International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 03 Issue: 08 | Aug-2016 www.irjet.net p-ISSN: 2395-0072

"Optimization of Wind Loads Affecting Skyscrapers using ANSYS"

Kavana B C[1], M S kalappa[2]

[1]MTech student, dept of civil Engineering, MCE, Hassan, Karnataka, India
[2]Professor, dept of civil Engineering, MCE, Hassan, Karnataka, India

---***---

Abstract - *In present days, the field of civil construction 'Skyscrapers' are taking its tall in a competitive way. Starting from the date 1885 with 'Home Insurance Building' at Chicago, USA to latest 'Burj Khalifa' at UAE, we have seen many numbers of skyscrapers being built. In this study, we will be looking forward on the common dispute faced by skyscrapers that is effect of 'Wind Energy' on these buildings.The present study is about providing a building with aerodynamic structure to avoid wind loads acting on them and verifying them with numerical analysis CFD. To avoid those loads we approach with the two well-known theories in the field of aerodynamics i.e., 'Coanda Effect & Vortex Generation'*

Key Words: Skyscrapers, Wind Energy, CFD Analysis, Aerodynamic Structure.

1. INTRODUCTION

As we study deeper about skyscrapers, we notice that wind loads have been the dispute for those structures from the very beginning of their construction. However, by the laws of aerodynamics we can minimize the wind loads on the structure of the building.

1.1 SKYSCRAPER

Any building which is continuously habitable and whose structure is above 150m height from ground usually comes under 'skyscraper' category. In addition, buildings above 600m are considered to be 'Super-tall' buildings. Age of skyscraper started from 1885 with Home Insurance Building which is claimed to be the first tallest building habitable constructed. Today we have Burj Khalifa which is above 800m.

1.2Aerodynamics

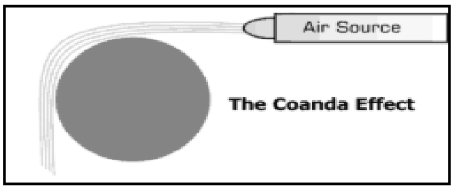

Fig-1: Coanda effect.

It is a study of fluid flow around any solid body in motion. However, in our skyscrapers they are stagnant and wind flow effects are measured around them still in this case fluid is moving so we can still consider as flow effects around a body. Understanding the flow of fluid around a building will allow us to calculate the forces & moments acting on it. Using which we can eliminate the wind loads acting on the building by providing them with the suitable structure with minimal lose in volume of the building.

1.3 COANDA EFFECT

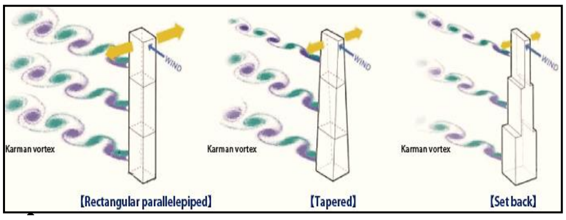

Fig-2: Karman vortex effect.

The tendency of fluid in motion to stick to the convex surface is known as Coanda effect. The study of this effect is usually implied in the field of aviation to study the airfoil and their characteristics. Using this effect on the skyscrapers by making suitably small convex portions throughout the building outer surface, the wind is made to stick to the building outer contours.

1.4 VORTEX GENERATION

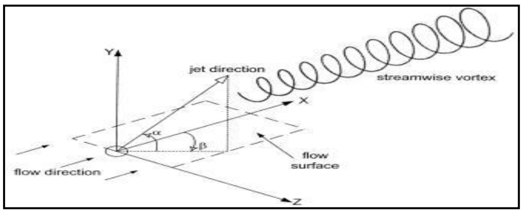

Fig-3: Vortex generation.

When the two or more fluids in motion with different velocity meet, the vortex is generated. This vortex generation has drawback that is known to producing more drag with more velocity, but when reverse vortex is generated it is familiar to be cancelling out each other, which in return compensates the amount drag generated.

Vortex shedding frequency N is given by,

$$N = S\frac{U}{b} \quad \ldots.(1)$$

Where,

$S \rightarrow Strouhal\ Number \qquad b \rightarrow Building\ Width$
$U \rightarrow Wind\ Speed$

Peak response due to vortex excitation,

$$U_{CRIT} = \frac{N_\gamma b}{S} \quad \ldots.(2)$$

Where,

$N_\gamma \rightarrow No\ Vortex\ Shedding$

$$Magnitude\ of\ Peak\ Response \propto \frac{1}{density * damping} \quad \ldots.(3)$$

2 METHODOLOGY

Numerical Technique is called as CFD 'Computational Fluid Dynamics' which is linked with the study and simulation of fluid flow, forces & heat transfer rates. Numerical analysis is turned out to be a very powerful tool for flow simulation and calculation for industrial and non-industrial solicitation area.

CFD were used mainly in aeronautical & aerospace industries, but now in the present field of construction skyscrapers have been a trend from past four decades, field of civil has started to use wind tunnels and flow analysis software's to finalize design criteria. CFD codes work in following steps like Pre-Processor, Solver & Post-Processor.

2.1.1 DESIGNING

Making use of SOLIDWORKS design a scaled model of building (1:100) and see to that all the required counters are closed before exiting the modeling phase.

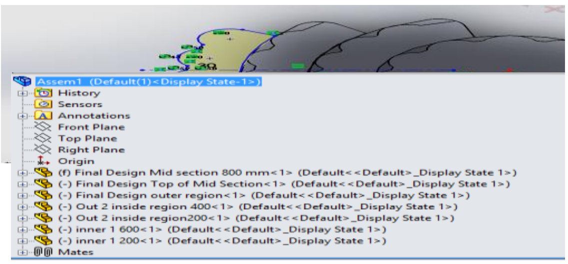

Fig-4: Solid works initial 2D drawing.

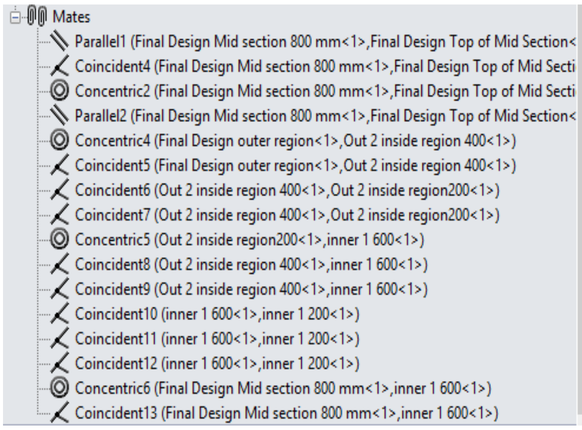

Fig-5: Assembly options.

2.1.2 CREATING CONTROL VOLUME

Importing the model to the analysis workbench create equally distribute control volume. The control volume created in this experiment is as follows:

Table-1: Domain size.

Coordinate	Length in mm
X_1	500
Y_1	1500
Z_1	500
X_2	-500
Y_2	-1500
Z_2	-500

2.1.3 MESHING OF THE DOMAIN

After creating the control volume for the flow field it is required to be meshed to carry on the analysis and iterations as required by the numerical analysis methodology.

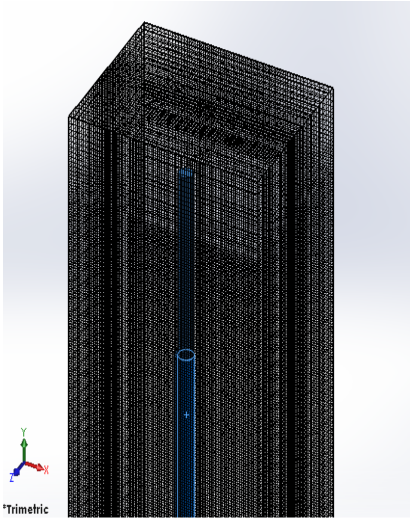

Fig-7: Mesh.

2.1.4 PRE-PROCESSORS

After meshing of the model is done boundary condition are to be set to provide the flow parameters, and conditions to be set for the wall of the building. The following are the parameters set for carrying out this entry analysis work.

Table-2: Boundary Condition.

Sl. No.	Parameters	Condition set
1.	Analysis Type	External Flow
2.	External Flow Condition	Exclude both Cavities without flow condition & Internal space
3.	Reference Axis	X
4.	Fluids	Gases-Air
5.	Flow Type	Laminar & Turbulent
6.	Wall Condition	Adiabatic
7.	Roughness of wall	Default
8.	Pressure	101325 Pa
9.	Temperature	293.2 K
10.	Vector	3D Vector
11.	Velocity	X Direction 15m/s
12.	Turbulence	2%
13.	Goals	Static Pressure
		Total Pressure
		Velocity in X
		Force on X
		Friction Force on X
14.	Results	Flow Trajectory
		Surface Plot

2.1.5 SOLVER

Initialize all the above mentioned condition and check for the problem statement once all the conditions are set run the iteration.

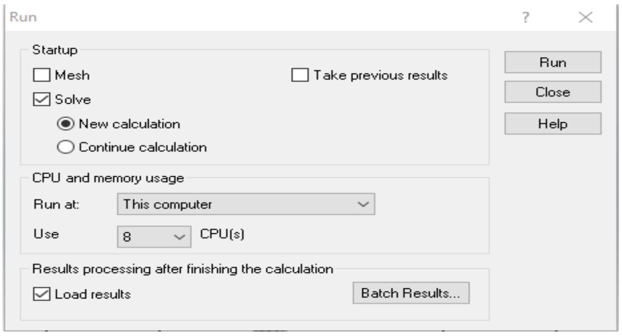

Fig-8: Final simulation setup.

2.1.6 POST PROCESSING

After the solving is done, generate the flow trajectory setting the parameter to 20 Nodes. Then extract the file to Excel Sheet to generate the graph. All the graphs of respective design are shown in appendix section of the report.

2.2 INITIAL DESIGN OF SET BACK

Table-3: Initial design building pressure table.

Pressure (Pa)		Velocity_(X) (m/s)	
Trajectory Length (m)	Flow Trajectories	Trajectory Length (m)	Flow Trajectories
-0.21	101343.46	-0.21	14.95
-0.21	101343.61	-0.21	14.95
-0.19	101344.25	-0.19	14.93
-0.18	101344.95	-0.18	14.91
-0.16	101346.42	-0.16	14.84

-0.15	101347.88	-0.15	14.77
-0.13	101350.31	-0.13	14.63
-0.12	101352.79	-0.12	14.48
-0.11	101356.86	-0.11	14.19
-0.09	101361.19	-0.09	13.87
-0.08	101367.43	-0.08	12.95
-0.06	101373.99	-0.06	11.90
-0.05	101379.00	-0.05	10.92
-0.04	101385.05	-0.04	10.08
-0.04	101386.77	-0.04	9.80
-0.03	101386.73	-0.03	9.75
-0.03	101385.21	-0.03	9.75
-0.03	101382.03	-0.03	9.73
-0.03	101379.95	-0.03	9.71
-0.03	101378.55	-0.03	9.70
-0.03	101377.58	-0.03	9.70

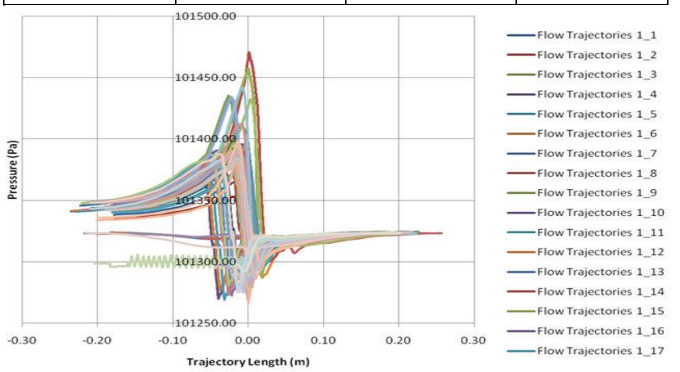

Chart-1: Initial design building pressure distribution.

2.3 FINAL DESIGN

Table-4: Final design building pressure table.

Pressure (Pa)		Velocity_(X) (m/s)	
Trajectory Length (m)	Flow Trajectories	Trajectory Length (m)	Flow Trajectories
-0.03	172790.92	-0.03	14.80
-0.03	173621.92	-0.03	14.77
-0.02	175278.17	-0.02	14.71
-0.02	176931.10	-0.02	14.66
-0.02	179995.70	-0.02	14.48
-0.02	183576.77	-0.02	14.26
-0.02	187146.74	-0.02	14.04
-0.02	192389.56	-0.02	13.61
-0.02	197653.20	-0.02	13.17
-0.02	203461.88	-0.02	12.67
-0.02	211031.50	-0.02	11.97
-0.02	218508.38	-0.02	11.27
-0.02	226603.56	-0.02	10.34
-0.01	235047.19	-0.01	9.23
-0.01	242886.91	-0.01	8.16
-0.01	248168.45	-0.01	7.01
-0.01	251126.54	-0.01	5.97
-0.01	244069.56	-0.01	5.74
-0.01	231085.60	-0.01	5.95
-0.01	217381.27	-0.01	6.35
-0.01	206492.74	-0.01	6.65

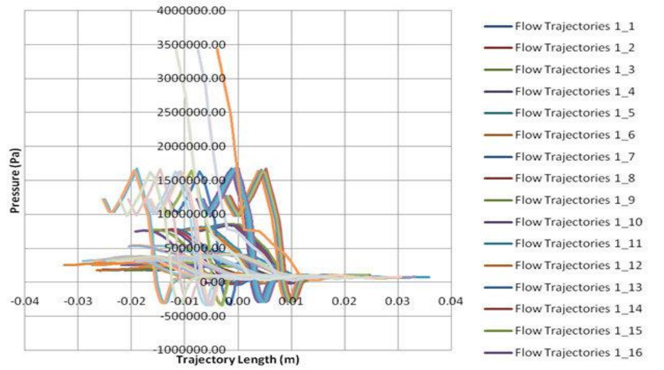

Chart-2: Final design building pressure distribution.

2.4 CALCULATIONS

*All the values presented in this calculation is taken out from the CFD analysis represented in appendix section of this report

Substituting Values in Equation no.(5)

$$V_w = (0.2^2 + 0.3^2)^{1/2} = 0.36$$

2. Substitute V_w in equation (4)

$$\lambda_w = \frac{1}{1.3} e^{0.75^{2*3.5*0.36}} = 1.56$$

First order and second order momentum values taken down by the flow analysis.

3 RESULTS

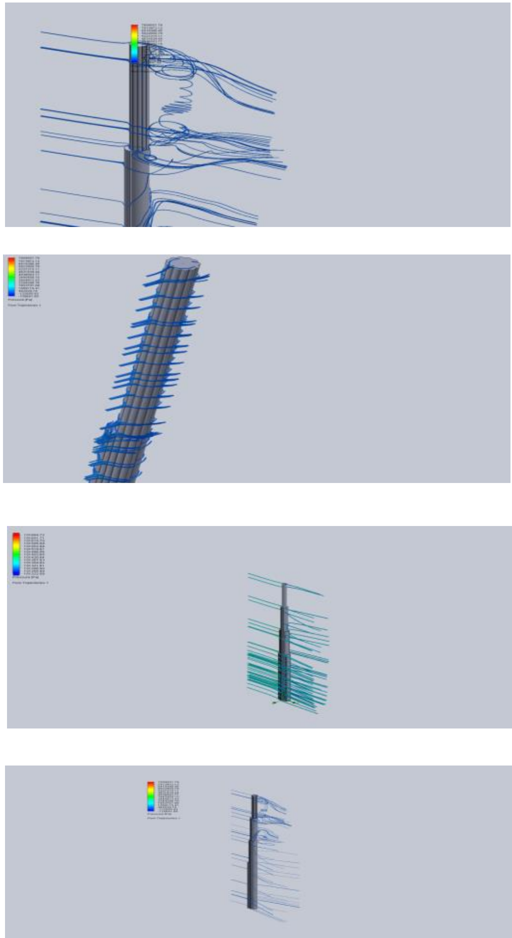

Fig-9: Initial and final design building results

CONCLUSION

Wind tunnel testing will go about as a capable device to outline compositional and structure configuration of a building. Utilizing several rounds of varied forces and parameter wind tunnel allows us to refine architecture of building. Such as CFD analysis also helps in noticing them and correcting parameters which can be done using Reynolds number values as we have done in this course of work.

By this work we can conclude that skyscraper with the following design section can be more aero elastic load barer. Comparing with all other possible design as done we can clearly see that damping values are of 30% lesser and the structure can withstand better loads with this architecture.

Pedestrian comfort including of thermal comfort to the oscillation control of structure due to wind loads are increased with the architecture of the building.

REFERENCES

[1] Soligo, M.J., Irwin, P.A., Williams, C.J., and Schuyler, G.D., 1997 **"A comprehensive assessment of pedestrian comfort including thermal comfort"**, Proceedings of the Eighth U.S. National Conference on Wind Engineering, Baltimore MD, June.

[2] F. Liu, S. Cong, **"Reliability analysis of portal frame based on space overall ultimate bearing capacity",** Applied Mechanics and Materials, Vols. 501-504, pp. 634-639, 2014.

[3] C. M. Guo, Y. S. Bai, W. Li, **"Coupled simulation and analysis of wind environment around buildings"**, Advanced Materials Research, Vols. 953-954, pp. 497-500, 2014.

[4] S. Medvecka, O. Ivankova, **"Effect of inclination of columns on the change of the stiffness of a high-rise building"**, Applied Mechanics and Materials, Vol. 769, pp. 9-12, 2015.

[5] Aerodynamic mitigation of wind-induced uplift forces on low-rise buildings: A comparative study.

[6] Seismic analysis of high-rise mega-braced frame-core buildings.

[7] Irwin, P.A. 1988 **"Pressure model techniques for cladding loads"**, Wind Engineering and Industrial Aerodynamics, 29, pp 69-78.

[8] ASCE, 2003 **"Outdoor human comfort and its assessment"**, State of the Art Report, prepared by a Task Committee of the Aerodynamics Committee of the American Society of Civil Engineers, chaired by P. Irwin, published by ASCE, Reston, Virginia.