# Flow through Backward-facing Step

Manish Tajpuriya

Department of Mechanical and Aerospace Engineering

IOE Pulchowk Campus, Tribhuvan University

## Introduction:

The backward-facing step is a widely studied structure as it has a wide range of applications including aerofoil at a high angle of attack, separation flow around a vehicle, spoiler flow, and flow around buildings. It is a basic structure that helps us understand 3 important phenomena, flow separation, vortex formation and reattachment. This report includes a transient simulation of a flow through a backward-facing step. It includes both laminar and turbulent simulation of air and the change in reattachment length in both cases. The evolution of the recirculation zone in laminar and turbulent flow is analysed in this report. This report also includes the variation of velocity along the cross-section around the reattachment point.

## Problem Statement:

The backward-facing step is a standard test problem in CFD. Given below is the domain where a uniform flow of air enters from the left with a Reynolds number of 200.
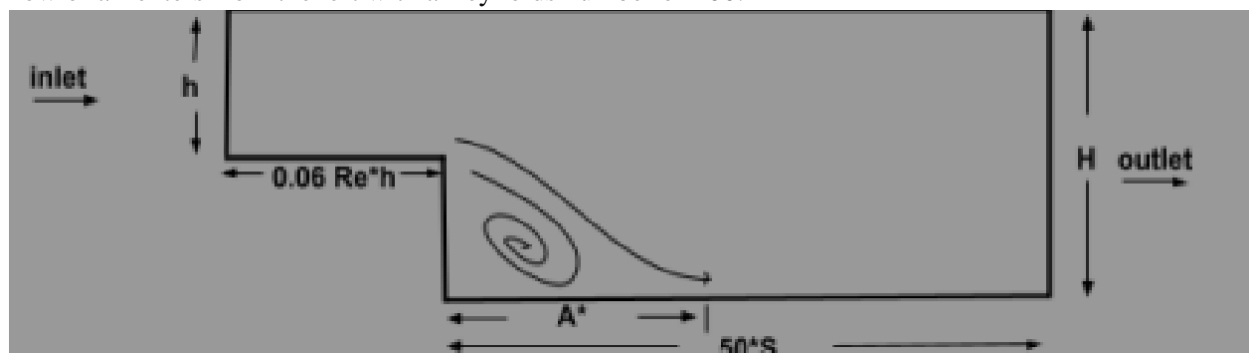


H = 30mm
H/h = 2 = Expansion ratio

S = 15 mm Re = 200

$\rho$ = 1.247 kg/$m^3$

$\nu$ = 1.76*10-5 $m^2$/s

A* = Reattachment length

**a)** Simulate the flow using a transient solver. Find the reattachment length for this phenomenon. Would this length be longer for turbulent flows? Explain your answer.

**b)** Plot the velocity profile ( on a line) along the cross-section at the reattachment point. Plot the velocity profile before and after the reattachment point. Explain the profiles obtained.

Explain the necessary theory, formula, calculations, OpenFOAM data, plots/contours and conclusion in the report. (Quality of the report is one of the criteria of evaluation)

# Methodology:

## Software Used:
- OpenFoam V11
- BlockMesh - Mesh Generation
- ParaFoam - postprocessing

## Mesh Generation:

For Mesh Generation, an open-source software Blockmesh was used. Blockmesh is a multi-block mesh generator software. The channel was divided into 3 blocks in the following manner:
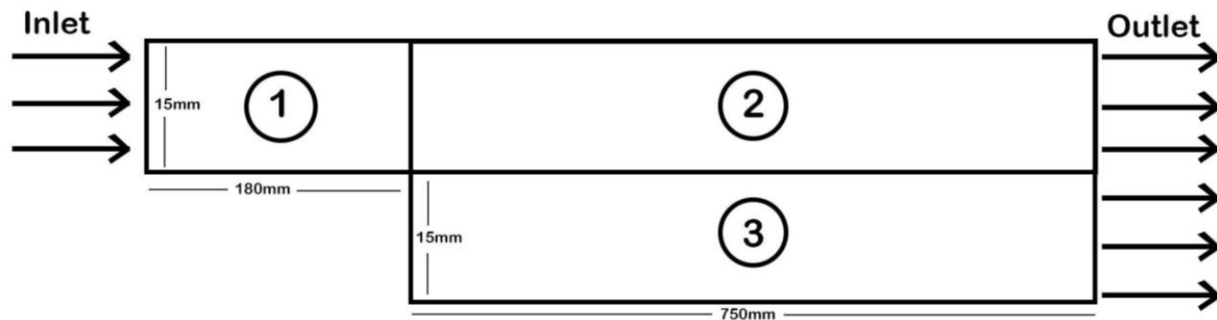


Fig1- Block Description

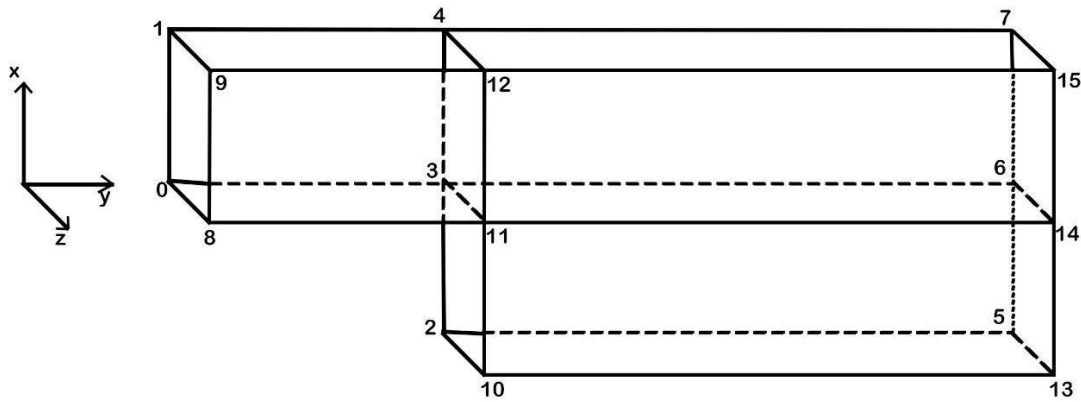Then, each vertices of the block was named in the following manner:



Fig- Vertices

Then with the origin at Vertex 0, each vertices were assigned a coordinate in the following manner:
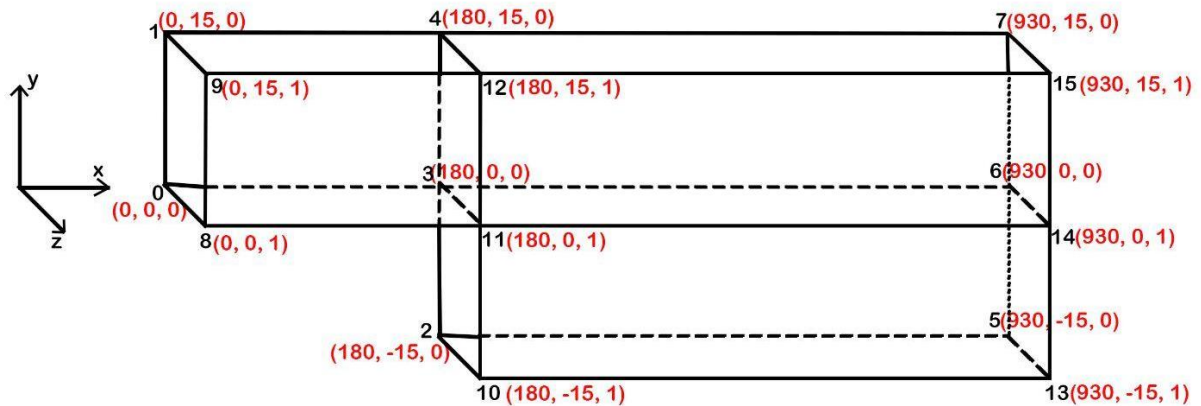


Fig- Coordinates of vertices

All this information was included in the "blockMeshDict.txt" file under the "system" sub-directory. A 3-dimensional mesh was generated because OpenFOAM always operates in a three-dimensional Cartesian coordinate system. It solves the case in 2D by specifying a special empty boundary condition on boundaries normal to the 3rd dimension (for which no solution is required).

For the grading of Mesh, "*simplegrading*" was used to prevent the complexity of calculation and to reduce the computation time, with the expansion ratio of 1 in all directions.

| Block | Grading | No. of elements in x-direction | No. of elements in y-direction | No. of elements in z-direction |
|-------|---------|-------------------------------|-------------------------------|-------------------------------|
| 1 | Simplegrading(1 1 1) | 50 | 30 | 1 |
| 2 | Simplegrading(1 1 1) | 300 | 30 | 1 |
| 3 | Simplegrading(1 1 1) | 300 | 30 | 1 |

After running "blockMesh" command, the mesh was generated following mesh was generated:
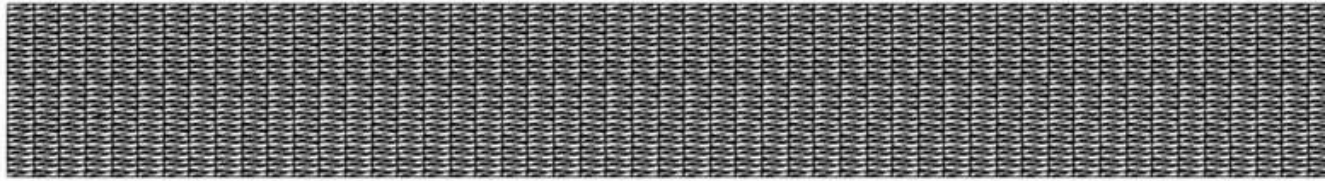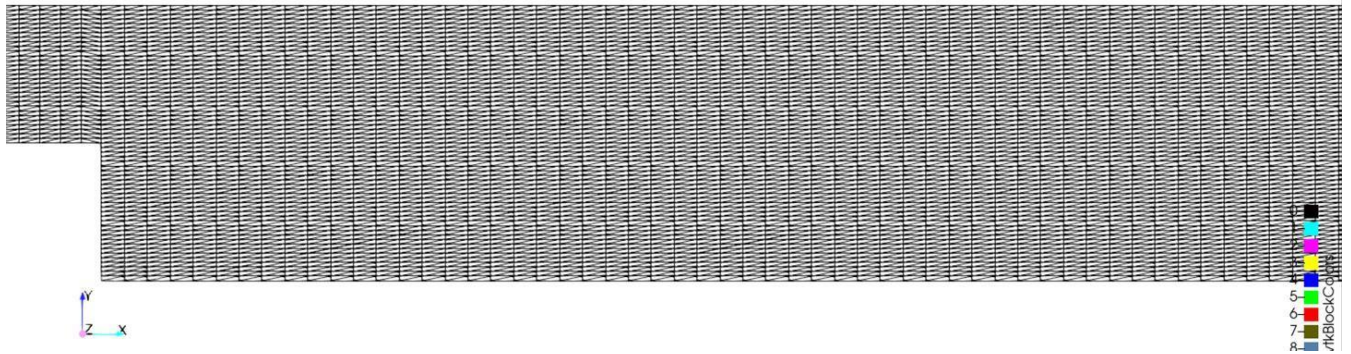


Fig- meshing of block 1



Fig- meshing of block 2 & 3

## Boundary and Initial condition:

After the generation of Mesh, the initial condition was specified in the 0 sub-directory. This sub-directory contains p and U, kinematic pressure (p) and velocity (U) fields respectively. The Boundary conditions were set in the following ways:

| patch | U | p |
|---|---|---|
| inlet | fixedValue (0.23 m/s) | zeroGradient |
| outlet | zeroGradient | fixedValue (0 ) |
| upperWall | noSlip | zeroGradient |
| lowerWall | noSlip | zeroGradient |
| Front and Back | empty | empty |

Fig: Boundary Condition

The inlet Velocity (U) was determined by using the following formula:

$$Re = \frac{d\,|U|}{\nu}$$

$$\text{Or, } U = \frac{d}{Re * \nu}$$

For, d= 15mm = 0.015m , Reynolds Number (Re) = 200 and Kinematic viscosity ($\nu$)= $1.76 * 10^{-5}\ m^2/s$, we get,

U = 0.23467 m/s

## Physical Properties:

Under the "*constant*" sub-directory, "*physicalProperties*" file contained the given information:

```
viscosityModel  constant;


nu         [0 2 -1 0 0 0 0] 1.76e-05;
rho              1.247;
```

Where, *nu* represents the kinetic viscosity of air, i.e. $1.76 \times 10^{-5}\ m^2/s$ and *rho* represents the density of air, i.e. 1.247 kg/$m^3$.

## Momentum Transport:

Under the "*constant*" sub-directory, "*momentumTransport*" file contained information about the type of simulation, in this case, the laminar type was chosen.

```
simulationType        laminar;
```

## Control:

Under the "*system*" sub-directory, "*controlDict*" file contained information about the control of time and reading/writing of solution data:

```
application     foamRun;
Solver          incompressibleFluid;
```

OpenFOAM version 11 has a general application"foamRun" which works for a very large field of solver. And under solver, "*incompressibleFluid*" is chosen. And laminar and turbulent modelling is now defined at the "*momentumTransport*" file, whereas the transient and steady-state solver is defined at "*fvschemes*".

For time-step calculation, the following formula was used:

$$Co = \frac{\delta t \, |U|}{\delta x}$$

Where,

$$\delta x = \frac{d}{n} = \frac{0.015}{50} = 0.0003$$

To achieve temporal accuracy and numerical stability, a Courant number of less than 1 is required, hence, keeping Co=1, we get,

$$\delta t = \frac{1 * 0.0003}{0.23467}$$
$$\delta t = 0.001 \text{ s}$$

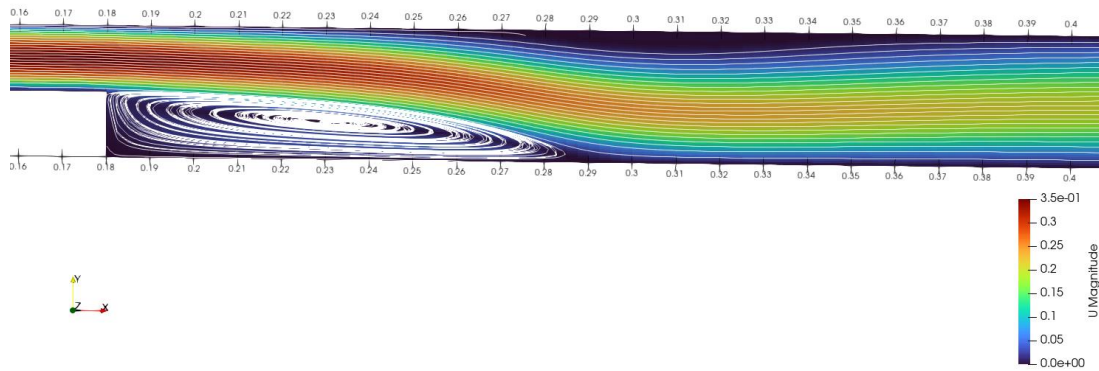Hence, the timestep was kept at 0.001 seconds.

## Post-processing:



Fig: Streamline and velocity profile

### Reattachment Length:

Reattachment length is the length at which the separated fluid re-attaches with the wall, in our case, the air separates at a distance of 180mm from the origin, and then again re-attaches to the wall at 290mm. Hence, the reattachment length was found to be 110mm.
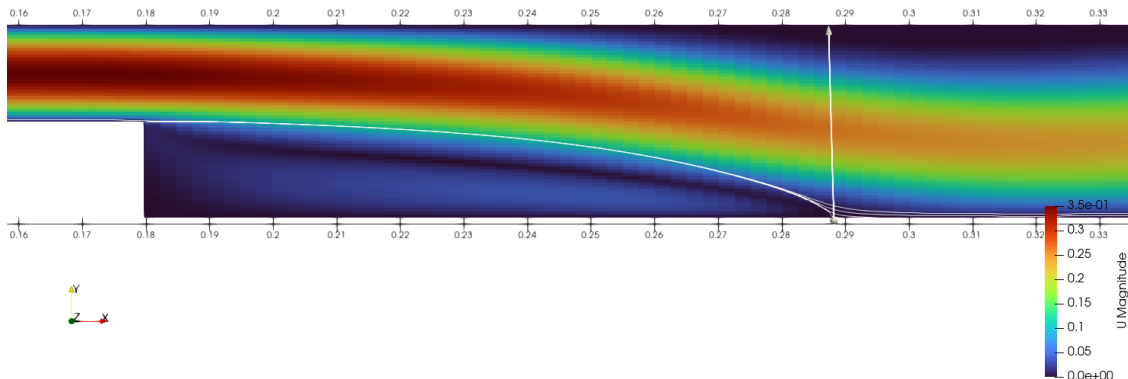
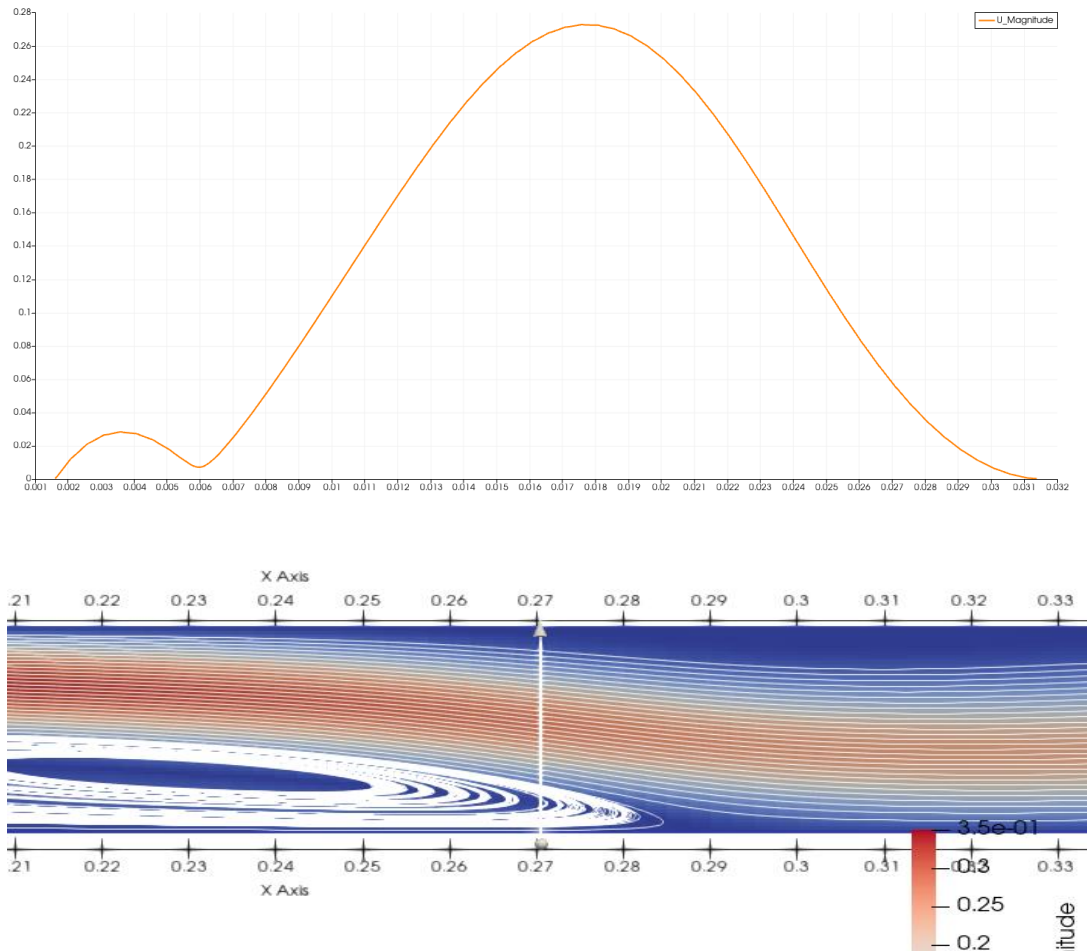Fig: Reattachment Length = 110mm

## Velocity Profile:

**Before Reattachment Length:**





Fig: U-plot before reattachment length (x=270mm)

At x=270mm, the velocity magnitude (U) was plotted across the cross-section and the above curve was obtained. The X-axis represents the length of the cross-section, whereas the Y-axis represents the magnitude of velocity.

In the initial part of the curve, there is instability in velocity as seen from the graph, this is because of the recirculation zone. The curve becomes stable only after crossing the height of 6mm. It reaches its maximum velocity of 0.275m/s at 18mm from the bottom.
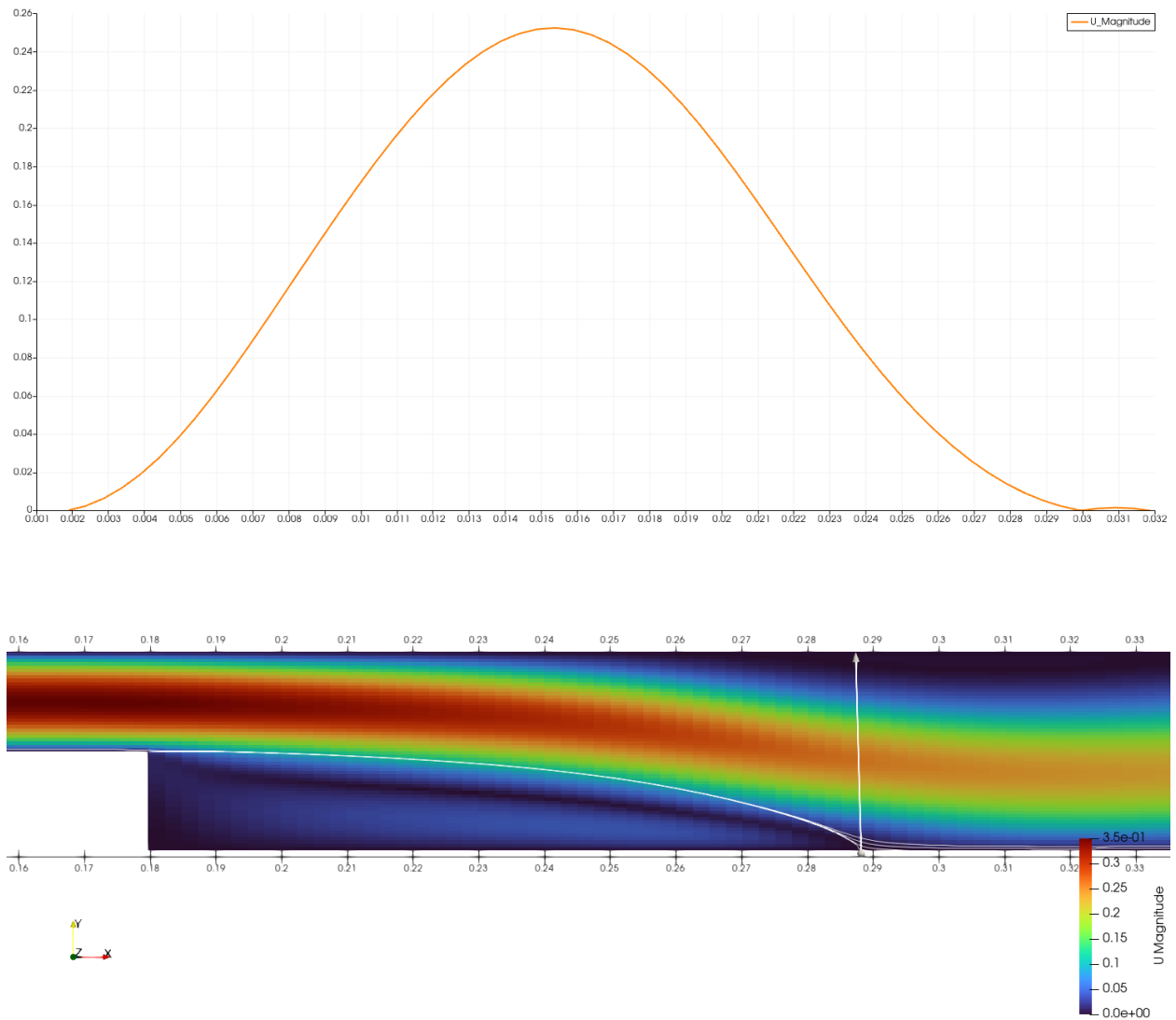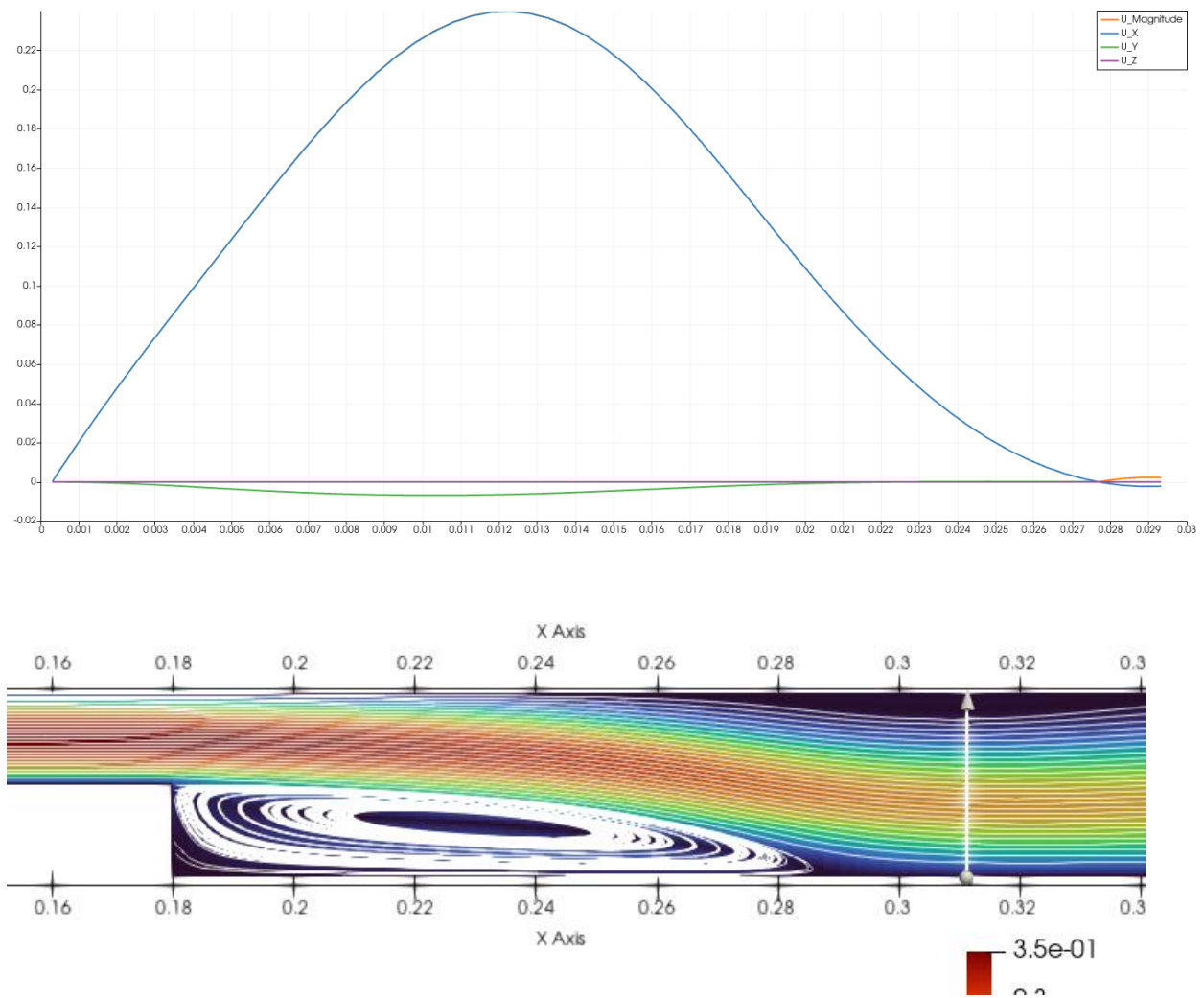
**At Reattachment Length:**





Fig: U-plot at reattachment length (x=290mm)

At x=290mm, the velocity magnitude (U) was plotted across the cross-section and the above curve was obtained. The X-axis represents the length of the cross-section, whereas the Y-axis represents the magnitude of velocity.

At the reattachment point, the velocity distribution across the cross-section is uniform. And there is no instability in the velocity profile.

It reaches its maximum velocity of 0.25m/s at 15mm from the bottom.

**After Reattachment Length:**



Fig: U-plot after reattachment point(x=310mm)

At x=310mm, the velocity magnitude (U) was plotted across the cross-section and the above curve was obtained. The X-axis represents the length of the cross-section, whereas the Y-axis represents the magnitude of velocity.

As seen in the curve, we can see that the velocity rises slowly in the upper part of the cross-section. This is because of another recirculation zone that is formed in the upper part of the channel at this cross-section. It reaches its maximum velocity of 0.24m/s at 12mm from the bottom.
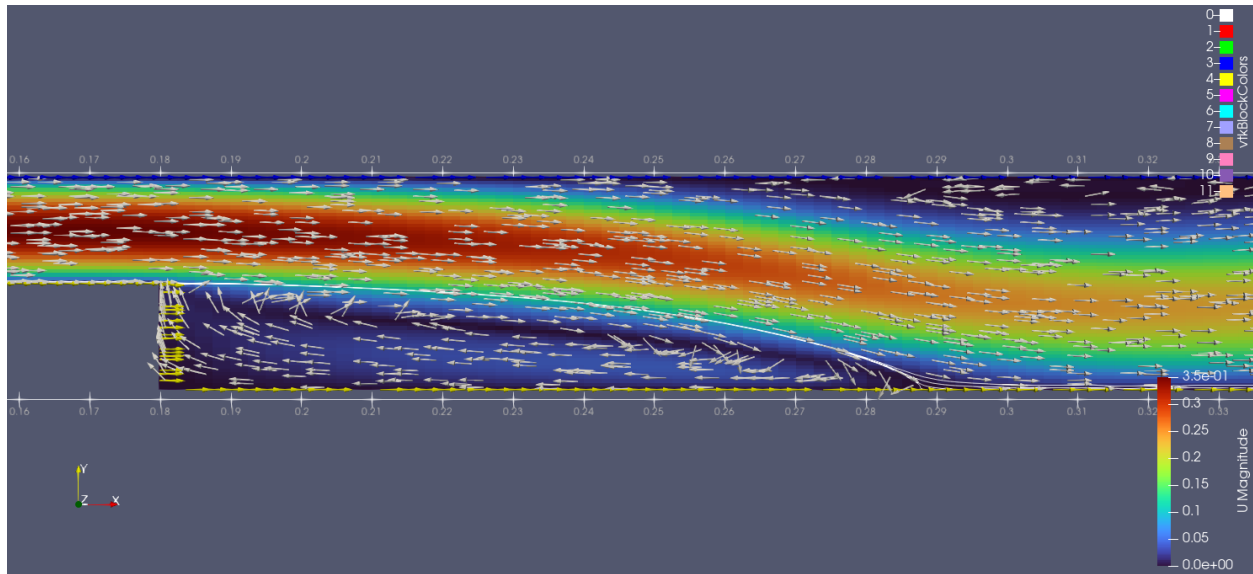
Fig: Glyph of Velocity

# Turbulence Modelling:

Turbulence modelling is part of general momentum transport which is concerned with models for the viscous stress in a fluid. For turbulence modelling, the following changes were made:

## Mesh Refinement:

The mesh size was made even more fine for more accurate modelling. An expansion ratio of 2 was used in the x-direction in blocks 2 and 3 to make the recirculating zone more finer.

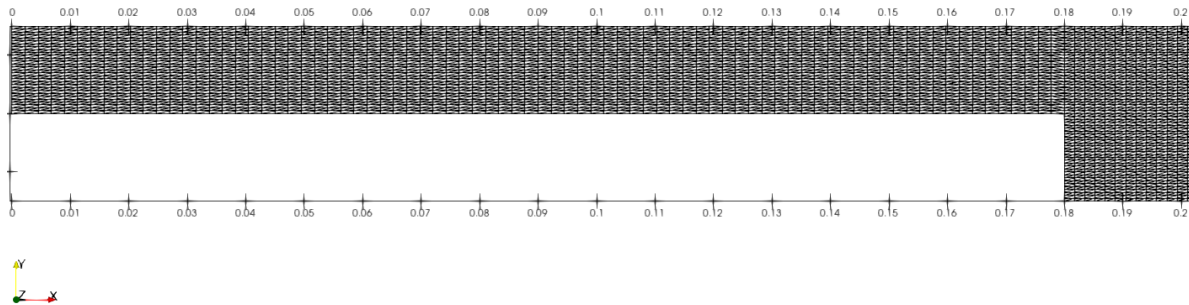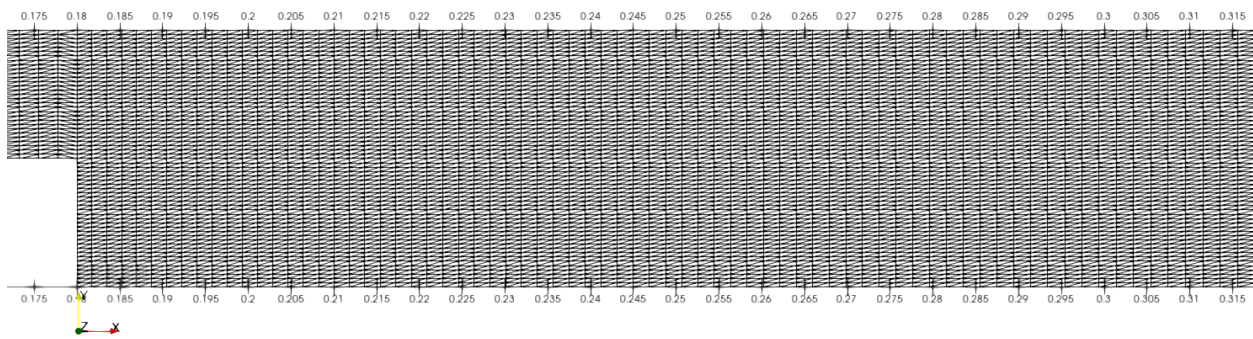| Block | Grading | No. of elements in x-direction | No. of elements in y-direction | No. of elements in z-direction |
|-------|---------|-------------------------------|-------------------------------|-------------------------------|
| 1 | Simplegrading(1 1 1) | 80 | 30 | 1 |
| 2 | Simplegrading(2 1 1) | 300 | 30 | 1 |
| 3 | Simplegrading(2 1 1) | 300 | 30 | 1 |

Fig: Meshing of block 1



Fig: Meshing of block 2 & 3

## Momentum Transport:

Under the "momentumTransport" file, instead of laminar simulationType, RAS(Reynolds Averaged Simulation) was chosen.

```
simulationType          RAS;

RAS
{
    model               kEpsilon;

    turbulence          on;

    printCoeffs                 on;
}
```

Under the RAS, kElipson turbulence model is chosen because it is a standard turbulence model for incompressible flows.

For this model, the value of k (turbulent kinetic energy) and (turbulent dissipation rate) needs to be given as the initial value under the "0" sub-directory.

These values were calculated based on the following formula:

$$k = \frac{3}{2}(|U| * I)^2$$

Where,
I = turbulent Intensity = U'rms/U = 5% = 0.05 (estimate)
U = inlet velocity = 0.23467 m/s

Therefore, we get,

$$k = 0.0002 m^2/s^2$$

Now, for the dissipation rate,

$$\varepsilon = C_\mu^{0.75} \frac{k^{1.5}}{l_m}$$

Where,
$C_\mu$ = 0.09
$l_m$ = Prandtl mixing length = 10% x step-height = 0.1 x 15mm = 0.0015m
k = 0.0002

Therefore, We get,

$$\varepsilon = 3.09 \, m^2/s^3$$

Putting these values in the folder *'k'* and *'epsilon'* under the 0 sub-directory, in both initial internalfield and inlet value to initialize the simulation.

## Post Processing:
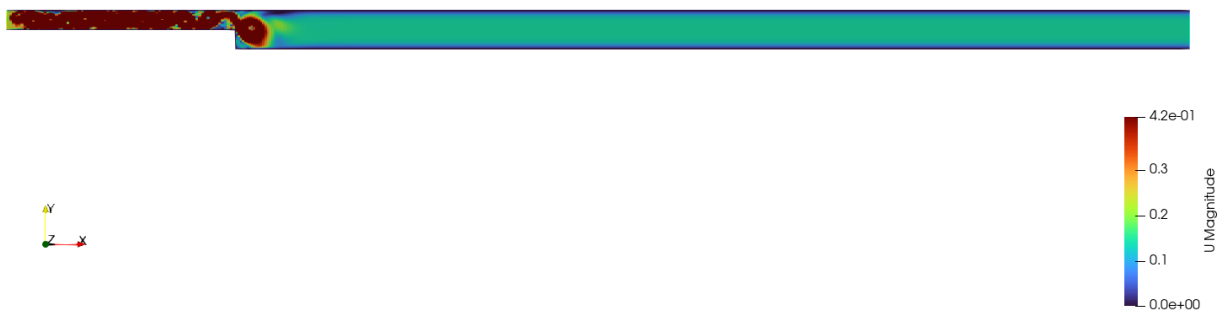


Fig: U profile at t=0.5sec
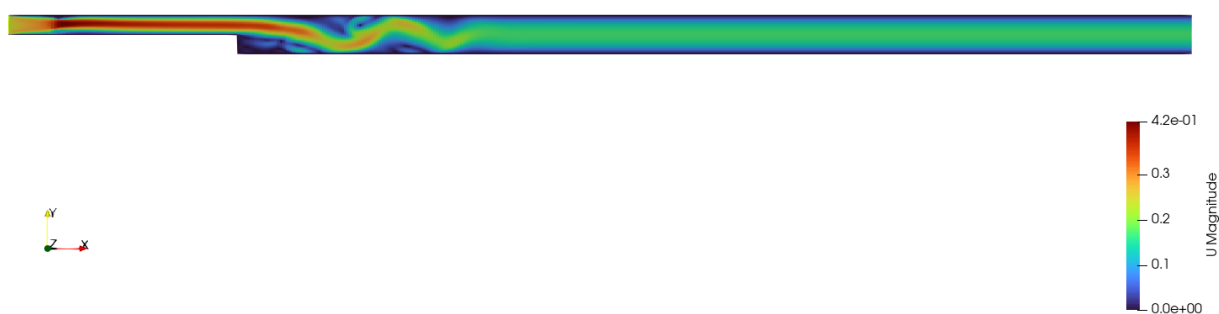
Fig: U profile at t=1sec



Fig: U profile at t=1.5sec



Fig: U profile at t=2sec
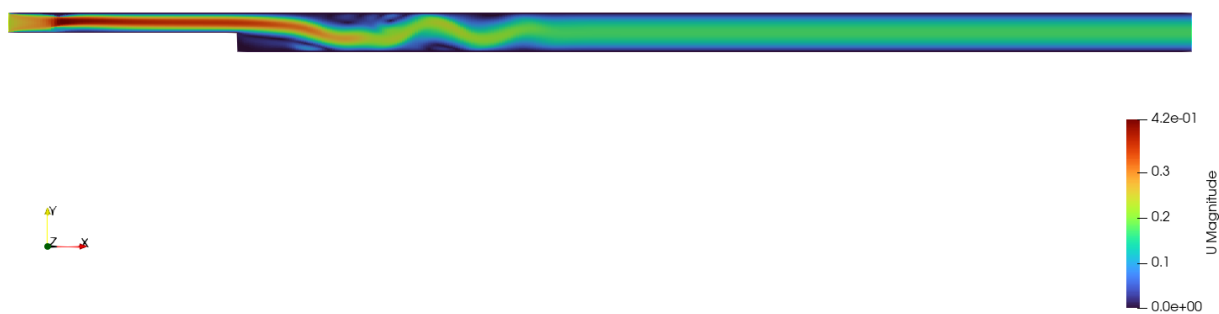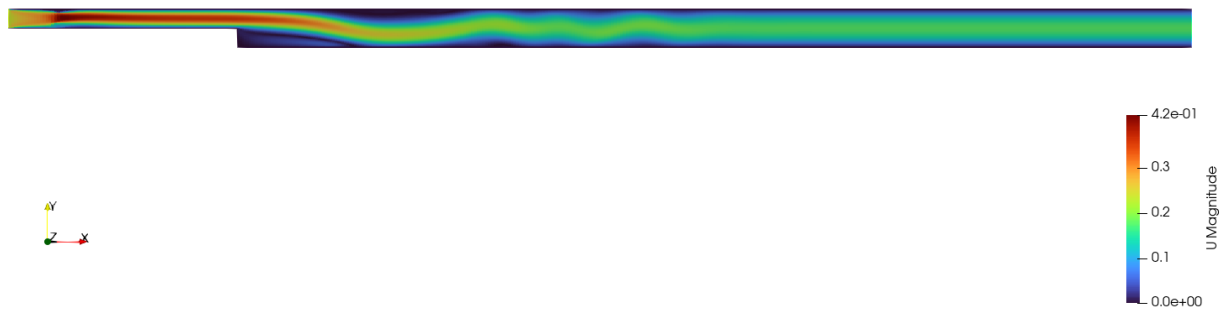


Fig: U profile at t=3sec
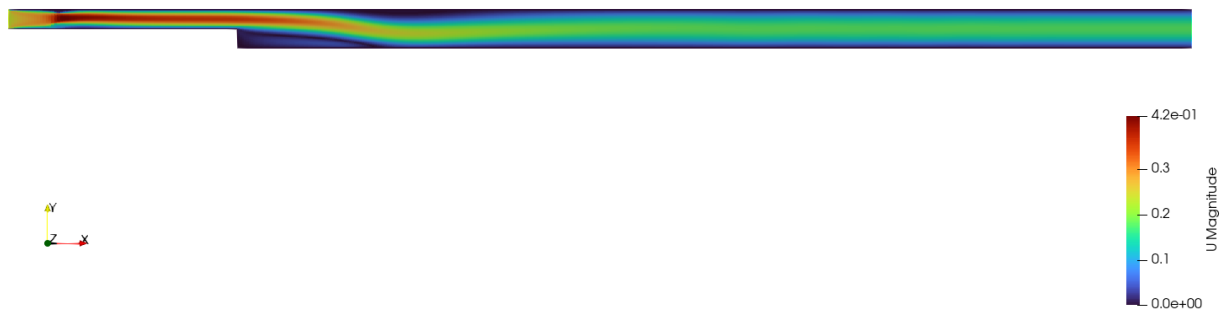
Fig: U profile at t=4sec



Fig: U profile at t=7sec



Fig: U profile at t=15sec

As seen in the above velocity profile, the flow becomes steady after 7 seconds, before that the flow was unsteady and turbulent.

## Reattachment Length:

In turbulence modelling, it was found that the reattachment length has gotten longer. The fluid reattaches with the wall at a distance of 302mm from the origin after separating from a distance of 180mm. This means the reattachment length was 122mm which is longer than the laminar model's 110mm.
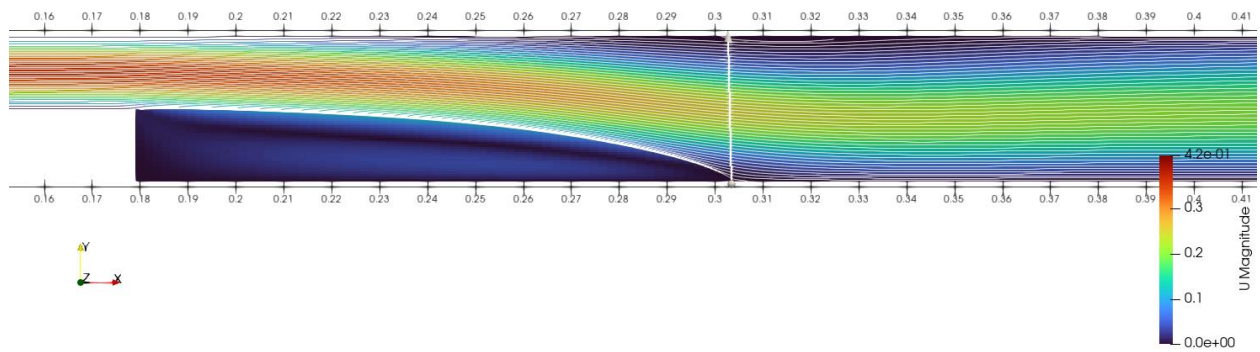
Fig- Reattachment Length

## Velocity Profile:

**Before Reattachment Length:**
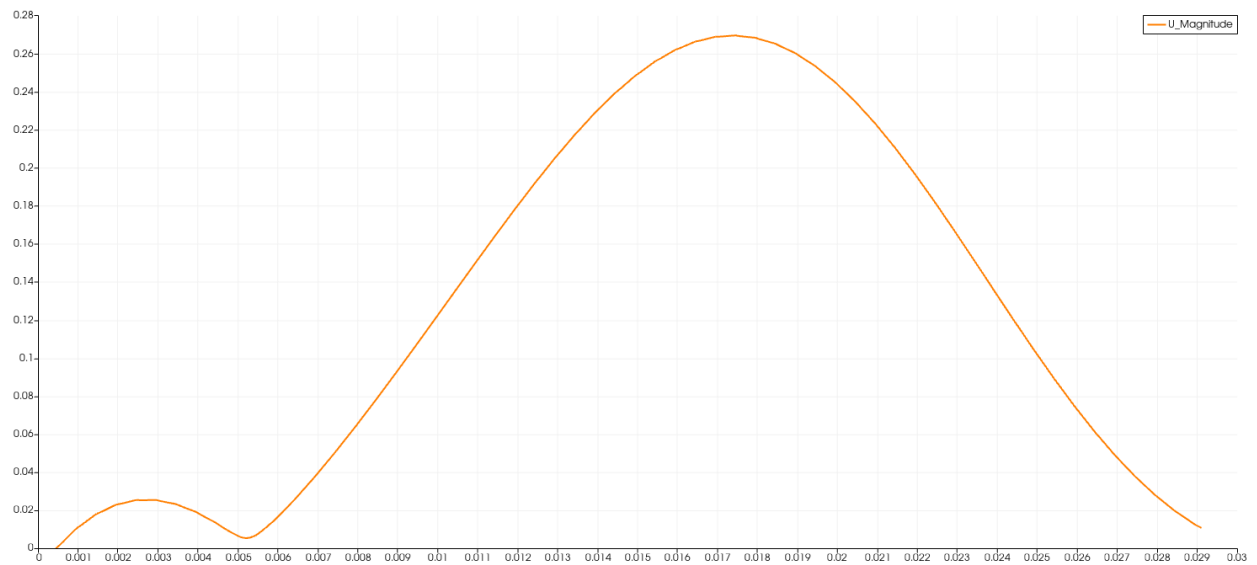
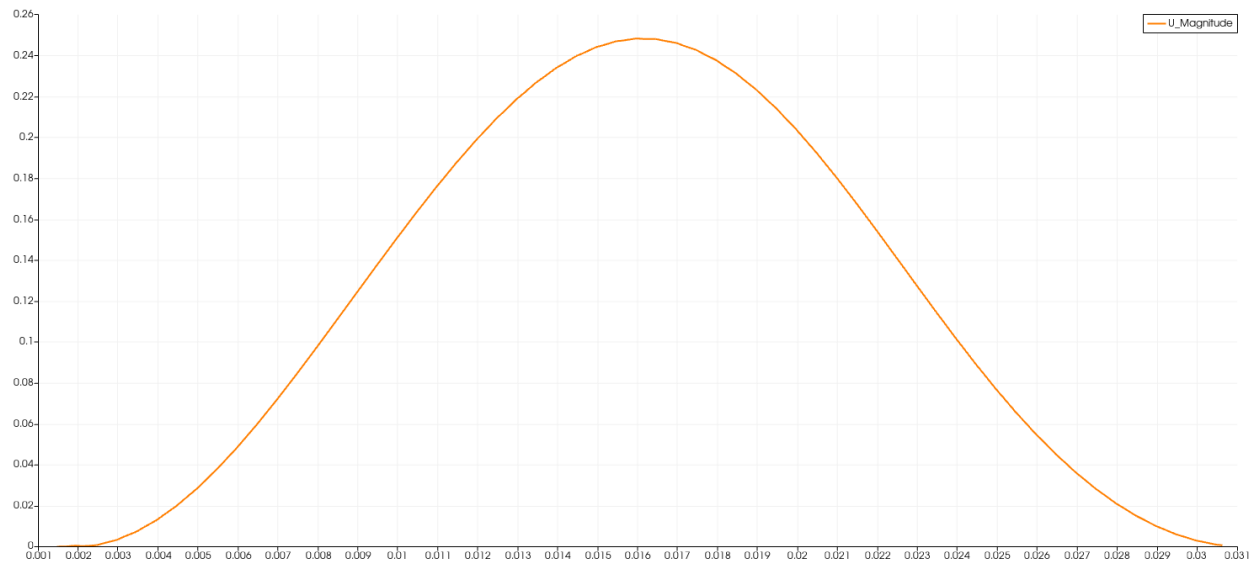

Fig- U-plot before reattachment length(x=280mm)

At x=280mm, the velocity magnitude (U) was plotted across the cross-section and the above curve was obtained. The X-axis represents the length of the cross-section, whereas the Y-axis represents the magnitude of velocity.

In the initial part of the curve, there is instability in velocity as seen from the graph, this is because of the recirculation zone. The curve becomes stable only after crossing the height of 5.5mm. It reaches its maximum velocity of 0.27m/s at 17.5mm from the bottom.

**At Reattachment Length:**



Fig- U-plot before reattachment length(x=300mm)

At x=20mm, the velocity magnitude (U) was plotted across the cross-section and the above curve was obtained. The X-axis represents the length of the cross-section, whereas the Y-axis represents the magnitude of velocity.

At the reattachment point, the velocity distribution across the cross-section is uniform. And there is no instability in the velocity profile.

It reaches its maximum velocity of 0.25m/s at 16mm from the bottom.

**After Reattachment Length:**
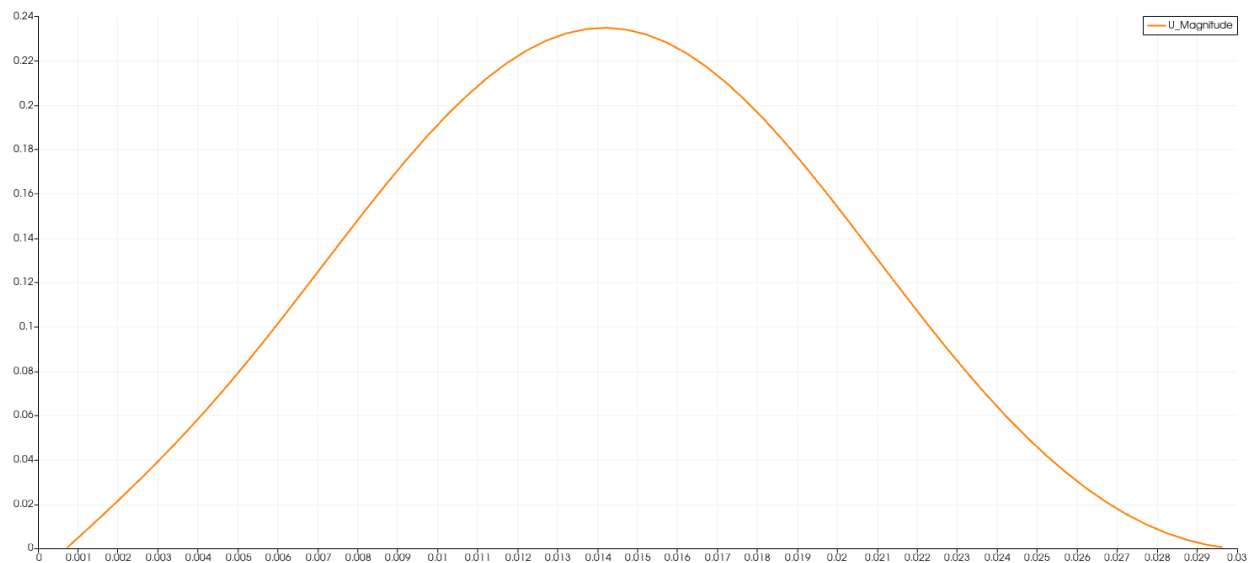


Fig: U-plot before reattachment length(x=310mm)

At x=310mm, the velocity magnitude (U) was plotted across the cross-section and the above curve was obtained. The X-axis represents the length of the cross-section, whereas the Y-axis represents the magnitude of velocity.

As seen in the curve, we can see that the velocity rises slowly in the upper part of the cross-section. This is because of another recirculation zone that is formed in the upper part of the channel at this cross-section. It reaches its maximum velocity of 0.24m/s at 14mm from the bottom.
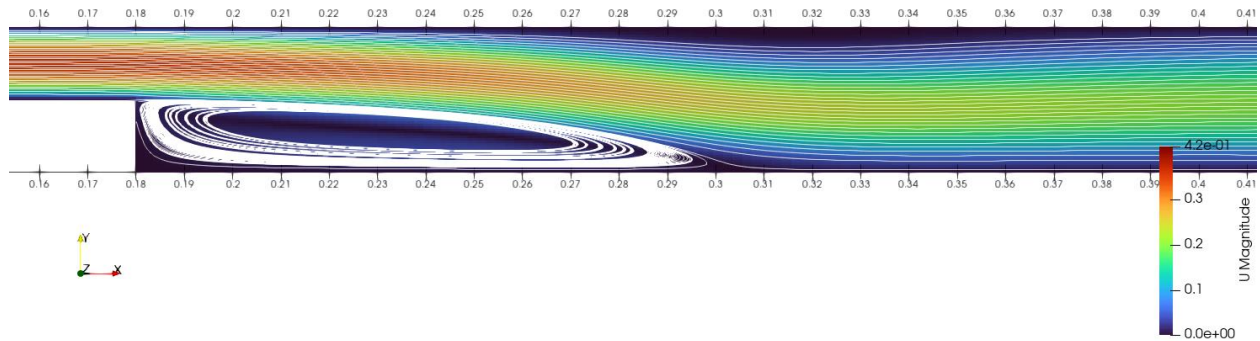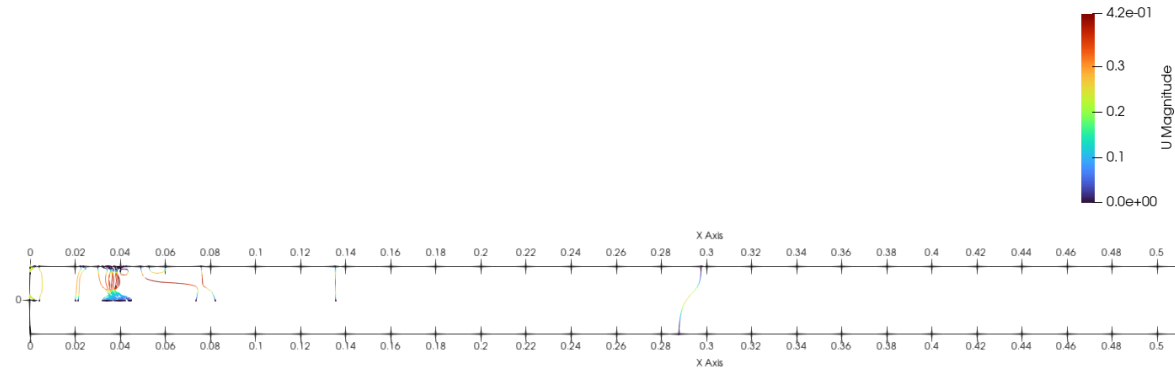
**Streamline:**



Fig:  Streamline flow

**Contour:**



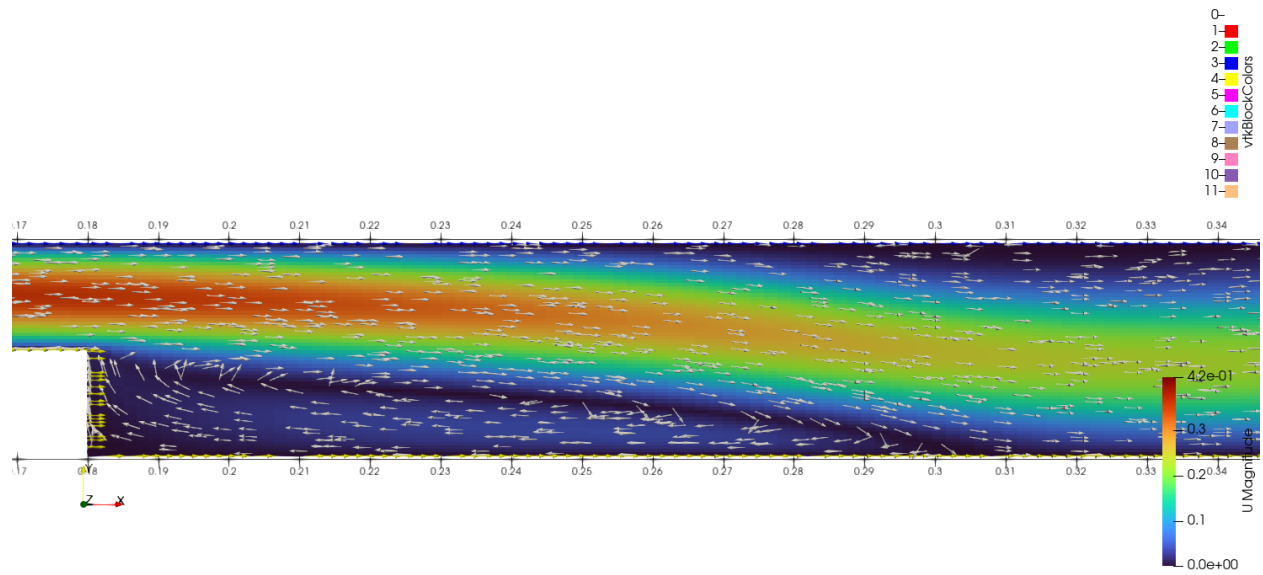Fig: Contour

**Glyph:**



Fig: Glyph

# Result and Conclusion:

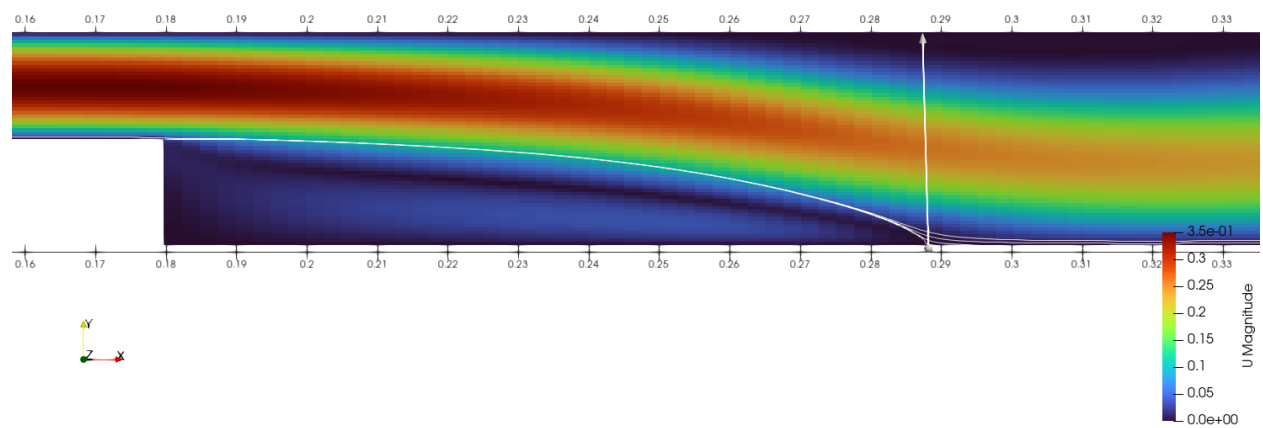a)  Comparison of Reattachment length between laminar and turbulent flow:
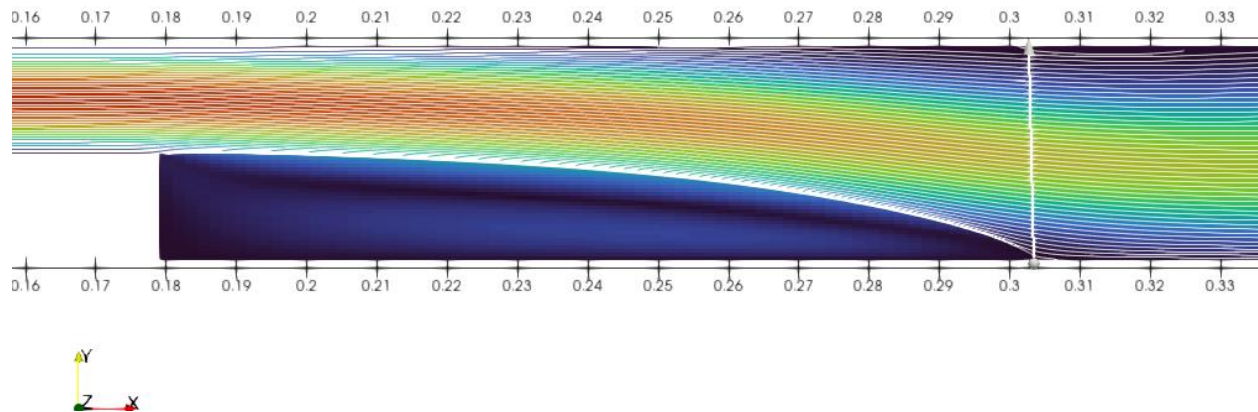


Fig: Laminar flow

Fig: Turbulent flow

Hence, it is concluded that the reattachment length is longer for turbulent flow. As, the reattachment length for turbulent flow was found to be 122mm and for laminar flow was found to be 110mm. The velocity distribution at the reattachment point is more uniform and stable than before and after the reattachment point.

# References:

1. https://doc.cfd.direct/openfoam/user-guide-v11/index
2. https://www.openfoam.com/documentation/user-guide
3. https://www.sciencedirect.com/science/article/abs/pii/S2451904918300167#:~:text=Backward%2DFacing%20Step%20(BFS)%20flow%20is%20one%20representative%20model,the%20flow%20around%20buildings%2C%20etc.

# Appendix:

1. Reynolds Average Simulation:
   Notes on CFD: General Principles - 6.9 Reynolds-averaged simulation
2. KElipson turbulence model:

   The correlations of this model are [11]:

   $$U_i \frac{\partial k}{\partial x_i} - \frac{\partial}{\partial x_i}\left[\frac{v_T}{\sigma_k} \frac{\partial k}{\partial x_i}\right] = P - \varepsilon, \tag{3}$$

   $$U_j \frac{\partial \varepsilon}{\partial x_j} - \frac{\partial}{\partial x_i}\left[\frac{v_T}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i}\right] = \frac{\varepsilon}{k}(C_{\varepsilon 1} P - C_{\varepsilon 2} \varepsilon), \tag{4}$$

   $$v_T = C_\mu \frac{k^2}{\varepsilon}, \tag{5}$$

   $$P_k = -\overline{u_i' u_j'}\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right), \tag{6}$$

   $$\overline{u_i' u_j'} = \frac{2}{3}\delta_{ij}k - v_T\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) \tag{7}$$

   where $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1$ and $\sigma_\varepsilon = 1.3$ are the constants of the model and $P_k$ signifies the production rate of the turbulence kinetic energy. $v_T$ is the Boussinesq eddy viscosity, while the turbulence Reynolds stress tensor is calculated with the help of the generalized Boussinesq hypothesis (Eq. (5)).

3. Navier-Stokes Equation:
   https://en.wikipedia.org/wiki/Navier%E2%80%93Stokes_equations