



York University CFD Analysis

Balagan, Herman – 217136490

A report submitted in fulfillment of the requirements of the Aerodynamics course as Lassonde School of Engineering

Abstract

Understanding the aerodynamic interactions of airflow around buildings is critical for urban planning, pedestrian comfort, and structural stability. This study employs computational fluid dynamics (CFD) to simulate and analyze airflow around two structures in close proximity. A scaled-down model (100:1) was used to optimize computational efficiency while maintaining geometric and kinematic similarity. The simulation parameters included a uniform airflow velocity of 10 m/s, an ambient temperature of 20°C, and atmospheric pressure of 101325 Pa. Advanced turbulence models were incorporated to enhance solution accuracy. The results indicate significant aerodynamic concerns, including high-velocity wind corridors, stagnation zones, and strong pressure differentials, which may lead to pedestrian discomfort and safety issues. Recommendations are proposed to mitigate these adverse effects before construction, including architectural modifications and urban planning strategies.

Table of Contents

<i>1. Introduction</i>	4
1.1 Background	4
1.2 Objectives	4
<i>2. Experimental Apparatus and Methodology</i>	5
2.1 Computational Framework	5
2.2 Methodology	6
<i>3. Results and Discussion</i>	6
3.1 Problem Identification	6
3.2 Computational Results	6
3.2.1 Individual Building Analysis	6
3.2.2 Combined Building Analysis in Ansys	9
3.3 Engineering Implications and Design Recommendations	20
<i>4. Conclusion</i>	20

1. Introduction

1.1 Background

Aerodynamic interactions between structures play a pivotal role in determining pedestrian comfort and environmental quality in urban settings. The air velocity, turbulence intensity, and vorticity fields influence the accumulation of airborne debris and the formation of wind tunnels, which can pose risks to pedestrians. Computational fluid dynamics (CFD) enables a comprehensive analysis of these flow phenomena by solving the Navier-Stokes and continuity equations at each discretized element within a defined computational domain. In high-density urban areas, these effects are exacerbated by the increasing height and proximity of buildings, making CFD analysis an essential tool for predicting environmental impact.

$$\begin{aligned} -\frac{\partial P}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) &= \rho \frac{\partial u}{\partial t} + \rho(\vec{v} \cdot \nabla)u \\ \rho g - \frac{\partial P}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) &= \rho \frac{\partial v}{\partial t} + \rho(\vec{v} \cdot \nabla)v \\ -\frac{\partial P}{\partial x} + \mu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) &= \rho \frac{\partial w}{\partial t} + \rho(\vec{v} \cdot \nabla)w \end{aligned} \quad [1]$$

Equation 1: 3D Navier Stokes

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0$$

Equation 2: 3D continuity

This study leverages CFD simulations to assess the impact of building placement on airflow characteristics and its implications on pedestrian wind comfort, air pollutant dispersion, and structural wind loads.

1.2 Objectives

The primary objective of this study is to evaluate airflow patterns, including velocity distribution, turbulence effects, and pressure differentials, around two buildings

individually and in proximity. The analysis aims to:

- Identify regions of concern, such as high-velocity wind corridors and stagnation zones.
- Assess the impact of airflow on pedestrian comfort and building functionality.
- Propose engineering solutions, including structural modifications, to optimize pedestrian comfort and structural aerodynamics.
- Validate CFD results using known empirical data and propose further experimental verification methods.

2. Experimental Apparatus and Methodology

2.1 Computational Framework

The CFD analysis was conducted using ANSYS Fluent, a finite volume-based software for solving fluid dynamics problems. The computational domain was discretized using an unstructured mesh, optimized for computational efficiency while maintaining accuracy. The simulations assumed incompressible, steady-state airflow governed by the Reynolds-averaged Navier-Stokes (RANS) equations. The $k-\omega$ SST turbulence model was employed to capture both near-wall effects and free-stream turbulence accurately.

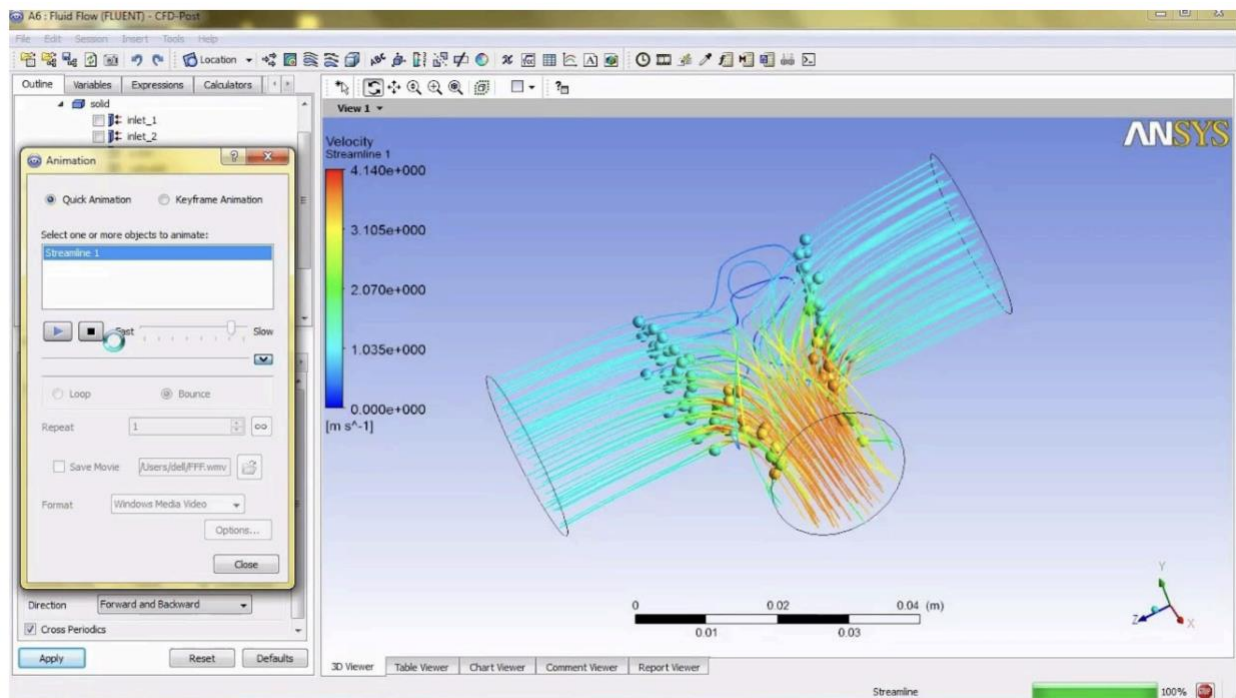


Figure 1: Fluent Workbench

2.2 Methodology

1. **Geometry Modeling**: The building structures were designed within ANSYS Fluent's modeling tool. A 100:1 scale was applied to reduce computational complexity while preserving dynamic similarity. The computational domain was extended sufficiently to avoid boundary-induced inaccuracies.
2. **Mesh Generation**: The computational domain was meshed using an adaptive grid with refined resolution near building edges, wake regions, and high-gradient flow zones. A grid independence study was performed to ensure mesh convergence.
3. **Boundary Conditions**: The following conditions were applied:
 - Inlet velocity: 10 m/s (uniform)
 - Outlet: Pressure outlet at atmospheric pressure (101325 Pa)
 - Wall boundaries: No-slip condition applied to building surfaces
 - Turbulence intensity: Set at 5% to simulate realistic atmospheric conditions.
4. **Simulation Execution**: The simulations were run iteratively until residuals for momentum and continuity equations converged below a threshold of 10^{-6} . The SIMPLE pressure-velocity coupling algorithm was used for solution stability.
5. **Post-Processing**: Velocity vectors, pressure contours, and streamlines were analyzed to interpret flow characteristics. Turbulence kinetic energy and vorticity distributions were also examined for further insight into flow separation and recirculation zones.

3. Results and Discussion

3.1 Problem Identification

Airflow behavior around structures significantly influences pedestrian comfort. Wind velocities in excess of 10 m/s can cause discomfort, while velocities approaching 20 m/s render areas unsuitable for pedestrian traffic. Similarly, pressure differentials impact building access points, potentially making doors difficult to open or prone to abrupt closing forces. Wind-induced oscillations may also affect building stability and long-term structural integrity.

3.2 Computational Results

3.2.1 Individual Building Analysis

- **Wide Building**:
 - Localized velocity extremes varied between 2 m/s and 23 m/s, though pedestrian-level velocities remained between 8-15 m/s.
 - Pressure gradients were similar to the tall building, reinforcing concerns over door usability and pedestrian exposure to wind forces.

- Vorticity contours showed concentrated flow separations behind the structure, similar to the tall building.
- Wake turbulence was more pronounced due to the wider cross-section, leading to enhanced mixing effects.

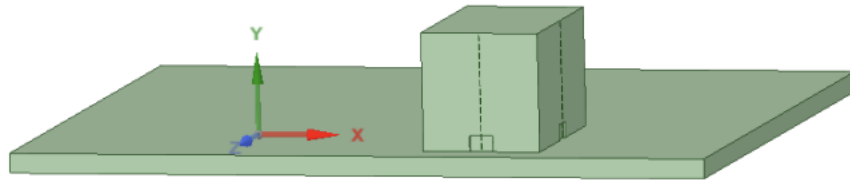


Figure 2: Wide Building Model

- **Tall Building:**
 - Velocity magnitudes near the base remained within the 3-9 m/s range, with peaks at 16 m/s in higher elevations.
 - Pressure distribution analysis revealed localized low-pressure zones near doorways, indicating a potential for abrupt door closure.
 - Vorticity analysis confirmed flow recirculation behind the structure, suggesting potential debris accumulation zones.
 - Turbulence kinetic energy plots indicated increased shear stress at the windward facade, which could impact structural loading.

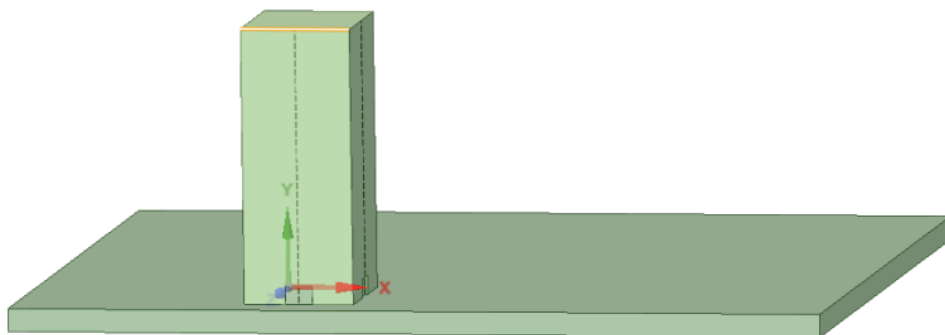


Figure 3: Tall Building Model

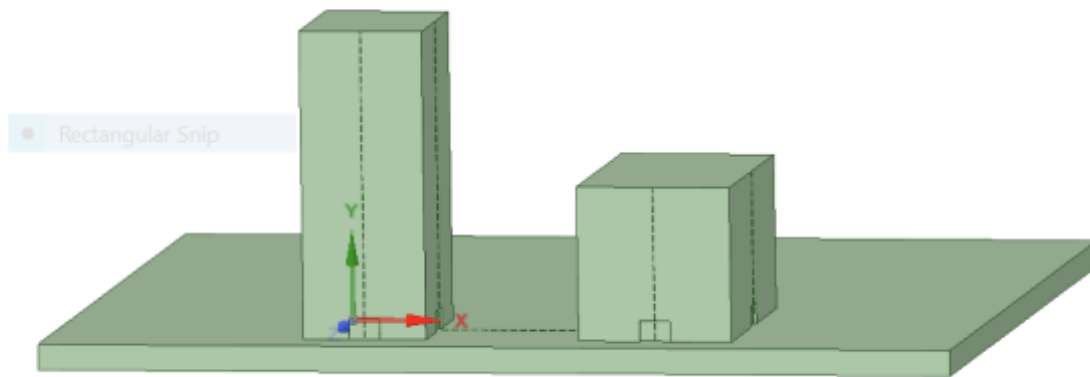


Figure 4: Both Buildings Model

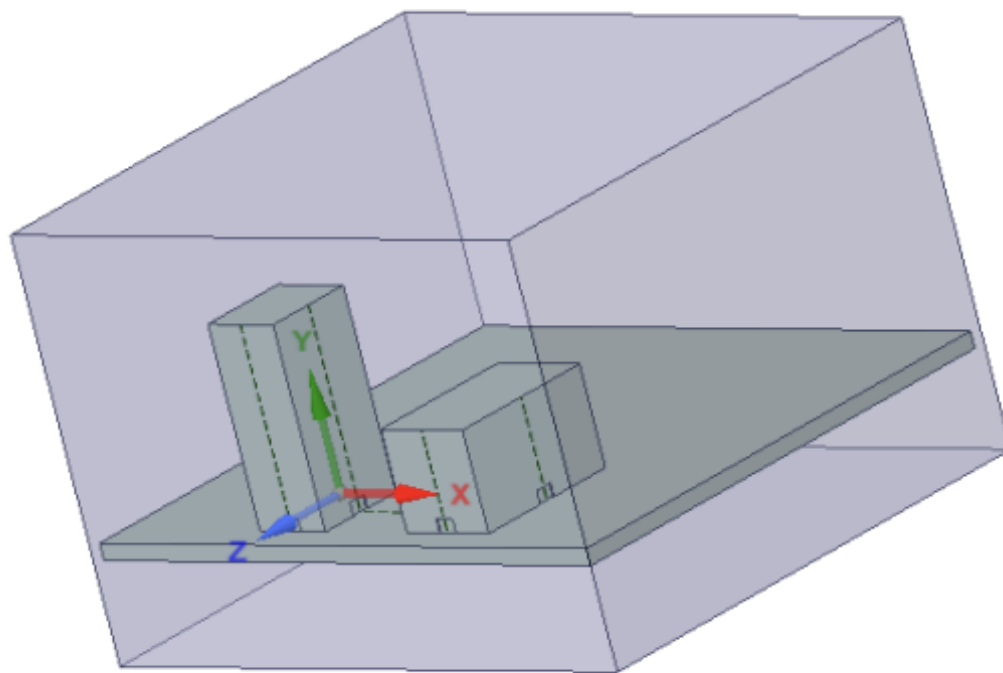


Figure 5: Domain (Consistent for all 3 cases)

3.2.2 Combined Building Analysis in Ansys

When both buildings were modeled in proximity, distinct flow interactions emerged:

- ****Wind Tunnel Effect:**** The velocity field between the buildings reached 20 m/s, making the passage unsuitable for pedestrian use.
- ****Pressure Distribution:**** Similar to individual analyses, the pressure differential near entrances remained suboptimal, necessitating potential design modifications.
- ****Recirculation Zones:**** Airflow separation behind the buildings resulted in substantial stagnation zones.
- ****Vortex Shedding Phenomenon:**** Increased unsteady vortices were observed, potentially affecting long-term structural fatigue of the buildings.

Mesh Generation:

Details of "Mesh"	
Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Export Format	Standard
Export Preview Surface Mesh	No
Element Order	Linear
Sizing	
Size Function	Curvature
<input type="checkbox"/> Max Face Size	Default (6.746e-002 m)
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default (3.373e-004 m)
<input type="checkbox"/> Growth Rate	Default (1.20)
<input type="checkbox"/> Min Size	Default (6.746e-004 m)
<input type="checkbox"/> Max Tet Size	Default (0.134920 m)
<input type="checkbox"/> Curvature Normal Angle	Default (18.0 °)
Bounding Box Diagonal	1.34920 m
Average Surface Area	0.100010 m ²
Minimum Edge Length	3.e-003 m

Figure 6: Mesh Settings

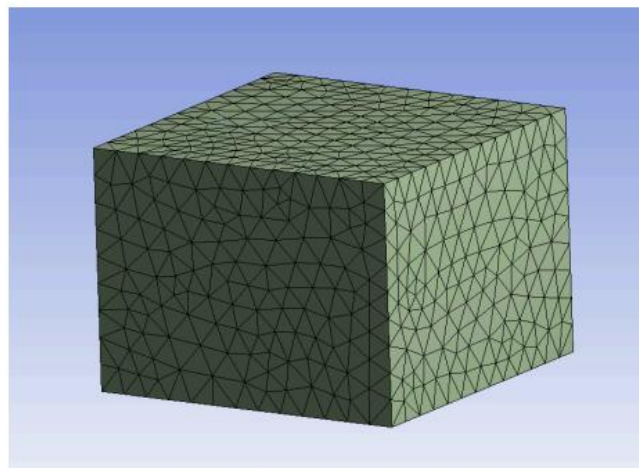


Figure 7: Mesh surrounding the buildings

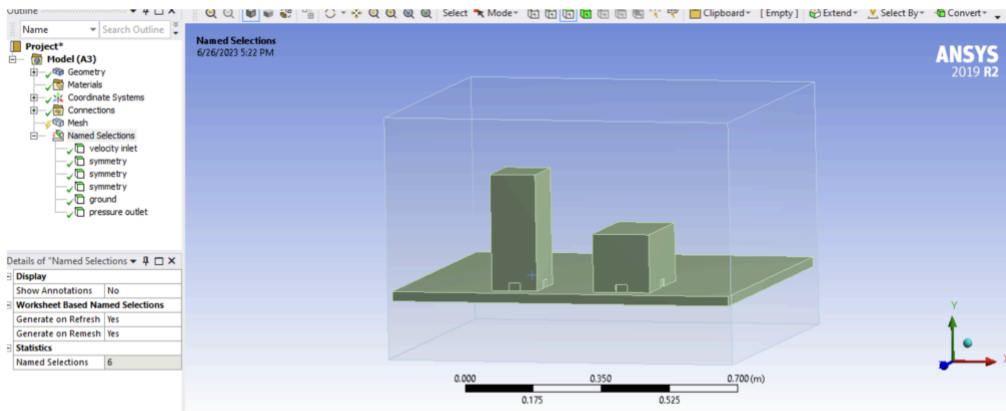


Figure 8: Named Selections

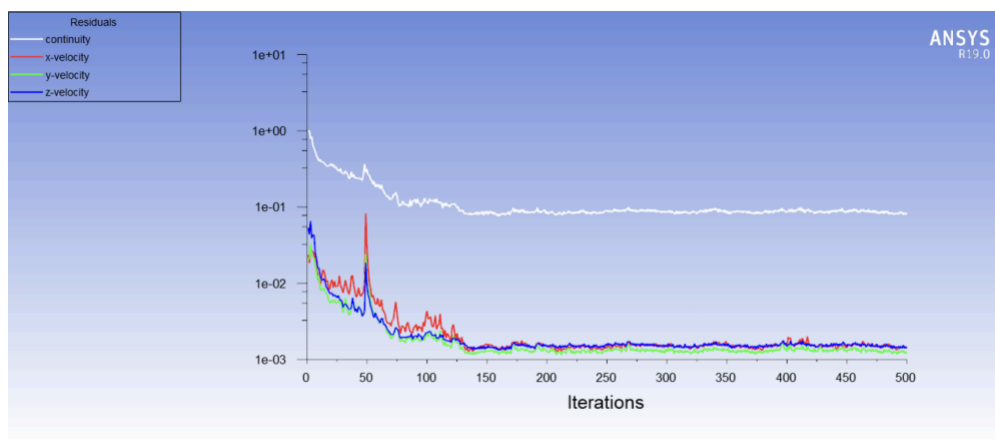


Figure 9: Number of Iterations, convergence

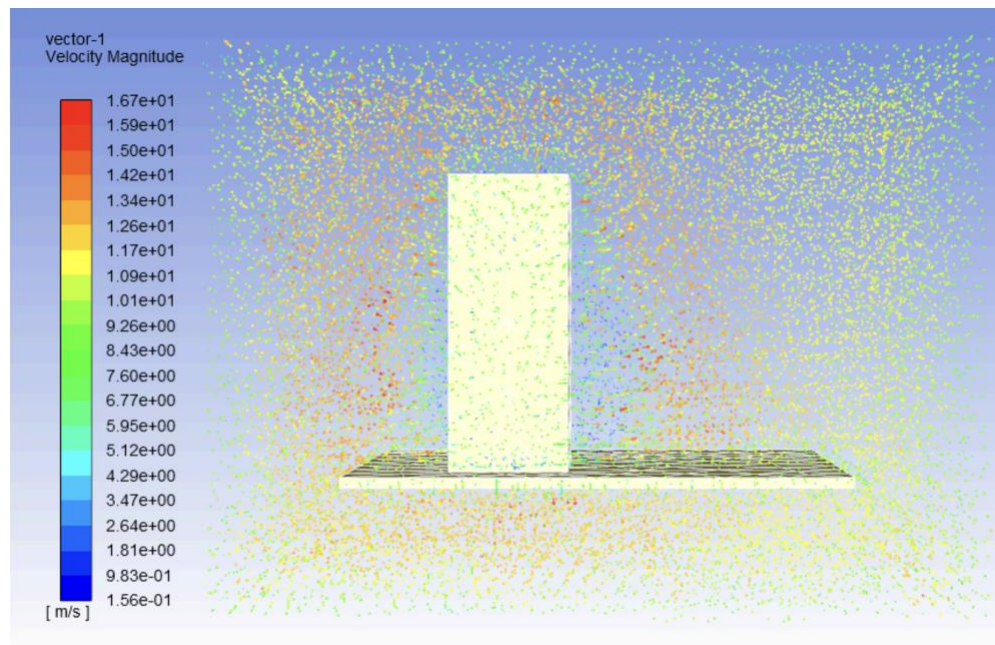


Figure 10: Tall Building Velocity Magnitude (Front View)

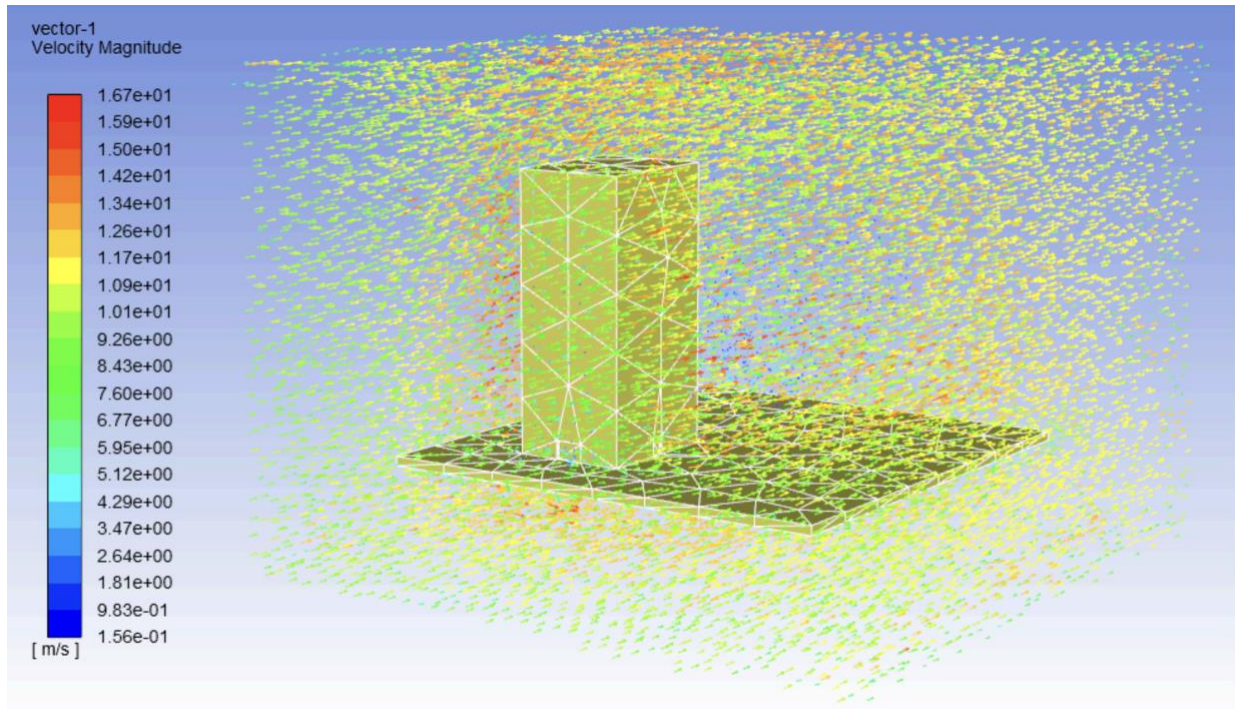


Figure 11: Tall Building Velocity (Angle 2)

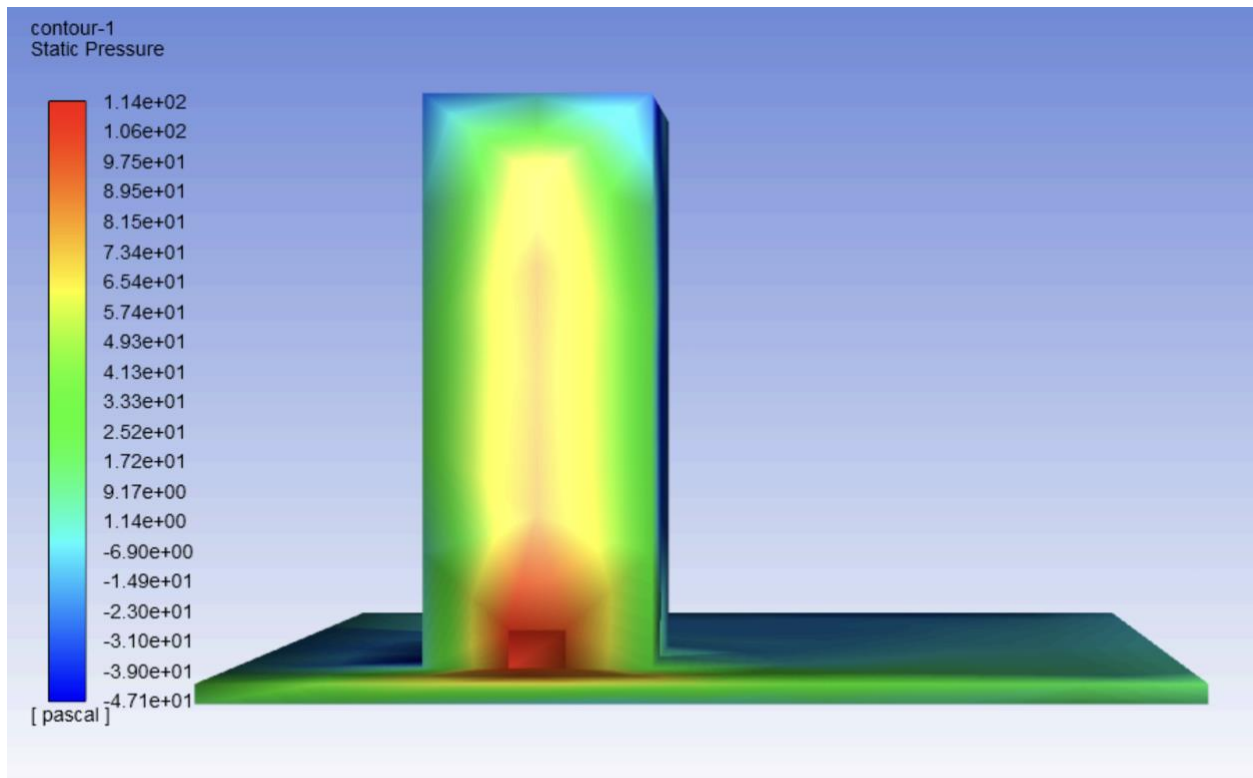


Figure 12: Tall Building Pressure Distribution (Front View)

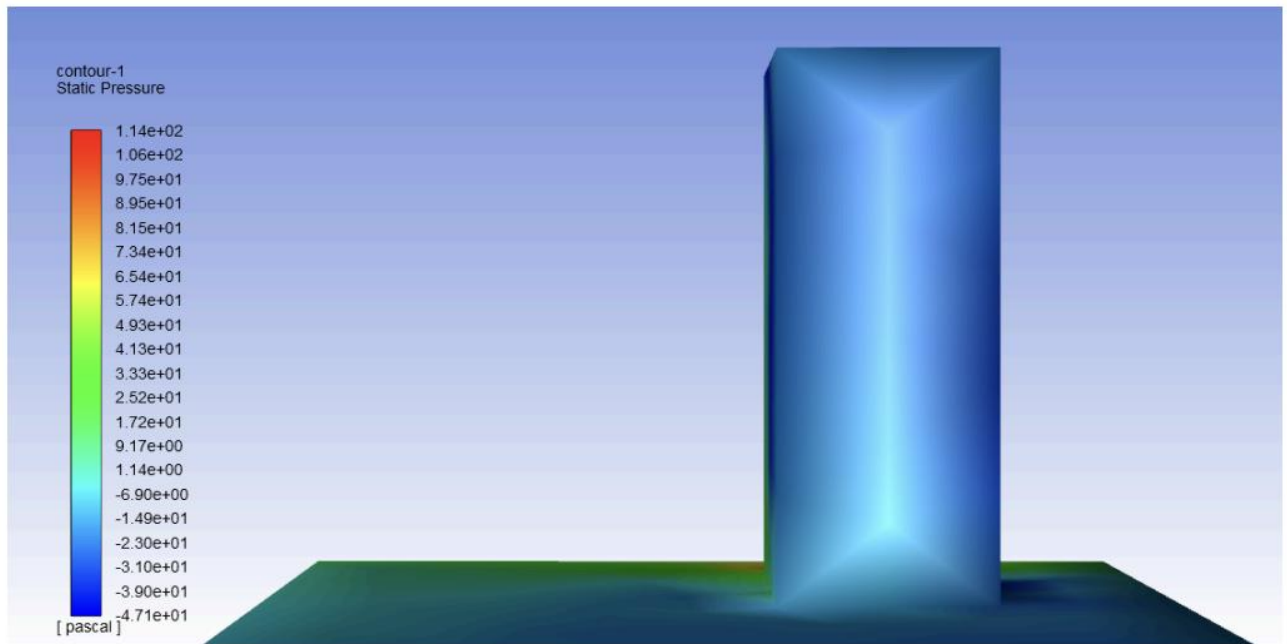


Figure 13: Tall Building Pressure Distribution (Back View)

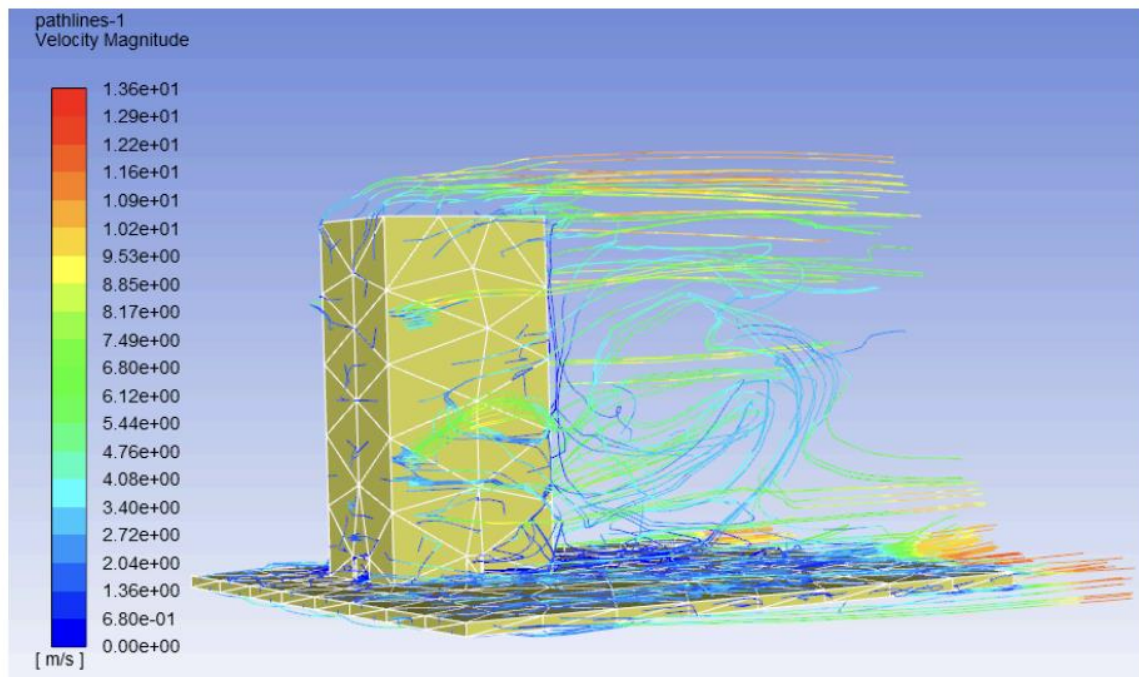


Figure 14: Tall Building Vorticity

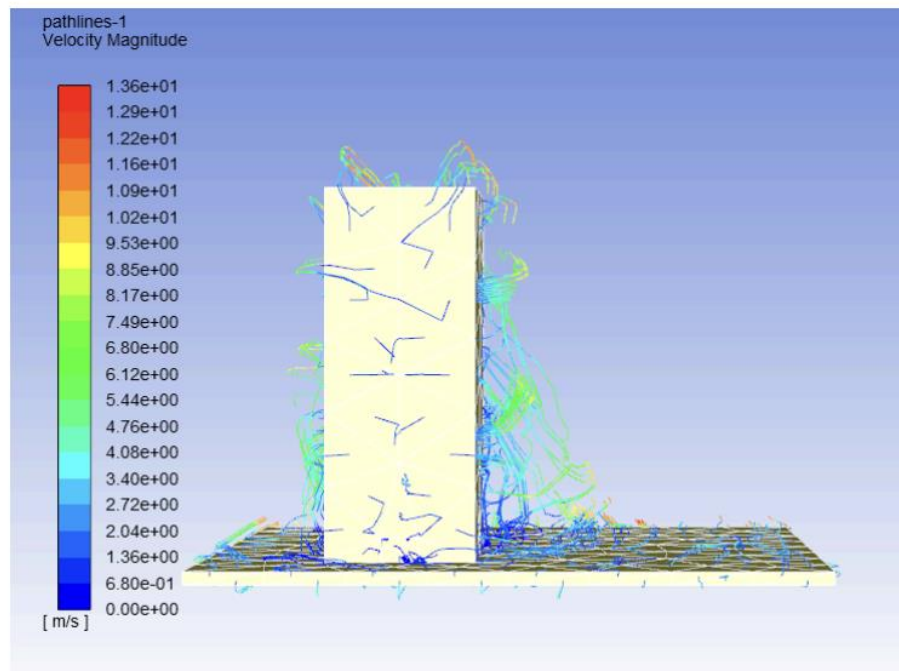


Figure 15: Vorticity (Front View)

Wide Building Analysis:

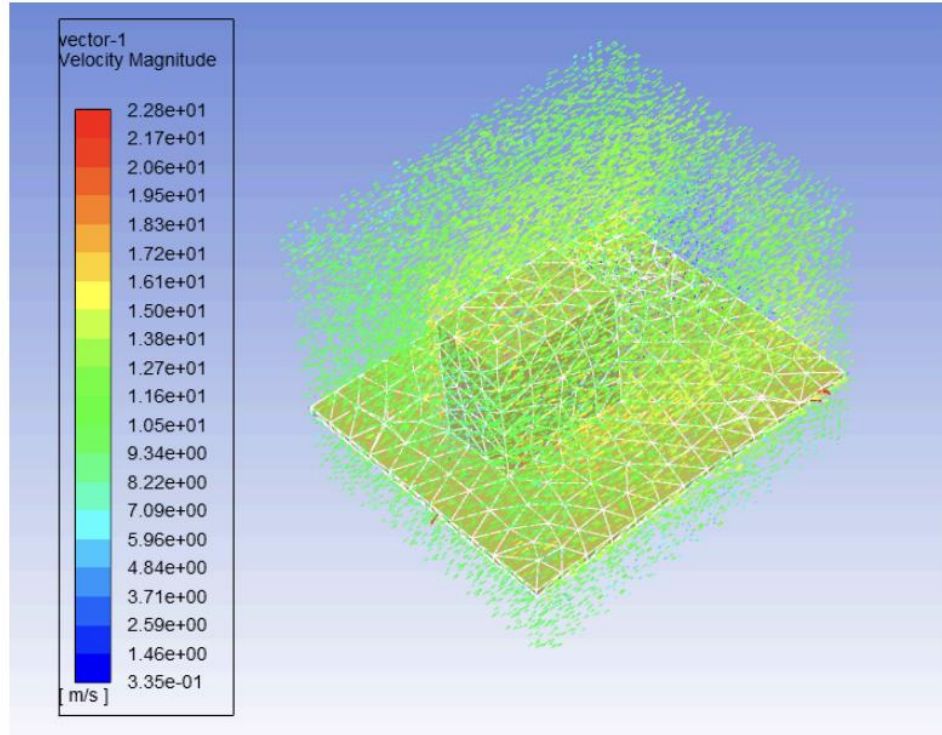


Figure 15: Velocity Field (note the outliers on the edge of the ground)

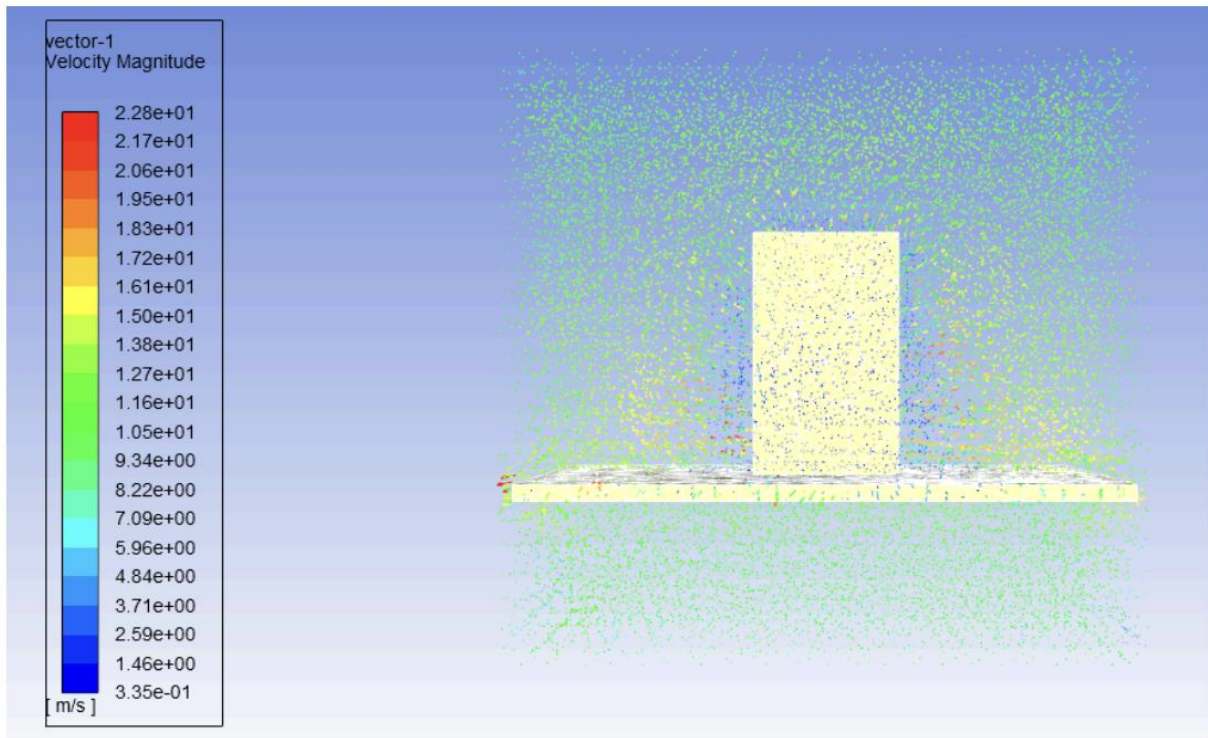


Figure 16: Wide Building Velocity Field (Front View)

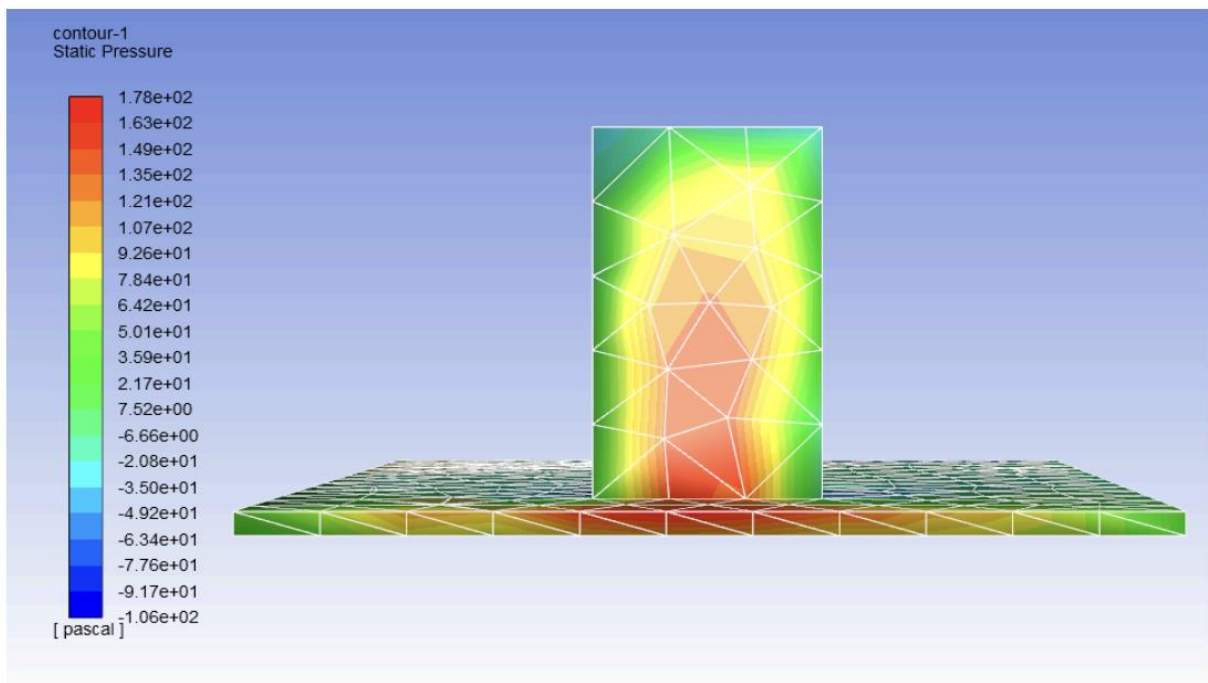


Figure 17: Wide Building Pressure Distribution (Front View)

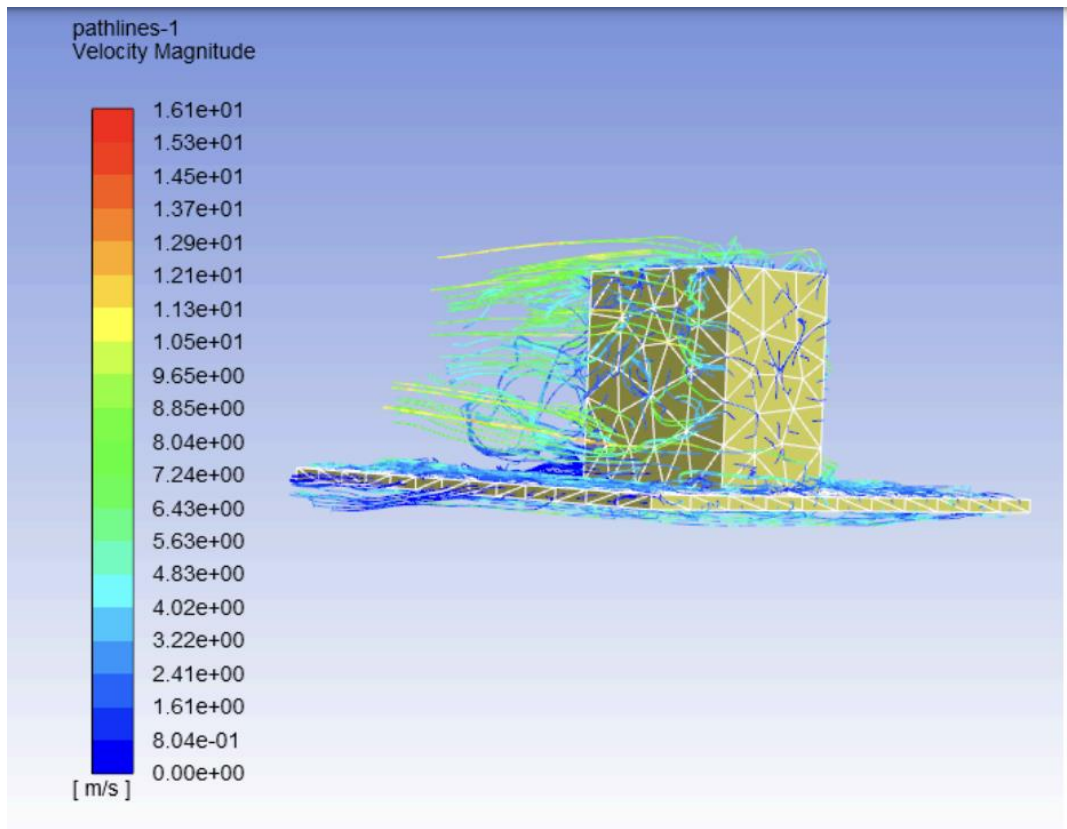


Figure 18: Wide Building Vorticity

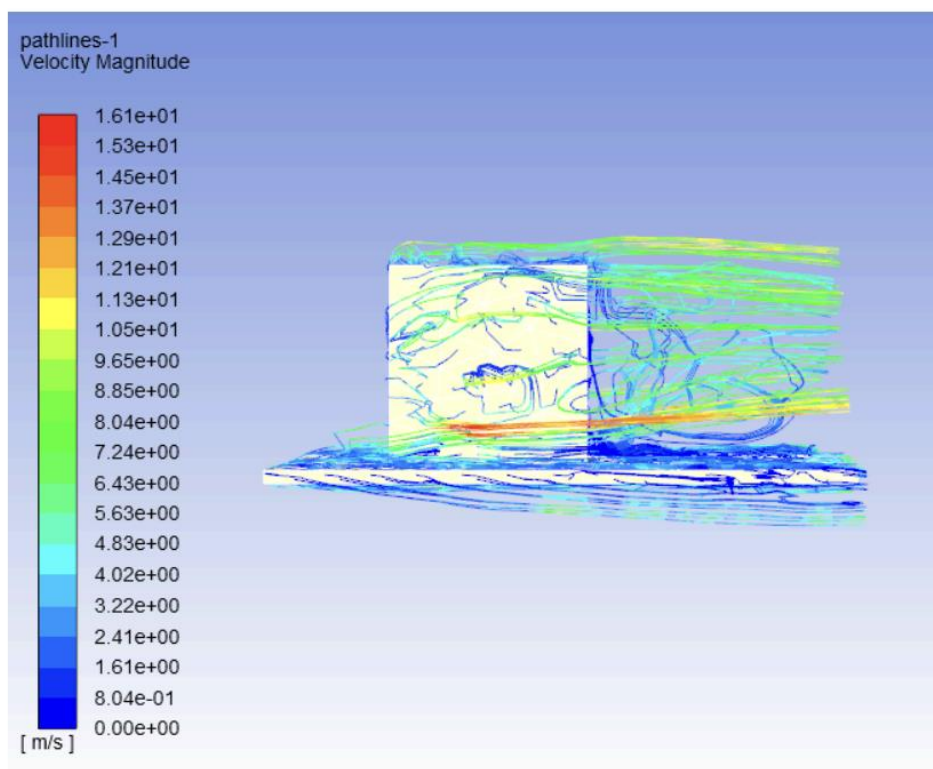


Figure 19: Wide Building Vorticity (Side View)

Both Buildings Analysis:

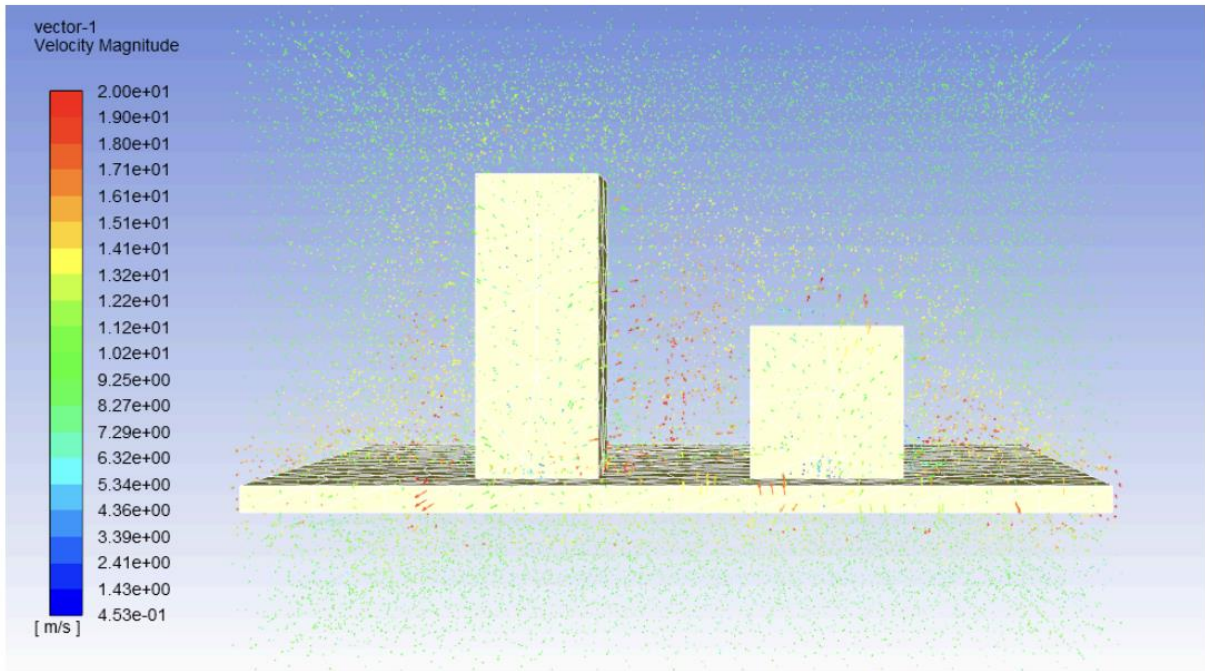


Figure 20: Both Buildings Velocity Field (Front View)

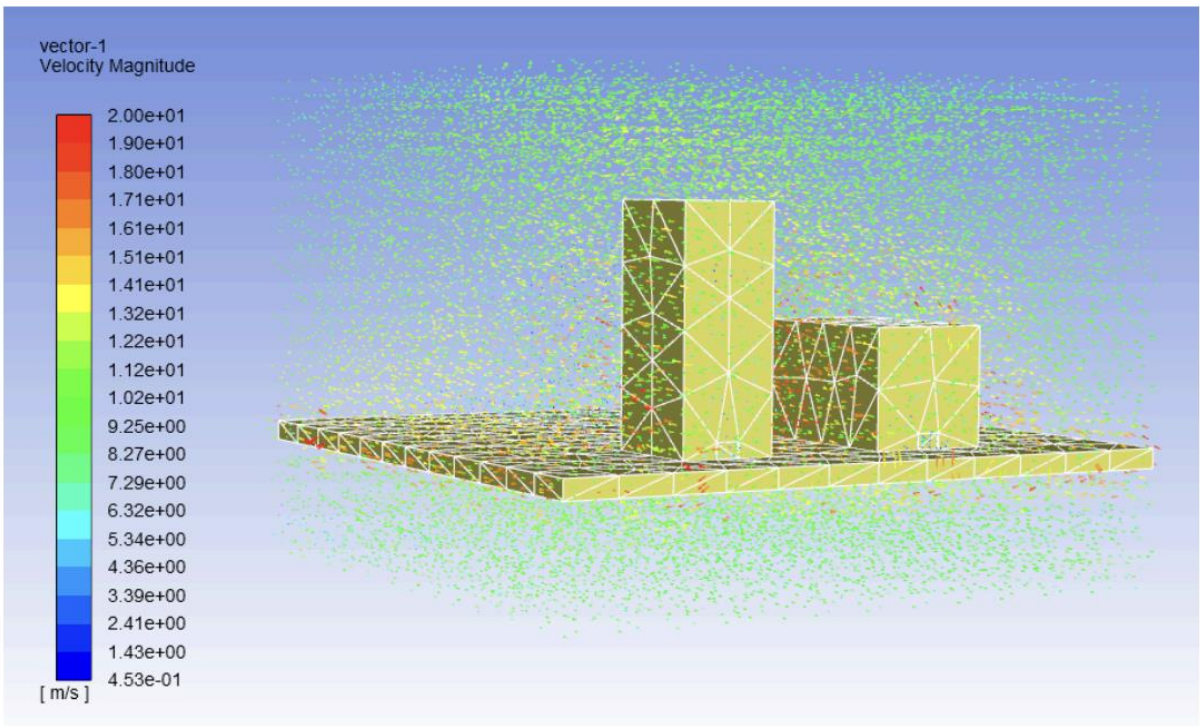


Figure 21: Both Buildings Velocity Field (Separate Angle)

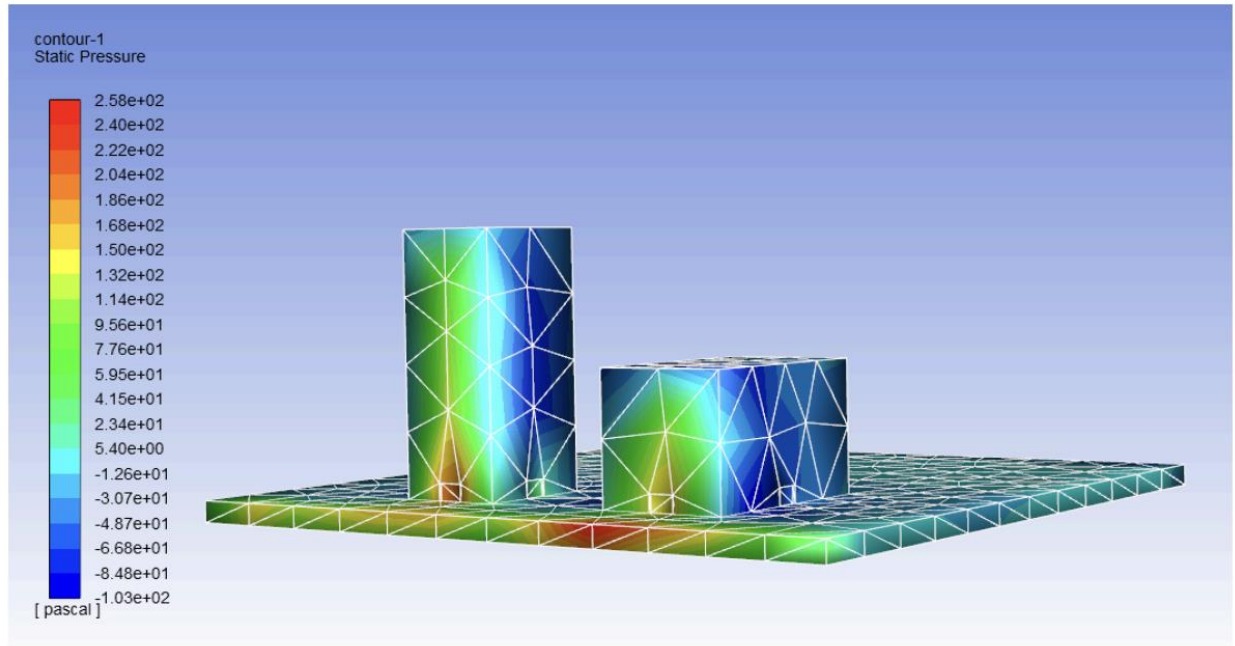


Figure 22: Both Buildings Pressure Distribution

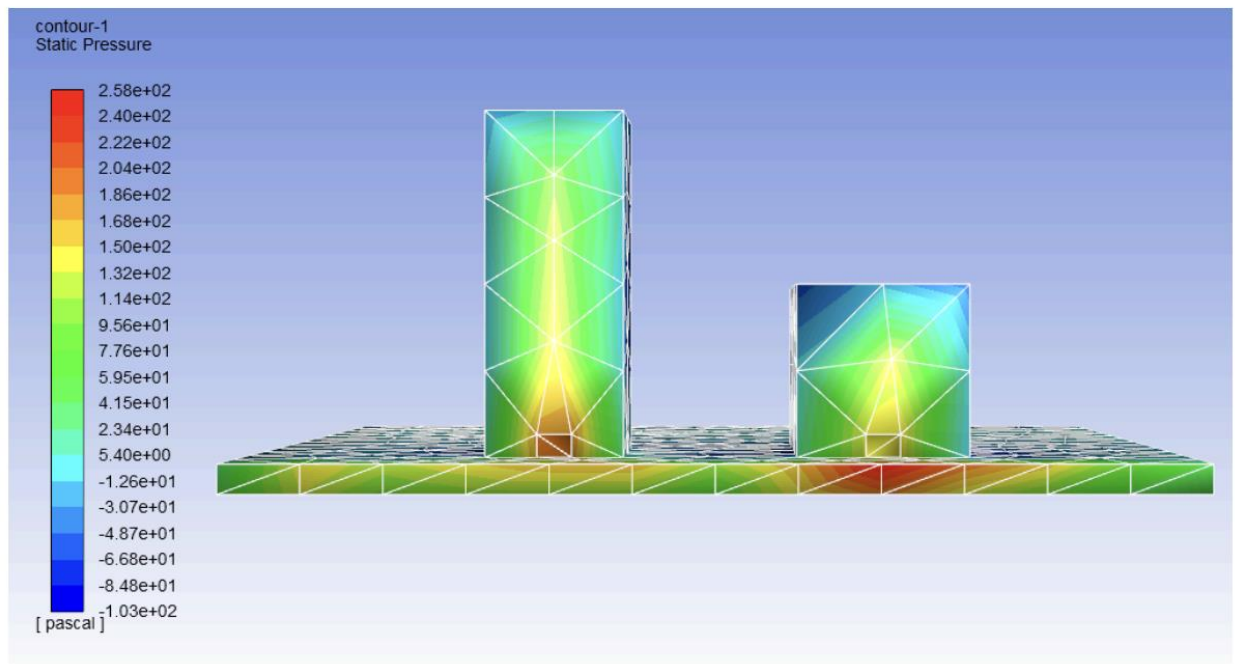


Figure 23: Both Buildings Pressure Distribution (Front View)

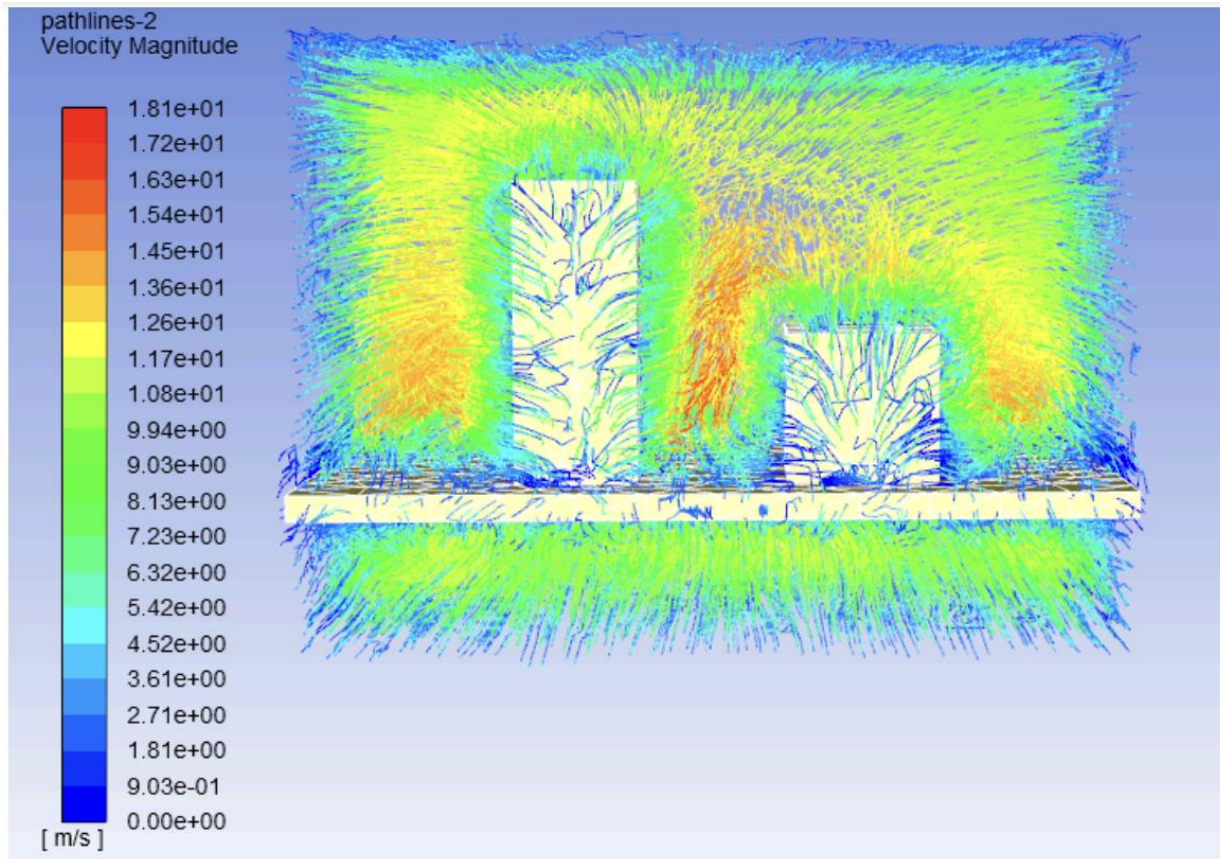


Figure 24: Both Buildings InDepth Vorticity

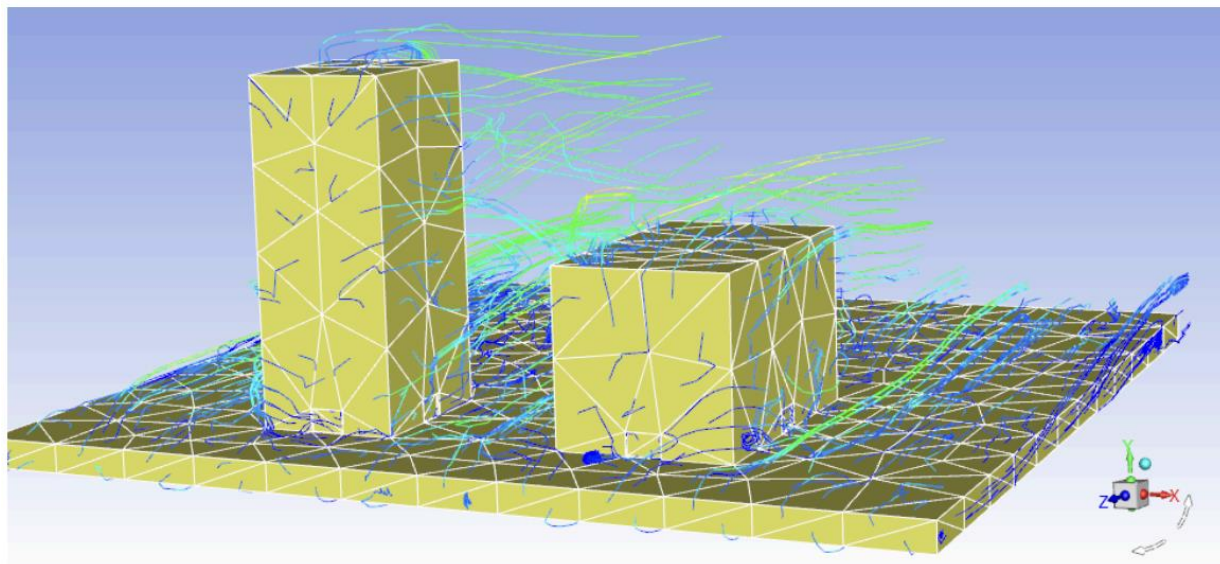


Figure 25: Both Buildings Vorticity

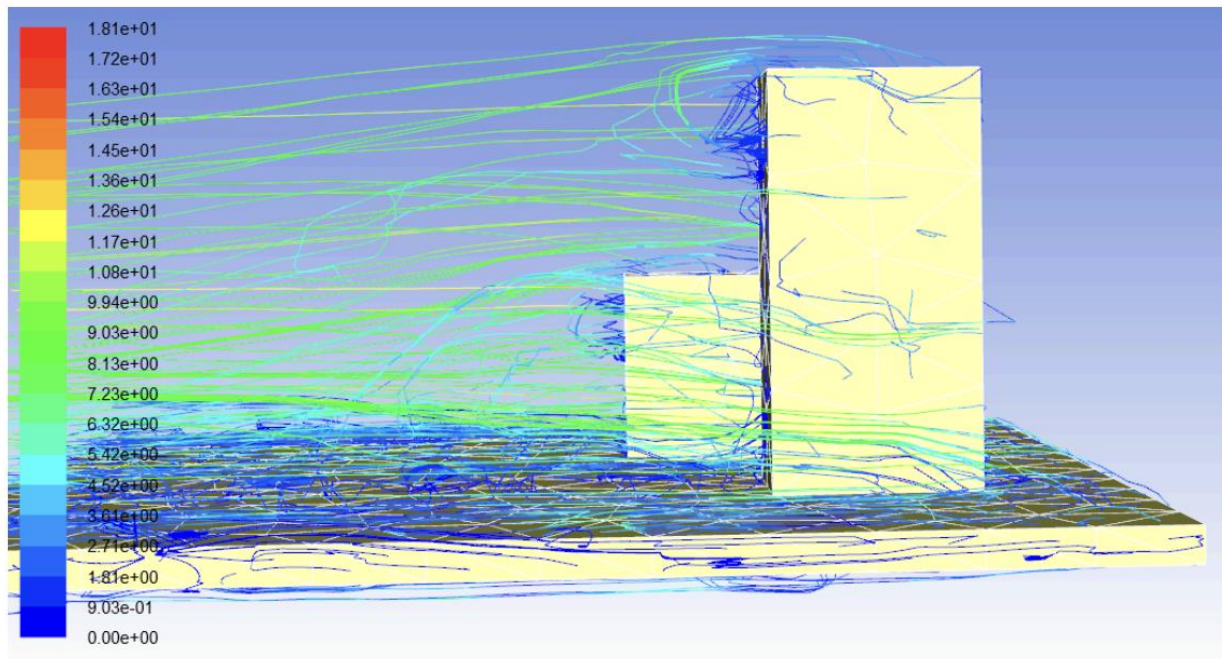


Figure 26: Both Buildings Vorticity (Side View)

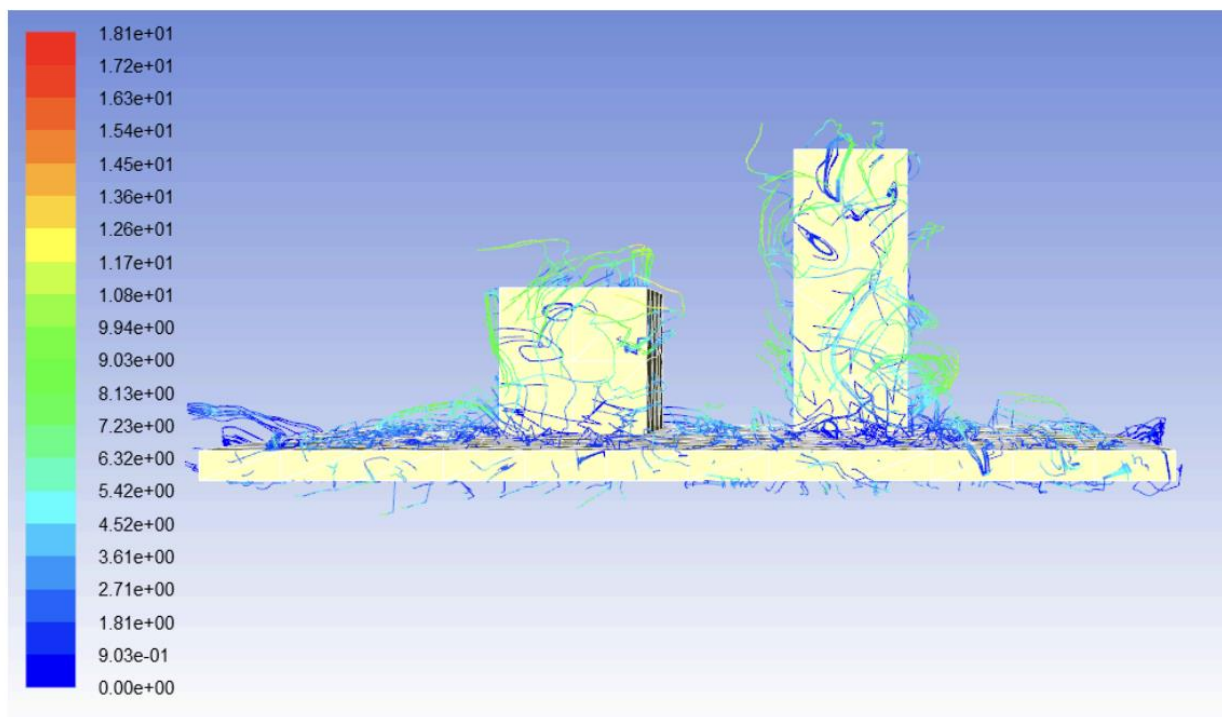


Figure 27: Both Buildings Vorticity (Rear View)

3.3 Engineering Implications and Design Recommendations

- **Mitigating High-Velocity Wind Tunnels:** Wind deflectors or architectural modifications, such as extended canopies or strategically placed barriers, can disrupt accelerated flow between buildings.
- **Optimizing Entrance Design:** Recessed entryways or vestibule designs can minimize pressure-induced door malfunctions.
- **Addressing Recirculation Zones:** Landscaping elements or baffle structures may be implemented to mitigate stagnant airflow zones.
- **Structural Reinforcement Against Vortex Shedding:** Incorporating tuned mass dampers or aerodynamic modifications can reduce oscillatory forces on buildings.

4. Conclusion

This CFD study provides a comprehensive analysis of airflow interactions between adjacent buildings, highlighting key aerodynamic concerns. The findings emphasize the necessity for preemptive design considerations to mitigate high wind velocities, pressure differentials, and turbulence effects. Future work should incorporate transient simulations and wind tunnel experiments to validate computational findings and refine design strategies.