International Journal of Emerging Technology and Advanced Engineering
(E-ISSN 2250–2459, UGC Approved List of Recommended Journal, Volume 7, Special Issue 2, December 2017)

Effect of Plan Shape on the Wind Pressures On buildings - A CFD Approach

A. Balaji Rao¹, M. Durga Abhishek²
¹Associate Professor, CBIT, Gandipet, Hyderabad-500075, India
²ME-Structural Engineer, CBIT, Gandipet, Hyderabad-500075, India
¹beyaarl@yahoo.com, ²crike.abhi@gmail.com.

Abstract-- Lateral loads i.e. wind load, seismic load, govern the design of tall buildings and their computation is of paramount importance for the efficient analysis of structures. For very tall buildings, wind loads are more predominant than seismic loads and the present wind load code IS 875 (PART-3) provides provisions for design pressure and force coefficients for some standard shapes. But with the present trend of adopting complex geometries for buildings, the present specifications are inadequate for the computation of wind loads. For such cases, wind tunnel testing, which is required to generate equivalent atmospheric turbulence properties and boundary layer flow inside the wind tunnel can be adopted, but is too costly and time consuming. In such a scenario, computational fluid dynamics, an analytical tool comes handy and provides a reasonable and economical solution.

Computational Fluid Dynamics, popularly known as CFD, basically involves obtaining numerical solution for the fluid problems often governed by Navier Stoke equations. It needs high speed computing systems and efficient algorithms. In the present work, an attempt is made to predict the wind pressures on buildings of various shapes with various floor heights and make a comparative study. K-epsilon turbulence model is considered for the analysis and software ANSYS - FLUENT is used for CFD analysis.

Keywords-- Computational fluid dynamics (CFD), Boundary layer, K-epsilon, UDF - Velocity profile.

I. INTRODUCTION

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 the earth’s surface. These eddies give wind its gusty or 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 gustiness tends to decrease with height.

The characteristics of wind pressures on a structure are a function of the characteristics of the approaching wind, the geometry of the structure under consideration, and the geometry and proximity of the structures upwind. The pressures are not steady, but highly fluctuating, partly as a result of the gustiness of the wind, but also because of local vortex shedding at the edges of the structures themselves. The fluctuating pressures can result in fatigue damage to structures, and in dynamic excitation, if the structure happens to be dynamically wind sensitive. The pressures are also not uniformly distributed over the surface of the structure, but vary with position.

The purpose of the present study is to investigate the dynamic behaviour of tall structures of various shapes when subjected to wind. For the simulation part domain size and mesh size influences the accuracy of the result. The boundary conditions and wall condition around the bluff body should be considered. The main focus of the present study is to reduce the unsteadiness of wake region around the structure, which creates high pressures, by considering the appropriate shape of the structure. Aerodynamic forces on tall building models with same area were using the pressure contours generated on various faces of models are calculated. [3]

II. OUTLINE OF MODELS CONSIDERED

A. Configuration of tall building models:

The tall buildings used for the experiments are square, circle, ellipse and parabolic shapes. The pressure contours are generated for various angles of attacks like 0, 90 and 180. The height of the structures considered are 150m, 195m and 240m. Selection of models were based on the aerodynamic nature of the buildings.

B. Computational Fluid Dynamics

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyse problems that involve fluid flows.
Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g., tests. The software used for the CFD problems are ANSYS FLUENT.

C. Reynolds Transport Theorem:

In fluid dynamics, it is more common to work with a control volume, defined as a region in space chosen for study. The size and shape of a system may change during a process, but no mass crosses its boundaries. A control volume on the other hand, allows mass to flow in or out across its boundaries, called as control surface.

As it is usually more convenient to work with the control volumes. And thus there is a need to relate the changes in a control volume to the changes in a system. The relationship between the time rates of change of an extensive property for a system and for a control volume is expressed by the Reynolds transport theorem.

\[ N_t = (N_1)_t + (N_2)_t \]

\[ N(t+\Delta t) = (N_2)(t+\Delta t) + (N_3)(t+\Delta t) \]

The net equation becomes,

\[ \frac{dN}{dt}(\text{System}) = \frac{\partial}{\partial t}(\text{CV}) + \text{Rate of out flow} - \text{Rate of inflow} \]

\[ \frac{dN}{dt}(\text{System}) = \frac{\partial}{\partial t} \int p \rho dV + \int p \rho (v.n) dA \]

Where: \( N = \) quantity to be conserved.
\( n = N \) per unit mass.
\( V_r = \) relative velocity.

From the above we can find the linear momentum equation,

\[ \sum F_{cv} = \frac{\partial}{\partial t} \int \rho v dV + \int \rho v (v.n) dA \]

Where \( F_{cv} \), \( i = F \) surface, \( i + F \) body, \( i \)

And F surface, \( i = \int T i dA = \int (\tau_i n) dA \)

\( F \) body, \( I = \int_{cv} b_i \rho dV \)

Now substituting the above 2 equation in linear momentum equation and converting the surface integral into volume integral

\[ \frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = \frac{\partial \tau_{ij}}{\partial x_j} + \rho b_i \]

The above equation is termed as NAVIERS EQUATION.

D. Navier Stokes Equation:

The above equation is not very useful to us as it is, because the stress tensor \( \tau_{ij} \) contains 9 components, six of which are independent because of symmetry. Thus in addition to density and the three velocity components, there are six additional unknowns, for a total of 10 unknowns. Of course to be mathematically solvable, the number of equations must be equal to the number of unknowns, and thus we need six more equations. These equations are called constitutive equations, and they enable us to write the components of stress tensor in terms of the velocity field and pressure field.

The first thing we do is to separate the pressure stresses and the viscous stresses. When a fluid is at rest, the only stress acting at any surface of any fluid element is the local hydrostatic pressure \( P \), which always acts inwards and normal to the surface.

When a fluid is moving, pressure still acts inwardly normal, but viscous stresses may also exist. So decompose it into 2 parts.

\[ \tau_{ij} = \tau_{ij} \text{ (Hydrostatic) } + \tau_{ij} \text{ (deviatory)} \]

From the definition of Newtonian fluids, Stress is linearly proportional to rate of deformation.

So we introduce \( \tau_{ij} =C_{ijkl} \).

Where \( C_{ijkl} \) has 81 combinations, so we require 81 equations.

\[ \tau_{ij} = \tau_{ji} \]

From symmetry these reduces to 36 combinations.

By substituting the pressure terms and assuming the stokes hypothesis we get the final equation as

\[ \rho \frac{\partial u_i}{\partial t} = - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_l} \left( \mu \frac{\partial u_i}{\partial x_l} \right) + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) \]

This equation is called Navier stoke equation.

This equation is valid for both compressible and incompressible fluids. The above equation is a coupled nonlinear differential equation.
When we solve the equation we get the velocity and pressure. As the equation is of coupled form, to obtain the solution we must use computational fluid dynamics (CFD).

III. MODELING

The main purpose of this study is to conduct CFD tests for tall buildings of different shapes. The method by which CFD application works can be broken down into three stages. a. Pre – Processor, b. Solver, c. Post -Processor

A. Domain Size:

Though the domain size very much affects the final solution, not much literature is available on the topic. It plays a vital role in forming fully developed flow or developed flow. It also must address the mathematical constraints under which CFD software solves the governing differential equation. Wool house (Wool house 2006) suggests the computation domain is determined by a blockage factor of 3%, which is defined as the ratio of the upwind building face area of buildings to the corresponding area of the computation domain, when the height and side spacing are equal, and the downward boundary distance in the range of 10 to 30 times the height. This method generally gives a bigger computation domain. So, fixing domain size is based on trial and error method until the results are converged, but distances Y1 and Y2 are important for the flow to develop.

The basic rules followed for fixing the domain are[4]

1. X1 = 2 to 5w
2. X2 = 12 to 20w
3. H = 2 to 5h
4. Y1, Y2 = 5w

Where X1 = distance from inlet to the structure front face
X2 = distance from the structure back face to the outlet.
Y1, Y2 = distance perpendicular to the wind flow

B. Mesh Size:

Fine meshing is used on the inner and outer surface of the building and near the corners of the building[1]. The measure of elements distortion depends upon the quality of element which is defined as the determinant of the Jacobian matrix. For obtaining the height of the first cell some calculations are done. If Reynolds number is more than 5 x 105, flow over a surface will be turbulent. For most of wind flow problems flow is turbulent. Now,

\[ \text{Re} = \frac{\rho v L}{\mu} \]

Where

\[ L = 4A/P \]

At Standard Sea Level density, \( \rho = 1.225 \text{ kg/m}^3 \).
Dynamic viscosity, \( \mu = 1.79 \times 10^{-5} \text{ Pa-s} \approx 1.8 \times 10^{-5} \text{ Pa-s} \).
Skin friction on a plate, \( C_f = 0.058 \times \text{Re-0.2} \)

\[ \tau_w = 0.5 \times C_f \times \rho \times U^2 \]

Where, \( U \) is the free stream velocity (usually taken outside of the boundary layer or at the inlet).

\[ U_r = \sqrt{\frac{\tau_w}{\rho}} \]

From the above equations the cell height \( y \) can be obtained as

\[ y = \frac{y + \mu}{\sqrt{\tau_w}} \]

The mesh properties chosen are as follows
Element size near the structure: 2.5mm (As calculated and varies for different models)

1. Relevance centre : Fine
2. Smoothing : low
3. Curvature angle : 18 degrees
4. Growth rate : 1.850
5. Inflation : smooth
6. Transition ratio : 0.272

C. Boundary Layer Velocity Profile:

A user defined function (UDF) is written in C-language, for obtaining the parabolic velocity profile. The wind speed chosen in the model is 44 m/s. [2]

Pre-defined macros are used to write UDF which allows accessing the field variables, material property and cell geometry data. The UDF written in C-program is linked with the solver by interpret method.

Power law equation is used to obtain the parabolic velocity profile,

\[ U_y = 84.7 \times \left( \frac{y}{400} \right)^{0.142} \]

Where \( U_y \) = velocity at height \( y \)
Height of the boundary layer is considered as 400m for wind speed of 44m/s.

IV. RESULTS AND DISCUSSIONS

The wind pressures on buildings of four plan shapes i.e. Rectangular, circular, elliptical and curved are predicted using the analytical models. 50, 65 and 80 floor buildings are considered and a comparative study is made and the results are presented and discussed in the present section.
Figure 1 shows pressure contours for various building shapes. From the figure, it can be noticed that the positive pressure on the windward face of the building is very high at the middle 1/3rd portion and maximum suction pressure is acting at the top 1/3rd portion of the leeward face. This suction pressure is due to sudden variation of pressure gradients at the top of the structure. The depth of suction pressure will vary depending on the shape of the structure. The difference in high negative pressure on the side faces of the structures causes the building to oscillate in normal direction of the building causing vortex shedding phenomenon.

From Figure 2, it can be observed that the average pressure is the least for the circular building (600 N/mm²) and maximum value of 1200 N/mm² is for the building curved in plan. It can also be noticed that the difference in average pressure is increasing as the height of the structure is increasing.

From the fig.3, it can be noticed that the circular building has the least negative pressure of -600N/mm² and elliptical buildings have the highest negative pressure of 1950N/mm². Figures 3 and 4 indicate that the percentage increase in pressure form 50 to 80 floors is the highest for curved shape buildings, about 26.9% whereas for circular and rectangle shapes, it is about 16.6%.

From Figure 4, it is seen that the pressure variation along the height of the structure for elliptical structure has the least pressure of 1400 N/mm² and for curved shape the maximum pressure is of 1700 N/mm² at the bottom level.
As the height increases, there is a constant increase of pressure for all the shapes but after a particular height there is a sudden drop in pressure and goes to suction pressure. This suction pressure is maximum for circular structure and the remaining three shaped buildings are subjected to same suction pressure and is less by 83.3% compared to circular shaped buildings.

This also causes undeveloped flow in wake region which causes the unsteadiness of the structure increasing building oscillations.

The development of flow in wake regions is shown in fig. 7 for various shapes considered. Circular shape being the best aerodynamic shape, has least average pressure of 600 N/mm² on the wind ward face and -600 N/mm² on the side face, but it should be noticed that the height of suction pressure is 35.3% more than other shapes, which plays a critical role in designing the tall structures.

Figures 5 and 6 present the variation of pressure with height of the buildings of 65 and 80 floors respectively. The negative pressure is increasing as the height of the building increases for all the considered shapes.

The percentage increase in negative pressure between 50 and 80 floors is 70.8% and between 50 and 65 floors is 38.46%. As the height of the building increases the suction pressure on top of the building is doubled.

V. CONCLUSIONS

The percentage difference of averaged pressure on wind ward face, when compared with circular structure is 50% more for curved, 33% more for elliptical and 16% more for rectangular structures.

The percentage difference of averaged pressure on side faces, when compared with circular structure is 70% more for curved, 68.42% more for elliptical and 8% more for rectangular buildings.

Circular shape being the best aerodynamic shape with least average pressure but is subjected to maximum negative pressure at the top.

The percentage pressure difference between the circle and remaining shapes being 83.3%.
The percentage increase in negative pressure between 50 and 80 floors is 70.8% and between 50 and 65 floors is 38.46%.

REFERENCES

[1] Mohd. Mohsin Khan and Dr. Amrit Kumar Roy. CFD simulation of wind effects on industrial RCC chimney. ISSN Print: 0976-6308 and ISSN Online: 0976-6316

