

Wind analysis of tall buildings using OpenFOAM CFD

Ratish Dixit

Bachelors of Technology student, School of Mechanical and Building Sciences, VIT University, Chennai-600127, Tamil Nadu, India (email: ratish.dixit2016@vitstudent.ac.in)

Abstract

Urbanization along with the rapid increase in demand for the house or any other structural needs with the decreased land availability to comply with such needs, the need for the construction of tall slender buildings often referred to as skyscrapers have drastically escalated. Therefore, for the sake of safe construction and durability of the building, it makes necessary for the designers to study all the affects of air and pressure on the buildings before the project begins. Therefore, an attempt is made in this project to study the effect of neighbouring tall buildings. An isolated rectangular building model has been considered as the based model in the analysis. An atmospheric boundary layer based wind profile has been adopted using a user defined function in the simulations.

Key Words: Wind analysis, Openfoam, CFD, Skyscrapers, Tall buildings

1. INTRODUCTION

Computational Fluid Dynamics (CFD) has found its application in the plethora of engineering applications in the recent times. From aerospace applications to the designing of optimum cooling systems for computers, it has a widespread usage.

However, very limited work has been done on the application of CFD in simulating the wind behaviour around tall structures. The CFD of tall buildings is only performed in complementary to the tunnel test. The level of CFD in this aspect still has not been achieved. Skyscraper is a term given to the tall slender buildings over 600 meters. Urbanization along with the rapid increase in demand for the house or any other structural needs with the decreased land availability to comply with such needs, the need for the construction of tall slender buildings often referred to as skyscrapers have drastically escalated.

Therefore, an attempt is made in this case study to observe and examine the effect of wind flow around buildings which will make a new step in the future of construction of tall buildings and their effective analysis.

2. METHODOLOGY

The case study was initiated with a scientific literature survey using the research papers available in the similar field.

The next step was to decide the dimensions of the domain and main building. Further it was decided to study the wind flow pattern around the building in different orientations.

The next step was to design the buildings in the domain. This was completed in the *Salome* software.

Next, meshing was done using the commands in the *Blue CFD core terminal*. The quality of meshing was verified using the *Paraview* software.

After running the case study in *Blue CFD Core terminal*, post processing was performed in the *Paraview* software.

3. GEOMETRY

The first step is the design of the domain where our building will be situated for the analysis. The domain was a large size intentionally to ignore any wind effects with the surface of the walls to render the real analysis erroneous.

Dimensions of domain are as follows:

$$\text{In } x - \text{axis} = 1500 \text{ meters}$$

$$\text{In } y - \text{axis} = 3400 \text{ meters}$$

$$\text{In } z - \text{axis} = 1200 \text{ meters}$$

Dimensions of the buildings are as follows:

Small building (Height = H/2)

$$\text{In } x - \text{axis} = 150 \text{ meters}$$

$$\text{In } y - \text{axis} = 100 \text{ meters}$$

$$\text{In } z - \text{axis} = 300 \text{ meters}$$

Big building (Height = H)

$$\text{In } x - \text{axis} = 150 \text{ meters}$$

$$\text{In } y - \text{axis} = 100 \text{ meters}$$

$$\text{In } z - \text{axis} = 600 \text{ meters}$$

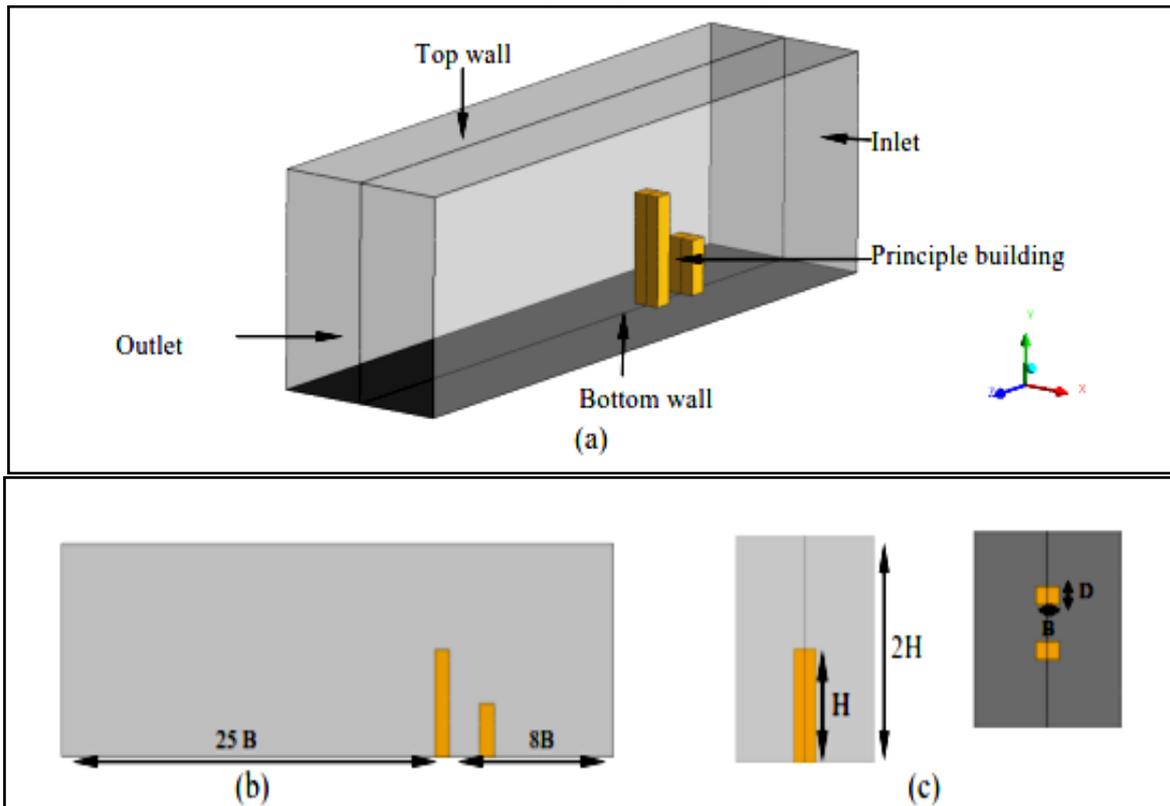


Figure 1: Geometry common to all cases

4. CASE SET UP

CASES	PARTICULARS
1	Isolated building
2	Two building of same height in the vicinity
3	A building of height $H/2$ upwind of a building of height H
4	A building of height $H/2$ downwind of a building of height H

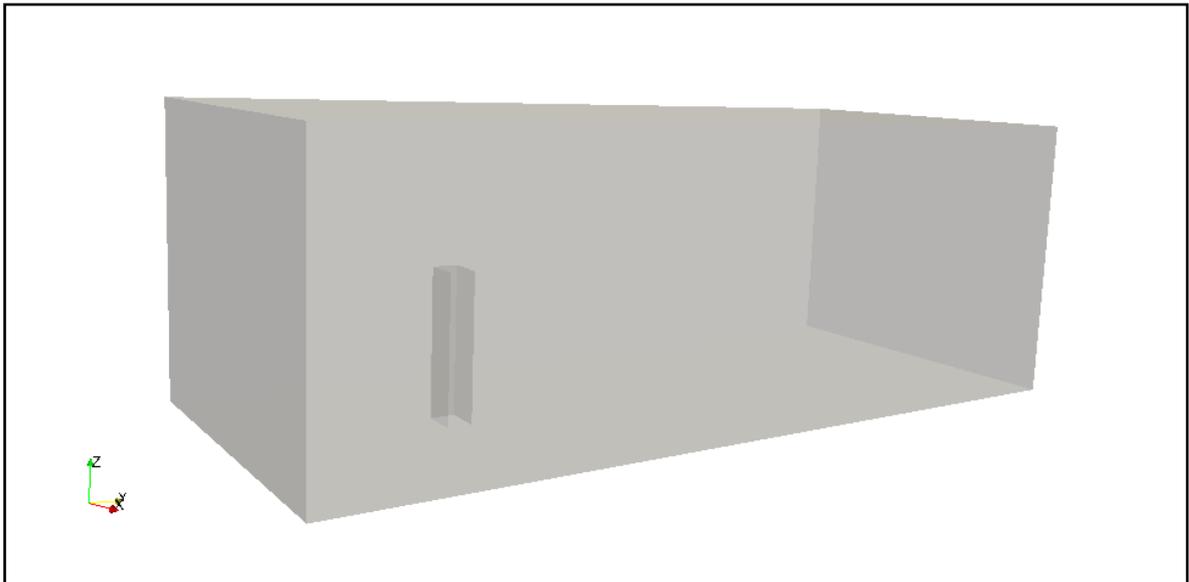


Figure 2: Case 1 geometry

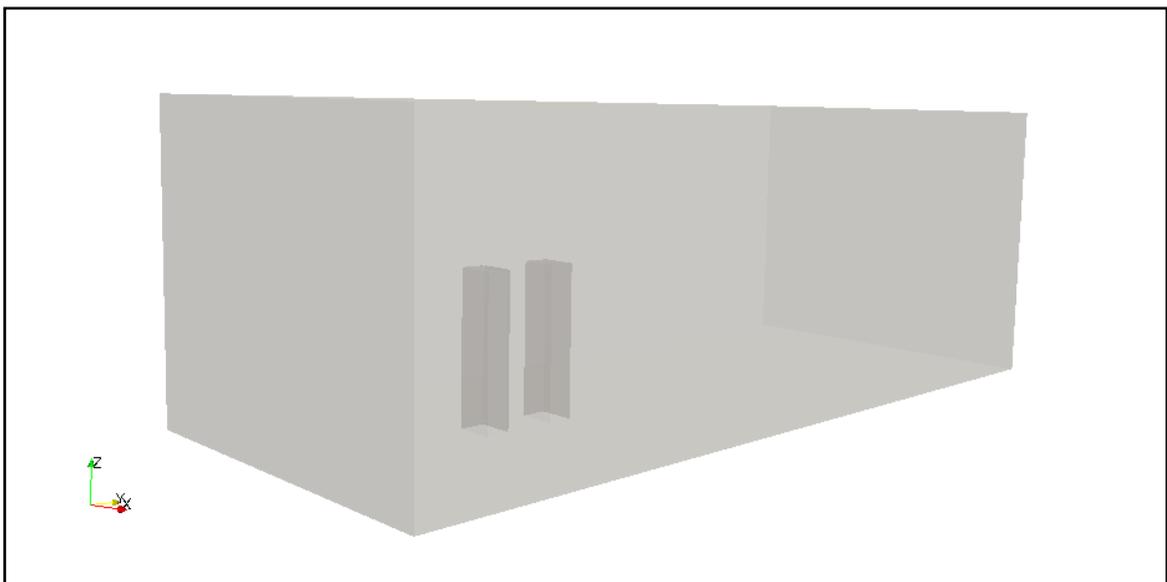


Figure 3: Case 2 geometry

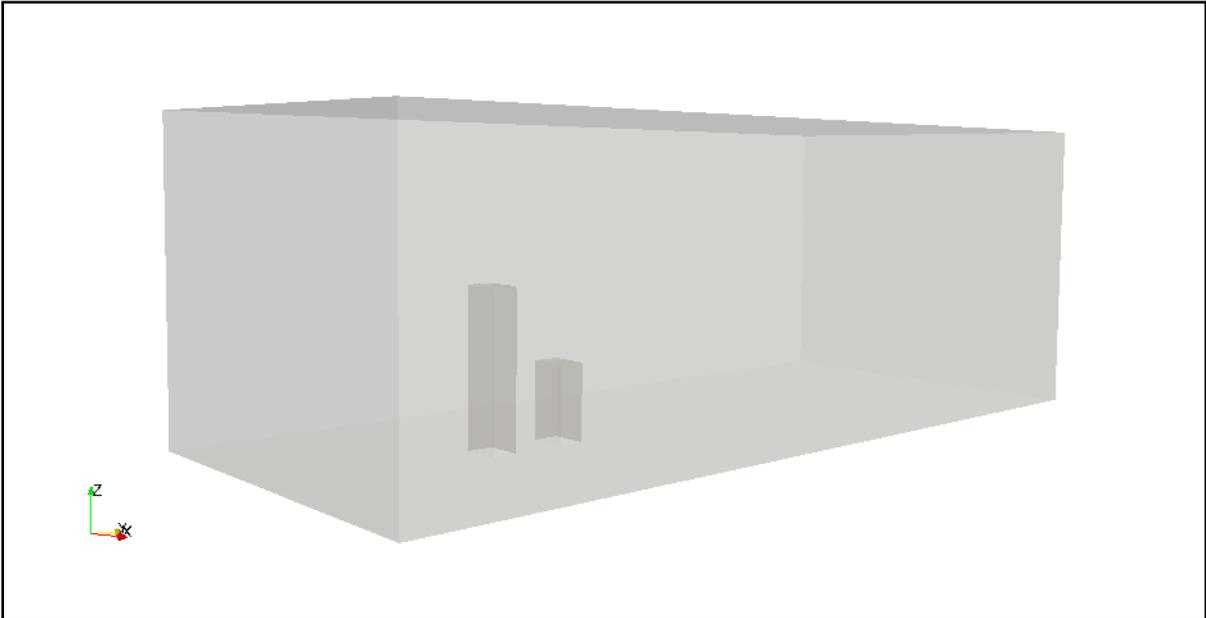


Figure 4: Case 3 geometry

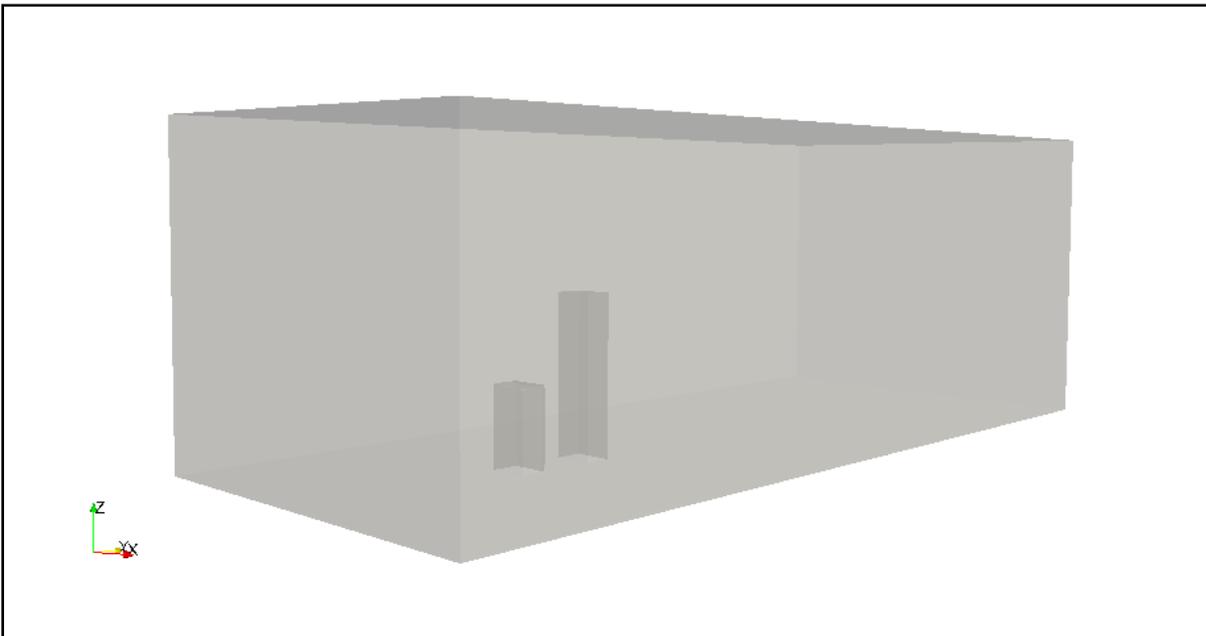


Figure 5: Case 4 geometry

5. MESHING IN OPENFOAM

Snappy Hex Mesh was generated for the case study. This mesh gave a better result than the other types of mesh.

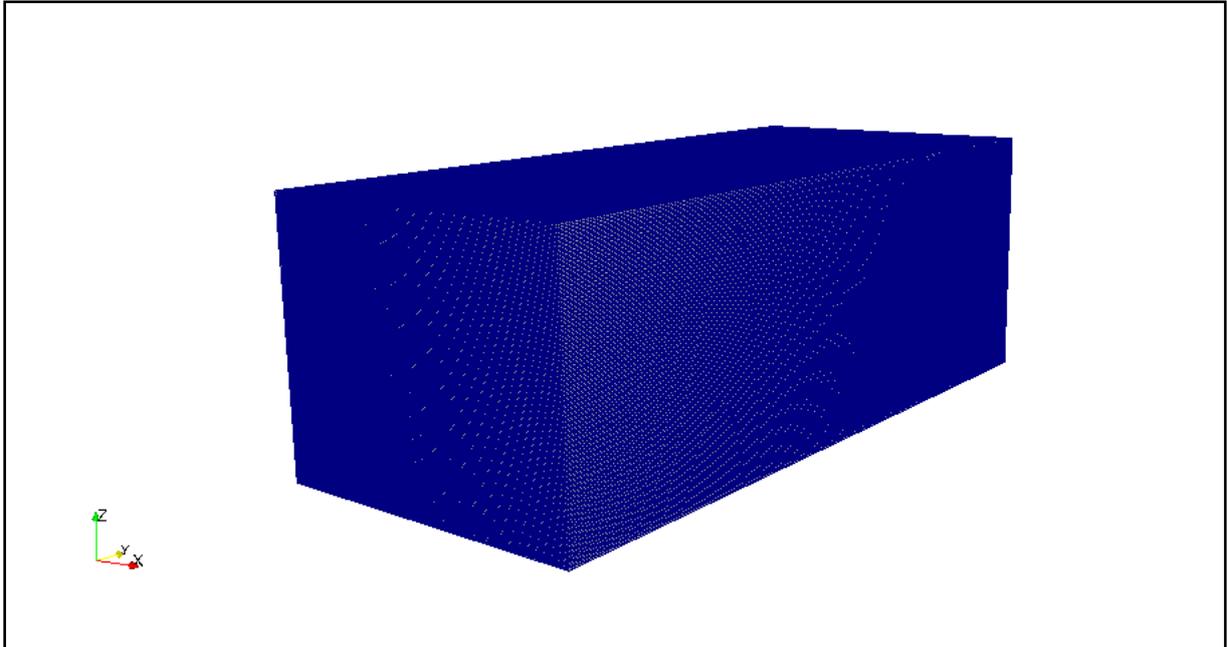


Figure 6: Snappy Hex Mesh

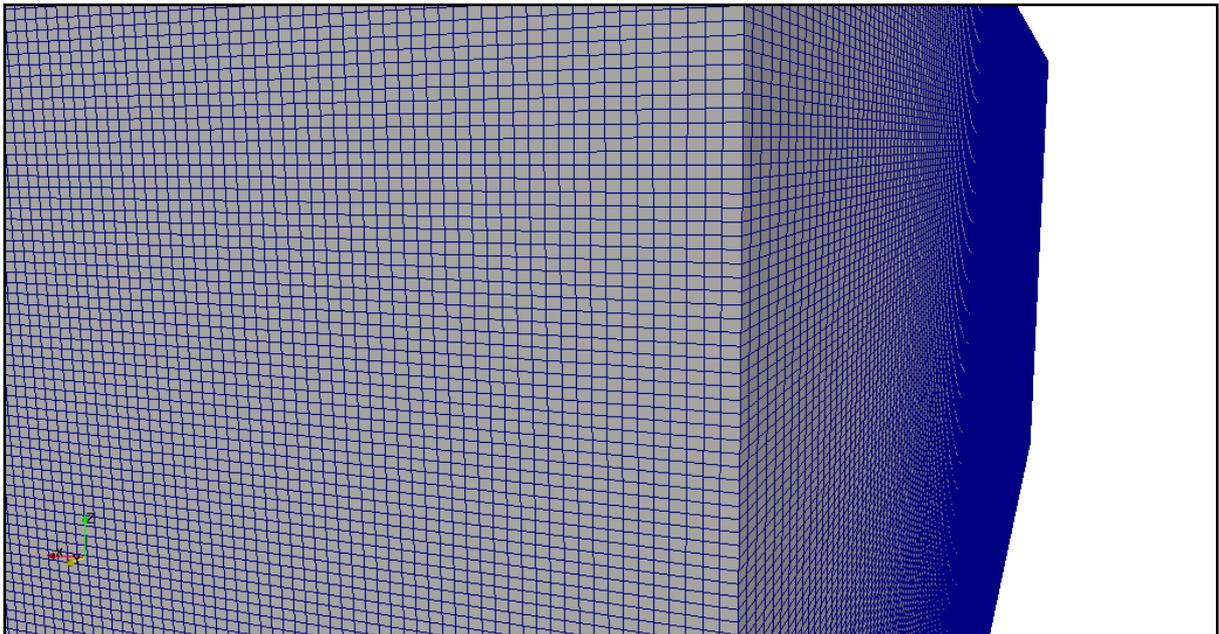


Figure 7: Enlarged view of the mesh

6. SOLVER USED IN THE CASE

The following solver was used to solve the case study with the following values of required quantities:

RAS – Model: k – epsilon turbulence model

Solver Type: Simple Foam

Inlet Velocity: $40 \frac{m}{s}$

Epsilon: 2.11

K: 24

Kinematic viscosity: $1.5 \times 10^{-5} \frac{m^2}{s}$

The formulas used to calculate these values are:

$$R_e = \frac{vL}{\vartheta}$$

$$k = \frac{3}{2} UI^2$$

$$\omega = \frac{\sqrt{k}}{L}$$

$$\epsilon = \frac{C_\mu^{0.75} \times k^{1.5}}{L}$$

$$C_\mu = 0.09$$

$$L = 0.16 R_e^{-1/8}$$

$$L = 0.038 D_h$$

$$D_h = \frac{4A}{P} = \frac{2xy}{x+y}$$

Where

R_e is reynolds number

v is inlet velocity

L is the characteristic length

k is the turbulent kinetic energy

ε is the turbulent dissipation rate

ω is the specification rate

ϑ is the kinematic viscosity

D_h is the hydraulic diameter

7. RESULTS

CASE 1

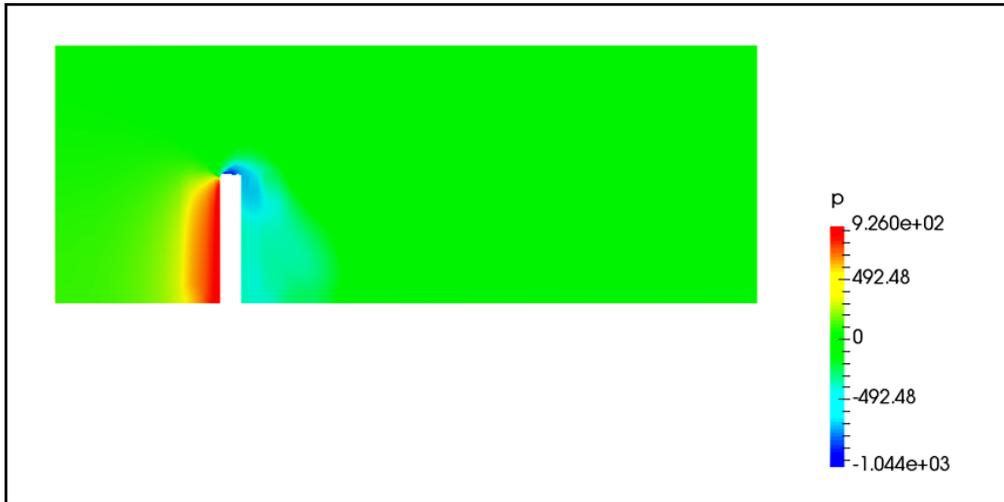


Figure 8: Pressure at mid plane

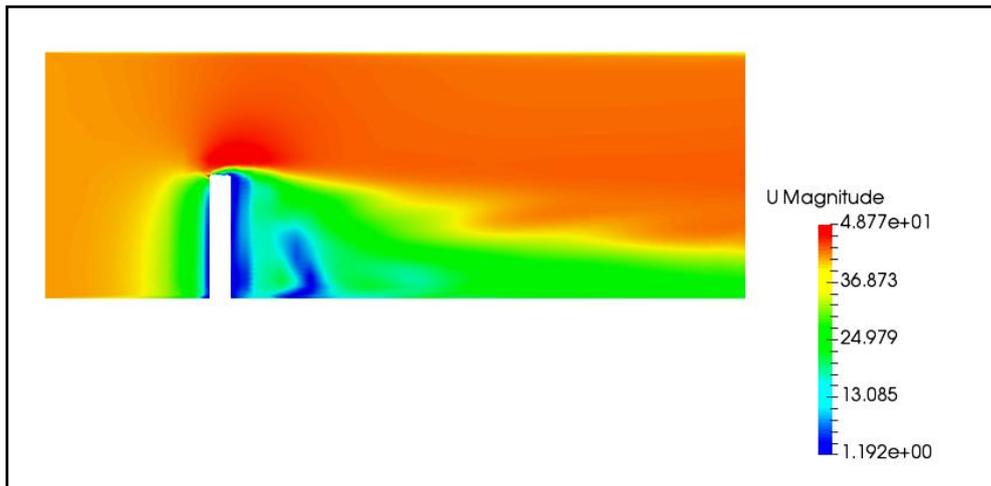


Figure 9: Velocity at mid plane

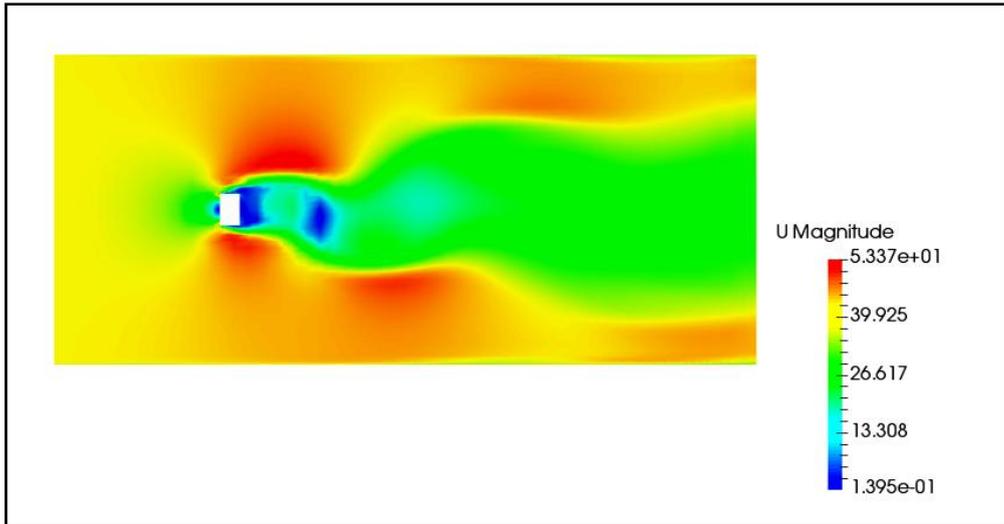


Figure 10: Velocity at 100 m height

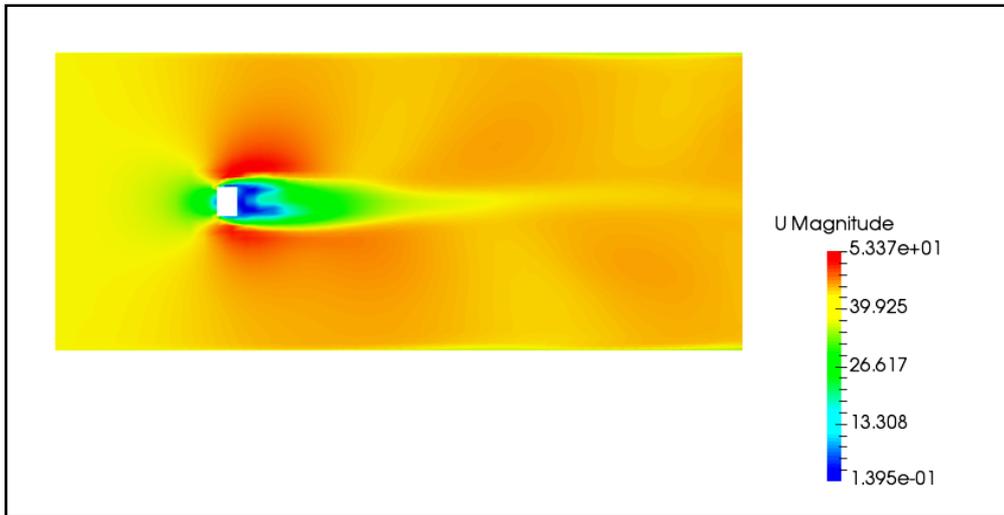


Figure 11: Velocity at 500 m height

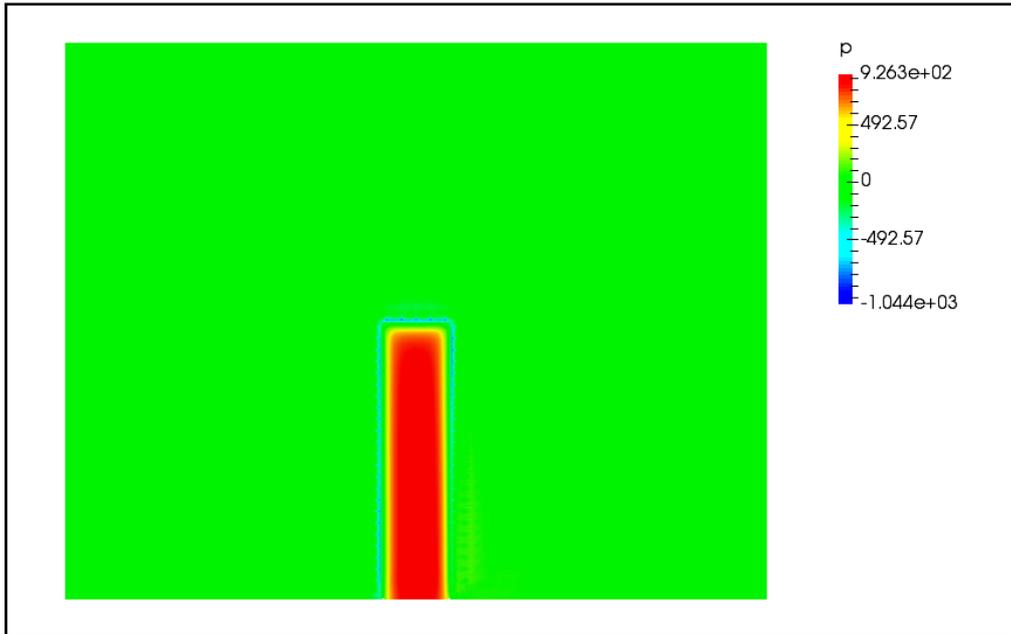


Figure 12: Pressure at windward side

CASE 2

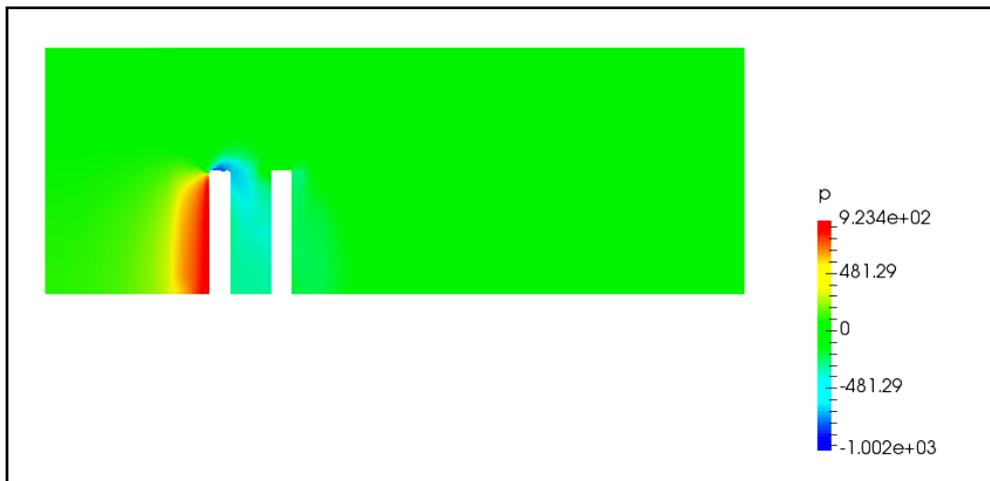


Figure 13: Pressure at mid plane

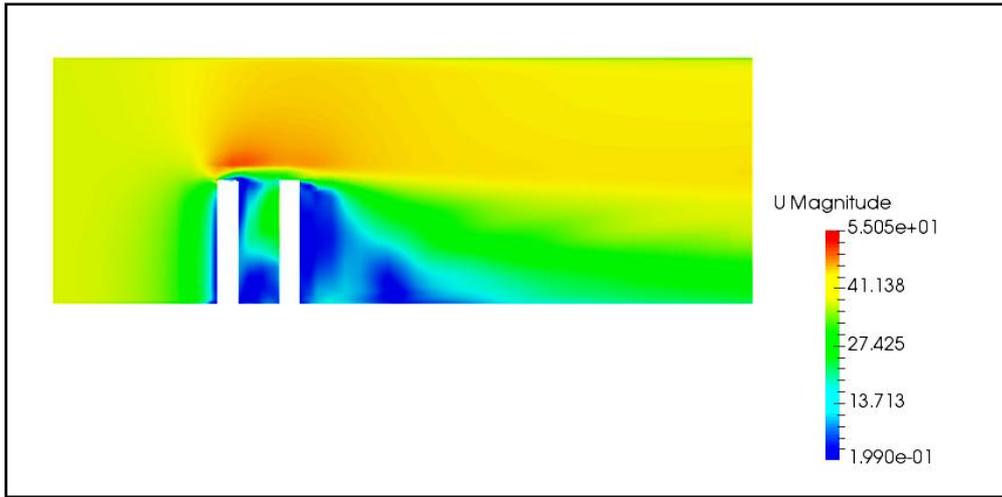


Figure 14: Velocity at mid plane

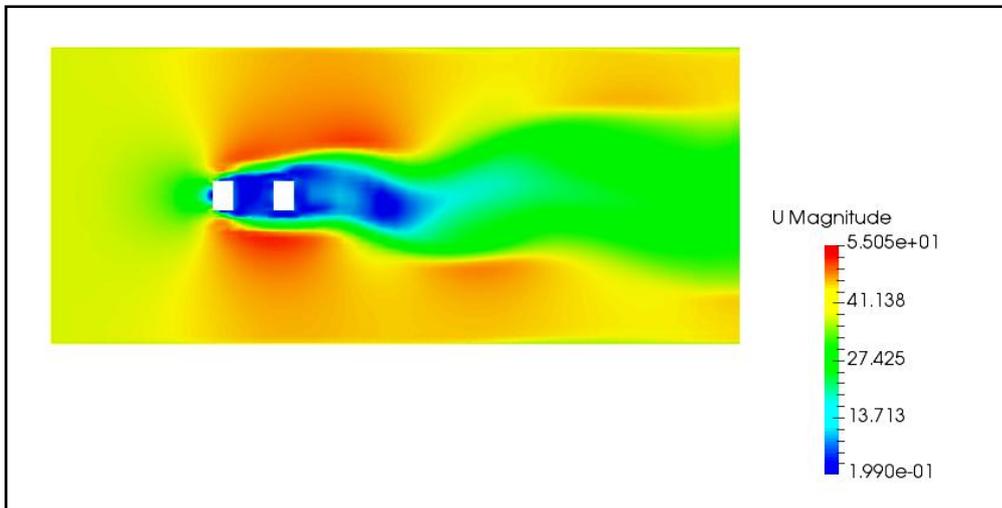


Figure 15: Velocity at 100 m height

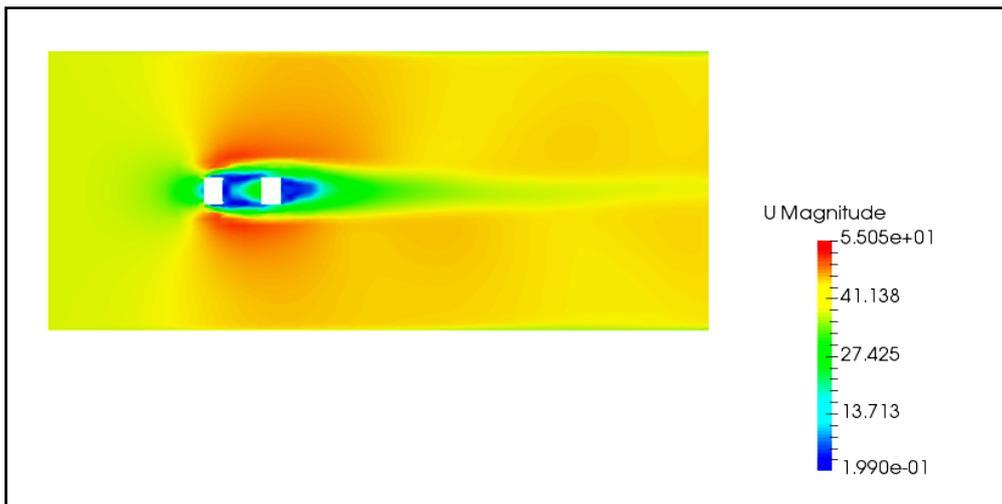


Figure 16: Velocity at 500 m height

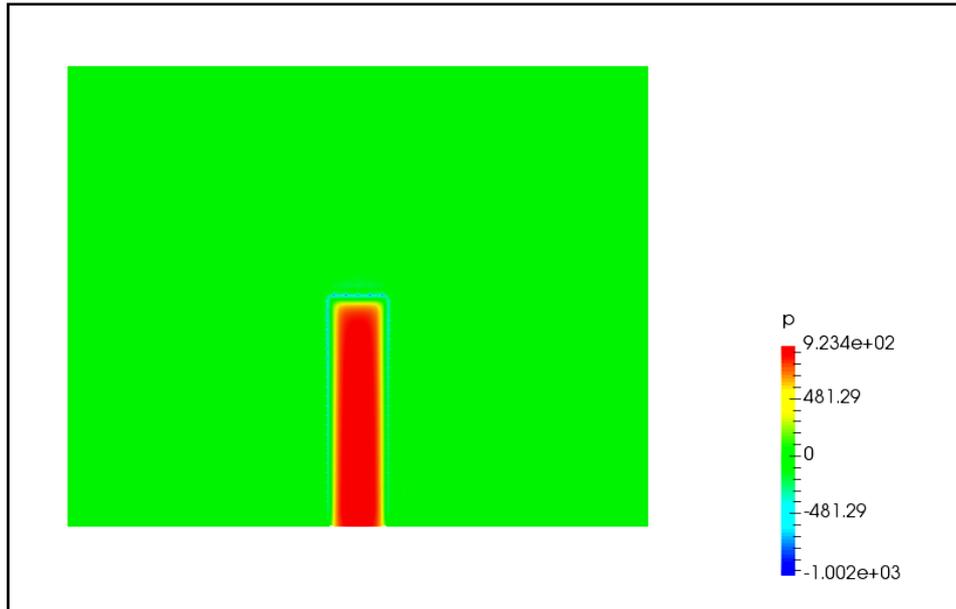


Figure 17: Pressure at windward side

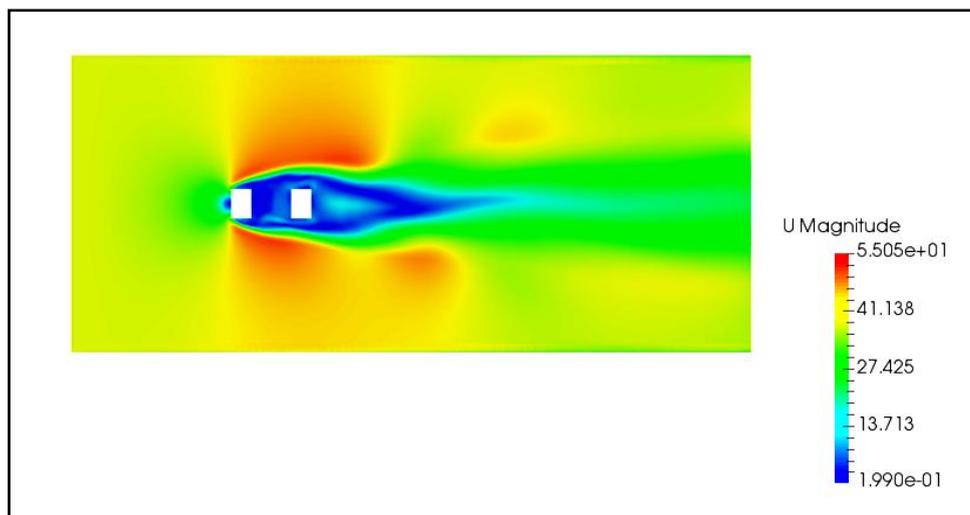


Figure 18: Vortex shedding

CASE 3

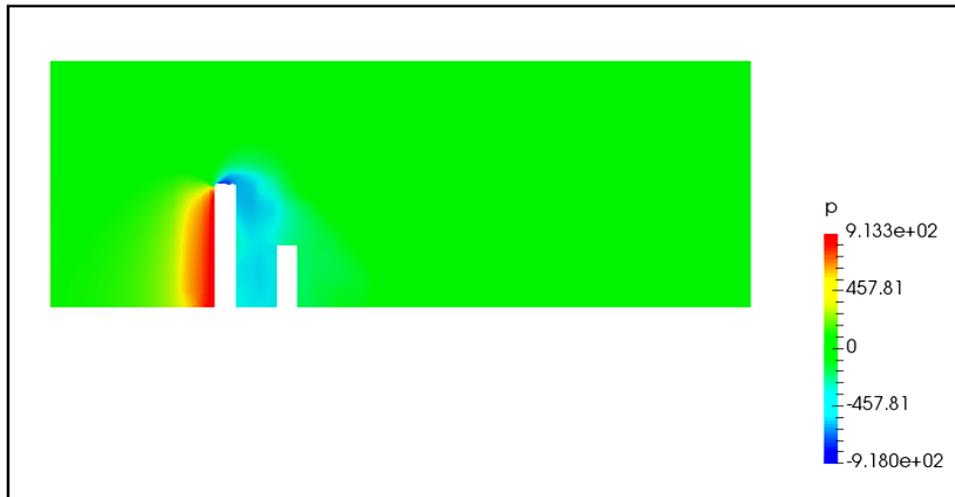


Figure 19: Pressure at mid plane

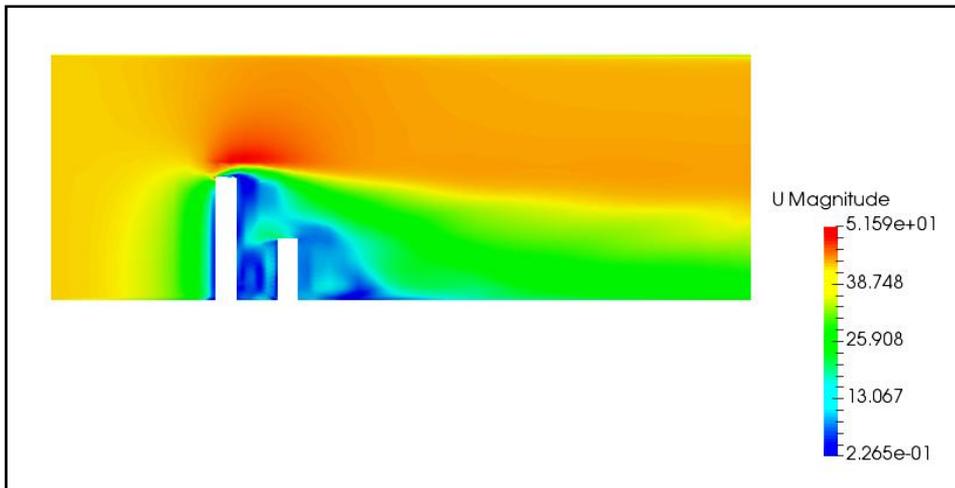


Figure 20: Velocity at mid plane

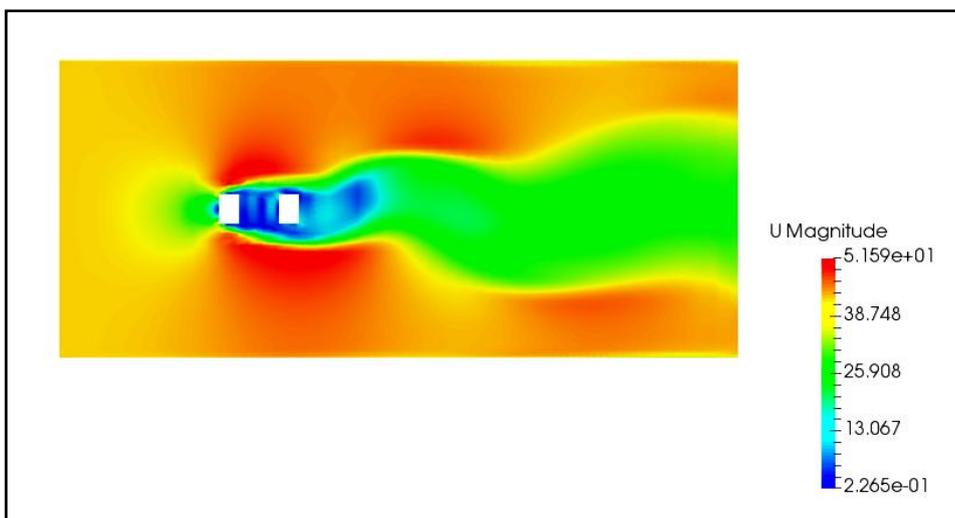


Figure 21: Velocity at 100 m height

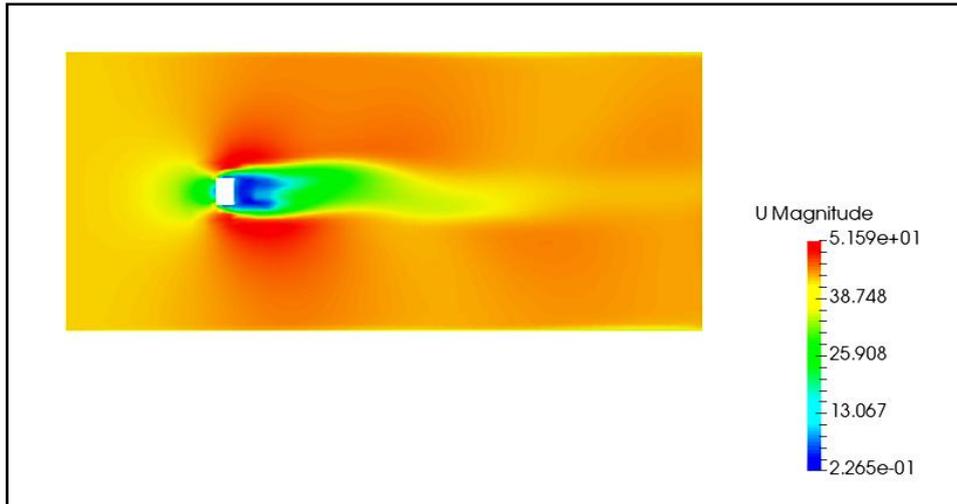


Figure 22: Velocity at 500 m height

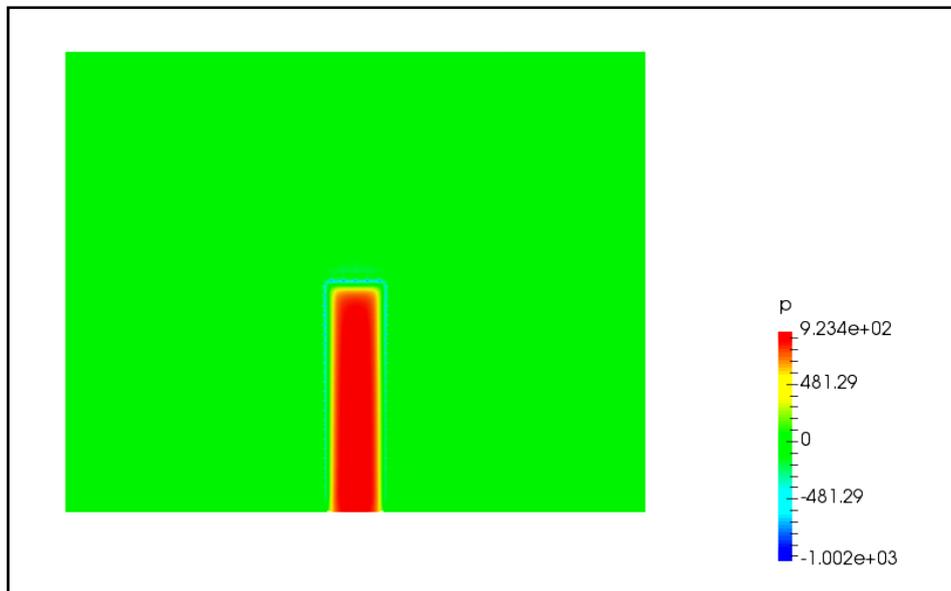


Figure 23: Pressure at windward side

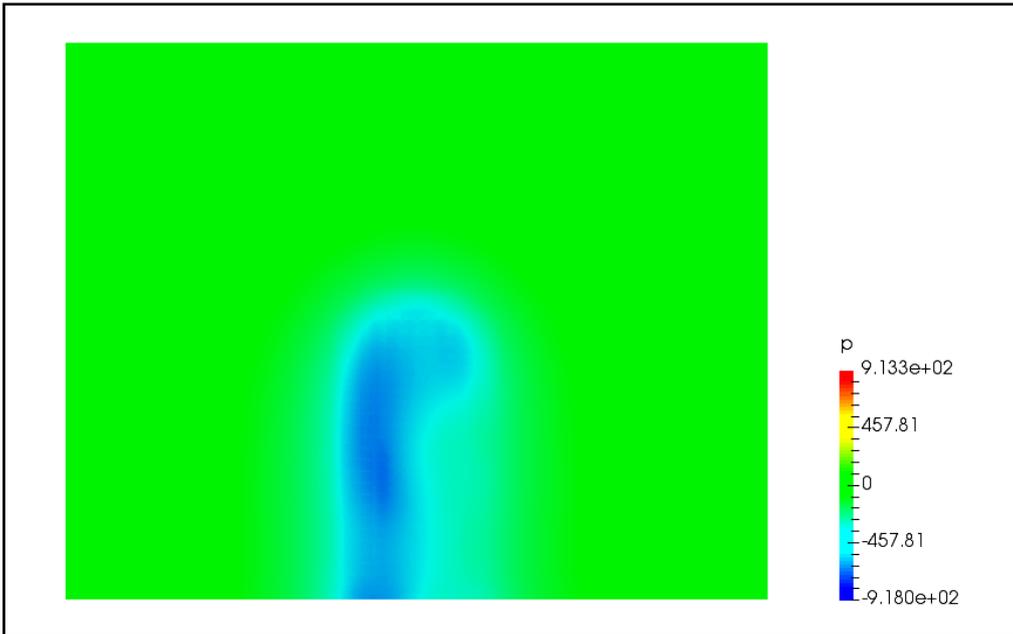


Figure 24: Pressure on smaller building

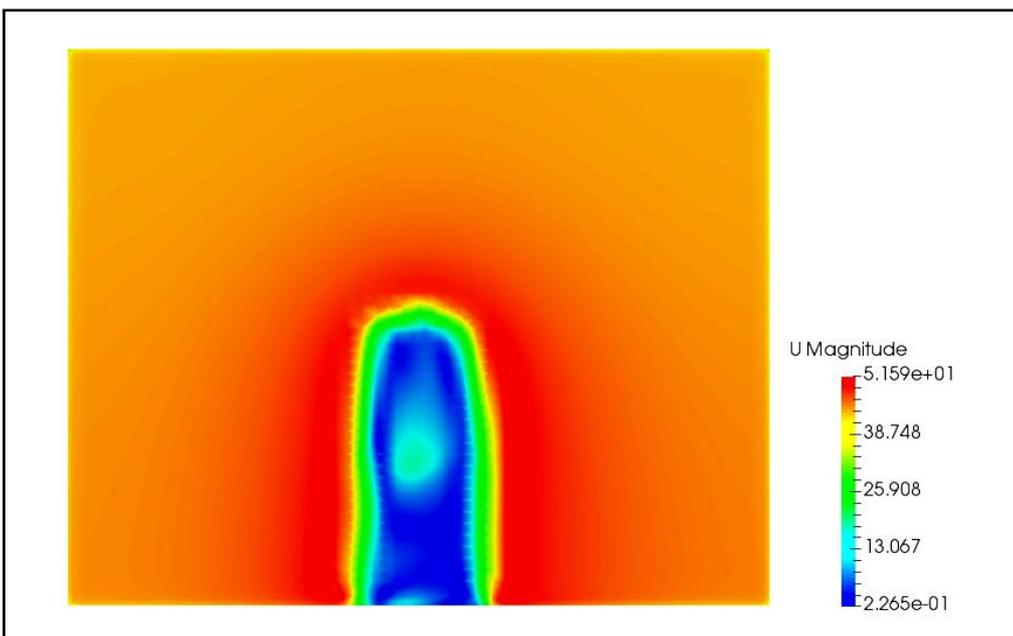


Figure 25: Velocity profile at smaller building

CASE 4

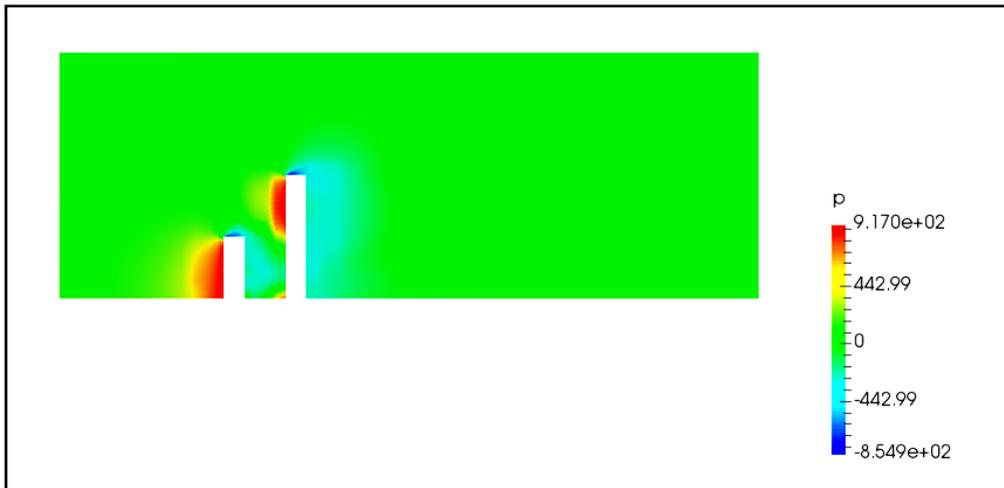


Figure 26: Pressure at mid plane

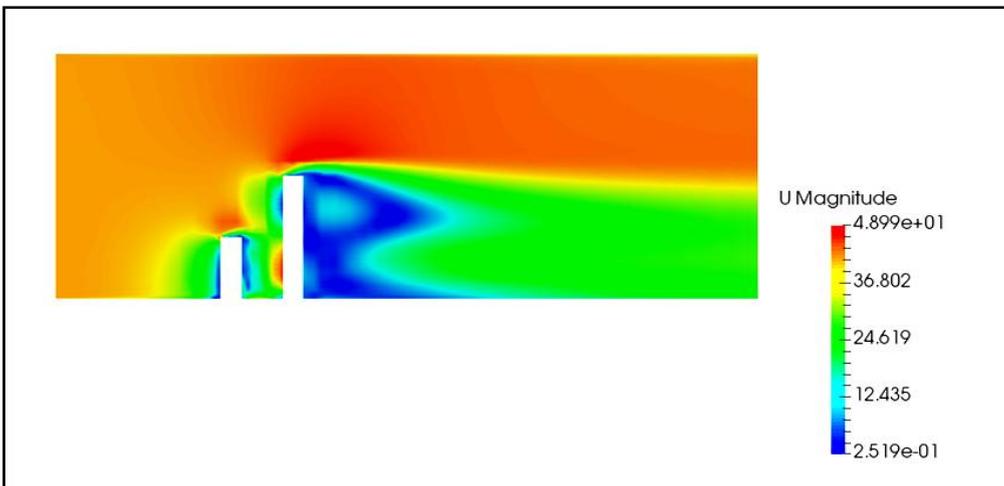


Figure 27: Velocity at mid plane

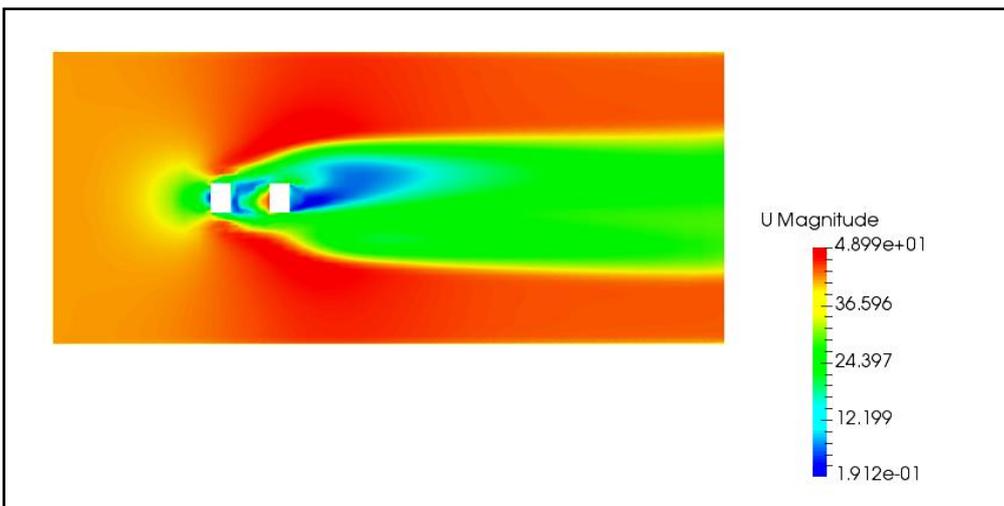


Figure 28: Velocity at 100 m height

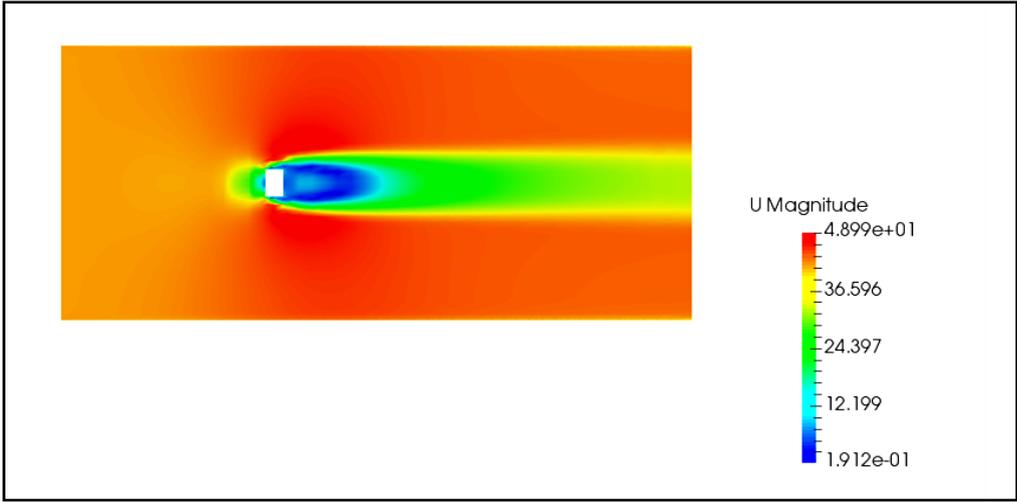


Figure 29: Velocity at 500 m height

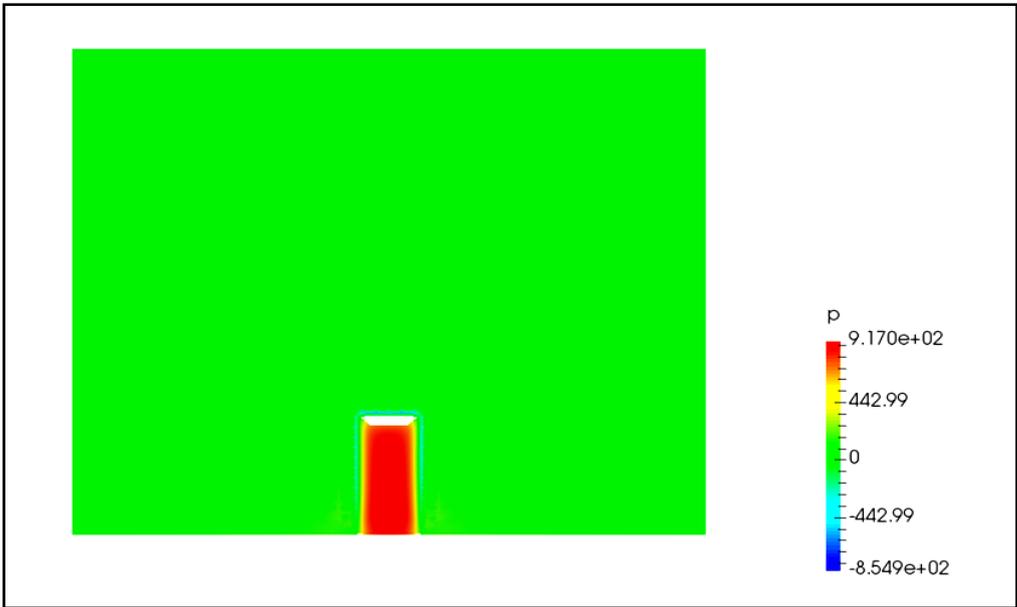


Figure 30: Pressure at windward side

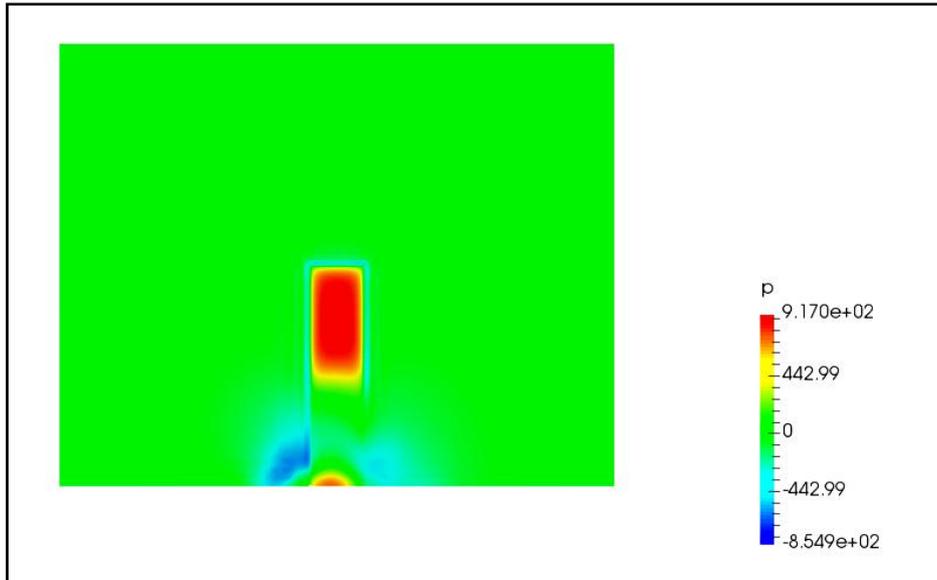


Figure 31: Pressure at larger building

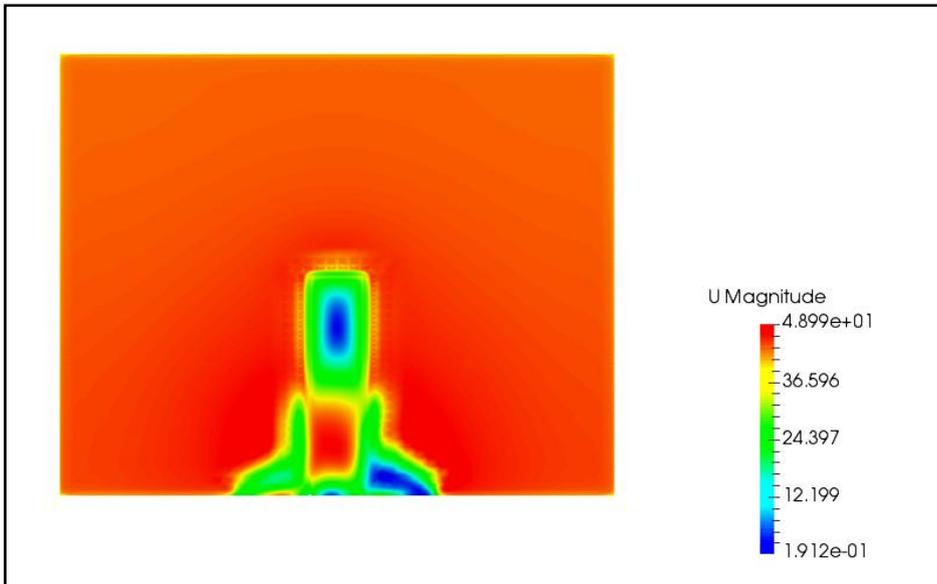


Figure 32: Velocity profile at larger building

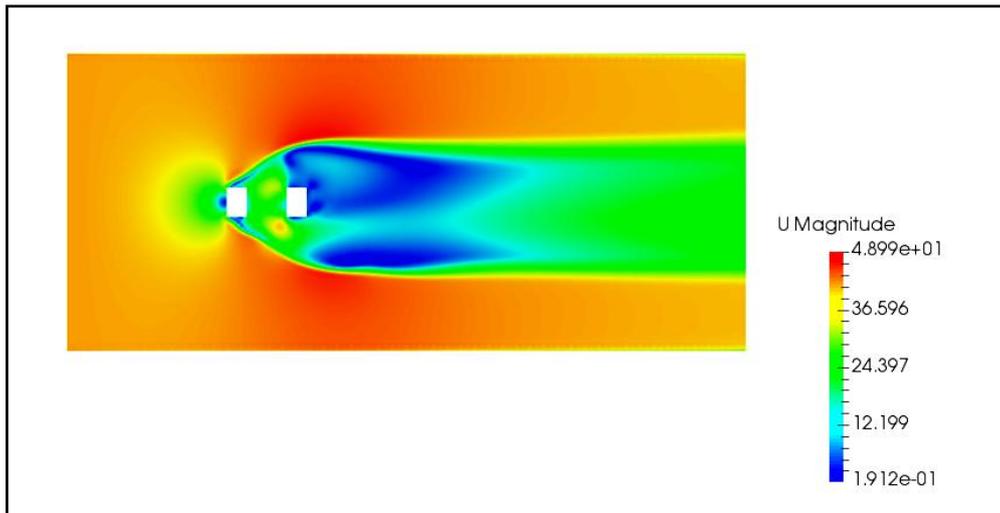


Figure 33: Vortex shedding

8. CONCLUSION

We have hence concluded the detailed research performed on this case study to examine the wind flow pattern around tall buildings using openfoam CFD software. This data can further be used to design such types of buildings in near future and study the effect of such buildings in a city environment and on neighbouring buildings.

9. REFERENCES

1. Application of CFD in wind analysis of tall buildings, Damith Mohotti, Priyan Mendis, Tuan Duc Ngo