CFD Analysis of wind flow behavior around Tall Stacks: A Review

[1]D. R. Adwani, [2]M. S. Joshi, [3]S. M. Ranade, [4]A. M. Patil

[1]M-Tech, Thermal Student, [2]Professor, [3] Professor, [4]Sr. CFD Engineer

[1, 2, 3]Department of Mechanical Engineering, [4]B & H Department

[1, 2, 3]Walchand College of Engineering, Sangli, India

[4]Thermax Ltd., Pune, India

Abstract: More and more concerns are made over the safety issues which are directly and indirectly related to the human lives**.** One of such big issue is failure of high elevated structures experiencing high velocity winds with turbulence. Turbulence is complex phenomenon which is difficult to explain mathematically as its generation depends on various parameters. Geographical conditions, Terrain type, surface roughness, Interference etc are some the parameters. Because of uncertainty in such parameters, what nature fluctuating components of wind is carrying is hard to say many times. Also with heavy demands, industrialization is increasing and due to this wind flow behavior on one structure is very well affected by surrounding structures. It is been seen that due to interference, drag, lift and moment on structures increased. Also the vibrations due to interference can have some dangerous consequences like generation of vortex shedding, vibrations of wind when matched with natural frequency of structures can cause its failure. This review paper deals with the effects on stacks in atmosphere and CFD approach needed for the simulation.

***Index Terms* – CFD, Lift, Drag, Stacks**

I. INTRODUCTION

Large numbers of tall chimneys are to be constructed every year, as large scale industrial developments are taking place all around the world. The primary and the key function of stacks is to discharge harmful gases to a higher elevation such that the gases do not pollute the atmosphere we live in. Height of chimney or stack has been increasing since last few years because there have been complaints of increasing air pollution. Due to their tall slenderness shape with the circular cross section, chimneys have different structural problems and are not treated in a same way as done with other form of tower structures. Design and analysis of stacks depends on numerous factors such as wind force, surrounding conditions, materials type used and shape and cross sectional area of the chimney [1]. Tall chimneys form crucial part of maximum industries and power plants. Damage to the chimneys ultimately leads to shut down of the industry and power plants

1.1 Need of Validation

The proper design and construction of chimneys will create self-standing structures to resist wind load and other forces acting on them. Common practice is to consider the effects of wind and earthquake separately in the design. When it comes in a business of designing and constructing of such structures, organization needs to fix the height of such chimneys with almost care. Considering engineering and economics, justice to safety is a priority. Factors such as type of stack construction, stack dimensions, outlet velocity at stack, dust collector installation, geographical location and last but not the least size of ultimate plant are some of the economic and engineering factors affecting the design of stacks [2]. Improper termination of chimney leads to formation of eddy zone which causes pollutants to enter the lower atmosphere causing threat to living beings [3]. After studying various literatures on types of loading on stacks, Mr.Praveen Kumar & Dr.Ajay swarup in their paper, "Review of self supporting steel chimney" has concluded that wind effect is more unpredictable than that of earthquake effect. Hence base and top diameter of chimney, height of chimney and its thickness are crucial factors in its design and analysis. Before its design these parameters need to be carefully considered. For overcoming wind forces some engineers stated the guy roped steel chimney and comparative studies between self supportive and guy roped chimneys has shown that later one gives less moment at base than previous [4]. At any point on the chimney, the wind load exerted is the sum of quasistatic and dynamic load component. If wind blows at mean steady speed, producing steady displacement in a structure then the force exerted by the wind is the static load. The component which causes oscillation of structures is the Dynamic component of wind which is produced due to many reasons. According to requirement and safety index, height with precise wind load effects and vibration analysis need to experimented and validated. There are many structures which affect the flow around it. Due to the interference effect of surrounding structures chimney has to experience Vortex Shedding which may cause failure of stack. Thus the stacks with structures around it must be analyzed in wind tunnel. With advancements in technology customers are sometimes asking for CFD analysis of such structures for validation of results done with the help of wind tunnel.

II. LITERATURE REVIEW

Siva et al. [1] presents the along and across wind effects on a 275m tall RCC lined chimneys for Ist & VIth wind zones of India and the results say that in shell completed condition, for zone I, which has basic wind speed of 33 m/s, across winds are dominating and for highest wind zone of VI, which has basic wind speed of 55 m/s, along wind loads are governing or dominant as compared to that of the across wind loads.

Jigar and Atul [2] has attempted to analyze the industrial steel chimney for the wind forces considering topographic factor and risk factor. With the increase in height, the wind forces have become important while designing and analyzing such structures. In this paper, an author has attempted to analyze the industrial steel chimney for the wind forces considering chimney with and without guy ropes.

Bohidar et al. [3] in his paper deals with wind load effects on tall chimneys. Large scale movement of air due to thermal currents is wind. From ancient times, it had played and continues to play crucial role in design of tall structures because static and dynamic loads are exerted whose effects on a slender structure like chimney are significant. At any point on the chimney, the wind load exerted is the sum of quasistatic and dynamic load component. Without deep knowledge of basic wind engineering concepts, chimneys' response can't be predicted.

Roy and Khan [6] has estimated the design wind loads, provided some guidelines for structures with unconventional geometry such as industrial chimneys. CFD analysis along the model creation, meshing, setting up physics with the turbulence models has been done to find the alternative to wind tunneling testing.

Manohar [7] in his book has explained in details the chimney sizing taking into account safety and environmental issues. Besides wind effects on tall chimneys along with the causes of failure is stated. Finally the need of model simulation is explained and also the limitations

Mohd. Mohsin Khan [8] has studied wind flow behavior around stack with and without interference. He had reduced the geometry of model in ratio 1:600 and along with turbulence model and pressure velocity coupling he gave boundary conditions for CFD analysis of stacks.

J. Revuz et al [9] has compared various computational domain sizes i:e small, medium, proposed and large. After comparing they have concluded the optimum size for computational domain. Optimum size will at last affects the mesh count and hence computation time and hardware cost

H K Versteeg and W Malalasekera [11] in their book, "An introduction to Computational Fluid Dynamics" has explained many concepts on discretization and turbulent models based on finite volume method.

S. Chakraborty [10] has explained how the dimensional and flow similarity is essential if we are reducing the model by some ratio.

III. Loads on Stacks in Wind

Seasonal changes, typhoons, waves in ocean, structural failures etc are some of the crucial aspects effecting living things on earth. Large scale movement of air due to thermal currents is wind. It has very important role of transporting and dispersing pollutants. As it exerts static and dynamic loads on slender structure like chimney, wind effects is most important factor in its design. To predict wind effects precisely by analytical procedures is highly impossible because of winds' uncertain variability and therefore approximate design techniques are used by the designer [5]. Along wind effects and across wind effects are two major topics in dairies of researchers dealing with effects of winds on manmade structures. Following table shows their immediate cause and effects

Table 3.1.Various loads on stacks with causes and effects

Loads	Cause	Effects
Static component on Stack	Wind Flowing at mean steady state	Steady displacement in structure
Dynamic Component on Stacks	Fluctuations component in velocity and turbulence	Gusts, Vortex Shedding and Wake buffeting

Along wind effects:

- Stack is considered as cantilever
- Drag component of wind is considered and moments

Across wind effects:

- Lift component is considered
- Occur due to velocity dependent forces and vortex shedding which make the chimney to oscillate in direction perpendicular to wind flow

IV. CFD Staircase to Analysis for Stacks

4.1 Modeling

Stacks with accurate dimensions are modeled in any modeling software like CATIA, Design Modeler etc. If stack's dimensions are too large then they are reduced in some ratio, so that while meshing it the elements and nodes are within limit to control the computational time. When geometry of prototype is reduced or increased, parameters change and are obtained by the concept of similarity to accommodate the changes taken place due to change in dimensions.

4.1.1Fluid Domain

While analyzing the wind flow around stacks there must be controlled space in which behavior is studied. For every problem of CFD analysis there is optimum fluid domain size which comes with experience. Various people have studied and tried to define the standards for defining the size of fluid domain.

Some of the findings for optimum computational domain are,

- An upwind fetch, 20 B or 2 H, whichever is the greater.
- A downwind fetch, 30 B or 3 H, whichever is the greater.
- A top clearance, 4 H.
- Side clearances, 26 B by Revuz [6] (B is a width and H is the height of structure)and

5 H, 15 H, 5 H and 5 H respectively by Franke. Et. Al [7]

4.1.2Similarity

If dimensions are reduced or increased then results taken from CFD post are not valid until before simulation parameters are manipulated by using similarity. It is concept which shows the change in parameters when the dimensions are reduced or increased. Following three criteria are necessary for making sure that CFD output can be extrapolated for prototype.

4.1.2.1 Geometric Similarity

When dimensions of corresponding structures are in fixed constant ratio, then the condition of Geometric Similarity is met.

4.1.2.2 Kinematic Similarity

When homogenous particles of structures lie at the same points at the same time, then the condition of kinematic similarity is met

4.1.2.3 Dynamic Similarity

When Forces on structures at same corresponding points are in constant ratio then the condition of Dynamic Similarity is met.

For facing the multiple variables at the same time Mr. Buckingham gave method to group the variables that matters for that particular process. This method is called Buckingham pi theorem and that group of variables is called Dimensionless number or entity or Pi terms.

4.2Meshing

CFD for finding solutions at various points uses concept of discretization of geometry into various elements and nodes. It uses various techniques which are pre-defined in software. Near stacks inflation layers are made to capture boundary layer. It is done by software available in ANSYS itself or can be done by separate software. Mesh near stacks need to be fine and with good quality as accuracy of results is much needed at that place while when we go away from stacks coarse mesh is acceptable. Thus the conclusion is fine mesh with good quality gives the good quality over the coarse mesh. But there must be limit to the number of elements and number of nodes otherwise the computational time will increase exceptionally with not much increase in accuracy. Increasing the mesh number i.e. doing mesh finer and finer after sometime will not increase the accuracy of results and at that point and after that no accuracy can be increased. Finding out the results which are independent on mesh is called grid independent test. This test helps to find the best results from CFD.

4.3Boundary Conditions

Every step in analysis is crucial but this step has input of physics. Setup tab defines the conditions, in which stacks are at every instant like the velocity of air, type of turbulence models, pressure-velocity coupling etc. Shih, et al. 1995, concluded that Realizable k-ε turbulence model, SIMPLE algorithm for pressure velocity coupling and second order for pressure interpolation gives the more accurate results. Thus setting up the physics has for such situations are reached by matching the simulating conditions with the pre-defined definitions of models. Also many times it goes more with experiencing as many as problems someone can.

At high heights the velocity is much greater than the velocity at ground level. So, constant velocity profile can’t give the expected results. In such cases user defined function (UDF) help to create varying velocity. This is done in any programming language and has to fed as input at velocity tab of software. This concept is broadly called as Atmospheric Boundary Layer (ABL). The layer of an air directly above the earth surface in which the effects of surface (Friction, heating and cooling) are felt on the time scale and , in which the significant fluxes of momentum, heat or matter are carried by the turbulent motions on the scales of order of depth of boundary layer or less. The region of frictional influence is called the atmospheric boundary layer and it is similar in many respects to the turbulent boundary layer on a flat plate at high wind speeds. The Boundary Layer wind tunnel is different from ABL in two different ways; firstly, thermal turbulence coexists with mechanical. Secondly, boundary layer interacts with mean flow is affected by the rotation of earth. [8]

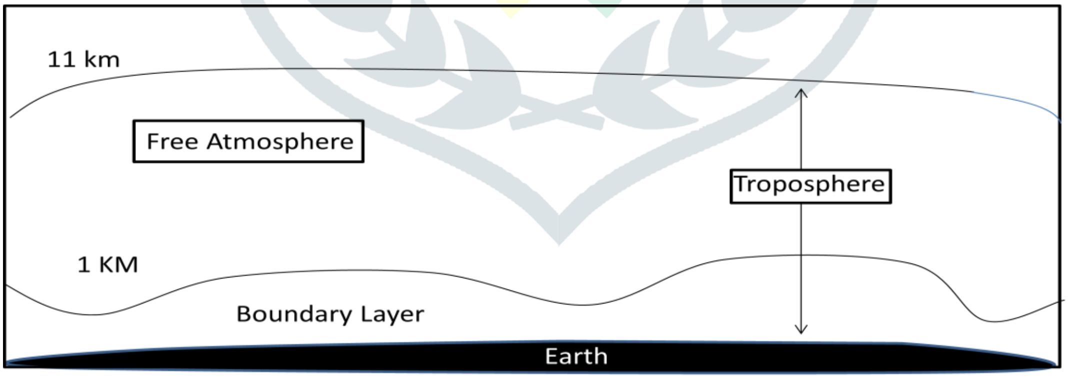

Figure 4.1.Atmospheric Boundary Layer

4.4 CFD Post

Simulation results are seen in CFD post, fourth and the step that is essentially depends on the art of interpretation of the person. In this case, for finding out the drag, lift and moment, Force report tab in solution is used. There in, is a need to give direction to find corresponding forces. If on stack the wind is coming from X-direction then defining direction vector X in force report tab we get the drag force acting the stack. Defining vector that is vertical to wind flowing direction on stack, gives the lift force acting on the stack. Similarly moment can be found on the stack. Besides velocity contour for finding out the maximum velocity magnitude and location, Pressure contours, streamline, and vector plots etc. can be found as per the requirement. Also for finding out the vortex shedding at particular velocity transient simulations can be carried out.

Interference Factor- It’s the ratio of drag force or lift force on the stack when it is simulated alone i.e. without any structures around or without interference to the ratio of drag force or lift force on the same stack with interference. As discussed with heavy industrialization the problem of interference is increasing day by day and hence it is becoming necessary to find this factor and try to keep it as close to unity. For assuring the results got from CFD are up to mark or not, different countries has laid down standards to find out the above calculated forces by considering various factors like in Indian standards topography factor, risk factor and terrain and structure size factor. Country to country these standards vary and accordingly boundary conditions are varied in CFD approach.

V. Fluid Structure Interaction (FSI)

After finding the forces along the stacks in CFD post, these forces are transferred to static structural part of the ANSYS. It will help to find out the maximum deflection that can happen with the forces that are exerted by the fluid on stacks. This is known as one way coupling. Again transferring the calculated deflection in static structural analysis into fluent and examining that after deflection, how stack is behaving in wind is said to be two-way coupling. Stacks can be considered have negligible deflection and hence one way coupling is enough for complete analysis of stacks.

References

[1]K. Siva, V. P. Rohini and C. H. Srikanth, "Study of wind load effects on tall REC chimneys", IJAET/Vol.III/ Issue II/April-June, 2012/92-97

[2]H. L. Von Hohenleiten & R. H. Kent, “Economic and Engineering Considerations in the Design of Stacks for Good Gas Dispersion”, ISSN: 0096-6665 (Print) (Online) Journal 2012.

[3]J. F. SCHULZ, “Factors involved in the design of high rise chimney and chute systems”, The flinkote Company East Rutherford, New Jersy

[4]R. Boopathiraja et al, “Comparative Design and Analysis of Self Supporting and Guyed Steel Chimney”, IJIRST –International Journal for Innovative Research in Science & Technology| Volume 3 | Issue 07 | December 2016

[5]S. Manohar, "Tall Chimneys", Tata McGraw-Hill,1985

[6]J. Revuz et al, “On the domain size for the steady-state CFD modeling of a tall building, Wind and Structures, Vol. 15, No. 4 (2012) 313-329

[7]Franke, J. (2007), Introduction to the prediction of wind loads on buildings by CFD, Wind effects on buildings and design of wind sensitive structures, Springer Wein, New York

[8] Roland B. Stull, An Introduction to Boundary Layer Meteorology, 1999.