Peak pressure computation on low rise building using inflow turbulence generator

R. Panneer selvam [1]

[1]*University of Arkansas, Fayetteville, AR 72701, USA, email: rps@uark.edu*

SUMMARY:
A well validated computational fluid dynamics model (CFD) provides confidence in using it for practical problems. One of the major challenges is the computation of peak pressure using CFD. Here CFD peak pressures are computed using inflow turbulence generator. The peak pressure coefficients (C_p) at the building corner for 45-degree flow angle are reported with inflow turbulence generator based on synthetic eddy method. The building considered is Texas Tech University low rise building. Eight different grid configurations are considered for flow computation. The grids are developed using multilevel grids to save storage space and computer time. Three to four multilevel are considered for flow computation. The opensource software openFOAM is used for computation. The computed C_p are in reasonable comparison with wind tunnel measurements for 1:50 scale testing and with a maximum error of 43% for 1:6 scale testing. Flow visualizations are used to show the conical vortex development. Further works are identified for improvements.

Keywords: computational fluid dynamics, inflow turbulence, large eddy simulation

1. INTRODUCTION

To reduce building damages, understanding the range of peak pressure on building is important. The current code regulations are based on wind tunnel (WT) testing. The extreme peak pressure coefficients (C_p) for the Texas Tech building (TTU) from field measurements and wind tunnel (1:6 scale) were reported to be -16 and -8 at the corner of the building by Mooneghi et al. (2016). The mentioned WT measurements are for 1:6 scale and other WT testing reported in the literature are for smaller size building. Hence there is a challenge in measuring accurate C_p from wind tunnel. Other alternative technology is computational fluid dynamics (CFD). If CFD is used with inflow turbulence generators, it may be a viable tool. Recent years Selvam and his group (Selvam, 2022, Mansouri et al. 2022a & b) investigated eleven different inflow turbulence generators for wind engineering applications. They computed peak C_p for flow along the short side of the TTU building and they compared with 1:6 scale WT measurements for certain inflow turbulence generators. Of the eleven different inflow generators tried, many methods introduced spurious pressures or/and lost energy at the building location comparing to the inflow location. They selected synthetic eddy method with Gaussian shape function as an optimum method to use. The computed C_p minimum along the centreline of the building is shown in Figure 1 along with WT measurements. They compared well with WT measurements. In the figure WT6 refers to 1:6 scale wind tunnel results from Florida International University (FIU). The details of input parameters, mesh details and other computational details considered are reported in Mansouri et al. (2022a). A current state of peak pressure computation will be reviewed in the presentation.

The next step is further validation of peak pressures with WT in other flow angles. This will pave the way for computing peak pressures for field measurement set up in the future. From the literature it is found that only handful of researchers reported peak pressure at the corner and a review will be provided in the full paper. One of the major challenges for other flow angles is grid generation. When orthogonal meshes are used for the computation of flow along the shorter side, a grid spacing of H/16 or below computed the C_p minimum in par with WT measurements as reported in Mansouri et al. (2022). For corner peak pressures there is no clear guidelines on what level of minimum grid spacing is needed. Hopefully this work will address some of those issues. When the flow is non-orthogonal to the sides of the building, orthogonal grid cannot be used. On the other hand, the inflow turbulence is generated at the inflow assuming the mean flow is along the x-direction and the grid used to be orthogonal. The orthogonal grid provides high accuracy in transporting the turbulence. Hence different grid orientation has to be merged properly as shown in Figure 2. In Figure 2, multilevel grid generated using OpenFOAM CFD software is shown in Figure 2a and a combination of nested and multilevel grid is shown Figure 2b. The performance of this type of grid and the level of accuracy of C_p at the critical locations with reasonable computing power is investigated in this work. When two level grids are merged, the amount of error introduced in merging different grid spacing (h and 2h) is not known clearly. As an initial investigation the C_p at the corner of the roof is compared with FIU-WT measurements for few different grid configurations.

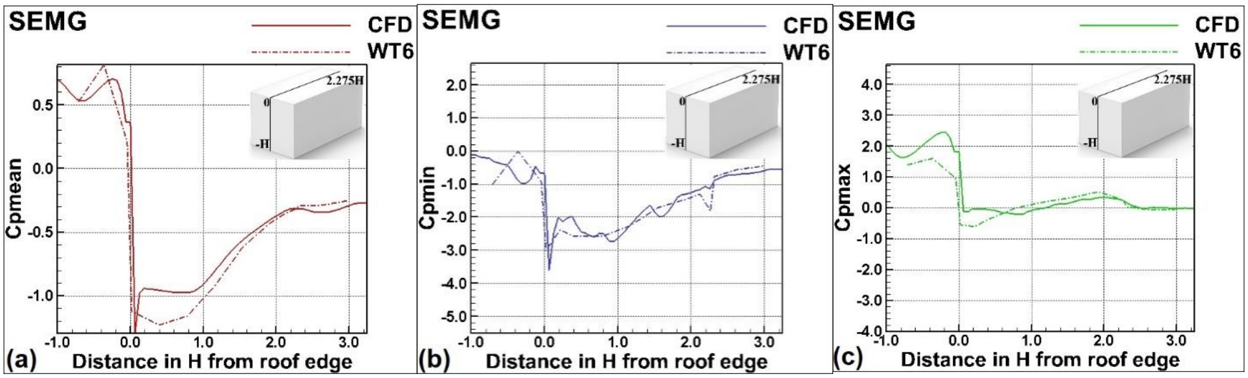

Figure 1. Comparison of CFD pressure coefficients with WT (1:6 scale) measurements for flow along the short side.

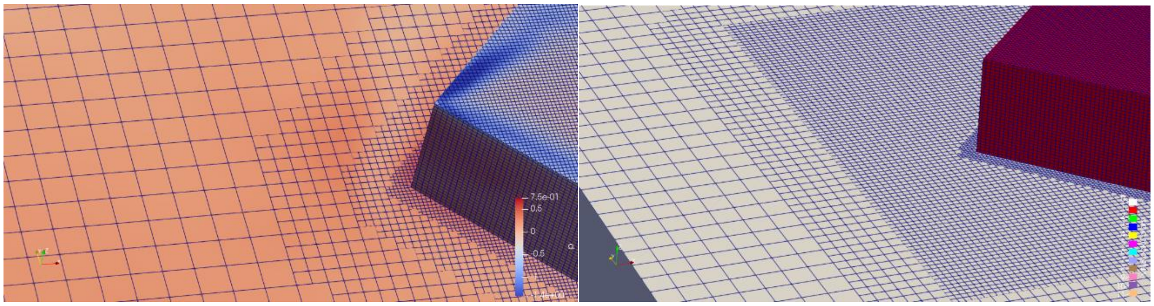

Figure 2. CFD grid (a) multilevel grid and (b) multilevel and nested grid option

2. RESULTS

For this study, a computational region of 13.75Hx10Hx5H is considered and the building is kept for 45 angular flow as shown in Figure 2. Here H is the height of the building. Nine different grids are tried so far by varying the level of refinements, h_{max} and h_{min} as reported in Table 1. Here h is the grid spacing. For comparison of CFD to WT negative peak C_p coefficients, only A1 and A2 points shown in Figure 3a are considered. The inflow turbulence generator parameters considered for synthetic eddy method (SEM) are the same as reported in Mansouri et al. (2022). The CFL condition is kept less than one and the computation is done for a non-dimensional time of 50-time units. The pisoFoam solver is selected for solving the Navier-Stokes equations (NS) equations. Few runs (h_{max}=H/8 and H/16) diverged before it reaches the specified maximum time to run. From the time varying velocity plot at the inlet, it is found that for these diverged runs, the maximum amplitude from the mean ranged from 70% to 90%. These details will be provided in the conference. More detailed analysis is needed why the jobs diverged for H/8 & H/16 grid spacing far away. Some runs were done using serial computing, and some runs were done using parallel computing with 4 processors. Further runs are underway, and the improved results will be reported in the full paper and in the conference.

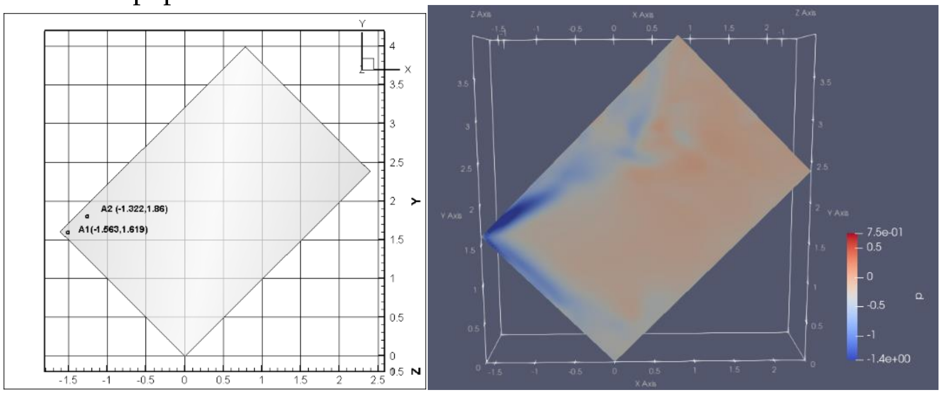

Figure 3. (a) Corner points A1 to A2 are considered for CFD and WT comparison. (b) Pressure contour plot on the roof

Table 1. Computed CFD peak negative C_p for A1 and A2 points, used multilevel and grid details

Multi levels	h_{max}	h_{min}	C_{pmin} @ A1	C_{pmin} @ A2
1	H/16	H/32	-4.54	-3.46
3-run1	H/4	H/32	-3.38	-4.88
3-run2	H/4	H/32	-3.22	-3.14
3-run3	H/4	H/32	-2.78	-3.14
3-run4	H/4	H/32	-2.54	-3.21
3	H/8	H/64	-3.21	-3.47
4-run1	H/4	H/64	-4.4	-4.12
4-run2	H/4	H/64	-3.59	-3.824

The roof pressure contour plot in Figure 3b shows the roof corner pressure developments and the orientation of the building with respect to x-axis of the computational domain. Figure 4 shows the conical vortex developments at time 40 and 50 units. The inflow turbulence at the inflow is different at each computer run. The reported peak C_p in Table 1 are from 10s to 50s of the computer output. From the computer runs, we could say that the highest negative C_p from CFD is -4.54 for point A1 and -4.88 for point A2. Tieleman et al. (1996) reports C_p in the range of -4 to -8 for WT testing of 1:50 to 1:100. Mooneghi et al. (2016) reports a value of -8 for point A1 and -6 for point A2. For the 1:50 scale WT testing, the computed C_p are in good agreement but for 1:6 scale testing, there is an error of 43% for A1 and 19% error for A2. The Tieleman work reports variation in turbulent intensity and length scales in the WT testing. These issues will be considered in the further investigations. Since this is a challenging problem and several factors influence the peak pressure further study is underway.

3. CONCLUSIONS AND FUTURE WORK

The reported CFD C_p are from preliminary investigations. The reported corner peak pressure coefficients have 19% to 43% error, and they are unacceptable. Several factors influence the flow and further study is warranted. The effect of several factors listed below will be investigated for further improvements and the next level of results will be reported in the conference and in the full paper. The factors identified for investigation: (1) The considered computational domain around the building is similar to 90-degree angular flow reported in Mansouri et al. (2022). The effect of larger domain needs to be investigated. The effect of length of computational region before the building needs also to be investigated (2) The effect of inflow parameters on peak pressure are investigated by varying the length scale and the Reynolds stresses. (3) The length of computer run needs to be increased to see that effect on C_p. (4) Consider much smaller grid spacing around the building as well as at the inflow. (5) The effect of sudden grid spacing change when using multilevel grid generator on turbulence transport needs to be quantified and (6) Finally, are the points A1 and A2 taken from Mooneghi et al. (2016) is the proper one for CFD needs to be critically investigated. May be more points around the corner need to be probed.

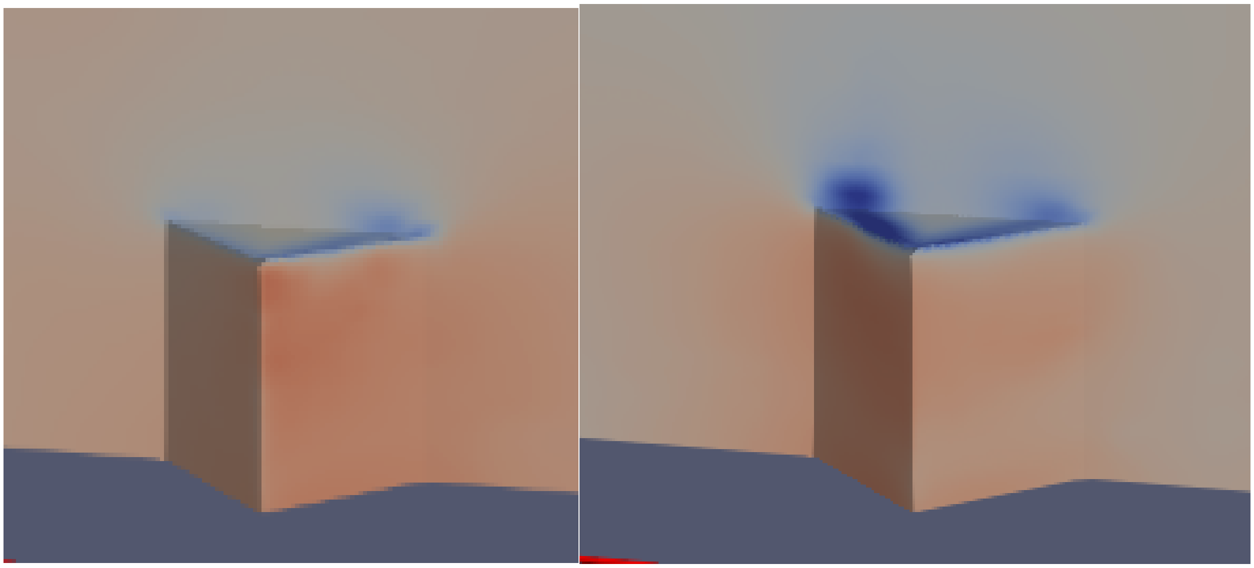

Figure 4. Roof corner vortex developments shown using pressure contour for time 40 and 50 units.

REFERENCES

Mansouri, Z., Selvam, R. P., and Chowdhury, A. G., 2022a. Performance of different inflow turbulence methods for wind engineering applications, Journal of Wind Engineering and Industrial Aerodynamics, 229, 105141

Mansouri, Z., Selvam, R.P. and Chowdhury, A.G., 2022b. Maximum grid spacing effect on peak pressure computation using inflow turbulence generators, Results in Engineering, 15, 100491

Mooneghi, M.A., Irwin, P., Chowdhury, A.G., 2016. Partial turbulence simulation method for predicting peak wind loads on small structures and building appurtenances, Journal of Wind Engineering and Industrial Aerodynamics, 157, 47-62

Selvam, R.P., 2022. Computational fluid dynamics for wind engineering, Chichester, Wiley-Blackwell.

Tieleman, H.W., D. Surry, D., Mehta, K.C., 1996, Full/model-scale comparison of surface pressures on the Texas Tech experimental building, Journal of Wind Engineering and Industrial Aerodynamics, 61, 1-23