RESILIENT INFRASTRUCTURE
June 1–4, 2016

# COMPUTATIONAL MODELING OF TORNADIC LOAD ON A TALL BUILDING

Zoheb, Nasir
Ph.D. Candidate in Western University, Canada

Bitsuamlak, Girma
Associate Professor in Western University, Canada; Associate Director WindEEE Research Institute
Canada Research Chair in Wind Engineering Tier II

## ABSTRACT

Numerical simulations are carried out to analyse the impact of tornadic load on a tall building. For the present study a one-celled tornado replicating a real EF-2 scale has adopted. A standard tall building based on the Commonwealth Advisory Aeronautical Research Council (CAARC) is used. For detail analysis, the tornado is placed in three different locations with respect to the center of the building. These locations are at the tornado center, at the core radii and outside core radii. As the building has rectangular cross section (plan wise), two different orientation of the building with respect to the center of the buildings are considered. Irrespective of the orientation of the building, higher suction obtains when the center of the building coincides with the center of the tornado and it started to decrease as the tornado center moves away from the building center. This happens due to the ground pressure distribution which dominates the overall pressure distribution along the faces of the building. After comparing the pressure distribution on the roof it obtains that, suction is higher for short building than tall building.

Keywords: Computational modeling, Tornado, Tall building.

## 1. INTRODUCTION

Tornado is one of the most devastating natural peril. Each year 800-1000 tornadoes forms on average in North America alone, causing close to 80 fatalities and 1500 injuries, and property damage worth of a billons (on average). In 2015 alone, for instance, 1148 were confirmed cases of Tornado in North America causing 36 fatalities.

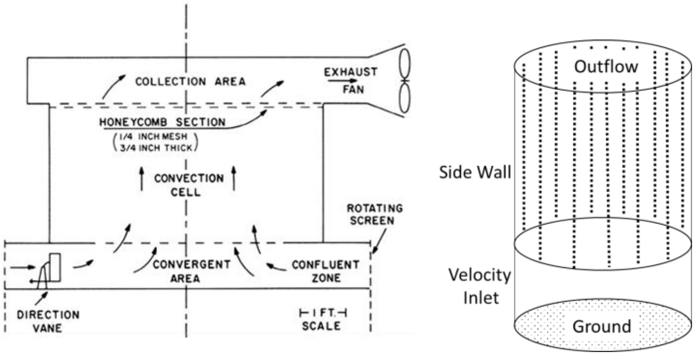
Various research is reported in literature that attempts to analyse the origin and the flow structures of tornadoes of different intensities. Due to the high swirling velocity near the ground and limitation in proper engineering instruments, core analysis of tornadic flow structure has been mostly limited to scaled-down laboratory based and numerical modeling. Some of the previous work focusing on assessing the impact of tornadic flow over engineering structures include: Jischke and Light (1983), Bienkiewicz and Dudhia (1993) and Mishra et al. (2003) measured the surface pressure distribution and flow structure experimentally over scaled down building models in the presence of tornado like vortex and straight line winds and reported that the pressure distribution is quite different among the two wind systems. Wang et al. (2001) and Fouts et al. (2003) reported the rapid change in pressure distribution with respect to the core position. Sarkar et al. (2006) and Sengupta et al. (2006) analysed the force coefficients on a cubic model. Hangan and Kim (2008) simulated tornadoes of different intensity in a experimental model. Haan et al. (2010) analyzed the impact of tornadic load and atmospheric boundary layer in a low rise building with the gable roof. Yang et al. (2011) analysed the impact of tornadic load over a tall building for a laboratory based model. Natarajan and Hangan (2012) analysed the impact of translation and surface roughness over a tornadic structure. Some of the numerical modeling includes the following: Selvam and Millet (2003) analysed the impact of tornadic load over a cubic building using large eddy simulation. They obtained that, translating tornadoes have higher force coefficients on the side walls and roof of the building than a stationary wind. More recently Refan et al. (2014) developed a consistent scaling method through the mini WindEEE Dome experiments and field measurements. The same technique was adopted recently by Nasir et al. (2014), to analyse the difference in pressure distribution and flow structure over a short rectangular bluff-body for different locations with respect to the core center. They obtained from their study that, suction pressure is higher for core center location of the bluff-body and it keep decreasing as it moves farther away from the core center. The present study extends the same approach to assess the tornado induced tall building aerodynamics.

## 2. METHODOLOGY

### 2.1 Numerical model for tornado simulator and topographical change

For the present study, a numerical model depicting the Purdue tornado simulator is used (see Figure 1a). The actual Purdue tornado simulator is cylindrical in shape, where flow enters into the simulator from the bottom. Guide vanes are provided at the inlet to provide desired angle in the inflow. Once the swirling flow enters inside the simulator through the confluent region it reroutes vertically upward in the convection region. To emphasize this vertical movement of swirling flow, an exhaust fan is installed at the outlet to suck out the air from the simulator. Keeping the main operational mechanism in mind, some modification is made into the simplified numerical model (see Figure 1b)

The simplified numerical model is also cylindrical in shape. Despite of providing with the guide vanes, a computer code has implemented to achieve the desired angle in the inflow. A “shear free” sidewall was provided and at the outlet “outflow” boundary condition has been provided. Purdue tornado simulator is a laboratory scaled simulator which is the modified version of the Ward’s tornado simulator. Purdue tornado simulator is around 2m in height and 0.4m in radius. As a result, the original simulator is scaled up in the present numerical analysis keeping the aspect ratio similar to the laboratory scaled model.



(a) (b)

Figure 1: (a) Laboratory and (b) simplified numerical model of Ward's tornado vortex chamber.

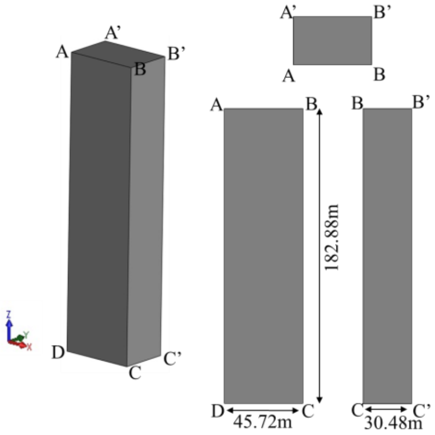


Figure 2: Dimension of the CAARC building

A standard tall building based on the Commonwealth Advisory Aeronautical Research Council (CAARC) is used. The CAARC building is rectangular in shape with dimensions 30.48m (width) by 45.72m (length) by 188.82m (height).

### 2.2 Geometric scaling factor for full scale simulation

As already mentioned in the introduction that one of the main feature of this current study is the full scale numerical modeling. As a result, a geometric scale factor is required to scale up the Purdue tornado simulator. The scale factor for the tornado simulation is chosen by a matching process developed by Refan et al (2013) based on experiments carried out at the mini WindEEE Dome. First, the maximum tangential velocity of numerically simulated tornadoes is determined for different swirl ratios ranging from one to two-celled tornadoes. Then, the horizontal distance from the tornado center and height above the ground corresponding to the maximum tangential velocities for each simulated tornado are obtained and compared with actual tornado data. Since according to fluid dynamic solution there can be only one length scale, it is expected that both the horizontal distance and height of the maximum tangential velocity will converge to a similar length scale value for a particular swirl ratio. Thus, a specific swirl ratio that represent the target tornado is identified.

For the present study, an Enhanced Fujita scale-2 (EF-2) tornado is chosen that took place in Happy, Texas in 2007. The reason for choosing such an EF-2 scale tornado is that, in nature about 90% tornadoes are EF- 2 scale. CFD simulations for various swirl ratios are carried out and the geometric scaling between the field measurements and the CFD is plotted as shown in Figure 3 following the procedure explained earlier.

The core radius and height of the maximum tangential velocity point for the actual tornado and the CFD simulated tornado converged for swirl ratio 0.4 at scale factor of 4222 (see Figure 3) for the target tornado case.

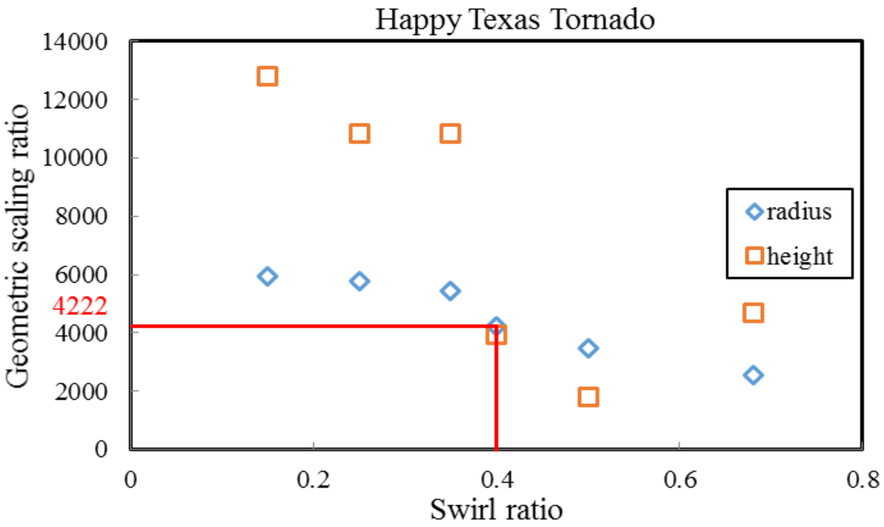


Figure 3: Determining scaling factor for small-scale numerical depicting Purdue tornado simulator.

### 2.3 Meshing for computational domain

For the simulation, commercial software STAR-CCM+ is used. The main advantage of this software is the availability of octagon mesh elements which simulates more efficiently cylindrical shape domains.

Although, the base mesh size is kept comparatively coarse because of the large domain size, the mesh size near the ground and also around the building are kept very fine (wall Y-plus < 2) to capture the sharp velocity changes near the solid region (see Figure 4a, b and c).

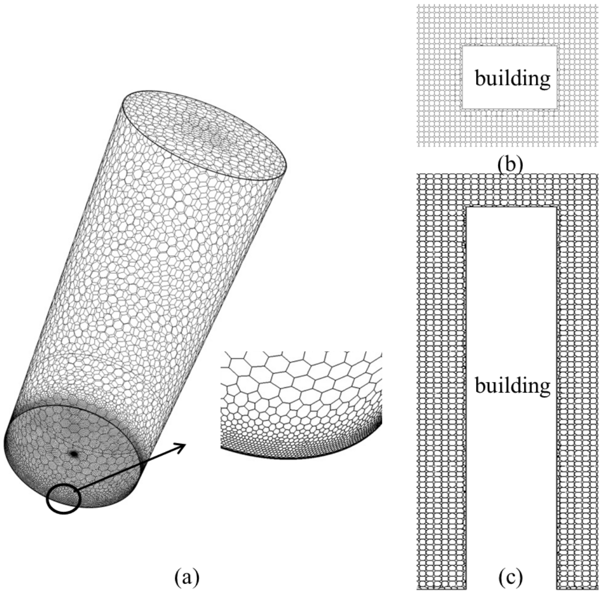


Figure 4: Mesh distribution: Finer mesh close to the ground (a), and around study building plan view (b) and elevation view (c).

### 2.4 Boundary condition for tornado simulator and building

The following boundary conditions for tornadic flow are used: an exhaust fan at the outlet of the laboratory model is replaced by the outflow boundary condition in the numerical model and at the inlet the flow velocity had two components (radial and tangential) in order to produce the swirling flow field. The equation for the radial and tangential velocity components are as follows:

[1] $$V_r = V_1 \times \left(\frac{z}{z_1}\right)^{1/7}$$

[2] $$V_t = \frac{2H_o}{R_o} \times S \times V_r$$

where, $V_r$ and $V_t$ are the radial and tangential component of velocity at 'z' height from ground surface respectively. 'S' is the swirl ratio which actually determines the intensity of model scale tornado. Here, $S = \tan\theta/2a$; $\theta$ is the inflow angle at the inlet and 'a' is the aspect ratio. $H_o$ and $R_o$ are the inlet height and radius of inlet respectively. $V_1$ and $z_1$ are the reference velocity and height respectively. For the current full-scale numerical model, reference velocity, $V_1$ and height, $z_1$ of 10m/s and 106m are chosen respectively based on the actual data obtained from the tornadic event that took place in Happy, Texas in 2007.

### 2.5 Turbulence model used for simulation

A 3D computational modeling is adopted for numerical simulation using Reynolds Averaged Navier-Stokes (RANS) equations and for turbulence model Reynolds Stress Model (RSM) is used. RSM turbulence model is chosen over k-epsilon and other two equation models because of its better accuracy for rotating flows.

The two equation models are based on the Boussinesq assumption which postulates that, the Reynolds stress tensor must be proportional to the strain rate tensor. However, for complex flow, such as tornadic flow this particular assumption does not work because of the curvature effect. Again Large Eddy Simulation (LES) turbulence model is not considered because of the nature of the simulations which makes it computationally costly.

## 3. RESULTS AND DISCUSSION

### 3.1 External pressure distribution

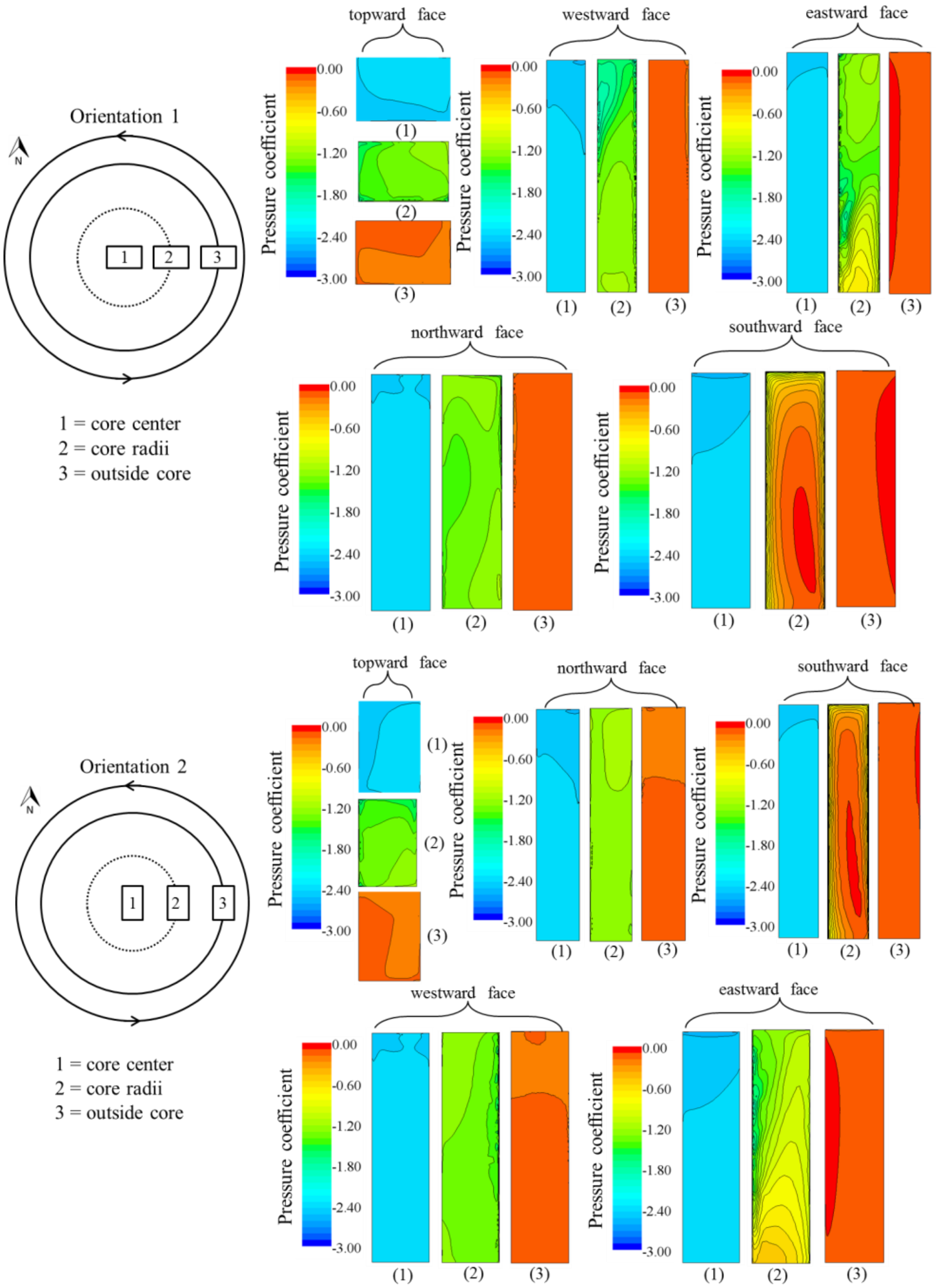


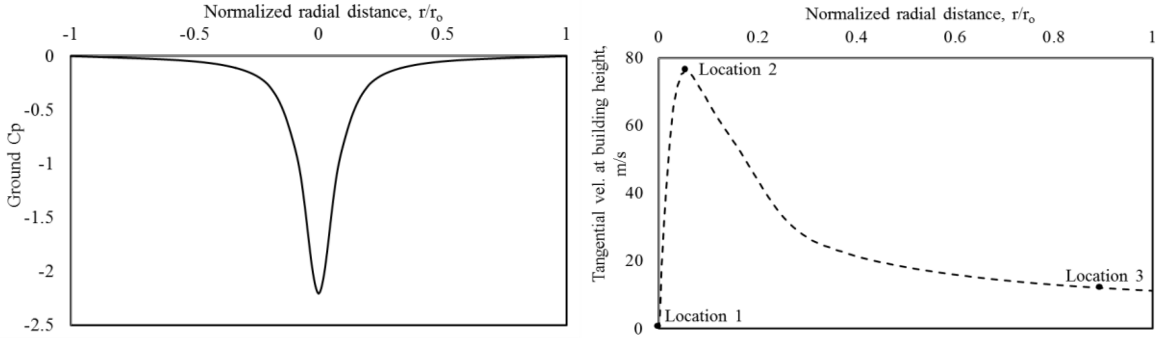
Figure 5: Cp distribution along the faces of the building

For the present study pressure coefficients are measure along the outer surfaces of the building. Pressure coefficient, Cp on the ground is obtained using following equation,

[3] $$C_p = \frac{p - p_o}{\frac{1}{2}\rho {u_o}^2}$$

Here, ‘$p_o$’ is the maximum pressure at the ground, ‘$\rho$’ is the density of air and ‘$u_o$’ is the reference velocity which in this case is the maximum horizontal velocity at the height of the building but in the absence of the building. As the building is rectangular in shape, therefore two different orientation of the building are considered. In orientation 1, the longer edge of the building is perpendicular to the tangential component of the tornadic flow, however in orientation 2, the longer edge is parallel to the tangential component of the flow (see Figure 5).

Irrespective of the orientation of the building, maximum suction obtains on the outers surfaces of the building for location 1 where center of the building coincides with the center of the tornado core. However, suction started to decrease as the tornado moves away from the building.



(a) (b)

Figure 6: Ground Cp and tangential velocity distribution

As a result, lowest suction obtains for location 3 where center of the building is far away from the tornado core center. This indicates the overall suction on the outer surfaces of the building is dominated by the ground pressure distribution which occurs due to the angular momentum of the flow (see Figure 6a).

Now for location 1, suction along all the surfaces of the building are almost identical. However, for location 2, where center of the building coincides with the core radii (location of maximum tangential component of the tornadic flow) location of the tornado, due to the anticlockwise rotation of the flow and maximum tangential flow distribution, suction is lowest on the southward face of the building irrespective of the orientation of the building. Also in this location, the flow approaches the building at a slight angle because of the presence of the inward radial component of the flow near the building (see Figure 7). As a result, suction on the northward and westward faces of the building is higher than the southward and eastward faces of the building due to the recirculation of the flow.

Lastly for location 3, where tornado core center is far away from the center of the building, suction on all the surfaces are very small and almost identical except on the corner of southward and eastward face. This is due to the incoming tornadic flow which approaches the building at a sharp angle (almost $45^0$). Although the flow creates recirculation on the back of the building, but still it does not increase the suction significantly due to the weak tangential component of the flow (see Figure 6b).

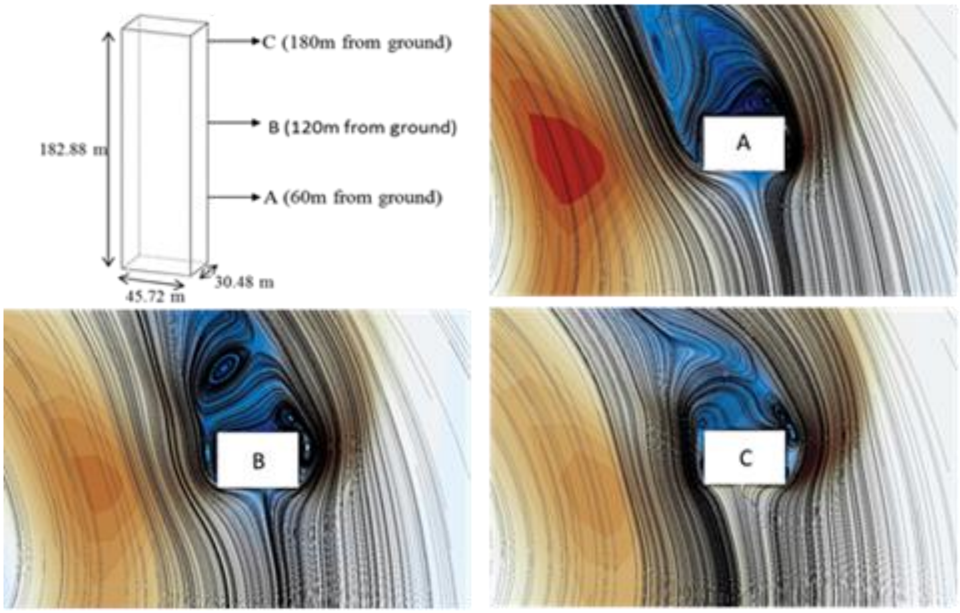


Figure 7: Flow distribution at core radii

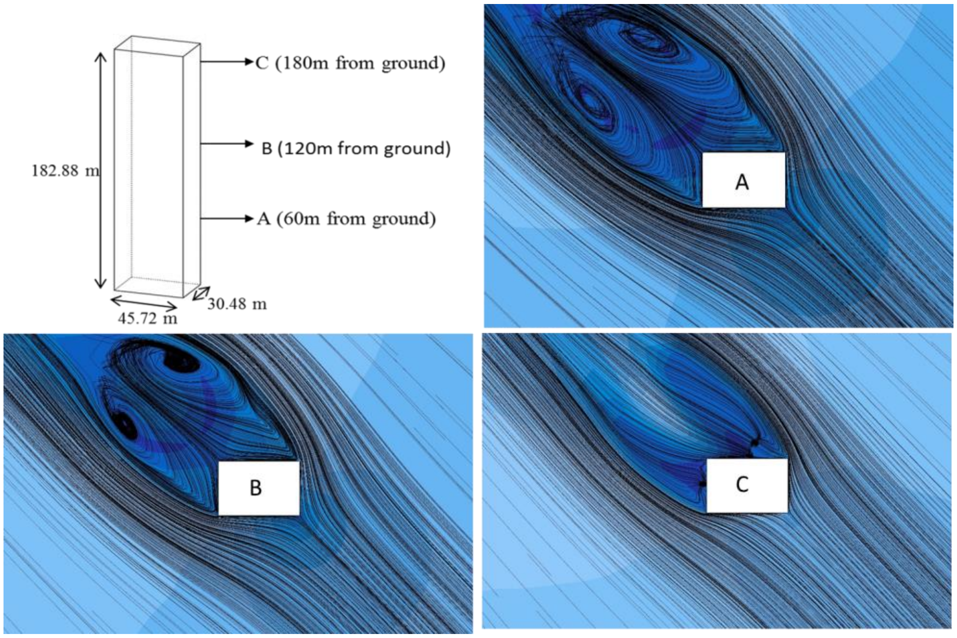


Figure 8: Flow distribution at outside core

### 3.2 Comparison of Cp on the roof

Pressure distributions on the roof are compared between short and tall building. For the short building pressure distribution has obtained from the previous study of Nasir et al. (2014). For both the cases location 1 is chosen where center of the building coincides with the center of the tornado core. The reason for adopting such location is the higher suction than other two locations. For both the buildings, Cp is measured along the centerline parallel to the longer edge of the building. In this case Cp is measured using the same equation used in the previous case, however to make the comparison logical reference velocity is chosen as the maximum horizontal velocity at the height of the short building but in the absence of the building. As the buildings have different dimensions for the edges, so to facilitate the comparison radial distance is normalized by the length of the tall building ($x_{\text{tall-building}}$).

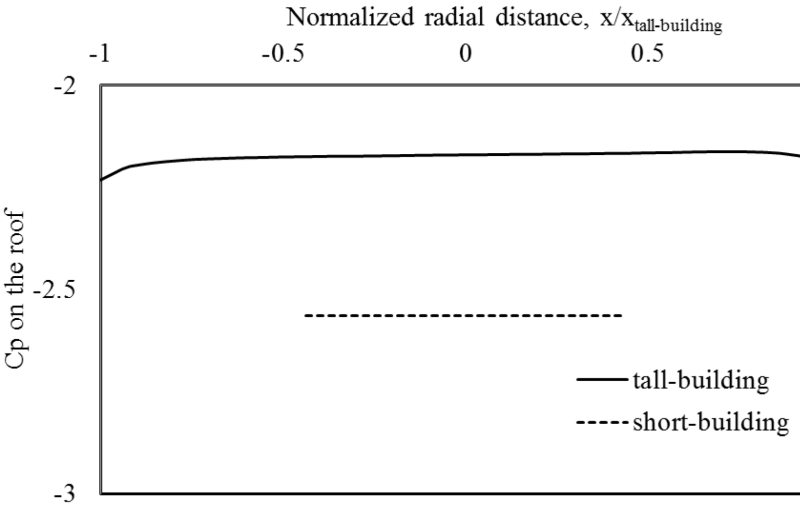


Figure 9. Cp comparison between tall and short building

From the comparison it can be obtained that, for both the cases pressure distribution is quite uniform along the roof. However, higher suction is obtained for the short building than the tall building. This is because of the over pressure distribution on the ground which in turn dominates the pressure distribution along the faces of the building. As the height if the tall building is around 18 times higher than the short building, the domination of the ground Cp is lees for tall building.

## 4. CONCLUSION

Some concluding remarks can be obtained from the present study which are as following,

- Irrespective of the orientation of the building, suction is higher for core center location compare to other locations.
- For core radii location, suction was lesser in southward face compare to other faces of the building because of the anticlockwise rotation of the tornadic flow which provides a flow towards the southward face of the building and counteract the suction pressure due to the angular momentum of the flow.
- For outside core location, suction is almost uniform along the all the surfaces except at the corner of the southward and eastward face, because anticlockwise flow approaches the building at angle which creates lesser suction on this corner.
- Suction on the roof is higher for short building than the tall building because of the domination of the ground Cp distribution.

## REFERENCES

Bienkiewicz, B., Dudhia, P., 1993: Physical modeling of tornado like flow and tornado effects on building loading. *Proc. 7thU.S. National Conf. on Wind Engineering*, UCLA, Los Angeles, 95-106.

Church, C., Burgess, D., Doswell, C., Davies-Jones, R., 1993: *Tornado: Its structure, dynamics, predictions and hazards*. Washington, D.C., USA, American Geophysical union.

Dagnew, A., Bitsuamlak, G.T. 2014: Computational evaluation of wind loads on standard tall building using a large eddy simulation. *Wind & Structures*, 18(5), 567-598.

Fouts, L., James, D.L., Letchford, C.W., 2003: Pressure distribution on a cubical modeling tornado like flow. *Proc. 10thIntl Wind Engineering Conf.,* Texas, Tech. Univ., Lubbock, Tex.

Haan, F.L., Jr., Sarkar,P.P., Gallus, W.A., 2008: Design, Construction and Performance of a Large Tornado Simulator for Wind Engineering Applications. *Engineering Structures*, v. 30 1146-1159

Hangan, H., Kim, J-D., 2008: Swirl ratio effects on tornado vortices in relation to the Fujita scale. *Wind and Structures,* **11-4,** 291-302.

Jischke, M.C., Light, B.D., 1983. Laboratory simulation of tornadic wind loads on a rectangular model structure. In: Proceedings of Sixth International Conference on Wind Engineering, vol. 1, Australia & New Zealand.

Jischke, M.C., Light, B.D., 1983: Laboratory simulation of tornadic wind loads on a rectangular building. *J. Wind. Eng. Ind. Aerody.,* **13 (1-3),** 274-282.

Mishra, A.R., James, D.L., and Lecthcford, C.W., 2003: Comparison of pressure distribution on a cubical model in boundary layer and tornado like flow fields. *Proc. Americas Conference on Wind Engineering* , Baton Rouge, LA, USA.

Nasir, Z., Bitsuamlak, G., Hangan, H., 2014: Computational modeling of tornadic load on a bluff-body, 6th International Symposium on Computational Wind Engineering, Hamburg, Germany, June 8-12, 2014

Natarajan, D., Hangan, H., 2012: Large eddy simulation of translation and surface roughness effects on tornado like vortices. *J. Wind Eng. Ind. Aerodyn.* **104-106,** 577-584.

Refan, M., Hangan, H., Wurman, J., 2013: Reproducing tornadoes in laboratory using proper scaling. *J. Wind Eng. And Industrial Aerodynamics,* **135,** 136-148..

Sarkar, P.P., Haan, F.L., Balaramudu, V., Sengupta, A., 2006: Laboratory simulation of tornado and microburst to assess wind loads on buildings. *Proc. ASCE Structures Congress,* ASCE, Reston, Va.

Selvam, R.P., Millet, P.C., 2003: Computer modeling of tornado forces on a cubic building using large eddy simulation. *Arkansas Academy of Sci.* **57,** 140-146.

Sengupta, A., Haan, F.L., Sarkar, P.P., Balaramudu, S.V., 2006: Transient loads on buildings in microburst and tornado winds. *Proc. The fourth International Symposium on Comp. Wind Engr. (CWE2006),* Yokohama, Japan.

Wang, H, Letchford, J, D, Snow, R.J., 2001: Development of a prototype tornado simulator for the assessment of fluid-structure interaction. *Proc. 1st Americas Conference in Wind Engineering,* Clemson, SC.

Yang, Z., Sarkar, P., Hu, H., 2011: An experimental study of high-rise building in tornado-like winds. *J. Fluids and Structures,* v. 27 471-486.