Numerical Fluid Mechanics II

Summer Semester 2018

DELIVERABLE TASK II:

Turbulent flow over a square cylinder

LSTM
Erlangen

Submitted By:

Kudeti Siddhartha (22433020)

JishnuJayaraj (22448952)

Mohammad Moataz (22455424)

Vasu Kolli (22290271)

Contents

INTRODUCTION

We modeled the incompressible viscous turbulent flow around a square cylinder using the OpenFoam toolbox. We used the RANS k-epsilon model for solving this case.

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).

This is a 3D-case in which the dimensions of the domain are as follows,

Length = 96 cm , Height = 56cm, Width = 40cm and Diameter of square cylinder = 4cm.

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
- Outlet pressure = 0

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 ineach 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.

RAS Model

Turbulent Flows fluctuates on a broad range of time and length scales. Hence the simulation of such flows are difficult and therefore we use RANS(Reynolds Averaged Navier Stokes equation) to model the equations and is used to describe the flow field.

In RANS approach to turbulence, all of the unsteadiness in the flow is averaged out and regarded as part of turbulence. The flow variables, are represented as sum of 2 components (average component and fluctuating component)

The solver (pimpleFoam) reads the 'turbulence properties' dictionary. Within the file 'simulationType' keyword controls the type of turbulence modelling to be used.(laminar,RAS,LES)

RAS(Reynolds averaged simulation) modelling
The governing equations are solved in ensemble-averaged form, including appropriate models for the effect of turbulence.

The choice of RAS modelling is specified in a 'RAS' sub directory. The following entries can be edited according to user requirements.

'RASModel' name of RAS turbulence model
'Turbulence' switch to turn the solving of turbulence modelling on or off
'printCoeffs' switch to print model coeffs to terminal at simulation start up
<RASModel> Coeffs optional dictionary of coeffs for the respective RASModel, to override the default coeffs

We are using kepsilon (2 equation)RAS model
It solves 2 extra pde for solving the flow equation. It make use of turbulent kinetic energy(per unit mass) and a model equation for epsilon to solve

Advantage of k epsilon

- Relatively simple to implement in openfoam library
- Leads to stable calculations that converge relatively easily
- Reasonable predictions for many flow

Disadvantages

- poor predictions for
  - swirling and rotating flows
  - Flows with strong separation
  - Axisymmetric jets
  - Fully developed flows in non circular ducts
- Valid only for turbulent flows and requires wall functions to be computed for implementation

Calculation of k-epsilon

Given turbulence intensity $I = \frac{u'}{U} = 0.02$

It is known that the velocity fluctuation $u' = \sqrt{\frac{2}{3}k}$

Also turbulent kinetic energy can be calculated as $k = \frac{3}{2}(u')^2 = \frac{3}{2}(UI)^2 = \frac{3}{2}(0.54 * 0.02)^2 = 0.00017496$

Given the turbulence length scale $l = 0.1\, d = 0.1 * 0.04 = 0.004$

We can finally calculate epsilon as $\varepsilon = C_\mu^{3/4}\frac{k^{3/2}}{l} = (0.09)^{3/4}\frac{(0.00017496)^{3/2}}{0.04667} = 9.5 * 10^{-5}$

Grid convergence study

This study provides a consistent manner to provide an error band on the grid convergence of the solution. It Enables a systematic way to estimate the discretization error.

The sizes of three grids chosen for the study are :

- 480*280*8
- 240*140*8
- 120*70*8

The steps are as follows,

a) Define representative cell, mesh/grid size,

   h= [$\Delta x_{max} \Delta y_{max} \Delta z_{max}$]^(1/3)

   Therefore,

   - h1 = $[\frac{96*56*40}{480*280*8}]^{\frac{1}{3}}$ = 0.58480
   - h2 = $[\frac{96*56*40}{240*140*8}]^{\frac{1}{3}}$ =0.928317
   - h3 = $[\frac{96*56*40}{120*70*8}]^{\frac{1}{3}}$ =1.473612

b) Coefficients of drag (cd) is taken as the variable of interest for the study,

Grid size	Cd
480*280*8	1.456136
240*140*8	1.604905
120*70*8	1.771364

The grid refinement factor 'r' should be = $\frac{h_{coarse}}{h_{fine}}$ ≥ 1.3. In our case we have a constant grid refinement factor of 1.5874.

c) Calculation of the order of convergence,

Since we have a constant grid refinement factor the order of convergence p simplifies to

P= $\left[\frac{1}{\ln(r)}\right]\left[ln\left|\frac{cd3-c}{cd2-cd1}\right|\right]$ = 0.243139

d) The error estimates for the observed order p,

Relative error (e_a^{21}) = $\left|\frac{cd1-c}{cd1}\right|$ = 0.1021669

Relative error (e_a^{32}) = $\left|\frac{cd2-cd3}{cd2}\right|$ = 0.1037189

e) The Grid Convergence Index (GCI) :

GCI^{21} = $\frac{\mathrm{Fs}.e_a^{21}}{r^p-1}$ = 1.074

GCI^{32} = $\frac{\mathrm{Fs}.e_a^{32}}{r^p-1}$ = 1.0903

Where, factor of safety (Fs) = 1.25 for comparison over three or more grids

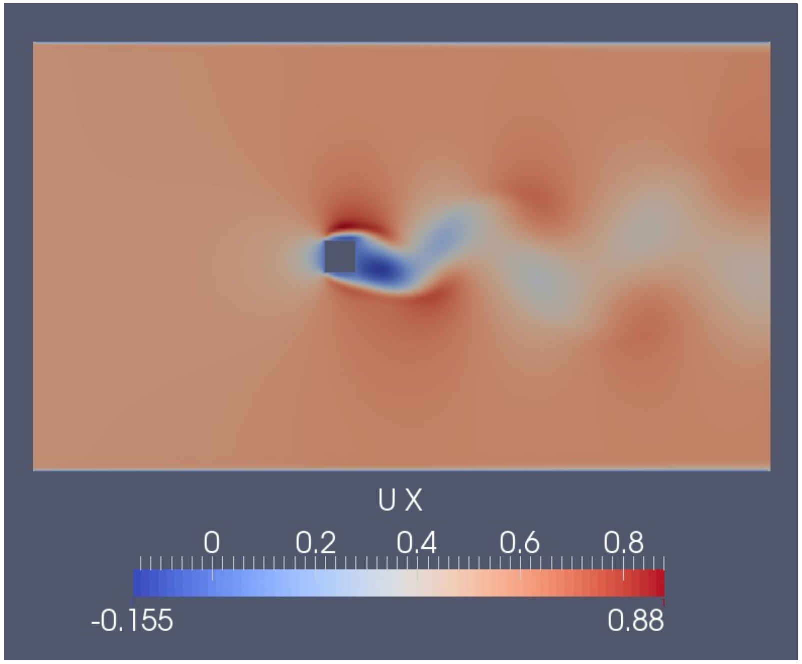
f) Asymptotic range of convergence :

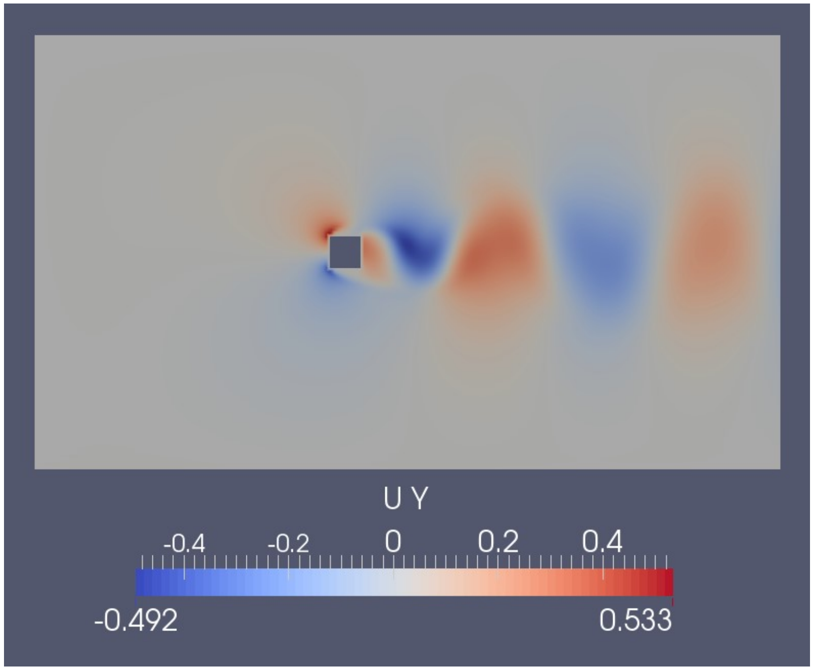
It is important that each grid level yield solutions that are in the asymptotic range of convergence for the computed solution. If the solutions are in the asymptotic range of convergence then the mesh can be considered as fine enough. This can be checked by ,

$$\frac{GCI^{32}}{r^p\ GCI^{21}} = \frac{1.0903}{r^p * 1.074} = 0.907 \approx 1$$

This means that the second finest grid will have approximately 90% of the finest grid accuracy and that is acceptable in our case.

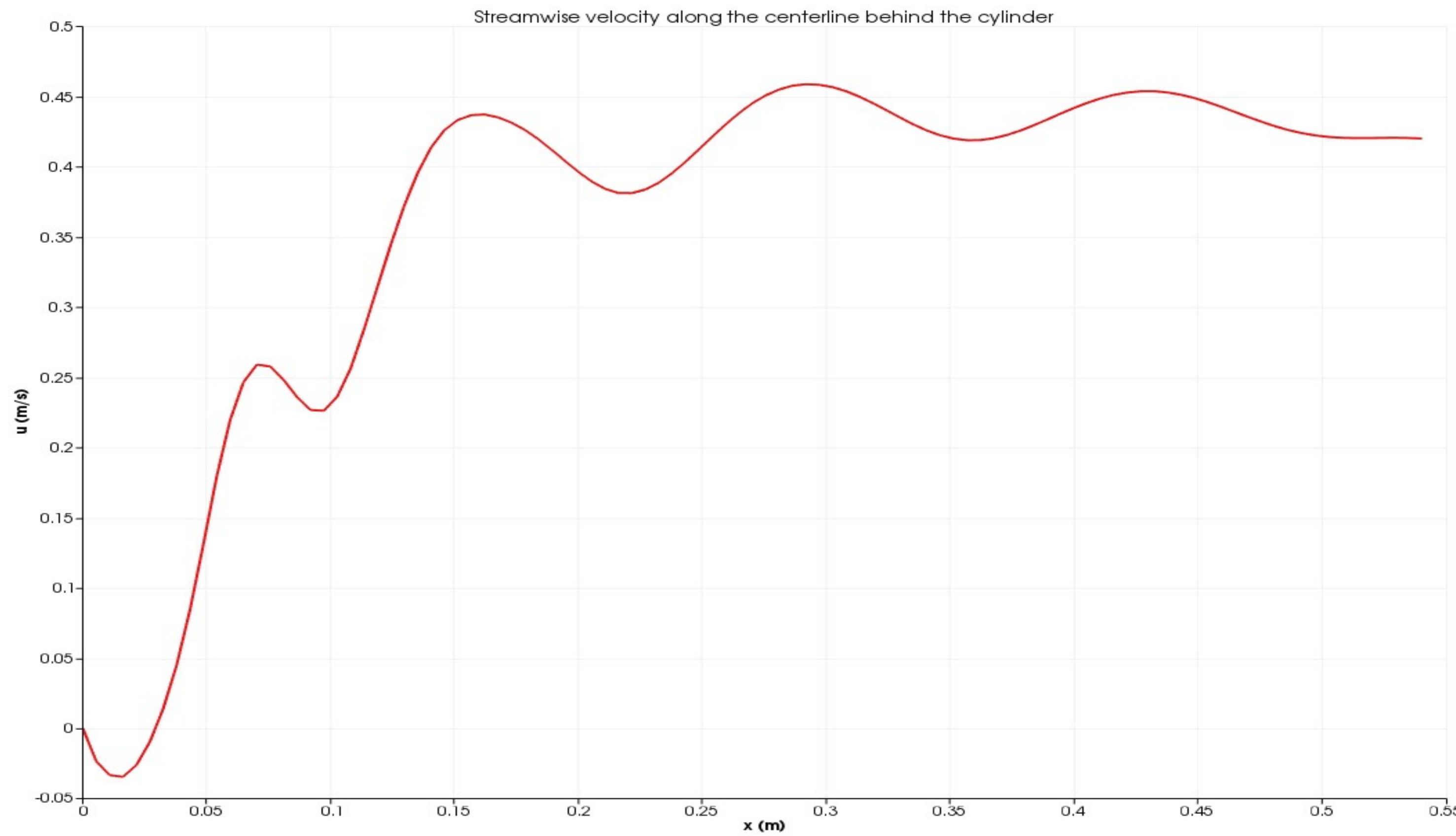
Velocity contour





The Flow speed outside the wake region is much higher than inside. A pair of counter rotating vortices are observed downstream the obstacle. They result from the shear layer formed on top and bottom surface of the square block and they roll inside because of the low velocity wake region. These vortices of opposite sign roll up in a alternative manner resulting in vortex shedding.

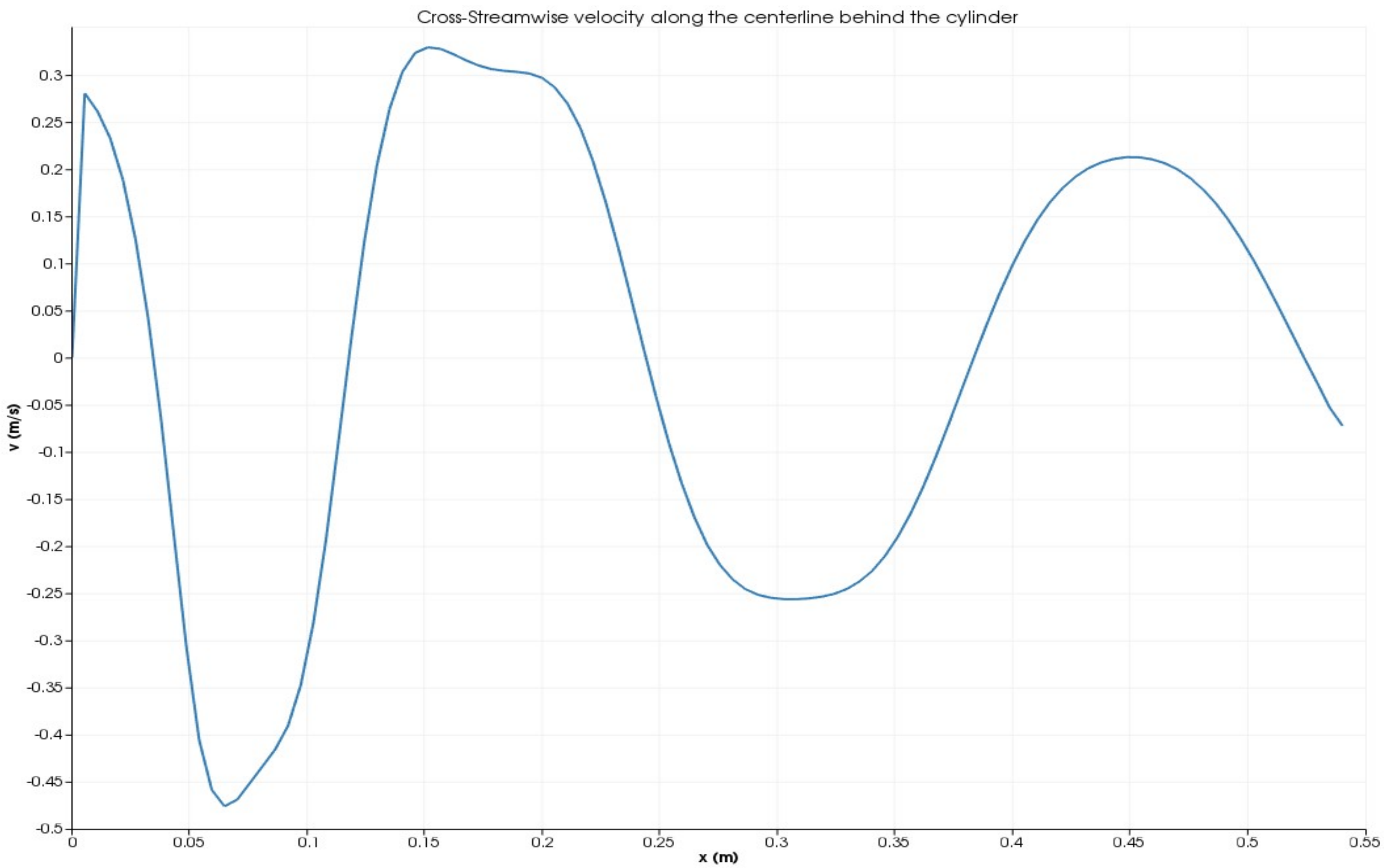
U-plot on the centreline behind the cylinder



Just downstream the cylinder the streamwise velocity is negative due to the recirculation.

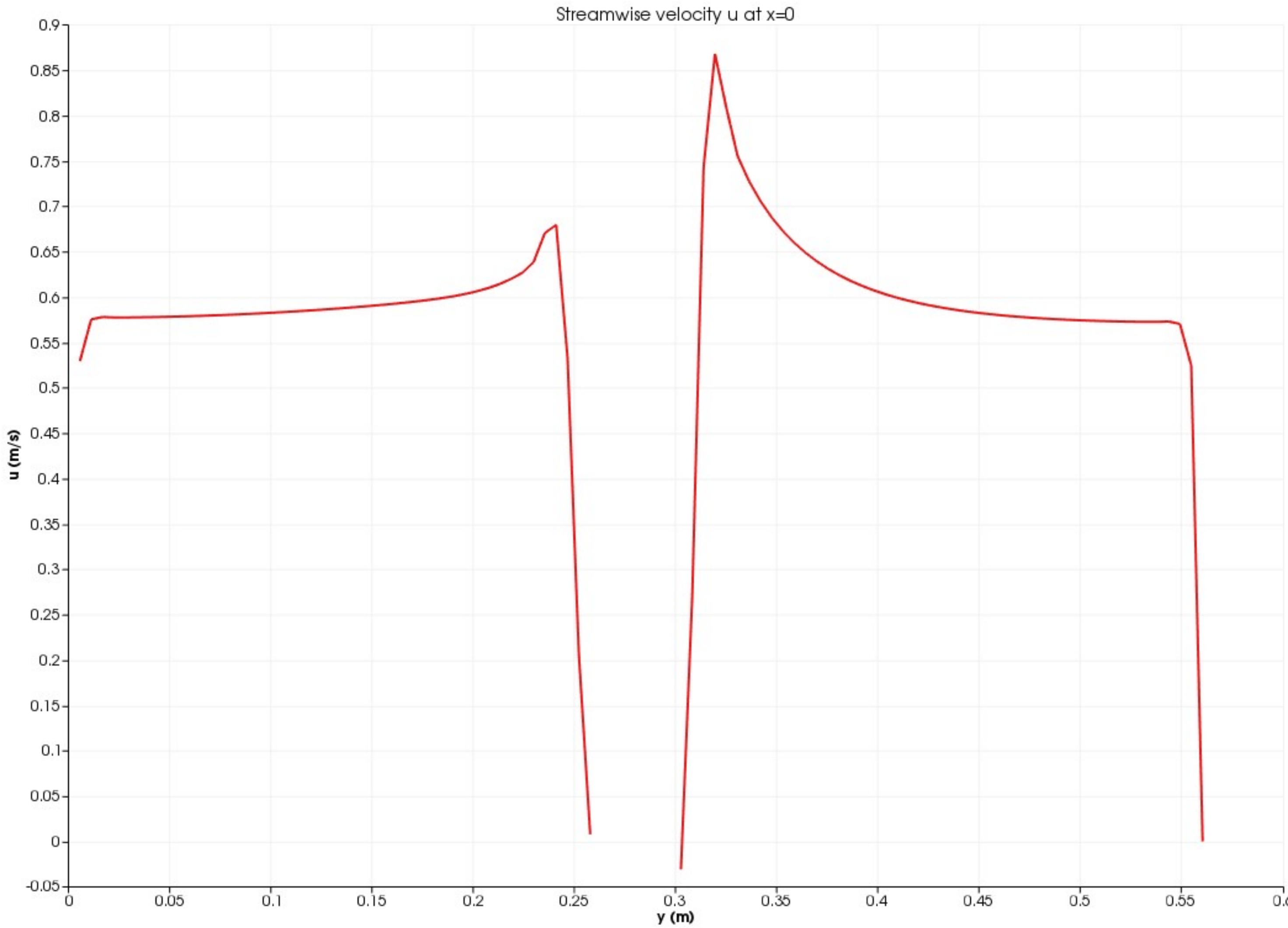
From the plot the recirculation length is approximately 0.04m downstream the cylinder. After which the velocity becomes positive again and increase gradually till it reaches a value near the inlet velocity (0.54 m/s) . We can observe that the oscillations that happen due to the vortex shedding are damped as we go downstream because the disturbance created by the obstacle is becoming less effective and the vortex energy is dissipating and joins the main stream.

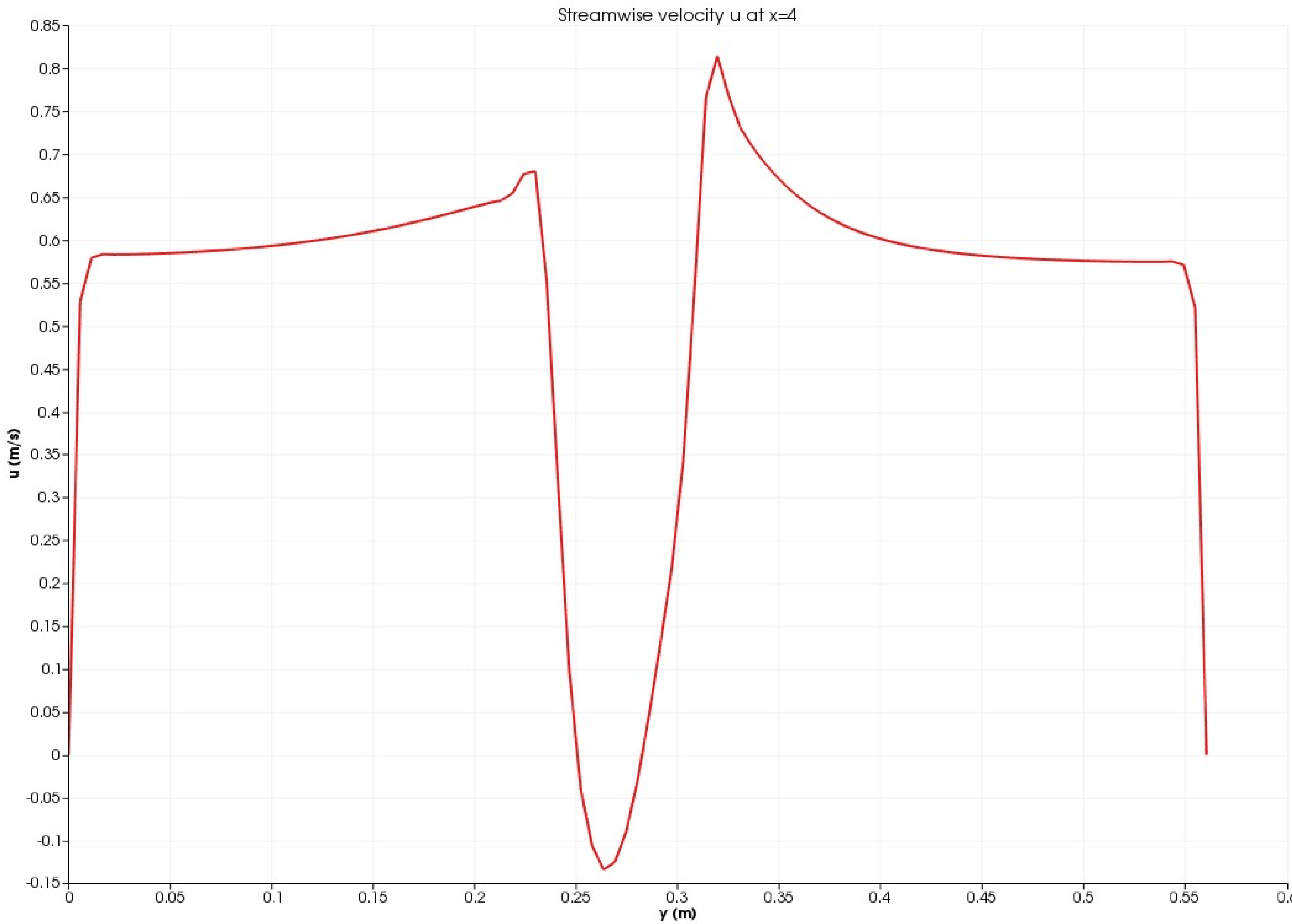
V plot on the centreline behind the cylinder

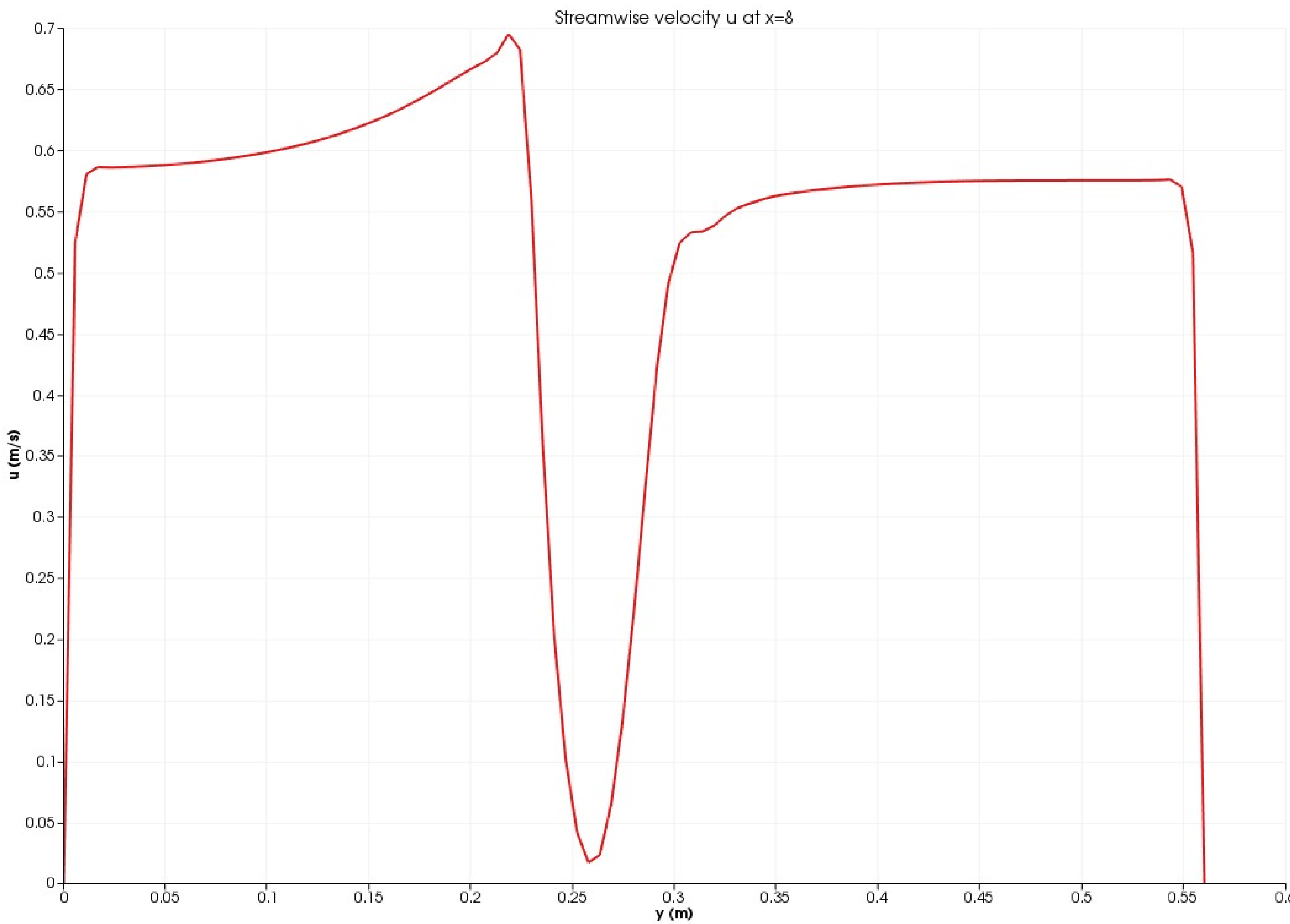


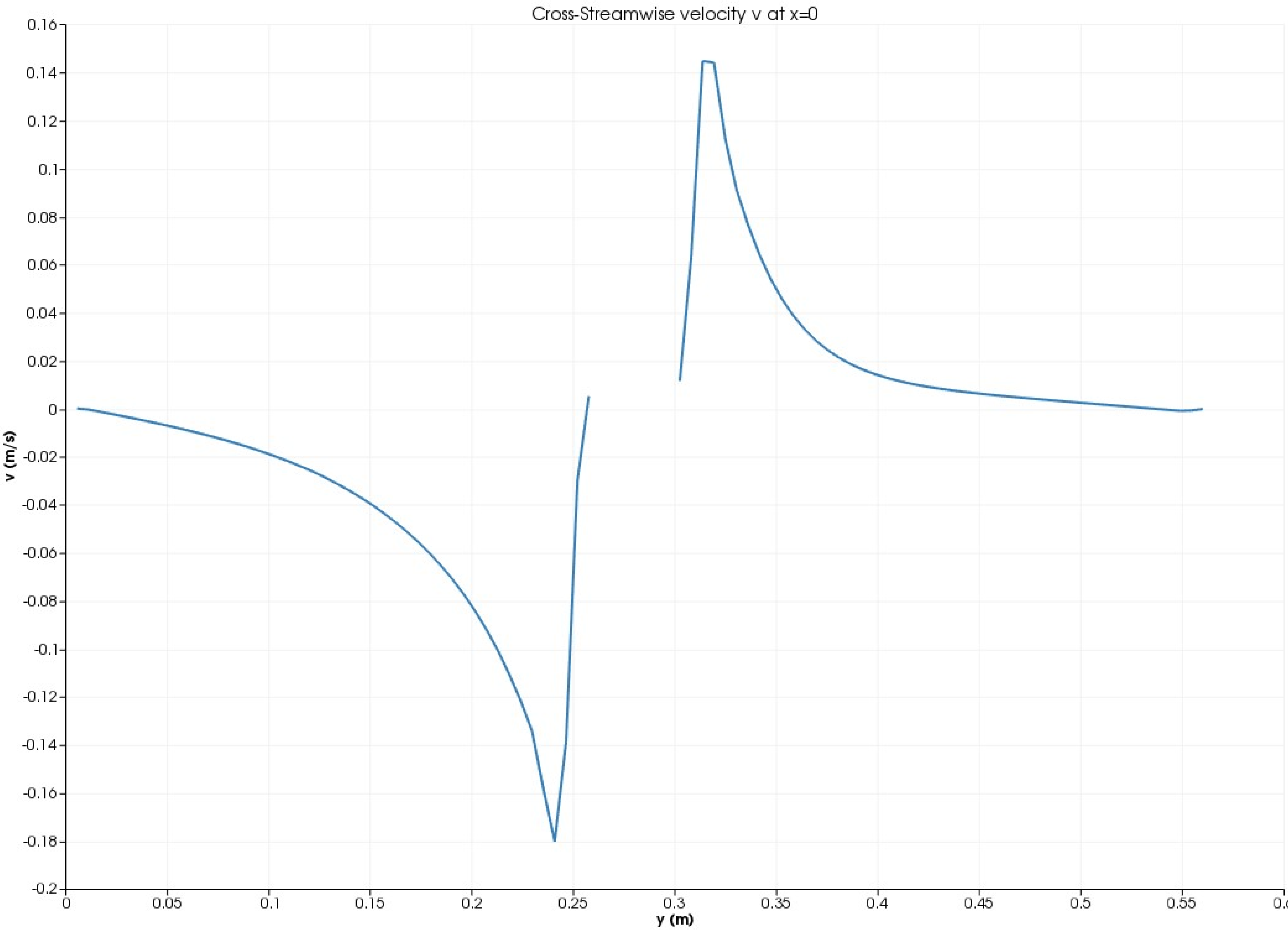
For the laminar case the cross-streamwise velocity along the centreline behind the cylinder would have been zero. However, in our case since the flow is turbulent there exists vortex shedding and we can notice large fluctuations of velocity in cross-stramwise direction that are damped as we go downstream because the disturbance created by the obstacle is becoming less effective and the vortex energy is dissipating and joins the main stream.

Streamwise and Cross-streamwise Velocity profiles at x=0,4 and 8 cm

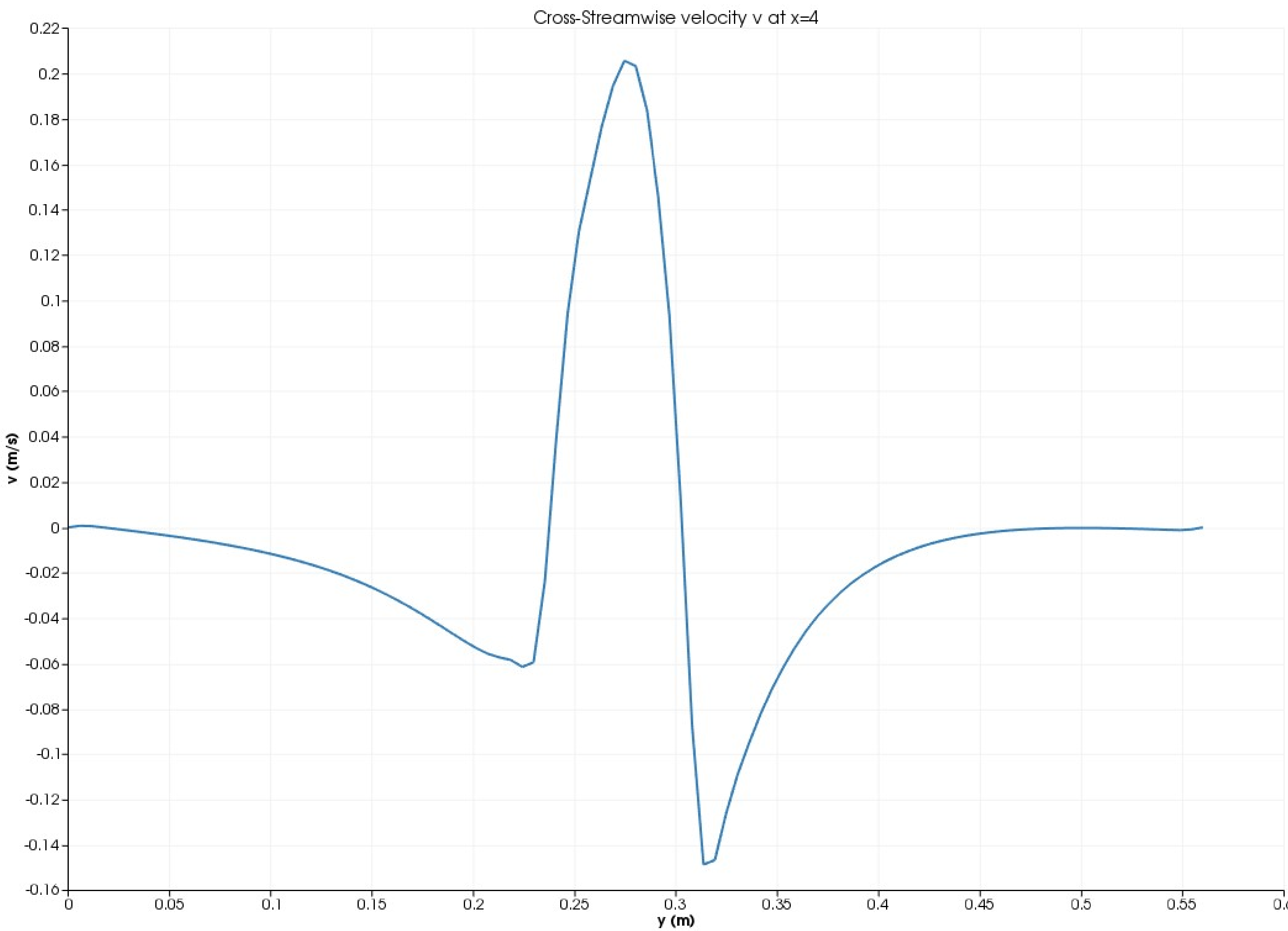



Streamwise velocity u at x=4
u (m/s)
y (m)
0.85
0.8
0.75
0.7
0.65
0.6
0.55
0.5
0.45
0.4
0.35
0.3
0.25
0.2
0.15
0.1
0.05
0
-0.05
-0.1
-0.15
0
0.05
0.1
0.15
0.2
0.25
0.3
0.35
0.4
0.45
0.5
0.55
0.6

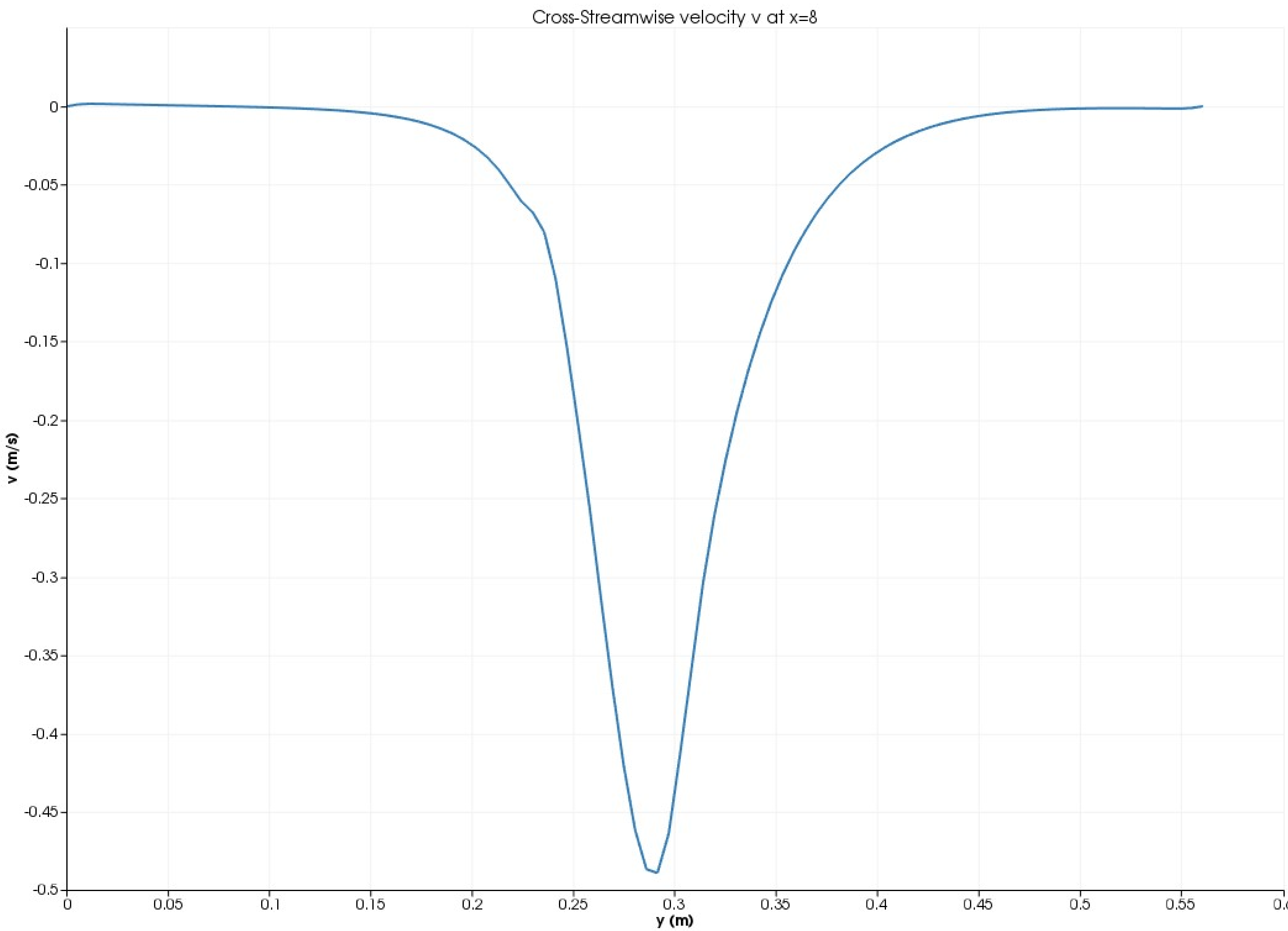

Streamwise velocity u at x=8
u (m/s)
y (m)
0
0.05
0.1
0.15
0.2
0.25
0.3
0.35
0.4
0.45
0.5
0.55
0.6
0.65
0.7



Cross-Streamwise velocity v at x=0
v (m/s)
y (m)
0.16
0.14
0.12
0.1
0.08
0.06
0.04
0.02
0
-0.02
-0.04
-0.06
-0.08
-0.1
-0.12
-0.14
-0.16
-0.18
-0.2
0
0.05
0.1
0.15
0.2
0.25
0.3
0.35
0.4
0.45
0.5
0.55
0.6



Cross-Streamwise velocity v at x=4
v (m/s)
y (m)
0.22
0.2
0.18
0.16
0.14
0.12
0.1
0.08
0.06
0.04
0.02
0
-0.02
-0.04
-0.06
-0.08
-0.1
-0.12
-0.14
-0.16
0
0.05
0.1
0.15
0.2
0.25
0.3
0.35
0.4
0.45
0.5
0.55
0.6



Cross-Streamwise velocity v at x=8
v (m/s)
y (m)
0
-0.05
-0.1
-0.15
-0.2
-0.25
-0.3
-0.35
-0.4
-0.45
-0.5
0
0.05
0.1
0.15
0.2
0.25
0.3
0.35
0.4
0.45
0.5
0.55
0.6

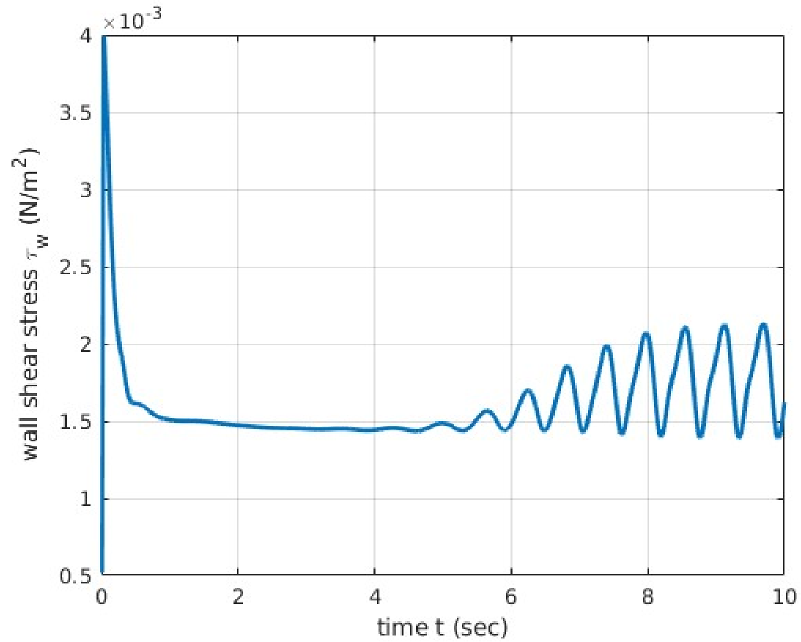
General Observations on velocity graphs at x=0,4 and 8 cm:

a) The presence of the phenomenon of vortex shedding leads to velocity fluctuations and non-symmetry of the velocity profiles. They will always by higher at one side than the other.
b) The effect of disturbance created by the cylinder is high near the cylinder and fades gradually as we go downstream. This can be easily noticed by checking how the velocity profiles gets closer and closer to the fully developed symmetric turbulent profile as we go downstream.

Applying these general observations on each graph we can notice the following

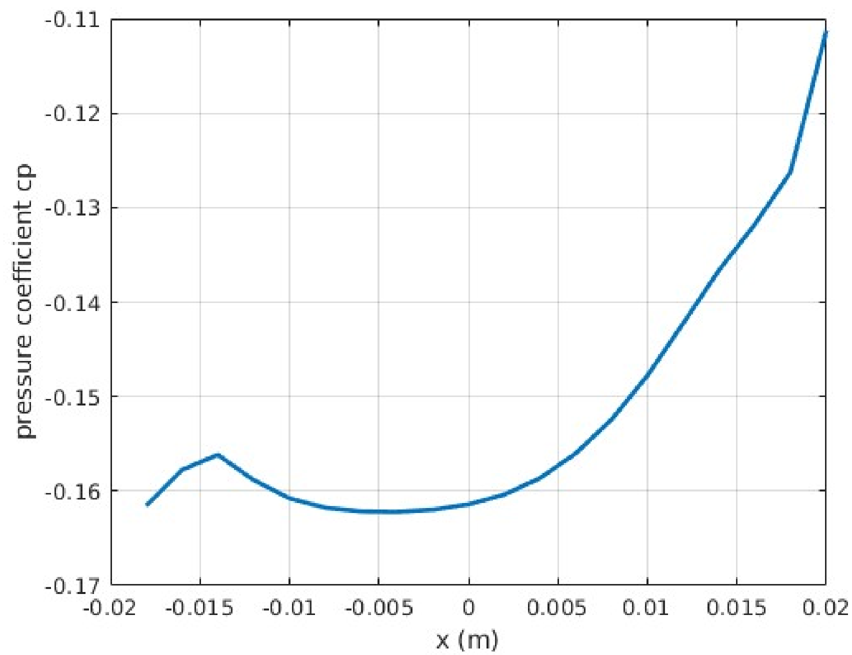
Drawing	How the turbulent case deviates from the laminar
u at x=0 cm	• Zero velocity at cylinder and domain walls due to no slip condition • Non-symmetric increase in the magnitude of velocity on both sides of the cylinder due to vortex shedding. • The maximum peak has a high value because at this point the effect of the disturbance created by the cylinder is high.
U at x=4 cm	• Zero velocity at the domain walls due to no slip condition • Negative minimum velocity values near the domain centreline as the recirculation is still detected at that point • Similar non-symmetry as at x=0 (i.e. the higher value is on the same side) since both lines cross the same vortex. • The maximum peak has less value than at x=0 because the disturbance effect starts to get less.
U at x=8	• Zero velocity at the domain walls due to no slip condition • Positive minimum velocity values near the domain centreline as this point is downstream the recirculation region. • The maximum peak has less value than at x=0 and x=4 because the disturbance effect is getting less and less.
V at x=0	• Zero velocity at cylinder and domain walls due to no slip condition • Non-symmetric increase in the <u>absolute</u> magnitude of velocity on both sides of the cylinder due to vortex shedding.
V at x=4	• Zero velocity at the domain walls due to no slip condition • Non-symmetric increase in the <u>absolute</u> magnitude of velocity on both sides of the cylinder due to vortex shedding and because the line x=4 crosses 2 opposing regions as can be seen from the v contour
V at x=8	• Zero velocity at the domain walls due to no slip condition • From the v contour it can be shown that the line x=8 crosses only one region and that’s why it looks a little symmetric.

Wall Shear Stress



Time interval(seconds)	Flow regime	Comments
0-4	Laminar	• No vortices formed • Nearly constant shear stress
4-8	Transition	• Amplitude of the vortex shedding gradually increases causing velocity fluctuations • The shear stress values fluctuate accordingly
8-10	Turbulent	• Amplitude of the vortex shedding become nearly constant • The shear stress values fluctuate in a constant manner accordingly

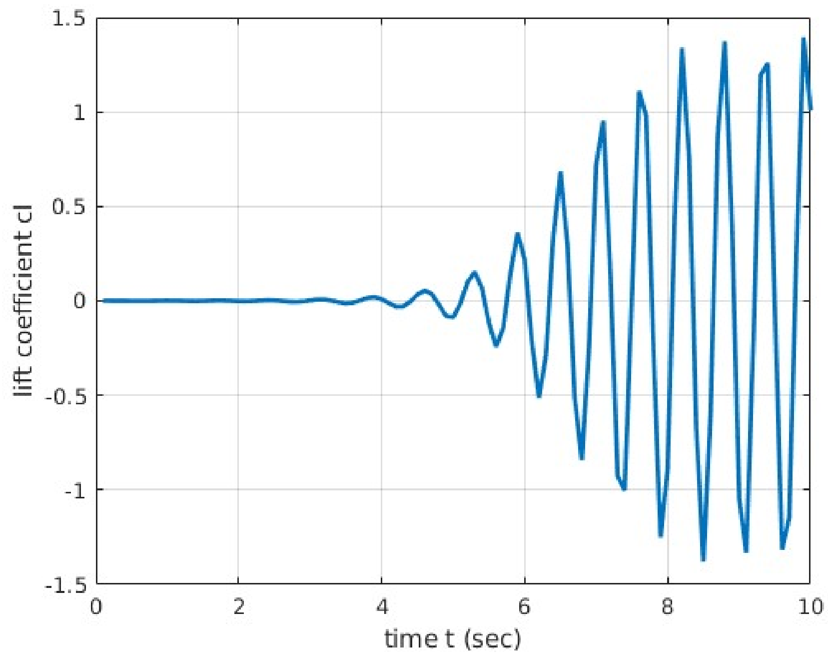
Pressure coefficient



As the flow impinges the front wall of the cylinder, the front face of the square experience a high pressure values. Then the sharp upstream corners forces the flow to separate causing two large recirculation at top and bottom areas of the cylinder. This strong curvature of stream lines induces a strong pressure gradient. Then the curvature of stream lines because of downstream corners increases the pressure value.

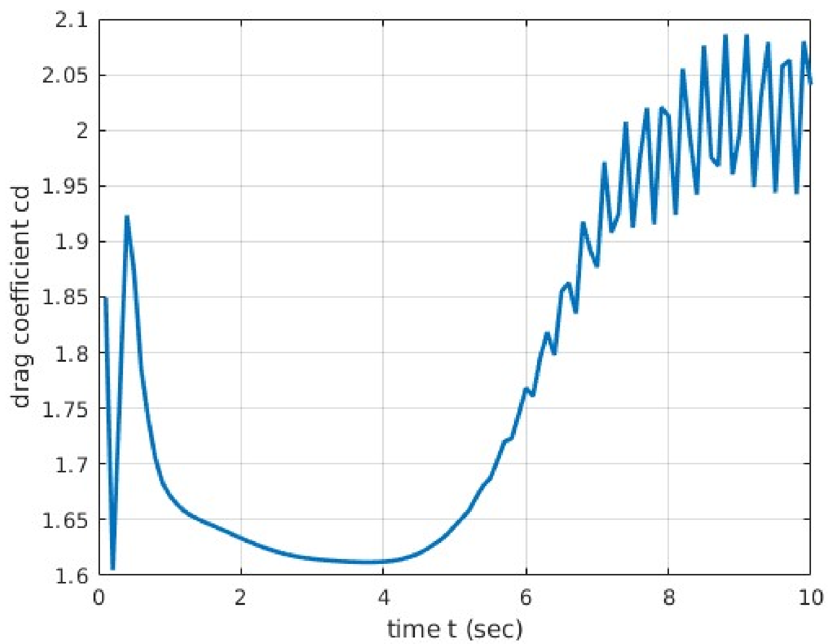
Because of the flow separation at the upper edge of the cylinder we have a negative pressure and pressure coefficient. Downstream the edge of the cylinder the fluid tends to recirculate due the pressure difference and this results an gradual increase in pressure as well as the pressure coefficient along the cylinder wall as we go downstream.

Lift Coefficient



Time interval(seconds)	Flow regime	Comments
0-4	Laminar	• No vortices formed • Nearly zero lift coefficient
4-8	Transition	• Vortices are formed and they start to shred • Amplitude of the vortex shedding gradually increases • The lift coefficient starts oscillating with the increase in amplitude
8-10	Turbulent	• Amplitude of the vortex shedding become nearly constant • So the lift coefficient oscillates with a constant amplitude

Drag Coefficient



Time interval(seconds)	Flow regime	Comments
0-4	Laminar	• No vortices formed • The drag coefficient decreases gradually till it reaches a minimum value
4-8	Transition	• Vortices are formed and they start to shred • Amplitude of the vortex shedding gradually increases • The drag coefficient increases gradually due to the disturbances in the flow • We can notice some small oscillations in Cd due to the formation of vortices
8-10	Turbulent	• The Amplitude of the vortex shedding become nearly constant • The mean value is much larger than the one in laminar regime • So the drag coefficient oscillates with a constant amplitude

Strouhal Number

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. Strouhal number is a dimensionless number describing oscillating flow mechanisms.

From the coefficient of lift plot, we can notice that at a particular point of time the amplitude of the vortex shedding becomes constant. From the graph we calculated the time period required for one oscillation and calculated the frequency of vortex shedding. The time period of oscillation is calculated and found to be ,

T= 0.52 seconds,

Frequency (f)= 1/T= 1.904 Hz

Strouhal number = $\frac{\mathrm{L} * \mathrm{f}}{U}$ = 0.04 * 1.904/0.54 = 0.133

For large Strouhal numbers (order of 1), viscosity dominates fluid flow, resulting in a collective oscillating movement of the fluid "plug". For low Strouhal numbers (order of 10^{-4} and below), the high-speed, quasi steady state portion of the movement dominates the oscillation. Oscillation at intermediate Strouhal numbers is characterized by the buildup and rapidly subsequent shedding of vortices.

Bonus Task

If LES is selected, the choice of LES modelling is specified in a LES sub-dictionary which requires the following entries.

- LESModel: name of LES turbulence model.
- delta: name of delta δ model.
- <LESModel>Coeffs: dictionary of coefficients for the respective LESModel, to override the default coefficients.
- <delta>Coeffs: dictionary of coefficients for the delta model.

For incompressible flows, different ‘LESModel’ can be chosen like

‘Smagorinsky’- The Smagorinsky SGS model , ‘WALE’-The Wall-adapting local eddy-viscosity (WALE) SGS model., ‘SpalartAllmarasDES’-SpalartAllmarasDES DES turbulence model for incompressible and compressible flows,

For incompressible flows we can choose

‘SpalartAllmarasDDES’- SpalartAllmaras IDDES turbulence model for incompressible and compressible flows

‘dynamicKEqn’ - Dynamic one equation eddy-viscosity model

We are using ‘oneEqEddy’ (One equation eddy-viscosity model for incompressible flow)

Final results and conclusions

There are 3 different source of vortices production in cylinder. The first two sources are created at the surface by both development of boundary layers due to oncoming flow on the front of the square block. 3th boundary layers will then separate at the tip of the block and produces shear layes of opposite signs and thus creating a votice flow. The third source is the back flow region formed behind the cylinder which is formed due to the interaction between cylinder , shear layer and vortex roll up that occur downstream.

The presence of the phenomenon of vortex shedding leads to velocity fluctuations and non-symmetry of the velocity profiles. They will always by higher at one side than the other.

The effect of disturbance created by the cylinder is high near the cylinder and fades gradually as we go downstream.