

Validation of Wind Pressure on Multi-Storied Building by Using CFD Simulation and Shape Optimization

Rabna P.K¹, Ashly Sajan²

P.G Student, Department of Civil Engineering, Jai Bharath College of Management and Engineering Technology,
Ernakulam, Kerala, India¹

Associate Professor, Department of Civil Engineering, Jai Bharath College of Management and Engineering
Technology, Ernakulam, Kerala, India²

ABSTRACT: Wind loading on buildings, being an important parameter in Multi-Storied building design, needs to be carried out with at most precision. As most of the existing wind design codes have their own limitations in providing necessary guidelines for the wind designing, such as height limits of the buildings, the expected practice is to conduct wind tunnel tests to determine the wind-induced loads on the buildings. However, the cost of the wind tunnel test is comparatively high and conducting wind tunnel tests at the preliminary design stage is uneconomical. This paper investigates the Computational Fluid Dynamics (CFD) simulations on the design wind loads of a high rise building and indicates that the CFD technique can be used to predict the mean wind load of an isolated tall building with reasonable accuracy. A CFD simulation gives more or less a realistic picture of the physical phenomenon of wind loading on high rise buildings. Also, the equation for determining the wind pressure on building from the design wind speed of the locality will be validated using the CFD simulation results.

I. INTRODUCTION

It is desired to study wind effects on tall buildings for design consideration, because wind load is one of the critical loads which are considered in design of tall buildings. It is seen in many design cases that severe condition occurs under wind load for tall buildings. Wind affects such tall buildings in two way i.e., statically and dynamically. Generally pressure due to wind load is acting on different surfaces of buildings. Pressure variation of some regular (square, rectangular, circular etc.) plan shaped building are given in relevant codes (IS: 875 (Part-3):1987.) and design of such regular plan shaped tall building is done using these relevant codes. But, pressure variation on irregular plan shaped tall building may vary with respect to regular plan shaped, so it is desired to study wind induced effects on irregular plan shaped tall building. Only wind effects due to perpendicular wind angles are considered in relevant codes, but it is seen from previous researches that, wind effects due to skew wind incidence angles also need to calculate. Pressure variation may change noticeably in case of wind effects due to skew wind incidence angles.

Pressure variation may change noticeably in case of wind effects due to skew wind incidence angles. Computational Fluid Dynamics (CFD) is much matured technique in the field of wind engineering in recent decades. Pressure distribution on different faces of building and wind flow pattern could be predicted by this technique very usefully and the results can be used for any structural design.

Objective of this study are:

- To validate the wind pressure calculation as per IS 875 (Part 3) using CFD simulations and the results from the Wind Tunnel Experiment.
- To compare the wind pressure on the aero dynamic structure using IS 875 (Part 3) and the CFD simulations.
- Shape Optimization for High rise building using CFD simulations.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 6, June 2019

II. METHODOLOGY

Winds loading are complex live load that varies both in time and space. The wind induced on building is response of tall buildings can reduce by means of aerodynamic. The use of aerodynamic building forms are an effective method for reducing the wind loads on buildings.

Methods considered are:

- Analytical method (using gust factor approach from IS-875(Part3)-1987).
- Software Analysis (using ICEM, CFD and ANSYS FLUENT 19.0 tool).

Steps involve:

- An extensive study of literature carried out of wind pressure to understand the present advance in computation on wind loads on high rise building.
- Coupled Field Analysis is conducted to execute the whole work using ANSYS Simulation software Package.
- ANSYS ICEM, Fluent and Mechanical will be used for Meshing, CFD, and Structural simulation. ANSYS Fluent will be used for simulating the fluid flow.
- Coupled Field analysis for estimating the wind pressure on the building walls and its deflection using ANSYS Fluent and Mechanical.
- The wind tunnel experiment results will be validated using analysis of CFD simulation.

III. MODELLING AND ANALYSIS

III.1 DATA COLLECTION

With the data collection phase, the initial data has been collected from IS 875: Part 3-1987, National Institute of Wind Engineering (CWET). As per Indian code of practice, IS 875 the wind speed has been defined on the basis of basic wind speed. The wind speed for this study has been chosen as 39 m/s which are the basic wind speed at Trivandrum.

III.2 SHAPE OPTIMIZATION

In order to further study the wind resistance of multi-type high-rise building structures, 8 different geometry is selected with projected area plan - 900 m² and height of 225m by analytical method (using IS 875 (part 3) - 1987) and numerical analysis (using ANSYS FLUENT 19.0). A comparison of results obtained from CFD simulations to the predictions given by the Indian Wind Design Standards (IS875 (part3)-1987) is presented.

The shape optimizations are focused on the basis of same plan area and height. The plan area is fixed at 900 m² and height of the building at 225 m tall. A Square Building, Triangular Building apex facing, Triangular Building base facing, Elliptical Building horizontal, Elliptical Building vertical, Square Building plus shape, and Hexagonal Building are established.

III.3 COMPUTATIONAL DOMAIN

The computational domain defines the region where the flow field are computed numerically used the assigned models. The building geometry for the computational domain was based on wind tunnel study. The structure considered 75-story building model.

The Computational Domain has been fixed as per Bilal, 2015. Two surfaces 2 m and 30 m from exterior of building is also considered for better results near vicinity. The overall computational domain was 21B × 47B where the structure was cantered at 10.5B from the inlet and 10.5B from the south wall condition. A distance of 36B in the downstream direction was chosen to allow for the reduction of the disturbances experienced by the flow.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 6, June 2019

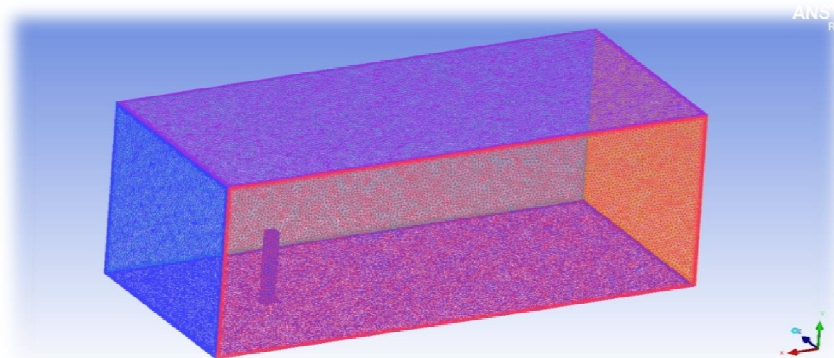


Fig: 3.1: 3D modeling of building within the Computational Domain

The overall computational domain was $18.75B \times$ by $47B$ where the structure was centered at $10.5B$ from the inlet. The 8 geometries of same plan area (900 m^2) and height (225 m) are kept same. The buildings are placed at a distance 300 m from inlet and 1080 m from outlet.

III.4 DETAIL OF MESHING OF THE GEOMETRY

In order to analyse fluid flows, flow domain is split into smaller sub domains. Then the governing equations are discretized and solved inside each sub domain. The sub domain is called nodes, elements, or cells. The collection of all elements and cells are called a mesh or grid. They are two classifications of meshes that depend upon the connectivity of the nodes:

- Structured Meshes: They are characterized by regular connectivity which can be expressed as arrays. These elements are usually quadrilaterals in 2D and hexahedra in 3D.
- Unstructured Meshes: They are characterized by irregular connectivity and cannot be expressed as an array in a computer. It allows for the use of any variation of geometric shapes which can fit the model of geometry.

The quality of mesh checked in ICEM-CFD and if this value is above 0.5 the quality of mesh referred as good. The greater number of cells gives the better CFD results, but as the number of cells increases with the calculation time.

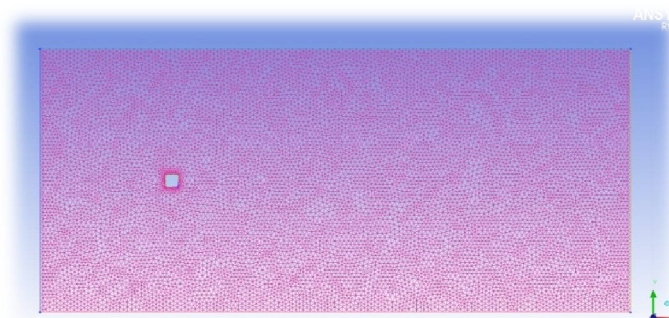


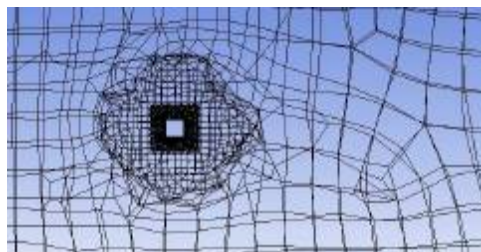
Fig. 3.2 Meshing of computational domain and square building

International Journal of Innovative Research in Science, Engineering and Technology

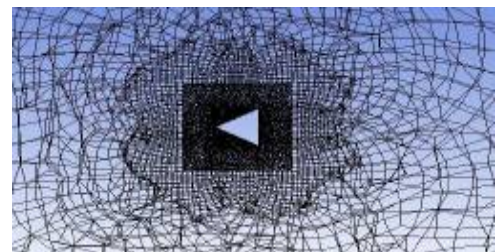
(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 6, June 2019



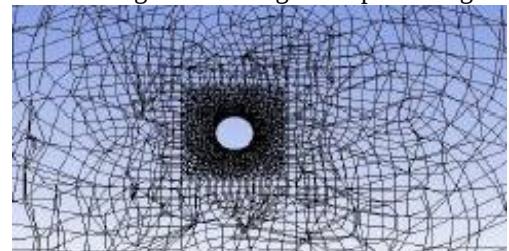
Square Building



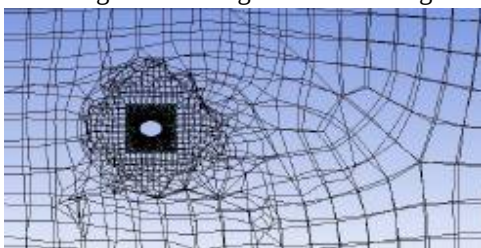
Triangular Building with apex facing



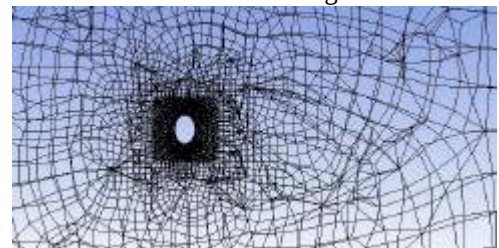
Triangular Building with base facing



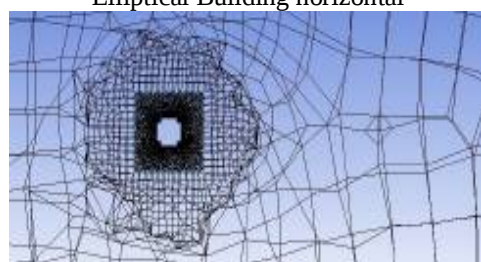
Circular Building



Elliptical Building horizontal



Elliptical Building vertical



Square Building plus shaped



Hexagonal Building

Fig. 3.3 Meshing of Computational Domain of all the Geometry

III.5 BOUNDARY CONDITIONS

The boundary conditions defined as determines at what point the surroundings is deemed to no longer have a significant effect on the computational domain. When setting the inflow boundary conditions, the mean velocity profile and information about the turbulence quantities are needed to create an equilibrium boundary layer. For the top and lateral boundaries wall condition can be used at boundaries do not affect the calculated results around the target building. For outlet boundary condition, constant pressure conditions are recommended. Symmetry conditions are used for the top and lateral boundaries.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 6, June 2019

III.6 NUMERICAL ANALYSIS USING CFD

The numerical analysis is carried out by the boundary domain as per Bilal, 2015. The size of the computational domain has been fixed to 1410x630x562.5m³. The geometry of the CFD model includes the modelling of computational domain and principal building. The geometry has been modelled in ANSYS Design Modeller 19.0. The meshing are done for discretization into tetrahedral elements. On conducting a grid independence study, the number of elements for meshing is confirmed and which is carried out in ANSYS Mesh.

Using the Continuity Equation and 3D RANS Equation, the CFD code executes. Viscous laminar k- ω SST model is used for turbulence modelling. The pressure velocity coupling runs on SIMPLE algorithm and pressure interpolation is second order. Conservation of mass and momentum equations uses the first order upwind discretization scheme. The cell-zone condition is defined as fluid (air). Boundary condition is set as velocity inlet for inlet, pressure-outlet for outlet. UDF velocities defined as the velocity profile are assumed to be parabolic in shape.

Table 3.1 The parameters for CFD analysis using ANSYS Fluent

Computational Domain (LxWxH)	1410x630x562.5 m ³
Principal Building (BxDxH)	30x30x225 m ³
Mesh	unstructured Tetrahedral
Turbulent flow calculation	RANS Equation
RANS Turbulence Equation	k- ω SST model
Discretization Techniques	SIMPLE algorithm
Pressure Interpolation	Second Order

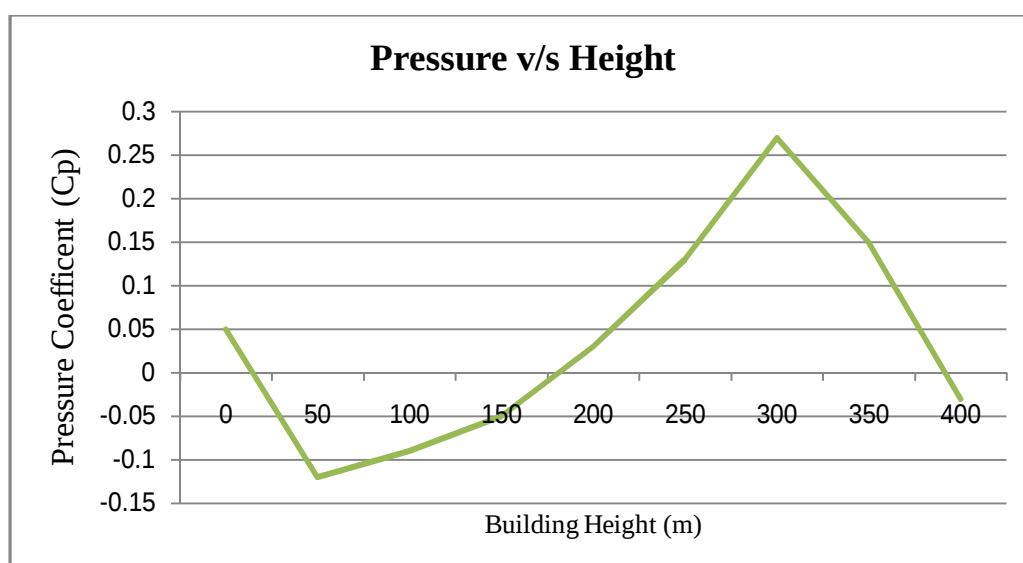


Fig. 4.1: Pressure variation from CFD

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 6, June 2019

III.7 EXPERIMENTAL VALIDATION OF WIND TUNNEL TEST

The experimental validation aims that proving of the Computational Fluid dynamics can be a substitute to the conventional Wind Tunnel test. In the experimental analysis, two parameters were compared to check, the pressure on the face of the building and the flow pattern. A similar situation is arranged in numerical analysis using CFD analysis using ANSYS Fluent 19.0. In CFD analysis, the results such as flow pattern and pressure contour were computed. The results from experimental analysis and numerical analyses are compared.

From the experimental analysis of Wind Tunnel test, the pressure impinged on the building surface due to wind load was conducted using a Wind Tunnel test set up. The testing using Wind Tunnel is always done at reduced scale ratio of 1:400 to 1:600. By experimental study the scaling has been fixed to 1:600. The principal building modelled for analysis of pressure acting on the building due to effect of wind load. In order to tap the pressure, pressure tubes were inserted into the holes left while modelling the building. The pressure tubes connected to the manometer. The manometric liquid was methanol. The effect of wind hitting the structure causes pressure on the building surface. The pressure variation was clearly depicted in variations in the manometer. The pressure was noted down and plotted.

Table 6.1: The pressure values from the wind tunnel test

Front Face Of Building				
Port No.	Height	H	Pressure	Pressure coefficient (Cp)
1	25	136.3	1056.31	0.0987
2	50	134.0	1038.49	0.1345
3	75	135.6	1050.89	0.1096
4	100	138.8	1075.69	0.0598
5	125	140.3	1091.19	0.0286
6	150	140.8	1087.31	0.0364
7	175	142.0	1100.49	0.0099
8	200	140.0	1084.99	0.0411
9	225	136.5	1057.86	0.0956
10	250	132.0	1022.99	0.1656
11	275	129.5	1003.61	0.2046
12	300	128.3	994.31	0.2232
13	325	128.0	991.99	0.2279
14	350	128.8	998.19	0.2155
15	375	140.7	1090.41	0.0302

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 6, June 2019

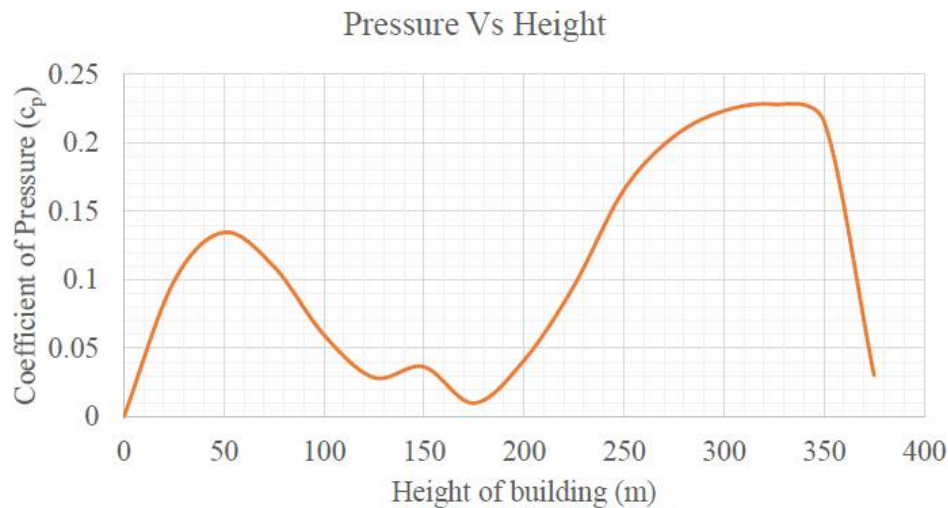


Fig. 4.1: Pressure variation from Wind Tunnel test

IV. EXPERIMENTAL RESULTS

In this study, focuses on different fluid parameters namely vorticity, Pressure drop in front and behind the building and Turbulence kinetic energy. These parameters are selected because of their immense importance in quantifying the fluid dynamics.

IV.1 VORTICITY

Vorticity is mathematically defined as the curl of velocity field. In simple terms, the measure of local rotation of the fluid. This results in the phenomena of vortex shedding. The frequencies of the vortices are dependent on the shape of the blunt body and the velocity of the fluid flow or wind hitting this body. The vortices create low pressure at the downwind side of the building. For structures like tall and uniform in size and shape, the vibrations can be damaging and ultimately lead to fatigue failure. Masts or tower is highly susceptible to vibrations induced by vortex shedding.

Table 4.1: The Vorticity of Different Geometry Studied

Square	3.003
Triangle apex	2.557
Triangle base	2.900
Circular	6.134
Ellipse horizontal	3.054
Ellipse vertical	4.059
Square plus	4.922
Hexagon	4.620

International Journal of Innovative Research in Science, Engineering and Technology

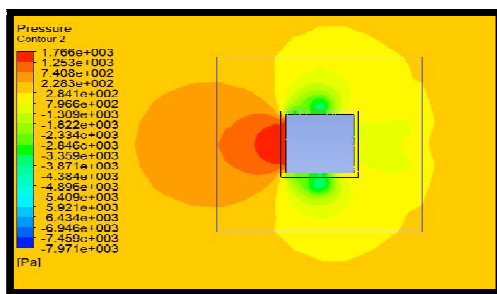
(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

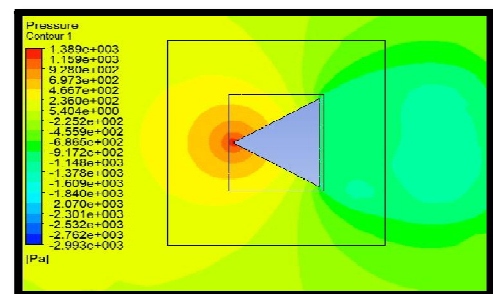
Vol. 8, Issue 6, June 2019

IV.2 PRESSURE DROP

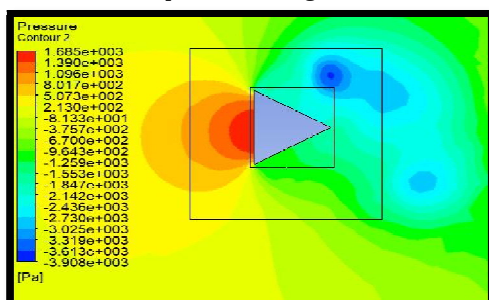
When the wind load hitting on the surface of the tall building which results the development of pressure on the surface facing directly to the wind. The energy of hitting gets dissipated around corners. Comparatively high-pressure regions are formed in front of building and low-pressure region at behind building. The structural integrity is depends upon the drop-in pressure in the vicinity of building, mainly in front and behind of building in the direction of wind hitting. The minimum pressure drops are experienced by ellipse in horizontal direction, followed by circular building and ellipse in vertical direction. The maximum drops in pressure are experienced by triangular building with base facing wind. The results in pressure drop shows that the sharp edges can result in greater pressure drop.



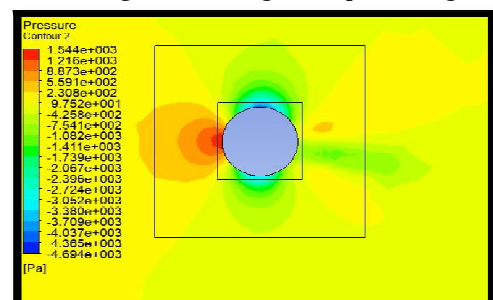
Square Building



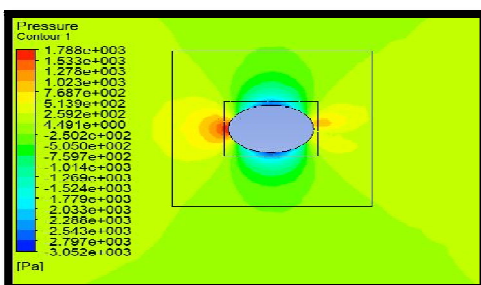
Triangular Building with apex facing



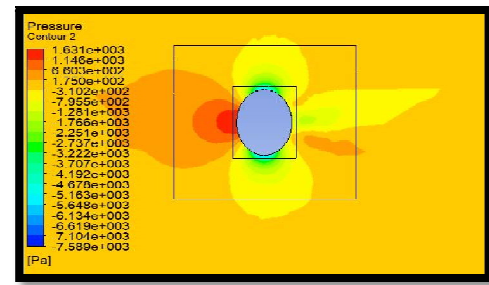
Triangular Building with base facing



Circular Building



Elliptical Building horizontal



Elliptical Building vertical

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

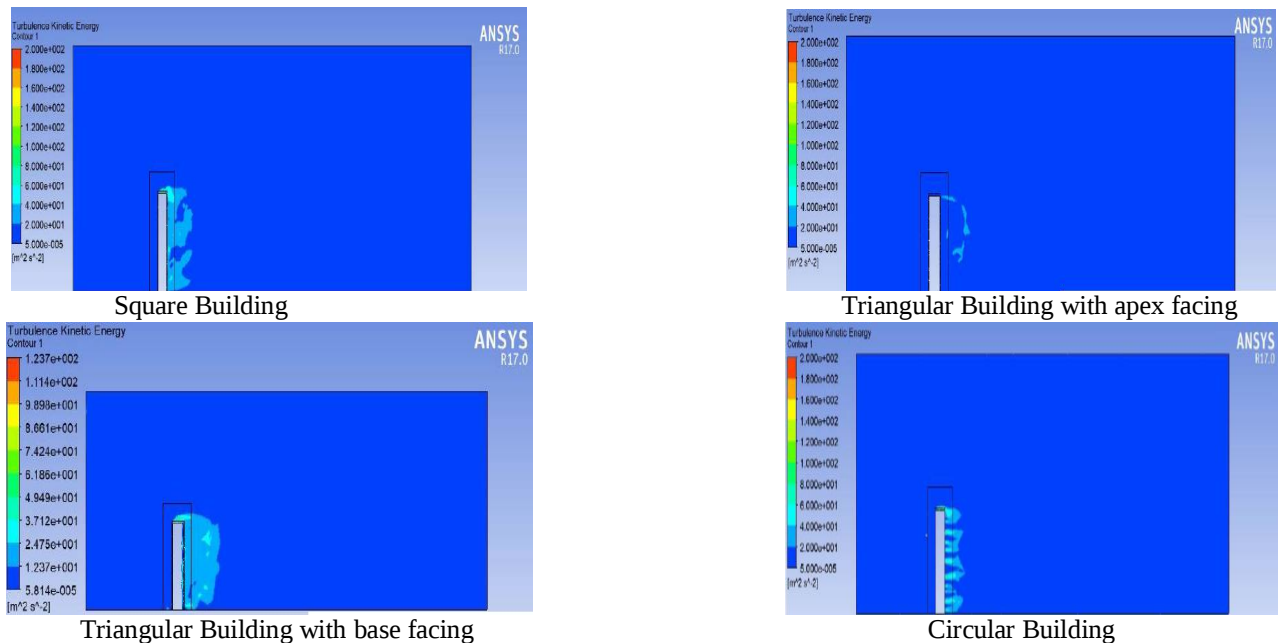
Vol. 8, Issue 6, June 2019



Fig. 4.1 The pressure contours of different geometry

IV.3 TURBULENCE KINETIC ENERGY

Turbulent flow are characterized by highly transient particles called eddies. Eddies can be very small and also very large. Larger the size, higher is the energy content of eddies. Turbulence Kinetic energy is directly related to vorticity. The kinetic energy is increases with increase of vortex formations. Larger the size of eddies more is the kinetic energy possessed by the eddies. The drag caused by the building depends on the energy possessed by the eddies. Turbulence Kinetic energy is possessed by triangular building with apex facing the wind.



International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 6, June 2019

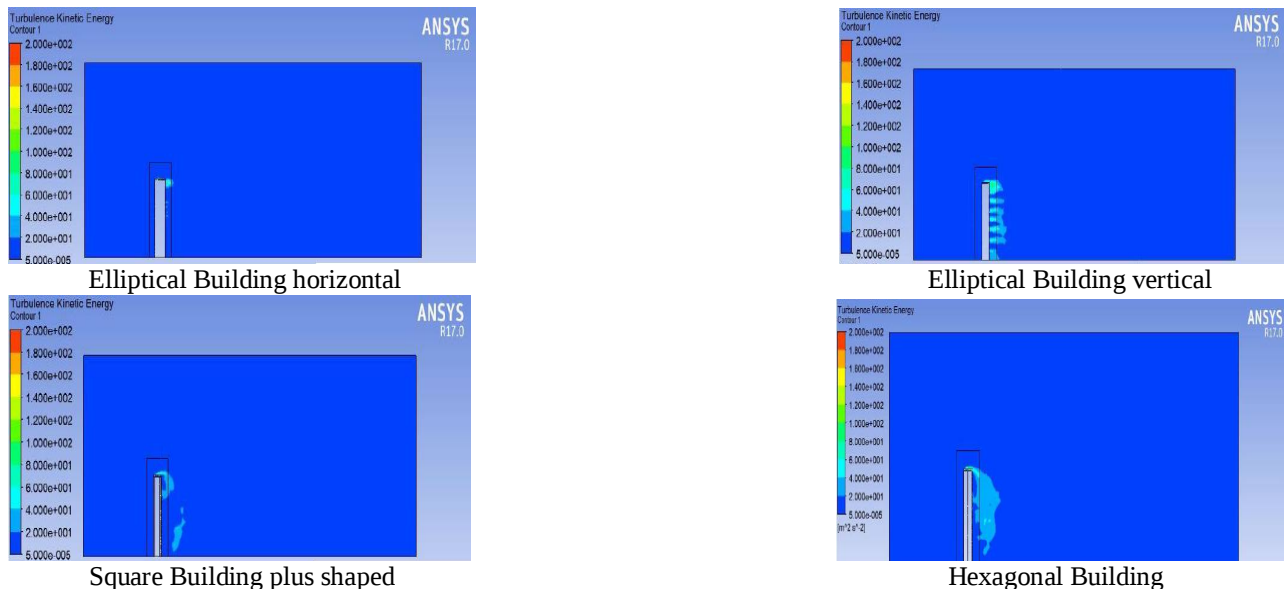


Fig. 4.2: The Turbulence Kinetic Energy for different geometry

V. CONCLUSION

❖ Comparison Results Of Wind Tunnel Test And CFD

This study has been conducted to validate the results from Wind Tunnel test and numerical analysis using Computational Fluid Dynamics. The Coefficient of Pressure versus height of the building (fig.4.1 and 4.2) shows that conformity with each other. So that Computational Fluid Dynamics can be a substitute to the Wind Tunnel testing. It proves that advantages over Wind Tunnel test:

- No requirements of physical setup.
- An initial investment for commercial software no cost involved.
- Can model most physical phenomenon.
- Can gather lots of data by in one simulation.

The time and cost on numerical simulation by CFD is less than compared to the wind tunnel test. Then CFD can be used for computing effect of wind load on high-rise structures.

❖ Inference From CFD Analysis

- The vortices provide low pressure at the downwind side of the building. Lowest vorticity is at triangle building with apex facing wind and highest is at circular building.
- The minimum pressure drops is at ellipse in horizontal direction, followed by circular building and ellipse in vertical direction. The maximum pressure drop is at triangular building with base facing wind. The sharp edges can result greater pressure drop.
- The kinetic energy is increases with increase of vortex formations. Least Turbulence Kinetic energy is at triangular building with apex facing the wind and more for square fillet.

Of the 8 different geometry considered, elliptical building oriented in the horizontal direction provide the least pressure drop with a comparatively lower turbulence. Then ellipse in horizontal is the optimized shape and is considered for fluid-structure interaction by incorporating different structural systems.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 6, June 2019

REFERENCES

- [1] Bilal Assaad, "Wind Effect on Super-Tall Buildings Using Computational Fluid Dynamics and Structural Dynamics" May 2015
- [2] Saurabh Ranka and Prof. V.S Shingade, "Analytical and Numerical Analysis of Wind Load on Tall Buildings" International Journal of Science, Engineering and Technology Research (IJSETR) Volume 5, Issue 11, November 2016
- [3] Okafor Chinedum Vincent, "Application of Computational Fluid Dynamics Model in High-Rise Building Wind Analysis-A Case Study" Advances in Science, Technology and Engineering Systems Journal Vol. 2, No. 4, 197-203 (2017)
- [4] Chunming Liu, Liang Liu and Chengbin Liu, "Analysis of wind resistance of high-rise building structures based on computational fluid dynamics simulation technology" International Journal of Heat and Technology Vol. 36, No. 1, March, 2018
- [5] A.K. Roy and Mohd. Mohsin Khan, "CFD Simulation of Wind effects on Industrial Chimneys" Civil Engineering Conference-Innovation for Sustainability (CEC - 2016), September 09 -10, 2016
- [6] C. L. Fu, S. M. Lee and C. M. Cheng, "Validation of CFD simulations on the wind loads for tall buildings' preliminary design" The Fourth International Symposium on Computational Wind Engineering (CWE2006), Yokohama, 2006
- [7] K. Rama Raju, M.I. Shreef, Nagesh R Iyer and S. Gopalakrishnan "Analysis And Design Of Rc Tall Building Subjected To Wind And Earthquake Loads" The Eighth Asia-Pacific Conference on Wind Engineering, December 10-14, 2013, Chennai, India
- [8] Dr. Salah R. Al Zaidee and Alfhadhel B. Kasim "Validation of Computational Fluid Dynamics, Turbulent Flow Approach, for Wind Flow around Building" International Journal of Science and Research (IJSR) ISSN, Volume 6 Issue 3, March 2017.