Journal of Engineering Research and Application
http://dx.doi.org/10.55953/JERA.2023.2104
Volume 2, June 2023
pp 27-38

Investigation of wind effects on star shape building model

Astha Verma[2]* , Vishal Kadian[1], Suraj Rai[1], Rishab Aggarwal[1] and Raghav Siakri[1]
[1] Department of Civil Engineering, Punjab Engineering College
[2] Gobind Ballabh Pant University of Agriculture & Technology, India
* Corresponding Author asthaverma.civil@gbpuat-tech.ac.in

ABSTRACT

Building construction in India has been on rise since last two decades. More recently construction of tall buildings ranging from 10 to 50 stories have been proposed in many cities. For these buildings, wind analysis is as important as earthquake analysis, since in taller buildings, wind loads are dominant than earthquake loads at greater heights. There is a rising demand for tall structures to fulfil the occupancy requirements in congested metropolitan areas. In India, wind analysis is done according to IS 875(Part III):2015. generally engineers rely on wind tunnel experiments to understand the behaviour of winds on structures. However, for arriving at the best orientation several tests have to be conducted which is a tedious job. In this project, a necessity to adopt a different approach such as Computational Fluid Dynamics (CFD) simulation is proposed. The objective of this project is to study the effect of wind on a statement design model which is computed using AutoCAD 3D and ANSYS CFX (using CFD methodology). The modelling of building is done on AutoCAD (3D Modelling) software. The building is consisting of two cross – sections – square (at the lower portion) and plus (in the upper portion) of the building. Various contours and streamlines and their interpretation are discussed in detail.

1. INTRODUCTION TO TALL BUILDINGS

There is no universally accepted definition of a “tall building”. One of the main areas of application of CFD in wind engineering is in the environmental aspect. In the past few decades, research on the application of CFD has been conducted extensively in areas such as pedestrian wind comfort and safety, exterior building surface heat transfer, pollutant dispersion around buildings, and natural ventilation of buildings. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. CFD has been used effectively in modelling aerodynamics effect on automotive. Therefore, it has shown a considerable accuracy in simulating atmospheric boundary layer effect. This highlights the possibility of using a similar approach to simulate the wind behaviour around the buildings. There is no universally accepted definition of a “tall building”. One of the main areas of application of CFD in wind engineering is in the environmental aspect. In the past few decades, research on the application of CFD has been conducted extensively in areas such as pedestrian wind comfort and safety, exterior building surface heat transfer, pollutant dispersion around buildings, and natural ventilation of buildings. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or

www.jera.co.in
ISSN2583-3987

turbulent flows. CFD has been used effectively in modelling aerodynamics effect on automotive. Therefore, it has shown a considerable accuracy in simulating atmospheric boundary layer effect. This highlights the possibility of using a similar approach to simulate the wind behaviour around the buildings.

Fig. 1- Representation of Tall Buildings

1.1 CFD tools over Wind Tunnel:

There are several advantages of CFD tools over Wind Tunnel as follows. The advantages of CFD tools over Wind Tunnel **is** Easy alternative analysis – CAD design of buildings in the CFD domain can be altered quickly and remodelling done immediately. Physical models require more time and effort for adjustment, especially if the design changes occur long after the initial wind tunnel modelling or the wind tunnel is booked for other projects.

Fig. 1- Representation of Tall Buildings

2. Methodology

2.1 Basic Equations

Easy alternative analysis – CAD design of buildings in the CFD domain can be altered quickly and remodelling done immediately. Physical models require more time and effort for adjustment, especially if the design changes occur long after the initial wind tunnel modelling or the wind tunnel is booked for other projects.

2.2 Standard k - ε model

K – ε model is a two – equation model of computational fluid dynamics used to replicate flow characteristics for turbulent flow conditions. This model assumes the turbulent viscosity of wind considered as 10 m/s in this study; to be isotropic. Turbulence is less pronounced in this model in comparison to SST k – ω model.

Turbulent kinetic energy,

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij}E_{ij} - \rho \times \in \quad (1)$$

Energy Dissipation,

$$\frac{\partial(\rho\times\in)}{\partial t}+\frac{\partial(\rho\in\mu_{i)}}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\in}\frac{\partial\in}{\partial x_j}\right] + C_{1\in}\frac{\in}{k}2\mu_t E_{ij}E_{ij} - C_{2\in}\rho\frac{\in^2}{k} \quad (2)$$

2.3 Standard $k-\omega$ Model

It is a two – equation model, generally used for flows having low Reynolds's number used to interpret turbulent flow conditions. For open channel flow problems, k – ω gives best result.

Turbulent kinetic Energy,

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho\in\mu_{i)}}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\tau_k\frac{\partial k}{\partial x_j}\right]+G_k - Y_k + S_k \quad (3)$$

Specific dissipation rate,

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho\omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\tau_\omega\frac{\partial\omega}{\partial x_j}\right]+ G_\omega - Y_\omega + S_\omega \quad (4)$$

2.4 SST k-ω Model

SST acronym for Shear Stress Transport, is used to obtain better flow separation prediction for adverse pressure gradient conditions.

Turbulent kinetic energy,

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho\omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\tau_k\frac{\partial k}{\partial x_j}\right]+ \widetilde{G} - Y_k + S_k \quad (5)$$

Specific Dissipation rate,

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho\omega u_i)}{\partial x_i} = \frac{d}{\partial x_j}\left[\tau_k\frac{\partial\omega}{\partial x_j}\right]+ G_\omega - Y_\omega + D_\omega + S_k \quad (6)$$

3. Numerical Simulation

3.1 CFD Pre – simulations

Power law is used for interpreting the results in ANSYS in set – up and solution steps. The inputs for the set – up step are α = 0.147, Z_{ref}= 1 m, and U_{ref} = 10 m/s. Power law is used for calculation of pressure (Equation 7).

$$P = U_{ref}\left(\frac{Z^\alpha}{Z_{ref}^\alpha}\right) \quad (7)$$

The program is run for about an hour before the results are evaluated. The variable coefficient of pressure is then found out (Equation 8). V_{ref} is assumed to be 10 m/s.

$$C_p = \frac{(P - P_{ref})}{(\frac{1}{2}\rho_\alpha V_{ref})} \quad (8)$$

Thus, value of pressure density comes out, $p_a = 1.225\ kg/m^3$.

The wind load effect on Model is analysed for 17 faces. Windward side is that side which is facing the on – coming wind. The side resisting this on – coming wind is called as the Leeward side.

3.2 DESIGN OF STATEMENT MODEL BUILDING

3.2.1 ANSYS

ANSYS is an American company based in Canonsburg, Pennsylvania. It develops and markets CAE/ multi-physics engineering simulation software for product design, testing and operation and offers its products and services to customers worldwide. ANSYS Mechanical finite element analysis software is used to simulate computer models of structures, electronics, or machine components for analysing strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. The uses of ANSYS are numerous in number, but finally all of them leads to one single concept called profitability to the organization. I will explain how, let's consider you are manufacturing a product. In the first place you need to manufacture some prototypes of the product and test it for all kinds of worse situation that could happen to hamper or damage the product. In some cases you need to conduct few tests to optimize the material required, the design, the loads if any acting on the product, the natural frequency or its temperature controlling mechanism, and many more. In order to know every detail of your product you need to manufacture prototypes. This would eventually lead to increase the cost to company.

3.2.2 IS CODES

IS 875 (Part III) : In our project, we have considered IS Code 800: 2007, IS 875 (Part 1) for design loads (other than earthquake), IS 875 (Part 2) deals with imposed loads on buildings produced by the intended occupancy and IS 875 (Part 3) for code of practice that provides for information on wind effects for buildings and structures, and their components. For our project, we will focus majorly on IS 875 (Part 3). 2.3.1 IS 875 (Part III) IS 875 (Part III) is code of practice for calculating wind loads for buildings and structures and it was last revised in 2015. This code provides data and procedure to calculate design wind pressure for different terrains and topographies at different heights. It also provides data of pressure coefficients for rectangular shaped buildings having roof tops of different slopes and of different height and width ratios at different angles of attack of wind. In addition, data of force coefficients for clad and unclad buildings of different shapes such as square, circular, triangular are given. Dynamics effects of wind on structures is addressed in brief in this code. Gust effectiveness factor method is the only method in this code for calculating load along wind direction.

However, IS 875 (Part III) does not give any information about across – wind responses of the structure. It only provides data of buildings with regular shapes like square, rectangular, circular etc. It doesn't have any information of any other shapes of structures liked winged structures. Dynamic effects of wind on structures are not fully addressed in this code. However, on the other hand, the necessity of such design is increasing because in some cities, the construction of tall buildings is increasing day by day. Seeing all the above drawbacks of IS 875 (Part III), there is a necessity for different approach to evaluate wind effects on tall structures. Wind tunnel experiments and CFD simulations are the alternate ways to address this problem.

3.3 Ansys Geometry Modelling

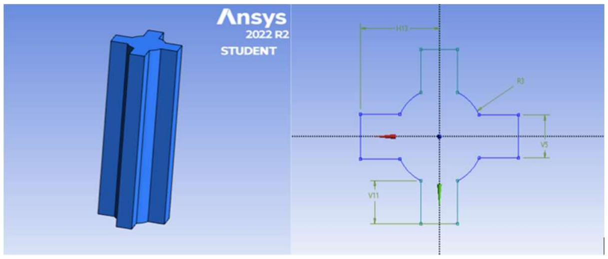


Fig. 2 Specifications of the model

1. (R3) Radius of Circle = 5.64 mm
2. (V11) length of side building = 5 mm
3. (V5) width of side building = 5 mm
4. Height of the model. = 60 mm
5. Dimensions = 10 mm x 10 mm

The CFD analysis of the wind flow surrounding the building will be performed. The terrain category was taken as 2 and the design wind speed was selected as 10 m/s. The lateral wind load analysis was performed according to the guidelines given in AS 11170.2:2011 and results are tabulated.

3.4 Design Procedure in ANSYS

Design Procedure - The ANSYS CFX mode is used for analysis of our model. The procedure in total is divided into 5 sections, namely, Geometry, mesh, setup, solution and result.

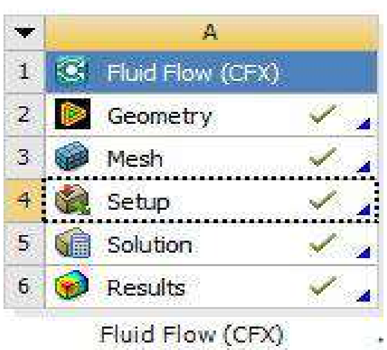


Fig. 4 symbols of ASYS CFS

- **STEP 1 - GEOMETRY**

We will make the domain around our model in accordance to the specifications given by IS 875 (Part 3). According to the code, the front face of the domain ahead of face A (Windward side) should be at a distance of 5h from the surface of face A (where h is the height of the model or building, i.e., h = 60 mm). The back face of the domain should be at a distance of (15h + 0.5 l). And the side faces of the domain should be at a distance of (5h + 0.5l) from the surface of sides. Also, the top of the domain should be at a distance of 5h from the top surface of the building. It is represented in the following figure.

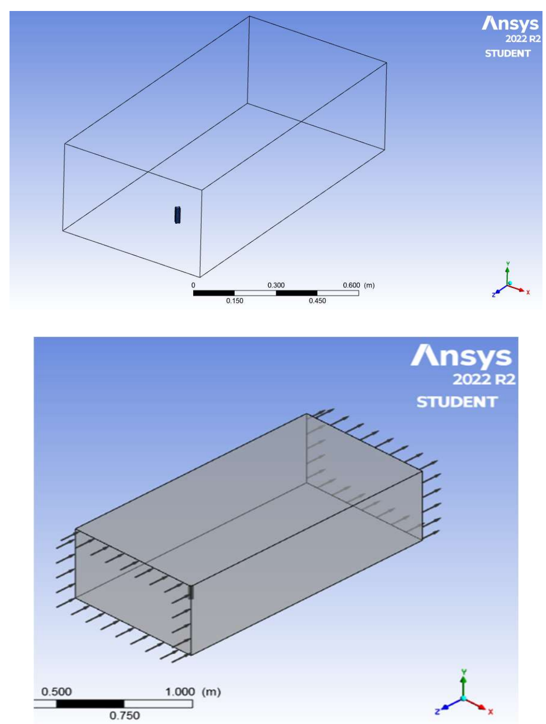


Fig.3 Representation of the domain around the design model

This marks the end of step 1.

- **STEP 2 - MESHING**

Generating a high quality mesh is extremely important to obtain reliable solutions and to guarantee numerical stability. In the CFD community, mesh generation is also referred to as grid generation, which has become a separate discipline in itself and remains a very active area of research and development. This is evident with many existing commercial codes on the market having their own powerful, built-in mesh generators, as well as with a number of independent grid generation packages available.

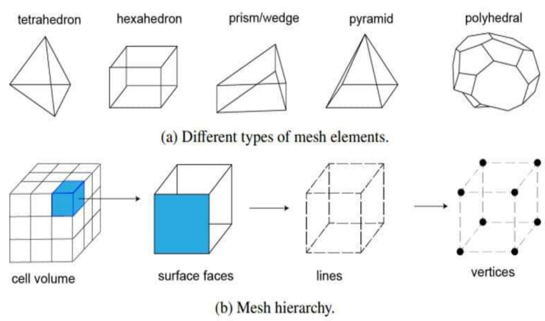
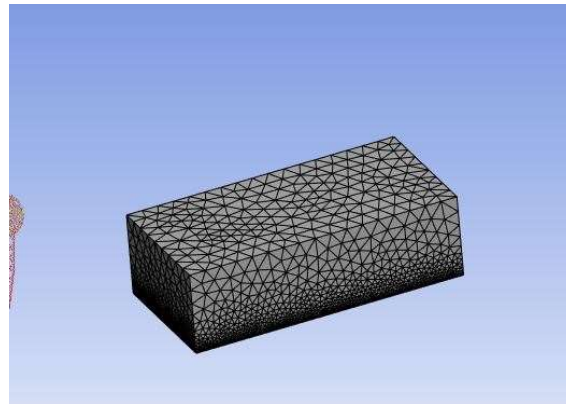
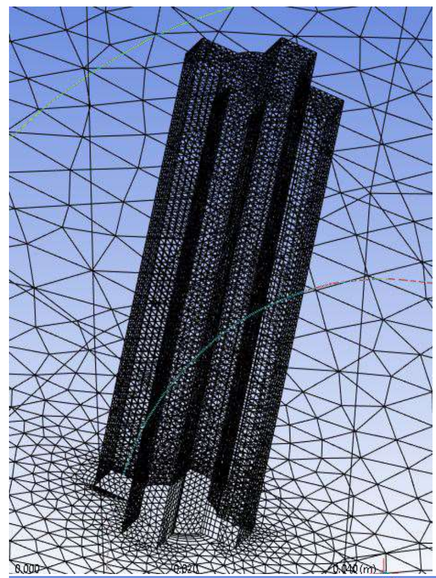
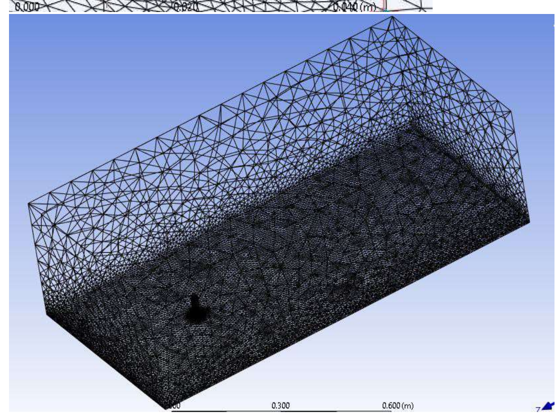


Fig.4 Mesh types







Generating a mesh for the first time is a daunting process, where decisions have to be made on the arrangement of discrete points (nodes) throughout the computational domain, and the type of connections for each point. The quality of the mesh leads to either success or failure of the numerical simulation. For our model, the following specification are considered as below.

Mesh specifications

Face sizing 1 = 8×10^{-4} m (for building and ground)

Face sizing 2 = 8×10^{-3} m (other surfaces)

Inflation is done for full domain and then separately for whole building

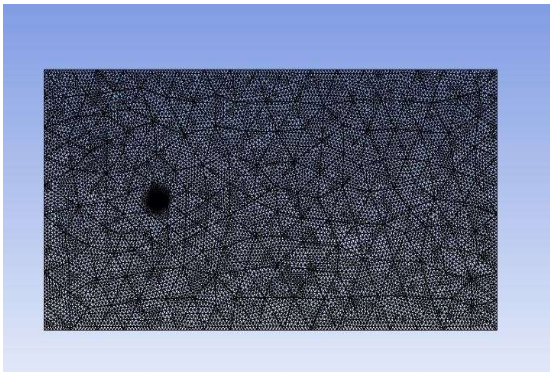


Fig. 5 Meshing diagram of buildings

- **STEP 3 - SETUP**

In the set up step, the following inputs are provided,

- $\alpha = 0.147$
- $Y_{ref} = 1$ m
- $U_{ref} = 10$ m/s

Wind blow due to the pressure difference over a different part of the earth and Coriolis force. Due to the ground roughness atmospheric boundary layer is formed near the ground. And the wind becomes highly turbulent due to ground obstacles. The degree of ground roughness and drag due to local projections that resist wind flow determines the vertical profile of wind speed. Gradient height is the height at which the drag effects disappear, and gradient velocity is the corresponding velocity. The atmospheric boundary layer is the height at which topography influences wind speed. As per Power Law, the wind speed profile, as shown in Fig. within the atmospheric boundary layer is given by

Using power law, we calculate pressure,

Pressure on each side of building face.

$$P = U_{ref} \times (Y/ Y_{ref})^{\alpha}$$

- **STEP 4 – SOLUTION**

The program is run for about an hour for the results to be evaluated. Reynolds number is observed. After finish Contours, Streamlines and Pressure coefficients can be obtained.

- **STEP 5 - RESULTS :** Details Streamlines, Pressure contours on different faces are created.

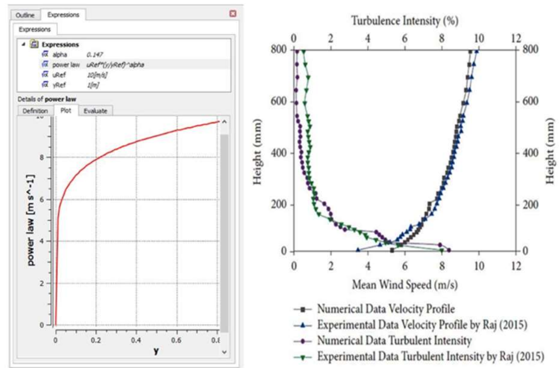


Fig. 6 Velocity Profile and Turbulent Intensity

3.5 VALIDATION

A study on CFD analysis for tall building having rectangular cross – section was performed using ANSYS and it was verified that the results got were in good agreement with IS 875 (Part 3) and were most accurate for wind incidence angles 0° and 90° (Ashok, J., Charles, S., & Umarani, C., 2018). Thereby in this study for validation purposes, a tall building having a square cross – section throughout has been taken.

An isolated reference model named Square Model is designed and analysed using ANSYS: CFX. Square Model is designed such that the entire model is formed with a single square cross – section having dimensions 100 mm x 100 mm throughout the height of 192 m (Fig. 3). Coefficient of pressure, Cp values for each face (A, B, C, D) of Square Model is recorded after analysis using ANSYS: CFX mode and is as listed in Table below. It is compared with the acceptable values as given in IS: 875 (Part III) – 2015.

TABLE 1– Comparing Cp values of Model X with acceptable Cp values in accordance to IS: 875 (Part III) – 2015

Wind loading code	FACE A	FACE B	FACE C	FACE D
By Ansys CFX	0.68	-0.34	-0.67	-0.65
IS:875 PART (III)	0.8	-0.25	-0.8	-0.8

4. ANALYSIS OF MODELLED TALL BUILDING

4.1 CFD Simulations

A body can be considered as an aerodynamically "bluff" when flow streamlines do not follow the surface of the body similar to the case of streamlined body, but detach from it leaving regions of separated flow and wide trailing wake. Most of the manmade structures including tall buildings are aerodynamically bluff bodies which have been designed to withstand wind forces. Therefore, it is important to understand the flow pattern around the buildings in order to validate the model results in wind simulations.

The simulations were able to capture flow separations and vortex formation in the wake of the bluff body quiet well. In the vicinity of those regions high velocity flow can be observed. Leeward side of the building is generally in the wake where the lower negative pressure presents. This cause drag forces on the wall of the building in the leeward direction. Separated flows get re-attached at rear stagnation point in the leeward direction of the building.

The zone between rear stagnation point and the building is significantly turbulent and low in wind pressure. The wind pressure and forces on the building is generally estimated by assuming non-compressible fluid scenario and using the equation given below where, C_p and C_f are pressure and force coefficients, ρa is the air density, P and F are pressure and force at the location of interest, P_{ref} ,V_{ref} and A_{ref} are static pressure ,velocity and area at the reference location. The pressure coefficients (C_p) of the outer surface of the building obtained through the CFD simulations. Standards assume that the pressure on the buildings increases with the increasing height. However, CFD results show that the pressure on the windward surface of the building increases with the increasing height measured from the ground.

4.2 Faces

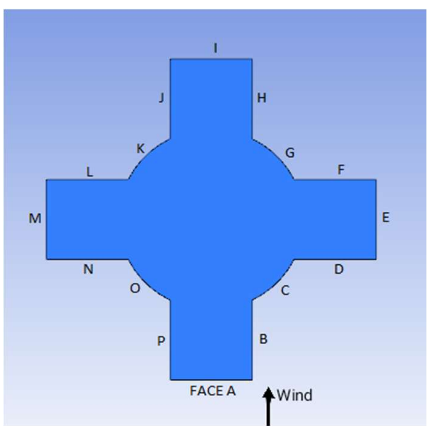
In our project, we have considered the modelled as mentioned in Chapter 2. The wind load effect on this building or model will be analysed for 17 faces. The faces are names as shown below. Windward side is that side which is facing the oncoming wind. The opposite side to it is called as the Leeward side. The same have been shown in the diagram below. Majorly we have classified and named the four directions to have faces A, B, C and D as the "front faces" as shown in the figure. These remaining faces will be classified and named as "side faces" for these front faces. As explained in the figure below, each direction or front face will have 2 side faces.

4.3 Pressure Contours

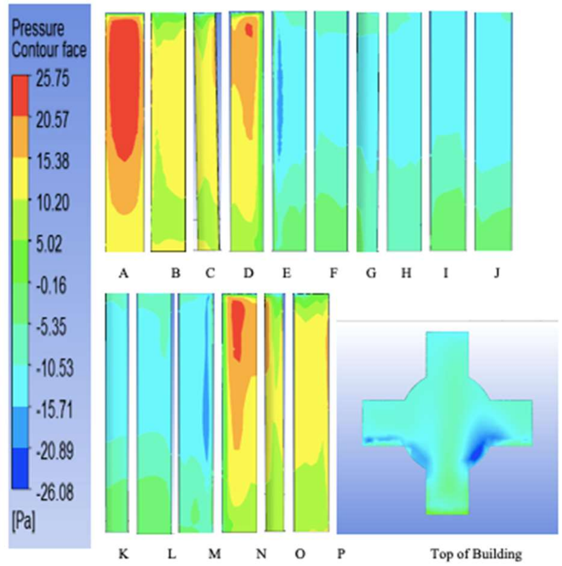
Pressure contours are obtained on analysis in ANSYS: CFX mode. Pressure contours are used to depict the change in pressure values on a surface. A contour plot with color bands has discrete coloured regions while the display of a variable on a locator (such as a boundary) shows a finer range of colour detail by default. The instructions that follow will illustrate a variable at the outlet and create a contour plot that displays the same variable at that same location.

4.4 Streamlines

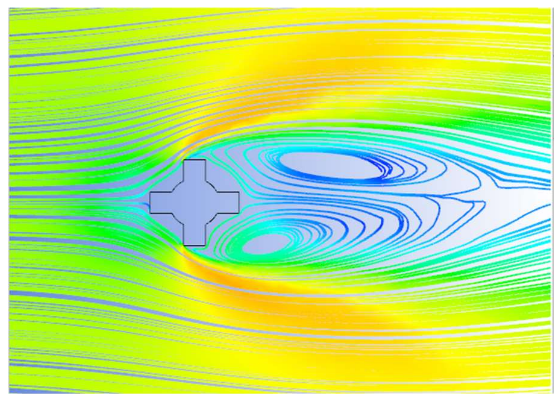
Wind is a phenomenon of great complexity because of the many flow situations arising from the interaction of wind with structures. Wind is composed of a multitude of eddies of varying sizes and rotational characteristics carried along in a general stream of air moving relative to earth's surface. These eddies give wind its gutsy and turbulent character. The gustiness of strong winds in the lower levels of the atmosphere largely arises from interaction with surface features. The average wind speed over a time period of the order of ten minutes or more, tends to increase with height, while the gutsiness tends to decrease with height. The wind in our study is taken to be a fluid body. Streamline is a path of imaginary particles suspended in the fluid and carried along with it. The streamlines are represented as below.



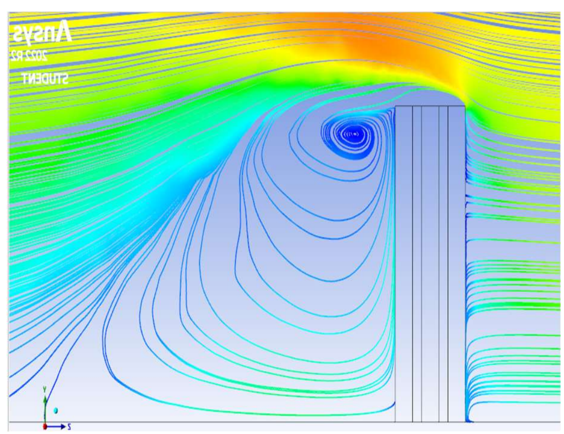
Face Details



Observation diagram for Contours



Observation diagram for Horizontal Mid plane Streamlines



Observation diagram for Vertical Side plane Streamlines

4.5 Pressure Coefficient (C_p)

The positive value of Cp indicates that the face is in the windward region, whereas the negative value indicates that the face is in the leeward region. C_p for square building model, only the windward face has positive pressure distribution, rest all faces are under the effect of negative pressure distribution that is due suction on the leeward side and side faces of the building model.

5. CONCLUSIONS

In-trinational codes only provide pressure coefficient CP values for shapes like square and rectangle. After using a square to validate the study, the objective of this research is to determine the pressure coefficient for various regular shapes. Various methods such as wind tunnel testing, gust factor method are used to determine the behavior of wind effect on high – rise buildings for slenderness, serviceability, resilience resistance, comfort, and sustainability conditions. However, CFD supersedes the most popular method for analysis of wind flow characteristics in tall building called wind tunnel method as it eliminates scaling, and provides direct reporting. The following major conclusion are drawn from this study. From our work, we can give the following conclusions.

- From Velocity Streamlines we can see that the direction of wind flow is upward to a height of one-third of the building model and downward to a height of two-thirds of the building model. It has been discovered that as a model's number of sides increases, so does its ability to resist wind loads.
- Pressure experiences and flow patterns are also provided via ANSYS CFX. Information about flow separation and vortex development in the negative pressure zone was obtained from flow patterns.
- Due to direct wind contact, it was observed that each building's windward face had the most positive pressure. In weak zone vortices are formed.
- Due to the creation of vortices, the pressure coefficient for the side faces of each building was more negative than the pressure coefficient for the rear faces. And was the same for opposing faces when the wind direction passed through the axis of symmetry of any shape.
- The pressure is found to be lowest at the top face, close to the windward side, and then progressively rises. When wind moves away from the shape's axis of symmetry, two clearly defined vortices are visible.

6. FUTURE SCOPE

We will analyse our model along with new models which would be a variation of this model in ANSYS and later on in STAAD PRO and ETABs and verify our results. Also, further cases will be considered wherein the building is rotated from the vertical axis at 15°, 30°, 45°, 60°, 75° and 90° and discusses the contours and the results obtained from our design problem. Once the pressure parameters on the surface of the building are obtained from CFD analysis, it will be used as an input in ETABs model to compare the code predicted results.

REFERENCES

[1] A. Verma, R.K. Meena, H. Dubey, R. Raj, S. Anbukumar, Wind effects on rectangular and triaxial symmetrical tall building having equal area and height, vol. 2022, 2022.

[2] R.K. Meena, R. Raj, S. Anbukumar, Wind excited action around tall building having different corner configurations, Adv. Civ. Eng. 2022 (2022), https://doi.org/10.1155/2022/1529416.

[3] S. Pal, R. Raj, S. Anbukumar, Bilateral interference of wind loads induced on duplicate building models of various shapes, Lat. Am. J. Solids Struct. 18 (5) (2021), https://doi.org/10.1590/1679-78256595.

[4] S.K. Nagar, R. Raj, N. Dev, Experimental study of wind-induced pressures on tall buildings of different shapes, Wind Struct. An Int. J. 31 (5) 441–453, https://doi.org/10.12989/was.2020.31.5.431

[5] Montazeri, H. and Blocken, B. (2012) "CFD Simulation of Wind-Induced Pressure Coefficients on Buildings with and without Balconies: Validation and Sensitivity Analysis", Building and Environment, Vol.60, pp.137-139