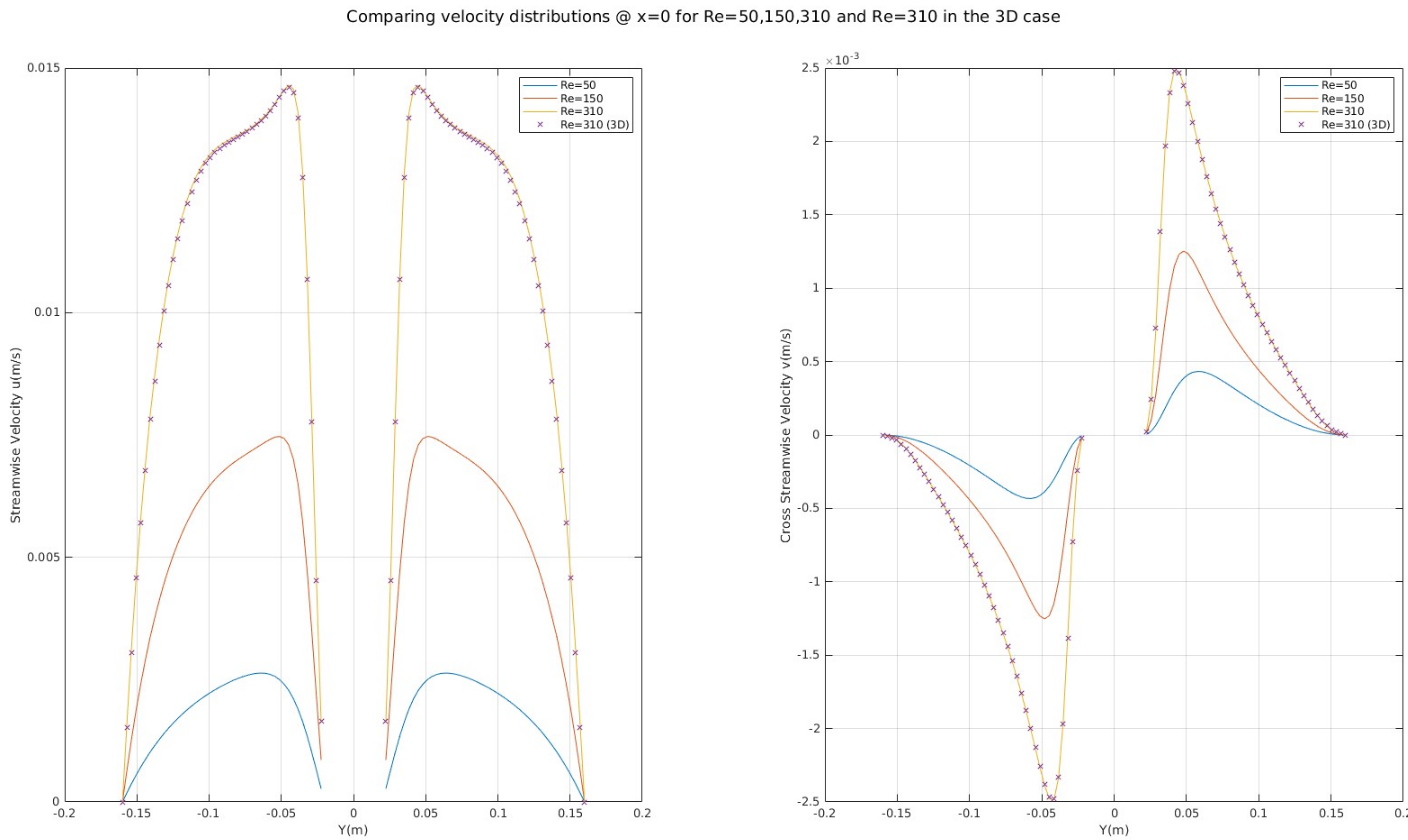
Streamwise velocity (u)

- The velocity is zero on the wall due to the no slip boundary condition.
- The negative velocity just behind the wall shows that there is recirculation of the stream occurring. The recirculation length is the length behind the wall is the point where the flow changes its direction (the point at which the curve crosses the x-axis) .
- The velocity increases gradually after the recirculation as we go farther from the cylinder.
- The negative velocity behind the cylinder keeps increasing with Re. So, the recirculation length is greater for higher Reynolds numbers.

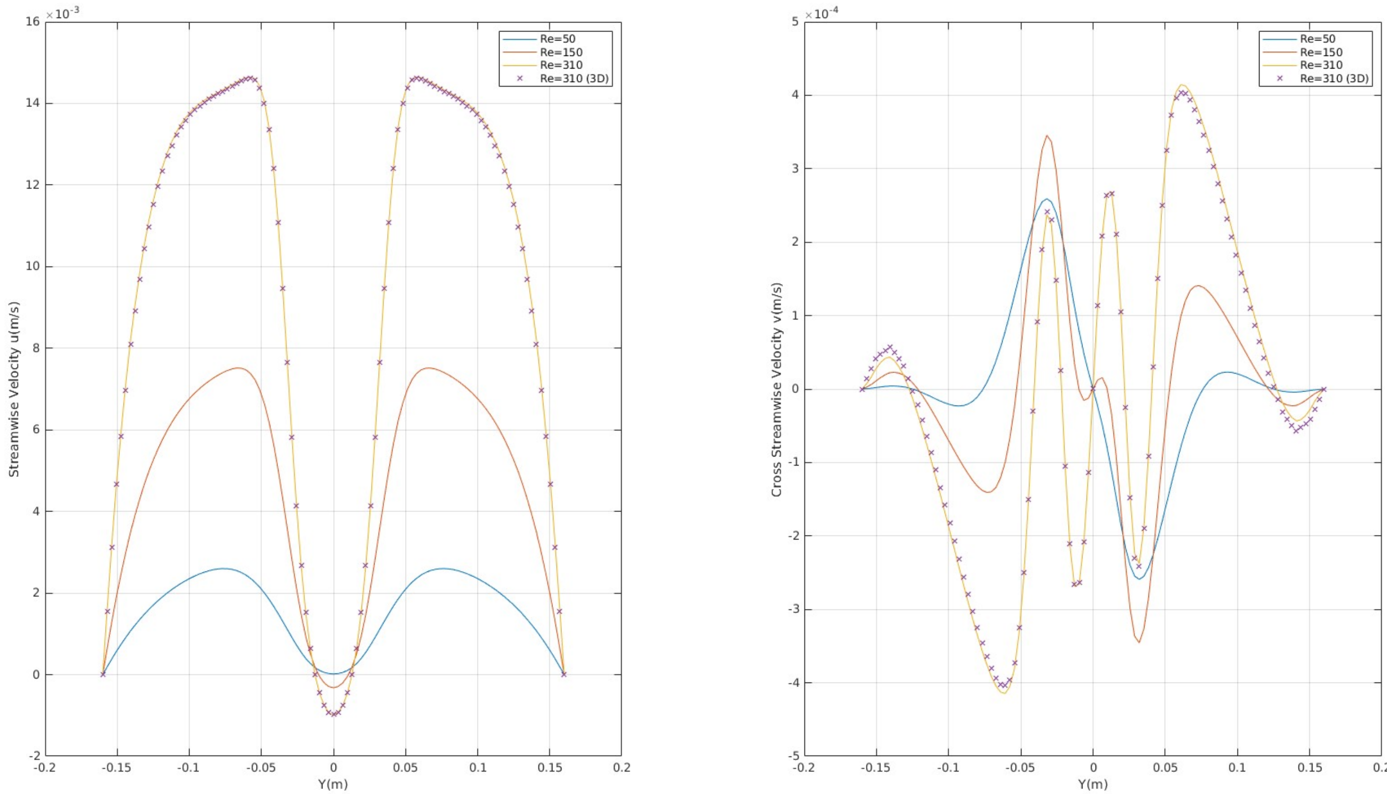
Cross-Streamwise velocity

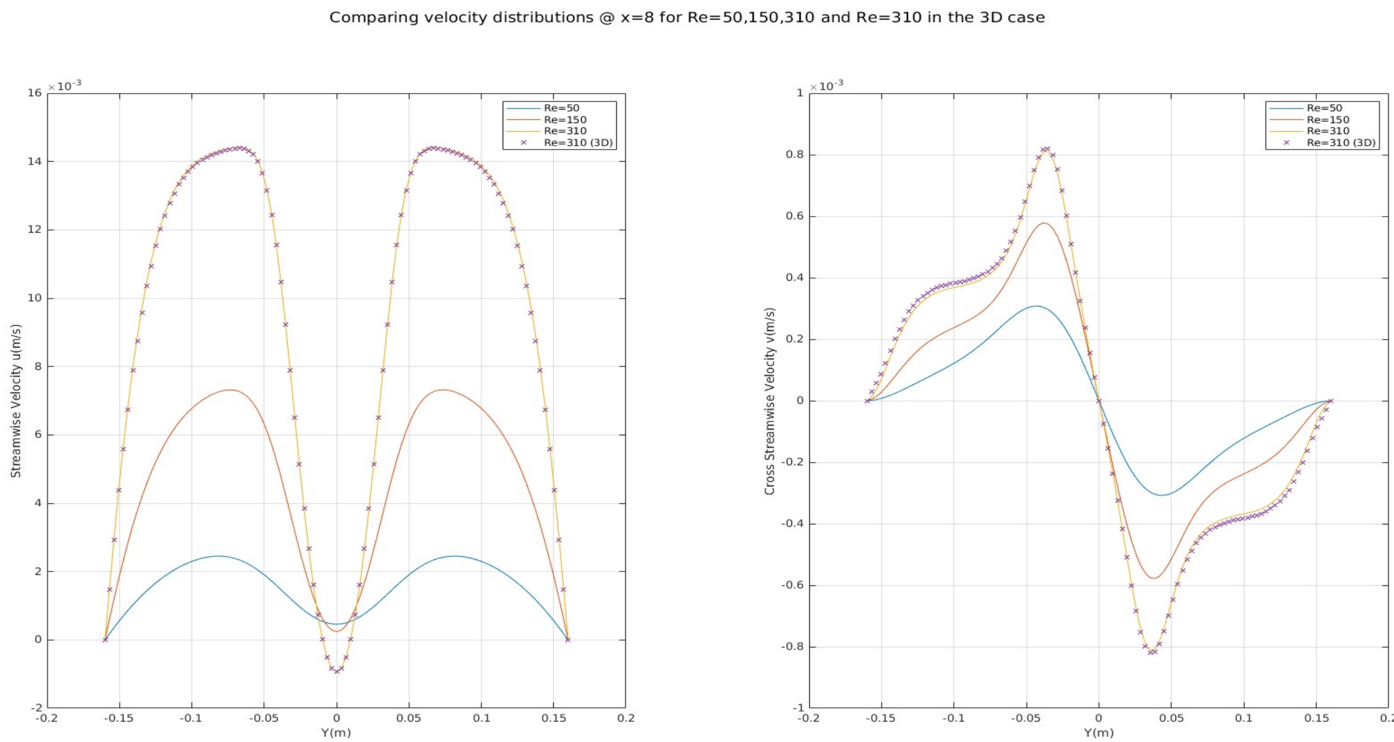
- The wake occurs close to the square cylinder for higher Reynolds number.
- But for the lower Reynolds number the wake occurs little bit farther away from the cylinder.
- The wave pattern occurring behind the cylinder, downstream the flow is due to the pressure differences of the fluid up and down the cylinder.
- In this case, when the wake width is smaller than the integral length scale as seen in the highest Reynolds number the wake grows linearly with distance downstream (i.e. proportional to x), whereas when the wake width is larger than the integral length scale as seen in lowest Reynolds number, it grows diffusively (i.e. proportional to $x^{1/2}$).

Velocity contour at locations x=0,4,8



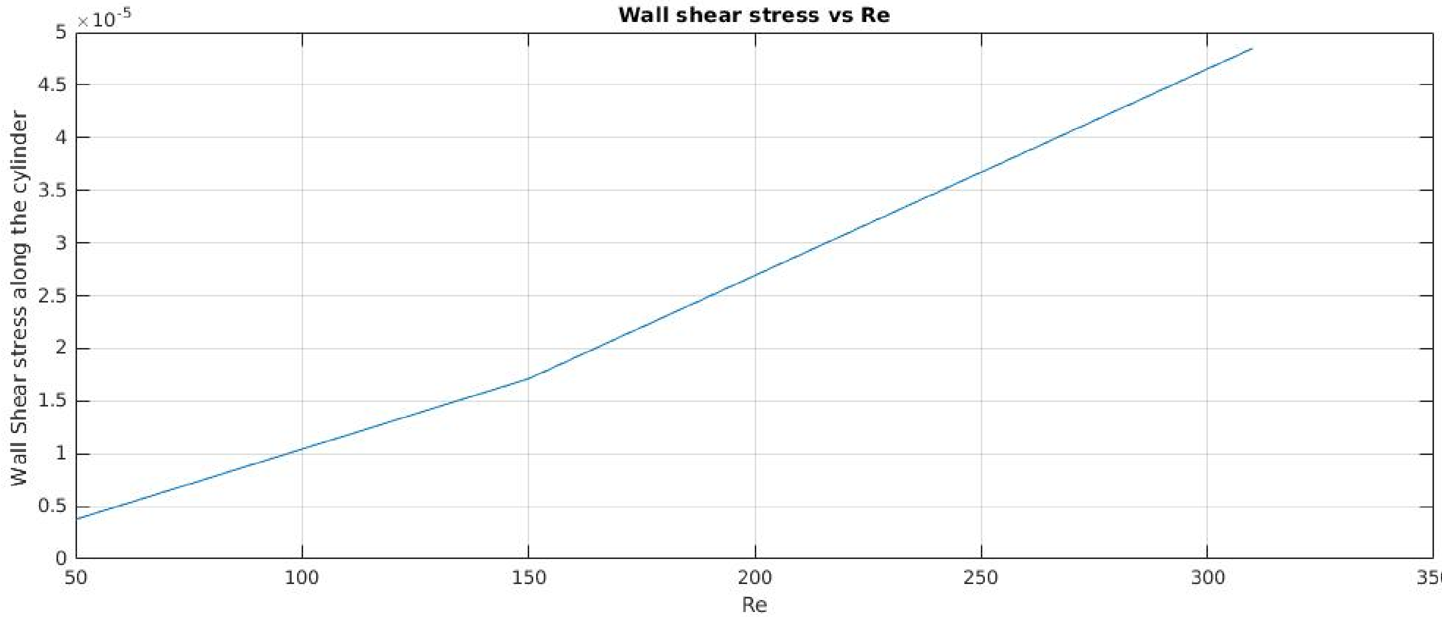
Comparing velocity distributions @ x=4 for Re=50,150,310 and Re=310 in the 3D case





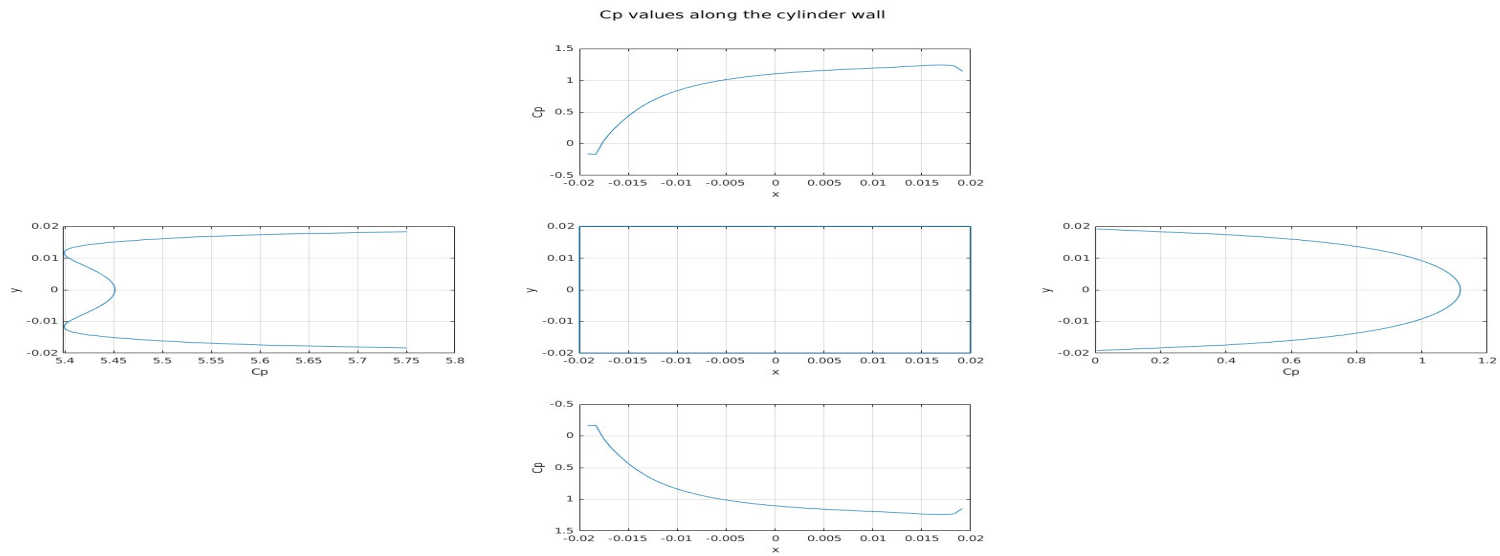
- In all the above plots the general shape of the graph remains same, only the magnitude and position of the maximum velocity changes.
- The magnitude of maximum velocity increases with increase in Reynolds number, Also they shift closer to cylinder walls with increase in Reynolds number
- The frequency of change of cross-stream velocity is high at x=4 but it diffuses as flow progresses to x=8

Wall Shear Stress



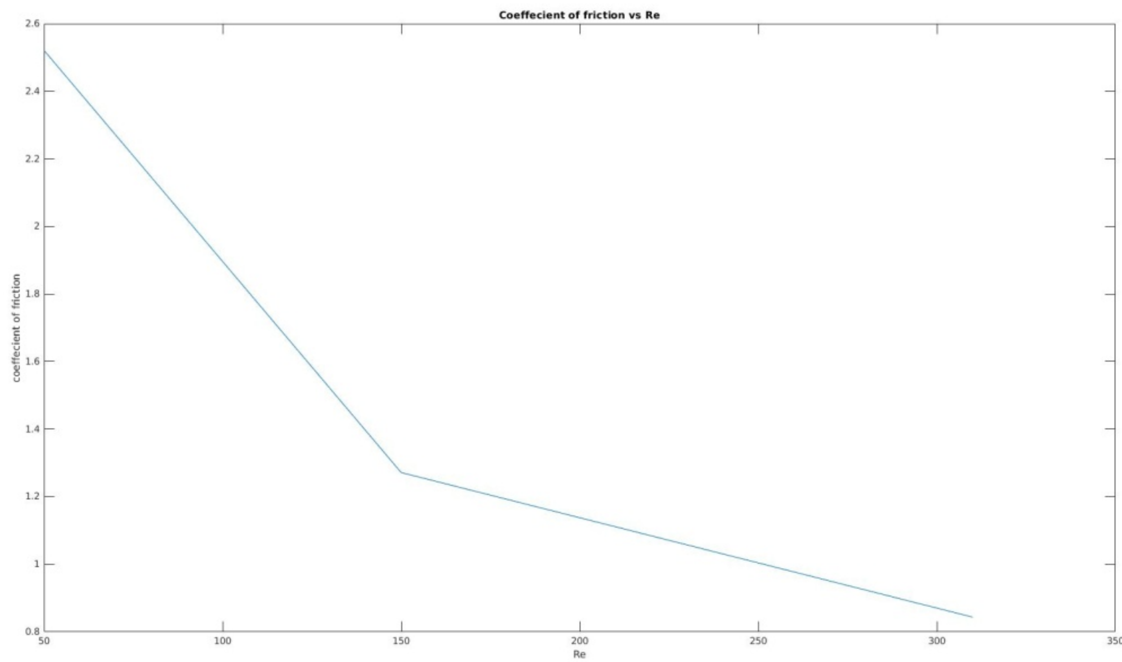
- The shear stress depends on the normal gradient of the tangential velocity (dU/dy) in a layer near the wall called the viscous (sub)-layer.
- With increasing Reynolds number the wall shear stress increases.

Pressure Coefficient



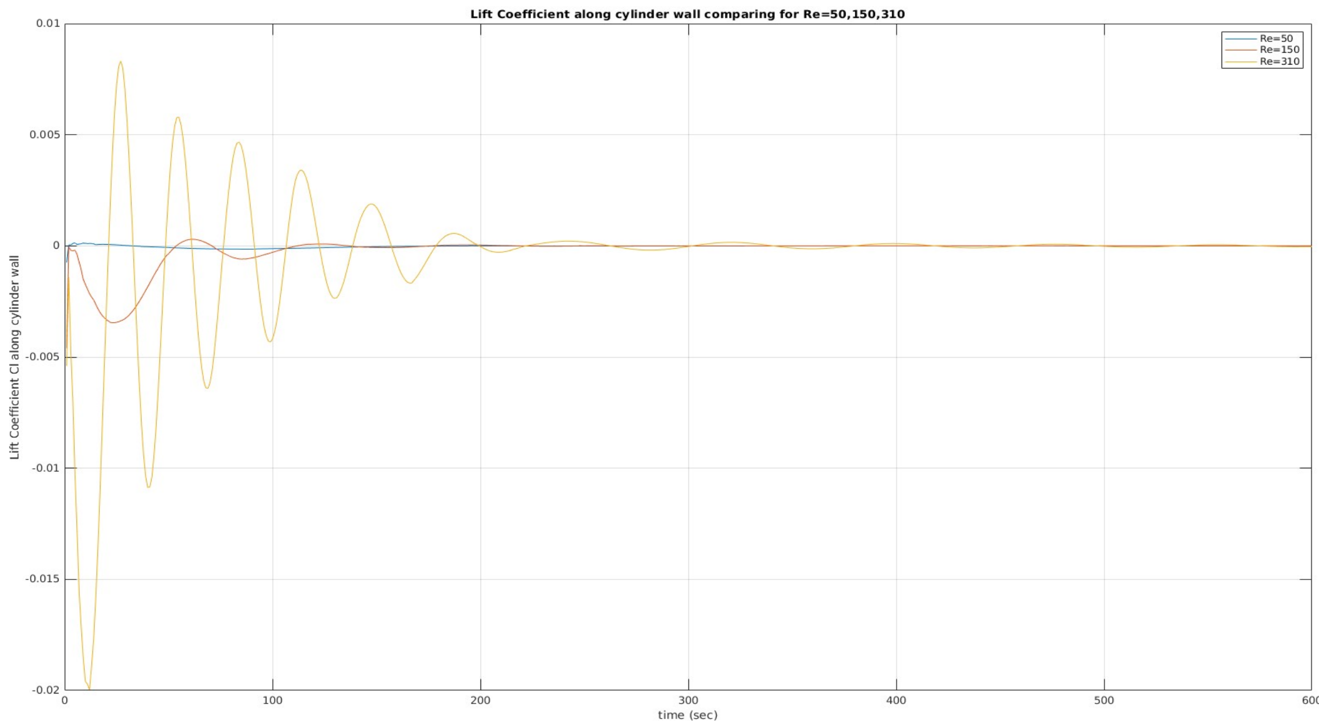
- The pressure coefficient is maximum behind the square block at the centre and reduces symmetrically as we move around the edge.
- The pressure coefficient increases along the flow direction on the top and bottom surface.

Coefficient of friction (Cf)



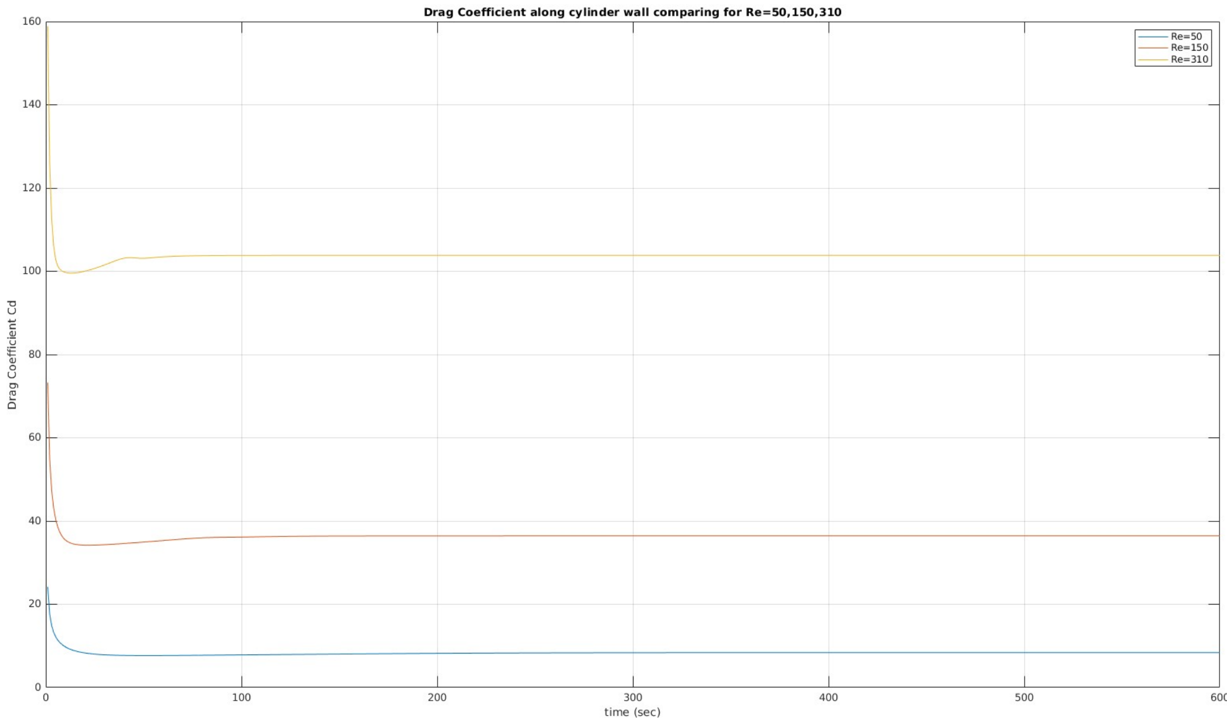
- The coefficient of friction decreases with increase in Reynolds number.
- This is due to higher stream velocities for flows with higher Re and as the coefficient of friction is inversely proportional to the square of free stream velocity.
- While the turbulent layer grows with the increase in Re the laminar layer thickness decreases. These results in a thinner laminar boundary layer which, relative to laminar flow, depreciates the magnitude of friction force as fluid flows over the object.

Coefficient of lift



- Coefficient of lift quantifies the amount of lift force or thrust experienced by the body in fluid stream.
- Higher Cl means higher lift force. In the plot we can see that there is an oscillating component to the lift force along the length of the fluid flow for higher velocity. This is because at higher velocity wakes are formed which oscillates downstream and this creates a oscillating thrust force.

Drag coefficient (Cd)



- Drag coefficient is a dimensionless quantity that quantifies the amount of resistance or drag experienced by a body in fluid flow environment.
- In the plot of Cd we can see that as the velocity of fluid increases (higher Reynolds number) the coefficient of drag also increases.

Strouhal number

Strouhals number is a dimensionless parameter that describes oscillation in flow over bodies. The Strouhal Number can be important when analyzing unsteady, oscillating flow problems. The Strouhal Number represents a measure of the ratio of the inertial forces due to the unsteadiness of the flow or local acceleration to the inertial forces due to changes in velocity from one point to another in the flow field. The Strouhal Number can be expressed as

$St = \omega l / v$ *where,*

St = *Strouhal Number*

ω = *oscillation frequency*

l = *characteristic length*

v = *flow velocity*

- For Re = 310 there is an oscillating velocity component for cross-stream velocity, using this frequency we get a Strouhal number = 0.054.

Conclusion

After simulating an incompressible, viscous flow around a cylinder using PIMPLE solver in Open Foam. Computations have been carried out for Reynolds number 50, 150 and 310 having different flow regimes. And calculated different flow parameters at different locations of the flow field and the following conclusions have been made:

- At very low Reynolds number the flow was creeping flow and the streamlines passed around the body and joined downstream forming the normal laminar flow regime
- As Reynolds number increases the flow stream lines undergo flow separation from the body and creates disturbance in downstream.
- These vortices formed at higher Re are stable vortices which diffuse in further downstream. But if the Reynolds number is further increased then it leads to formation of laminar vortex streets.
- The results for the 3D simulation for Re=310 can be seen in the velocity plots. We can notice that the stream wise velocity curve of Re=310(3D) is similar to that of 2D.
- We can notice a change in Cross-stream velocity curve of Re=310(3D) with 2D velocity plot because there is a flow in z-direction due to which the magnitude of oscillations of vertical velocity(v) component reduces.