Comparative study of experimental and numerical analysis of fluid parameter around tall buildings

Sreelakshmi K Nair ¹ and Mohammed Thowsif ²

¹M. Tech. Graduate, ²Assistant Professor, Department of Civil Engineering, TKM College of Engineering, Kollam, Kerala- 691005, India.

Abstract. The rapid growth of the urban population and the consequent pressure of limited space have influenced city residential developments, which ultimately resulted in the sudden sprouting of tall and vertical structures. The structural system of present day has become less stiff and lighter than earlier, hence are usually more sensitive to the effect of lateral loads acting on it, especially wind loads. In order to ensure serviceability of the structure, the analysis and design must be carried out efficiently. From being termed a simple static drag force, wind force gained new dimension taking into consideration the dynamic aspects, thus demanding more attention in structural analysis and design. This paper deals with the comparative study of fluid parameters around tall structures when subjected to the traditional experimental analysis by Wind Tunnel test and the most modern approach of numerical simulation using Computational Fluid Dynamics (CFD). The comparison of result obtained via both of the above-mentioned approach and the further extend of possibilities in adopting CFD is also discussed.

Keywords: Wind analysis, wind tunnel, CFD, tall structures.

1. Introduction
The past century witnessed a rapid growth in urban population. The lack of space to accommodate the population demanded the construction of vertical structures that are tall enough. But, till date there is no perfect categorization for the term tall buildings in terms of height or number of floors. Tallness is a relative matter dependent on the consequent perception and circumstances. Though there is no special categorization, from Structural Engineer’s point of view, a tall building can be generally defined as the one because of its height, is affected by lateral forces due to wind and earthquake to an extent that they play a critical role in structural design. Due to increased height and use of lightweight building materials, lateral loads have become crucial in the design of high-rise structures than gravity loads. Hence it has become mandatory to incorporate the effect of wind forces while analysis to ensure the serviceability and stability of the tall structures. Initially only IS codes were available for computation, which was applicable only to predefined shapes. Further research has developed experimental methods like Wind tunnel test for analysis of pressure around buildings, subjected to a large set of computation involving the topography of existing buildings and modelling of the proposed building. But there exists great complication in making even slight changes in experimental model. To overcome such difficulties the concept of numerical simulation using CFD can be done efficiently.

Blocken[1] describes that as a result of more than 50 years’ investigation, air flow around buildings have proven to be more susceptible and complicated, generally showing large variability with respect to wind features, building configuration and location characteristics. Kwon et al.[2] has conducted a
long-term meteorological monitoring and wind tunnel tests, with the aid of regression methods, eventually developing wind profiles using power law/exponential law and logarithmic law. These profiles prove vital in selecting turbulent models, which is the key component in governing the reliability of numerical simulation using Computational Fluid Mechanics (CFD) techniques. Many researchers have inspected the effectiveness of various turbulent models in CFD such as Reynolds Averaged Navier Stokes (RANS), Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS). Since the computational cost is comparatively high for DNS, most studies were limited to RANS and LES. Initially Yamada et al. [3] studied the 2D air flow over a square surface mounted obstacle, followed by calculation of 3D flow around structures by Hirt et al.[4]. Simulation using CFD for 3D buildings began with isolated cubical shaped structures to quantify the velocity and pressure distribution with the studies of Murakami et al. [5] and Baskaran et al.[6]. According to Blocken[1], with the progress of research, a firm foundation was laid for CFD analysis, emphasizing on grid resolution study, setting boundary conditions and convergence.

2. Experimental analysis

In the experimental analysis, two parameters are compared to check the comparability of both ways of testing, namely the pressure on the face of the building (tapped using Wind Tunnel set up) and the flow pattern (captured using a recirculating water channel visualization). The model considered for study includes the surrounding topography. The study is done on the principal building surrounded by three interfering buildings in the direction of prevailing wind direction. The dimensions of building are described in table 1.

| Buildings considered                  | Dimensions       |
|---------------------------------------|------------------|
| Principal Building (BxDxH)            | 30x30x225 m³     |
| Interfering Building model-1(b1xd1xh1)| 15x10x112.5 m³   |
| Interfering Building model-2(b2xd2xh2)| 10x15x112.5 m³   |
| Interfering Building model-3(b3xd3xh3)| 20x20x168.75 m³ |

The plan of the dimensions of buildings discussed above is shown in fig. 1. The 30 m x 30 m building is the principal tall building. Three buildings act as interfering building with dimensions mentioned in table 1. The buildings are so chosen that the interfering building is three-quarter the height of principal building, plan area being the same and the other two interfering buildings are half the height of principal building and are so oriented that the longer face is exposed to wind in one case and shorter face in another case.

Figure 1. The plan of principal and interfering buildings considered for study
2.1. Wind tunnel test

Wind Tunnel testing is an experimental tool that has been used to examine the aerodynamic effects of wind on a solid object since the end of the 19th century. According to Baals et al., [7] it was during 1740-1750, an English mathematician, Benjamin Robins first adopted the idea of moving the air past an object that is stationary for the purpose of simulating its movement in the air. Later on, in 1804, Sir George Cayley is said to have made improvements in the design of his whirling arm system and built a small glider. But the aircraft models on the end of a whirling arm were prone to very high turbulence due to the centrifugal forces. Hence, reliable relative velocity between the model and air could not be determined. Further, it was hard to set up instruments to measure small forces exerted on models while they were in motion at high speeds. This brings the first enclosed Wind Tunnel invented and operated by Francis H. Wenham, a Member of the Aeronautical Society of Great Britain in 1871. Since then, the Wind Tunnel testing techniques have been widely used in Aeronautical Engineering. Soon it has also revolutionized the Automobile Industry and had become inevitable in Civil Engineering too.

![Figure 2. Wind Tunnel test set up used for the study](image)

![Figure 3. Manometer for measuring pressure and slot for inserting model](image)

![Figure 4. a) 3D rendered model in SOLIDWORKS b) The cut section of principal building depicting holes for pressure tubes](image)
The models for experimental testing must be precise in dimension (i.e., prototype of actual case reduced geometrically to scale of 1:600) and so the most acceptable method of model making is 3D printing of the models. The sketching of the models is done using SOLIDWORKS 16, with the holes imprinted to ensure the passage of 2 mm outer diameter calibrated pressure tubes. The model is fixed to a wooden board. In order to tap the pressure, pressure tubes are inserted into the holes left while modelling the building. The model is printed using 3D printing. The file from SolidWorks is processed for 3D printing. The printing is done using Ultimaker 3D printer. The printed model is shown in fig. 5.

![Models fixed on wooden board and placed inside the Wind Tunnel](image)

**Figure 5.** Models fixed on wooden board and placed inside the Wind Tunnel a) side view b) front view (principal building with pressure points numbered)

The model is tested for analysis in Wind Tunnel test set up to monitor the variation in pressure on the principal building. The Wind Tunnel can also be used for finding out the flow pattern using smoke visualization. But water channel visualization can give a better flow pattern and is preferred in this study.

2.2. **Water channel visualization**

The flow visualization experiments were carried out in a recirculating water channel having dimensions 1.5m x 2.5m. Fine Aluminium powder particles (size: 80microns) are sprinkled over the water surface and the water bed was illuminated to visualize the flow over the object clearly. Two paddle wheels rotate in opposite directions powered by a motor and create a flow in the central test section of the channel where the models are kept and studied. Flow velocity is measured by introducing a sufficiently small, lightweight particle in the center of the test section and measuring the time taken by the same to cover a distance of 60 cm. Flow velocity is calibrated against the rotational rpm of the paddle wheel so as to enable setting of a desired flow velocity. Flow visualization pictures were captured using a video camera (SONY DSR-PD150) and was recorded at 25 frames/sec.

![Flow visualization water channel](image)

**Figure 6.** Flow visualization water channel
In case of visualization, cross-section is primary. The velocity is determined by averaging the values after ten trials. Correspondingly the Reynolds Number is also determined. The flow pattern is monitored and yields a clearer pattern than smoke visualization. The experimental setup of water channel visualization, the principal building with adjacent topography is shown in fig. 6. For oscillating cylinder cases, a frame-by-frame analysis has been carried out to obtain the mode and mechanism of vortex shedding over a complete cycle of oscillation.

3. Numerical simulation
The numerical analysis of the same model has been carried out setting the boundary domain as per Bilal[8]. The size of the computational domain has been fixed to 47B x 21B x 18.75B. The dimensions for CFD analysis is as same as the model for experimental analysis. The geometry of the CFD model includes the modelling of computational domain, principal building and interfering buildings. The geometry has been modelled in ANSYS Design Modeller 17.0.

![Figure 7. The geometry of the buildings modelled in Ansys Design Modeller](image)

The meshing is done for discretization into tetrahedral elements. On conducting a grid independence study, the number of elements for meshing is confirmed and meshing 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. As pressure-based solver is used, for both the convection terms and viscous terms of governing equation i.e. Conservation of mass and momentum equations, uses first order upwind discretization scheme. The cell-zone condition is defined as fluid (air). Boundary conditions are set as velocity inlet for inlet, pressure-outlet for outlet, symmetry for front and back faces of domain. The velocity profile is assumed to be parabolic in shape, defined as UDF velocity.

![Figure 8. The tetrahedral mesh generated around the buildings in ANSYS 17.0](image)
4. Results and discussions
The study comprises of comparison of the pressure and flow patterns by experimental analysis and numerical simulation. The table 2 gives the value obtained from manometers attached to Wind tunnel.

**Table 2. Pressure values derived from Wind Tunnel test at front face of principal building**

| Port No: | Actual Height (m) | Scaled Height (mm) | Head (mm) | Pressure (pa) | Cp     |
|----------|-------------------|--------------------|-----------|---------------|--------|
| 1        | 15                | 25                 | 136.3     | 1056.31       | 0.0987 |
| 2        | 30                | 50                 | 134.0     | 1038.49       | 0.1345 |
| 3        | 45                | 75                 | 135.6     | 1050.89       | 0.1096 |
| 4        | 60                | 100                | 138.8     | 1075.69       | 0.0598 |
| 5        | 75                | 125                | 140.8     | 1091.19       | 0.0286 |
| 6        | 90                | 150                | 140.3     | 1087.31       | 0.0364 |
| 7        | 105               | 175                | 142.0     | 1100.49       | 0.0099 |
| 8        | 120               | 200                | 140.0     | 1084.99       | 0.0411 |
| 9        | 135               | 225                | 136.5     | 1057.86       | 0.0956 |
| 10       | 150               | 250                | 132.0     | 1022.99       | 0.1656 |
| 11       | 165               | 275                | 129.5     | 1003.61       | 0.2046 |
| 12       | 180               | 300                | 128.3     | 994.31        | 0.2232 |
| 13       | 192               | 325                | 128.0     | 991.99        | 0.2279 |
| 14       | 210               | 350                | 128.8     | 998.19        | 0.2155 |
| 15       | 225               | 375                | 140.7     | 1090.41       | 0.0302 |

The manometers embedded in the holes of the model, on subjected to wind forces, showed variation in heads denoting the value of pressure. Since it is always preferred to compare the dimensionless quantities, the coefficient of pressure is computed. The plot of pressure (represented in terms of Coefficient of Pressure) versus height from experimental analysis is computed from the above-mentioned values. Whereas, in case of CFD analysis, the value of Cp verses height of building is directly obtained after the computational analysis. The values obtained in both analyses is cumulated into single graph as shown in fig. 9.
The experimental analysis by water channel visualization yields the flow pattern, i.e., the occurrence of turbulence and eddy formation in the vicinity of object. The same condition can be computed in CFD to get the turbulence pattern and even quantify the turbulent kinetic energy.

![Figure 10. Comparison of turbulence around the buildings, obtained from a) Water Channel Visualization b) CFD analysis](image)

5. Conclusions
The study emphasizes on the results obtained from experimental analysis and numerical computations. From the result of coefficient of pressure vs height of building, the plot seems almost similar with a slight deviation in the beginning. It may be because of the limited domain for a fully developed flow in case of experimental test. The results of flow pattern obtained from water channel visualization and CFD analysis are also similar, but CFD analysis giving a clear picture of the distribution in intensity of turbulence in the vicinity of object. From the study it can be thus efficiently concluded as CFD analysis proves advantages over Wind Tunnel test without compromising accuracy.

To the structural engineer the efficient and reliable result may be justified for conducting experimental studies, while the developer may be more concerned with economics than with academic accuracy. The advantages of CFD is that it doesn’t require the preparations of physical set up and except the cost of a commercial software license for an initial investment, not much cost is involved. Further CFD ensures the flexibility of choosing any geometry and surroundings. More fluid parameters can be calculated by just solving the simulation once. Hence on a larger perspective, the time and cost requirements on numerical simulation are less compared to the setting charges involved in experimental analysis. Hence, the most modern approach of CFD analysis can be used in computing effect of wind load on tall structures as well. CFD in Civil Engineering is a comparatively young initiative on which more research areas are opened up.

References

[1] Blocken B 2014 50 years of Computational Wind Engineering: Past, present and future J. Wind Eng. Ind. Aerodyn. 129 pp 69-102
[2] Kwon D K and Kareem A 2013 Comparative study of major international wind codes and standards for wind effects on tall building Engg. Structures51 pp 23-35
[3] Yamada T and Meroney R N 1971 Numerical and wind tunnel simulation of airflow over an obstacle Nat. Conf. on Atmospheric Waves, American Meteorological Society (Salt Lake City, Oct. 12-15)
[4] Hirt C W and Cook J L 1972 Calculating three-dimensional flows around structures and over rough terrain. J. Comp. Phy.10 pp 324-340
[5] Murakami S and Mochida A 1988 3-D numerical simulation of airflow around a cubic mode means of the k-ε model J. Wind Eng. Ind. Aerodyn. 31 pp 283-303
[6] Baskaran A and Stathopoulos T 1989 Computational evaluation of wind effects on buildings, *Building and Environment* **24(4)** pp 325-333

[7] Baals D D and Corliss W R 1981 *Wind Tunnels of NASA, NASA SP-440*, National Aeronautics and Space Administration, Washington

[8] Bilal A 2015 *Wind effect on super-tall building using Computational Fluid Dynamics and Structural Dynamics* Master Thesis- Florida Atlantic University