Numerical prediction of deflection of a cylindrical hollow structure due to steady wind loads

C Shravankumar¹, S Gowtham², B V Lalith Kumar², V Praneeth², D Siva Krishna Reddy¹*
¹Research Assistant Professors, Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur, Chennai, India.
²Graduate Students, Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur, Chennai, India.
E-mail: sivakriv@srmist.edu.in

Abstract. A raising population and to meet the raising needs there is an increasing demand for tall structure both for commercial use and industrial purpose. Wind behaviour is a key design parameter for such structures and need to be assessed accurately in the preliminary and secondary design stages. This study is aimed at prediction and analysis of deflection of hollow structure due to steady wind loads. Hollow structures typically represent chimneys that are used in the coal fired stream power plant. A hollow cylindrical part with base diameter of 6 cm is fabricated and tested in wind tunnel at constant speeds of 10, 15 and 20 m/s. An accelerometer is mounted on top of body to measure the deflection. Next, the deflection of the body is predicted numerically using commercial ANSYS software. Initially Computational Fluid Dynamics (CFD) simulations are performed to predict the flow field and associated wind force acting on the body. The wind load is transferred to the structural solver to predict the deflection of the body. The predicted deflection compared well with the wind tunnel experiments. Further FSI simulations are performed by changing the thickness of the hollow structure. The results are analysed to study the effect of wind speed and thickness on the deflection. A cubic polynomial curve-fit for the deflection, as a function of the wind speed is developed.

1. Introduction
The advancements in globalisation and industrialisation have increased the demand for power generation and supply. Consequently, the need for chimneys and tall structures used in power plants and refineries are also increased. The purpose of chimney is to expel the poisonous exhaust flue gases to the higher elevation. Chimneys are expected to withstand adverse weather conditions like high speed and turbulent winds and seismic vibrations for a considerable period of time. Chimneys are constructed vertically or nearly vertical to ensure free flow of hot gases. They are designed tall to discharge exhaust gases at higher elevation as per pollution regulations to maintain quality air at ground level.

Various studies were performed by researchersto study the effects of wind forces acting on tall structures. Pavan et.al.[1] studied lateral displacement, bending moment and shear stress characteristics across the height of chimney for different seismic zone regions and wind velocity of 39m/s,44m/s,49m/s and 50m/s for two different chimneys of height 90 m and 110 m designed as per IS 6533. Bilal Assaad [2] used Reynolds-averaged Navier-Stokes and Large Eddy Simulations to a square prismatic prototype structure in which its dynamic properties have also been investigated. With proper modelling of the atmospheric boundary layer flow, these numerical techniques reveal important aerodynamic properties and enhance flow visualization to structural engineers in a virtual environment.Amitha [3] calculated limiting deflection and maximum lateral deflection of reinforced
concrete chimney with five different heights above 275m. The chimney was subjected to static self-load, maximum temperature of 45°C and along wind loads at 39m/s in a seismic zone II region as per Indian Standard code. Rakshith[4] has analysed deflection, stress, base moment of cantilever steel chimney designed as per Indian Standards considering gust loading, aerodynamic effects, vortex formation, vortex excitation, seismic and temperature effects in STAAD-PRO platform. Dynamic analysis of 3D tall concrete chimney was performed by varying its slope in the motive to obtain the optimum chimney geometry. Its deflection and stress is studied for a gust wind speed of 55 m/s in seismic zone III using gust factor method code in STAAD-PRO software by Chaithanya [5]. Dongshong Chen[6] has presented method to estimate the effects of fatigue and inelastic behaviour on structural safety of tall buildings in hurricanes wind environment. The results demonstrated that a ductile structure does have a significance reserve of safety against collapse due to wind action.

Many studies were conducted for wind tunnel testing some of which include, mode shape corrections considering long wind, cross wind and torsional responses for wind tunnel testing of tall buildings were studied by X.L. Xu [7]. Zhang[8] has investigated the effect of wind induced interference with different angle of attack on the pressure distribution on neighbouring building in a staggered arrangement. The pressures and base forces were obtained experimentally in a low speed boundary layer wind tunnel and used renormalization group (RNG) version of the k-ε model along with SIMPLEC algorithm based on the incompressible RANS equation for numerical investigation. Samali [9] performed Wind tunnel testing and its reliable results with respect to modelling requirements was justified by its pressure coefficient and wind forces measured at 16 different elevations of the scaled down building model by.

Hence, Fluid structural interaction (FSI) method was decided to perform the numerical simulation study of the deflection caused in the chimney through wind and structural loads. The cross section of the wind tunnel fluid domain was measured 0.6m x 0.6m. The scaled down model blockage area is ensured to occupy less than 5% and is fabricated using mild steel for the calculated scaling factor 1:300 refer figure 1. for dimensional specifications.

**Figure 1.** Chimney Geometry: (a) Isometric View (b) Front view (c) Top View.
2. Methodology

A scaled down chimney model is designed as per the dimensions of the wind tunnel section. The blockage normal area of the chimney is calculated to occupy within 5% of wind tunnel fluid test section to obtain more accurate experimental results [10]. The geometry is then fabricated and wind tunnel experiment is performed. FSI study is performed for the scaled down geometry. Experimental and numerical results are compared for the result deviations. The obtained results are extrapolated and curve fitting is performed for the simplification and estimation of the wind load effects on the chimney.

2.1 Simulation Methodology

ANSYS – Fluent is used for simulating the flow field around the chimney. Simulations are performed on structured grids, which are generated using ANSYS Workbench meshing tool. The incompressible RANS equation are solved with pressure-based algorithm under steady state condition. κ-ω SST model is used to close the turbulence terms in RANS equation. SIMPLE algorithm is employed to couple pressure and velocity. For both convection and viscous terms second order discretization schemes is used. At inlet, free stream conditions are imposed. At outlet, zero-gauge pressure is applied. No-slip boundary condition is applied on chimney walls.

To solve the near wall region a fine mesh is used and the γ+ is maintained to be less than 5 in majority of the portions. The first grid point in the wall-normal direction is placed at a distance of 1.468e-5 m from wall. The below figures 2(a, b) shows the schematic of the structured grid used in the simulations.

![Figure 2. Schematic of the structured grid used in simulations : (a) Top View and (b) Computation Domain.](image)

Three cases were simulated with along wind velocity 10 m/s, 15 m/s and 20 m/s. The Turbulent properties (k and ω) for the three velocities are given in the table below (Table 1)

| Velocity(m/s) | Turbulent kinetic energy (m^2/s^2) | Turbulent dissipation rate (s^-1) |
|--------------|-----------------------------------|----------------------------------|
| 10           | 0.001855                          | 2074.688                         |
| 15           | 0.00278                           | 3112.033                         |
| 20           | 0.003712                          | 4149.377                         |

2.2 Structural simulation

The structural simulations were performed in ANSYS Static Structural. Geometry and solution is imported from the Fluid Flow (Fluent) module to static structural module. Standard structural steel material data is imported to the model. A fine hex dominant all quad element structural mesh method is performed with 41417 nodes and 8495 elements. Standard earth gravity, fixed support at the base mountings were assigned as boundary conditions. Fluid solid interfaces are selected and fluid pressure is imported from ansys fluent to the designated surfaces. Pressure mapping from fluent to static
structural is ensured to be 100% accurate else setup file meshing is further improvised and selection of fluid-solid interface is more refined.

2.3 Wind tunnel testing

Wind tunnel test was performed in horizontal subsonic open loop wind tunnel established in SRM Institute of Science and Technology by Department of Aerospace Engineering shown in figure 4(a). The dimension of the test section is measured to be 0.6m x 0.6m x 1.5m and maximum safe operating velocity is limited to 20m/s. Scaled down chimney is fabricated using mild steel and is fastened to the wind tunnel base through bolts and lock nuts. National Instruments NI9234 data acquisition module is used to log data into NI Signal Express 2015 software. NI-DAQ977 is used as data acquisition console to extract data from accelerometer and log the analogue signal into the software as pictured in figure 4(b). Accelerometer with sensitivity 100 mV/g is mounted at the top most point of the model to record deflection of the model. Testing is done at 10m/s, 15m/s and 20m/s at steady state condition and the acceleration values are obtained which is further calculated to obtain deflection values.

3. Results

3.1 CFD simulation results

CFD results for the three simulation are obtained relatively quickly due to the fact that this study is done for a steady fluid flow case with well refined mesh. The figures 5 and 6 below show the velocity contour and pressure contour around the cylinder at a height of 20 cm from the bottom of the chimney respectively.

The computed flow field in terms of velocity contours for freestream velocity of 10 m/s are shown in Figure  5 (a)(b). Flow of air decelerates due to obstruction caused by chimney. As a result, velocity of air at the nose stagnation point approaches to zero. From nose stagnation point, expansion of flow occurs. Hence Velocity of air progressively increases as distance from the nose stagnation point.
increases. It attains a maximum value of 31 m/s at an angle of 76°. Further downstream, flow separates and results in formation of low velocity region behind the cylinder. The low velocity region is known as near wake. A shear layer sheds the wake from outer expanding flow. The shear layers from top and bottom side of the cylinder colloid and results in the formation of vortex shedding. The qualitative features of the flow remain same for freestream velocities of 15 and 20 m/s. As the velocity of freestream increases, the strength of vortex shedding also increases.

Figure 5. Velocity Contour at a height of 0.2m from the bottom of the chimney at: (a), (b) 20m/s (c), (d) 15m/s (e), (f) 10m/s.
Figure 6. Pressure Contour at a height of 0.2m from the bottom of the chimney at: (a)10m/s (b)15m/s (c)20m/s
Figure 6 shows gauge pressure contours around the cylinder for the three cases. As the flow decelerates in the nose stagnation region, pressure in this region higher. Flow expansion results in decrease of pressure in circumferential direction. Minimum pressure is observed at an angle of 76° for 10, 15 and 20 m/s respectively. In the wake region, pressure of air is low. Over all, front side which faces the air experiences higher pressure. As the freestream velocity increases, gauge pressure at nose stagnation point also increases. Its value is 67.51, 150.9 and 266.45 Pa for 10, 15 and 20 m/s respectively.

The figure 7 (a, b) below show the variations of coefficient of drag (Cd) and coefficient of lift (Cl) respectively with respect to the freestream velocity.

![Coefficient of Lift vs Velocity](image1)

![Coefficient of Drag vs Velocity](image2)

**Figure 7.** Variation in forces with velocity : (a) Cd vs Velocity and (b) Cl vs Velocity.

The results obtained concludes that the drag and the lift force increases with increase in velocity. This is as expected since both drag force and lift forces are directly proportional to the freestream velocity. Hence the results obtained after the FSI analysis should result a similar behaviour in deflections and stresses on the chimney.

### 3.2 Structural Simulation Results

Total deformation data is obtained from the solution after coupling fluent and static structural domain and is tabulated in Table 2. Maximum deflection is observed at the top most point of the chimney for all the velocity conditions. For a wind velocity of 10m/s and 15m/s the ansys simulated FSI study has resulted 5.0291µm and 7.9175µm of deflection respectively. The maximum deflection is observed to occur at wind velocity of 20m/s and the deflection with respect to wind velocity is also observed significantly higher after 15m/s.

![Structural Deflections for Wind Velocity](image3)

**Figure 8.** Structural deflections for wind velocity (a)10m/s (b)15m/s (c)20m/s.
Table 2. FSI analysis results.

| Velocity (m/s) | Deflection (µm) |
|---------------|-----------------|
| 10            | 5.0291          |
| 15            | 7.9175          |
| 20            | 18.618          |

3.3 Wind tunnel testing Results

An analogue signal was generated by the accelerometer and logged by the data acquisition setup; acceleration (m/s\(^2\)) vs time (s) and magnitude vs frequency (Hz) signals were produced while performing the experiment. Analog signal processing was performed to interpret these data signals to obtain deformation results and are tabulated in Table 3.

![Figure 8](image-url)  
**Figure 8.** Data Acquisition signals from accelerometer for wind velocities (a)10m/s (b)15m/s (c)20m/s.
Table 3. Wind tunnel testing data.

| Velocity (m/s) | Sampling          | Acceleration ($\ddot{x}$ g) | Frequency ($\omega$ Hz) | Deflection ($x$ µm) |
|---------------|-------------------|------------------------------|-------------------------|---------------------|
| 10            | Low sampling      | 0.02890                      | 35.48                   | 5.7110              |
|               | High sampling     | 0.02600                      | 35.03                   | 5.2698              |
| 15            | Low sampling      | 0.02041                      | 33.90                   | 4.4177              |
|               | High sampling     | 0.03733                      | 34.82                   | 7.6590              |
| 20            | Low sampling      | 0.09300                      | 33.87                   | 16.7489             |
|               | High sampling     | 0.07200                      | 32.70                   | 19.7550             |

Data from FSI analysis and experimental wind tunnel setup are compared and error is recorded in Table 5. The error obtained from experiment and the numerical simulations were considered under limits and legit of the FSI analysis for the hollow structures such as cylinders is proven. The obtained data is extrapolated in the graph and curve fitting is performed as pictured in figure 9 (a, b). This study methodology can further be implemented for real time tall structures and analysis can also be performed dynamically for transient wind loads.

Table 5. Comparison between ansys and experimental data.

| Velocity (m/s) | Ansys simulation data (µm) | Experimental data (µm) | Relative Error (µm) | Error (%) |
|---------------|-----------------------------|------------------------|---------------------|-----------|
| 10            | 5.0291                      | 5.2698                 | -0.2407             | 4.56754   |
| 15            | 7.9175                      | 7.659                  | 0.2585              | 3.375114  |
| 20            | 18.618                      | 19.755                 | -1.137              | 5.7555    |

Figure 9.(a) Deflection vs velocity graph (b) Curve fitting graph.

The curve fitting results were evaluated for relative error to validate the accuracy of curve fitting methodology and is tabulated in Table 6. The curve fitting error with actual results are under acceptable limits hence this methodology is inferred to be valid.

Table 6. Comparison between curve fitting and ansys data.

| Velocity (m/s) | Curve fitting deflection data (µm) | Ansys deflection data (µm) | Relative Error (µm) | Error (%) |
|---------------|----------------------------------|----------------------------|---------------------|-----------|
| 10            | 5.025                            | 5.0291                     | 0.0041              | 0.081525521 |
| 15            | 7.908                            | 7.9175                     | 0.0095              | 0.11998737 |
| 20            | 18.601                           | 18.618                     | 0.017               | 0.091309485 |
4. Conclusion
The extrapolated curve fitting equation is found to be $y = 0.1562x^2 - 3.3284x + 22.689$ where $y$ represents maximum deflection(µm) of the model at the top and $x$ represents velocity(m/s) which can be aided in estimation of deflection at different wind velocities. The possible reasons for the error occurred between FSI analysis and experimental results could be slight non rigidity of the base being mounted in the wind tunnel setup whereas software evaluates fixed support as completely rigid which is almost impossible in experimental conditions. Software assumes both fluid and material properties to be homogeneous which is not true in real testing and manufacturing. The data signal from the DAQ could have internal noise which is filtered manually though cannot be completely eliminated. The proposed methodology is justified legit and all results are well defined under acceptable limits. Extrapolation of results can reduce the computation complexity to a greater extent and maximum deflection can be estimated for different wind velocities. Hence, major catastrophes could be prevented without any loss in industrial and human resources.

Nomenclature
CFD = Computational Fluid dynamics
FSI = Fluid Structural Interaction
κ = Turbulent Kinetic Energy, J/kg
ω = Specific turbulent dissipation rate, 1/s
SST = Shear Stress Transport
C_d = Drag Coefficient
C_L = Lift Coefficient

5. References
[1] Kumar M P, Raju P M, Babu N V and Roopesh K 2017 A Parametric Study Resistance of Ste. International Journal of Civil Engineering 8
[2] Assaad B 2015 Wind Effect on Super-tall Buildings Using Computational Fluid Dynamics and Structural Dynamics Doctoral dissertation, Florida Atlantic University
[3] Amitha Baiju and Geethu 2016 Analysis of tall RC chimney as per Indian Standard Code International Journal of Science and Research 5
[4] Rakshith B D, Ranjith A, Sanjith J and Chethan G 2015 Analysis of Cantilever Steel Chimney As Per Indian Standards Journal of Engineering Search and Applications 5
[5] Chaithanya Varada Prasad K, Radhika K L, Anuradha P and Santhosh Reddy B 2018 Dynamic analysis of a tall chimney International Journal for Research in Applied Science & Engineering Technology 6
[6] Chen D 1999 Vulnerability of tall buildings in hurricanes. Faculty of Graduate Studies University of Western Ontario
[7] Xu Y L and Kwok K C S 1993 Mode shape corrections for wind tunnel tests of tall buildings Engineering Structures 15 387-392
[8] Zhang A and Gu M 2008 Wind tunnel tests and numerical simulations of wind pressures on buildings in staggered arrangement Journal of Wind Engineering and Industrial Aerodynamics 96 2067-2079
[9] Samali B, Kwok K S, Wood G S and Yang J N 2004 Wind tunnel tests for wind-excited benchmark building Journal of Engineering Mechanics 130 447-450
[10] Chen TY and Liou L R 2011 Blockage corrections in wind tunnel tests of small horizontal-axis wind turbines Experimental Thermal and Fluid Science 35 565-569
[11] Khan M. and Roy A K 2017. CFD Simulation of Wind Effects on Industrial RCC Chimney Int J CivEngTechnol. 8 1008-102