

“Drag Coefficient of Tall Building by CFD Method using ANSYS”

Sharma P K¹, Dr. Parekar S R²

¹ME Student, Department of Civil Engineering, AISSM'S, Pune, Maharashtra, India

²HOD, Department of Civil Engineering, AISSM'S, Pune, Maharashtra, India

Abstract - Wind is a more effective lateral load than Earthquake for a tall building owing to larger time period. It is suggested in literature, to analyze a building over 200 meters for wind loads. Generally, wind tunnels are used for analyzing only some important tall buildings because of higher modeling cost and unavailability of suitable wind tunnel. Computational Fluid Dynamic provides a worthy alternative to simulate the prevailing conditions and give fairly accurate results. Thus, a comparison is carried out using ANSYS 16.0 Fluent for checking the CFD analysis. Checking the effect of shape of building on wind analysis, drag coefficients are found out using CFD analysis for different shapes of building in plan and compared with the values given in IS 875-Part III 2015(table-25).

Key Words: CFD Analysis, Drag Coefficient, Ansys16.

1. INTRODUCTION:

Wind Engineering is studied universally by wind tunnel, full scale test, analytically and by using Computational Fluid Dynamics (CFD). In wind tunnel studies, scaled models of structures are subjected to scaled atmospheric wind in a controlled laboratory set-up. Then sensors installed on the model can measure the physical quantities of interest such as shear, moment, pressure etc. Most of the complex architectural and structural innovations are being confirmed through wind tunnel tests. With full-scale studies, actual buildings already built will be used to get instrumented and put into test in natural wind flow for several months in order to get decent measurements. Full-scale studies are good to improve our understanding of the science as well as the simulation in wind tunnel. But these types of studies are not practical in day-to-day life. With analytical studies, the structures are modeled in structural dynamic sense and the wind flow is modeled as stochastic time series and thereafter, the response of the structure is obtained by random vibration techniques. In Computational Fluid Dynamics (CFD) studies, like in analytical studies, the structures are modeled in structural dynamic sense. But the wind flow is modeled using basic fluid dynamic equations such as continuity, energy and momentum equations.

1.1 Drag Coefficients:

Air resistance, also known as drag, is a force that is caused by air, the force acts in the opposite direction to an object moving through the air. It is where air particles hit the front of the object slowing it down. The more surface area, the more air particles hit it and the greater the resistance.

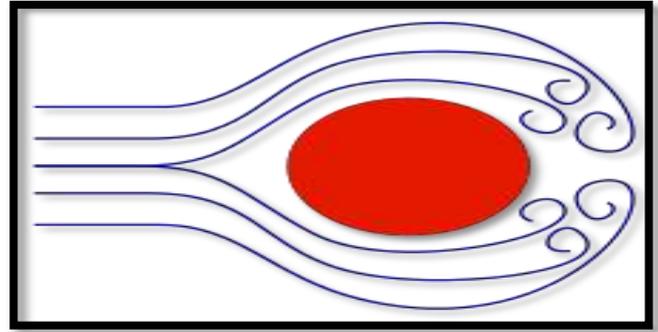


Fig - 1: Drag Coefficient

Given below is the drag equation;

$$F_d = C_d \frac{1}{2} \rho v^2 A$$

Where,

F_d = drag force (N).

C_d = drag coefficient.

ρ = density of fluid (1.2 kg/m³ for air at NTP).

v = flow velocity.

A = characteristic frontal area of the body.

It is observed from the above equation that,

- Drag Coefficient is directly proportional to the drag force on the structures.
- Drag coefficient is inversely proportional to the density of fluid i.e higher drag coefficient is in water than in air.
- Drag Coefficient is inversely proportional to the velocity of fluid.

According to IS 875 Part III, a non-dimensional coefficient such that the total wind force on a body is -

the product of the force coefficient, the dynamic -pressure of the incident design wind speed and the reference area over which the force is required. When the force is in the direction of the incident wind, the non-dimensional coefficient will be called as 'drag coefficient'. Referring of IS code 875 Part III, the value of force coefficients apply to a building or structure

as a whole, and when multiplied by the effective frontal area A_e , of the building or structure and by design wind pressure, P_d gives the total wind load on that particular building or structure.

$$F = C_f * A_e * P_d$$

Where,

F = is the force acting in a direction specified in the respective tables.

C_f = is the force coefficient for the building.

2. PROCEDURE OF ANALYSIS:

2.1 Domain of size:

Creation of domain of size as given in the paper by, M. Gomes (2005).

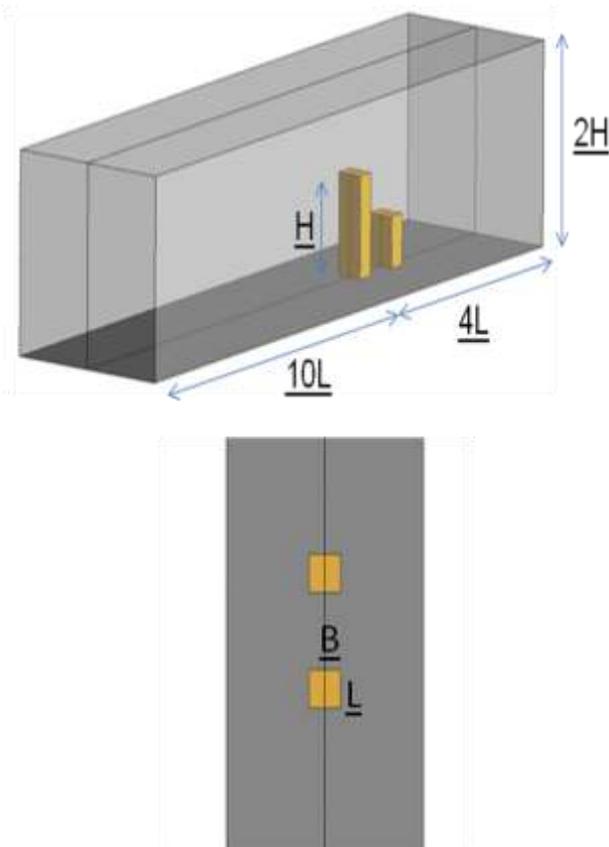


Fig - 2: Domain size

2.2 Mesh Generation:

CFD requires the subdivision of the domain into a smaller, non overlapping subdomains in order to solve the flow physics within the domain geometry that has been created; this results in the generation of a mesh of cells viz. elements. The accuracy of CFD solution is strongly influenced by the number of cells in the mesh within the domain.

2.3 Boundary Condition:

- The inlet boundary condition permits flow to enter the solution domain in one direction only. It can be a velocity inlet, pressure inlet or mass flow inlet. $V_x = V$ and $V_y = V_z = 0$.
- The outlet boundary condition permits flow to exit the solution domain. It also can be a velocity inlet, pressure inlet or mass flow inlet.
- Side walls, Top and Ground boundary condition For stationary walls, the default consideration is to assume that the no-slip condition implies, which simply means that the velocities are taken to be zero at the solid boundaries.

2.4 Setup:

Turbulence models are used to determine the effect the fluctuation on mean flow. Commonly used is two equation turbulent model.

Standard *K-epsilon model* by Launder and Spalding (1974) is used in ANSYS; where,

K is the turbulence kinetic energy and is defined as the variance of fluctuations in velocity.

Epsilon (ϵ) is the turbulence energy dissipation i.e. the rate at which the velocity fluctuations dissipate.

Finite Volume Method (**FVM**). FVM is a common approach used in CFD codes. The FVM approach requires interpolation and integration for methods of order higher than second are more difficult to develop in 3D.

2.5 Post Processing:

Finally, the last sets of results were selected. Then fluid velocity were checked from the nodal solutions. Further, the desired planes of the models having pressure coefficients were selected which were then compared with the results from the IS 875 PART-III.

3. RESULT:

Drag Coefficients for different shapes:

IS 875 Part III have given values of drag coefficients for different shapes of buildings in plan. Similar shapes of buildings of height to width ratio 10, are modeled in ANSYS 16.0 and analyzed using CFD and drag coefficients are obtained and compared.

Table -1: Comparisons of drag Coefficient.

PLAN SHAPE	IS875-PART3	CFD Analysis
Circle	0.5	0.430
Diamond	1.2	1.242
Triangle	1.0	0.942
Square with round edge	0.6	0.427
Hexagon	1.4	1.011
Octagonal	1.2	0.928

Following are the figures of drag coefficient obtained at different shape of building.

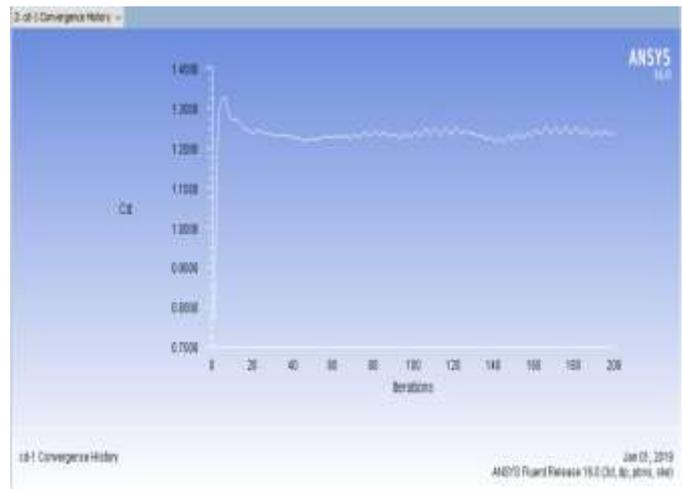


Fig - 5: Drag coefficient for Diamond shape

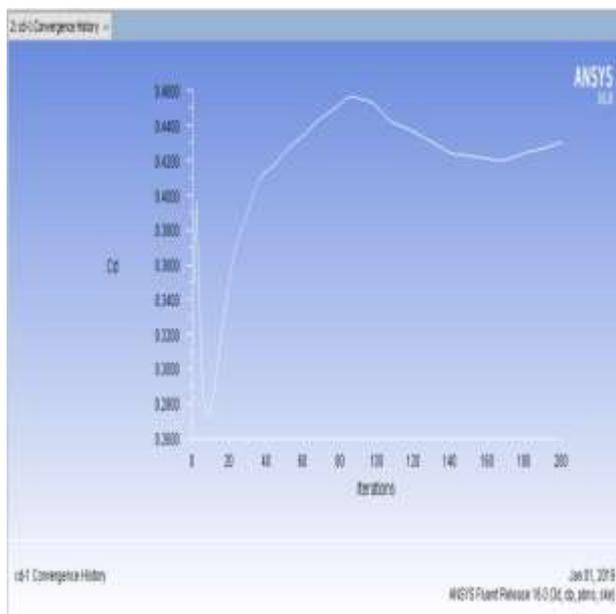


Fig - 3: Drag coefficient for circular shape

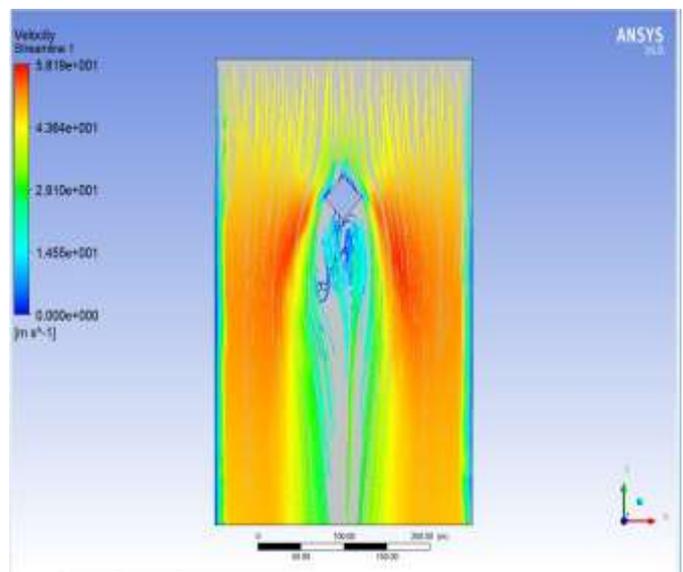


Fig - 6: Velocity Streamlines for Diamond shape

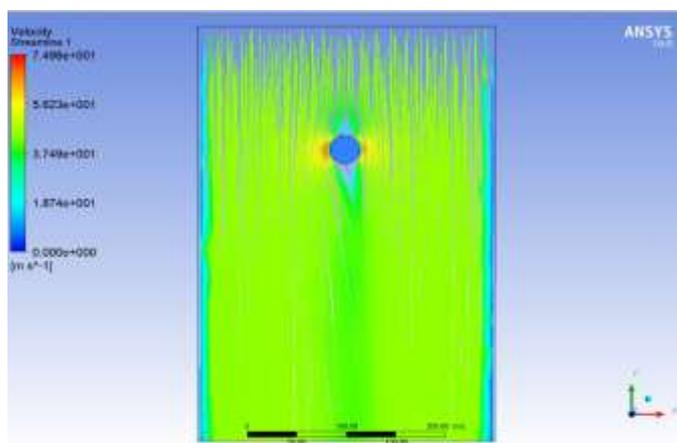


Fig - 4: Velocity Streamlines for circular shape

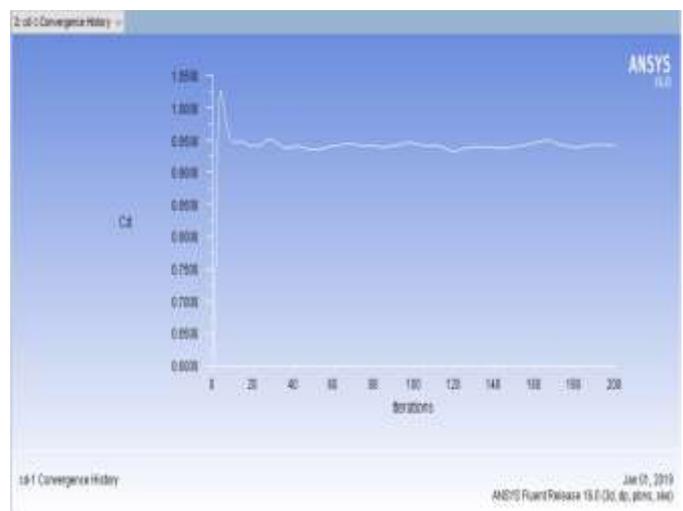


Fig - 7: Drag coefficient for Triangle shape

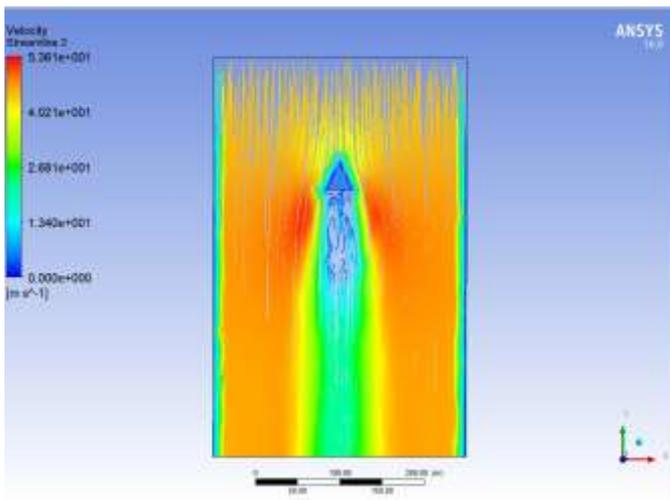


Fig - 8: Velocity Streamlines for Triangle shape

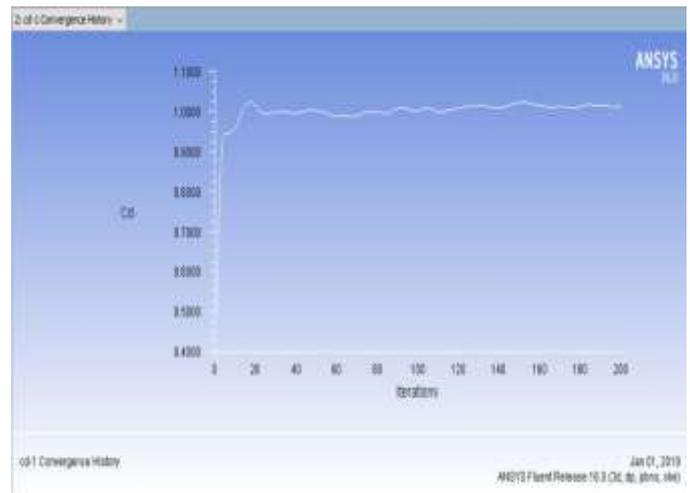


Fig - 11: Drag coefficient for Hexagonal shape

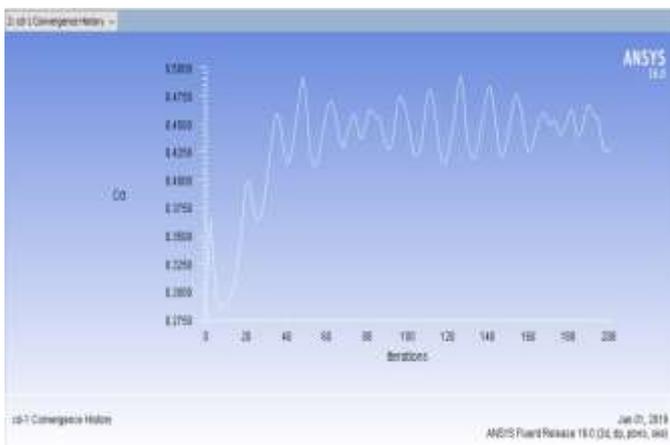


Fig - 9: Drag coefficient for Square with Round edge shape

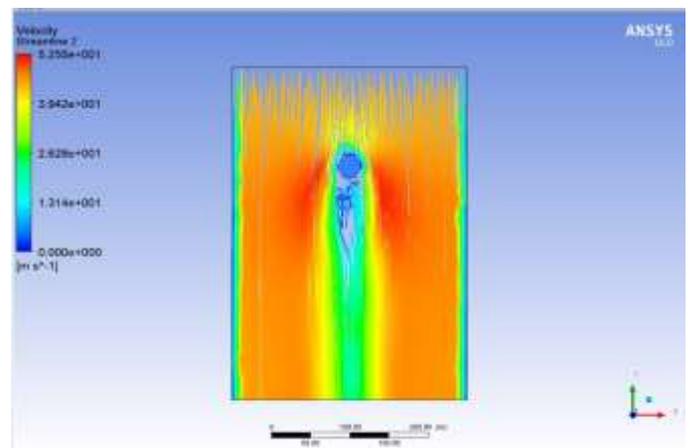


Fig - 12: Velocity Streamlines for Hexagonal shape

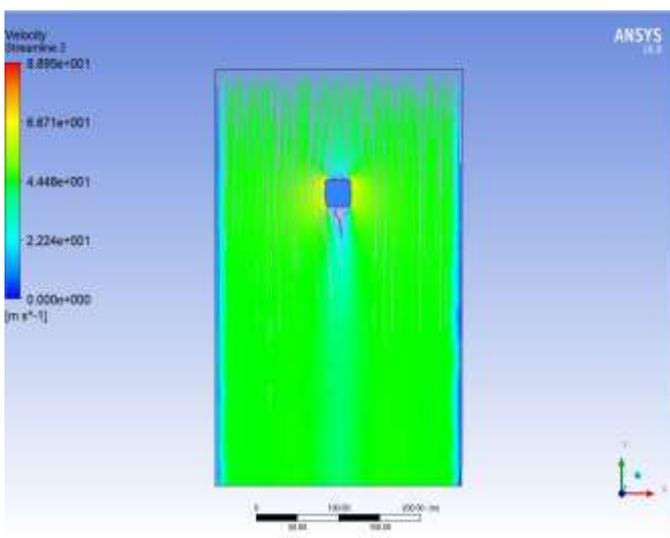


Fig - 10: Velocity Streamlines for Square With Round edge shape

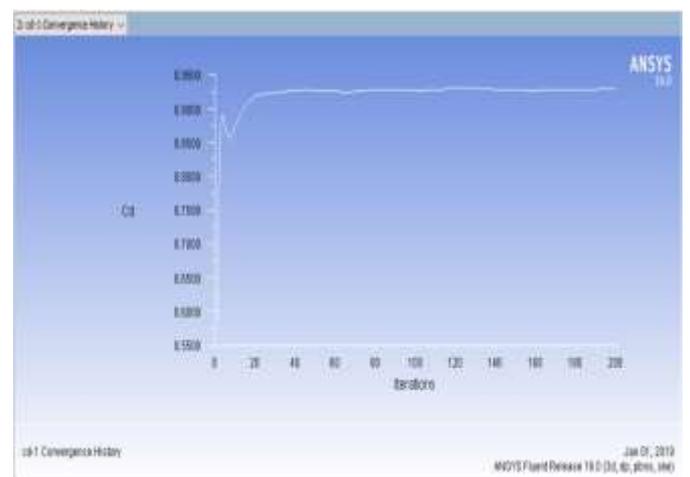


Fig - 13: Drag coefficient for Octagonal shape.

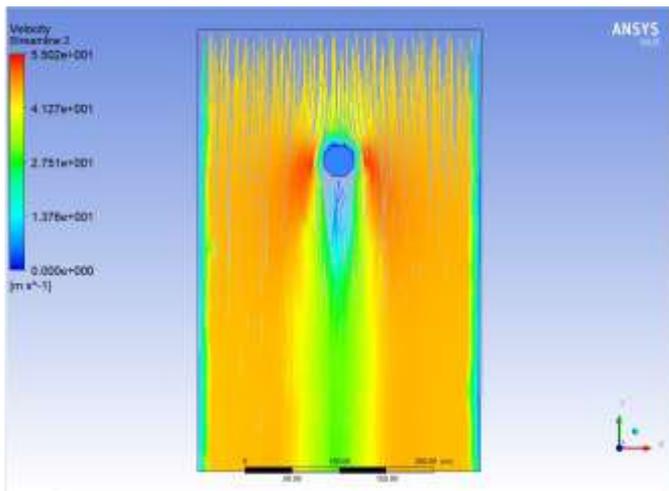


Fig - 14: Velocity Streamlines for Octagonal shape.

4. CONCLUSIONS:

Validation of CFD analysis considering drag coefficients given in IS 875 part III of different shapes of building in plan is carried out. The CFD analysis varied within a range of 05-08 %.

REFERENCES:

- [1] P. A. Irwin."Wind engineering challenges of the new generation of super-tall buildings". (2009) J. Wind Eng.Ind.Aerodyn.97, 328-334.
- [2] D. K. Kwon, A. Kareem, "Comparative study of major international wind codes and standards for wind effects on tall buildings". Engineering Structures 51 (2013) 23-35.
- [3] C. Anoop et al. "Suitability of IS 875 provisions for assessing wind loads on high rise buildings: scope and limitations". VI National Conference on Wind Engineering 2012, Dec. 14-15
- [4] E.Vafaeihosseini, A. Sagheb, P. K. Ramancharla. "Computational Fluid Dynamics Approach for Wind Analysis of High rise Buildings". Structural Engineering World Congress Report No: IIIT/TR/2011/-1.
- [5] J. K. Sevalia S.A. Vasanwala , "Comparative Study of wind flow effect on tall building using Computational Fluid Dynamics".8th Biennial Conference on Structural Engineering Convention (SEC-2012).
- [6] "Indian Standard Code 875 -Part III 2015".