Flow past circular cylinder

by
Suryansh Prakhar

CFD Final Project

Course Instructor: Prof. Rajat Mittal

Johns Hopkins University
May 2022

Contents

List of Figures

1 Introduction

Flow past circular cylinder is one of the simplest flow problem that involves internal boundary. It is also a good first choice to test any CFD code since numerous published results exists to verify the results. The problem setup is shown in Fig 1 . Cylinder diameter is 1 and is used to normalize length scales in the problem. Domain dimensions are taken as 12D x 6D unless other wise mentioned.

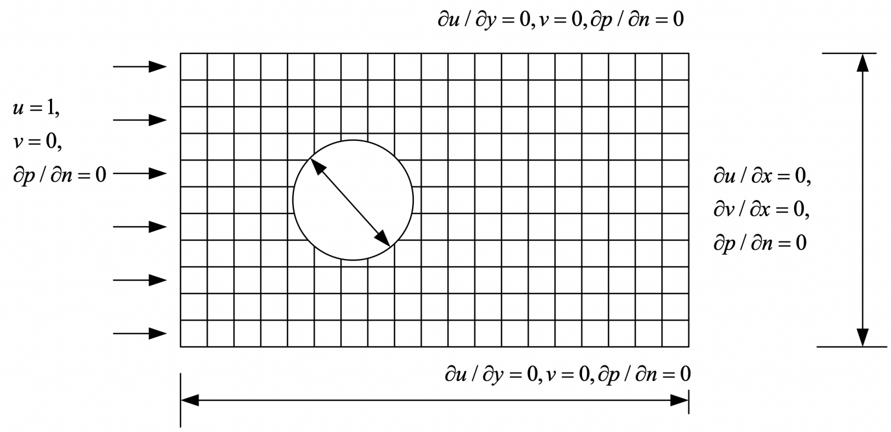


Figure 1: Problem sketch

2 Numerical Scheme

2.1 Equations

Navier Stokes equation is broken into two parts using Fractional step method. Thus we have three set of equations to solve, namely Advection diffusion equations

$$\partial_t u + \partial_x(Uu) + \partial_y(Vu) = \frac{1}{Re}[\partial_x^2 u + \partial_y^2 u]\,, \tag{2.1}$$

$$\partial_t v + \partial_x(Uv) + \partial_y(Vv) = \frac{1}{Re}[\partial_x^2 v + \partial_y^2 v]\,, \tag{2.2}$$

and Pressure Poisson equation

$$\partial_x^2 p + \partial_y^2 p = \frac{1}{\Delta t}[\partial_x u + \partial_y v]. \tag{2.3}$$

Both space and time coordinates are discretized to second order accuracy assuming non uniform spacing. Time is discretized using Adams–Bashforth 2nd order accurate with respect to convective term and Crank-Nicolson 2nd order accurate with respect to diffusion term. All spatial discretization is done using central differencing. Thus Momentum equations become,

$$\frac{u^* - u^n}{\Delta t} + \frac{3}{2}\partial_x(Uu)^n - \frac{1}{2}\partial_x(Uu)^{n-1} + \frac{3}{2}\partial_y(Vu)^n - \frac{1}{2}\partial_y(Vu)^{n-1} = \frac{1}{2Re}[\partial_x^2 u^* + \partial_x^2 u^n + \partial_y^2 u^* + \partial_y^2 u^n]\,, \tag{2.4}$$

$$\frac{v^* - v^n}{\Delta t} + \frac{3}{2}\partial_x(Uv)^n - \frac{1}{2}\partial_x(Uv)^{n-1} + \frac{3}{2}\partial_y(Vv)^n - \frac{1}{2}\partial_y(Vv)^{n-1} = \frac{1}{2Re}[\partial_x^2 v^* + \partial_x^2 v^n + \partial_y^2 v^* + \partial_y^2 v^n]\,, \tag{2.5}$$

Here * denotes intermediate step velocities, n and n-1 velocities at 2 previous time steps. We compute mass flux at inlet and outlet using u^* and get $m_{diff} = m_{out} - m_{in}$. This value is substracted from u^* at last cell center column, $u^*(nx, :)$. This ensures compatibility condition is satisfied.

The pressure poisson becomes

$$\partial_x^2 p^{n+1} + \partial_y^2 p^{n+1} = \frac{1}{\Delta t}[\partial_x u^* + \partial_y v^*]. \tag{2.6}$$

For any quantity ϕ, the first order derivatives are calculated using

$$\partial_x \phi = \frac{\phi_e - \phi_w}{x_e - x_w} \tag{2.7}$$

Small n,s,e and w denote quantities on face while capital N,S,E and W denote quantities at cell center. If ϕ is only available at cell center, it is first interpolated to get values at face and then derivatives are calculated. To calculate second order derivatives,

$$\partial_x^2 \phi = \frac{\partial_x \phi|_e - \partial_x \phi|_w}{x_e - x_w} = \frac{\frac{\phi_E - \phi_P}{x_E - x_P} - \frac{\phi_P - \phi_W}{x_P - x_W}}{x_e - x_w} \tag{2.8}$$

u,v and p values are stored at cell center and in addition to that we also need to store face velocities for u and v.

Since mesh is non uniform, we first need to define weights $\lambda_e, \lambda_w, \lambda_n, \lambda_s$ for each cell which will be used in linear interpolation to get cell center values at face for computing derivatives.

A matrix called iBlank is stored which is 0 inside solid cells and 1 in fluid cells. In addition to that we also need IM,IP,JM,JP matrix to know location of solid - fluid boundaries. This value will be 1 if neighboring cell is a boundary, and this matrix will help in modifying coefficients for Gauss Sidel iterative solver. Since internal body in this project is always at rest, so if a cell just outside body has velocity u, then the cell just inside solid will have velocity -u. Thus we can modify coefficients in equations by multiplying values at ϕ_e, ϕ_w etc face by (1-IP(i,j)), (1-IM(i,j)) etc to make them 0 near boundary and these terms coefficient must be added to ϕ_p term by multiplying them by IP(i,j), IM(i,j) etc .

Gauss -Sidel is used as iterative solver for both ADE and PPE. The convergence criteria for PPE is 10^{-3} and for ADE is 10^{-5}. dt is taken as **0.001** for all simulation results in this project.

2.2 Residual tracking

Residual is tracked for both ADE and PPE for every iteration for each time step and total number of iterations vs residual is shown in results. The first set of iterations starts from 0 initial guess, hence residuals start from highest error and takes longest to converge, but successive iterations take in old value as initial guess and as cylinder approach steady state , this initial residuals are low and convergence speeds up. For Gauss Seidel iterative scheme , we can write

$$[A]\phi = b\,. \tag{2.9}$$

After every iteration, we will get ϕ'. The residual for every iteration is taken as

$$\epsilon_{res} = |b - [A]\phi'| \tag{2.10}$$

2.3 Lift and Drag calculation

Using pressure value at each time step, total drag force is pressure integration in x direction and lift is pressure integrated in y direction. The pressure values are taken at each grid just before the boundary. Therefore for drag, pressure at each location is multiplied by $(1 - iBlank(i+1, j))$ and $(1 - iBlank(i-1, j))$. Since pressure inside cylinder is 0, this will only keep pressure values at left and right boundary of cylinder respectively, stored in drag force left and right. Since direction of normal is opposite on both sides, these values are multiplied by y grid segment at each location and subtracted to get total drag force. Similarly lift is also calculated. These are divided by $\frac{1}{2}\rho U_{in}^2 D_{cylinder} = 0.5$ to get drag and lift coefficient for cylinder. The mean value of lift and drag is calculated near steady region to compare with published results.

2.4 Shedding frequency calculation

A probe was placed in the simulation at x=4.5D and y=3D for all the simulations to get velocity and pressure time history. Then FFT was performed in the steady region to get dominant frequency which will give shedding frequency and also Strouhal number for cylinder since U and D is 1 for cylinder.

3 Channel Flow

3.1 No shear stress boundary

First we test a case without internal body, so we start with flow between two walls and we take no sheer boundary condition on top and bottom surface. As we see in Fig 2, the u velocity is uniform everywhere. One thing to point out is that since we are enforcing global mass conservation , even if we look the flow at intermediate step when inlet velocity should not have reached the outlet, the flow is 1 everywhere. So we cant have flow with finite velocity at inlet and no velocity at outlet. If I comment out global mass conservation part, then pressure poisson wont converge. The residual gets stuck at some value.

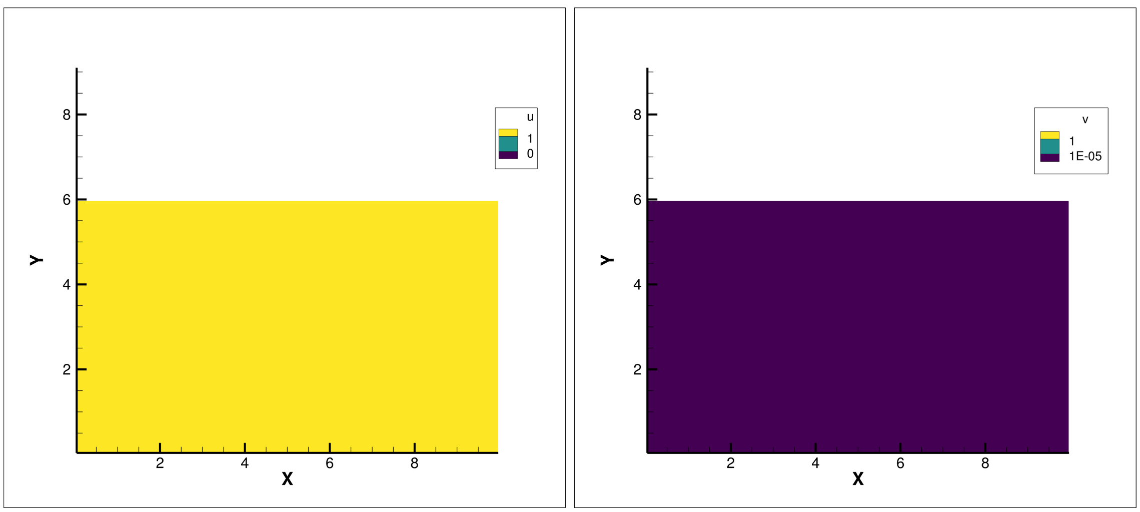


Figure 2: u (left) and v(right) contours for flow with no cylinder at Re=150, no shear stress boundary

3.2 No slip boundary

Next, we test same flow but with no slip boundary condition. Domain size for pipe is 10 x 1 and Re=10. Uniform mesh (N_x, N_y)=(200,80) was used. Fig 3 countors are taken at t=20 sec. Note parabolic profile developing similar to pipe flow, the maximum value of u at centerline is 1.4935 which is very close to theoretical value of $1.5u_{in}$=1.5 .

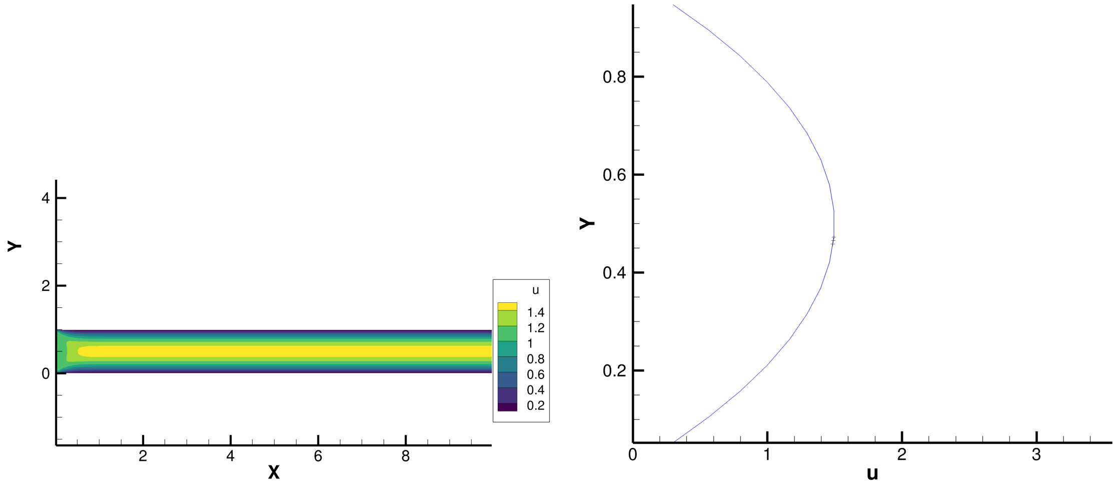


Figure 3: u contour(left) and u velocity profile at x=2(right) for pipe /duct flow at Re=10 , no slip boundary

4 Flow past cylinder at Re=150

4.1 Velocity, Pressure and Vorticity Contours

As with any CFD analysis, we first start with grid convergence test to make sure we are using fine enough mesh and using finer mesh wont change results significantly. This is shown in Table 1. Domain length for all study unless other mentioned is L_x=12, L_y=6. dt= 0.001 for all simulations. The drag coefficient is computed from taking mean of drag coefficient when flow reached steady state and shedding frequency is computed from v velocity time history probed at location 1.5D behind cylinder on centerline. We see results of 360 x 120 mesh and Non uniform mesh of 240 x 96 very close to each other. Comparison with published results is shown in table 2. Uniform mesh of 360 x 120 is shown in Fig 4 and is used in simulations for Re=150 case.

Fig 5 shows residual tracking throughout the simulation. The first iteration for PPE has very large residual and takes longest to converge, but since next values takes guess from earlier values, the convergence of successive steps increases rapidly. ADE converges in few iteration for every timestep, hence it has negligible convergence history compared to PPE.

Next we have velocity, pressure, vorticity and countours at t=6,12,30 and 70 sec shown in figures 6 to 9.

4.2 Drag,Lift and Shedding Frequency

Drag and lift curve is shown in is shown in Fig 12. The mean drag coefficient in steady region is 1.391 and mean lift coefficient is -0.05.

Table 1: Effect of mesh size on shedding frequency, f and Drag coefficient, c_D

mesh size	c_D	f
64 x 64 (Uniform Mesh)	0.97	0.239
128 x 128 (Uniform Mesh)	1.37	0.199
160 x 120 (Uniform Mesh)	1.478	0.214
240 x 120 (Uniform Mesh)	1.46	0.199
360 x 120 (Uniform Mesh)	1.39	0.215
240 x 96 (Non uniform Mesh)	1.38	0.2

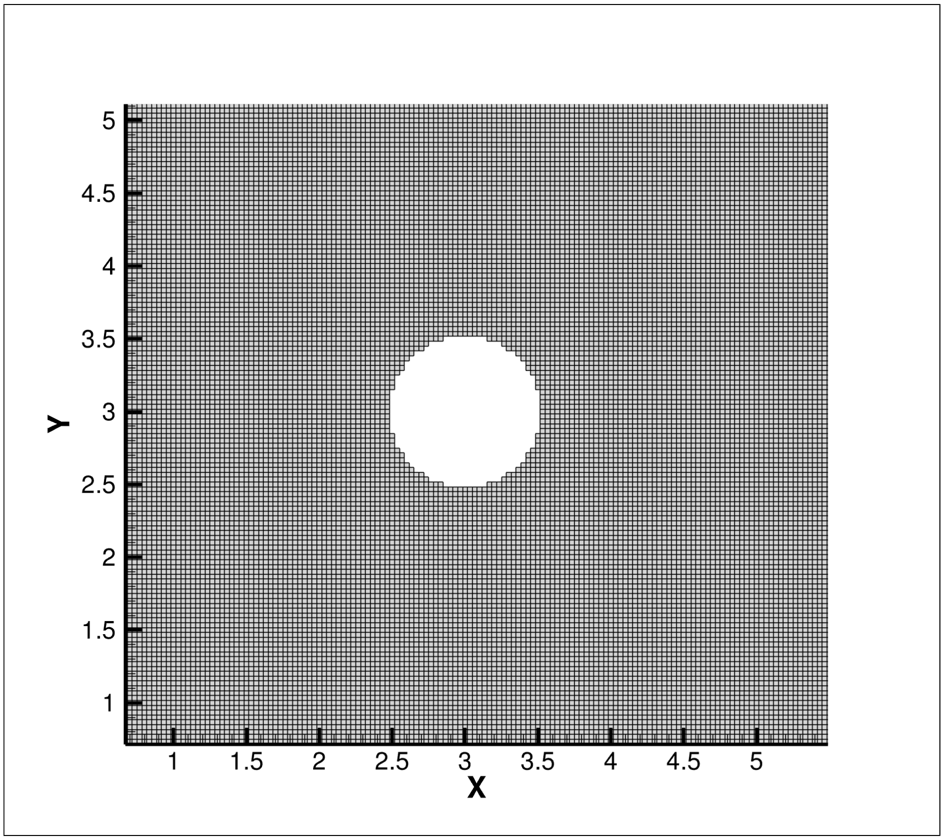


Figure 4: Uniform Mesh for cylinder at Re=150

Table 2: Comparison of shedding frequency, f and Drag coefficient, c_D with published results for Re=150

-	$Numerical$	Ref[2]
c_{pD}	1.391	1.037
f	0.215	0.187

Surface pressure (measured from right end of cylinder in counter clockwise direction is shown in Fig 13.

Velocity and pressure time history at probe location 1.5D behind cylinder on centerline is shown in Fig 14. The FFT of v velocity probe data gives dominant frequency gives shedding frequency of 0.2153. Since U=L=1, this is also equal to St number.

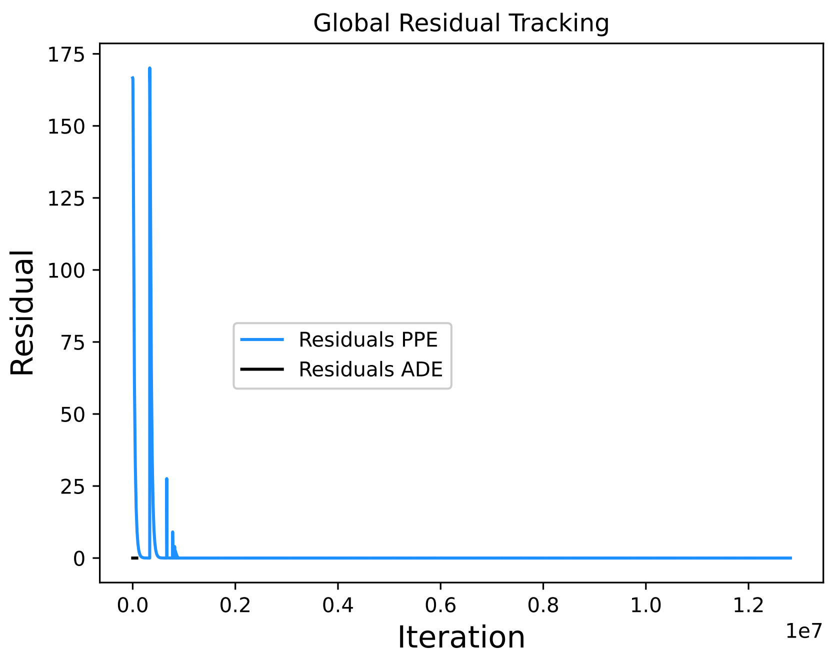


Figure 5: Residual history cylinder at Re=150

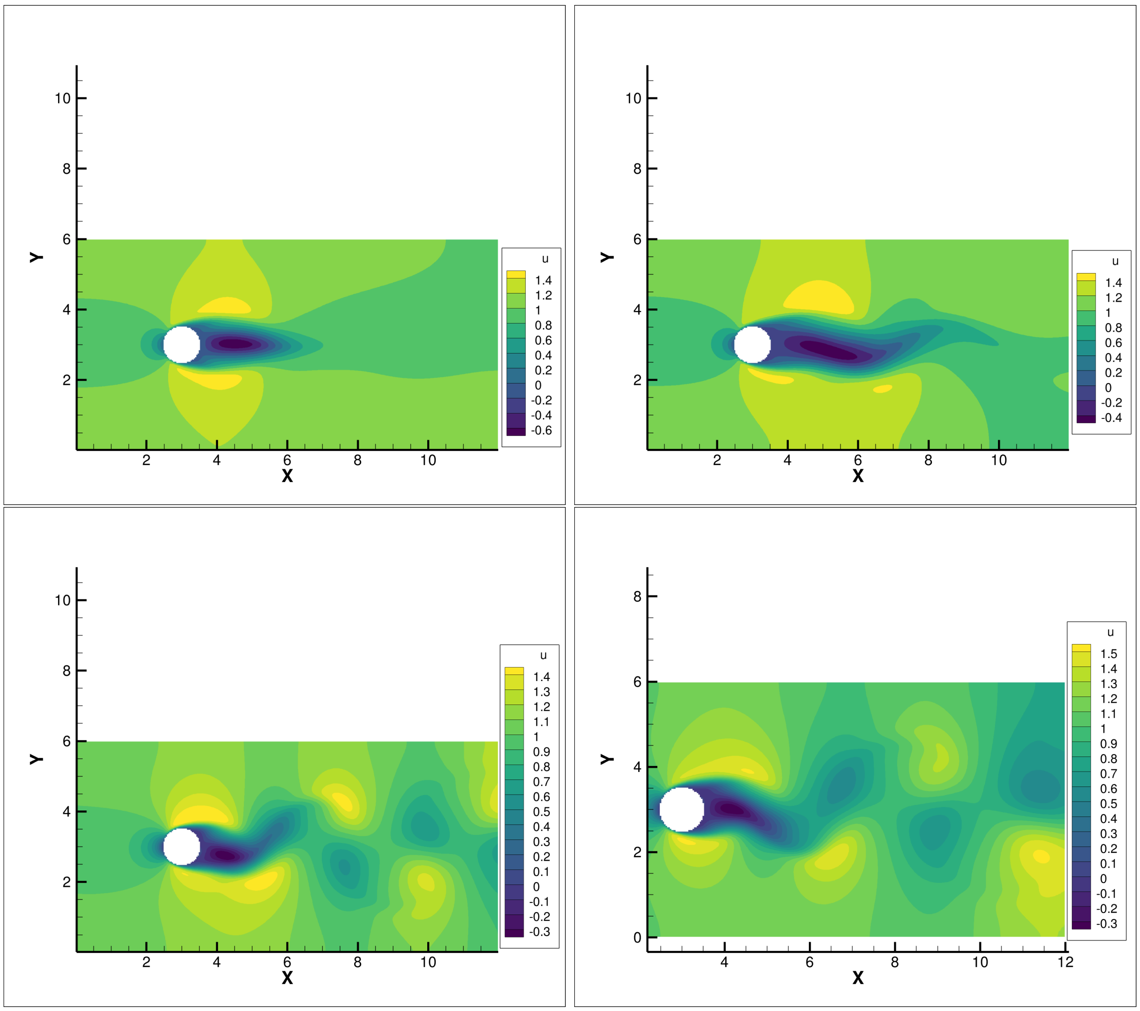


Figure 6: u contours for cylinder at Re=150 at t= 6,12,30 and 70 sec

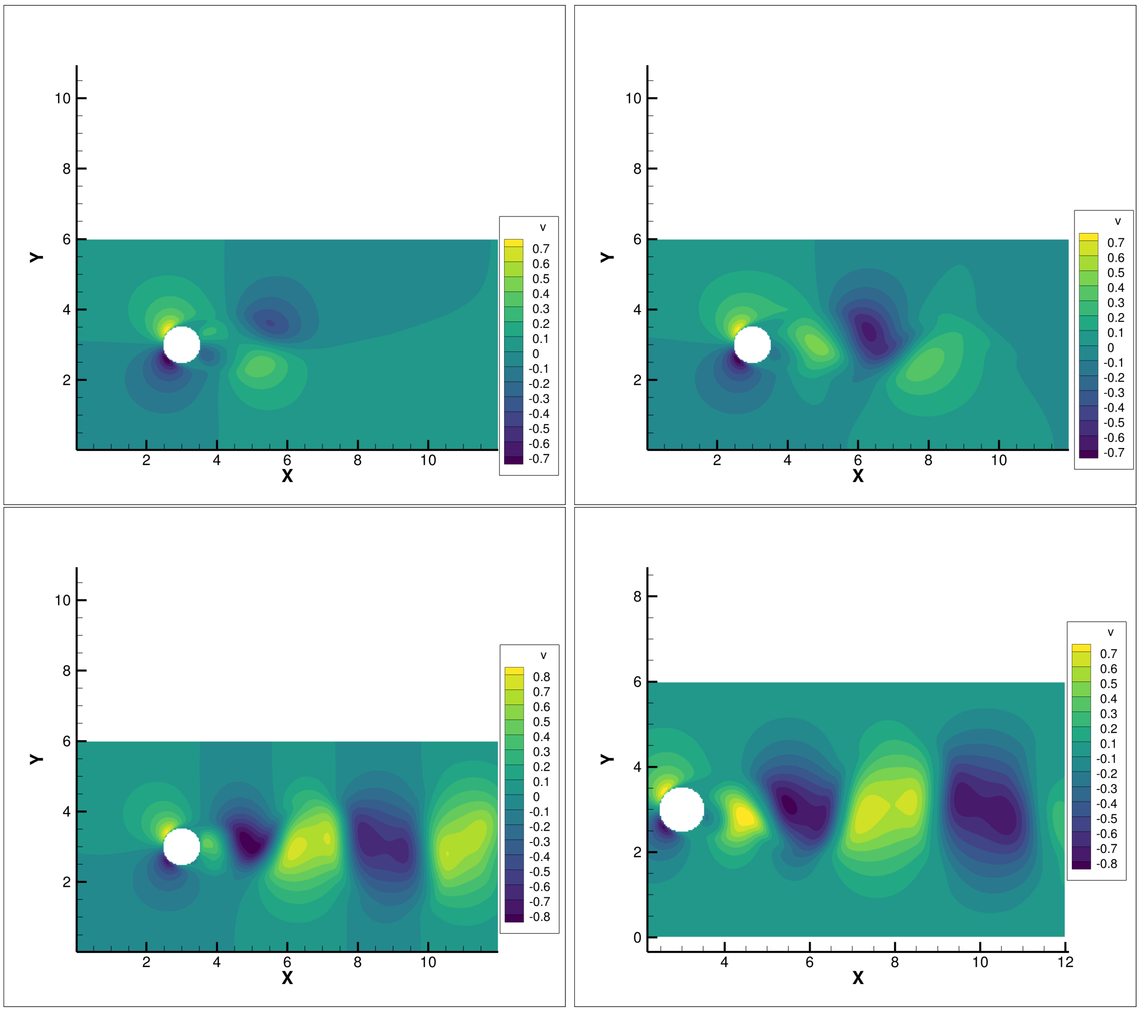


Figure 7: v contours for cylinder at Re=150 at t= 6,12,30 and 70 sec

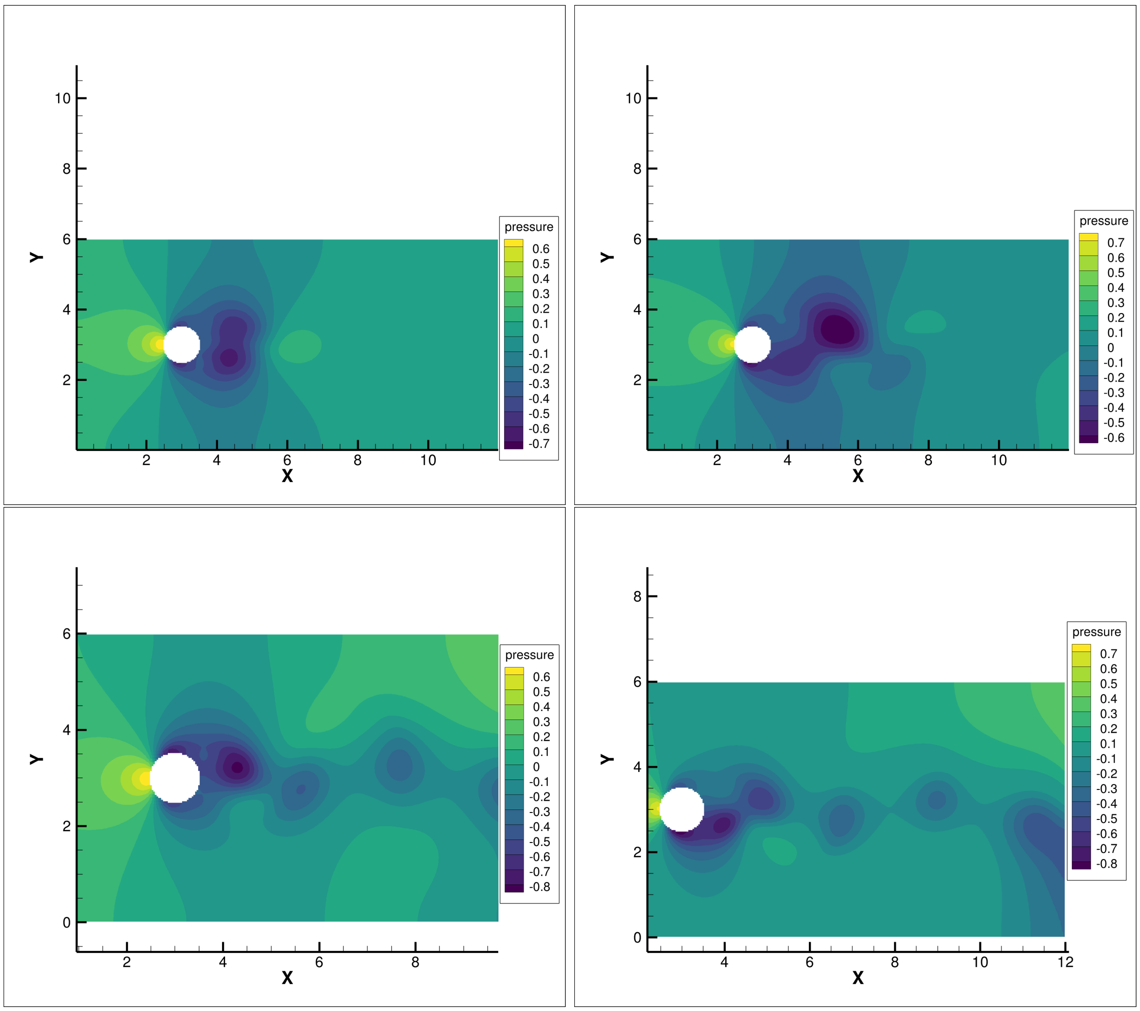


Figure 8: Pressure contours for cylinder at Re=150 at t= 6,12,30 and 70 sec

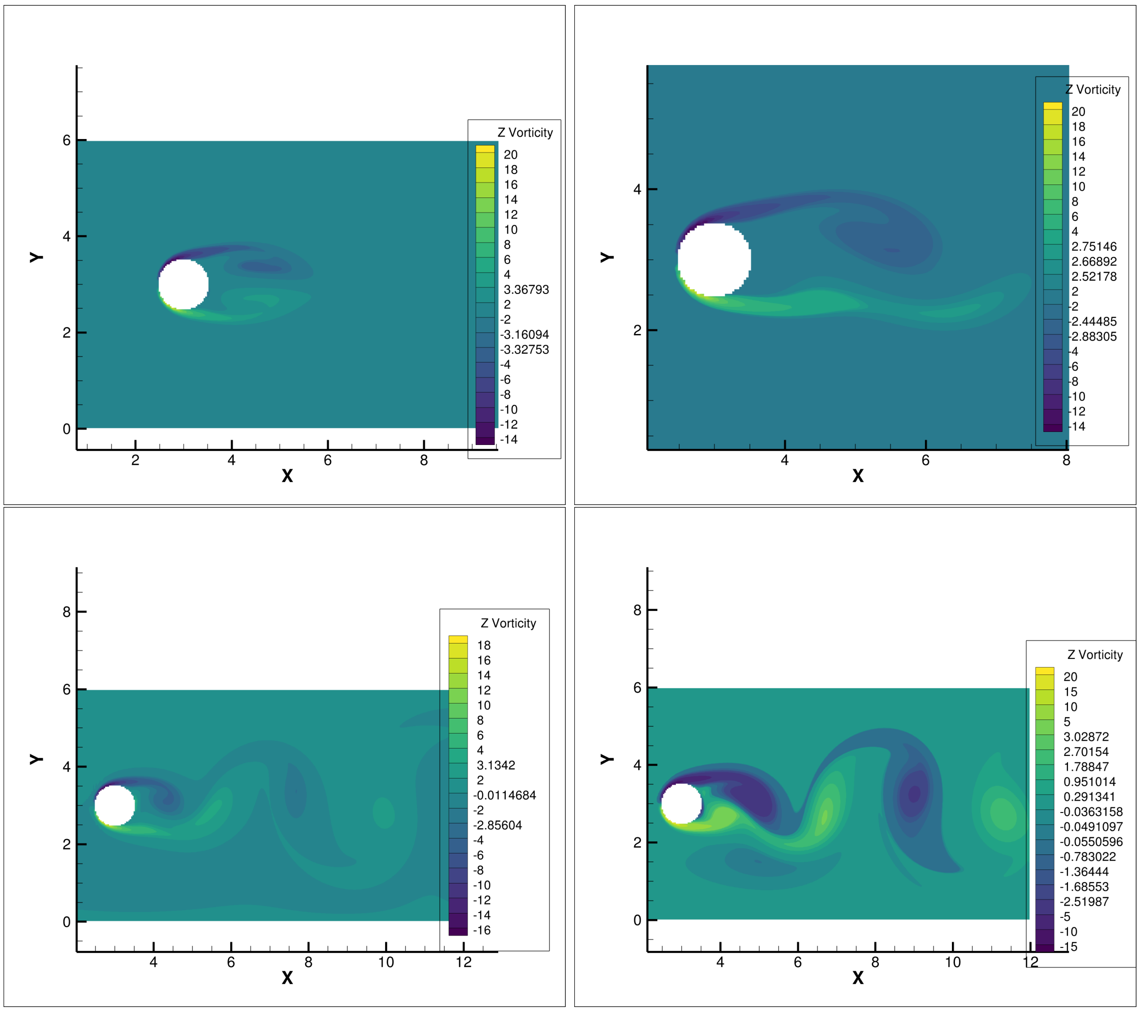


Figure 9: Vorticity contours for cylinder at Re=150 at t= 6,12,30 and 70 sec

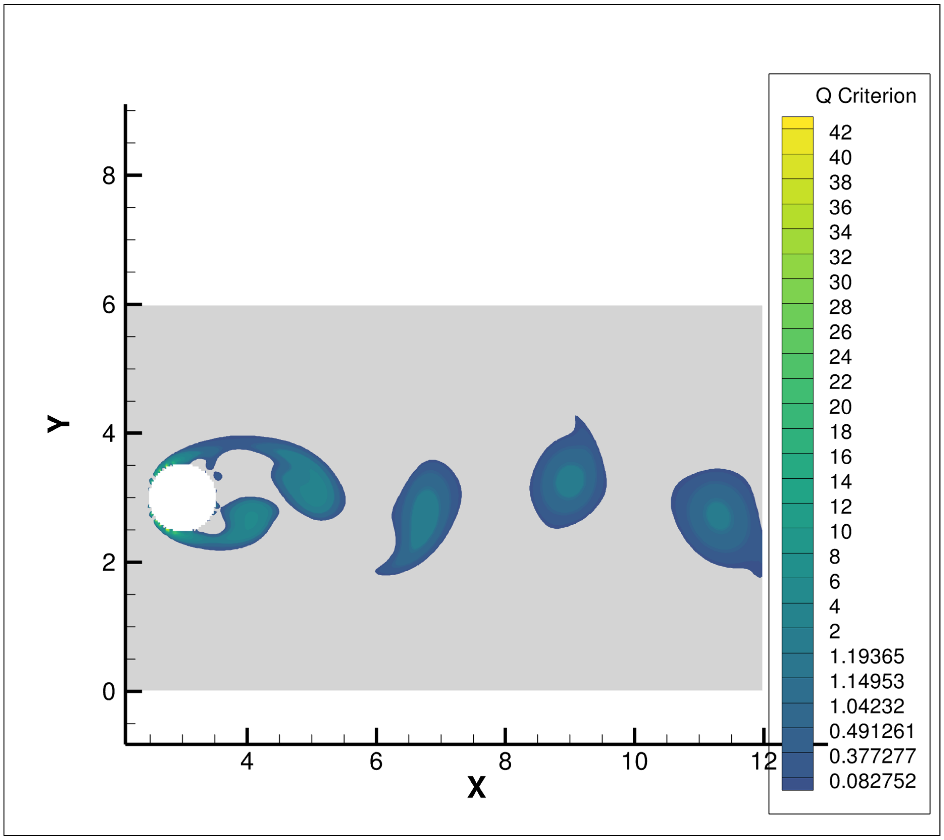


Figure 10: Q criterion cylinder at Re=150 at t=70 sec

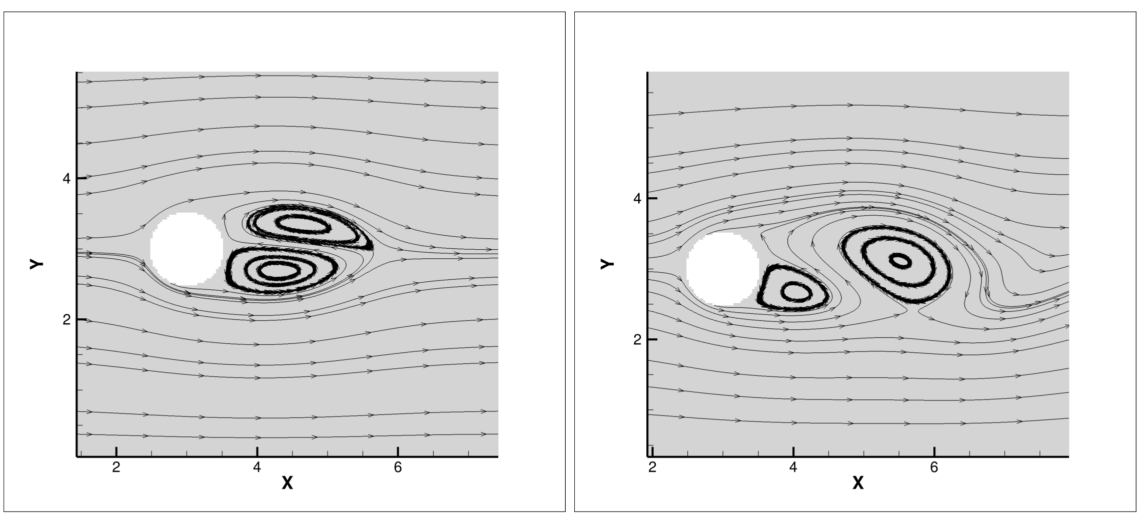


Figure 11: streamlines for cylinder at Re=150 at t=6 and 12 sec

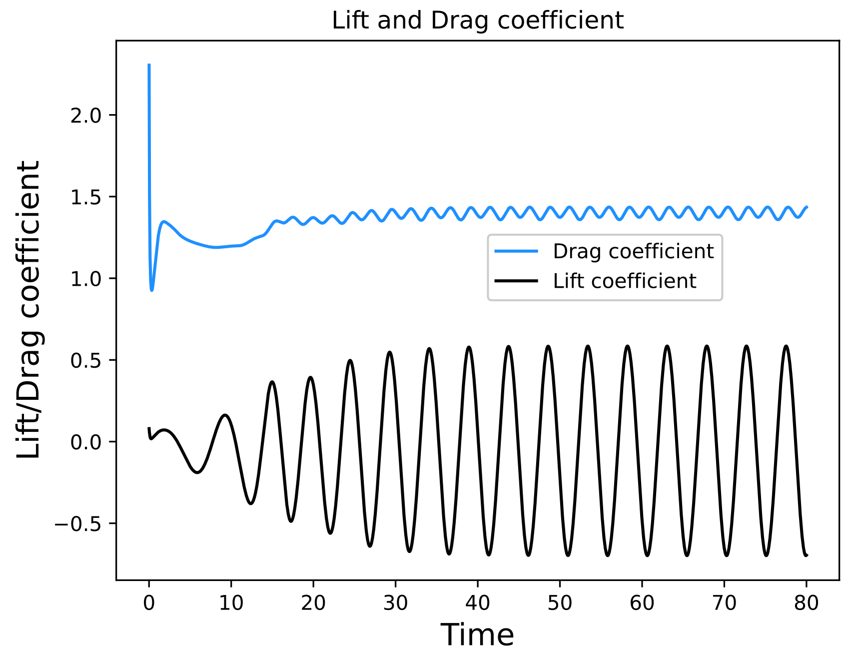


Figure 12: Drag and Lift coefficient for cylinder at Re=150

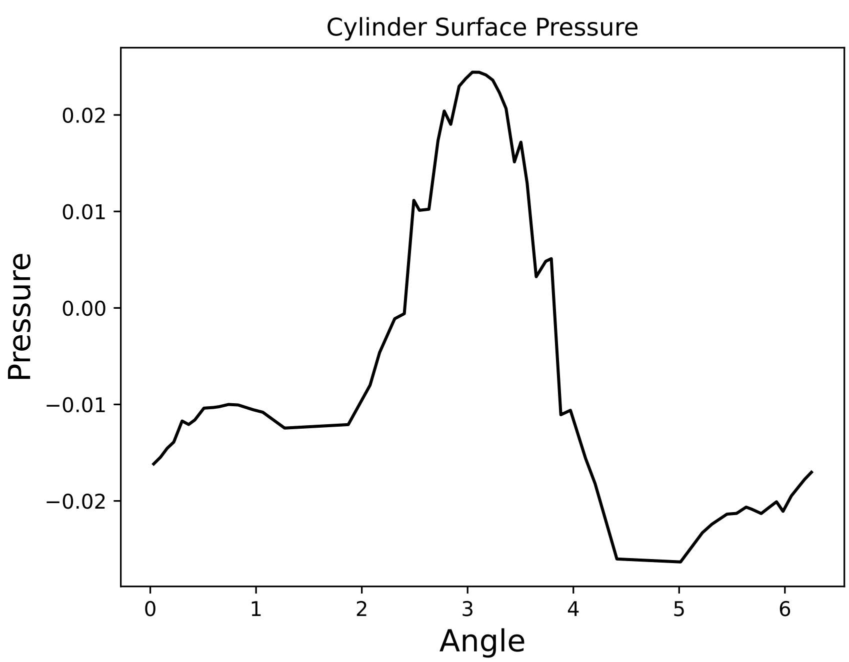


Figure 13: Surface pressure for cylinder at Re=150

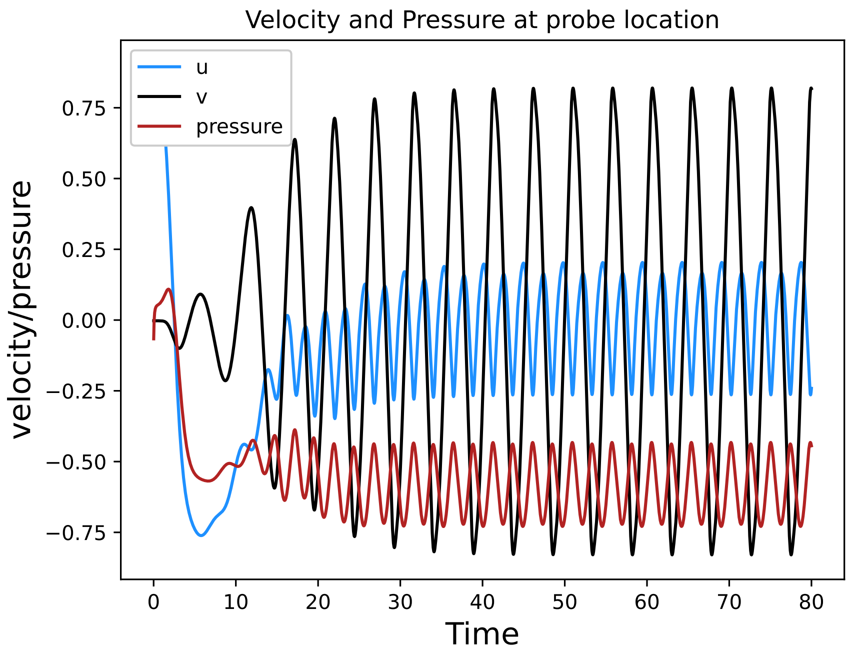


Figure 14: velocity and pressure time history at probe location (x,y)=(4.5,3)

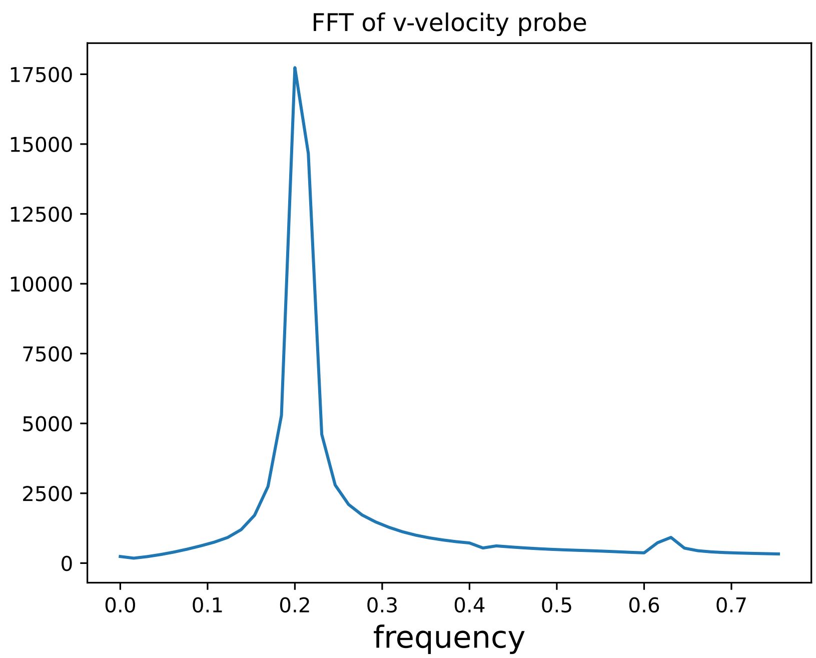
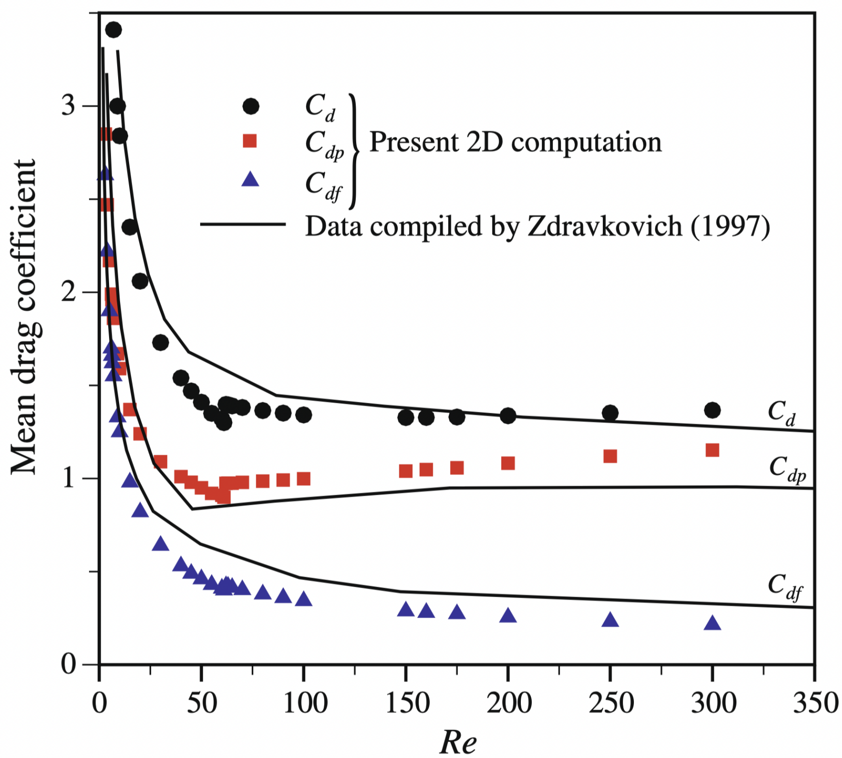


Figure 15: FFT of v time history in steady state region



(a) Mean Drag

Figure 16: Pressure drag data for cylinder by Ref[2] used for comparision

5 Flow past cylinder at Re=300

5.1 Velocity, Pressure and Vorticity Contours

The non uniform mesh shown in Fig 18 is used for both Re=300 and 1000 case simulation. This mesh is generated using Dr. Seo cartesian non uniform mesh generator code. Grid size is 240 x 96. We know boundary layer scales roughly as $\frac{1}{\sqrt{Re}}$, so Re=300 and 1000 will have boundary layer thickness of roughly 0.05 and 0.03 respectively. Grid spacing near cylinder is 0.025, smaller than BL thickness.

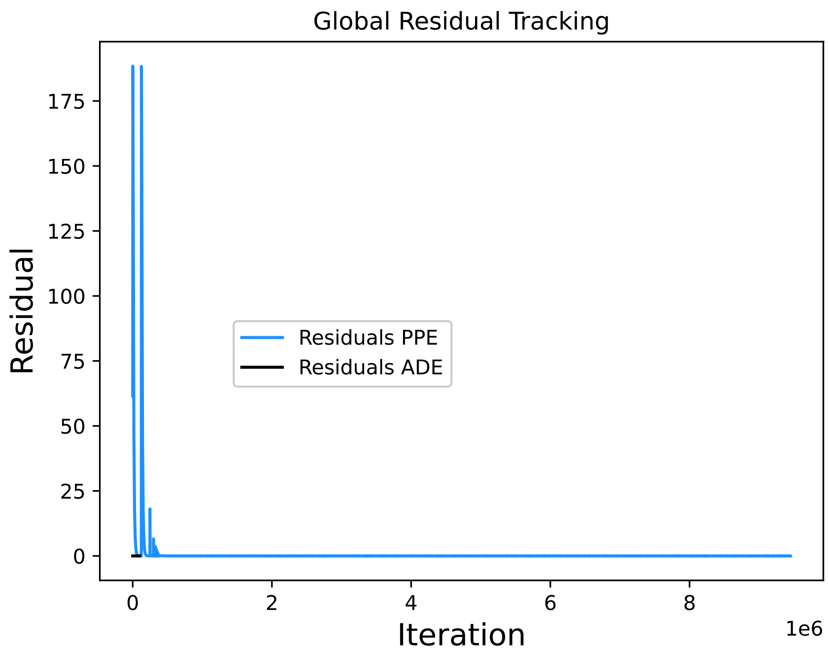


Figure 17: Residual history cylinder at Re=300

The velocity, pressure, vorticity and contours at t=6,12,30 and 70 sec shown in figures 19 to 22.

5.2 Drag,Lift and Shedding Frequency

Drag and lift curve is shown in is shown in Fig 24. The mean drag coefficient in steady region is 1.544 and mean lift coefficient is -0.11.

Surface pressure (and measured from right end of cylinder in counter clockwise direction is shown in Fig 25.

Velocity and pressure time history at probe location is shown in Fig 26. The FFT of v velocity probe data gives dominant frequency gives shedding frequency of 0.2235.

Table 3: Comparison of shedding frequency, f and Drag coefficient, c_D with published results for Re=300

-	$Numerical$	Ref[2]
c_D	1.544	1.159
f	0.2235	0.21

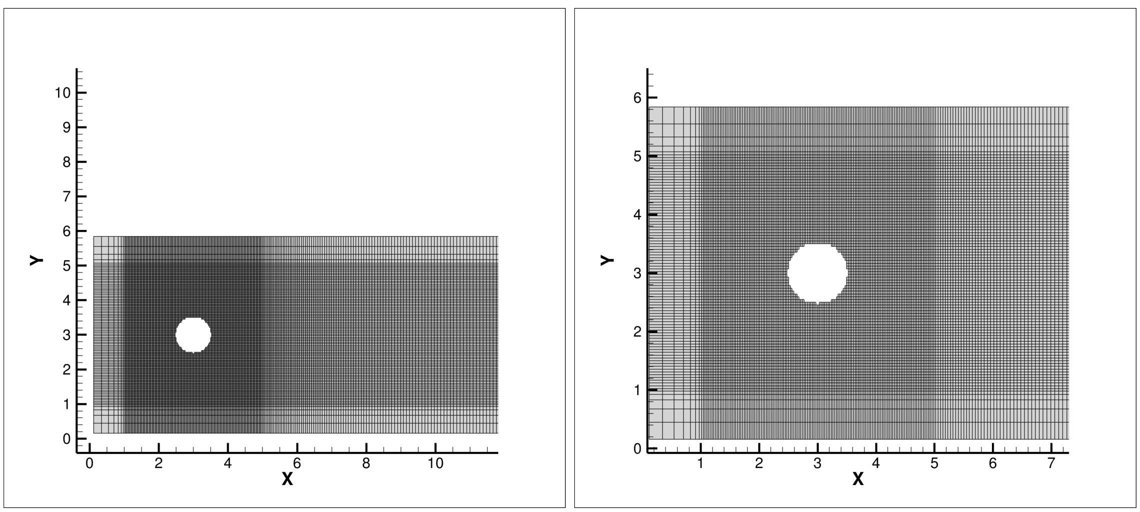


Figure 18: Non uniform mesh

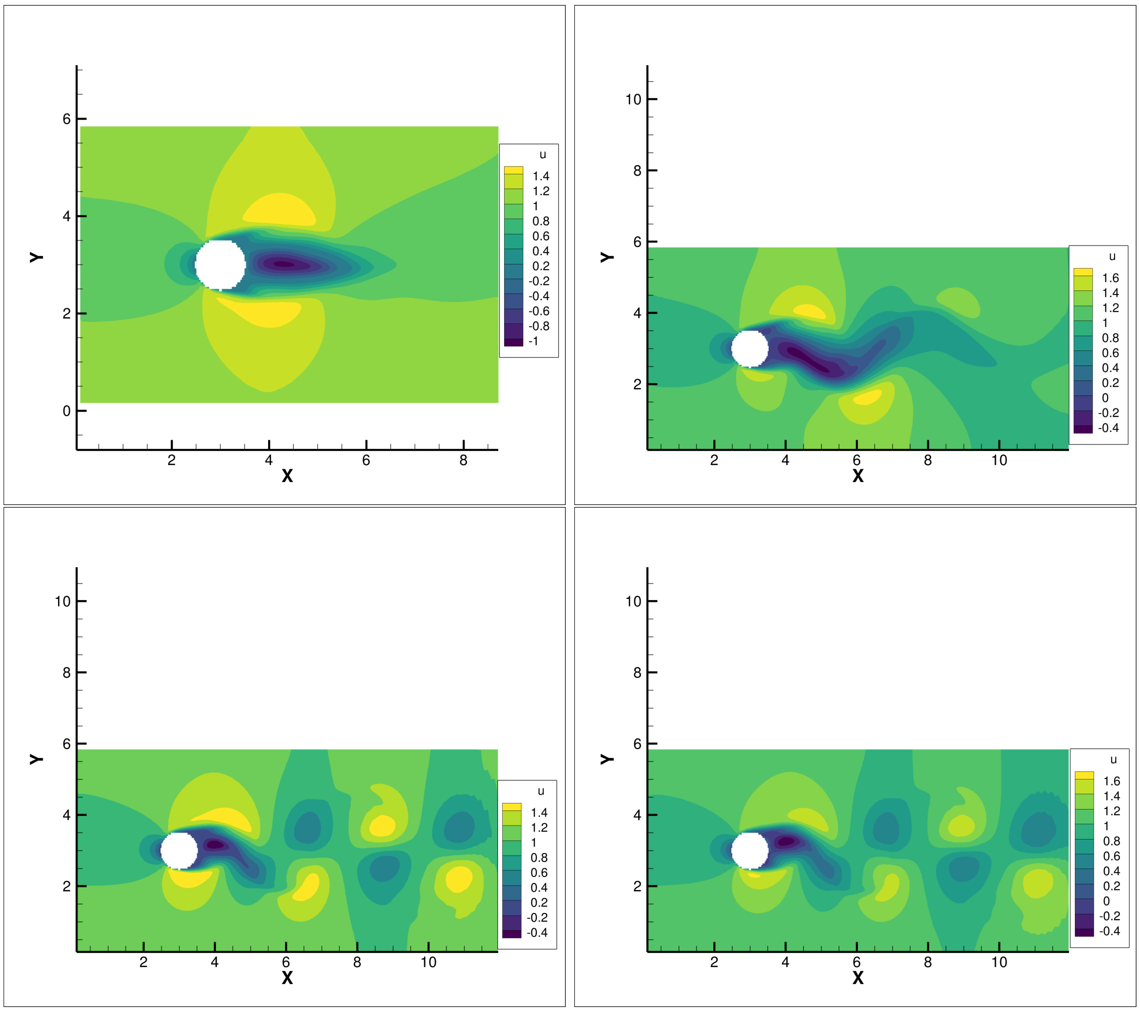


Figure 19: u contours for cylinder at Re=300 at t= 6,12,30 and 70 sec

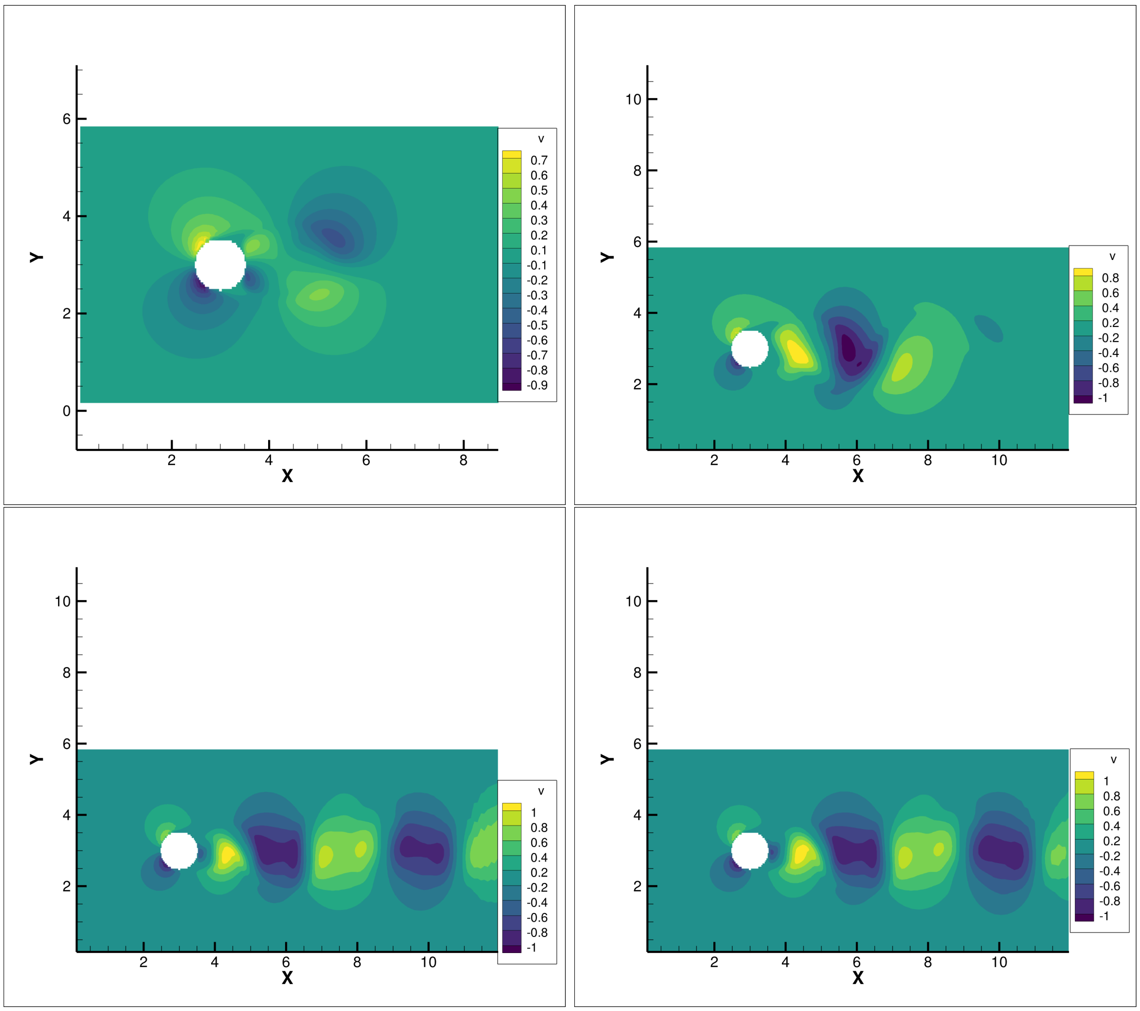


Figure 20: v contours for cylinder at Re=300 at t= 6,12,30 and 70 sec

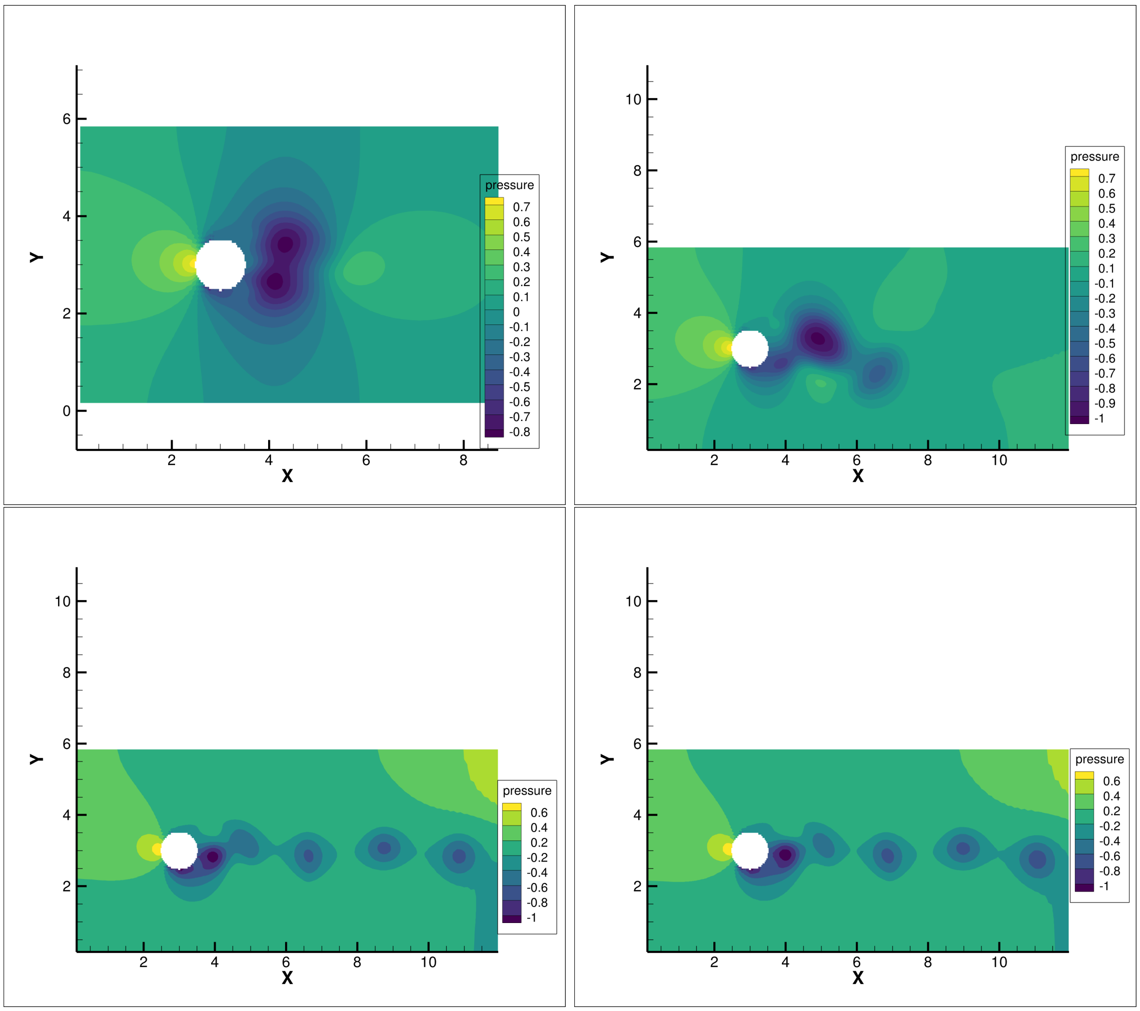


Figure 21: Pressure contours for cylinder at Re=300 at t= 6,12,30 and 70 sec

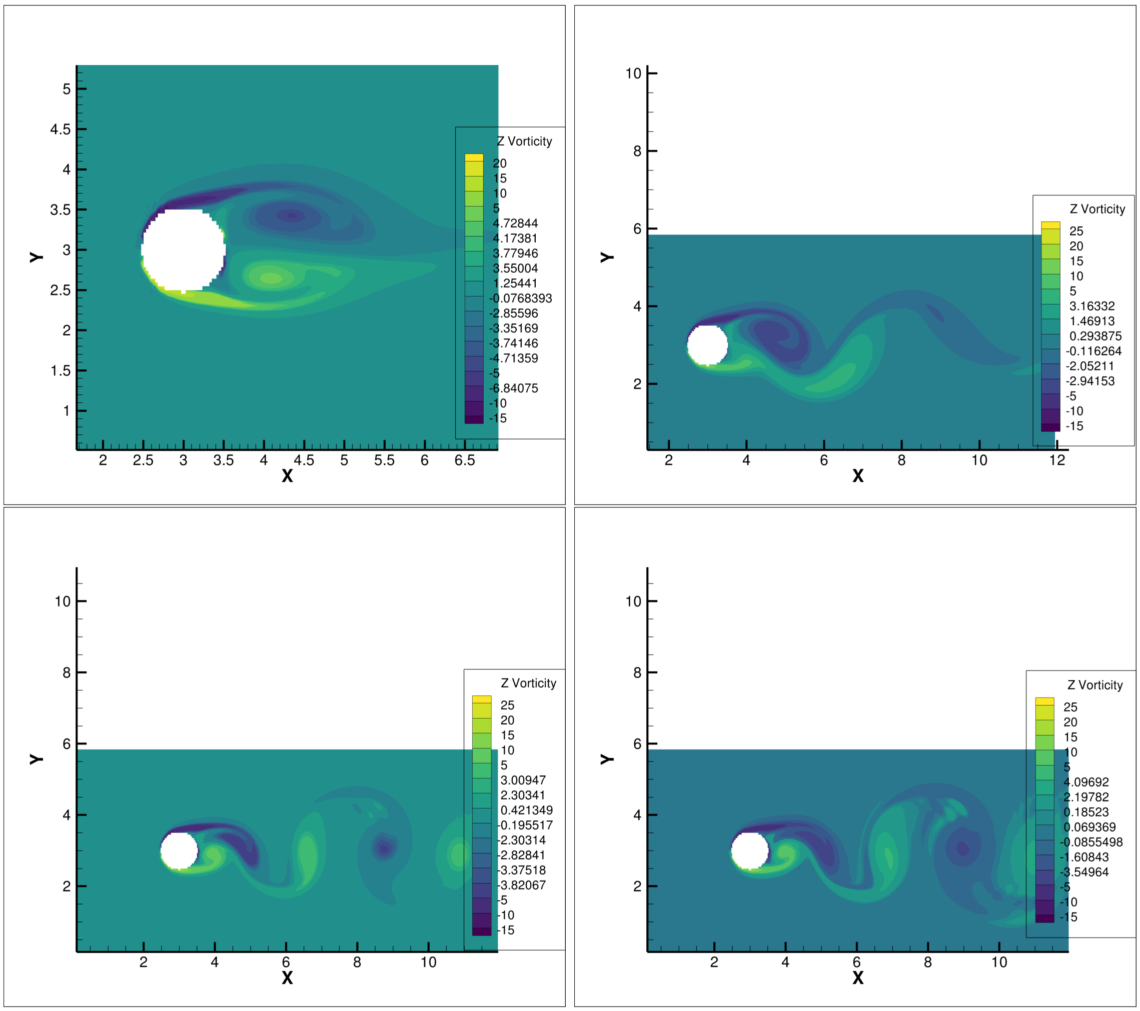


Figure 22: Vorticity contours for cylinder at Re=300 at t= 6,12,30 and 70 sec

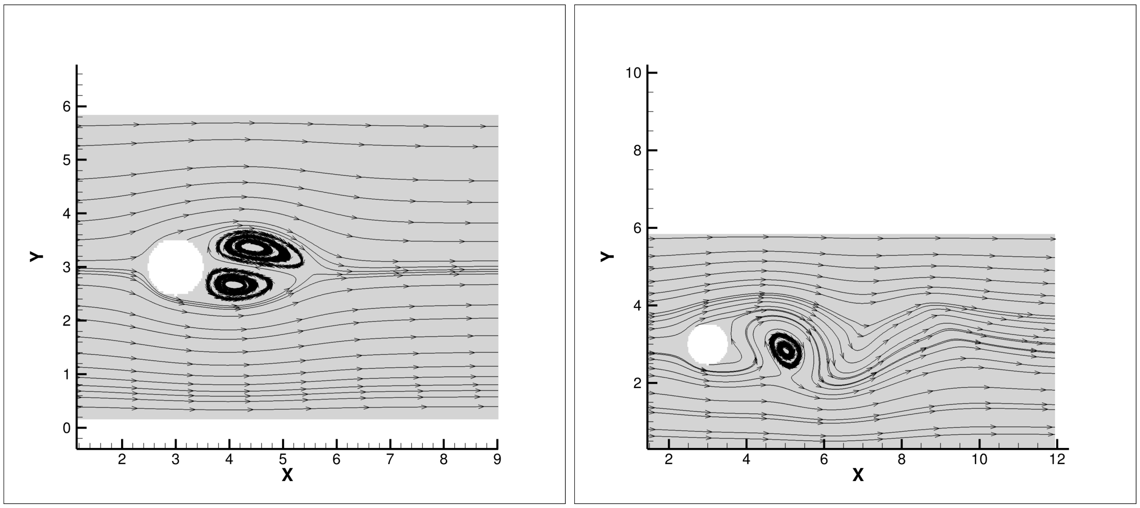


Figure 23: streamlines for cylinder at Re=300 at t=6 and 12 sec

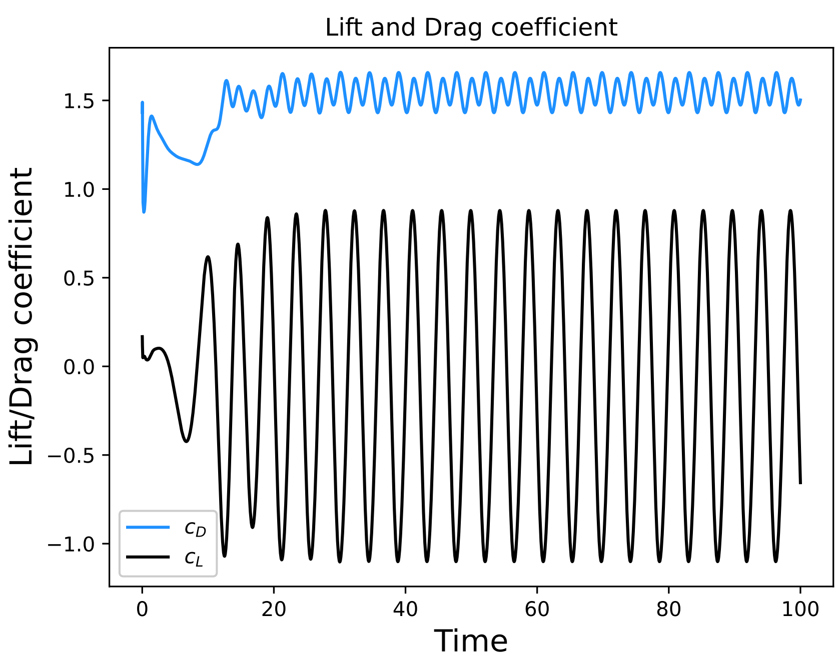


Figure 24: Drag and Lift coefficient for cylinder at Re=300

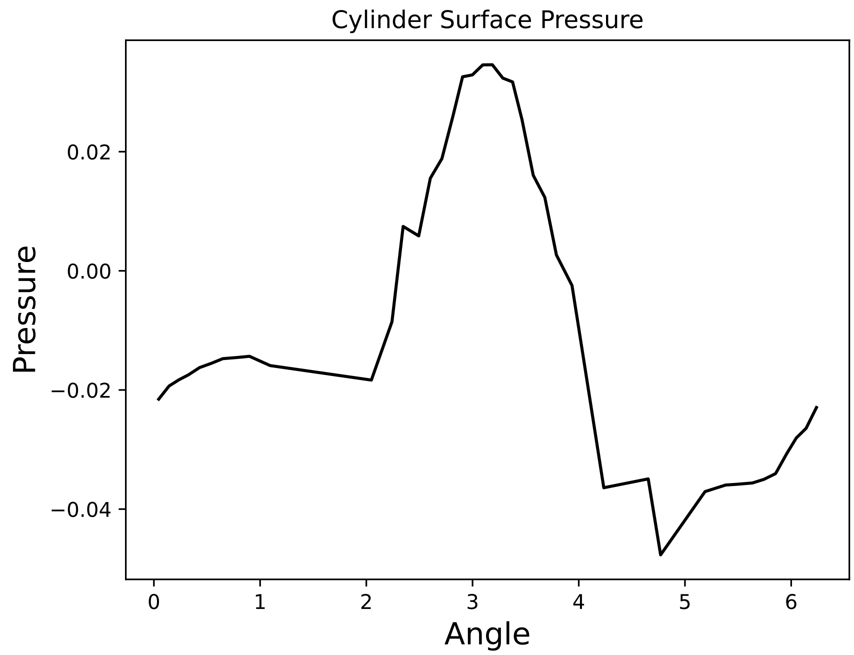


Figure 25: Surface pressure for cylinder at Re=300

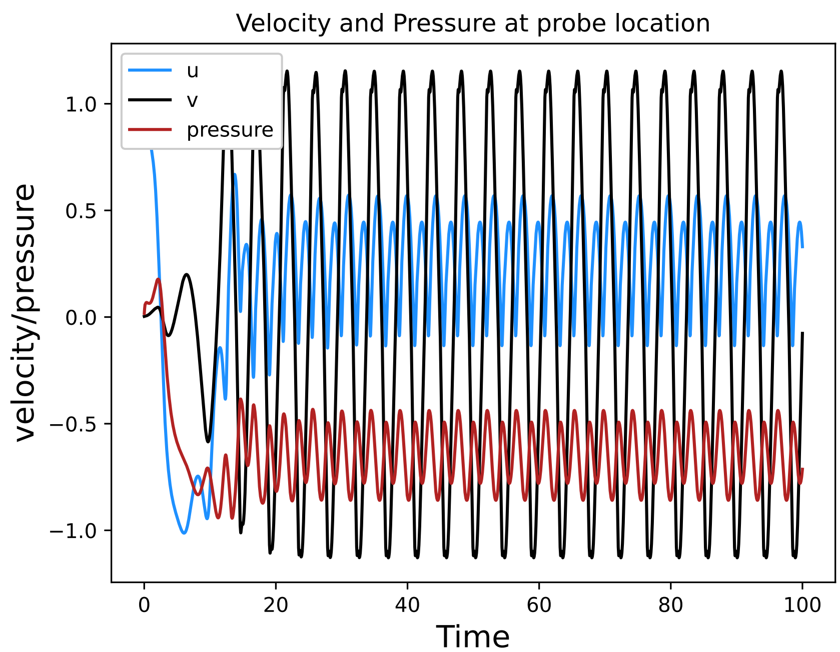


Figure 26: velocity and pressure time history at probe location (x,y)=(4.5,3)

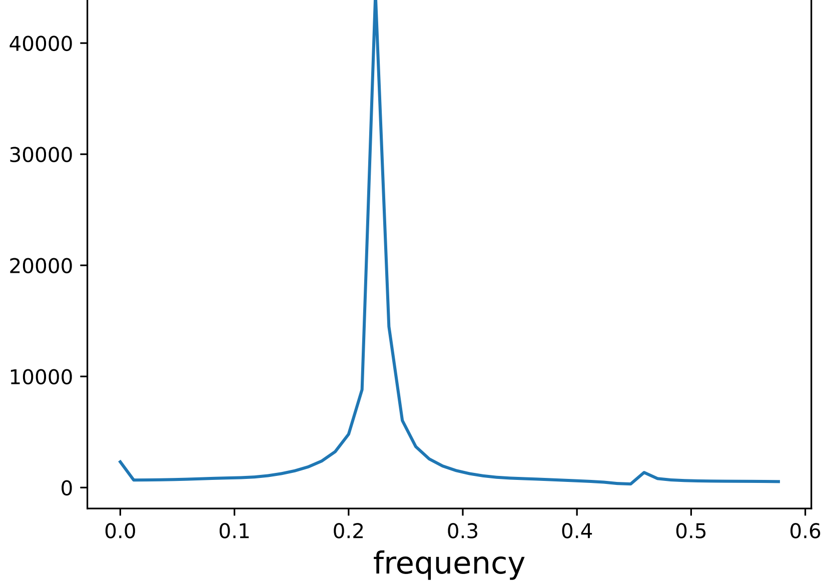


Figure 27: FFT of v time history in steady state region

6 Flow past cylinder at Re=1000

6.1 Velocity, Pressure and Vorticity Contours

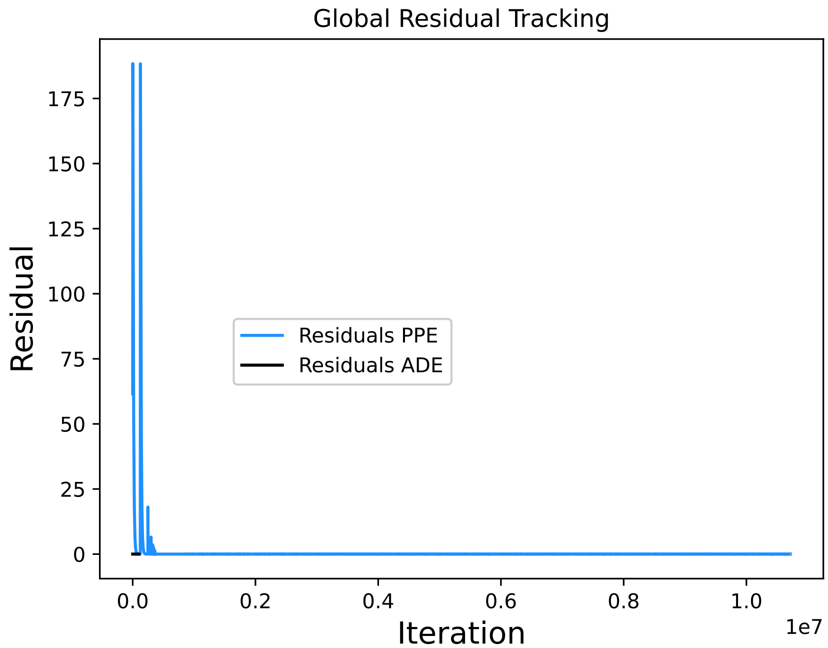


Figure 28: Residual history cylinder at Re=1000

Now we have velocity, pressure, vorticity and Q countours at t=6,12,30 and 70 sec

6.2 Drag,Lift and Shedding Frequency

Drag and lift curve is shown in is shown in Fig 35. The mean drag coefficient in steady region is 1.476 and mean lift coefficient is 0.279.

Surface pressure (and measured from right end of cylinder in counter clockwise direction is shown in Fig 36.

Velocity and pressure time history at probe location is shown in Fig 37. The FFT of v velocity probe data gives dominant frequency gives shedding frequency of 0.258. Since we notice small fluctuations in velocity probe data, there is another very small peak around frequency of 0.5 and 0.8 to reflect that.

Table 4: Comparison of shedding frequency, f and Drag coefficient, c_D with published results for Re=1000

-	$Numerical$	Ref[3]
c_{pD}	1.476	1.34
f	0.279	0.236

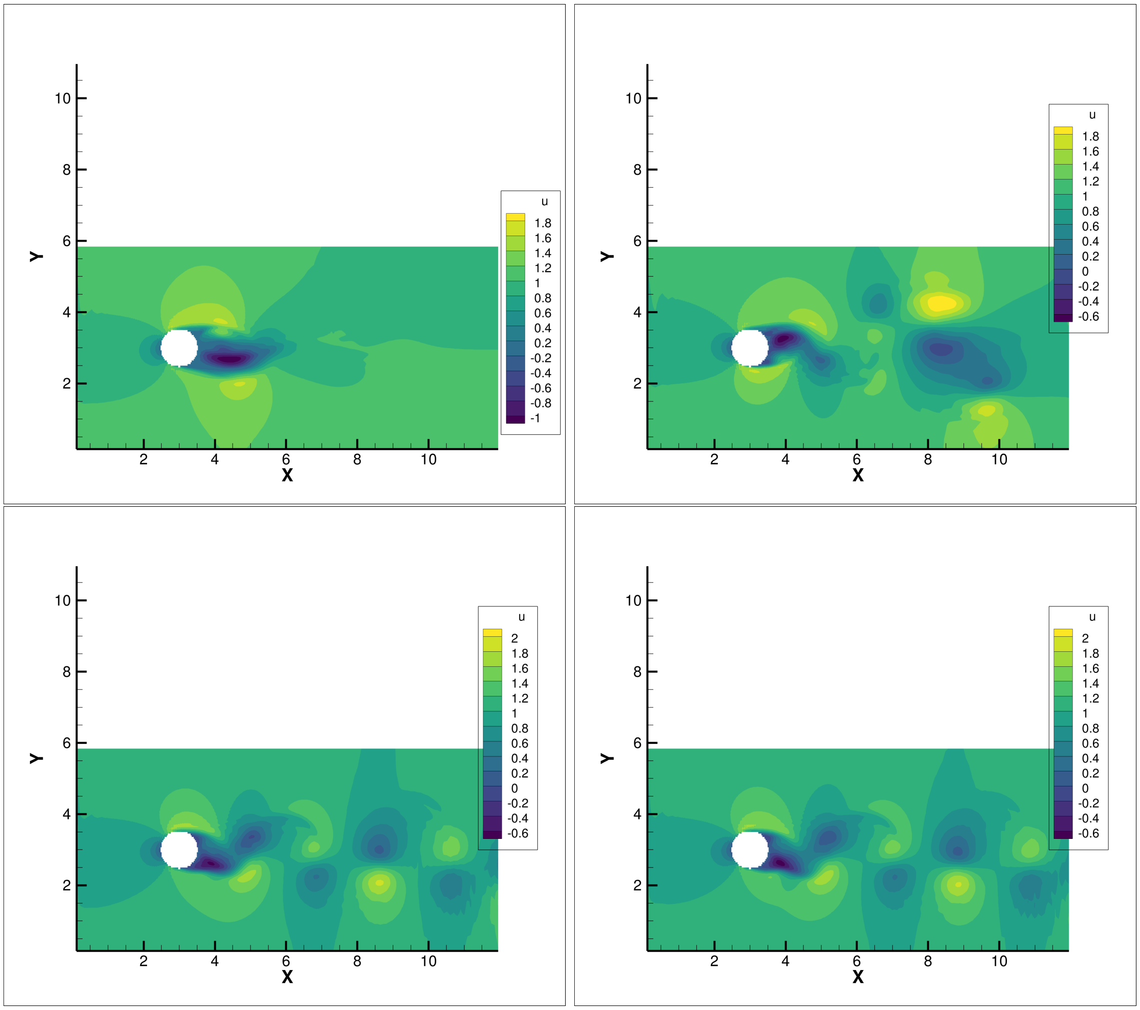


Figure 29: u contours for cylinder at Re=1000 at t= 6,12,30 and 70 sec

6.3 Comparison of Re=150, 300 and 1000 results

As Re is increased, we see more frequencies in velocity probe data, which shows up as small peaks in FFT of v velocity. Instabilities set in quicker in higher Re cases, hence we see shedding earlier as Re is increased. Drag is also higher for Re=300 and 1000 compared to Re=150 case.

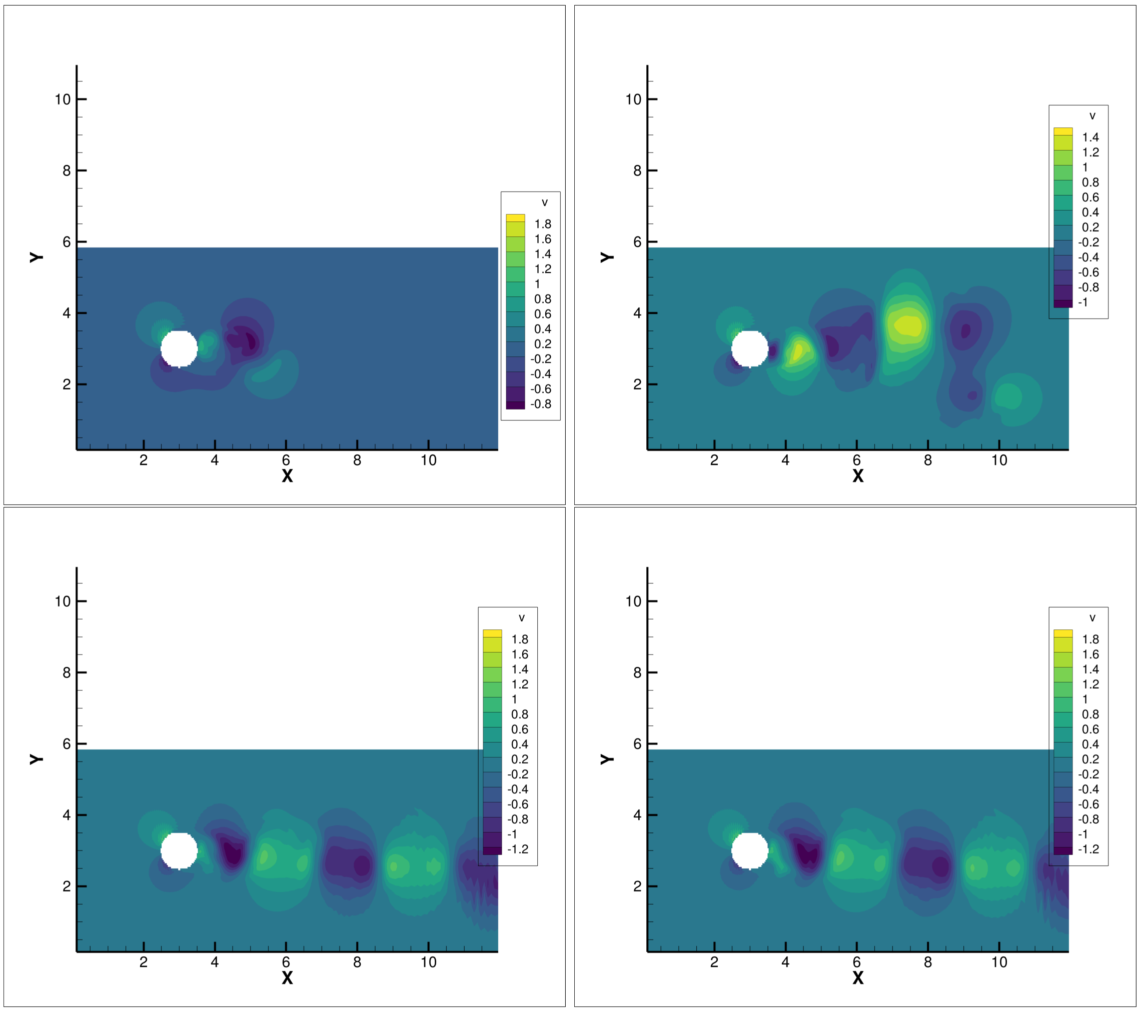


Figure 30: v contours for cylinder at Re=1000 at t= 6,12,30 and 70 sec

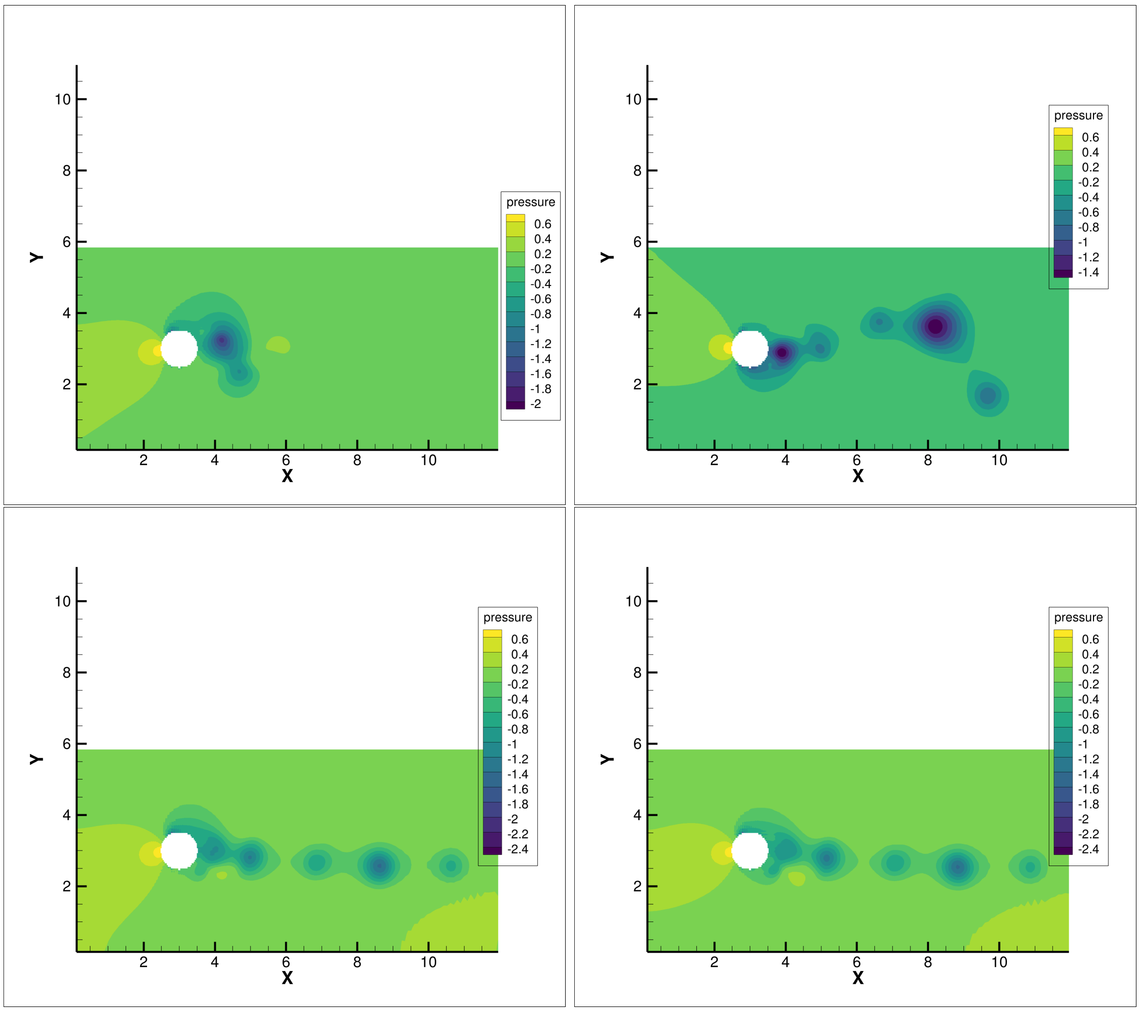


Figure 31: Pressure contours for cylinder at Re=1000 at t= 6,12,30 and 70 sec

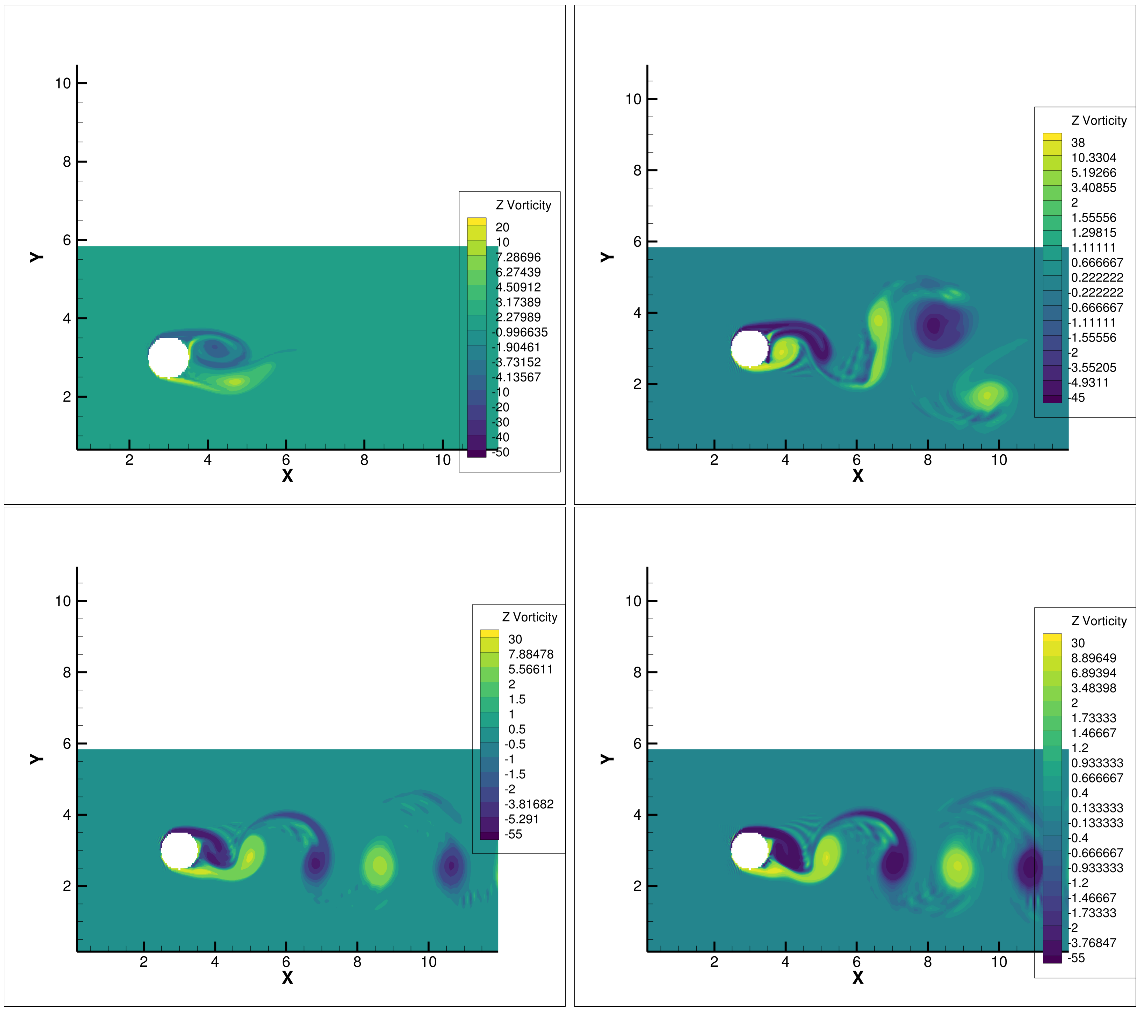


Figure 32: Vorticity contours for cylinder at Re=1000 at t= 6,12,30 and 70 sec

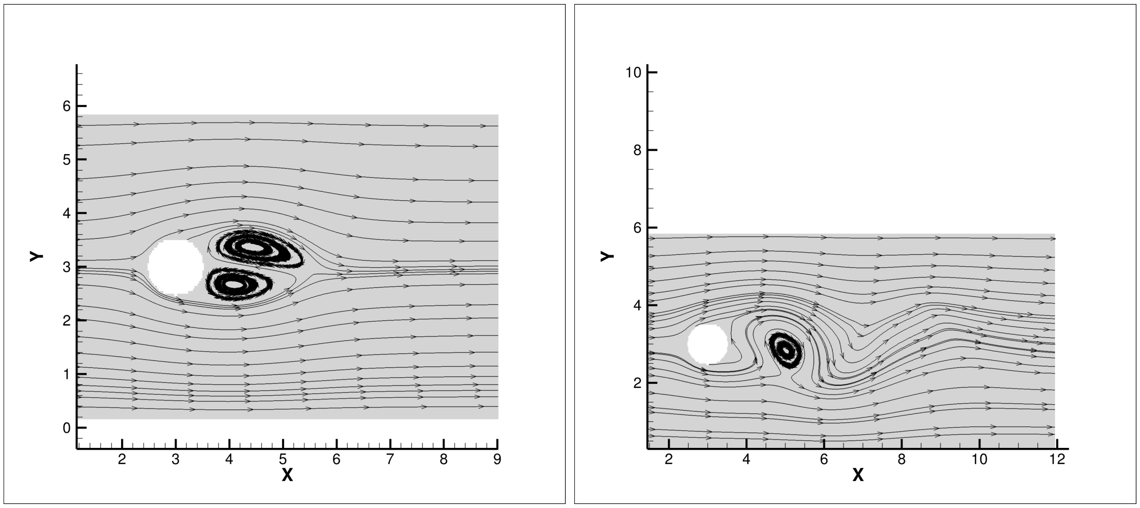


Figure 33: streamlines for cylinder at Re=1000 at t=6 and 12 sec

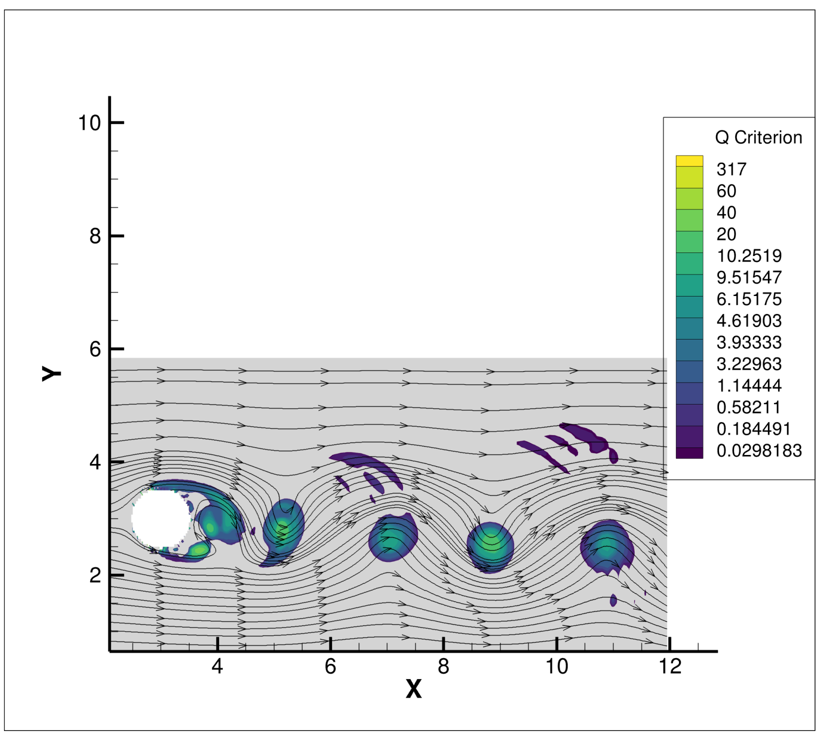


Figure 34: Q criterion and streamlines for cylinder at Re=1000 at t=70 sec

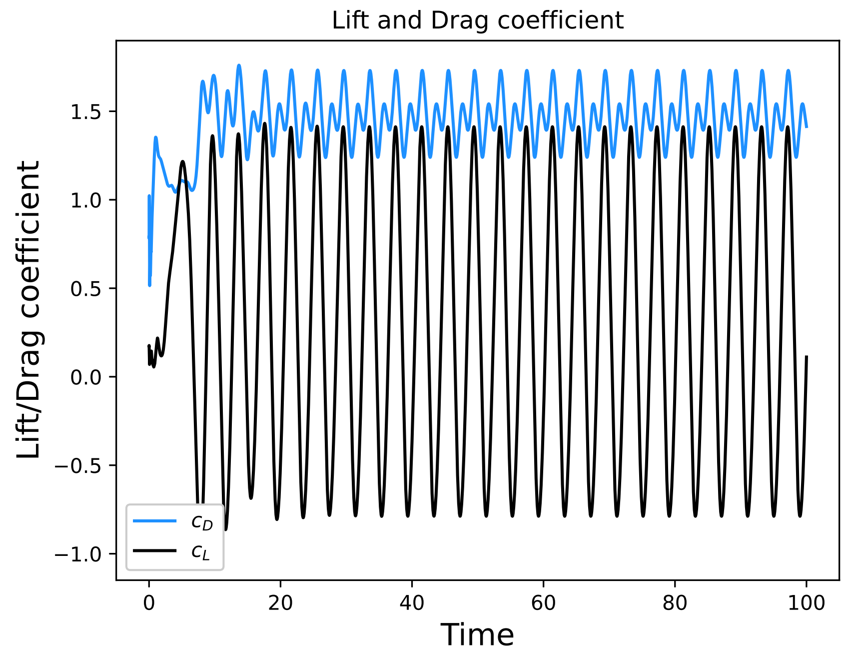


Figure 35: Drag and Lift coefficient for cylinder at Re=1000

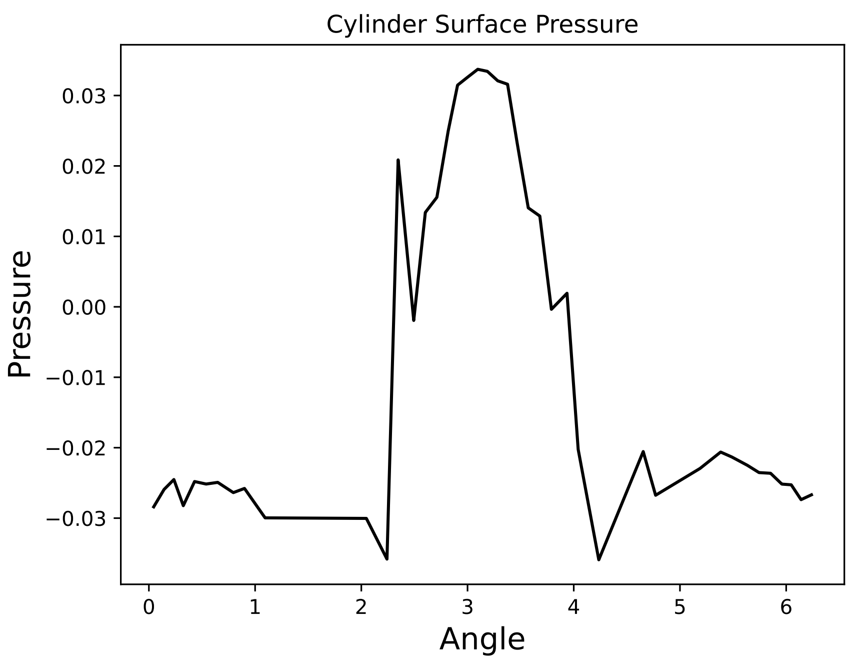


Figure 36: Surface pressure for cylinder at Re=1000

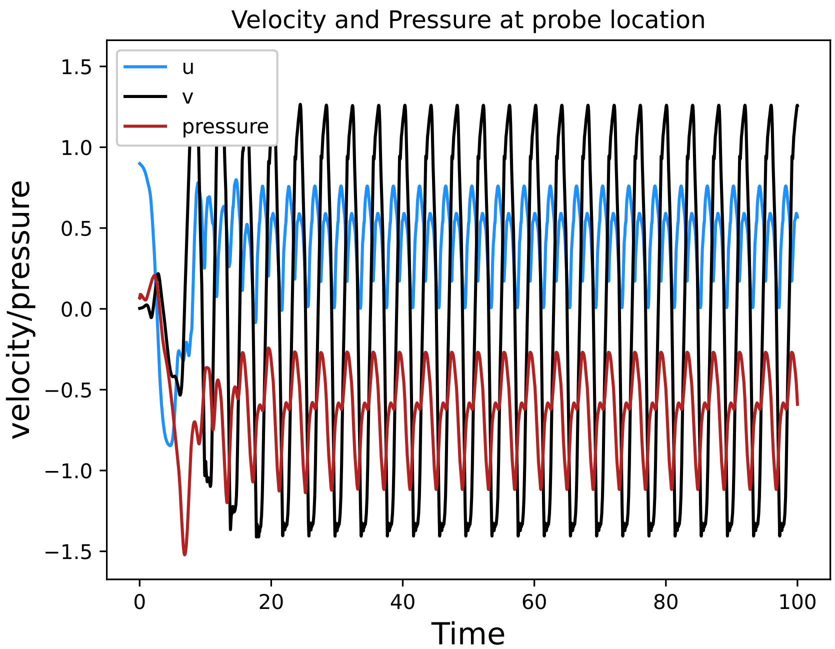


Figure 37: velocity and pressure time history at probe location (x,y)=(4.5,3)

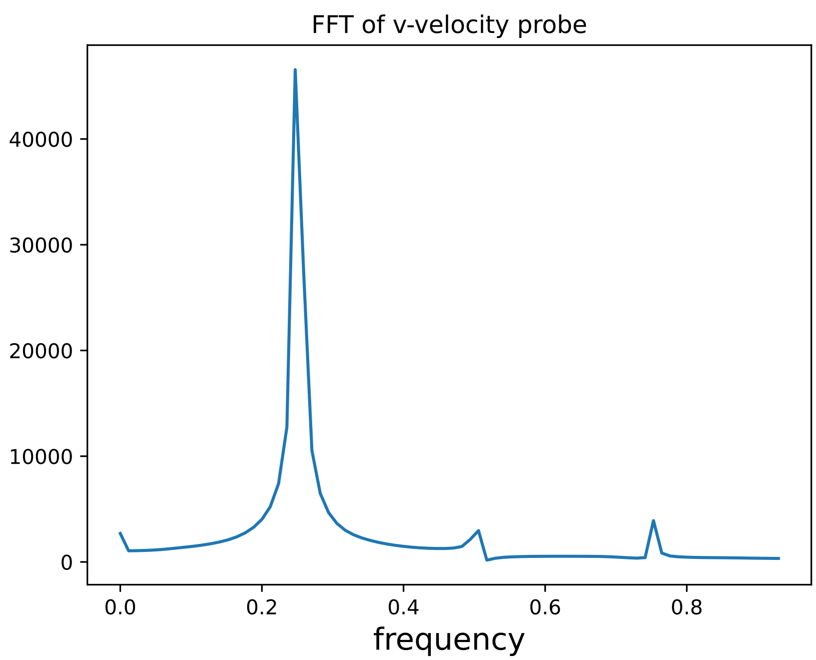


Figure 38: FFT of v time history in steady state region

7 Flow past ellipse at Re=300

7.1 Velocity, Pressure and Vorticity Contours

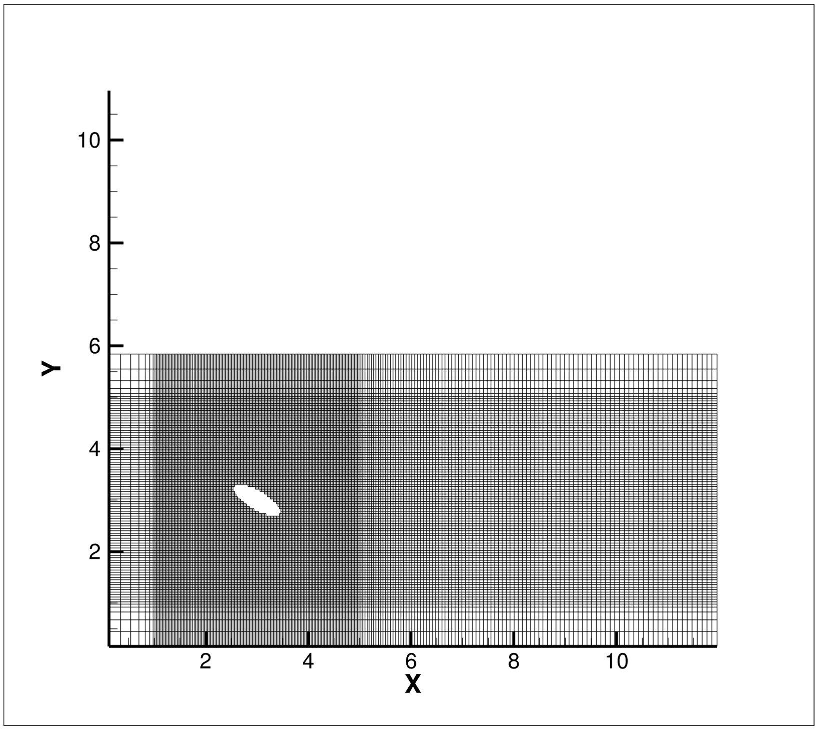


Figure 39: Geometry and mesh of domain

Now we have velocity, pressure, vorticity and Q countours at t=6,12,30 and 70 sec

7.2 Drag,Lift and Shedding Frequency

Drag and lift curve is shown in is shown in Fig 46. The mean drag coefficient in steady region is 0.654 and mean lift coefficient is 0.679.

Velocity and pressure time history at probe location is shown in Fig 47. The FFT of v velocity probe data gives dominant frequency gives dominant shedding frequency of 0.329.

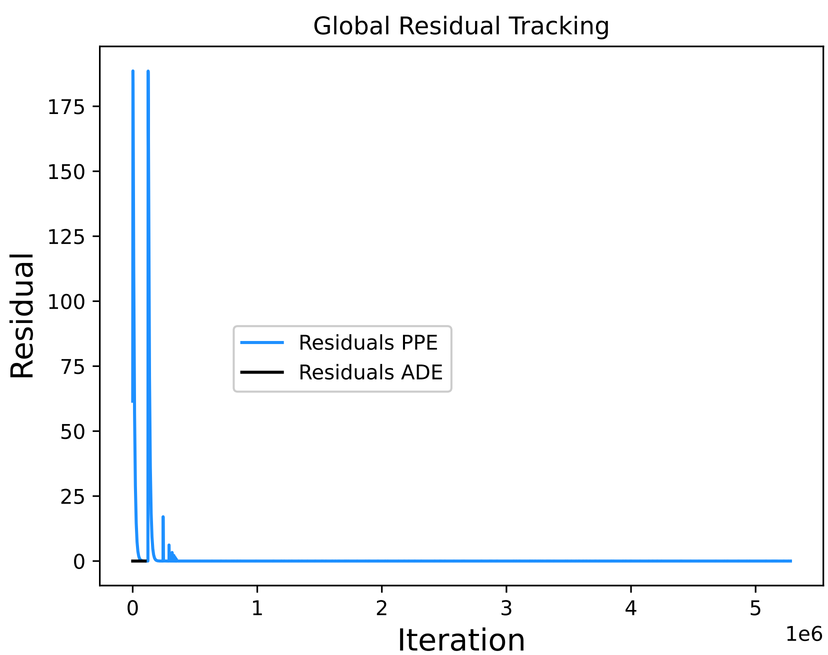


Figure 40: Residual history ellipse at Re=300

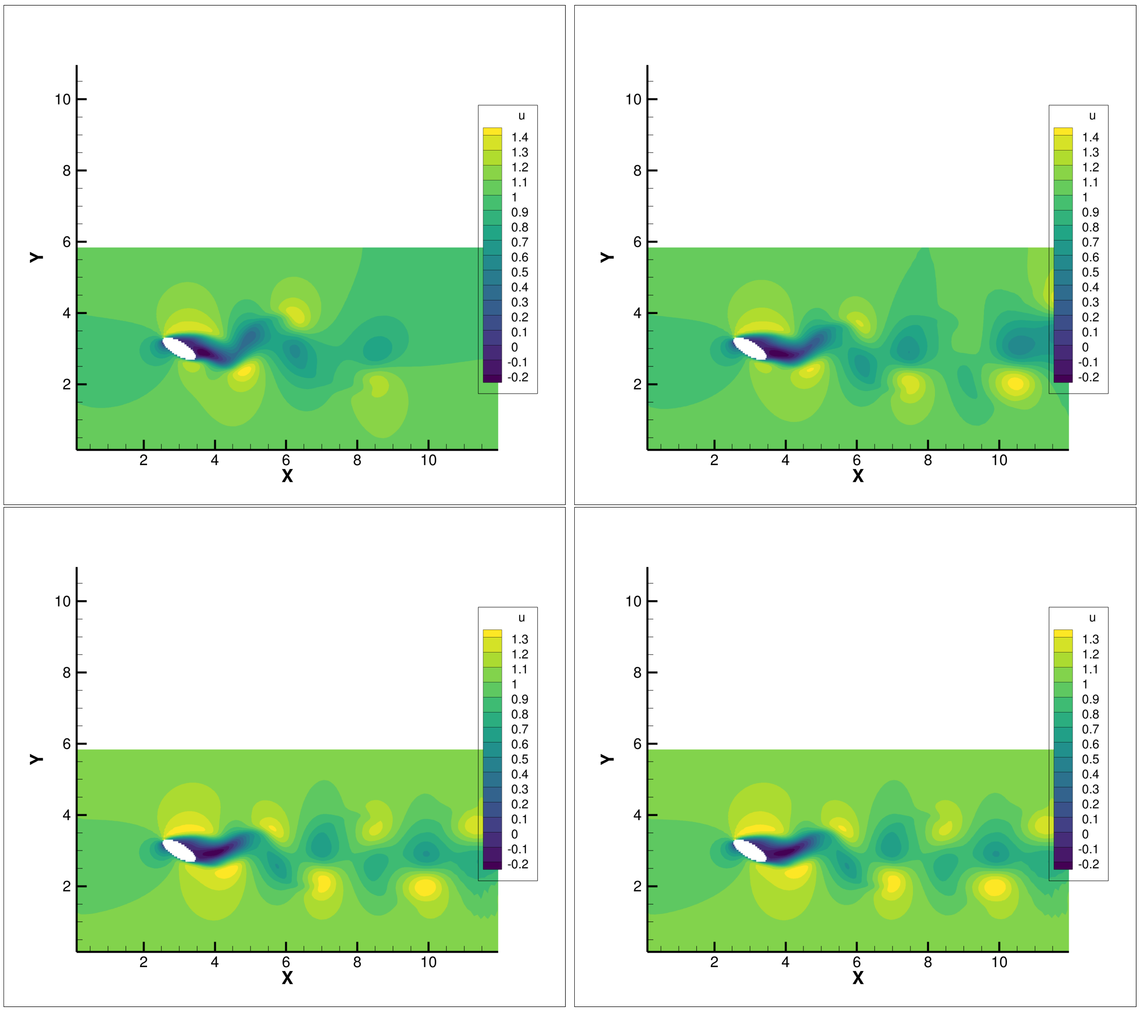


Figure 41: u contours for ellipse at Re=300 at t= 6,12,30 and 70 sec

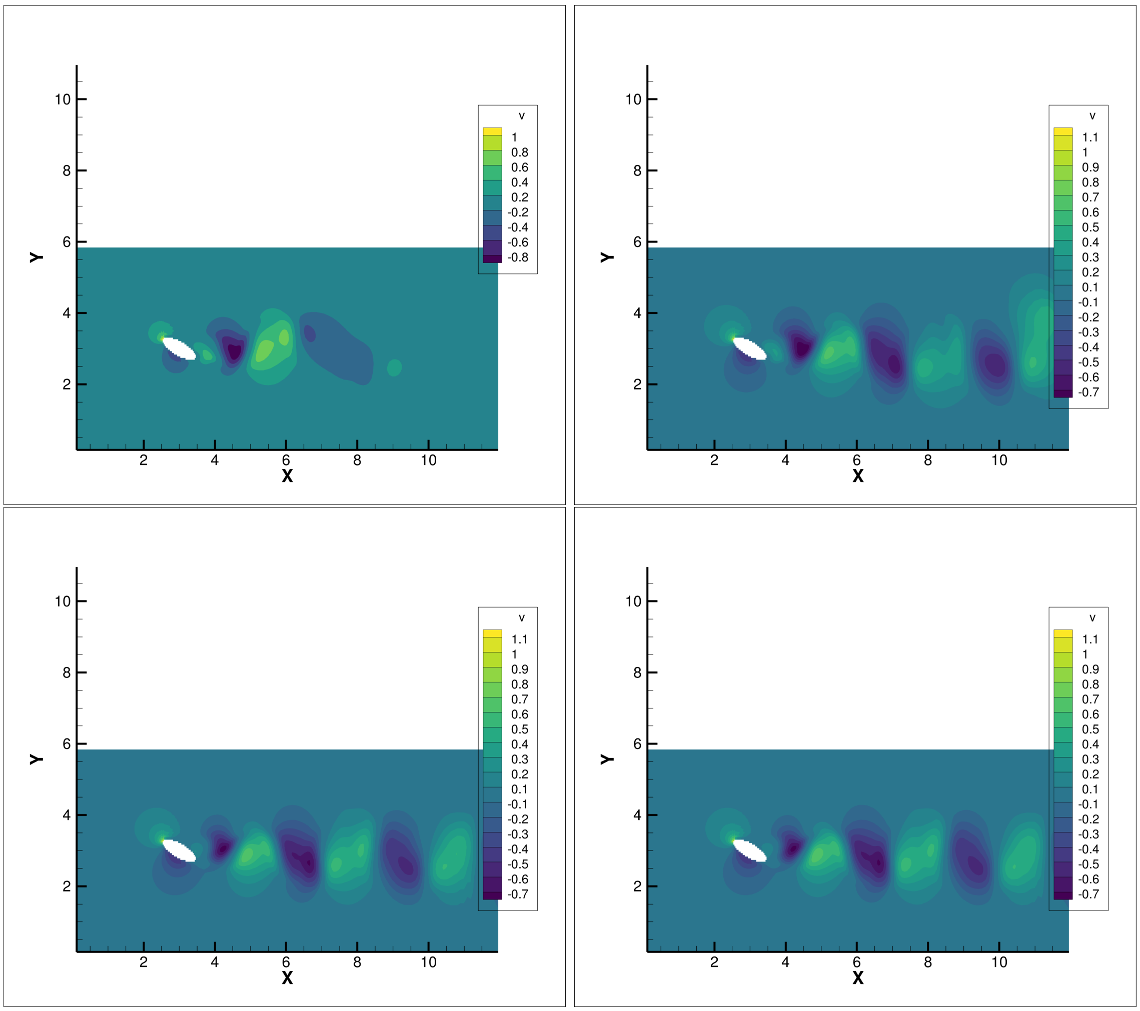


Figure 42: v contours for ellipse at Re=300 at t= 6,12,30 and 70 sec

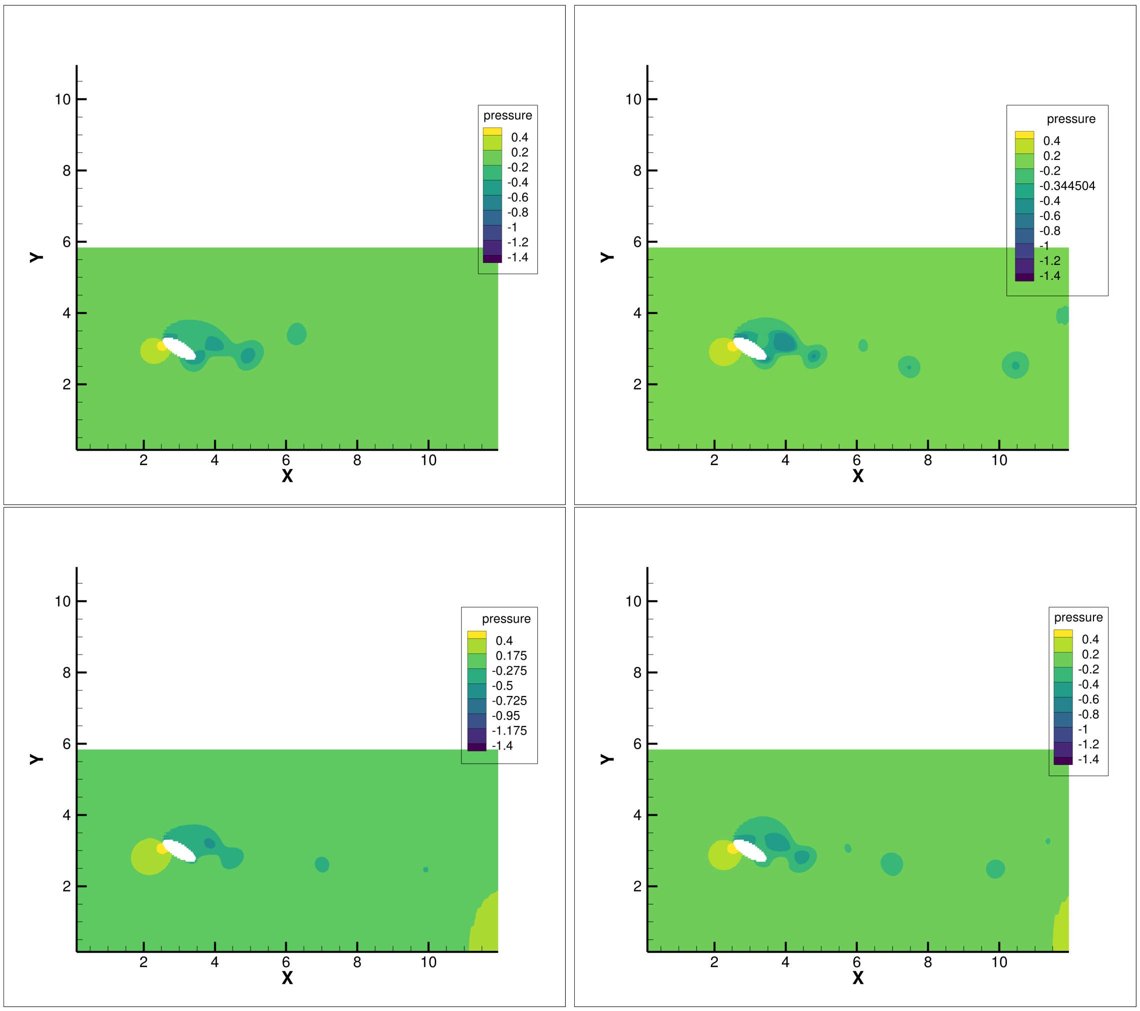


Figure 43: Pressure contours for ellipse at Re=300 at t= 6,12,30 and 70 sec

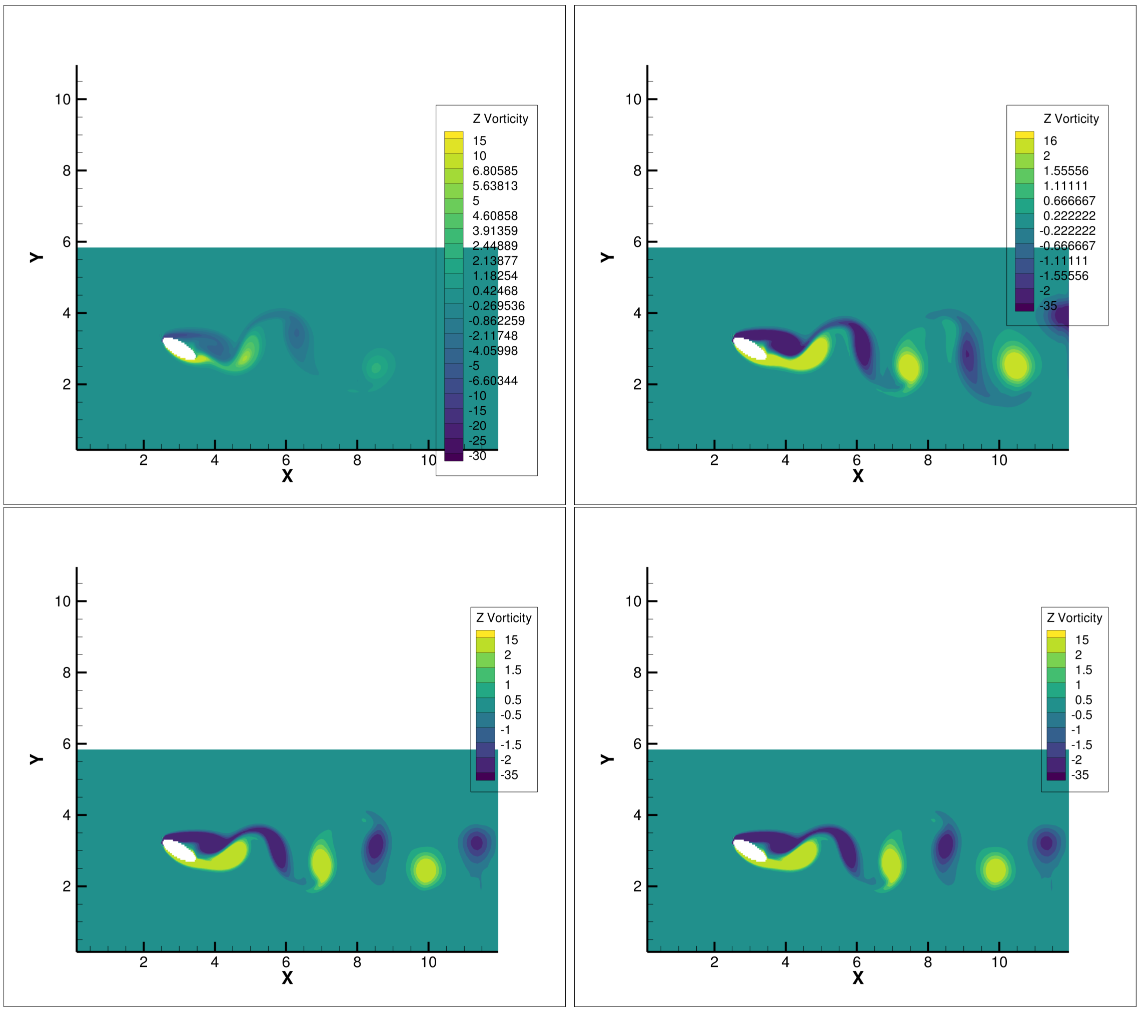


Figure 44: Vorticity contours for ellipse at Re=300 at t= 6,12,30 and 70 sec

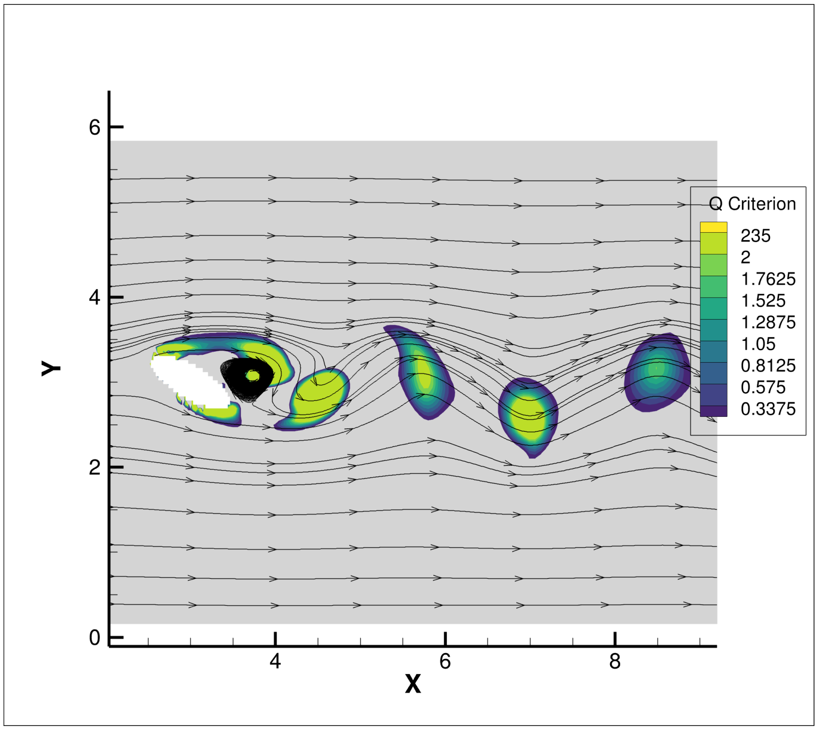


Figure 45: streamlines and Q criterion for ellipse at Re=300 at t=70sec

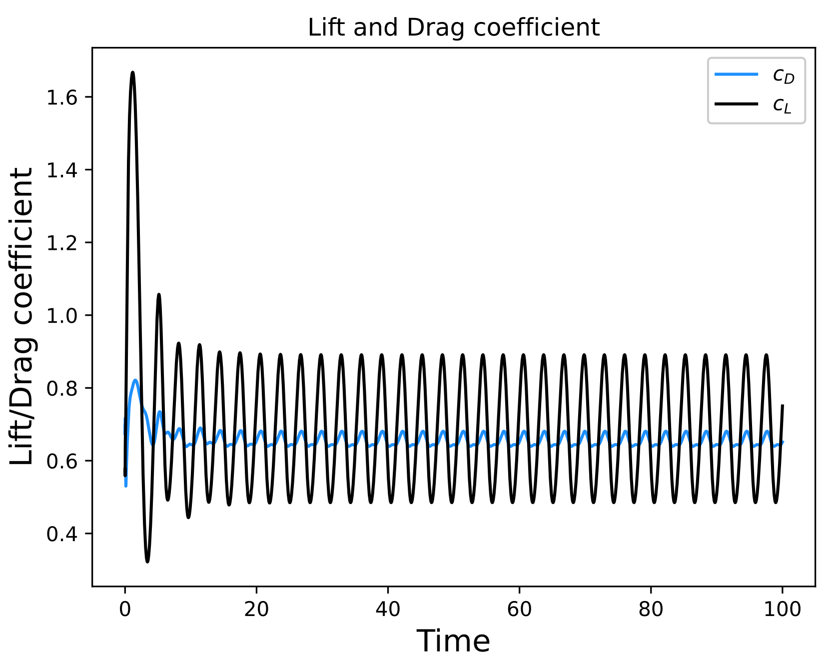


Figure 46: Drag and Lift coefficient for ellipse at Re=300

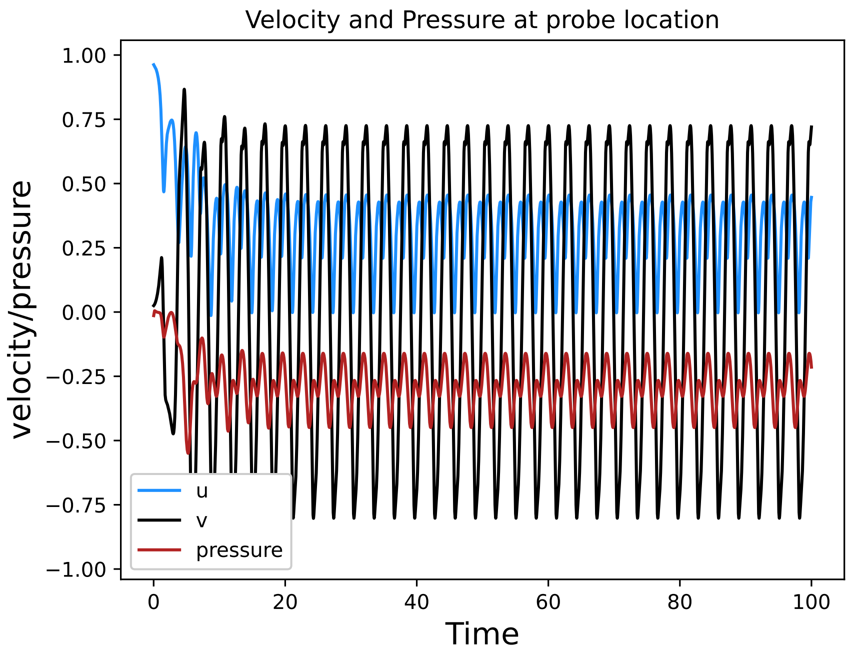


Figure 47: velocity and pressure time history at probe location (x,y)=(4.5,3)

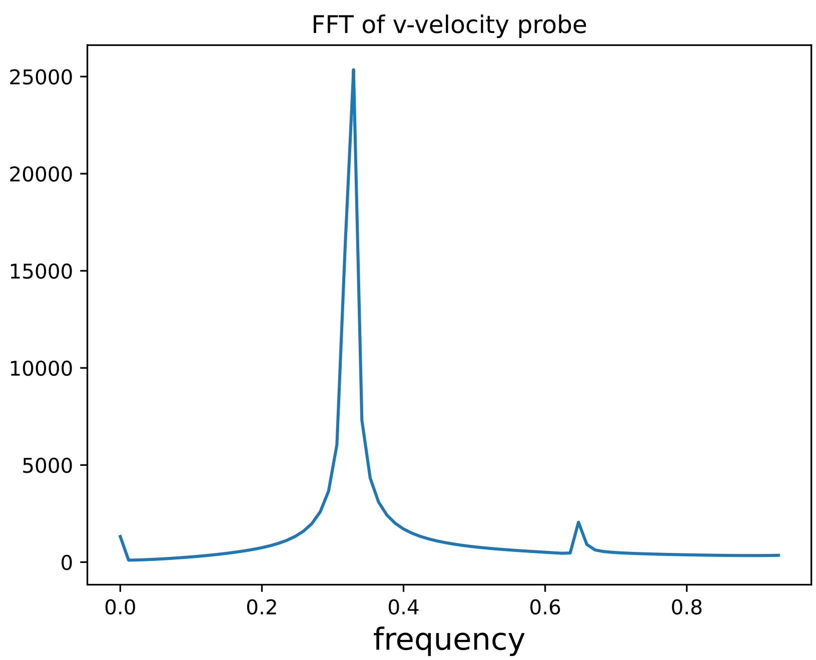


Figure 48: FFT of v time history in steady state region

8 Effect of domain size

To see how domain size may affect our results, Re=150 case is taken. dx and dy are fixed as 0.05. First length of domain in x is changed while keep height of domain fixed to 6D, keeping cylinder fixed at (3D,3D) for all case. The results are shown in Table 5. All simulation are run for 35 sec and steady region is used to compute average drag and shedding frequency.

Table 5: Effect of x domain length on shedding frequency, f and Drag coefficient, c_D

L_x	c_D	f
5	1.263	0.2799
7	1.429	0.225
10	1.470	0.187
12	1.45	0.187
15	1.45	0.187

Now length of domain is fixed to 12D and height is changed and same as before, dx and dy are fixed as 0.05. The results are shown in Table 6. When height of the domain is changed, the cylinder is also moved so to be symmetric in y direction. We see that when top boundary is far from cylinder , like 5D away from center of cylinder, c_D value converges better. Note that height of domain is 6D when L_x is varied, so the results of drag coefficient in Table 5 when L_x is increased to max of 15 and in Table 6 when L_y is increased to max of 12 differ by 0.7. But the results shown earlier for Re=150 in section 4 is on fine mesh, so even with L_y=6, the results match results from literature.

Table 6: Effect of x domain length on shedding frequency, f and Drag coefficient, c_D

L_y	c_D	f
3	1.94	0.25
5	1.53	0.25
10	1.38	0.187
12	1.38	0.187

Drag coefficient depends on refinement near cylinder, because when mesh is coarse, it will be approximated by larger rectangles near surface which will change cylinder shape and hence drag too. Shedding frequency for buff bodies does not seem to vary a lot for same Re, like for cylinder and ellipse at Re=300, we see much more change in drag and lift coefficient rather than shedding frequency.

9 Flow past any bluff bodies

As long as we can provide iBlank locations to the code, we can provide any stationary boundary to the code to simulate flow past it.

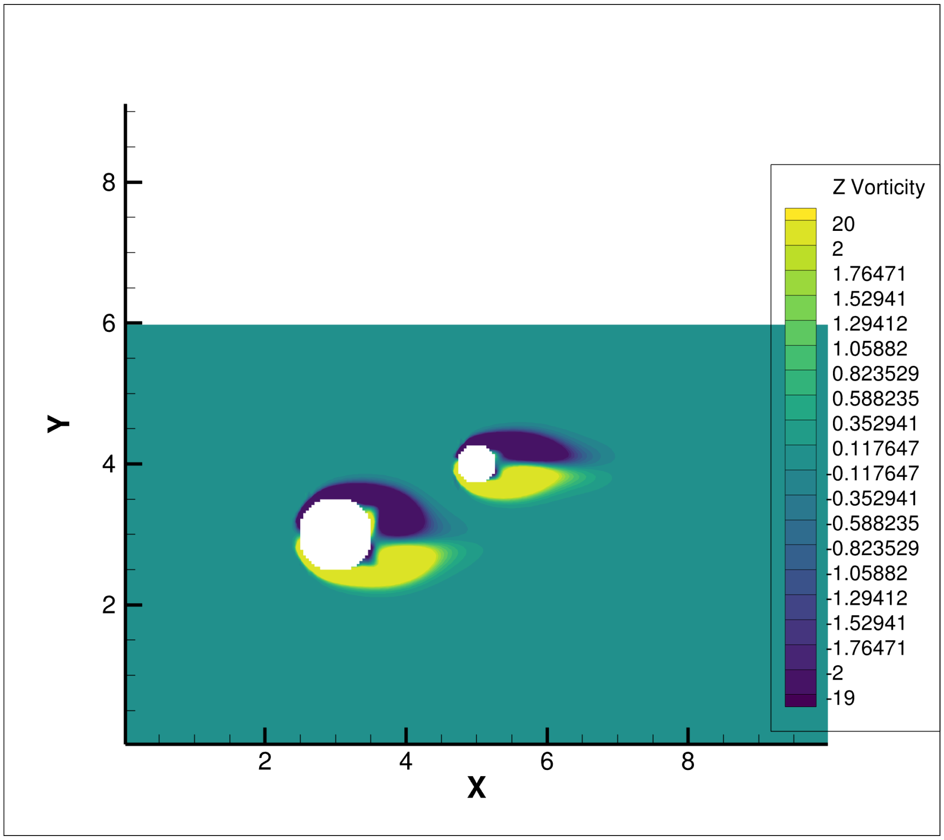


Figure 49: Flow past two small cylinder, Re=150, t=2 sec

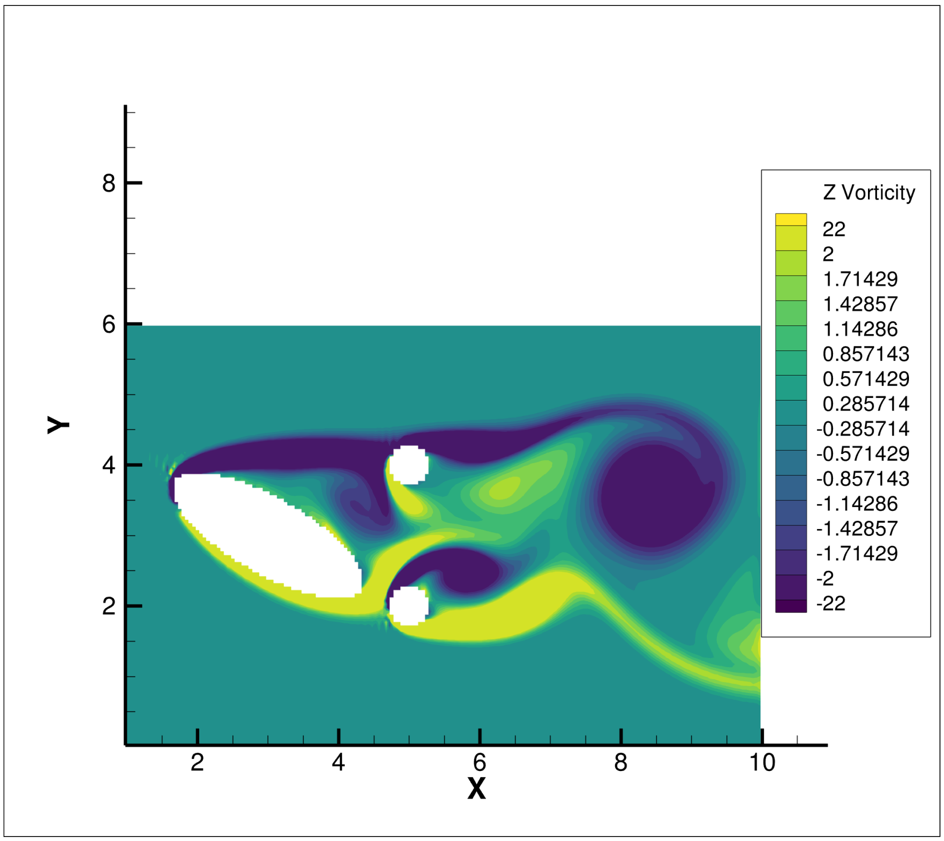


Figure 50: Flow past an eliipse and two small cylinder, Re=150, t=17 sec

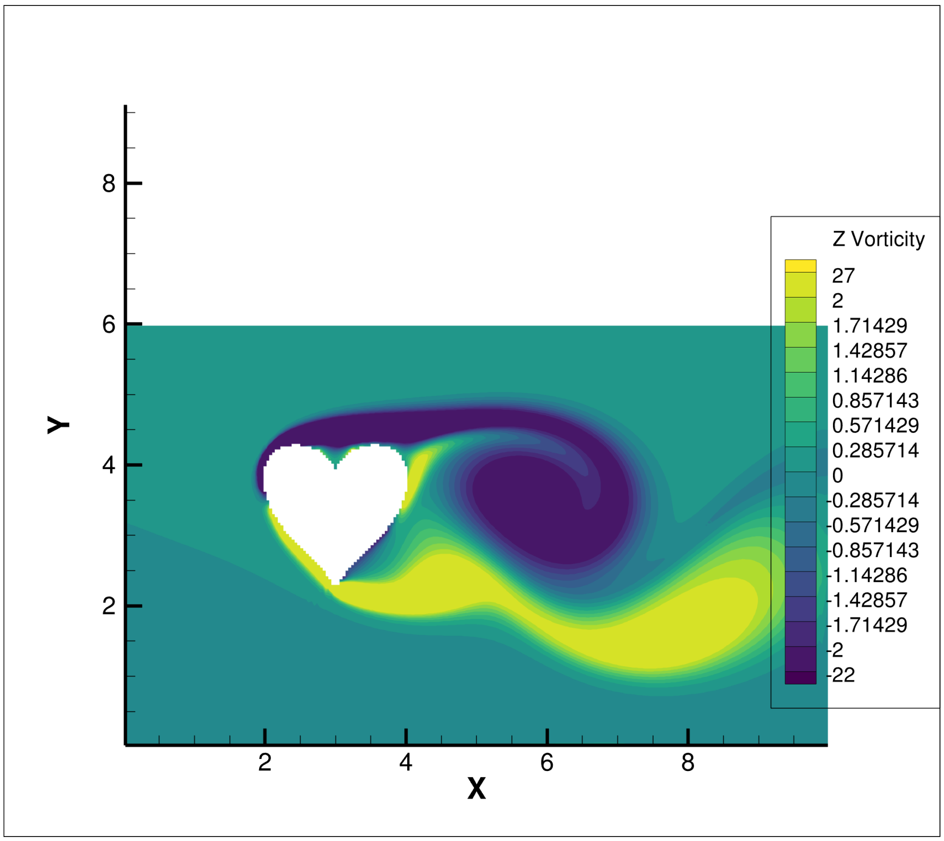


Figure 51: Flow past an 2D heart shape, Re=100, t=10 sec

10 References

1. Code Link: https://drive.google.com/drive/folders/1OiqDnRrSq-UvlUptPPiRAa1HfSTHeqoa?usp=sharing
2. Numerical simulation of laminar flow past a circular cylinder - B.N. Rajani, A. Kandasamy, Sekhar Majumdar
3. Numerical calculation of laminar vortex-shedding flow past cylinders - R. Franke, W. Rodi, B. Schonung