International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 07 Issue: 01 | Jan 2020 www.irjet.net p-ISSN: 2395-0072

Tall-Building Structure Shape Optimization using "Computational Fluid Dynamic (CFD)"

Samir Kumar Singh[1]

[1]M.Tech in Structural Engineering, Dept. of Civil Engineering, Indira Gandhi Institute of Technology, Odisha, India.

Abstract - *Wind plays an important role in tall buildings. So it is necessary to analyze wind behavior around and over these tall buildings. In tall buildings, wind pressure can reduce by adopting the suitable shape of buildings. CFD (computational fluid dynamics) simulation is a suitable option for study the sensitivity of wind pressure over tall buildings. Analyzing the different building shape with constant air flow around the buildings; after that, we can select one of the building shapes. In this paper, we are using ANSYS 17.2 for CFD simulation. Here we analyze turbulent flow; so we are taking a turbulent model which is "realizable k-ε turbulent model". We are taking realizable k-ε turbulent model because it is more accurate than SST k- ε, standard k- ε, RNG k- ε models. A series of wind test have been carried out to determine the wind characteristics of many buildings with various configurations.*

***Key Words*:** Ansys, CFD, Tall Buildings, Wind analysis, Turbulent flow.

1. INTRODUCTION

A tall building cannot be defined in specific terms related just too height or the number of the floors. A measurable definition of a tall building cannot be universally applied because the tallness of a building is a matter of community's circumstance and their consequent perception [1]. From the structural engineer point of view, a tall building can be defined when the structure is affected by the lateral loads. Due to lateral forces building sway or drift at the top of the building relative to its base. Drift is the magnitude of lateral displacement. The lateral forces on buildings are increase with the height of the building. Wind and earthquake are the important lateral forces which play an important role in structural design. Therefore these forces considered from the very beginning of the design process. In tall buildings, wind loads are one of the most symbolic forces which are responsible for natural disasters.

Wind forthcoming a building is a complex issue; where the flow pattern generated around the building is complicated. Large wind pressure fluctuations develop on the surface of a building due to the mean flow, the flow separation, the vortex formation, wake development characteristics of wind. As a result, large aerodynamic loads are imposed on the structural system and the fluctuation of force also large on the elevation of the building. Due to these fluctuating forces; a building will vibrate in linear as well as a torsional mode. This is called vortex shedding. The amplitude such of such oscillations is dependent on the nature of aerodynamic forces and the stiffness characteristic and dynamic characteristic of the buildings [2]. Therefore, the building shape is also the most important aspect of a tall building design. Effects of wind on tall buildings mainly depend on the magnitude of wind speeds and directions of wind. The response of buildings to wind loads basically 3 types as illustrated in Figure 1; these are:-

1. Along with wind response
2. Crosswind response
3. Torsional effects

1.1 Along with wind response

Due to wind flow on the face of building the building displace horizontally in the direction of wind flow. This displacement is occurring due to drag forces. Pressure fluctuations on the windward face and leeward face (back face of the building) because of these drag forces.

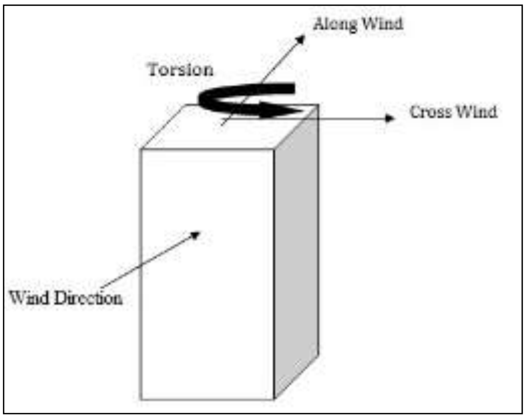

Fig -1: Wind response directions

Some techniques such as the "gust loading factor" approach are developed to predict forces as well as a response in the along wind direction. The gust factor approach is used in international standards codes to calculate the dynamic forces along wind direction and their effects on high rise buildings [3].

1.2 Cross wind response

The crosswind response means deformation perpendicular to the wind flow direction due to wind forces. Tall buildings have opposed the streamline of wind that causes the flow to separate from the surface of the structure, rather than follow the body contour. The wake flow thus created behind the building produce various degrees of periodicity, ranging from virtually periodic with a single frequency to fully random. In each of the cases, at any given instant, the wake flow is asymmetrical. The lateral turbulent fluctuations in the oncoming flow may also contribute to the crosswind forces. The complex nature of the crosswind loading which results from an interaction of incident turbulence, unsteady wake effects, and building motion has inhibited reliable theoretical predictions.

1.3 Torsional effects

Torsional or twisting moments are induced in the buildings when the wind flows normal to the face of prismatic type buildings by variations in the fluctuating wind velocity around the face of the buildings. The larger building has larger width also, so the greater will be the fluctuating torsional moment. For the turbulence, buffeting and vortex shedding these fluctuating torsional moments are generate. Because of the eccentricity between the instantaneous aerodynamic center and building's center of rigidity; torsional motions of a building occur.

Most of the modern buildings and structures usually result in a natural frequency in a torsional mode that is greater than the lowest translational natural frequency. Hence, torsional motions will develop only after lateral motions are induced in the structure. By structural modifications; the effects of the torsional moment can be controlled. Due to the lack of firm guidance from design codes to estimate the torsional effects accurately, structural engineers depend on "wind tunnel tests" to evaluate the torsional behavior of tall buildings.

2. COMPUTATIONAL FLUID DYNAMICS (CFD)

One of the main end products of this analysis is to find the pressure variations on the surface of the building using a rigid model approach. However, wind tunnel testing has its own limitations, including the disguise of Reynolds number due to the use of scaled models. **Mr. E Achenbach** has been investigated the effect of Reynolds number on the pressure coefficients of circular cylinders [4]. The output showed that pressure values changed according to the different Reynolds numbers. The scaled down models used in wind tunnel and full-scale wind flow have significantly different values of Reynolds numbers. Wind analyzer generally adopts rough surfaces to simulate turbulent flow which is comparable to real life condition; however, this is a rudimentary method as the degree of roughness required to simulation the real wind behavior is unknown. Therefore, significant data studies are required before the simulation for selecting the roughness of elements. And another additional limitation of wind tunnel test is time requirement, cost and need for technical expertise. Computational Fluid Dynamics is a good solution for the above difficulty.

In the modern world, Computational Fluid Dynamic (CFD) is progressively used in wind engineering to analyze wind loads, pedestrian level wind comfort and wind energy harvesting, pollutant dispersions. The main advantage of CFD is the capability to simulate full-scale models within full-scale atmospheric boundary layer (ABL) flows by creating a virtual domain. In wind tunnel test parameters are only calculated at selected points, but CFD can provide detailed information about wind pressure, velocity, streamline, drag force, lift force, streamline of flow, etc. CFD is more flexibility than conventional wind tunnel method because wind tunnel test requires material, skilled labor, time resources[5].

Computational Fluid Dynamic (CFD) is a branch of fluid mechanics that use to analyze and solve problems that involve fluid flows by using numerical analysis and data structures. To simulate the free stream flow of the fluid, computers are used to perform the required calculations. High-speed super-computers are required when the simulation is a complex problem. Some time high-speed super-computers are used to calculate an accurate solution. In CFD analysis, the examination of fluid flow in accordance with its physical properties such as velocity, pressure, temperature, density, and viscosity is conducted.

3. TURBULENT FLOW

In fluid dynamic when the flow motion is disorderly changes in pressure and flow velocity; then the flow is called as turbulence or turbulent flow. Turbulence is commonly observed in our daily life such as fast flowing rivers, billowing in storm clouds, smoke form by a chimney and most fluids flows occurring in nature and created in engineering application are turbulent. When the excess kinetic energy generated in part of the fluid flow; then turbulence is generated in fluid flow. Because of the kinetic energy of fluid overcomes the damping effect of the fluid's viscosity. Turbulence is easier to create in low viscosity; but more difficult in a highly viscous fluid.

The Reynolds number (R_e) is an important dimensionless quantity in fluid mechanics; which is used to describing the flow characteristic whether it is laminar flow or turbulent flow. If the Reynolds number is low then the flows are laminar flow and if the Reynolds number is high then the flow is turbulent flow. In case of flow through pipes of diameter 'D' in a fully developed region, laminar flows occur when Reynolds number (R_e) is less than 2300 and turbulent flow occur when Reynolds number (R_e) is greater than 2900. In case of flow over a Flat plate; if the Reynolds number (R_e) is above the value of ($2*10^5$) to ($3*10^5$) then the flow is called as turbulent flow. Normally Reynolds number (R_e) is above ($5*10^5$) is taken as turbulent flow.

Reynolds number can be defined as;

$$R_e = \frac{\rho v L}{\mu}$$

Where,

ρ = Density of the fluid (SI units: kg/m^3)

v = Characteristic velocity of the fluid with respect to the object (m/s)

L = Characteristic linear dimension (m)

μ = Dynamic viscosity of the fluid (Pa□or N□/m ^{2} or kg/(m□)).

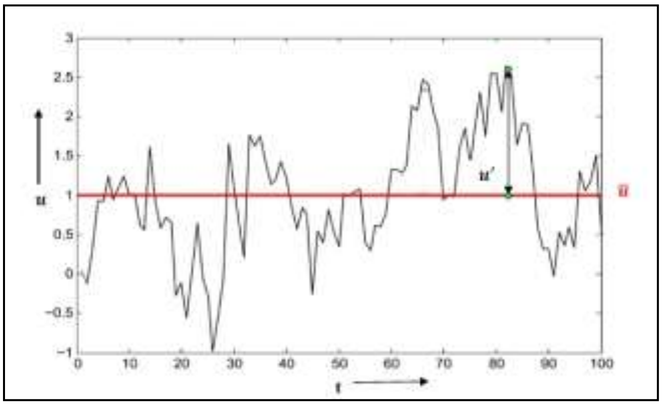

Fig -2: Velocity versus time

Refer to **Fig -2,**

Where,

t = time

u = velocity

$\bar{u}$ = Reynolds average velocity

u' = fluctuation velocity

As shown in **Fig -2**; this is a graph of velocity versus time. As it is a turbulent flow; so the velocity is fluctuating at a high rate. We consider an average velocity '$\bar{u}$' and fluctuation about the average velocity is "u'"; which is called as Reynolds average velocity and fluctuation velocity. I can decompose the velocity 'u' into the average velocity '$\bar{u}$' and fluction velocity 'u''.

We can write,

$$u = \bar{u} + u'$$

The above equation is called as "Reynolds decomposition". In most engineering problems we need to calculate only in "Reynolds average velocity" and "Reynolds average pressure".

3.1 Reynolds Average Navier- Stokes Equation

In Physics, the Navier-Stokes Equation named by 'Claude-Louis Navier' and 'George Gabriel Stokes'. This equation describes the motion of viscous fluid substances. Navier-Stokes Equation arises by applying 'Newton's second law' to 'fluid flow motion', and then together with the assumption that the stress in the fluid is the sum of the fluid is the sum of 'diffusing viscous' term and a pressure term which can describe viscous flow. Navier-Stokes Equations are very useful for describing and analyzing the physical phenomena of scientific and engineering. This equation can model the ocean currents, water flow in a pipe and air flow around a wing can help to design of aircraft and cars, the study of blood, etc.

Navier-Stokes Equations for x-direction moment of conservation is written below,

$$\rho \frac{\partial u}{\partial t} + \rho \left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = -\frac{\partial p}{\partial x} + \mu \nabla^2 u$$

Here, $\rho \frac{\partial u}{\partial t}$ is an extra term comes dure to unsteadiness in the flow: the flow is not really steady here because we have these fluctuations. So we have to extend the Navier-Stokes Equation to 'unsteady case' and we will get this extra term $\frac{\partial u}{\partial t}$.

4. TURBULENCE MODELS

In CFD simulations 'turbulence modeling' is a key issue. All most all engineering applications are turbulent and hence turbulence model is very important in CFD simulations. Turbulent models are constructed for use of the mathematical model to predict the effects of turbulence. In case of simple flow only, the equations governing turbulent flow can be solved directly. In most of the real-life turbulent flows; CFD simulations predict the evolution of turbulence by using the turbulence model. Constitutive relations predict the statistical evaluation of the turbulence model, and these constitutive equations are simplified by turbulence models.

4.1 Two equation turbulence models

Two equation turbulent models are one of the most common types of turbulence models. Models like the k-epsilon model and k-omega model are commonly used for most types of engineering problems; hence these models have become industry standard models. Two equation turbulence models are also very much use in research work and new refined two-equation models are still in underdeveloped.

According to the definition, two equation models include two extra transport equations to represent the turbulent properties of the flow. These extra transport equations allow a two-equation turbulence model to account for history effects like convection and diffusion of turbulent energy. Mostly one of the transported variables is 'turbulent kinetic energy (k)'. The second transported variable varies depending on what type of two-equation model it is. Commonly used are 'turbulent dissipation (ε)' or the specific turbulence dissipation rate (ω).

The first variable kinetic energy (k) determines the energy in the turbulence and second variable can be thought of as the variable that determines the scale of turbulence (i.e; length-scale or time-scale). In complex flows, it is impossible to guess the distribution of length scales. It may be difficult to define the distance to the wall and the local level of 'l' is affected by convection and diffusion processes. One approach to avoid having to prescribe 'l' building on the modeling, by adopted for 'k' is to obtain 'l' from its own separate equation. Thus, we devise a transport equation for some length scale containing a variable of the form "$k^a\ l^b$". The most popular choice of the second variable has been esplion (ε) which is the dissipation rate of 'k'.

We know that

$$\varepsilon = \frac{k^{3/2}}{l}$$

Here, a= 3/2 and b= -1

Other choices of the variable have been proposed which is length scale (l) or frequency (f).one alternative that has been widely tested is $\omega = \varepsilon/k$

Generally, two types of two-equation models are mostly used. Which are,

1. k-epsilon model (k-ε)
2. k-omega model (k-ω)

5. METHODS

5.1 Outline of the model

This simulation was conducted in "computer lab; civil engineering department" of Indira Gandhi Institute of Technology, Sarang. I am using ANSYS 17.2 software for my simulation work. For CFD simulation I am using ANSYS FLUENT tool.

In this simulation my interest to find the best shape for the tall building when turbulent flow around the tall building using aerodynamic. In other words, this simulation is conducted for tall buildings shape optimization using aerodynamic. Here I am taking six different shapes of the building to find which shape is better. Different shapes of the building are a circular shape, square shape, rectangular shape, square with four edge chamfer shape, Y shape, and oval shape. Again I take six different dimensions of oval shape building for better analysis. In my simulation, I am taking a total of eleven number of the different shape of buildings. Floor area of each building model is 100 m² and the height of the building model is 50 m. Flow domain shape varies according to the shape of the building, in some case it is square in shape and in other it is rectangular. In our case, I consider floor area of the flow domain is 2500 m² and height is 70 m.

Wind specification:-

Wind velocity = 50 m/s

Viscosity of wind = 1.7894×10^{-5} kg/ms

Density of wind = 1.225 kg/m³

Let's take a building model of square shape.

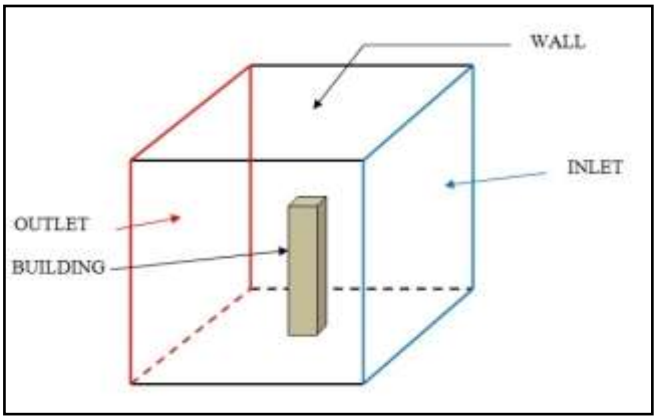

Fig -3: flow configuration and boundary locations for the square building

The flow configuration and boundary locations for the square building are shown in **Fig -3**. In this simulation turbulent wind flow from the inlet to the outlet within the flow domain. We consider turbulent wind flow velocity is 50 m/s. I want to find the pressure around the building wall after turbulent air flow around the building within the flow domain. On which building shape the pressure on the building wall is small, that one is the best aerodynamic shape of the tall building.

In my case there are three boundary conditions are present in my model.

1. Inlet
2. Outlet
3. Wall

Here I am taking six different shapes of the building shown in **Fig -4.** All building has the same floor area which is 100 m² and the same height also which is 50m. One by one I will check which building shape reduce the wall pressure. In which building small pressure acting on the wall will be the best shape for our tall buildings.

The input parameters I will use are:-

The cross-sectional area of the building is 100m².

Free stream velocity is 50 m/s.

The density of air is 1.225 kg/m³

The viscosity of air is $1.7894*10^{-5}$ kg/ms

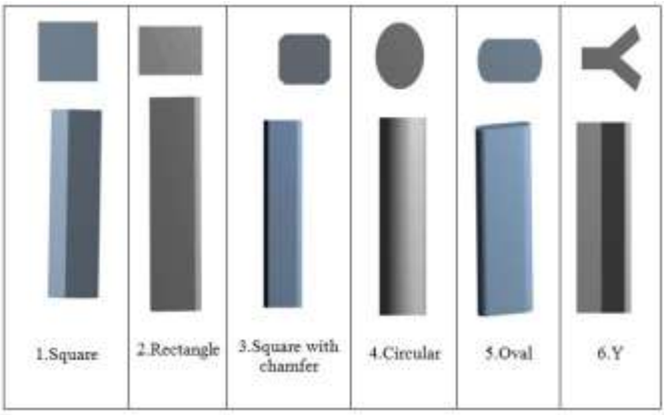

Fig -4: Different shapes of the building

6. RESULT AND DISCUSSION

6.1 Test result for square building

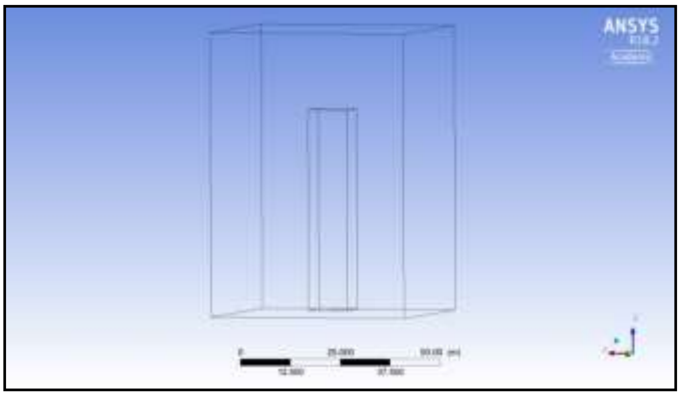

Fig -5: Square shape building model

As shown in **Fig -5;** I take a square shape building and start my simulation. After applying wind load our interest to find out the net pressure acting on the surface of the building. I found that the maximum pressure on the surface of the building is 3071.43 Pa.

The velocity of wind also changes in different place within flow domain. The velocity of the wind varies from 0 m/s to 72.8261m/s.

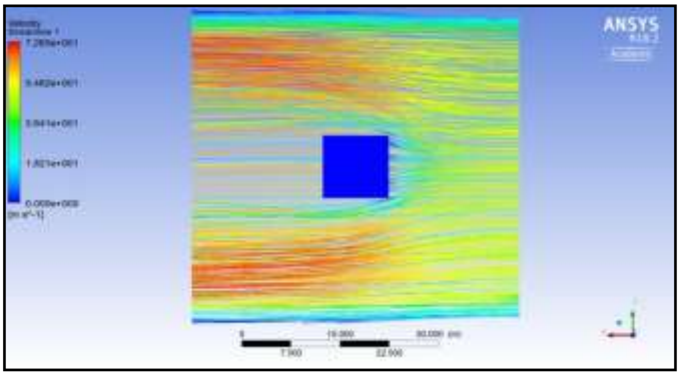

Fig -6: Velocity streamline

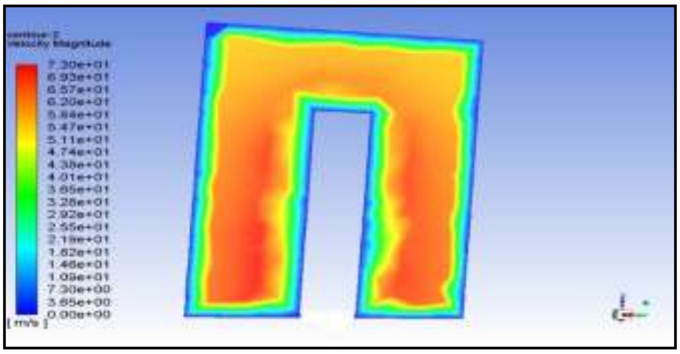

Fig -7: Velocity contour

2D represented of velocity contour of wind flow is shown in **Fig -7**. From the above figure, we found that velocity is decreased near the wall. Velocity is almost zero at the wall of the building. The velocity of wind gradually increases from the wall and after some distance from the wall; the velocity is at the pick. After approaching the maximum velocity the wind speed again decreases.

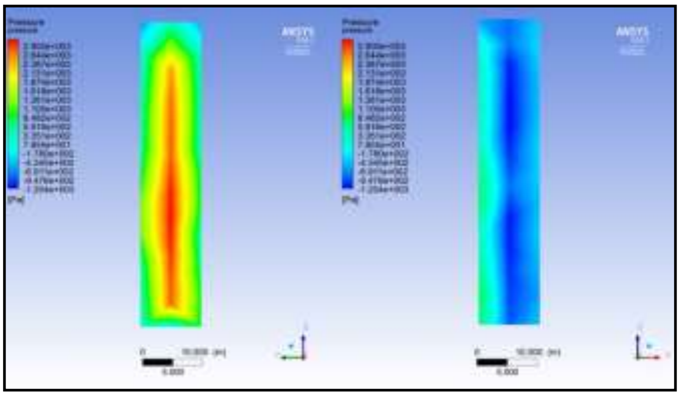

Fig -8: Surface pressure

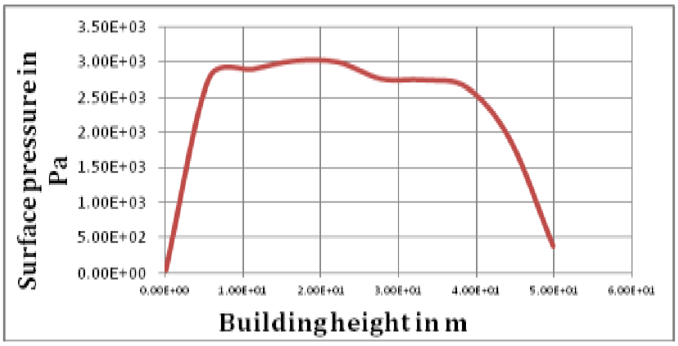

Chart -1: Surface pressure

By pressure coefficients, the surface pressure on a building is generally described. A pressure coefficient describes the relative pressure throughout a flowing fluid in fluid dynamics and it is a dimensionless number. In aerodynamics and hydrodynamics, the pressure coefficient is used. Pressure coefficient has its own unique pressure coefficient at every point in a fluid field. Pressure coefficient calculated through normalization with velocity pressure in the undisturbed approaching wind at reference height; which is often chosen as the height of the building.

Pressure coefficient can represent as "C_p".

$$C_P = \frac{\text{change from the free stream pressure (gauge pressure)}}{\text{dynamic pressure in the free stream}}$$

$$C_P = \frac{P - P_\infty}{0.5\rho v_\infty^2}$$

Where

P = static pressure at the point at which pressure coefficient is being evaluated.

P_∞ = static pressure in the free stream.

v_∞= free stream velocity

Chart -2: Pressure coefficient

Chart -2 graphically represents the pressure coefficient against the building height. In this simulation maximum pressure coefficient for square shape building is '1.97'.

6.2 Comparison of all shapes building

I have already conducted simulation against the different shapes of the building, and those are a square shape, a rectangular shape, square with chamfer shape, a circular shape, Y shape. Now I am going to compare and study which is the best shape for the tall building.

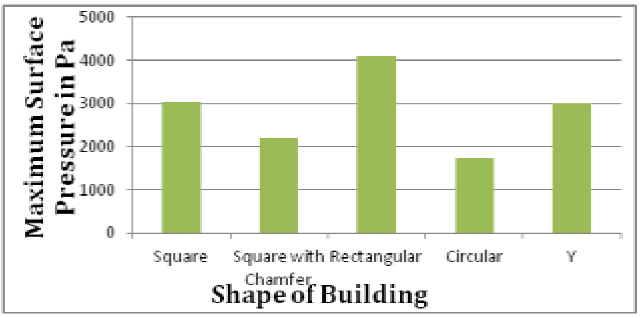

Chart -3: Surface Pressure of different shapes building

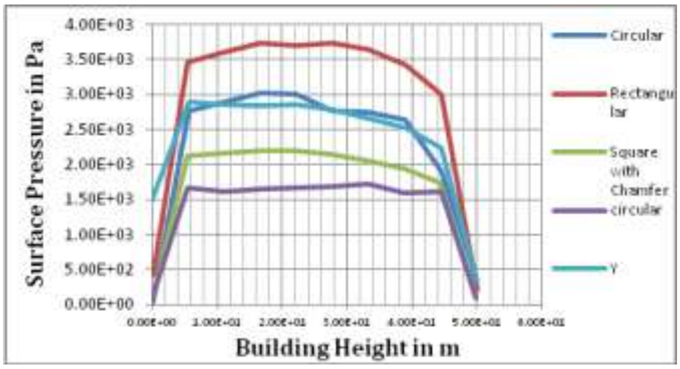

Chart -4: Surface Pressure of different shapes building

In **Chart -4**, I represent the surface pressure against the different shape of the buildings. Here also clearly shown that surface pressure acting very little on the circular shape of the building, and the maximum amount of surface pressure acting on the rectangular shape building. On square with chamfer shape building; surface pressure is little larger then the circular shape and Y shape building's surface pressure is larger than square with chamfer shape building. On square shape, surface pressure is less than the rectangular shape. Maximum surface pressure on the different buildings is given in **Table -1**.

Table -1: Maximum Surface Pressure

Building Shape	Maximum Surface Pressure (in Pa)
Square	3071.95
Square with Chamfer	2202.78

Rectangular	4096.06
Circular	1737.46
Y	2995.53

6.3 Test result for oval shape building

We already see that the circular shape of the building is the best shape because of the pressure coefficient very low in the circular shape building. Now I am going to modify the circular shape building to increase the floor area. So I manipulate the dimension of circular shape building to convert to **oval** shape building as shown in **Fig -9.**

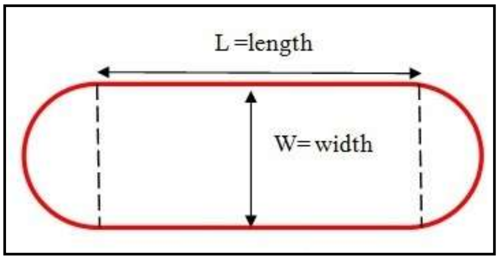

Fig -9: Oval shape building

Here I am going to change the dimension of the length (L) but the width will be constant at 10m. With different dimension, I will conduct the simulation to check the pressure parameter. I take a Y shape building and start my simulation. After applying wind load our interest to find out the net pressure acting on the surface of the building. Here I will conduct four different dimensions of oval shape building. The dimensions of the length (L) of five oval shape model are 0.5W, 1.0W, 1.5W, 3W. And the width (W) of five different models will be the same which is 10m. Height remains the same for all buildings which are 50m.

Table-2: Maximum Surface Pressure of oval building

Oval dimension (L)	MAXIMUM SURFACE PRESSURE IN (Pa)
0.5W	2299.05
1.0W	2321
1.5W	2345.05
3.0W	2390.22

After simulation all the four building I found maximum pressures as shown in **Table-2.** Here I found that if I increase the length the building then surface pressures are also increasing accordingly.

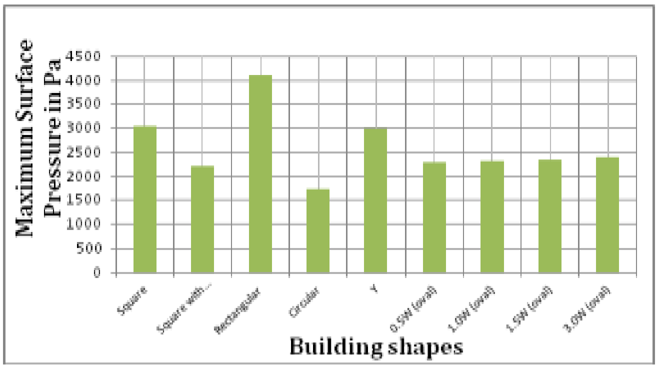

Fig -10: maximum surface pressure of all buildings

7. CONCLUSIONS

First I consider five different building shape which is a circular shape, square shape, rectangular shape, square with chamfer shape, and Y shape. After simulation, these five shape building I get minimum surface pressure on the circular building which is 1737.46(Pa), and maximum surface pressure on the rectangular shaped building which is 4096.06(Pa). And the floor area of both circular and the rectangular building is $100m^2$. After I conduct the same simulation on the oval shaped building. I take four oval-shaped building with different length, i.e; 0.5W, 1.0W, 1.5w, 3.0W. Height remains the same for all buildings which are 50m. Here I get maximum surface pressure on the oval building which length is 3.0W, and surface pressure value is 2390.22(Pa).

On rectangular shaped building I get 4096.06(Pa) of surface pressure which floor area is $100m^2$ only. But in the case of the oval-shaped building, I get 2390.22(Pa) of surface pressure which floor area is $378.5m^2$. So oval shape is more suitable for tall building than the other four models (i.e; square shape, rectangular shape, square with chamfer shape, and Y shape) because surface pressure is small even the floor area is larger than another model. So I take oval shape as the best shape for a tall building which is the best shape to resisting the turbulent air flow.

REFERENCES

[1] Structural analysis design of tall buildings by Bungale S. Taranath.

[2] wind analysis and design of tall buildings, the state of the art (ICSECM2017 – 463)

[3] Li, Q.S., J.R. Wu, S.G. Liang, Y.Q. Xiao, C.K. Wong, 2004. Full-Scale Measurements and Numerical Evaluation of Wind-Induced Vibration of 1 63-Story Reinforcedconcrete Tall Building. Engineering Structures.

[4] Distribution of local pressure and skin friction around a circular cylinder in cross flow up to Re = $5*10^6$ By E. Achenbach

[5] Comparison of Reynolds Averaging Navier-Stokes (RANS) turbulent models in predicting wind pressure on tall building ; By D. Mohotti , K. wijesooriya

AUTHOR

Name :- Samir Kumar Singh
BE (Civil Engineering)
ME (Structural Engineering)