Comparison of Numerical Prediction of Pressure Coefficient on Rectangular Small Building

[1]Neel M. Patel, [2]Satyen D. Ramani

[1]PG Student, Department of Civil Engineering, [2]Asst. Professor, Department of Civil Engineering,
[1,2]SAL Institute of Technology and Engineering Research, Ahmedabad, Gujarat, India

Abstract: In this study, 3D numerical simulation of rectangular small buildings having same plan area and height but different side ratio were carried out. The simulation was carried out in ANSYS CFX. For the simulation purpose, two different turbulence models (Standard K-Epsilon and Shear Stress Transport (SST)) are used. At the end, comparison of Pressure Coefficient (PCOE) results obtained from numerical simulation using CFD approach and the experimental results have presented. Also variation of turbulent kinetic energy obtained in numerical study is plotted for both turbulence models.

Keywords: CFD, 3D Numerical Simulation, ANSYS CFX, k-epsilon model, SST model, Pressure Coefficient, Rectangular Building

---

I. INTRODUCTION

Generally, wind load is considered in designing high rise buildings. Building designers usually calculate the wind load on building using IS codes. For some special cases, there is no codal provision to determine the wind load in that case experimental study is required to determine the wind load. But experimental tests (full scale study) are costly as well as time consuming. So in that case numerical simulation using CFD (Computational Fluid Dynamics) is very much helpful to determine the wind pressure and other quantities on any type of building.

During past decades so many Pressure distribution and wind responses of building are being investigated by many researchers through wind tunnel test and by numerical simulation. Satyen Ramani et al.[5] have applied computational method for the determination of Pressure coefficient using ANSYS Flotran. ANSYS Flotran simulation with standard k-epsilon model suggests the L2 (Distance between leeward face to outlet) = 12H and H (Total Domain Height) = 12H for better prediction of Pressure Coefficient. S. Murakami [6] used Numerical method to study velocity pressure flow field and wind generated forces around bluff body were studied by k-epsilon, ASM and LES turbulence model. For accuracy check, the comparison of the results for Pressure Coefficient with wind tunnel data was made. From the results it could be seen that LES turbulence model gives nearest results compared to wind tunnel. But LES needs high computational costs and time. P. J. Oliveira and B. A. Younis [9] studied the turbulent flow around full scale buildings. The tests on full scale, single span high eaves commercial glass house were used to quantify uncertainties with use of CFD to obtain wind load predictions. 2D flow field in mid length usually leads to over-estimation of suction pressure over the roof and leeward wall. The influence of grid size distribution on numerical error in appropriate grid distribution and reduction in peak pressure near ridge with grid refinement has been studied. For 2D simulation the domain size higher than 10H and upstream length greater than 15H appears to produce better result. 2D simulation predicts significant negative pressure but in 3D simulation, results are closer to experiment method. For 3D simulation, building should be placed at 3S(around 5H) from entry of domain and external boundary applied at distance of 5h from side and 4h from the roof. R.P. Hoxey and A. P. Robertson [7] have studied full-scale measurement of surface pressure on single span buildings. The full-scale measurements have indicated the importance of geometric parameters of building height, span and roof angle in assessment of wind load. By studying 13 different buildings the average pressure on windward is 0.43 which is insensitive to building geometry. But at some small areas of building pressure coefficient values goes up to 0.7 has been observed. Rocky Patel and Satyen Ramani [4] studied the optimum domain size required for 3D Numerical Simulation in ANSYS CFX. From observation, it can be seen that the side distance for the 3D domain size and distance between inlet to windward face should be 5h, optimum back distance between outlet to leeward face should be 15h and optimum height of 3D domain should be 10h. Where, h is the maximum height of structures on windward face.

II. TURBULENCE MODELS:

K-Epsilon (k- ε):

k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate).

The k - ε model introduces two new variables into the system of equations. The continuity equation is then:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j\right) = 0$$

And the momentum equation becomes:

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial \rho'}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] + S_M$$

Where, S_M is the sum of body forces, μ_{eff} is the effective viscosity accounting for turbulence, and ρ' is the modified pressure.

$$\rho' = p + \frac{2}{3}pk + \frac{2}{3}\mu_{eff}\frac{\partial U_k}{\partial x_k}$$

The k - ε model, like the zero equation model, is based on the eddy viscosity concept, so that:

$$\mu_{eff} = \mu + \mu_t$$

Where, μ_t is the turbulence viscosity. The k - ε model assumes that the turbulence viscosity is linked to the turbulence kinetic energy and dissipation via the relation:

$$\mu_t = C_\mu\, \rho \frac{k^2}{\varepsilon}$$

The values of k *and* ε *come directly from the differential transport equations for the turbulence kinetic energy and turbulence dissipation rate:*

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j k\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon + P_{kb}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j\, \varepsilon\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}(C_{\varepsilon 1}P_k - C_{\varepsilon 2}P_\varepsilon + C_{\varepsilon 1}P_{\varepsilon b})$$

Where,$C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k,σ_ε and C_μ are constants.

Shear Stress Transport Model (SST):

The k-ω based SST model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. The proper transport behavior can be obtained by a Limiter to the formulation of the eddy-viscosity:

$$V_t = \frac{\alpha_1 k}{(\max \alpha_1 \omega, SF_2)}$$

Where, $V_t = \frac{\mu_t}{\rho}$

$$F_1 = \tanh(arg_1^4)$$

With: $arg_1 = \min(\max\left(\frac{\sqrt{k}}{\beta' \omega y}, \frac{500\nu}{y^2\omega}\right), \frac{4\rho k}{CD_{kw}\ \sigma_{\omega 2}\, y^2})$

$$CD_{kw} = \max(2\rho \frac{1}{\sigma_{\omega 2 \omega}}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}, 1.0 * 10^{-10})$$

$$F_2 = \tanh(arg_2^2)$$

With: $arg_2 = \max\left(\frac{2\sqrt{k}}{\beta' \omega y}, \frac{500\nu}{y^2\omega}\right)$

Where, again F_2 is a blending function similar to F_1, which restricts the limiter to the wall boundary layer, as the underlying assumptions are not correct for free shear flow. S is an invariant measure of the strain rate.

III. Model Setup:

For simulation purpose, the domain size for ANSYS modeling is considered same as considered in literature [8]. Total length of domain is taken as 8.2 m long with the cross section of 1.2 m (width) × 0.85 m (height). Same height and area i.e. h=0.3m and a=10,000 mm^2 is considered for all the models. The models were placed at 6.1 m distance from inlet to measure the pressure coefficient on windward and leeward faces of the building. The schematic line diagram of model is shown in Fig. 1. All dimensions of building models are listed in Table 1 and input data is mentioned in Table 2. The plan and isometric views of models are shown in Fig. 2.

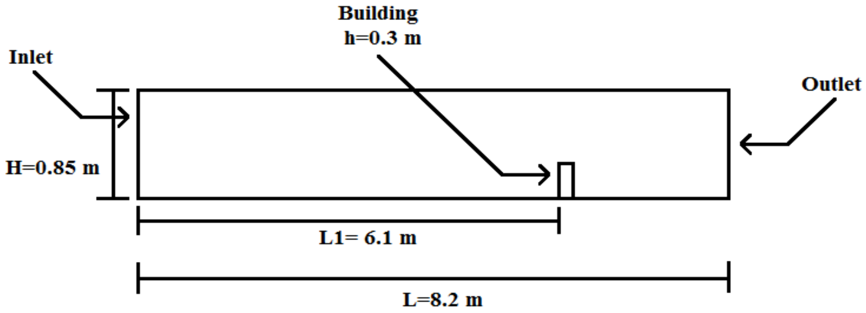
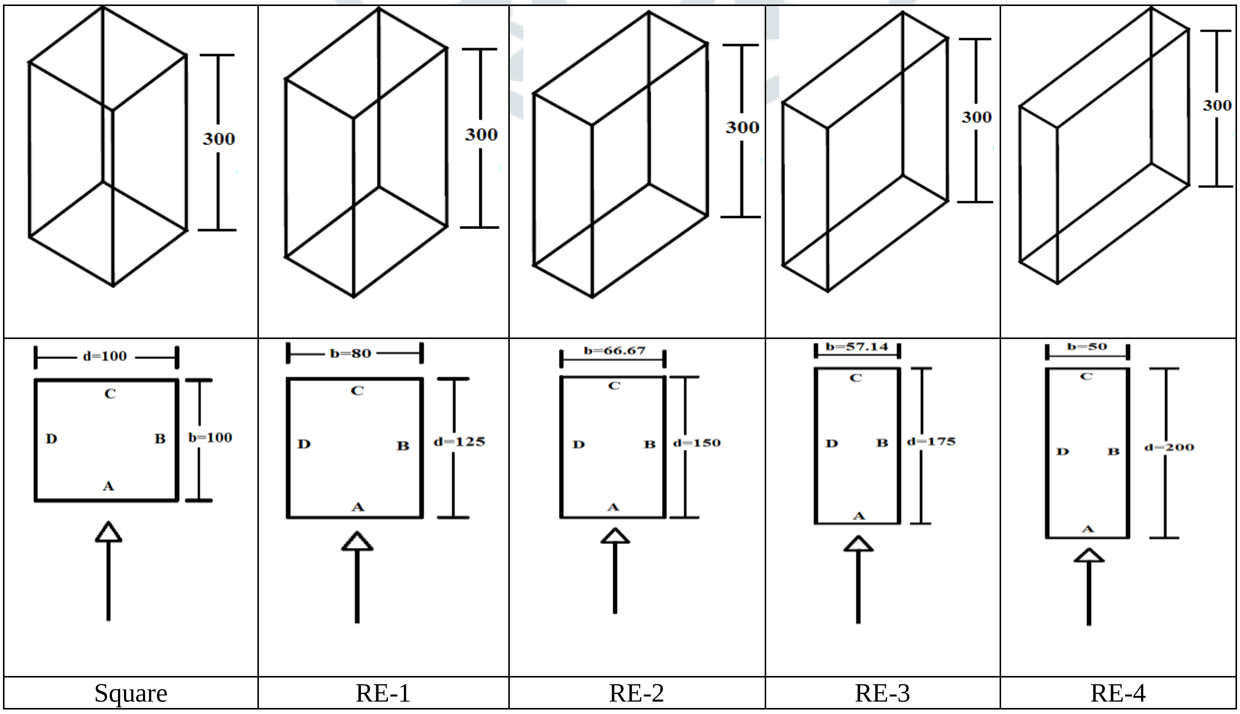


Fig. 1 Schematic Line Diagram of model

Table 1: The Dimensions of building models

Model	Width (mm)	Depth (mm)	Height (mm)
Square	100	100	300
Rectangular-1 (RE-1)	80	125	300
Rectangular-2 (RE-2)	66.67	150	300
Rectangular-3 (RE-3)	57.14	175	300
Rectangular-4 (RE-4)	50	200	300



Square RE-1 RE-2 RE-3 RE-4

Fig. 2 Plan and Isometric views of building models

Modelling of Atmospheric wind profile is done by Power law method. For modeling of wind flow characteristics applied power (n) of mean speed profile is 0.143. The free stream wind velocity at roof level of the building is maintained as 15 m/s. The simulated mean wind velocity profile is shown in Fig. 3. The equation for it is as follows.

$$V(y) = 15 \times \left(\frac{y}{H}\right)^{0.143}$$

Where, y = Height where the velocity measure
H = Reference height.

Normal and tangential velocity components are set to zero at solid boundary. All the walls of the building were chosen as no slip wall having roughness of 5 mm at the free stream velocity. For the study of pressure coefficient distribution around building, open terrain category is used.

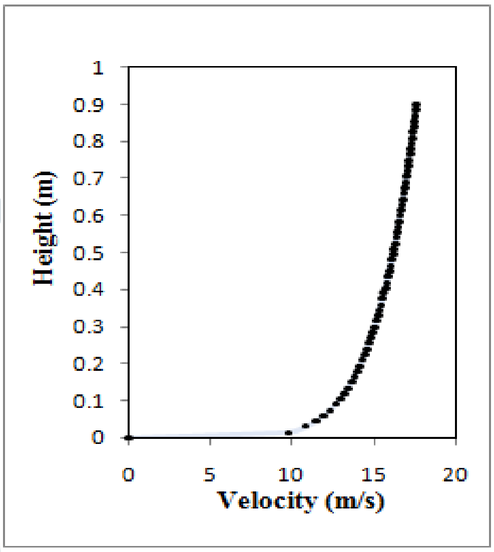


Fig. 3 Velocity Profile

Table 2: Input Data in ANSYS CFX

No.	Parameters	Value
1	Velocity Profile	$V(y) = 15 \times \left(\frac{y}{0.3}\right)^{0.143}$
2	Ground Roughness	5 mm
3	Turbulence Intensity at Inlet	5%
4	Density of Air	1.185 kg/m^3
5	Viscosity of Air	1.7594×10^5 kg/m. s
6	Turbulence Model	k- ε, SST

IV. PRESSURE DISTRIBUTION:

The elevated mean pressure at particular location of models is non-dimensionalized in the wind direction. The mean pressure coefficient along the wind direction is $1/2\rho v^2$, where ρ is the density of air which is 1.185 kg/m^3.

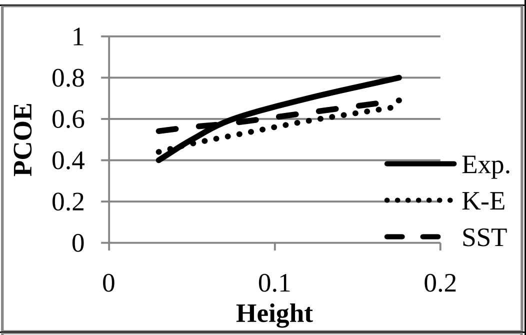
$$\text{Mean Pressure Coefficient} = \frac{\text{Mean Pressure}}{1/2\rho v^2}$$

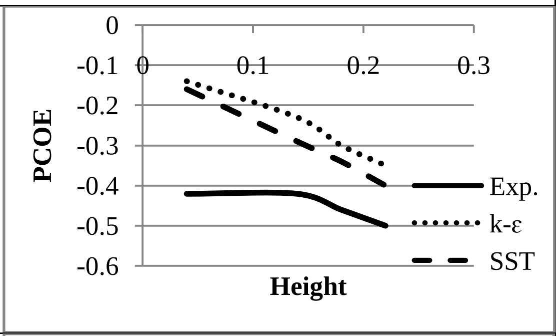
The general characteristics and effect of pressure distribution was measured at mid length of the building on Windward and Leeward faces of Square and Rectangular building models at wind incidence of 0°.

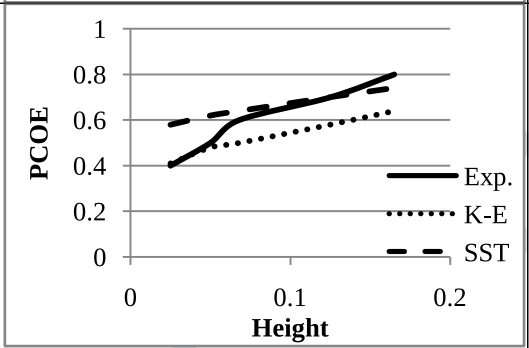
V. RESULTS:

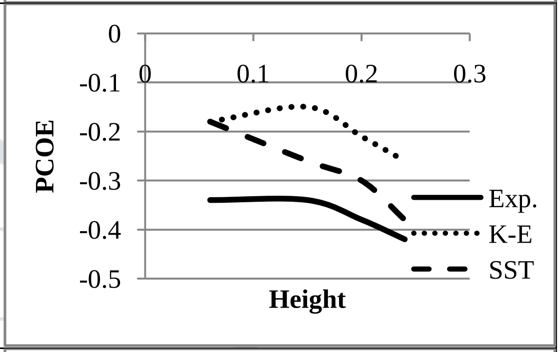
The comparison of pressure coefficient(PCOE) between Numerical simulation in ANSYS CFX using two turbulence models (k-ε and SST) with the given experimental data [8] are listed in Table 3.

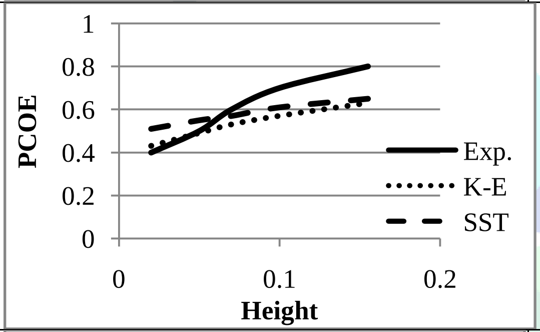
Table 3: Comparison of Pressure Coefficient at mid Plane of building

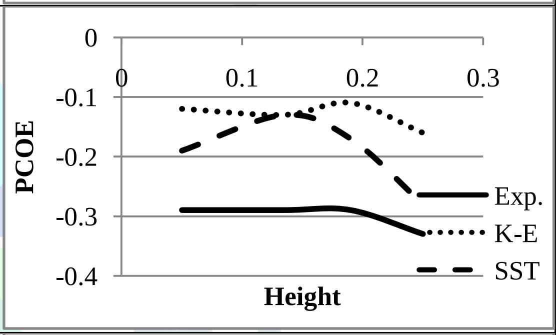
<table>
<tr><th>Model</th><th>Windward Face</th><th>Leeward Face</th></tr>
<tr><td>Square</td><td>


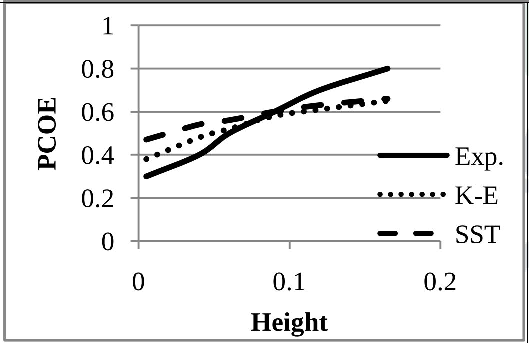
</td><td>


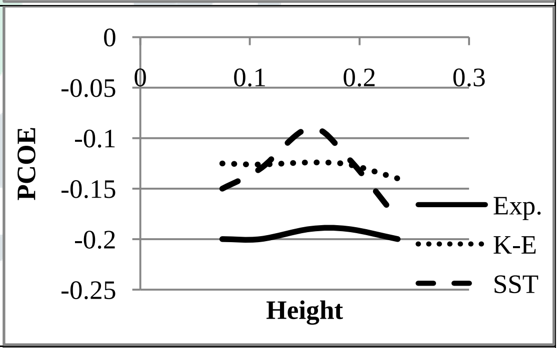
</td></tr>
<tr><td>RE-1</td><td>


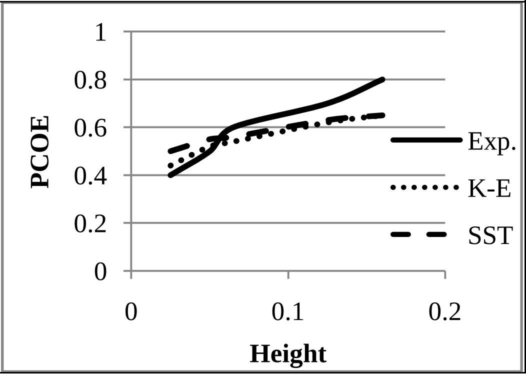
</td><td>


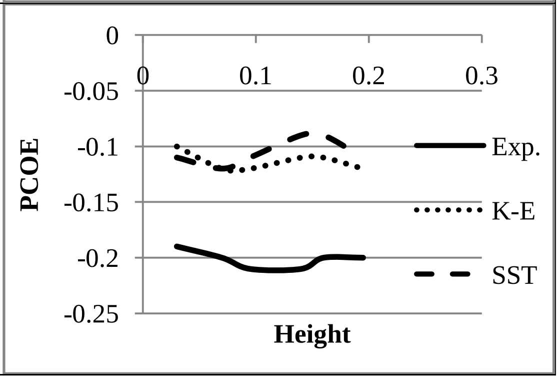
</td></tr>
<tr><td>RE-2</td><td>


</td><td>


</td></tr>
<tr><td>RE-3</td><td>


</td><td>


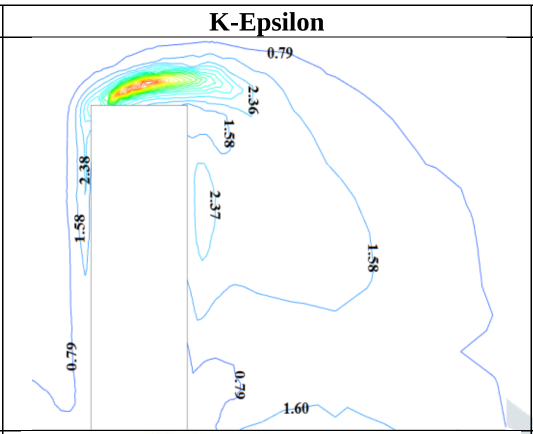
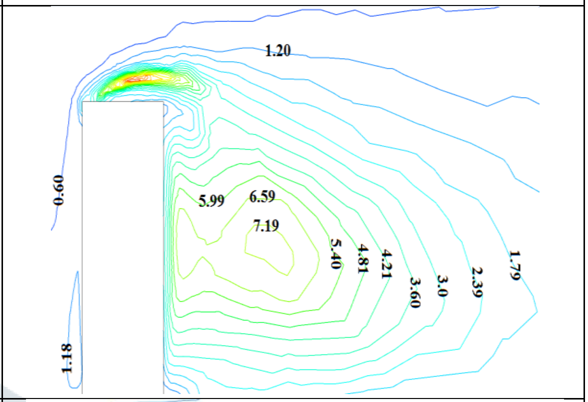
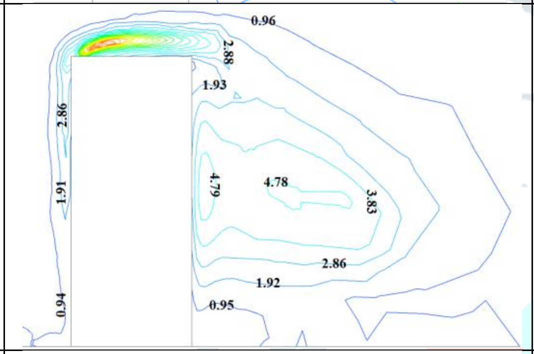
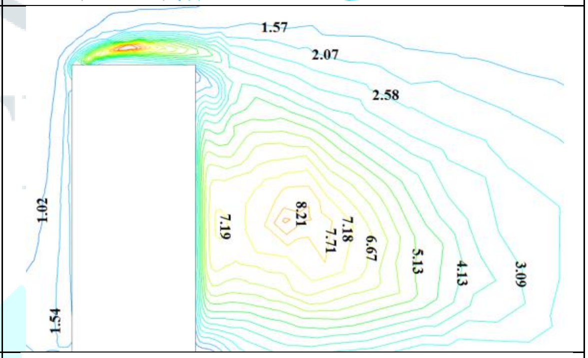
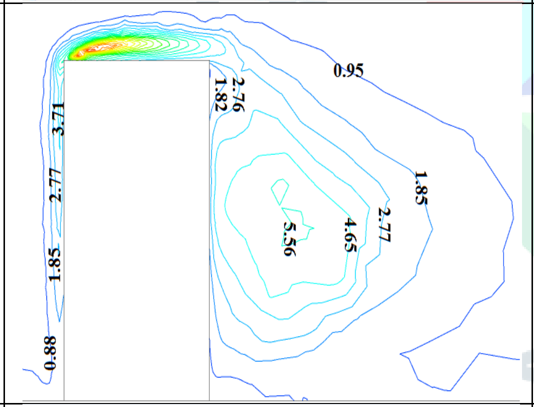
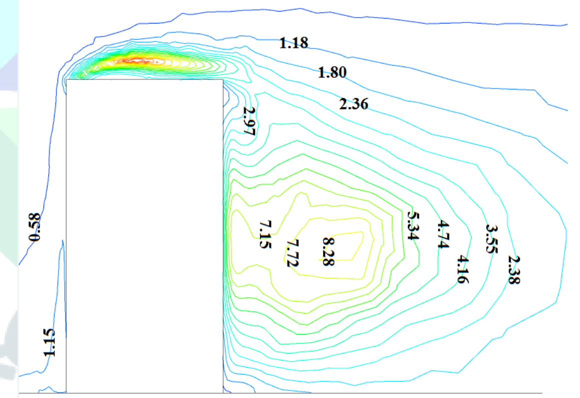
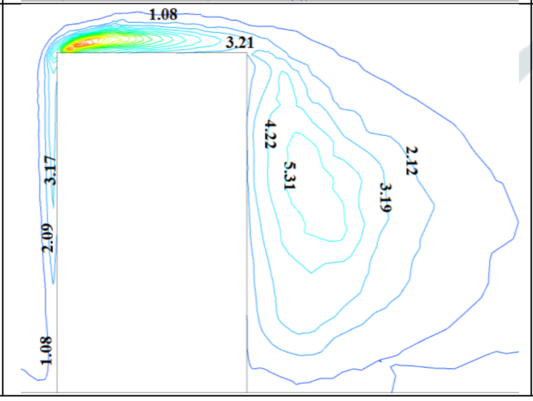
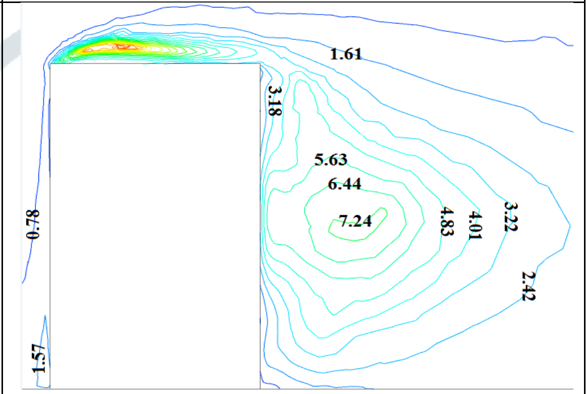
</td></tr>
<tr><td>RE-4</td><td>


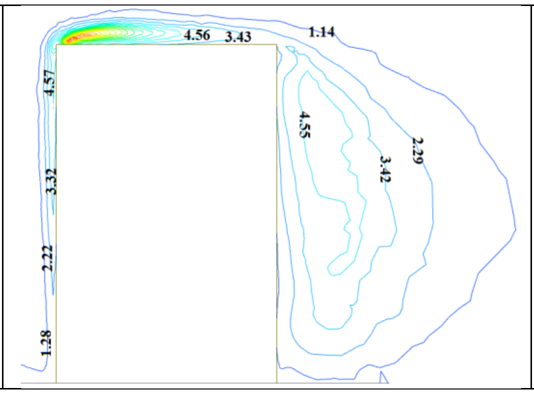
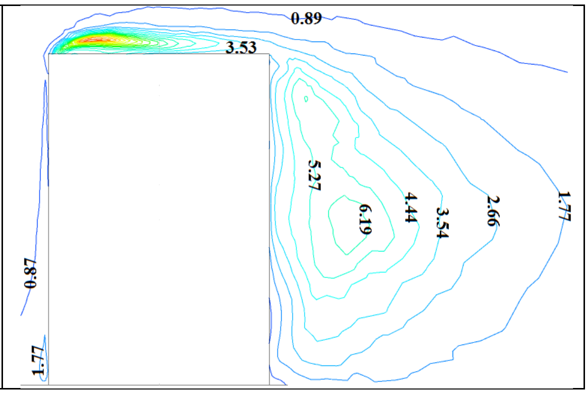
</td><td>


</td></tr>
</table>

The turbulence kinetic energy for two different turbulence models (k-ε and SST) at mid plane of the building is given in Table 4.

Table: 4 Comparison of Turbulence Kinetic Energy at mid plane of Building

Model	K-Epsilon	SST
Square		
RE-1		
RE-2		
RE-3		

RE-4		

VI. CONCLUSION

From the above graph, it can be seen that Shear Stress Transport (SST) turbulence model gives more accurate prediction of pressure coefficient compared to the Standard k - ε. The SST model combines advantage of both standard k-ε and k-ω model. SST model gives highly accurate prediction of onset and amount of flow separation under adverse Pressure gradients by inclusion of transport effect into formation of eddy viscosity. Mainly SST model is recommended for high accuracy boundary layer simulation.

From Table 3, it is clear that the results of PCOE on Windward obtained from CFX simulation are much consistent with experiment values regardless of the type of turbulence model. However, on the Leeward side, there is a significant difference between the experimental results and simulation result. Further, this difference is major for the Rectangular type buildings RE-3 and RE-4. This can be explained and visualized from the difference of blockage area perpendicular to incident wind. Compared to other models, these two models have lesser width normal to the wind direction. Being height constant, these results in lesser vertical blockage area at their windward face was estimate. It is also observed by authors in their past study and also supported by other literature that the simplified turbulence models may fail to predict suction on Leeward for these type buildings. The turbulent kinetic energy distribution around the corners may help to further investigate for the reasons behind underestimated prediction of suction or higher prediction in case of experimental study.

VII. REFERENCES

[1] Anderson J. D. (1995).*Computational Fluid Dynamics; The Basics With Applications*, McGraw-HILL International Editions, McGraw-Hill Book Co., Singapore.

[2] Ansys 13.0, Theory Refrence manual

[3] Indian Standard Code of Practice for Wind load i.e. IS – 875, Part-3

[4] Rocky Patel, Satyen Ramani (2015). “Determination of Optimum Domain Size for 3D Numerical Simulation in ANSYS CFX”

[5] Satyen Ramani, P. N. Godbole, L. M. Gupta, “Application of CFD determination of Pressure Coefficient Using ANSYA Flotran” Proceeding of 8th Structural Engineering Convocation, SVNIT, Surat.

[6] S. Murakami, A. Michida, Y. Hayashi and S. Sakamoto. “Numerical Study on Velocity Pressure Fields and Wind Forces for Bluff Bodies By k-ε, ASM and LES”. Journal of Wind Engineering and Industrial Aerodynamics, 41-44 (1992) 2841-2852.

[7] 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.

[8] J.A. Amin, A.K. Ahuja (2014) “Characteristics of wind forces and responses of rectangular tall buildings” *Int J Adv Struct Eng (2014) 6:66.*

[9] 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.