

Analysis of Tall Irregular Plan Shaped Building against Wind Loading

Rohan D. Pawar¹, P. M. Kulkarni²

P.G. Student, Department of Civil Engineering, Trinity College of Engineering & Research, Pune, India¹

Assistant Professor, Department of Civil Engineering, Trinity College of Engineering & Research, Pune, India²

ABSTRACT: Any tall building can oscillate in both the direction of along wind and across wind caused by the flow of wind. Modern tall buildings designed to satisfy lateral drift requirements, still may oscillate excessively during wind storm. These oscillations can cause some threats to the tall buildings with more and more height becomes more vulnerable to oscillate at high speed winds. Sometimes these oscillations may even cause discomfort to the occupants even if it is not in a threatening position for the structural damage. So an accurate assessment of building motion is an essential prerequisite for serviceability. There are few approaches to find out response of tall building to the wind loads. An analytical approach given by Gust Factor Method and mentioned in the IS: 875(part-3)-1987 is used which only applicable to a regular shape building but for an irregular shaped buildings and accurate response and behavior under high wind speed is provided from a prototype measurement in wind tunnel. For overcoming the shortcoming of analytical formulation, there is need to study the detailed analysis of effect wind load on tall structures. Main objective of project is to study effect of wind load on tall building of irregular plan shape using experimental method (wind tunnel test) and software analysis (computational fluid dynamics) and comparing the results obtained from these methods.

KEYWORDS: Irregular plan shape, Wind Load, Wind Analysis, CFD Analysis, Drag Pressure Coefficient.

I. INTRODUCTION

Buildings are defined as structure utilized by the people as shelter for living, working or storage. As due growth in population as well as urbanization and scarcity of land, we found need of much faster growth in residential and industrial structures in vertical direction. Wind is the flow of gases on a large scale. On Earth, wind consists of the bulk movement of air. Wind is caused by differences in pressure. When a difference in pressure exists, the air is accelerated from higher to lower pressure. In the field of structural engineering it includes strong winds, which may cause discomfort, as well as extreme winds, such as in a tornado, hurricane or heavy storm, which may cause widespread destruction. Tall structures are wind sensitive and prone to large wind-induced vibrations. Even small vibrations can induce fear within humans, as humans are very sensitive for such situations. So even wind does not affect strength of building elements, we must take care of vibrations within structures also. Extreme wind or tornado can cause destructions of structure which cause loss of human life, money, businesses and more. Nowadays due to high population and scarcity of land, need of tall building increase, also irregular architectural appearances increase the necessity to study wind behavior on the structure. Because we cannot predict wind behavior for every plan with analytical process, as wind behavior changes with plan atmospheric conditions. So we should analyze the building before starting their construction for various possible conditions of wind flow, so can design safe structure which withstand against heavy wind forces. The assessment of wind loads on such tall buildings becomes necessary for their design. For the evolution of wind load, IS875 (Part 3)-1987 provides pressure/force coefficients, which are based on wind tunnel studies on building model under uniform flow conditions. Further these coefficients are available for only selected geometries and for selected angle of wind incident, viz 0° and 90°, i.e., wind normal to any one face of the building. So we need to study wind pressure action on various different shape. There are mainly three methods to evaluate pressure coefficient for a structure 1. Analytical method, 2. Wind tunnel method, 3. Computational fluid dynamics method. In analytical method we can find values of pressure coefficient of selected shape i.e. square, rectangular, triangular etc. only. Direction of wind also restricted in this method. So we cannot use this method for evaluating values of pressure coefficient of complex shaped structures. In wind tunnel method, using scaled model of structure and creating atmospheric conditions within wind tunnel, we can analyze the actual structure against wind loading. But due to human or instrumental errors, results can be changed. This method is also expensive and not available easily. But nowadays numerical simulation by means of CFD (Computational Fluid Dynamics) is becoming a promising and powerful tool for predicting the behavior of structures in practical engineering cases.

II. LITERATURE SURVEY

1 Vermaet. al.(2013) carried out experimental study on rigid model of a square plan tall building made of Perspex sheet in a closed circuit wind tunnel under boundary layer flow. Wind pressure coefficient were calculated from wind pressure values measured at many pressure points on all 4-wall surfaces of the model. The results of experimental study were presented in the form of pressure contours.

2 Mufthaahmed et. al. (2015) carried out experimental study on triangular shaped tall building under stand-alone condition, pressure measurement were made on rigid model of tall building made up of Perspex sheet at many pressure points. The model was tested in an open boundary layer wind tunnel. The experimental study of pressure measurements were reported in form of contours of mean wind pressure coefficient for three wind incident angles. Cross sectional variation of mean pressure coefficient were also given at two sections for the model for six wind incident angles.

3 Biswarup Bhattacharyya et. al. (2014) conducted a comparative study between rectangular and 'E' plan shaped tall building under wind excitation in boundary layer wind flow condition. The outer plan dimension and height were kept same for both of the models but, plan area was different in nature. Aspect ratios of the both models were kept 2. Pressure distributions on different faces of both models were studied by Computational Fluid Dynamics (CFD) technique using ANSYS CFX software. k- ϵ turbulence model was used for creating turbulence flow. Study was conducted under different wind angles of 0°, 30°, 45° and 90°. Wind flow pattern under different wind angles were studied. Also pressure distributions and mean pressure on different faces of both the models were studied for each wind angle separately. Almost similar pressure distribution noticed on similar faces of both models whereas pressure distributions on dissimilar faces were too much different. Maximum positive pressure occurred on wind ward face of rectangular plan model at 0° wind angle but, maximum negative pressure occurred on side face of 'E' plan shaped model at 90° wind angle. Almost zero mean pressure occurred on similar faces of both model but, huge variation of magnitude of pressure from negative to positive is noticed on these faces which are also a critical condition under design criteria.

4 Ritu Raj et. al. (2013) carried out an experimental study on rigid model of tall building with varying cross-sectional shapes, but with equal floor area, in an open circuit boundary layer wind tunnel to measure wind forces action on them. It was observed that the base shear, base moment and twisting moments developed due to wind loads are not only influenced by wind directions, but also highly affected by cross- sectional shapes.

5 SouvikChakraborty et. al. (2013) carried out a comprehensive study of pressure developed on different faces of a '+' plan shape tall building. Computational Fluid Dynamics (CFD) technique was used to evaluate the pressure on different faces of the model for wind incidence angle of 0° and 45°. The analysis was carried out using ANSYS FLUENT with k- ϵ viscosity model and the results obtained were compared with the results of wind tunnel test. A general good agreement among the results has been observed for both the wind angles. Moreover, a square plan shape model was analyzed for normal incident angle to verify the reliability of the code. The faces of '+' plan shape model has demonstrated a different pressure distribution as compared to the square shape model with major difference observed in case of side faces. Flow pattern around the model was also studied in order to have an understanding on different phenomenon occurring around the building. The numerical analysis along with the experimental data can provide useful data for calculating wind load on such structures.

6 YadavVarshaet. al. (2016) In this paper, CFD simulation is carried out for a particular arrangement of group of tall buildings. Pressure coefficients on the same have been studied using Computational Fluid Dynamics (CFD) technique. Buildings of different heights are taken into consideration. The effect of different wind incidence angles on the pressure coefficients on all the faces of Principal Building has been investigated. Also, the pressure coefficient contours on all the faces of all buildings for wind incidence angles of 0°, 30°, 60° and 90° are plotted. ANSYS CFX is used for carrying out the CFD simulation.

7 ZhiwenLuo et. al.(2008) The effect of the surrounding lower buildings on the wind pressure distribution on a high-rise building is investigated by computational fluid dynamics (CFD). When B/H=0.1, it is found that the wind pressure on the windward side was reduced especially on the lower part, but for different layers of surrounding buildings, there was no great difference, which agrees with our previous wind tunnel experiment data. Then we changed the aspect ratio from 0.1 to 2, to represent different airflow regimes: skimming flow (SF), and wake interference (WI). It shows that the average C_p increases when B/H increases. For different air flow regimes, it is found that insignificant difference exists

when the number of the building layers is more than 2. From the engineering point of view, it is sufficient to only include the first layer for natural ventilation design by using CFD simulation or wind tunnel experiment.

8 H. Sarathkumaret.al. (2017) carried out experiment to evaluate wind pressure distributions on all four faces of a rectangular tall building with 1:1.5:7 ratio. The model is made up of acrylic sheet with a geometric scale of 1:300 with plan dimensions of 10cm × 15cm and height of 70cm. The model is tested using a Boundary Layer Wind Tunnel twelve angles (0°, 5°, 10°, 15°, 25°, 33.5°, 45°, 56.5°, 60°, 75°, 87.5° and 90°) of wind incidence under uniform flow condition. Mean and standard deviation of pressure coefficients, drag and lift coefficient along wind direction and perpendicular to wind direction, mean moment coefficient are calculated pressure measurement on the model.

9 Jigar K. Sevaliaet. al. (2012) carried out an attempt has been made to study the effect of different geometric plan configurations like Square, Circular, Hexagon, and Octagon of Tall Building having same plan area on Force Coefficient. To study the wind effect, 3-D Wind Flow condition around tall building has been developed using Computational Fluid Dynamics (CFD) Code namely Fluent / Gambit and then numerical computation has been executed to evaluate pressure coefficient and the wake region around the building. In this study, it was found that Wind Pressure Coefficient is maximum in case of square plan shape and it is minimum in case of circular plan shape of tall building. The octagonal plan shape of tall building with sharp windward edge is more effective in reducing wind pressure coefficient than hexagonal plan shape of tall building with sharp windward edge.

10 RavinderAhlawatet. al. (2015) carried out experimental study on the models of 'Y' plan shape tall building in an open circuit wind tunnel to investigate the wind loads generated on the building in isolated as well as interference condition. The base shear (Fx), overturning moment (My) and torsional moment (Mz) acting on the instrumented model were measured. For the isolated condition, the measurements were made for many wind incidence angles. For studying the interference effects, two similar building models were placed in side-by-side configuration and tandem configuration and the spacing between these models were varied. It is observed that the wind incidence angle greatly affects the wind induced loads on the 'Y' plan shaped building. Depending on the position of the interfering building, the inference effects may either be beneficial or may have an adverse effect.

11 Dr. Hassan Hemidaet. al. (2014) conduct wind tunnel experiments to investigate the flow around building for the possibility of harvesting wind energy from the above roof flow in built environment. The program consisted of measuring the velocity in the above roof flow and the surface pressure on a number of building shapes and configurations. These include: a high-rise building with a flat roof, a high-rise building with a tilted roof, a low-rise building with a tilted roof, an industrial building with a tilted roof. All building shapes were investigated in isolations considering different angles of flow attack. In addition to the above, the interference effects of the building configurations were also investigated. The velocities above the roof of building shapes at different heights were measured. Also the surface pressures on the roof and on one or two rings, depending on the building shape, at certain heights from the ground were measured. They also compared CFD data and wind tunnel data for every case.

12 Al ZoubiFeraset. al. (2010) carried out a wind tunnel test to measure cladding loads for a high-rise building of 295 m in height, which would be located in the business center of Chongqing Municipality, P. R. China. The rigid model was used to determine fluctuating local pressures on the exterior surfaces of the building. The wind tunnel test results show the critical zone of wind pressures on building surfaces in both standalone and interference conditions. The computational fluid dynamics (CFD) was conducted by using the FLUENT Code to compare with the wind tunnel test results, and the steady three-dimensional turbulent flow with Realizable k-ε as a turbulence model was used. The CFD results are agree with the wind tunnel test results in regards to distributions of wind pressures over a high-rise building's surfaces.

III. METHODOLOGY

As we know, in IS 456:2015 part 3, there is no any provision about how drag coefficient changes as shapes changes irregularly. The code only provide drag coefficient for some regular shape like square, rectangle, sphere etc. So, there is a need to find out pressure distribution and drag coefficient for different shape. The present study has been undertaken to bridge a few of the many existing gaps mentioned above, in field of wind induced response of tall building. In this study, mainly two irregular plan shapes namely (1)swastika with hexagon plan shape (2)swastika with square plan shape are considered on wind analysis.

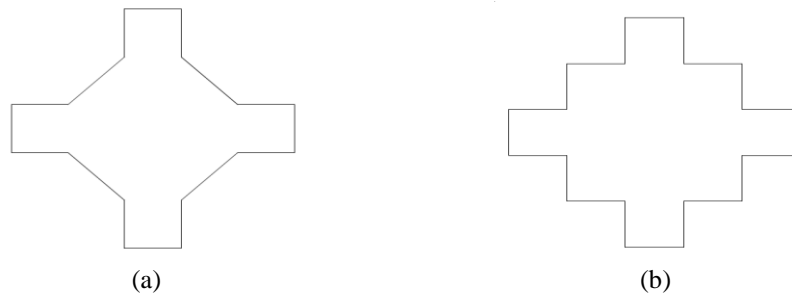


Fig.1: -(a) Swastika with hexagon plan shape (b) Swastika with square plan shape

In this study, mainly two methods are used to conduct analysis on tall irregular shaped building.

1. Software method:- In this method, analysis is carried out using Ansys Fluent 19.0 software. Computational fluid dynamics (CFD) analysis is carried out on these shapes. In this method, model of this shapes with surrounding boundary conditions are tested for various degree of wind incident angles i.e. $0^{\circ}, 15^{\circ}, 30^{\circ}, 45^{\circ}, 60^{\circ}, 75^{\circ}$. Performing analysis for several iterations for each wind incident angle, it gives values of pressure coefficient, pressure contours, velocity contours, drag coefficient.

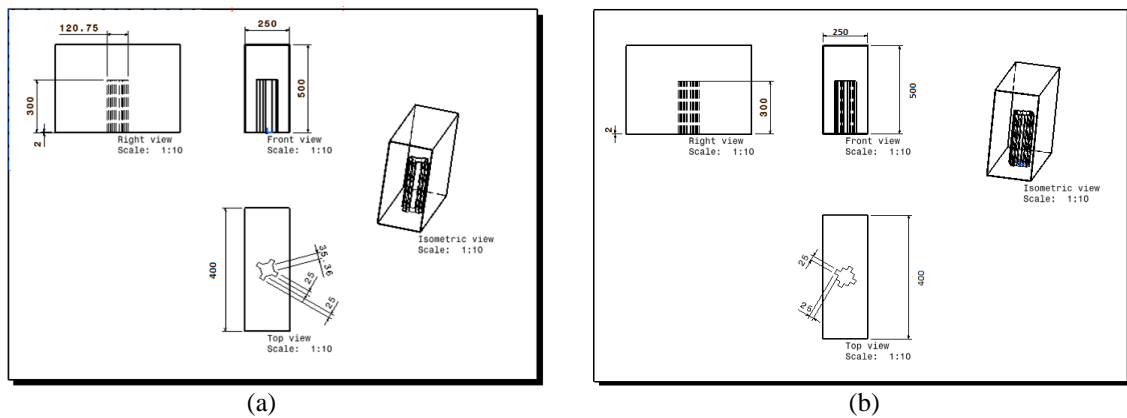


Fig.2: -(a) Model of swastika with hexagon plan shape (b) Model of swastika with square plan shape

2. Experimental method:- This method mainly used validate result of drag coefficient coming from software analysis. In this method, we manufactured own duct and model of scale 1:400. Normal wind tunnel is not sufficient for this reduced scale. In this method, we get values of force coming on structure at various points. Force sensor senses force impact on it, then Arduino Uno microcontroller convert this impact in single digit digital solution on computer screen. This force values are upper or lower non decimal number.

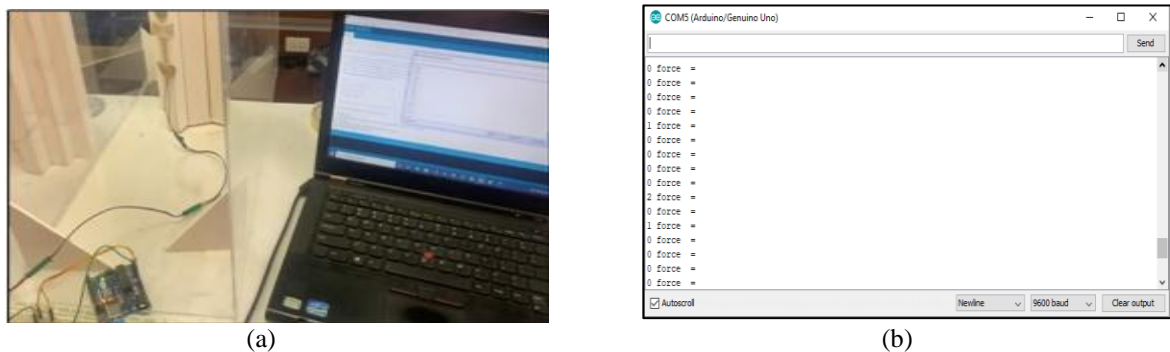


Fig.3: -(a) Performance set up for experiment (b) Result screen of experiment



IV. RESULTS

There are mainly two type of result of this study i.e. software results and comparative results. In this study we get the solution of drag coefficient from software analysis. 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 is called force coefficient. When force is in the direction of incident wind, the non-dimensional coefficient will be called as drag coefficient

	0 ⁰	15 ⁰	30 ⁰	45 ⁰	60 ⁰	75 ⁰
Swastika with hexagon plan	2.67	2.3	2.12	1.96	2.15	2.37
Swastika with square plan	2.48	2.44	2.16	1.99	2.13	2.32

Table 1: - Drag coefficient obtained from software analysis

Experimetal analysis gives us values range of forces which made impact of surface of the building. Force ranges are same for both shapes for different wind incident angles.

	0 ⁰	15 ⁰	30 ⁰	45 ⁰	60 ⁰	75 ⁰
Swastika with hexagon plan/Swastika with square plan	1N-2N	1N-2N	0N-1N	0N-1N	0N-1N	1N-2N

Table 1: - Drag forces results from experiment

V. CONCLUSION

Tall buildings are slender flexible structures and having large exposed area against wind, so need to be investigate against wind loading or wind induced oscillations along and across the direction of wind. Here are some conclusions which I understand from this study,

1. It is observed that, values of pressure coefficient given in the IS:875 part-3 are limited for certain shapes. Nowadays necessity or requirement of different unsymmetrical plan are increased, IS code recommendations are not suitable for design of such kind building.
2. Cross sectional area of structure can influence wind load coming on structure in large extent. Even area, material, surrounding environment are same, different elevation give different results.
3. Due to sloping part in structure, pressure applied on surface and air flow shows noticeable change in their behaviour.
4. As complexity of structure increases, values of drag coefficient changes. Greater the resistance to the wind, larger the drag force and lesser the resistance to wind, lesser the drag force coefficient.
5. As the angle of incidence of wind on structure changes, behaviour of pressure changes because edges and surface of structure divert the wind.
6. CFD can be adopted in wind analysis as it is able to model actual surrounding in real scale as compared to reduced scale in wind tunnel experiment. CFD can give accurate result as compared to wind tunnel experiment. Manual errors and instrumental errors can affect on result from wind tunnel methods.
7. CFD analysis results are close to result from wind tunnel experiments. So it will be cost effective and minimum error solution for wind analysis of structure.
8. Now days necessity of tall buildings are increased due factors like economy, population and so on. Tall buildings have greater impact of wind loads on it. Also there are many failure of structures due to oscillations induced in structure due wind. It shows that there is lack of interest in wind analysis. As there are lots of factors which affect result of wind analysis. While designing tall building, we should consider dynamic nature wind as well.

REFERENCES

- [1] S.K. Verma¹, A.K. Ahuja^{2*} and A.D. Pandey³- “Effects of wind incidence angle on wind pressure distribution on square plan tall buildings” J. Acad. Indus. Res. Vol. 1(12) May 2013
- [2] Muftha Ahmed Abdusemed¹, Ashok Kumar Ahuja²- “Wind Pressure Distribution on Triangular Shape Tall Buildings” IJIRSET, Vol. 4, Issue 8, August 2015
- [3] Biswarup Bhattacharyya, Sujitkumar Dalui- “Comparative study between regular and irregular plan shaped tall building under wind excitation by numerical technique” National Conference on Innovations in Design & Construction of Industrial Structures April 3-5, 2014
- [4] Ritu Raj, Ashok Kumar Ahuja-“ Wind Loads on Cross Shape Tall Buildings” Journal of Academia and Industrial Research (JAIR) Volume 2, Issue 2 July 2013
- [5] Souvik Chakraborty, Sujitkumar Dalui-“ Experimental and Numerical Study of Surface Pressure on '+' Plan Shape Tall Building” Research Gate
- [6] Yadav Varsha, Jignesh Arvindbhai Amin- “CFD Simulation on a Group of Tall Buildings” GRD Journals | Global Research and Development Journal for Engineering | Recent Advances in Civil Engineering for Global Sustainability | March 2016 e-ISSN: 2455-5703
- [7] Zhiwen Luo^{1,*}, Yuguo Li¹, Marcus Rosler² and Joachim Seifert²-“ SURROUNDING BUILDINGS AND WIND PRESSURE DISTRIBUTION ON A HIGH-RISE BUILDING” 9th International Symposium on Building and Urban Environmental Engineering, 7/10 July, Hong Kong
- [8] H. Sarathkumar, P. Vijaya Bhaskar Reddy-“Pressure Measurement Studies on a 1:1.5:7 Rectangular High Rise Building Model under Uniform Flow” IOP Conf. Series: Materials Science and Engineering 225 (2017) 012269
- [9] Jigar K. Sevalia^a, Dr. Atul K. Desai^b and Dr. S. A. Vasanwala^c-“ EFFECT OF GEOMETRIC PLAN CONFIGURATION OF TALL BUILDING ON WIND FORCE COEFFICIENT USING CFD” International Journal of Advanced Engineering Research and Studies
- [10] Ravinder Ahlawat and Ashok K. Ahuja-“Wind loads on ‘Y’ plan shape tall building” International Journal of Engineering and Applied Sciences (IJEAS) ISSN: 2394-3661, Volume 2, Issue-4, April 2015
- [11] Hassan Hemida², Anina Šarkić¹, Rüdiger Höffer³-“Wind Tunnel Test –Air Flow Around Building” final report of the Short Term Scientific Mission as part of the COST ACTION TU1304.
- [12] AL ZOUBI Feras †, LI Zheng-liang, WEI Qi-ke, SUN Yi-“ Wind tunnel test and numerical simulation of wind pressure on a high-rise building” Journal of Chongqing University (English Edition) [ISSN 1671-8224] Vol. 9 No. 1 March 2010
- [13] Indian Standards Codes, BIS (2015), Indian Standard Code of Practice for Design Loads (Other than Earthquake) for Buildings and Structures Part 3- Wind Loads [IS 875 (part3): 2015], Bureau of Indian Standard, India