CFD Study of Flow Characteristics and Pressure Distribution on Re-Entrant Wing Faces of L-Shape Buildings

Arun Kumar*, Ritu Raj

Department of Civil Engineering, Delhi Technological University (DTU), India

Received September 23, 2021; Revised December 22, 2021; Accepted January 16, 2022

Cite This Paper in the following Citation Styles

(a): [1] Arun Kumar, Ritu Raj, "CFD Study of Flow Characteristics and Pressure Distribution on Re-Entrant Wing Faces of L-Shape Buildings," *Civil Engineering and Architecture*, Vol. 10, No. 1, pp. 289-304, 2022. DOI: 10.13189/cea.2022.100125.

(b): Arun Kumar, Ritu Raj (2022). *CFD Study of Flow Characteristics and Pressure Distribution on Re-Entrant Wing Faces of L-Shape Buildings*. *Civil Engineering and Architecture*, 10(1), 289-304. DOI: 10.13189/cea.2022.100125.

Copyright©2022 by authors, all rights reserved. Authors agree that this article remains permanently open access under the terms of the Creative Commons Attribution License 4.0 International License

Abstract The purpose of this research is to investigate the wind flow pattern and pressure distributions on the re-entrant wing facades of an irregular L-cross sectional shape model over the range of wind incidence angles. Computational simulation with ANSYS (CFX) solver has been used for the study on model of the building of cross-sectional area 300m² and height 50m at a length scale of 1:100. ANSYS (CFX) is a finite element modelling CFD program of pressure-based solver technology suitable for low Mach No. (Ma) fluid flow. Homogeneous flow of steady wind with gustiness of 5% turbulence in boundary layer of atmosphere is taken for the study. Investigation has been conducted for a range of wind angle of attack from 0° to 180° @ 15° interval. The findings were confirmed by examining similar wind flow on a rectangular model of same cross-sectional area and height under same boundary conditions as applied to L shape model. The results were compared with the values given in various international wind design codes/standards. For a better understanding of the flow characteristics around the L shape model in terms of flow stagnation, flow separation, creation of wakes & vortices and reattachment of flow; illustrations of flow patterns around the model for certain specific wind incidence angles are presented. On the basis of study of surface pressure generated on the model envelope, coefficient of pressure on the re-entrant corner faces has been discussed. The pressure on these wing faces has been found critical between 75° & 180° wind incident angles.

Keywords Wind Pressure Coefficients, ANSYS (CFX); L-plan Shape, Standard k-ε Model

1. Introduction

With the rising use of new façade assemblies and unique shapes in the design of high-rise buildings, a precise description of wind action and its interaction with building shape is essential for a rational structural design approach. While considerable study has been done on regular cross-sectional shape buildings, on the basis of which many international wind design codes have been formulated, few research studies have been found on irregular-shape buildings.

A general estimation of magnitude of wind pressure on surfaces of a bluff body is the mean pressure coefficient (C_{pe}), which is pressure normalised by upwind velocity/dynamic pressure ($\frac{1}{2}\rho u^2$) at roof height, where ρ is density of air and u is mean wind velocity at roof height. It is a dimensionless measure independent of the velocity of flow. However, it depends on the shape and size of the bluff body for a similar wind flow characteristic. The oblique wind also causes a varied wind flow pattern around the body, as well as varying impact pressure on the surfaces themselves.

Baines [1] was the first to represent the influence of wind pressure distribution and flow pattern on tall

buildings. Positive pressure on the windward surface and negative pressures on the side surfaces, top and rear surface of a square-shape building were intuitively depicted as the contour of the mean wind pressure. Data for designing such simple-shaped buildings are available in various international codes/standards. But more complex tall buildings have necessitated study of the effect of wind on such buildings through available experimental research in wind tunnels. Computational simulations and field measurements on full scale buildings have also been conducted by other researchers. Various researchers have published their study publications on the topic of the wind effect on irregular-shape tall buildings. Distribution of pressure on various surfaces was observed to be different from that of cube-shape model. Mendis et al. [2] investigated the interference effect and also along and across wind impacts for high rise buildings, using wind tunnel testing and CFD. The experimental and analytical results were found to be within the acceptable limit of 20 percent to 25 percent. Fu et al. [3] published field measurements of boundary layer wind parameters for two super tall buildings across open country terrain and urban terrain. Full scale measurements were compared to wind tunnel test data and the results were found to be within acceptable range (20% - 25%). Tanaka et al. [4] used wind tunnel measurements to show the aerodynamic properties of different irregular cross-sectional shaped high-rise buildings. Amin and Ahuja [5] studied effect of various side ratios ranging from 0.25 to 4 on a rectangular model of similar area and height. The negative pressure on the side and downwind faces of the building is reported to be affected by wind incidence angles and side ratios. The wind pressure on the building faces is substantially affected by the cross-sectional shapes of the building, according to Kushal et al. [6]. The computational investigation of distribution of pressure on faces of tall building of square cross-section under three wind incidence angles is presented by Chakraborty and Dalui [7] using ANSYS (Fluent) solver. ANSYS (Fluent) is a finite volume modelling based CFD program having facility of customizable tools for study of complex fluid flow. Three wind incidence angles 0° , 30° and 45° were investigated and discussed by them. Through wind tunnel tests at various wind angles, Bhattacharyya et al. [8] investigated the distribution of pressure on the surfaces of an E-shape tall building. Chakraborty et al. [9] discussed result of wind effect on a plus shape tall building model conducted through wind tunnel study and simulation as well and; compared with the results for 0° and 45° wind angles. Differences in coefficient of pressure between CFD study and wind tunnel study were found to be within the acceptable range. Kheyari and Dalui [10] used CFD to investigate the wind forces on a high-rise building under interference effects. Roy and Bairagi [11] evaluated the pressure and velocity fluctuation of wind around stepped high-rise building with rectangular, square and triangular

shapes placed one above another. Mukherjee and Bairagi [12] studied the pressure and velocity distribution of wind around N shape tall building. Bairagi and Dalui [13] examined distribution of pressure on square tall building having a reduced offset of 20% at half the height of building and a reduced offset of 10% at one third height and two third height of building, each. For a 100 percent blockage condition, Pal S. et al. [14] assessed the interference effect of square and remodeled triangular shape buildings of equal plan area. The shape and orientation of interfering building are shown to have a significant role on pressure and force generated on main building. Through open circuit boundary layer experiments, Raj, Ritu, and Ahuja, A.K. [15] compared plus shape rigid high-rise building models of varying wing lengths with a square model having same cross-sectional area. It was reported that shear force at base, torque about the two vertical planes and about the plan depend on wind incidence angles as well as the wing lengths in plus shape buildings. Raj, Ritu, et al. [16] used ANSYS (CFX) solver to simulate H shape high-rise building model in isolated and interfering condition using the same shape model at various positions. In an open wind tunnel Pal S. et al. [17] studied wind generated mutual interference effect on twin square and fish shape tall building models placed at a distance of $0.1H$ (H being height of the model) with various combinations of orientation between forward-facing and back-facing and presented induced pressure and base shear on the models. S.K. Nagar et al. [18] evaluated the coefficient of pressure and interference factor for different interfering situations of square and H shape tall buildings of same floor area and discovered that H shape building is under more pressure than the square building. Kumar A, Raj R [19] studied pressure distribution on an irregular octagonal (oval) shape building using ANSYS (CFX) for varied wind incidence angles. It was reported that the flow characteristics is influenced by side ratio of the building and that inclined surfaces of the building are subjected to high pressure gradient.

Despite the fact that L shape is a highly typical arrangement, most of studies on such plan-shaped buildings have been limited to conventional wind directions. Stathopoulos T. [20] has examined the wind induced pressure for L shape of low-rise model through computational method for normal wind direction. The results between computational and experimental study were reported to be with good agreement between the two. Gomes et al. [21] used a wind tunnel test as well as CFD to evaluate wind pressure distribution on the facades of L and U shape buildings of identical height. They compared the results from a cube of same dimension and height and found different pressure distributions on the models than those of single cube. Amin and Ahuja [22] investigated pressure distribution on various sides of L and T shape tall buildings for varied wind angles in a wind tunnel test. The cross-sectional shape of the buildings was revealed to play

an important role in pressure distribution, particularly on the re-entrant wing side.

The pressure distribution on re-entrant faces depends on the respective lateral dimension (width of the faces) and height of the building. The influence of wind incidence angles on pressure distribution on faces of a L shape model (figure 1) of equal re-entrant wing faces for a prototype building of 50 m height and 300 sqm floor area have been investigated in the present study using a standard k-ε turbulent model on ANSYS (CFX) solver.

Figure 1. L Shape Model

2. Numerical Analysis

Navier-Stokes equations of (a) Continuity and (b) Momentum which governs the pressure and velocity in three-dimensional unsteady fluid flow are expressed as:

(a) Continuity equation

$$\frac{\delta u}{\delta x} + \frac{\delta v}{\delta y} + \frac{\delta w}{\delta z} = 0 \quad (1)$$

(b) Momentum Equation

X - Momentum

$$\begin{aligned} \rho \left(\frac{\delta u}{\delta t} + u \frac{\delta u}{\delta x} + v \frac{\delta u}{\delta y} + w \frac{\delta u}{\delta z} \right) = \\ = (-) \frac{\delta \rho}{\delta x} + \rho g_x + \mu \left(\frac{\delta^2 u}{\delta x^2} + \frac{\delta^2 u}{\delta y^2} + \frac{\delta^2 u}{\delta z^2} \right) \end{aligned} \quad (2)$$

Y - Momentum

$$\begin{aligned} \rho \left(\frac{\delta v}{\delta t} + u \frac{\delta v}{\delta x} + v \frac{\delta v}{\delta y} + w \frac{\delta v}{\delta z} \right) = \\ = (-) \frac{\delta \rho}{\delta y} + \rho g_y + \mu \left(\frac{\delta^2 v}{\delta x^2} + \frac{\delta^2 v}{\delta y^2} + \frac{\delta^2 v}{\delta z^2} \right) \end{aligned} \quad (3)$$

And Z - Momentum

$$\begin{aligned} \rho \left(\frac{\delta w}{\delta t} + u \frac{\delta w}{\delta x} + v \frac{\delta w}{\delta y} + w \frac{\delta w}{\delta z} \right) = \\ = (-) \frac{\delta \rho}{\delta z} + \rho g_z + \mu \left(\frac{\delta^2 w}{\delta x^2} + \frac{\delta^2 w}{\delta y^2} + \frac{\delta^2 w}{\delta z^2} \right) \end{aligned} \quad (4)$$

Where ρ is density and g 's are sum of body forces in the respective directions. Rest symbols are carrying usual meanings.

In computational fluid dynamics (CFD), wind simulation is created in virtual wind tunnel called domain within which model is kept alike as in the wind tunnel experiment. CFD models try to resolve the fluid flow around any bluff body, as specified by the aforementioned Navier-Stokes Equations, by modelling the flow at continuous finite grid locations. Researchers have created a number of mathematical models to define the flow properties and the influence of turbulence in the fluid flow. To replicate the near-real scenario that occurs in nature, a number of models have been developed. However, no model for determining the precise characteristics of turbulent flow has yet been developed. k-epsilon (k-ε) model is the most common among them for CFD study of wind flow on buildings. The actual k-ε turbulent model comprises several unmeasurable variable quantities and unknowns. This model has been further simplified (standard k-ε turbulence model) in which turbulence is described by two partial differential energy equations- Turbulence Kinetic Energy (k) and; dissipation of turbulence kinetic energy called the Turbulent Eddy Dissipation (ε). It is the simplest model, requiring the input of only the initial boundary conditions to the solver. Fluid shear, friction, buoyancy, and tiny eddies contribute to the Turbulent Kinetic Energy equation. Mathematically, it is defined as the mean kinetic energy per unit mass of eddies having dimension [L²T⁻²]. Physically, the Turbulence Kinetic Energy is the Root Mean Square (RMS) velocity fluctuations. Turbulence Kinetic energy is converted to turbulent energy, which dissipates (Turbulent Eddy Dissipation, ε) at the Kolmogorov micro scale by viscous forces, resulting in heat production. Dimension of Turbulent Eddy Dissipation, ε is [L²T⁻³] i.e., Turbulent Kinetic Energy per unit time. The energy equations are as follows.

(c) Turbulence Kinetic Energy

$$\frac{\delta(\rho k)}{\delta t} + \frac{\delta}{\delta x_i} (\rho u_i k) = \frac{\delta}{\delta x_j} \left(\frac{\mu_t}{\sigma_k} \cdot \frac{\delta k}{\delta x_j} \right) + 2\mu_t \cdot E_{ij} \cdot E_{ij} - \rho \epsilon \quad (5)$$

(d) Turbulence Eddy Dissipation

$$\begin{aligned} \frac{\delta(\rho \epsilon)}{\delta t} + \frac{\delta}{\delta x_i} (\rho u_i \epsilon) = \frac{\delta}{\delta x_j} \left(\frac{\mu_t}{\sigma_\epsilon} \cdot \frac{\partial \epsilon}{\partial x_j} \right) + \\ + \frac{\epsilon}{k} C_{\epsilon 1} \cdot 2\mu_t \cdot E_{ij} \cdot E_{ij} - C_{\epsilon 2} \rho \frac{\epsilon^2}{k} \end{aligned} \quad (6)$$

Where, u_i is velocity component in respective directions.

E_{ij} is rate of deformation i.e., strain rate.

μ_t is Eddy Viscosity and is represented as $\mu_t = C_\mu \rho \frac{k^2}{\epsilon}$

σ_k , σ_ϵ , $C_{\epsilon 1}$ and $C_{\epsilon 2}$ are adjustable constants. Values of these constants have been determined after multiple iterations of data fitting for a variety of turbulent flows as;

$$C_\mu = 0.09, \quad \sigma_k = 1.00, \quad \sigma_\varepsilon = 1.3, \\ C_{\varepsilon 1} = 1.44 \text{ and } C_{\varepsilon 2} = 1.92$$

3. Mean Velocity Characteristics

Wind does not flow evenly in nature. Due to friction generated by vegetation and structures at ground level, wind speed increases as height from the earth increases in a semi empirical relationship. As a result, the wind flow encountered in high-rise buildings is not constant along the height of building. The vertical distribution of horizontal mean wind flow is defined by the following empirical equations.

1) Parabolic Law:

$$u = u_{Ref} \sqrt{\frac{Z+22}{Z_{Ref}+22}} \quad (7)$$

Where, u_{Ref} is the reference wind speed in m/s

Z_{Ref} is the reference height usually taken as 10 m above ground.

u is longitudinal wind velocity (time averaged) at height Z above ground.

2) Power Law:

$$u = u_{Ref} \left(\frac{Z}{Z_{Ref}} \right)^\alpha \quad (8)$$

Where, value of α depends upon terrain category.

This is an upgraded formula over Parabolic Law. It is generally used for its simplicity. But it does not provide correct results in the bottom 10 m of ABL.

3) Logarithmic Law:

$$u = \frac{1}{k} u_0 L_n \left(\frac{Z-Z_d}{Z_0} \right) \quad (9)$$

Where, k is Von Karman Constant = 0.4

u_0 is Friction Velocity represented as $u_0 = \sqrt{\frac{\tau_w}{\rho}}$, τ_w

is wall shear stress and is represented as $\tau_w = \mu \left(\frac{du}{dy} \right)_{y=0}$

and; ρ is density of air.

Z_0 is aerodynamic roughness length which is a surface roughness parameter.

Z_d is the height of zero wind speed achieved above the ground.

In the bottom 10 meters of the earth are covered by the Logarithmic Law for the vertical distribution of horizontal mean wind speeds.

4. Model and Computational Domain

For computational simulation model is positioned inside a computational domain which is equivalent to a wind tunnel. The dimensions of the domain are created large enough to eliminate fluid stream reflections so that

any unusual fluid pressure adjacent to and on the model is avoided and; the blockage ratio remains below 3 %. At the same time, it is ensured that during the simulation, velocity variations, uplift forces, backwash and vortex formations in the wake zone are adequately formed. The domain size should also not be too large, since this would need greater number of grid elements for analysis, which will take longer time and demand more processing power for the solution to converge. In the current investigation recommendations of Revuz, J. et al. [23] have been adopted for the purpose. The domain inlet wall, side walls and the domain roof wall are kept at a distance of five times the height of the model from the respective faces of the model. The distance of domain outlet from the model leeward face is fixed at fifteen times the height of the model in order to produce a realistic wake and vortex behind the model. The domain length, $L = 10.3$ m, width, $B = 5.3$ m and height, $H = 3.0$ m created and model built with a scale of 1:100 positioned inside the domain is shown in figure 2. Homogeneous steady wind flow under atmospheric boundary layer at the inlet using power law wind profile with roughness coefficient, $\alpha = 0.143$ for terrain category-II [as per IS: 875 (Part 3): 2015] has been adopted. The gustiness effect is achieved by providing 5% turbulence intensity. 0.63 m/s uniform flow of wind velocity at the roof height of the model is attained. Figure 3 depicts the obtained velocity profile at the inlet.

Figure 2. Schematic Diagram of Domain

Figure 3. Velocity Profile

5. Validation

Figure 4 shows a rectangular plan model having same area in plan and same height as the L-plan shape model. Simulation results on this model under the same wind flow and boundary conditions were used to test the validity of the numerical technique investigated. The findings of the coefficient of pressure on the rectangular plan shape model faces were matched with several international codes/standards. The results are shown in

table 1 and are within acceptable limits, particularly for the upwind and downwind faces. Due to the anisotropic nature of turbulence and eddy viscosity, some larger discrepancies appeared on the side faces, whereas isotropic nature of boundary condition is used in the simulation. The numerical simulation, however, has caught the primary flow features, addressed later, which justifies the approach for further research on the L-plan shape model.

Figure 4. Rectangular Shape Model

Table 1. Comparison of C_{pe} on Faces of Rectangular Model

As per	Wind Angle	C_{pe} Face A	C_{pe} Face B	C_{pe} Face C	C_{pe} Face D
ANSYX (CFX)	0°	+0.68	-0.28	-0.48	-0.48
	90°	-0.43	-0.44	0.71	-0.23
IS: 875 (Part 3): 2015	0°	+0.8	-0.25	-0.8	-0.8
	90°	-0.8	-0.8	0.8	-0.25
ASCE/SEI 7-10	0°	+0.8	-0.5	-0.7	-0.7
	90°	-0.7	-0.7	0.8	-0.5
AS/NZS- 1170.2 (2002)	0°	+0.8	-0.5	-0.65	-0.65
	90°	-0.65	-0.65	0.8	-0.5
EN: 1991-1-4	0°	0.8	-0.55	-0.8	-0.8
	90°	-0.8	-0.8	0.8	-0.55
BS: 6399-2	0°	0.76	-0.5	-0.8	-0.8
	90°	-0.8	-0.8	0.76	-0.5

6. Computational Grid, Boundary Conditions

Discretized grid of computational domain and the model have a significant impact on the result of the simulation. The grid resolution is set to precisely capture crucial physical flow parameters on the model surfaces, flow separations, vortices generated and reattachment of flow etc. The faces of the model have been discretized at a finer level than the volume of domain, since our focus is to scan the model precisely. CFD gives good results if discretization is done with finer cell elements. However, when the cell numbers are increased in number computational duration are also increased, which may require more computational resources [24]. Hence, a tradeoff between the two is maintained. Various attempts of meshing techniques and boundary conditions were explored during the present study. By using the option of automatic method in the global setting, final meshing of domain volume with maximum element size of 90 mm was adopted. From the local setting on domain ground surface maximum element size was kept as 40 mm, and model walls & roof as 20 mm. Smooth inflation is applied to the model such that the domain grid meets the model walls orthogonally, allowing for a smooth simulation transition on the surfaces. Figure 5 depicts meshing. The total number of nodes produced is around 465000 and the number of elements is 2550000. The domain walls except ground and the model roof were defined as free slip walls ($\tau_{wall} = 0$; $u_{wall} = 0$), where τ_{wall} is representing fluid shear stress on wall and u_{wall} is representing component of fluid velocity normal to the wall. This means, component of fluid velocity parallel to wall has a predetermined value, which is computable. But,

component of velocity normal to wall as well as fluid shear on wall, are both zero. The model faces and the domain ground were specified as no slip walls ($u_{wall} = 0$), which means component of fluid velocity at the boundary of wall is zero and the wall is not moving i.e., wall velocity is also zero. To obtain different angle of attack of wind, model was rotated about its mass center of gravity, keeping the model boundary and flow parameters unchanged.

Figure 5. Meshing

Figure 6 (a). C_{pe} Contour on Faces of Rectangular Model for 0° Wind Angle

Figure 6 (b). C_{pe} Contour on Faces of Rectangular Model for 90° Wind Angle

Figure 7. Streamlines at Mid Height of Rectangular Model

7. Result and Discussion

7.1. Rectangular Model

Figure 6 (a) and (b) shows the mean pressure coefficients for the rectangular model for the two orthogonal wind directions. The mean pressure coefficients are positive at the stagnation points on the respective windward faces of wind orientation (face A for 0° wind angle & face C for 90° wind angle). The maximum values are in the upper heights of the respective windward faces and symmetrical from the vertical center line. Due to an increase in wind velocity along the height, the pressure distribution also increases along height. The flow separation takes place from the leading edges where wind flows sharply away at high speed. The pressure decreases creating negative pressure or suction pressure on side faces and roof. The leeward faces (face B for 0° wind angle & face D for 90° wind angle) are under almost uniform suction as expected. However, the pattern of the suction pressure distribution for both wind directions is different. For 0° wind angle suction is the minimum at the center of the leeward face whereas for 90° wind angle, it

is at the lower height of the leeward face. This is due to difference in intensity of horizontal and vertical vorticity generated in the wake region on account of different side ratio. The wake area on the leeward face for 0° wind angle & 90° wind angle can be seen in figure 7 (a) and (b).

7.2. L-Plan Shape Model

Coefficients of pressure on the faces of L-plan shaped model have been shown in figure 8 (a) to (m) for extended range of wind incident angles from 0° to 180° @ 15° . Following are the overall features and observed pressure distribution on these faces.

At 0° , the wind flow around the model is very similar to that of the rectangular model in the windward region. But the flow pattern is totally different in the leeward region. Unsymmetrical vortices, figure 9 (a), are formed behind the model with a smaller vortex in the L cavity behind face B2, whereas in the rectangular model two similar vortices are formed, figure 7 (a), behind the model. The upwind face A is subjected to similar positive mean coefficient of pressure of 0.68/0.69 for both the models - rectangular and L-plan shaped respectively. The remaining faces are under negative pressure.

Figure 8 (a). C_{pe} Contour on Faces for 0° Wind Angle Figure 8 (b). C_{pe} Contour on Faces for 15° Wind Angle Figure 8 (c). C_{pe} Contour on Faces for 30° Wind Angle Figure 8 (d). C_{pe} Contour on Faces for 45° Wind Angle

Figure 8 (e). C_{pe} Contour on Faces for 60° Wind Angle Figure 8 (f). C_{pe} Contour on Faces for 75° Wind Angle Figure 8 (g). C_{pe} Contour on Faces for 90° Wind Angle Figure 8 (h). C_{pe} Contour on Faces for 105° Wind Angle

Figure 8 (i). C_{pe} Contour on Faces for 120° Wind Angle Figure 8 (j). C_{pe} Contour on Faces for 135° Wind Angle Figure 8 (k). C_{pe} Contour on Faces for 150° Wind Angle Figure 8 (l). C_{pe} Contour on Faces for 165° Wind Angle

Figure 8 (m). C_{pe} Contour on Faces for 180° Wind Angle

(a): Streamlines at 0° Wind Angle

(b): Streamlines at 45° Wind Angle

(c): Streamlines at 90° Wind Angle

(d): Streamlines at 135° Wind Angle

(e): Streamlines at 180° Wind Angle

Figure 9. Streamlines L-Plan Shaped Model

Between wind incident angles of 0° and 60° , the inner faces B2 and C1 are under the wake region, figure 9 (a) and (b), and hence coefficient of pressure on these faces

are negative. Peak negative pressure coefficient (-1.50) occurs at wind incident angle of 45° on face B2. At wind incident angle of 75° the inner part near the re-entrant

corner of face C1 is subjected to suction due to formation of vortex in the region, but the outer part is subjected to positive pressure as the wind hits that region directly after separating from the outer edge of the wing face B2. Highly fluctuating pressure exists on both the inner faces B2 and C1 at wind incident 75° , figure 8 (f).

At 90° wind angle the inner faces B2 and C1 are subjected to positive pressure, figure 9 (c). On the inner part of face C1, positive pressure exists due to direct impact of wind. However, at the leading-edge suction occurs from where the wind separates. On face B2 the pressure is positive on its entire extent due to stagnation of flow at the re-entrant corner and recirculation of air from face C1. At wind incident angles between 105° and 135° the inner faces B2 and C1 are subjected to high wind pressure at the re-entrant corner. Variation of mean C_{pe} with wind incident angles has been drawn in figure 10. At an angle of 135° a large area of stagnation occurs at the re-entrant corner. The size of this area of stagnation depends upon the height and the relative dimension of the wing faces B2 and C1. Under uniform dimension of the wing faces, the area of this stagnation zone increases with the increase in the height [25]. This is owing to the fact that the flow has a tendency to contour the sides rather than flow into the cavity. Greater the wing length of face B2, higher the blockage and hence greater the positive pressure on the wing face C1. It is a unique characteristic of L-plan shape building.

At wind incident angle greater than 135° face B1 starts experiencing positive pressure. As the wind incident angle increases, the positive pressure on face B1 increases. Peak negative coefficient of pressure on face B1 is observed to be -1.67 at 105° wind angle. At an angle of 75° both the wing faces B2 and C1 are experiencing positive pressure and continue to be positive owing to the re-entrant corner effect. On face C2 suction occurs at 150° wind angle and

continues to be up to 180° wind angle. However, peak negative coefficient of pressure is observed on face C2 to be -1.77 at 15° wind angle.

At 180° wind incident angle both the upwind faces B1 and B2 are subjected to positive pressure. But the pressure distribution is not symmetrical about the vertical center line of the faces, figure 8 (m), unlike in the case of rectangular model. The pressure increases from the left edge of face B1 towards its right edge i.e., towards the re-entrant corner of wing faces B2 and C1. This can be credited to blockage of air flow by the wing face B2 and stagnation of wind at the re-entrant corner. On face B2 positive pressure also exists being the windward face except at the leading edges from where flow separation takes place. The pressure increases along the height due to increase in wind velocity along the height. The phenomenon also affects the wing face C1 and positive pressure exists on its entire extent unlike in the rectangular model where suction occurs on the side faces. Due to separation of wind from the leading edges, the side faces D and C2 are subjected to suction pressure progressively increasing from the near edge to the far edge. The downwind face is subjected to suction. It can be observed that the effect of recirculation on the face is not symmetrical, figure 9 (e), unlike in the rectangular model.

The influence of wind incident angles on the wing faces B2 and C1 is shown in figure 11 in terms of central vertical C_{pe} variations on faces. For clarity and understanding few wind incident angles are shown. It is observed that the pressure values on these faces are having a great influence of wind angles. It is noticed that there is little variation on suction along height whereas the positive pressure coefficient increases with height. The maximum value of C_{pe} is found to be above the height of $0.8 Z$ where Z is the height of the model.

Figure 10. Variation of Mean C_{pe} with wind angles on the re-entrant faces B2 and C1

Wind flow patterns for certain wind incident angles are shown in figure 9 (a) to (e). Flow separation and vortices can be seen from the flow patterns. It is observed that the vortices are not symmetrical as in the case of rectangular model, figure 7. Small vortex is created within the re-entrant corner for different wind angles other than where wind is directly hitting the corner (viz 135° wind angle). Streamlines along the central vertical plane for different wind incidence angles are shown in figure 12. The creation of upstream vortex near ground due to down

wash of wind after hitting the windward face, flow separation from the leading edges, recirculation cavity at roof and vortex on the downstream side of the model can be visualized. It is noticed that the pattern and intensity of vortices created for different wind angles are different. Major difference is found between 'before 90° ' wind incident angles and 'after 90° ' wind incident angles. At 45° wind incident angle the recirculation cavity at roof does not appear.

(a) Face B2

Figure 11. Variation of C_{pe} Central Vertical on Face B2 and Face C1 with Wind Angle

Figure 12. Streamlines along Wind Central on Vertical L-Plan Shaped Model

8. Conclusions

The present study shows that the wind incident angle has significant impact on the L shape building. For wind incident angles ranging 0° to 60° the re-entrant wing faces experience homogeneous suction pressure. Upon further increase of the wind incident angles between 75° to 180°, the pressure on the re-entrant wing faces increases due to stagnation of air. This is the unique characteristic of L shape buildings which plays an important role in design of cladding systems on re-entrant faces. The height of the building and the relative length dimensions of the wing faces play an important role in producing pressure on the wing faces.

Result of numerical simulation with standard $k-\varepsilon$ model showed a good agreement in case of normal wind incident angle. However, some differences cannot be ruled out for other directions of wind angles due to the non-isometric unsteady three-dimensional flow behavior of wind. Denser mesh arrangement in the particular flow region and improved modelling for mapping the three-dimensional non isometric unsteady flow could

enhance the quality of results. Nevertheless, the results obtained can provide useful information about wind pressure on re-entrant wing faces of L shape buildings for practical engineering calculations.

REFERENCES

- [1] Baines W. D., "Effect of Velocity Distribution on Wind Loads on a Tall Building," 1952.
- [2] Mohotti D., Mendis P., and Ngo Y. D., "Application of Computational Fluid Dynamics (CFD) in Predicting the Wind Loads on Tall Buildings - A Case study," in *23rd Australasian Conference of the Mechanics of Structures and Materials (ACMSM23)*, Dec. 2014, pp. 1–6.
- [3] Fu J. Y., Li Q. S., Wu J. R., Xiao Y. Q., and Song L. L., "Field measurements of boundary layer wind characteristics and wind-induced responses of super-tall buildings," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 96, no. 8–9, pp. 1332–1358, Aug. 2008, doi: 10.1016/j.jweia.2008.03.004.

- [4] Tanaka H, Tamura Y, Ohtake K, Nakai M, Kim YC, and Bandi E K, "Experimental Investigation of Aerodynamic Forces and Wind Pressure Acting on Tall Buildings with Various Unconventional Configuration," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 107–108, pp. 179–191, 2013.
- [5] Amin J.A and Ahuja A.K, "Effects of Side Ratio on Wind Induced Pressure Distribution on Rectangular Buildings," *Hindawi Publishing Corporation, J Struct*, pp. 1–12, 2013. doi: dx.doi.org/10.1155/2013/176739.
- [6] Kushal T, Ahuja A.K, and Chakrabarti A, "An Experimental Investigation of Wind Pressure Developed in Tall Buildings for Different Plan Shape," *Int. J Innov Res Studies*, vol. 1, no. 12, pp. 605–614, 2013.
- [7] Chakraborty S and Dalui S K, "Numerical Study of Surface Pressure on Square Plan Shape Tall Building," in *Symposium on sustainable infrastructure development (SID)*, Feb. 2013, pp. 252–258.
- [8] Bhattacharyya B, Dalui S.K, and Ahuja A.K, "Wind Induced Pressure on 'E' Plan Shaped Tall Buildings," *Jordon J Civil Eng*, vol. 8, no. 2, pp. 120–134, 2014.
- [9] Chakraborty S., Dalui S. K., and Ahuja A. K., "Experimental Investigation of Surface Pressure on '+' Plan Shape Tall Building," *Jordan Journal of Civil Engineering*, vol. 8, no. 3, pp. 251–261, 2014.
- [10] Kheyari P. and Dalui S.K., "Estimation of Wind Load on a Tall Building under Interference Effect," *Jordon J Civil Eng*, vol. 9, no. 1, pp. 84–101, 2014.
- [11] Roy K., and Bairagi A. K., "Wind Pressure and Velocity Around Stepped Unsymmetrical Plan Shape Tall Building Using CFD Simulation—A Case Study," in *Asian J. Civil Eng. (BHRC)* 17, 2016, vol. BHRC, no. 17, pp. 1055–1075.
- [12] Mukherjee A., and Bairagi A. K., "Wind Pressure and Velocity Pattern Around 'N' Plan Shape Tall Building- A Case Study," in *Asian J. Civil Eng*, 2017, pp. 1241–1258.
- [13] Bairagi A.K., and Dalui S.K., "Distribution of Wind Pressure Around Different Shape Tall Building," *Advances in Structures, System and Material*, pp. 31–38, 2020.
- [14] Pal S., and Ritu Raj, "Evaluation of Wind Induced Interference Effects on Shape Remodeled Tall Buildings," *Arabian Journal for Science and Engineering (Online Publication)*, 2021.
- [15] Ritu Raj and Ahuja. A.K., "Wind Loads on Cross Shape Tall Buildings," *JAIR*, vol. 2, no. 2, 2013.
- [16] Ritu Raj, Rana T., Anchalia, and Khola U., "Numerical Study of Wind Excited Action on H Plan-Shaped Tall Building," *International Journal of Emerging Technologies*, vol. 11, no. 3, pp. 591–605, 2020.
- [17] Pal S., Ritu Raj, and Anbukumar S., "Comparative Study of Wind Induced Mutual Interference Effects on Square and Fish-plan Shape Tall Buildings," *Sadhana (Indian Academy of Science)*, pp. 46–86, 2021.
- [18] Nagar S. K., Ritu Raj, and Dev N., "Experimental study of wind-induced pressures on tall buildings of different shapes," *Wind and Structure*, vol. 31, no. 5, pp. 441–453, 2020.
- [19] Kumar A., and Ritu Raj, "Study of pressure distribution on an irregular octagonal plan oval-shape building using cfd," *Civil Engineering Journal (Iran)*, vol. 7, no. 10, pp. 1787–1805, Oct. 2021, doi: 10.28991/cej-2021-03091760.
- [20] Stathopoulos T., "Numerical Simulation of Wind-induced Pressure on Buildings of Various Geometries," *J. Wind Engg. Indus. Aerodynam*, vol. 46 & 47, pp. 419–430, 1993.
- [21] Gomes M. G., Moret Rodrigues A., and Mendes P., "Experimental and numerical study of wind pressures on irregular-plan shapes," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 93, no. 10, pp. 741–756, 2005, doi: 10.1016/j.jweia.2005.08.008.
- [22] Amin J.A and Ahuja A.K, "Experimental Study of Wind-induced Pressure on Buildings of Various Geometries," *Industrial Journal of Engineering Science and Technology*, vol. 3, no. 5, pp. 1–19, 2011.
- [23] Revuz J, Hargraves D. M, and Owen J.S., "On the Domain Size for the Steady State CFD Modelling of a Tall Building," *Wind Structure*, vol. 15, no. 4, pp. 313–329, 2012.
- [24] Franke J., et al., "Impact of wind and storm on city life and built environment," *Recommendation on the use of CFD in Wind Engineering. In: COST action C14*, vol. Version 1.0, pp. 1–12, 2004.
- [25] Cook N. J., *The Designer's Guide to Wind Loading of Building Structures. Part 2: Static Structures*. London: Butterworths Press, 1985.