International Journal of Advance Research in Science and Engineering
Vol. No.5, Issue No. 04, April 2016
www.ijarse.com
IJARSE
ISSN 2319 - 8354

EFFECT OF WIND ON BUILDING ORIENTATION USING ANSYS CFX

Mahesh Bariya[1], Bhairav Thakkar[2]

[1]Undergraduate Student, [2]Associate Professor, Civil Engineering Department,
Navrachana University, Vadodara (India)

ABSTRACT

When we design high-rise buildings the orientation of building and shape of building influence the wind force velocity and consequently the design. Prediction of the wind direction and intensity of wind is difficult. The intensity and direction of wind for a maximum period of one year can be found using the wind rose diagram. This work present an approach of using the wind rose diagram along with computational fluid dynamics simulation of wind flowing over building to determine optimum orientation of building with design point of view. The simulations of wind on building are performed by ANSYS CFX.Wind pressures on the building considering various orientation of the building have been determined to be situated in Ahmadabad and an appropriate wind-rose diagram has been considered. This approach is expected to give significant insight and variation of wind pressures on building in various orientations and would allow more appropriate orientation. The column must be designed for higher loads even for shorter periods. However, this approach would be of immense help in determining fatigue in the frames and location of wind pressure release mechanisms.

Keywords: ANSYS CFX,Computational Fluid Dynamics, Wind analysis, Wind tunnel, Wind Rose Diagram

I. INTRODUCTION

Wind produces pressures different than the anticipated ones when the building orientation is not optimum. Consequently, it is recognized that a thorough wind flow analysis around the structure must be carried out in order to determine optimum loads on the structures [1]. While designing structures, the position and orientation of the structure must be decided keeping the flow around all edges of building in wind. This study presents an analysis carried out to study the position and orientation of building in such way that pressure exerted by wind on the building is Minimum.The models are subjected to wind blowing in various directions. A schematic of the building plans is shown in figure 1.

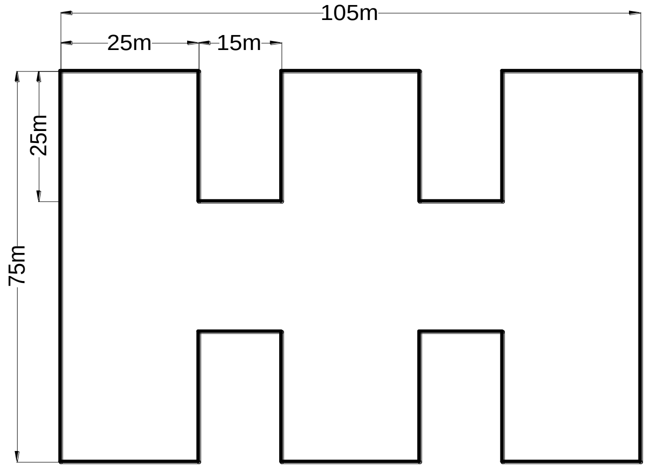

Figure 1 Plan of Building under consideration

A wind rose diagram shows the frequency and speed of wind blowing from each direction over the year. As you move outward on the radial scale, the frequency associated with wind coming from that direction increases. Each spoke is divided by color into wind speed ranges. The radial length of each spoke around the circle is the percentage of time that the wind blows from that direction. Wind rose diagram taken for In this sample Speed Distribution wind rose (05/01/2016 To 04/04/2016 wind rose from Ahmadabad, India), winds from the NW and NE directions are most common (up to 10% of annual hours).The winds from the N direction, wind speeds are most often in the 7.2-10.8 and 10.8-18 range (yellow and light blue) [2].From the wind rose diagram the appropriate velocity for each orientation were determined and consider for CFD simulation.

II. RESULT AND DISCUSSION

Wind rose diagram shows value and direction of wind over the year and the same have been used in CFD analysis. Wind rose diagram shows that NE, N and NW directions maximum wind velocity. ANSYS CFX analysis shows thewind velocities on the edge of building. Fig. 2,3,4,5 and 6 show the flow of wind around the building. Among 0^0,10^0,20^0,30^0 and 90^0orientations, the building experiences minimum pressure. When oriented at 10^0. It may be inferred that, the building should be oriented in a direction where the pressures are minimum.

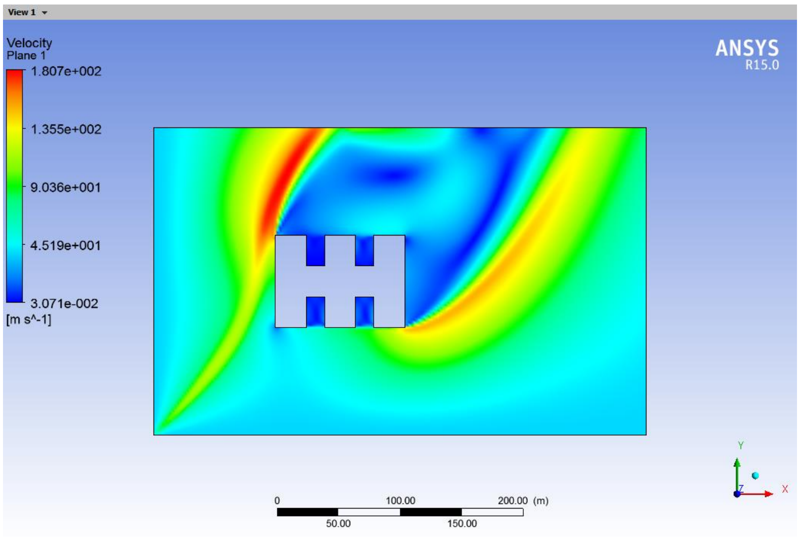

Figure 2Wind velocity on Building at 0^0

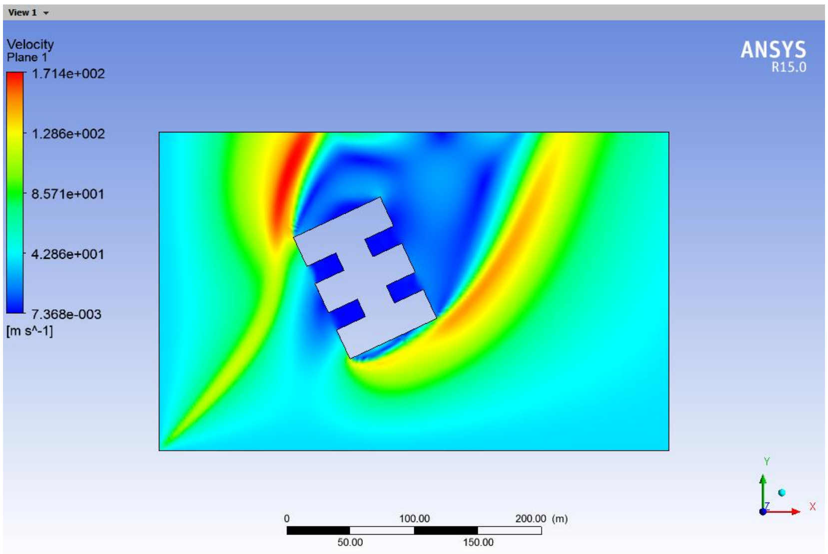

Figure 3 Wind velocity on Building at 10^0

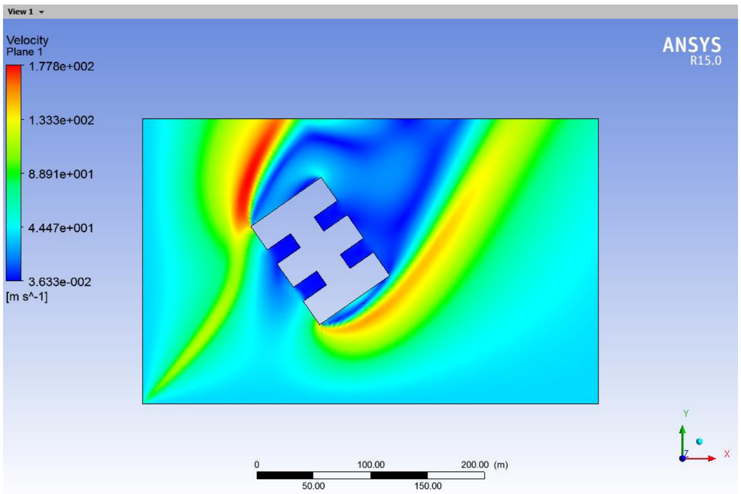

Figure 4 Wind velocity on Building at 20^0

International Journal of Advance Research in Science and Engineering
Vol. No.5, Issue No. 04, April 2016
www.ijarse.com
IJARSE
ISSN 2319 - 8354

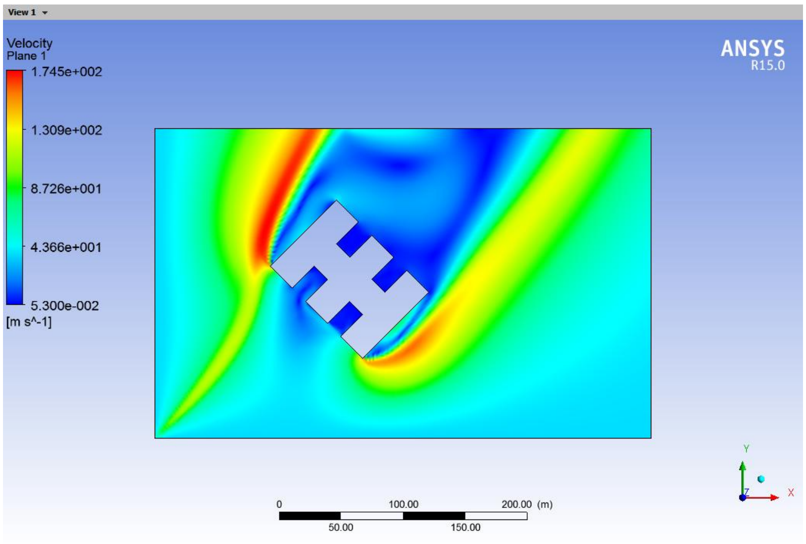

Figure 5 Wind velocity on Building at 30^0

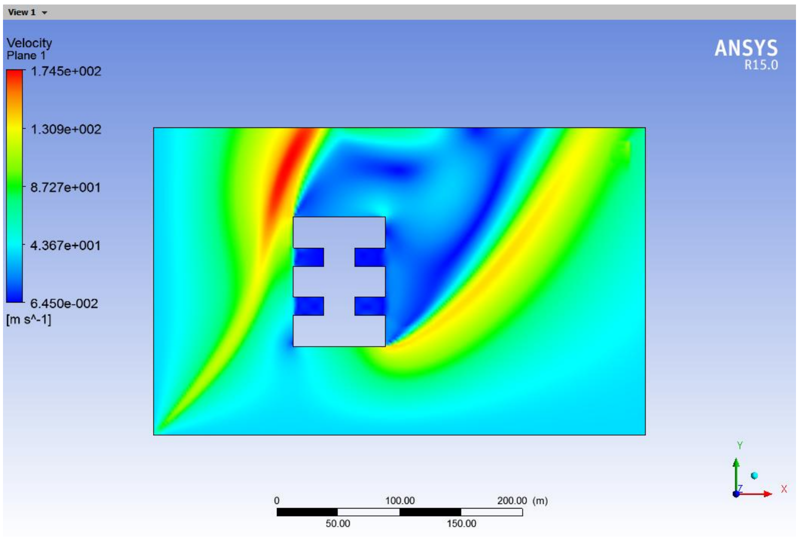

Figure 6Wind velocity on Building at 90^0

III. CONCLUSION

Computational fluid dynamics analysis of wind flow around a building has been simulation using Ansys CFX. Various orientations of the building have been considered and the most optimum orientation has been recognized. CFD has been found to be an excellent tool to determine the optimum building orientation over the distribution of wind in various directions over a year. The wind pressure can thus be effectively minimized and building design may be made more economical.

REFERENCE

[1]. IS: 875-1987 “Indian Standard Code of Practice for Design Loads (Other than Earthquake) For Buildings And Structures, Part 3 (Wind Loads)” Bureau Of Indian Standards, New Delhi.

[2]. https://envitrans.com/temppng/8ed8f95895123125fd68477b23c54f8b.png?id=808

[3]. ANSYS CFX Tutorials 12.1,November 2009

[4]. Anders TrondalSvendsen And Allan Michaelsen, (2008) “Aeroelastic Response Of High-Rise Buildings”,Master Thesis, Department Of Civil Engineering - Aalborg University.

[5]. A.G. Hansora, P.N. Nimodya, K.P. Gehlot, (2015) “ Numerical Analysis of Wind loads on Rectangular Shape Tall Buildings”, International Journal for Scientific Research &Development, Volume 3, Issue 03.

[6]. Ehsan Vafaei Hosseini, Azadehsgheb, P.K. Ramancharla, (2013)“Analysis of High-rise Building using Computational Fluid Dynamics Approach: ACase Study on 38-Storey High-rise Building”, A3C-12: AWARDS, CONVENTION & CONSULTANTS COLLOQUIUM, and January 11-12.

[7]. C.A.J. Fletcher, I.F. Mayer, A. Eghlimi, K.H.A. Wee,(2001) “CFD As A Building Services Engineering Tool”, International Journal on Architectural Science, Volume 2, Number 3, page.67-82.

[8]. Daniel Cóstola, Marcia Alucci, (2007) “Pressure Coefficient Simulated By CFD for Wind-DrivenVentilation Analysis”.

[9]. Huanga, S. et al 2007. "Numerical evaluation of windeffects on a tall steel building by CFD" Journal of Constructional Steel Research Volume 63, Issue 5, Pages 612-627.

[10]. SaddokHouda, Noureddine Zemmouri, Abdelmalek Hasseine, Rachid Athmani, Rafik Belarbi, (2012) “A CFD Model For Simulating Urban Flow In Complex Morphological Street Network”, The Online Journal of Science and Technology- January, Volume 2, Issue 1.

[11]. Blocken B, Stathopoulos T, Carmeliet J. (2007)“CFD simulation of the atmospheric boundary layer: – wallfunction problems”, Atmospheric Environment 41(2): 238-252.

[12]. Tomáš RENČKO, Anna SEDLÁKOVÁ, (2013) “Assessment Of Underfloor VentilationOf Historic Buildings Using ANSYS CFX”,Budownictwo o zoptymalizowanympotencjaleenergetycznym 1(11) , s. 98-106

[13]. Mr. Parag M. Thete, Prof. H. D. Thakare, (2014) “Performance Optimization Of Mixed Flow Impeller Using BackwardBlades”, International Journal Of Pure AndApplied Research In Engineering AndTechnology,Volume 2 (9): 402-408

[14]. Chandrakant D. Mhalungekar, Swapnil P.Wadkar,(2014) “CFD And Experimental Analysis Of VortexShedding Behind D-Shaped Cylinder”,International Journal of Innovative Research in Advanced Engineering, Volume 1 Issue 5.