

# Determination of Optimum Domain Size for 3D Numerical Simulation in ANSYS CFX

Rocky Patel<sup>1</sup>, Satyen Ramani<sup>2</sup>PG Student, Department of Civil, SAL Institute of Technology and Engineering Research, Ahmedabad, Gujarat, India<sup>1</sup>Asst. Professor, Department of Civil, SAL Institute of Technology and Engineering Research, Ahmedabad, Gujarat,  
India<sup>2</sup>

**ABSTRACT:**In the present study, 3D numerical study of wind load on a cube shaped building is carried out to determine domain size required for optimum results. Numerical simulation is carried out in ANSYS CFX package with 1.5 cm wall roughness and 5% initial turbulence. K-epsilon turbulence model is selected for simulating geometry induced turbulence. After determining optimum domain size a cube model is simulated with that domain size and pressure coefficient distribution on windward wall, leeward wall and roof is compared with wind tunnel data obtained by S. Murakami, A. Mochida, Y. Hayashi and S. Sakamoto.

**KEYWORDS:**3D wind Simulation, Numerical Simulation, k-epsilon model, ANSYS CFX, pressure coefficient distribution.

## I. INTRODUCTION

Wind load on structures are estimated with reference of standard codes of practice by structure designers. Code provisions are prepared on the bases of wind tunnel testing of typical shaped buildings. But with evolution of engineering and technology different shapes and geometries of structures are being introduced in construction field. So for particular special case for which there is no particular codal provision experimental study will be required for prediction of wind load on the different faces of structure. Experimental approach involves full scale study and wind tunnel testing. However full scale study of particular building will be costly as well as time consuming. For wind tunnel testing it is require to generate equivalent atmospheric turbulence properties and boundary layer flow inside the wind tunnel which is a bit tedious process as well as data acquisition and data handling is difficult in case of experimental method.

Numerical prediction of wind flow problems provides accurate simulation of required atmospheric boundary layer flow and turbulence conditions. More over data handling during post processing is easier in numerical simulation. Numerical simulation technique of simulating fluid flow is known as computational fluid dynamics which was initially used for aircraft designers but now for simulation of wind flow around engineering structures this method is used which was introduced as name of computational wind engineering. Computational wind engineering has come up with great possibilities in field.

Proper choice of boundary conditions, accuracy of discretisation methods, and turbulence model affects the quality of results. Even size of atmospheric wind domain at which wind flow is simulated is affecting the pressure distribution around the structure. Choice of larger size of domain will provide more accurate prediction of wind pressure coefficients but lead to long simulation process. On the other hand in case of small domain size it will take less time for simulation process but it will have effect of domain boundary condition.

Previously Satyen Ramani, P. N. Godbole and L. M. Gupta have carried out a parametric study for determination of optimum domain size for 2D simulation using ANSYS Flotran. On the basis of their study  $Y=12H$ ,

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

$L_2 = 12H$  and  $L_1 = 5H$  domain size is optimum for 2D simulation in ANSYS Flotran. Dimensioning notations are shown in fig 1.

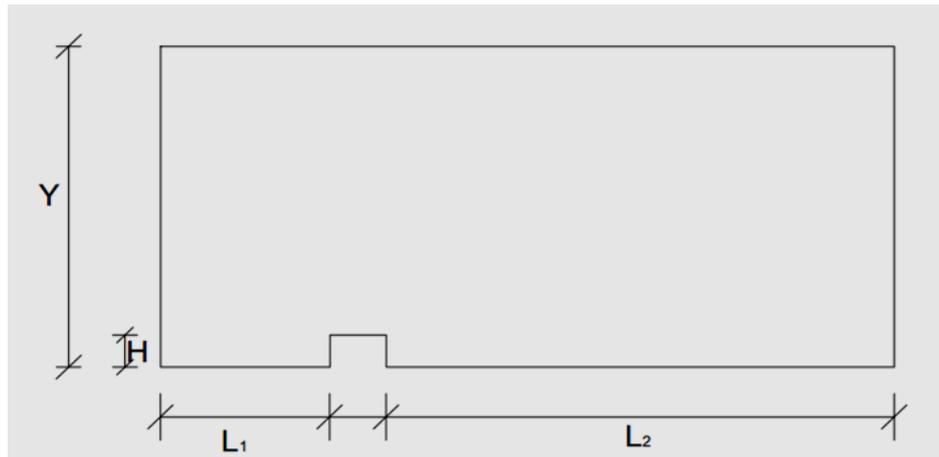


Fig 1. Dimensioning notation for 2d simulation domain.

Similar kind of parametric study was carried out and for 2D simulation in ANSYS CFX with k- epsilon turbulence model. Optimum size of domain for particularly ANSYS CFX can be taken as  $L_1 = 5H$ ,  $L_2 = 15H$  and  $Y = 10H$ .

However 2D simulation in any software package will have significant variation from actual pressure distribution on the building. But 2D analysis results have fair agreement with pressure distribution on mid length of long structures when wind direction is perpendicular to length of building. For detail numerical prediction of pressure distribution on the various parts of buildings 3D simulation is required. And for 3D simulation of any particular structure initial requirement is to decide the domain size. In the first part of paper structures having different height length and span ratios are simulated in 3D and pressure distribution around the structure is observed to decide optimum size of the domain. In the second part of paper results of 3D simulation in ANSYS CFX is compared to wind tunnel experiment result carried out by S. Murakami, A. Mochida, Y. Hayashi and S. Sakamoto to validate results of software.

## II. DETERMINATION OF 3D DOMAIN SIZE

For determination of optimum domain size for 3d simulation in ANSYS CFX three models are chosen to understand the extend turbulence due to presence of structure. Geometric configurations and assumed domain size are mentioned in Table 1. And dimensioning notations are shown Fig 2.

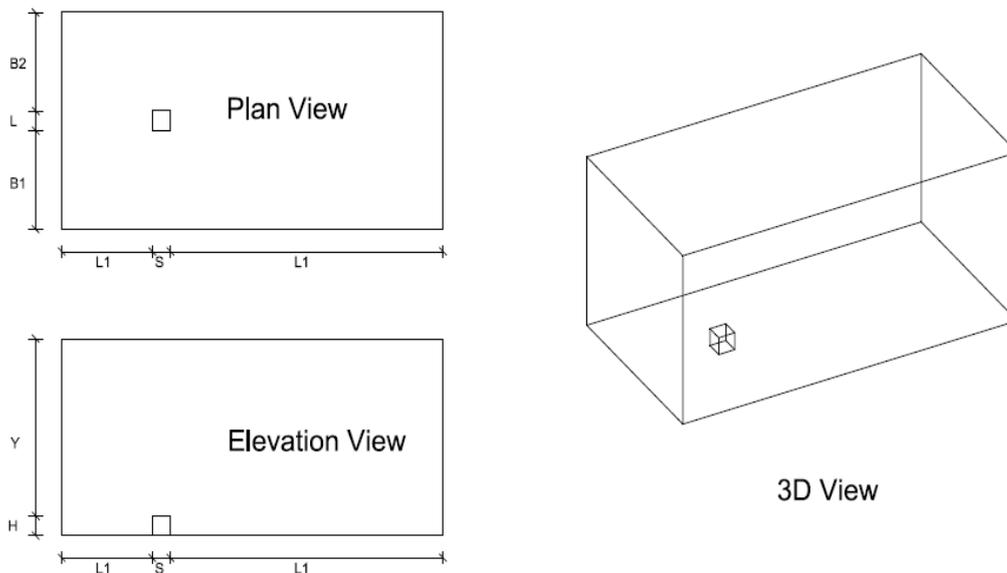
Table1 Geometric Configuration of models for determining domain size

Model	Structural dimensions			Assumed domain size			
	Span	Height	Length	$L_1$	$L_2$	H	$B_1=B_2$
Cube 1	5m	10m	1m	50m	150m	50m	50m
Cube 2	5m	5m	30m	25m	75m	25m	50m
Cube 3	5m	5m	5m	25m	75m	25m	50m

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015



**Fig 2.** Dimensioning notations for 3D simulation

Simulation is carried out for reference pressure of 1 atm to co-relate results with atmospheric wind condition. Other input parameters are mentioned in Table 2.

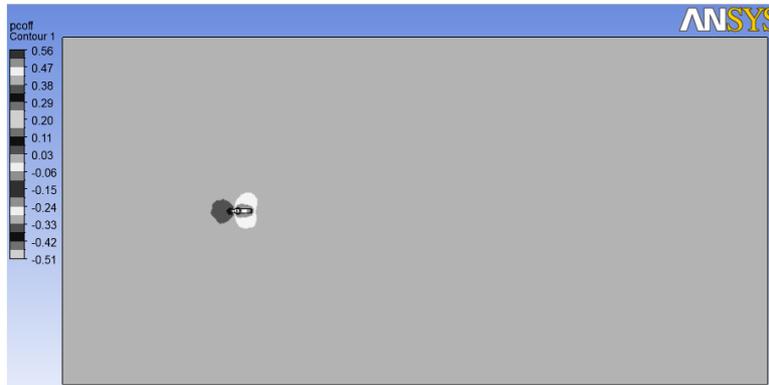
**Table 2** Input Data summary in ANSYS CFX

No	Parameters	Value
1	Velocity Profile	$9.02 \times \left(\frac{y}{5.86}\right)^{0.16}$
2	Ground Roughness	1.5 cm
3	Turbulence intensity at inlet	5 %
4	Density of air	1.182 kg/m <sup>3</sup>
5	Viscosity of air	$1.7594 \times 10^{-5}$ kg/m. s
6	Turbulence model	k-e

### III. RESULTS OF 3D PARAMETRIC STUDY FOR DETERMINING OPTIMUM DOMAIN SIZE

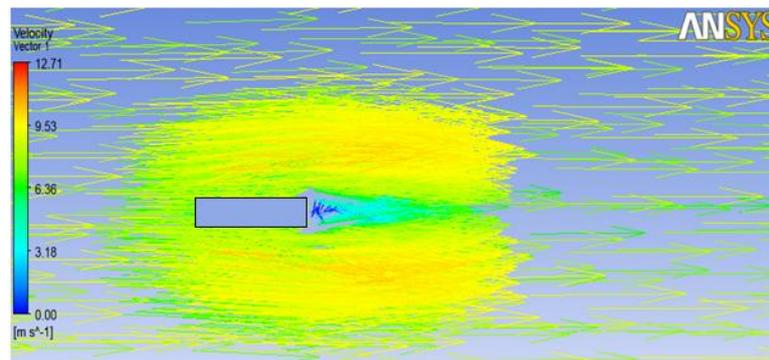
To understand extent of influence of the structure in the domain pressure coefficients are checked at different planes kept at different distances from the structure. And relation between geometry of structure and domain size is derived from above 3 cubical shaped structures are simulations.

**Figure 3** clearly shows pressure distribution contour and it shows that domain size selected for CUBE 1 is extensively large than required for computation of pressure distribution. This is unnecessarily increasing computation time with no significant increase in accuracy of result.



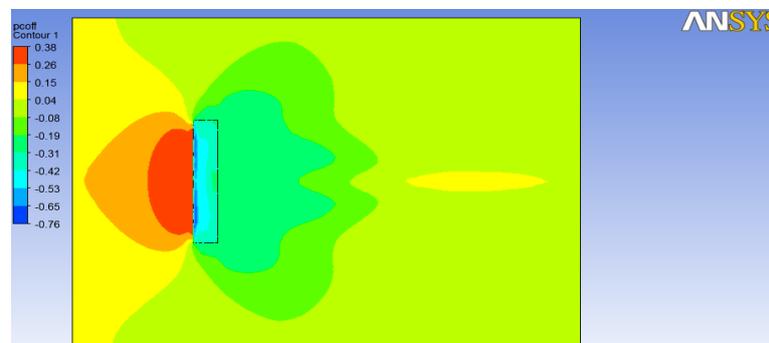
**Fig 3.** Pressure coefficient distributions on the ground for CUBE 1

**Figure 4** shows enlarged view of vector plot around CUBE 1. Congestion of vectors is due to refinement of meshing near walls of cube.



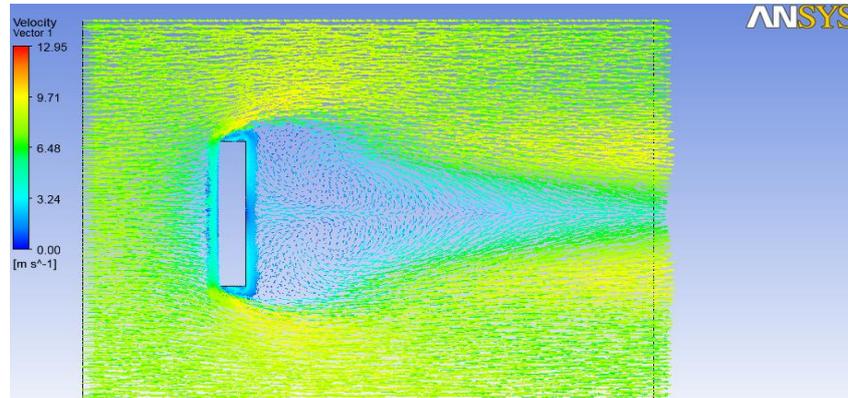
**Fig4.** Velocity vector plot around CUBE 1

**Figure 5** shows pressure distribution contour generated due to obstruction created due to CUBE 2. It clearly shows pressure increase on windward side of the structure and extend of suction on leeward side of structure as well as sides of structure.



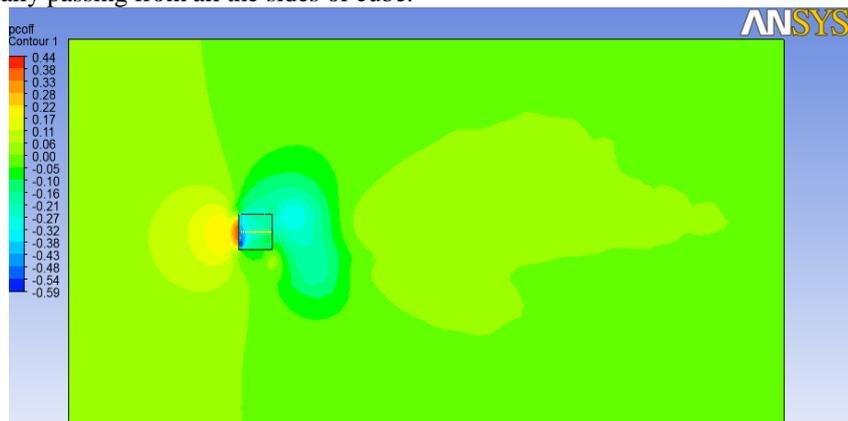
**Fig5.** Pressure coefficient distributions on the ground for CUBE 2

**Figure 6** shows vector plot of wind flow sound CUBE 2. On the leeward side direction of vector shows vortex generation which is creating suction on the leeward wall.



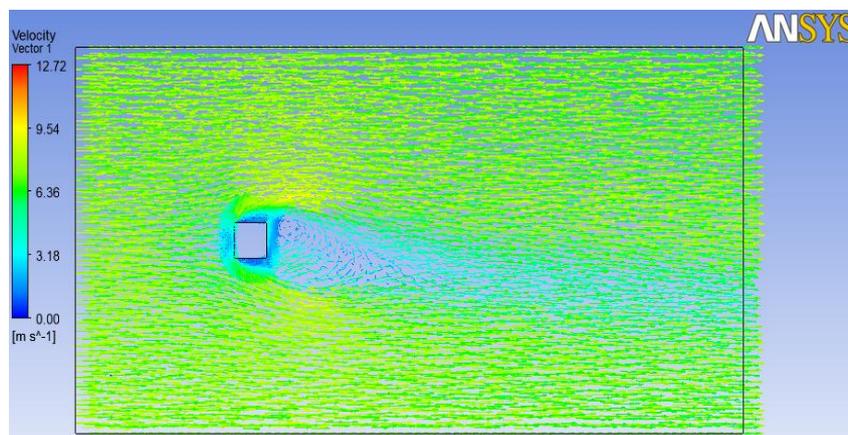
**Fig 6.** Velocity vector plot around CUBE 2

**Figure 7** shows pressure distribution contour around CUBE 3 model. In this model all dimensions of cube is equal so flow is equally passing from all the sides of cube.



**Fig 7.** Pressure coefficient distributions on the ground for CUBE 3

Figure 8 shows vector plot of flow around CUBE 3. This clearly shows that domain size selected for simulation of CUBE 3 is sufficient for required accuracy.



**Fig8.** Velocity vector plot around CUBE 3

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

## IV. 3D DOMAIN SIZE STUDY CONCLUSION

From above simulations we can conclude that height of the structure is not the only parameter which is affecting 3D domain size. Base dimensions of wind ward face which is obstructing the wind flow governs the domain size.

In CUBE 1 model which is having lesser length and more height most of the wind is flowing from the sides of structure and less wind is flowing from above the structure. So while it is obvious that provided domain size is more than required which is unnecessarily increasing simulation time.

In CUBE 2 model which is having more length compared to its height most of the wind will try to pass from above the structure. Providing more side distance will lead to more simulation time without any significant increase in accuracy.

In CUBE 3 model structure is a perfect cubical so wind will equally pass from sides as well as above the structure.

From the observation of result of above 3 models we can suggest that side distance on both the sides in 3D domain size should be at least 5 times minimum dimension of windward face of the structure. Front distance between inlet and windward face of structure should be at least 5 times minimum dimension of windward face of the structure. Optimum back distance between outlet and leeward face of structure should be 15 times minimum dimension of windward face of the structure. Optimum height of the 3D simulation of domain should be at least 10 times minimum dimension of windward face of the structure. So on that bases following optimum domain size can be suggested for above simulations.

**Table 3** Optimum domain sizes for given structural configuration

Model	Structural dimensions			Optimum domain size			
	Span	Height	Length	L <sub>1</sub>	L <sub>2</sub>	H	B <sub>1</sub> =B <sub>2</sub>
CUBE 1	5m	10m	1m	5m	15m	5m	5m
CUBE 2	5m	5m	30m	25m	75m	50m	25m
CUBE 3	5m	5m	5m	25m	75m	50m	25m

## V. 3D VALIDATION STUDY

S. Murakami, A. Mochida, Y. Hayashi and S. Sakamoto from Institute of Industrial Science, University of Tokyo, published a research paper on Numerical simulation on velocity-pressure field and wind forces for bluff bodies. They did a 3D numerical simulation to study pressure fields and wind induced forces on and around a cube shaped building. They used k-epsilon model, algebraic stress model and large eddy simulation for the numerical simulation and compared results with wind tunnel simulation data.

To examine accuracy of ANSYS CFX results same model is simulated in 3D with same boundary conditions and velocity input and results are compared with time averaged pressure field result of wind tunnel data.

### Geometric details, boundary conditions and input data:

Same as 2D simulation wind velocity profile is obtained by power law. Inlet velocity at height y is given by following expression.

$$V(y) = 10.7 \times (y/4)^{0.14}$$

Normal and tangential velocity are set to zero at solid boundary surfaces. Boundary condition near solid wall is described by wall roughness of 1.5 cm. At free stream boundary, flow is prohibited from crossing boundaries. Open terrain category is assumed for study of pressure coefficient distribution around building.

Domain size for 3D numerical simulation with k- epsilon turbulence model is taken on the basis of above study. Front distance of inlet boundary to windward wall face is kept five times height of cube, back distance between outlet boundary and leeward wall is kept 15 times height of cube. Side distance from sides of cube and side boundary is kept 5 times height of cube. Input data is mentioned in following table. Dimensioning notations are same as mentioned in Fig 2.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

Table 4 Geometric details

No.	Parameter	Value
1	Height of cube H	1 m
2	Span of cube S	1 m
3	Length of cube L	1 m
4	Front length of domain $L_1$	5 m
5	Back length of domain $L_2$	15 m
6	Height of domain H	10 m
7	Side distance in domain $B_1$	5 m

## VI. RESULTS AND DISCUSSIONS

Results of 3D simulation in ANSYS CFX are compared with the time averaged pressure coefficients at the vertical section of wind tunnel experimental result of S. Murakami and team. Pressure distribution given by k-epsilon turbulence model on the windward wall, Leeward wall and roof are shown in Figure 6.8, 6.9 and 6.10.

Figure 6.8 shows pressure coefficient distribution on the windward wall. Results of windtunnel testing results and 3D simulation in ANSYS CFX are fairly matching with each other.

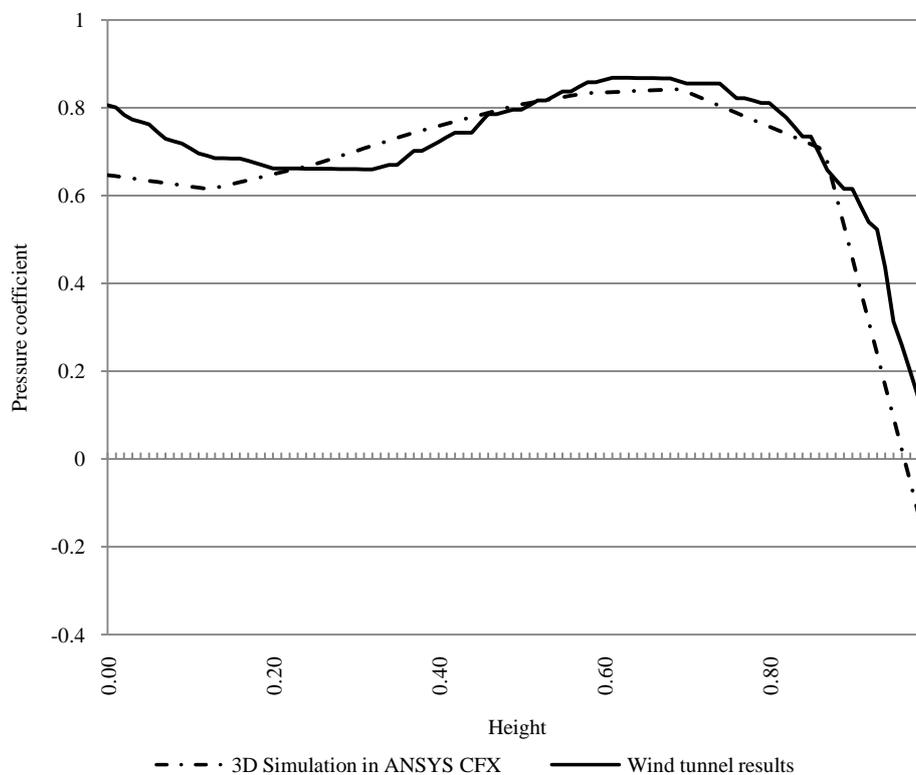


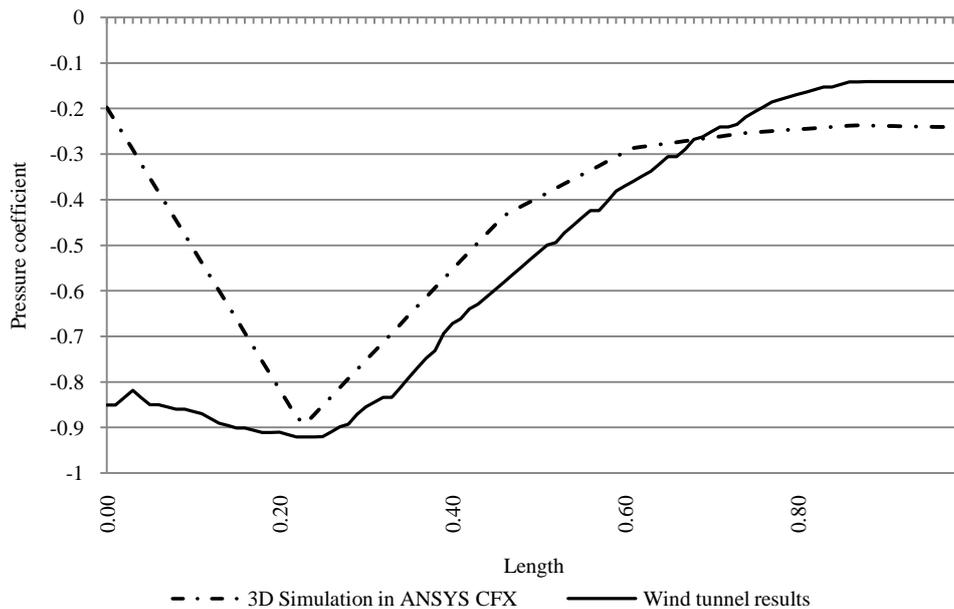
Fig 6.8 Comparison of Pressure coefficient distribution for windward walls on S. Murakami cube model

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

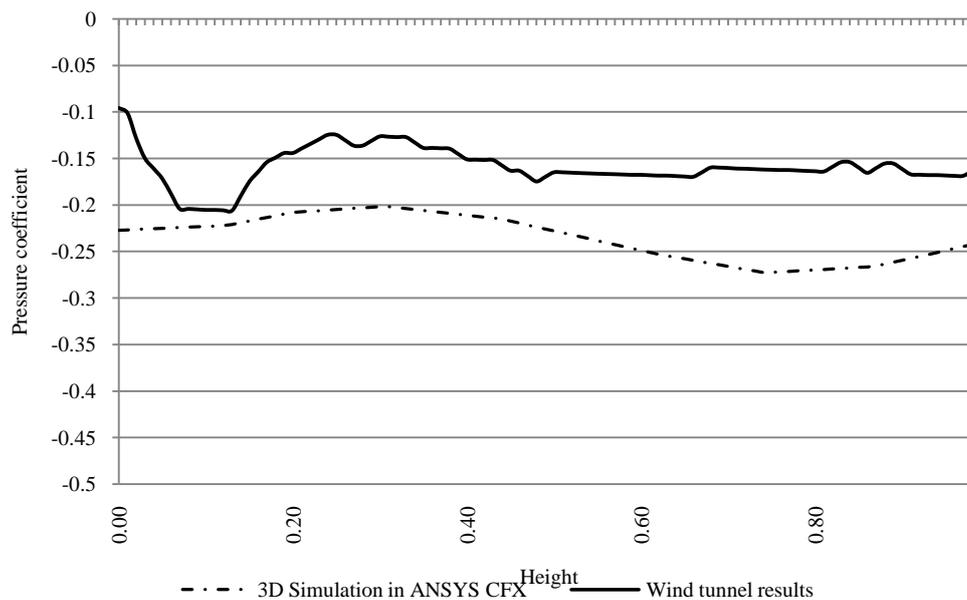
Vol. 4, Issue 6, June 2015

Figure 6.9 shows pressure coefficient distribution on the roof of model in 3d simulation in ANSYS CFX and wind tunnel testing results. On the windward edge there is a large suction in wind tunnel results due to flow separation and computations results shows equivalent suction at 20% length from the windward face. Results near windward wall can be refined by reducing mesh size.



**Fig 6.9** Comparison of Pressure coefficient distribution for roof on S. Murakami cube model

Figure 6.10 shows pressure coefficient distribution on the leeward wall of the cube. Computational results and wind tunnel results matches significantly on the leeward wall.



**Fig 6.10** Comparison of Pressure coefficient distribution for Leeward walls on S. Murakami cube model

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

## REFERENCES

- [1] S. Ahmad, M. Muzzammil, I. Zaheer, "Numerical prediction of wind loads on low buildings". International journal of Engineering, Science and Technology. Volume 3, No 5, 2011, pp. 59-72.
- [2] Anderson J. D. (1995).,"Computational Fluid Dynamics; The Basics With Applications"McGraw-HILL International Editions, McGraw-Hill Book Co., Singapore.
- [3] Biswas G. &Eswaran V. (2002), "Turbulent Flows-Fundamentals, Experiments and Modeling."Narosa Publishing House, Second Edition, New Delhi. First Edition.
- [4] Hoxey R.P. & Moran P. (1983)., "A Full-Scale Study of Geometric Parameters That Influence Wind Loads on Low-Rise Buildings",Journal of Wind Engineering and Industrial Aerodynamics, 13,277-288.
- [5] Satyen Ramani, P. N. Godbole, L. M. Gupta., "Application of CFD Determination of Pressure Coefficient Using ANSYS Flotran", Proceedings of 8<sup>th</sup> structural engineering convention, SVNIT, Surat.
- [6] Hoxey R.P., Robertson A.P., Basara B., Younis B. A. (1993).,"Geometric Parameters That Affect Wind Loads on Low-Rise Buildings: Full-Scale and CFD Experiments",Journal of Wind Engineering and Industrial Aerodynamics, 50, 243- 252.
- [7] Younis B.A., Oliveira P.J. (2000).,"On The Prediction of Turbulent Flows around Full-Scale Buildings",Journal of Wind Engineering and Industrial Aerodynamics, 86, 203-220.
- [8] S. Murakami, A. Michida, Y. Hayashi and S. Sakamoto., "Numerical Study on Velocity Pressure Fields and Wind Forces For Bluff Bodies By k- $\epsilon$ , ASM and LES"., Journal of Wind Engineering and Industrial Aerodynamics, 41-44 (1992) 2841-2852.
- [9] Hoxey R.P., Robertson A.P. (1994)., "Pressure Coefficients For Low-Rise Building Envelopes Derived From Full-Scale Experiments",Journal of Wind Engineering and Industrial Aerodynamics, 53, 283-297.
- [10] Hoxey R.P., Richards P.J. (1993)., "Flow Patterns and Pressure Field around a Full-Scale Building",Journal of Wind Engineering and Industrial Aerodynamics, 50, 203-212.
- [11] Satyen Ramani, P. N. Godbole, L. M. Gupta., "Application of CFD Determination of Pressure Coefficient Using ANSYS Flotran- A Parametric Study", National Conference on Wind Engineering, New Delhi, 2012.
- [12] Mathews E.H., Crosby C.P. Et Al. (1988)., "Numerical Prediction of Wind Loads on Buildings",Journal of Wind Engineering and Industrial Aerodynamics, 31,241- 250.
- [13] Y. Q. Zhang, A. H. Huber, S.P.S. Arya and W.H. Snyder., "Numerical Simulation To Determine The Effects of Incident Wind Shear and Turbulence Level on The Flow Around A Building",Journal of Wind Engineering and Industrial Aerodynamics, 46 & 47 (1993) 129-134.
- [14] S. Mandal, C.S.P. Ojha, P. Bhargava (2004)., "Wind Turbulence Modeling at Near wall Zone Using K-E Model: A Review", NCWE-2004.