Modifications of Wind Response of Tall-Building Caused by Interfering Effects: A CFD Approach


Abstract: The wind is an essential factor to consider in the design and construction of tall buildings. As buildings get taller, the wind’s impact becomes more significant, and the building’s stability and safety become more critical. The interfering effect is one of the significant consequences of a building that needs to be considered. The "interfering effect" is a phenomenon in wind engineering that occurs when an upstream structure affects the wind load on a downstream building. In the past, most of the studies of interfering effects were done with 2D or 3D simulations, only considering one or two parameters from the shape, height, and angle. Therefore, this research attempts to analyze the interfering effect qualitatively and quantitatively from the upstream building to a selected square-shaped principal building by varying the height of the interfering building with different shapes, namely, circular, cross, and triangular shapes with different orientations based on 3D CFD modeling. The commercial CFD package Midas NFX is used for this numerical analysis. The results from the base moment and base shear suggest that a safety factor for interfering effects should be considered in designing the building structures in the city area to ensure the stability of the building, and it is, for the worst-case scenario, 1.3. The pressure fluctuation results highlighted the importance of designing the connection of the cladding system to both compression and tension forces. The findings of the present paper will be crucial in ensuring the stability and safety of the building structures when those buildings are in a dense building environment.

Keywords: CFD Simulation, Interfering effect, Turbulence model, Wind response, Wind tunnel test

1. Introduction

Due to global urbanization, high-rise buildings are becoming a more common characteristic in cities to meet human thoughts and needs within limited areas. The wind is a major factor in determining the lateral load on tall buildings. The impact of wind loads on buildings can manifest in various ways, such as interfering effects, dynamic response, wind-borne debris, and aerodynamic instabilities. Among these impacts, the interfering effect is a significant phenomenon that may alter the wind flow characteristics around the building due to the influence of adjacent buildings [1]. Based on the shape and orientation of the upstream building, the interfering effect changes the wind load [2].

Previous research on interfering effects mostly used 2D or 3D simulations and only took into account one or two parameters related to shape, height, and angle [3,4,5]. A few research have concentrated on the interference effects of the height change of the interfering building [6]. Therefore, this study uses 3D CFD modeling to principal building. Accurate prediction of wind behaviour around buildings is crucial for optimizing building design. The wind analysis can be conducted based on currently available

v vary the height of the interfering building with various shapes, including circular, cross, and triangular shapes with different orientations, in order to analyze the interfering effect qualitatively and quantitatively from the upstream building to a chosen square-shaped
wind codes, computational fluid dynamics (CFD) simulations, and wind tunnel testing. Most wind codes have been unable to provide accurate results on high-rise buildings with irregular geometries and apply only to buildings less than 200 meters high and have a regular shape [7]. The other two methods, wind tunnel tests and Computational Fluid Dynamics (CFD), are frequently used to analyse tall buildings with irregular geometries as they have the flexibility to analyse any type of building. But the wind tunnel test method is expensive and time-consuming to set up and conduct tests. When considering the CFD simulation method, it gives adequately accurate results with limited resources. In this study, wind flows around the building are analysed by using Computational Fluid Dynamic (CFD) approach.

First, the CFD software was used to model the selected building structure from the literature [8]. There are three main approaches in CFD simulation for wind analysis: Reynolds-averaged Navier–Stokes equations (RANS equations), Large eddy simulation (LES), and Direct numerical simulation (DNS). According to the literature, DNS gives more accurate results but requires more computational power whereas RANS gives reasonably accurate results with less computational power. The RANS approach is utilized in this study to model the turbulent flow due to computer facility limitations. The findings were validated using published wind tunnel test results for the same building. The validated CFD model was used to identify the interference effect on a square-shaped principal building from the upstream building with different shapes, orientations, and heights. This study presents the comparison of the base moment, base shear, and pressure fluctuation of the principal building to assess the impact of the upstream building considering the interfering effects.

2. **Wind-induced Responses of Tall Buildings**

   As urbanization progresses and building technologies advance, modern tall structures are becoming slender, more flexible, lightweight, and exhibit lower damping properties. However, this trend also makes them more susceptible to wind-induced dynamic excitations, increasing their vulnerability [9]. Tall building wind design typically considers along wind, across wind, and torsional responses, as illustrated in Figure 1. Wind design codes and standards are employed during the preliminary design stages to estimate the impact of wind on tall buildings.

   In-depth investigation and analysis of the responses and behaviours of individual buildings are critical in developing and optimizing building designs. Wind loads are particularly significant for tall buildings, especially in cyclone-prone areas, as these structures are prone to wind-induced vibrations due to increased flexibility and limited damping properties [10]. Wind is generated by the differential heating of the atmosphere by the sun, resulting in large-scale wind patterns driven by differences in solar energy absorption between the equator and the poles. Wind effects on structures can be categorized into static and dynamic effects [11]. Static effects result in elastic bending and twisting of the structure, while dynamic effects cause vibrations or oscillations. The wind is characterized by a constant mean wind velocity and varying gust velocity, resulting in both mean and fluctuating wind forces.

![Figure 1 - Wind Responses of a Building](image)

2.1 **Along-Wind Response**

   The along-wind response of a building is primarily influenced by pressure variations on both the windward face (the frontal face exposed to the wind) and the leeward face (the back face) of the structure, as depicted in Figure 2. This response can be considered to have both a mean component caused by the mean wind speed, and a fluctuating component. The fluctuating component is caused by differences in wind speed from the mean and is typically composed of a random collection of eddies of various sizes [12]. The gust factor approach can provide reasonably accurate predictions of the dynamic response of buildings in the along-wind direction. The along-wind response of structures can be modeled as single- or multiple-degree-of-freedom systems [13].
First, the CFD software was used to model the Fluid Dynamic (CFD) approach. Adequately accurate results with limited consideration of the CFD simulation method, it gives tunnel test method is expensive and time-to-analyse any type of building. But the wind codes, computational fluid dynamics (CFD), simulation for wind analysis: Reynolds–Stokes equations (RANS), Large eddy simulation (LES), and Direct numerical simulation (DNS). According to wind codes, computational fluid dynamics standards are employed during the preliminary design stages to estimate the impact of wind on tall buildings under wind loads, which can be especially in cyclone-prone areas, as these wind effects on structures can influence the torsional response. These include computational studies that employed dynamic pressure and force data obtained from wind-tunnel models [16], as well as experimental investigations carried out on aero-elastic models with torsional degrees of freedom [17]. Cheung and Melbourne [18], Lythe and Surry [19], and Isyumov and Poole [20] conducted an investigation to explore whether torsional motions enhance the perception of motion when assessing accelerations near the peripheral sections of a tall building. The outcomes of these studies have provided valuable insights into the dynamic behavior of tall buildings under wind loads, which can be useful in enhancing their design and construction to ensure their safety and stability in high wind conditions.

### 2.4 Interference Effects

The impact of the surrounding building on the wind loading of a building considered is known as the interference effect. According to wind tunnel research, interfering effects do not always lower wind loads on a given building and can instead increase wind loads, which can have negative implications. In the 1930s, research on the "Interference effect" began on the Empire State Building [21]. This topic has been the subject of ongoing research since then. Several investigations have demonstrated that assessing an isolated structure according to wind codes has rather adverse wind effects.

Interfering effects can arise from various factors, including the orientation, height, geometry, and wind direction of surrounding buildings in relation to the tall building. The presence of neighboring buildings can significantly influence the wind flow patterns and pressure distribution around the tall building, leading to changes in wind loads. These effects can be complex and vary depending on the specific location and configuration of the tall building and its surroundings. Firstly, the Interference Factor (IF) was introduced by Saunders and Melbourne [22] to measure the interfering effect. Figure 3 explains the interference effect of the upstream building.

![Interference Effect](image)

**Figure 3 - Interfering Effect**

### 3. Numerical Modeling of Fluid Flow Around a Building

The CFD approach is a technique used to solve the Navier-Stokes equations, which describes the motion of fluids in the numerical domain. The equations are solved in a numerical grid that represents the geometry of the building and the...
surrounding fluid domain. CFD models are capable of simulating various fluid phenomena, such as turbulence, flow separation, and vortices. They can also provide detailed information about the flow field, such as velocity and pressure distribution, and can be used to predict the wind loads acting on the building.

3.1 Available Numerical Approaches

There are several numerical approaches available in CFD for wind analysis. Some of the commonly used approaches are, RANS, LES, and DNS. RANS approach averages the equations governing fluid flow over time [23], resulting in a time-averaged solution. It is commonly used for predicting mean wind loads on buildings [24]. The LES approach resolves the largest turbulent scales of motion and models the smallest ones [25]. It is commonly used for predicting wind loads on buildings with complex shapes. DNS is a CFD approach that solves Navier-Stokes equations directly without using any turbulence model. According to the literature, DNS gives more accurate results but requires more computational power. Besides, RANS gives reasonably accurate results with less computational power [23]. The RANS approach is utilized in this study to model the turbulent flow due to the available limited computer facility.

3.2 RANS Approach

The RANS approach is a widely used method for simulating fluid flows in CFD. In RANS, the governing equations for fluid flow are averaged over time, and the turbulent stresses are modeled using turbulence models. Various turbulence models based on the Reynolds Average Navier-Stokes Equations are:

- Zero equation model (Mixing length model)
- One equation model (Sapalarit-Allmaras)
- Two equation models (k-ε models)
- Three equation model (k-ε-ϕ by Kawamato)
- Seven equation model (Reynolds stress model)

The selection of a turbulence model is based on factors such as computer resources, accuracy requirements, problem type, and simulation time, which are all relevant considerations in choosing an appropriate turbulence model for computational fluid dynamics (CFD) simulation [26]. The 2 equation k – ε turbulence model is widely used in wind engineering problems because of its high reliability. Furthermore, in numerical simulation, 2 equation k – ε, and 2 equation k – ω (SST) are recognized as better turbulence models because they forecast the net pressure distribution throughout the perimeter better than the other turbulence models [27].

3.3 CFD Simulation of Flow around a Building Structure by using Midas NFX

The Commonwealth Advisory Aeronautical Council (CAARC) standard tall building is being used as the pre-determined building to compare the reliability of turbulence models with experimental data. The selected prototype building is rectangular and prismatic, with dimensions of 100 ft x 150 ft (30.48 x 45.72 m) for its sides and a height of 600 ft (182.88 m) [8]. Figure 4 shows the shape and dimensions of the CAARC building.

Figure 4 - Shape and Dimensions of CAARC Building [8]

Figure 5 shows the wind tunnel test data that five different institutions, namely the University of Bristol -England (Bristol), City University -England (City), Monash University-Australia (Monash), and National Aeronautical Establishment-Canada (a & b) (NAE) conducted on CAARC standard tall buildings [8].

In Figure 5, L is the perimeter of the principal building and X* is the considered distance along the perimeter.

Figure 5 - Wind Tunnel Test Data of CAARC Building

```plaintext

Figure 5 shows the wind tunnel test data that five different institutions, namely the University of Bristol -England (Bristol), City University -England (City), Monash University-Australia (Monash), and National Aeronautical Establishment-Canada (a & b) (NAE) conducted on CAARC standard tall buildings [8].

In Figure 5, L is the perimeter of the principal building and X* is the considered distance along the perimeter.
```
3.3.1 Computational Domain
To prevent the computational domain of the surface from affecting the surface pressure over buildings, it is necessary to maintain a sufficient distance between the building surface and the computational boundary. To ensure accurate modeling of the flow around both low-rise and high-rise buildings, it is necessary to maintain a minimum length of 5 times the height of the building for the upstream and downstream zones [28]. In Computational Fluid Dynamics (CFD) simulations of tall buildings, several variables have been suggested and adopted. To achieve accurate simulations, it is typically advised to maintain a computational domain with a width and height that are 6 times and 2 - 2.5 times the building width and height, respectively. Additionally, the downstream and upstream zones should be of length approximately 2 - 2.5 times and 1 - 1.5 times the building height, respectively.

Figure 6 illustrates the development of the CAARC model on the Midas NFX platform, with a computational domain of 945 m x 855 m x 500 m (L x W x H). To ensure adequate space for wind development, the downstream and upstream distances were set at 732 m and 183 m, respectively. The computational setting must meet specific requirements to ensure that the numerical model closely resembles the real model and avoids blockage effects. The blockage ratio (δ) should not exceed 5%, and in this study, the blockage ratio was 1.89% as determined by Eq 1.

\[
\delta = \frac{A}{A_0} \quad ...(1)
\]

The exponential wind profile was chosen to model the wind behavior around the CAARC standard tall building, which stands at a significant height within the atmospheric boundary layer. In wind tunnel studies, the velocity profile of the atmospheric boundary layer can be characterized by two laws: power law and log law. The power law describes a steeper variation in wind speed than the log law [29]. For this study, the power law given in Eq 2 was utilized.

\[
\frac{U(Z)}{U_H} = \left(\frac{Z}{Z_H}\right)^a \quad ...(2)
\]

Eq 2 defines \(U(Z)\) as the wind speed at height \(Z\), where \(Z_H\) is the building height of 180 m with a wind speed of 12.7 m/s as \(U_H\). The wind profile coefficient, \(a\), which is set at 0.3, is also taken into consideration in this equation.

3.3.2 Boundary Conditions
The boundary conditions for the fluid domain were set as follows:

- The inlet face was specified along the x-axis with a given velocity profile function.
- The outlet face was set to zero pressure.
- The velocity component \(V_x\) was set to zero at the faces along the y-direction.
- The velocity component \(V_z\) was set to zero at the top surface of the computational domain.

A no-slip condition was applied to the bottom surface of the fluid domain, as well as where the building surfaces come into contact with the fluid domain [30, 31].

As shown in Figure 7, mesh was generated by setting the element size growth rate as 1.05 and ratio between minimum and maximum element size as 2, the density of air as 1.25 kg m\(^{-3}\), and the viscosity of air as 1.79x10\(^{-5}\) kg/m.s, turbulent intensity as 0.003.

3.3.3 Sensitivity Analysis for Different Mesh Sizes
In order to ensure accurate numerical results, it is essential to choose an appropriate grid size prior to conducting simulations. To assess the...
impact of grid resolution on wind pressure coefficients, both the grid size and the number of grids were adjusted. Four models were developed with varying degrees of grid resolution using the $2k - \varepsilon$ model. The resulting wind pressure coefficients were computed using Eq. 3.

$$C_{pi} = \frac{P_i - P_\alpha}{\frac{1}{2} \rho U_\alpha^2} \quad \ldots(3)$$

The wind pressure coefficient at a specific point $i$ is given by Eq. 3, where $C_{pi}$ is the mean wind pressure coefficient, $P_\alpha$ is the static pressure at the reference height, $\rho$ is the air density, which is 1.25 kg m$^{-3}$ in this study, and $U_\alpha$ is the wind speed at the reference height. At the top of the CAARC building located at 180 m [32,33], the wind speed is 12.7 m/s. The mean pressure coefficients on each surface at 2/3 height of the CAARC standard tall building are presented in Figure 8.

![Figure 8 - Sensitivity Analysis of Different Grid Sizes for CAARC Standard Tall Building](image)

To assess the accuracy of the results, the National Aeronautical Establishment-Canada (NAE-a) case was used for sensitivity analysis. To quantify the accuracy of the results, the mean absolute percentage error (MAPE), mean absolute deviation (MAD), and mean squared deviation (MSD) were calculated using Eqs. (4), (5) and (6), respectively [34].

$$MAPE = \frac{\sum_{i=1}^{n} |C_{pitunnel} - C_{picFD}|}{n} \times 100\% \quad \ldots(4)$$

$$MAD = \frac{\sum_{i=1}^{n} |C_{pitunnel} - C_{picFD}|}{n} \quad \ldots(5)$$

$$MSD = \frac{\sum_{i=1}^{n} (C_{pitunnel} - C_{picFD})^2}{n} \quad \ldots(6)$$

Based on Table 1, it can be inferred that achieving a grid resolution of 0.5 m on the building surface would meet the necessary numerical accuracy standards.

### Table 1 - Sensitivity of Wind Pressure Coefficients to Grid Resolution

<table>
<thead>
<tr>
<th>Grid</th>
<th>Error (MAPE)</th>
<th>Error (MAD)</th>
<th>Error (MSD)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0m</td>
<td>14.10</td>
<td>0.08</td>
<td>0.01</td>
</tr>
<tr>
<td>1.5m</td>
<td>14.97</td>
<td>0.10</td>
<td>0.02</td>
</tr>
<tr>
<td>2.0m</td>
<td>15.01</td>
<td>0.10</td>
<td>0.02</td>
</tr>
</tbody>
</table>

3.4 Selection of the Most Reliable RANS Turbulence Model

The experimental data from wind tunnels were compared to each turbulent model independently. The distribution of $C_{pi}$ obtained using different turbulence models; $0$ equation model, $1$ - equation $k - \varepsilon$ model, $2$-equation $k - \varepsilon$ model, $2$ - equation $k - \omega$ model and $2$ - equation $k - \omega$ SST model were compared with the experimental data. Table 2 shows three errors calculated using Eqs. (4), (5) and (6). According to those results, the $2k - \omega$ SST model shows minimum MAPE, MAD and MSD values. The comparison results for the above five turbulence models are shown in Figure 9.

### Table 2 - Comparison between Numerical Results and Experimental Results

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>MAPE</th>
<th>MAD</th>
<th>MSD</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0$ eq</td>
<td>14.10</td>
<td>0.08</td>
<td>0.01</td>
</tr>
<tr>
<td>$1$ eq</td>
<td>15.01</td>
<td>0.10</td>
<td>0.02</td>
</tr>
<tr>
<td>$2$ eq</td>
<td>19.40</td>
<td>0.13</td>
<td>0.07</td>
</tr>
<tr>
<td>$2k - \omega$ SST</td>
<td>11.88</td>
<td>0.07</td>
<td>0.01</td>
</tr>
</tbody>
</table>

![Figure 9(a)](image)

![Figure 9(b)](image)
experience higher wind effects compared to other common plan shapes [35,36]. Therefore, for this study, building A has fixed geometries including length, width, and height of 24 m, 24 m and 150 m, respectively. The orientation ($\theta = 0^\circ, 45^\circ, 90^\circ$), height ($h = 50 m, 100 m, 150 m, 175 m$) and shape of building B will be changed while keeping the plan areas of the selected shapes remaining the same. As shown in Figure 10, building A is a fixed building, and the location of building B will be changed with the changing of the orientation. $X$ is the length between building A and building B, and it has a fixed value of 25 m (Minimum separation distance between tall buildings on the same site of 25 m or greater) [37]. $\theta$ is the angle between the two buildings.

For the parametric study, the mesh size was chosen as 0.5 m and the turbulence model as $2k-\omega$ SST, based on the previous analysis results. As shown in Figure 11, a total of 48 combinations were studied to identify their interfering effects. The responses (pressure distribution, interfering factor, base shear, and overturning moment) are compared in this study. First, the responses on the principal building were evaluated while keeping one angle and height unchanged for different plan shapes. From that, the critical shape which gives the highest interfering effect for that height and the angle was identified. Subsequently, the influence of angle was compared. Finally, using those results, the critical angle for the selected shape and the height was identified.
4.1 Base Moment Comparison

The maximum deviation was obtained at the 0-degree 175 m case with the triangular shape. To represent that, the base moment values of the 0 degree and 175 m cases were compared for all shapes and the comparison is shown in Figure 12. Deviation was calculated based on Eq. (7). The results indicate that the triangular shape has the maximum deviation of 94.67%, while the square shape has the minimum deviation of 88.38%.

\[
\text{Deviation} = \left| \frac{\text{Isolated condition} - \text{Interfering condition}}{\text{Isolated condition}} \right| \times 100\% \quad \ldots (7)
\]

Figure 11 - All the Combinations of Parametric Study

![Image of parametric study combinations](image)

4.2 Base Shear Comparison

The comparison of base shear for the 0-deg and 175 m cases for all shapes is shown in Figure 14. The results indicate that the circular shape has the maximum deviation of 99.62%, while the square shape has the minimum deviation of 88.38%.

![Image of base shear comparison](image)

Figure 12 - Base Moment due to Wind Loading for 0-deg 175 m Case

Next, results were compared for the triangular shape case with different angles, as shown in Figure 13. In this case, the maximum deviation of 94.67% is shown in the 0-deg case and the minimum deviation of 32.38% is shown in 90-deg case.

![Image of base moment comparison](image)

Figure 13 - Base Moment due to Wind Loading for 175 m Triangular Shape Case

![Image of base shear comparison](image)

Figure 14 - Base Shear due to Wind Loading for 0-deg 175 m Case
Next, results were compared for the circular shape 175 m case with different angles, as shown in Figure 15. In that, the maximum deviation of 99.62% is shown in 0-deg case and the minimum deviation of 0.19% is shown in 45-deg case.

According to base shear results, a maximum deviation of 94.67% is shown in the triangular 0-deg 175 m case and a minimum deviation of 0.04% is shown in the circular 0-deg 50 m case.

Based on the results, the maximum increments of the base moment and base shear in the triangular 90-deg and 175 m case were observed, and they were 32.38% and 31.30%, respectively. For all shapes at 0 degrees, the interfering effect initially increased with the height of the interfering building until it reached the height of the principal building. After that, the interfering effect decreased with further increases in the height of the interfering building.
4.3 Pressure Variation Comparison

Next, this work extended to study the variation of the mean $C_p$ along the height at the centerline of the windward face for each defined combination. Figure 18 shows the results of the analysis, which reveal that the presence of an interfering building causes a drastic variation in $C_p$ along the height of the building for all shapes. Maximum variation is shown in triangular shape and minimum variation shown in circular shape.

As shown in Figure 19, the variation of $C_p$ along the height of the interfering building differs significantly for different orientations in the case of a square shape with a height of 150 m.

The maximum variation is observed in the 0-deg orientation of the interfering building, while the minimum variation is observed in the 90-deg orientation of the interfering building.

As shown in Figure 20, higher interfering buildings create larger negative pressure zones. The angle between buildings also impacts the pressure, with larger angles resulting in positive pressure on the windward face of the principal building.

As shown in Figure 21, the windward face is positive, and the leeward and sidewalls are negative for the isolated model. However, when considering 150 m square shape cases where a building interferes with the isolated model, the pressure coefficient behaviour changes dramatically.

Figure 18 - $C_p$ Variation with Building Height in Windward Face of Principal Building: 150 m 0-deg Case

Figure 19 - $C_p$ Variation with Building Height in Windward Face of Principal Building: 150 m Square

Figure 20 - Pressure Variations for Triangular Shape Interfering with Building with Different Angles and Height

<table>
<thead>
<tr>
<th>Triangular shape</th>
<th>0°</th>
<th>45°</th>
<th>90°</th>
</tr>
</thead>
<tbody>
<tr>
<td>50m</td>
<td><img src="image" alt="Sectional view" /></td>
<td><img src="image" alt="Plan view" /></td>
<td><img src="image" alt="Sectional view" /></td>
</tr>
<tr>
<td>100m</td>
<td><img src="image" alt="Sectional view" /></td>
<td><img src="image" alt="Plan view" /></td>
<td><img src="image" alt="Sectional view" /></td>
</tr>
<tr>
<td>150m</td>
<td><img src="image" alt="Sectional view" /></td>
<td><img src="image" alt="Plan view" /></td>
<td><img src="image" alt="Sectional view" /></td>
</tr>
<tr>
<td>175m</td>
<td><img src="image" alt="Sectional view" /></td>
<td><img src="image" alt="Plan view" /></td>
<td><img src="image" alt="Sectional view" /></td>
</tr>
</tbody>
</table>
Figure 20 - Vertical Distribution of Pressure Variations for Triangular Shape Interfering with Building with Different Angles

Figure 21 - Pressure Variation around the Principal Building at the 2/3 Height: 150 m Square Shape Cases

Figure 22 - Pressure Variation around the Principal Building at the 2/3 Height for 0-deg 150 m Cases

5. Conclusions

In this research, Computational Fluid Dynamics (CFD) was used to qualitatively and quantitatively study the effect of an upstream building configuration on a downstream building. From the several turbulence models available in the RANS approach, 0 equation model, 1 - equation $k - \varepsilon$ model, 2 - equation Standard $k - \varepsilon$ model, 2 - equation $k - \omega$ model, and 2 - equation $k - \omega$ SST model were compared to predict the pressure variations around the building. From those models, the $2k - \omega$ SST model shows the least error (MAPD, MAD and MSD). It can be concluded that the $2k - \omega$ SST model is the most reliable turbulence model for this study. After validating and selecting the best turbulence model from the above models, the CFD analysis was performed to identify the interference effect on a square principal building from the upstream building with different shapes, namely, square, circular, cross, and triangular, with different orientations and heights. This study looked at the base moment, base shear, and pressure fluctuation of the principal building to assess the impact of the upstream building considering the interfering effects.

Based on the results, the maximum increments of the base moment and base shear in the triangular 90-deg and 175 m case were observed, and they were 32.38% and 31.30%, respectively. The results suggest that a safety factor for interfering effects should be considered in designing the building structures in the city area to ensure the stability of a building. And the maximum Cp variation in the triangular shape case was observed, and it was positive 0.8 to negative 0.3. The findings emphasized the significance of designing the cladding system's connection to both compression and tension forces. The results of the present study will be crucial in ensuring the stability and safety of the building structures when those buildings are going to be developed in a dense building environment.

For a particular orientation of the upstream building, the interference effects may change with the distance between the principal building and the interfering building. Also, it is recommended to consider the height of buildings beyond 200 m; this study is focused on the building height less than 200 m. Further, if there is a chance to conduct analysis by using LES or DNS other than RANS as a numerical method, it may give a more accurate prediction flow around the objects.

References


