Numerical Study of Low Rise Building in ANSYS Fluent
Ekta N Soni1 Satyen Ramani2
1 P.G Scholar 2 Assistant Professor
1,2 Department of Civil Engineering

SAL Institute of Technology & Engineering Research, Ahmedabad, Gujarat, India

Abstract—2-D numerical simulation of wind loads on low-rise building has been carried out. Numerical simulation is carried out on TTU building (13.7m x 9.1m in plan with 4.0m eave height flat roof) to check effect of influence of turbulence models used in ANSYS FLUENT software package. Also validation is done with available literature to check the results of ANSYS FLUENT software. The present study has a goal to explain the application of CFD to find wind generated pressure distribution on a particular structure using ANSYS FLUENT 13.0. Numerical simulation on flat roof (TTU building) is carried out to obtain wind loading effects. Results obtained from present study are validated with past data available. The numerical results shows a good agreement with the available data. The computed pressure coefficients have been validated with past numerical results of TTU building model.

Key words: low-rise buildings, wind loads, numerical simulation, gable roof, TTU building

I. INTRODUCTION

For a safe structural design of any particular structure it is necessary to consider wind effects. Although one cannot see the wind, it is a common observation that its flow is quite complex and turbulent in nature and has a tendency to exert differential velocity and pressure field around any obstacle likely to obstruct its flow path. The generalized estimation of wind loading is carried out by defining pressure coefficients. Pressure coefficients are non-dimensional parameter which is used to assess magnitude. Pressure coefficients are influenced by various parameters like, shape, structural geometry, incident wind profile, terrain roughness, turbulence in the wind, location of a particular structure etc. To evaluate these pressure coefficients, two basic methods are there. The first is experimental method in which structure is instrumented to record actual values called full-scale study or a model for prototype structure is investigated in wind-tunnel which is called wind-tunnel testing. The second method is theoretical, in which relevant country codes and standards are referred or a latest method called Computational Fluid Dynamics is used to carry out estimation.

Computational Wind Engineering (CWE) is the application of CFD to wind engineering problems either in 2-D or 3-D manner to study effects. Many problems in wind engineering can be handled by any of the three approaches, or a combination of these: (1) on-site measurements, (2) reduced-scale wind tunnel testing or (3) numerical simulation using CFD. The major focus of CWE branch is to find pressure coefficients around a 2-D or 3-D bluff body. The loading is the result of the pressure distribution on the structure in general. Simulation-based design instead of “build & test” is more cost effectively and more rapidly than experiments. Moreover CFD solution provides high-fidelity database for interrogation of flow field. Many researchers have made remarkable contributions in the field of computational wind engineering in the past.

P.J.Oliveira et.al (2000) [6] used the data obtained from tests on full-scale, single-span high eaves commercial glasshouse to quantify the uncertainties associated with the use of computational fluid dynamics to obtain wind load predictions for full-scale structures. It was found that performing the computations with the assumption of two-dimensional flow along mid-length plane leads to a serious overestimates of the roof suction loads. It is further shown that the use of a Reynolds-stress closure enables the prediction of flow separation on the windward side of the roof, with consequent changes in the wall static-pressure distribution. The standard k-ε model is found to predict no flow separation, contrary to experimental observation. Guidelines are suggested for suitable mesh distribution and for the efficient sizing of the computational domain relative to the building’s dimension.

Satyen Ramani et.al (2012)[3] made an attempt to demonstrate application of Computational Fluid Dynamics (CFD) technique to obtain pressure coefficients on a single roof gabled like closed structure subjected to atmospheric mean wind using ANASYS-Flotron module. Computational results are absolutely sensitive to various parameters like meshing size & patterns, application of boundary conditions, turbulence models, domain size used to model flow environment like height of domain, upstream length L1(distance between inlet plane to windward face), downstream length L2 (distance between leeward face to outlet plane).It was concluded that the guidelines given by Oliveira at al. for the meshing is also applicable to FEM based ANSYS Flotron simulations with some modification.

Satyen Ramani et.al (2012) [4] carried out a parametric study to present the influence of geometrical parameters like fluid domain sizes on pressure coefficient prediction.In this study, PCOE variations on WW, LW & roof due to changes in domain dimensions like L1, L2 and Y was studied.The quality of the result was measured against the requirement of any relaxation used in simulation analysis. In general, for ANSYS FLOTTRAN simulation with standard k-ε model L2=12H and Y=12H was suggested for better prediction of Pressure coefficient.

E.H. Mathews (1987) [5] used computational methods to predict wind-generated pressure distribution around a tall building & over a long building with 2-D simulation and the accuracy of the predicted pressure distribution was found to be acceptable. It was analyzed that the wind-generated pressure distributions around buildings can be predicted by numerical techniques. The author remarked that even the complex flows around complex building can be investigated efficiently by numerical technique for wind pressure prediction.

S.Murakami et.al (1992) analyzed the velocity-pressure fields and wind-induced forces on a building model with the help of numerical simulation. Three well-known
turbulence models which are k-ε Eddy Viscosity Model (k-ε EVM), Algebraic stress Model(ASM) and Large Eddy Simulation (LES) were used. Comparison of 2D and 3D computations were carried out with experimental data. The results of LES agree very well with experimental data in terms of the mean velocity distribution, mean surface pressure and turbulence statistics of velocity and pressure. It was also concluded that the results of k-ε EVM include serious discrepancies from experimental data, which are improved in ASM model. The results of 2D computation are different from 3D computations and experiment, 2D computation thus fails to reproduce unsteady surface pressure field and wind pressure forces on a building which is influenced by the unsteady 3D flow structures around a building.

S.Ahmad et.al (2011) did 2-D numerical simulation on flat roof (TTU building) and pitched roof in ANSYS FLUENT package and compared the obtained results with Cocharan (1992) in a wind tunnel testing for flat roof TTU building model. For pitched roof of 10°, 20° and 30° gable roof, the obtained results have been compared with Mathews et al (1988) on gable structures for 15°, 26° and 35° roof slopes.

Yongsheng Zhou et.al (1997) A two-layer method combining the k-ε model in fully turbulent regions with a one-equation model in near wall area has been tested in computing the wind conditions around a cubic building. Results were compared with those from k-ε model computation as well as from experiments. In contrast with the usual k-ε model approach, this method has been effective in predicting the flow separation above the roof surface and near the side walls of a cubic building. The numerical prediction of the wind-induced pressure on building surfaces, especially on roof and side walls, has also been improved as compared with the k-ε model method. The two-layer method is also superior compared with the k-ε model method under oblique wind conditions. The limitation of the two-layer model should also be noted. For example, the thickness of the inner layer in which the one-equation model can be valid might be too small for the separation areas near the leading corners on roof and beside the side walls.

Tedd Stathopoulos et.al (1995) presented the results of an investigation regarding the numerical evaluation of wind pressures on flat roofs by using the Reynolds averaged Navier-Stokes equations and the standard k-ε turbulence model. A low and a taller building model were considered and wind blowing from different directions was assumed. Data were presented for roof edges and corners. Experimental measurements have been carried out in a boundary layer wind tunnel and results were compared with the computed data. Over most of the roof surface area, the comparisons were good for normal wind conditions and the low building model. However, the present approach does not seem to yield adequate representations of available experimental data in the high vorticity region near the windward roof edges of the building, particularly for oblique wind conditions. Conclusions are made that the wind flow over the roof of a rectangular building is very complex since it involves high turbulence, severe pressure gradients, separation and possible reattachment. This makes the numerical modeling and evaluation of wind-induced pressures on roofs very difficult.

II. PRESENT STUDY

A. Turbulence Model Influence Study:

ANSYS FLUENT has one-equation models and two-equation models which are RANS based models and other two are Detached Eddy Simulation and Large Eddy Simulation. One-equation model is Spalart-Allmaras model. Two-equation models includes standard k-ε model, RNG k-ε model, Realizable k-ε model, standard k-ω model, SST k-ω model and Reynolds stress model.

In this study, TTU building (13.7m x 9.1m in plan with 4.0m eave height flat roof) model have been selected for numerical simulation for wind load prediction using four different turbulence models which are standard k-ε model, RNG k-ε model, standard k-ω model and transition SST model. Mesh with refinement is used for all models.

B. Validation Study:

TTU building model have been selected for numerical simulation for wind load prediction. To validate this model ANSYS FLUENT 13.0 software package was used. The FLUENT package is based on the partial differential equations that governs the movement of viscous fluid i.e. the Navier-Stokes equations (1) and the continuity equation (2).

\[ \rho \frac{\partial V}{\partial t} = -\nabla P + \rho \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \]

\[ \nabla V = 0 \]

Where, \( \rho \) is density, D/Dt is the substantial derivative, \( P \) is the pressure \( V \) is the velocity vector and \( \mu \) is the effective viscosity. The k-ε turbulence viscosity model is used for the simulation of turbulence in the flow. Values for k (turbulence kinetic energy) and ε (dissipation rate of turbulence) were obtained from elliptical partial differential equations.

C. Boundary Conditions:

The velocity profile at inflow is imposed using UDF (user defined function) in FLUENT. k-ε Turbulence viscosity model is used for numerical simulation. ks is a surface roughness parameter which is a measure of the surface roughness coefficient of the terrain kept 1.5 cm. The power law was used to simulate the inflow u- velocity profile of the atmospheric boundary layer. Inflow values of \( u\)- velocities are set to zero.

Boundary condition at outlet is 1 atm pressure and both normal and tangential velocity values are set to zero at solid boundaries. Sides of domain are symmetry. At top and bottom walls of domain are no slip walls.

In the present study, refinement has been carried out for meshing which is shown in Fig.3. A flow domain having boundary (front or windward wall) at the distance of 20m from inlet wall and outlet wall is at 60 m distance from the rear wall (leeward wall) also the height of the domain is kept 40 m. This domain size is used for solution of the TTU building has been shown respectively in Fig.1. The summary of the data required for computation through ANSYS FLUENT is shown in Table-1. Summary of data used in FLUENT is shown in Table-2. The results obtained in the present study have been compared with the numerical
results of S.Ahmad et al. (2011) and wind tunnel data of Cocharan (1992) having results on model of TTU building.

### Setup

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>General</td>
<td>Pressure-based solver</td>
</tr>
<tr>
<td></td>
<td>Steady time</td>
</tr>
<tr>
<td>Models</td>
<td>Viscous-Standard $k-\varepsilon$ model with standard wall functions</td>
</tr>
<tr>
<td>Materials</td>
<td>Fluid-air</td>
</tr>
<tr>
<td></td>
<td>Density- 1.225 kg/m$^3$</td>
</tr>
<tr>
<td></td>
<td>Viscosity-1.7894e-5 kg/m-sec</td>
</tr>
<tr>
<td>Boundary condition</td>
<td>Inlet- velocity inlet-turbulence intensity-5%</td>
</tr>
<tr>
<td></td>
<td>Wall roughness-1.5cm with no slip</td>
</tr>
<tr>
<td></td>
<td>Outlet- Pressure outlet</td>
</tr>
</tbody>
</table>

### Solution

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solution Methods</td>
<td>SIMPLE pressure velocity coupling</td>
</tr>
<tr>
<td></td>
<td>Power law discretization scheme</td>
</tr>
<tr>
<td>Iterate</td>
<td>Number of iteration-600</td>
</tr>
<tr>
<td></td>
<td>Profile update interval-0.001 sec</td>
</tr>
<tr>
<td></td>
<td>Reporting interval-40</td>
</tr>
</tbody>
</table>

### Table 1: Table For Computation through Fluent

<table>
<thead>
<tr>
<th>S.N</th>
<th>Parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Velocity Profile</td>
<td>$10.7 \times \left(\frac{y}{4.0}\right)^{0.14}$</td>
</tr>
<tr>
<td>2</td>
<td>Mean Wind Speed(m/s)</td>
<td>10.7</td>
</tr>
<tr>
<td>3</td>
<td>Power Law Coefficient, $\alpha$</td>
<td>0.14</td>
</tr>
<tr>
<td>4</td>
<td>Turbulence Intensity (%)</td>
<td>5</td>
</tr>
<tr>
<td>5</td>
<td>Roughness Length,$\ell_0$(cm)</td>
<td>1.5</td>
</tr>
<tr>
<td>6</td>
<td>Density of air(kg/m$^3$)</td>
<td>1.225</td>
</tr>
<tr>
<td>7</td>
<td>Viscosity of air(kg/m-sec)</td>
<td>$1.7594\times10^{-5}$</td>
</tr>
</tbody>
</table>

### Table 2: Summary of Data Used In the Fluent

### III. RESULTS AND DISCUSSIONS

#### A. Turbulence Model Influence Study:

Fig. 6, 7 and 8 compares the different turbulence models in Fluent. There is considerable difference between the 4 models predictions especially for roofs as in Fig. 8. The standard $k-\varepsilon$ model is a model based on model transport equations for the turbulence kinetic energy ($k$) and its dissipation rate ($\varepsilon$). The standard $k-\varepsilon$ model is valid for only fully turbulent flows. The Fig. 8. shows that the widely used $k-\varepsilon$ model is found to predict no flow separation. The rate of production of $k$ seems always to be exaggerated, leading to higher than expected levels of turbulence kinetic energy. Thus, when used for flat-roofed buildings, the standard $k-\varepsilon$ model either underestimates the extent of the region of the large-scale separation that occurs over the windward roof or fails to predict its occurrence altogether. In fluent, to improve the performance of $k-\varepsilon$ model, two variants are available called the RNG $k-\varepsilon$ model and the realizable $k-\varepsilon$ model. The RNG model has an additional term in its $\varepsilon$-equation that improves the accuracy for rapidly strained flows. Also the effect of swirl on turbulence is included in RNG model, enhancing the accuracy for swirling flows. These features make the RNG $k-\varepsilon$ model more accurate and reliable for a wider of flows than the standard $k-\varepsilon$ model.
Numerical Study of Low Rise Building in ANSYS Fluent

Fig. 6: Comparison of pressure coefficients for ww wall for TTU building for different turbulence models in ANSYS Fluent.

Fig. 7: Comparison of pressure coefficients for lw wall for TTU building for different turbulence models in ANSYS Fluent.

Fig. 8: Comparison of pressure coefficients for roof for TTU building for different turbulence models in ANSYS Fluent.

B. Validation:

The pressure coefficients obtained in the present study for the TTU building model have been compared with the pressure coefficients obtained by Cochran (1992) in a wind tunnel testing on TTU building model on and with S.Ahmad (2011) in numerical simulation on TTU building model in FLUENT package.

The three faces of the building are termed as windward wall, roof, and leeward wall. A graphical comparison has been made between the pressure coefficients on the edge of all these faces throughout its length. The pressure coefficients follow a similar trend with the Cochran and S.Ahmad (2011) results at the windward wall, leeward wall and roof which have been shown in Fig.9, 10 and 11. The results obtained in the present study are matching with an average variation of 13%, 39% and 30% respectively with S.Ahmad results and with an average variation of 16%, 12% and 3% respectively with Cochran (1992) results.

Moreover, the computational requirements are very much high for 3D compare to 2D. Also domain size and meshing are different so it will affect the results. In the present study, there is large variation of results at roof and at corner points. Also in the wind tunnel testing the peak pressure coefficients at the corner point do not match with the TTU Field value. The probable difference between Cochran and present study would be mostly due to (i) the steady (mean) wind modeling used in ANSYS. (ii) Further Cochran has not clearly mentioned the wind characteristics of Boundary layer flow generated by wind tunnel.
IV. CONCLUSIONS
The results obtained by the present study shows that the 2-D simulation using k-\(\varepsilon\) model for TTU building are matching fairly well (except at corner points) with the results of wind tunnel data. The average variation in results between present study and wind tunnel testing results of Cochran on windward wall, leeward wall and roof of TTU building model are 16\%, 12\% and 3\% respectively. The results obtained in the present study are matching with an average variation of 13\%, 39\% and 30\% respectively with S.Ahmad results. S.Ahmad result would arise from different type of discretization scheme, turbulence models used or due to non-similarity of inputted wind characteristics.

V. NOMENCLATURE
1) TTU Texas Tech University
2) 2D Two-dimensional
3) 3D Three-dimensional
4) CFD Computational Fluid Dynamics
5) \(\rho\) Density
6) \(V\) Velocity Vector
7) \(D/\text{Dt}\) Substantial Derivative
8) \(P\) Pressure
9) \(\mu\) Effective Viscosity
10) \(k_s\) Surface Roughness Parameter
11) \(\alpha\) Power Law Coefficient
12) \(C_p\) peak Pressure Coefficient (Peak)

REFERENCES