April 2017

Numerical modeling of tornado-like vortex and its interaction with bluff- bodies

Zoheb Nasir
The University of Western Ontario

Supervisor
Dr. Girma Bitsuamlak
The University of Western Ontario

Graduate Program in Civil and Environmental Engineering

A thesis submitted in partial fulfillment of the requirements for the degree in Doctor of Philosophy

© Zoheb Nasir 2017

Follow this and additional works at: http://ir.lib.uwo.ca/etd

Part of the Civil Engineering Commons, Computational Engineering Commons, and the Structural Engineering Commons

Recommended Citation
http://ir.lib.uwo.ca/etd/4451

This Dissertation/Thesis is brought to you for free and open access by Scholarship@Western. It has been accepted for inclusion in Electronic Thesis and Dissertation Repository by an authorized administrator of Scholarship@Western. For more information, please contact tadam@uwo.ca.
Abstract

The study the flow structure of tornado-like vortices and their impact over engineering structures is important due to the extent of tornado induced fatalities and damages observed each year in North America and around the world. In the present study, a numerical modeling approach inspired by the WindEEE Dome and the modified version of Ward’s Tornado Simulator has been developed. Using a full-scale numerical simulator, tornadoes of different intensities have been simulated for different swirl ratio values to study flow structures in comparison with previous studies. The effect of topographic features on the tornado-like vortex has been investigated for the first time. A new approach to quantify the changes in the tornadic wind field “speed-up” due to the topography has been developed. Once a confidence level is achieved on tornado flow structure under flat terrain and with topographic features scenarios, the interaction of the tornado-like vortex with bluff bodies is modeled. For this purpose, both low- and high rise buildings are considered. More emphasis has been given to the low-rise building and hence it has been further investigated for stationary and translating tornado under sealed and opened conditions. The open condition represents breaching on the building envelope due to, for example, wind born debris. Selected experiments have been conducted at WindEEE Dome for validating the numerical model.

For the low-rise building, the overall pressure distribution along the wall surfaces of the body are dominated by the external tornado pressure field (suction) near the ground surface that develops due to the high angular momentum of the flow. For large tornado size with respect to the study buildings, the Cp on the entire building surface resemble the near ground suction irrespective of the shape. Thus, indicating that at the tornado center, the effect of the interaction of tornado with the bluff body is minimum. This is not the case for smaller tornadoes. The tornado building interaction effects (i.e. aerodynamics) start to be dominant as the tornado center is located far away from the study building. A comparison with ABL flow reveals that the Cp magnitudes and their distributions are quite different for tornado center and core radius locations. For offset position of tornado center with respect to the building, the tornado induced Cp value starts to resemble the
ones from ABL-flow case. For opening case, ground suction dominates the overall pressure distribution for all the locations of tornado. As a result, irrespective of the number and location of openings, suction is higher both inside and outside the bluff-body at tornado center and it decreases as tornado moves away from the bluff-body (core radius and outside core regions). For the high-rise building, an additional vertical pressure variation, not seen in the shorter buildings, as the tornado vortex interacts with the building has been observed. The CFD study has been validated in comparison with WindEEE experiments. These studies are expected to contribute to addressing the lack of tornadic aerodynamic characteristics. In the long run this will contribute in enhancing the resiliency of the built environment and safety of our communities.

Keywords

Tornado, Swirl ratio, Numerical model, Tornado Simulator, Building, Opening, Hill-effect, WindEEE Dome, CFD.
Co-Authorship Statement

Chapter 3 is prepared for submitting in a journal paper under the co-authorship of Nasir, Z, Bitsuamlak, G and Hangan, H

Chapter 4 is prepared for submitting in a journal paper under the co-authorship of Nasir, Z, and Bitsuamlak, G

Chapter 5 is prepared for submitting in a journal paper under the co-authorship of Nasir, Z, Bitsuamlak, G, Refan, M and Hangan, H

Chapter 6 is prepared for submitting in a journal paper under the co-authorship of Nasir, Z, and Bitsuamlak, G
Acknowledgments

My first and foremost gratitude goes to my supervisor and mentor Professor Girma Bitsuamlak. His continuous support and time to time directions and suggestions always helped me to establish myself and my research work for last few years.

I also like to thank the StarCCM+ Software and Sharcnet (high performance computation system) engineers and scientists for their constant service and support in completing this research work.

I would like to thank and dedicate this dissertation to my wife and son for their constant support and patiently. I also like to thank my parents and my mother-in-law, who has been an inspiration over these years.

My list of gratitude extends with some special people from our research team. All these years, they helped me selfishly with their knowledge, ideas and also helped me a lot in completing this dissertation.

A special thanks to Anant Gairola. I wish him all the best for his future.
Table of Contents

Abstract .......................................................................................................................... i
Co-Authorship Statement .............................................................................................. iii
Acknowledgments ........................................................................................................ iv
Table of Contents .......................................................................................................... v
List of Tables .................................................................................................................. x
List of Figures ............................................................................................................... xi
Chapter 1 ...................................................................................................................... 1
  1 Introduction ............................................................................................................... 1
    1.1 General Introduction ............................................................................................. 1
    1.2 Motivation and Objective ...................................................................................... 3
    1.3 Thesis Layout ......................................................................................................... 9
References ................................................................................................................... 11
Chapter 2 ..................................................................................................................... 14
  2 Tornado Wind-field Simulation .............................................................................. 14
    2.1 Experimental and Numerical Modeling of Tornado Simulation ......................... 14
    2.2 Simplified Numerical Model ............................................................................... 17
    2.3 Governing Parameters ......................................................................................... 18
    2.4 Tornado Flow-structure for different Swirl-ratios ............................................... 19
References ................................................................................................................... 20
Chapter 3 ..................................................................................................................... 24
  3 Topographic Effect on Tornado-like Vortex ............................................................. 24
    3.1 Introduction ......................................................................................................... 24
    3.2 Methodology ....................................................................................................... 26
    3.2.1 Numerical Model ............................................................................................. 26
3.2.2 Fractional Speed-up Ratio (FSUR) definition ........................................ 30
3.2.3 Grid Independence Test ....................................................................... 32
3.3 Results and Discussion ........................................................................ 32
  3.3.1 Tornado field validation ................................................................... 32
  3.3.2 FSUR comparisons .......................................................................... 34
  3.3.3 Comparison of Cp at the ground level ............................................ 42
3.4 Conclusion ............................................................................................ 43
3.5 Acknowledgement ................................................................................... 43
References .................................................................................................. 43

Chapter 4 .................................................................................................... 48

4 Computational Modeling of Tornado-like Vortex Induced Mean Pressure
  Distribution on a Typical Building with Flat Roof ..................................... 48
  4.1 Introduction ......................................................................................... 48
  4.2 Methodology ....................................................................................... 50
    4.2.1 Numerical Model for Tornado Simulator and Bluff-body ............... 50
    4.2.2 Tornado Geometric Scaling Factor ............................................. 51
    4.2.3 Computational Domain and Bluff-body dimensions ..................... 52
    4.2.4 Boundary Conditions for Simulation ......................................... 53
    4.2.5 Turbulence Model Used for Simulation ....................................... 53
    4.2.6 Location of the study building with respect to the tornado core center... 54
    4.2.7 Grid Independence Test ............................................................ 54
  4.3 Results and Discussion ....................................................................... 55
    4.3.1 Flow Field Analysis ................................................................... 55
    4.3.2 Tornado induced pressure coefficients acting on the building surfaces .... 56
    4.3.3 Comparison of tornadic Cp distribution with those induced by synoptic
          flow ............................................................................................... 58
4.4 Validating with WindEEE ................................................................................................................. 62
  4.4.1 Numerical Model for WindEEE Simulator .................................................................................. 62
  4.4.2 Simplified Numerical Model ...................................................................................................... 64
  4.4.3 Comparing the Ground Pressure Distribution ....................................................................... 65
  4.4.4 Dimension and Location of low-rise building .......................................................................... 66
  4.4.5 Comparing the Pressure Coefficient Distribution .................................................................. 67
4.5 Conclusion .......................................................................................................................................... 69
References ............................................................................................................................................... 71

Chapter 5 .................................................................................................................................................. 74
5 Numerical Modeling of Tornadic Load on a High-rise building ....................................................... 74
  5.1 Introduction ..................................................................................................................................... 74
  5.2 Methodology .................................................................................................................................. 76
    5.2.1 Numerical Model for Tornado Simulator ............................................................................. 76
    5.2.2 Geometric Scaling for Full-Scale Simulation ....................................................................... 77
    5.2.3 Meshing for Tornado Simulation ........................................................................................... 78
    5.2.4 Boundary Condition for Tornado Simulator and Building .................................................. 79
    5.2.5 Turbulence Model for Simulations ........................................................................................ 80
    5.2.6 Grid Independence Test ......................................................................................................... 80
  5.3 Results and Discussion .................................................................................................................... 81
    5.3.1 External Pressure Distribution ............................................................................................... 81
    5.3.2 Comparing Pressure Distribution Using Numerical Model of WindEEE Dome Simulator .... 87
    5.3.3 Comparing the Force and Moment Coefficients .................................................................. 89
  5.4 Conclusion ...................................................................................................................................... 91
References ............................................................................................................................................... 92

Chapter 6 .................................................................................................................................................. 94
6 Tornado-like vortex induced pressure on a bluff-body with and without openings ....94

6.1 Introduction.................................................................................................................94

6.2 Methodology ...............................................................................................................96

6.2.1 Numerical model for tornado simulator.................................................................96

6.2.2 Tornado simulator and bluff body dimensions .......................................................97

6.2.3 Boundary conditions for tornado simulator and bluff-body .................................99

6.2.4 Turbulence Modeling...............................................................................................100

6.2.5 Grid Independence Test .........................................................................................100

6.3 Results and Discussion ..............................................................................................101

6.3.1 Tornado Field Validation .......................................................................................101

6.3.2 Pressure Coefficient Distribution .........................................................................102

6.3.3 Internal Pressure Distribution ..............................................................................103

6.3.4 Internal Pressure Comparison ..............................................................................105

6.3.5 Comparison of the mean internal pressure coefficient for stationary and translating tornado .................................................................................................................................107

6.3.6 Validating with Experimental Results ..................................................................110

6.3.7 External Pressure Distribution .............................................................................110

6.3.8 Net Pressure Distribution .....................................................................................112

6.3.9 Comparison of External Cp between Tornadic and ABL-flow over Bluff-body .................................113

6.4 Conclusion ..................................................................................................................114

References .........................................................................................................................114

Chapter 7 .............................................................................................................................119

7 Conclusion .......................................................................................................................119

7.1 Summary of the Findings ..........................................................................................119

7.2 Future Recommendation ............................................................................................121
Curriculum Vitae
List of Tables

Table 1-1. Enhanced Fujita Scale (Retrieved from Environment Canada) ........................ 2
List of Figures

Figure 1-1. Confirmed and probable tornadoes across Canada from 1980-2009 (Retrived from https://www.theweathernetwork.com/news/articles/tornadoes-in-canada-everything-you-need-to-know/25876/1/1) .................................................................................................................. 2

Figure 1-2. Speed-up for Synoptic flow ............................................................................................................ 7

Figure 1-3. Impact of tornado wind profile over the different types of building .................. 7

Figure 1-4. Internal volume correction for opening on the building ......................... 8

Figure 2-1. (a) Purdue Tornado Simulator and (b) Simplified Numerical Model .... 17

Figure 2-2. Computational Domain for Simulating Tornado ......................................................... 18

Figure 2-3. Comparison of Tornadic flow-structure ................................................................................ 19

Figure 3-1. (a) Laboratory, (b) simplified numerical model of Ward's tornado vortex chamber, and (c) boundary conditions .................................................................................................................. 26

Figure 3-2. Dimensions for (a) full computational domain and (b), (c) hills .................... 27

Figure 3-3. Location of tornado with respect to the crest of the hill ......................... 28

Figure 3-4. Mesh distribution - finer mesh close to the ground (a), and around study..... 29

Figure 3-5. FSUR calculation for synoptic flow ................................................................................ 30

Figure 3-6. FSUR calculation for tornadic flow over a hill ......................................................... 30

Figure 3-7. Comparison of flow-structure between tornadic simulation (a) with and (b) without the presence of the hill .................................................................................................................. 31

Figure 3-8. Velocity field distribution with and without the presence of the hill......... 32

Figure 3-9. Vertical flow structure of tornado-like vortex having swirl ratio 0.4 (one-celled vortex) ........................................................................................................................................... 33
Figure 3-10. Comparison of (a) ground Cp’ for $S = 0.40$ and (b) radial velocity along $R/R_0 = 0.1025$ for $S = 0.28$.

Figure 3-11. Comparison of FSUR at tornado at Location 1 along X-axis.

Figure 3-12. Comparison of Max. FSUR along X-axis.

Figure 3-13. Comparison of different velocity components of tornadic flow, for tornado at Location 1 along X-direction.

Figure 3-14. Comparison of FSUR at tornado at Location 2 along X-axis.

Figure 3-15. Comparison of FSUR and Max. FSUR at Location 2 (along X-axis).

Figure 3-16. Comparison of different velocity components of tornadic flow at Location 2 along X-axis.

Figure 3-17. Tornado center shift at Location 2.

Figure 3-18. Comparison of FSUR and Max. FSUR for tornado at Location 3 along X-axis.

Figure 3-19. Comparison of FSUR and Max. FSUR at tornado Location 2 along Y-axis.

Figure 3-20. Comparison of different components of tornadic flow at tornado at Location 2 along Y-axis.

Figure 3-21. Comparison of FSUR and Max. FSUR for tornado at Location 3 along Y-axis.

Figure 3-22. Vertical components of flow distribution for tornado at Location 3 along Y-axis.

Figure 3-23. Ground Cp comparison between steep and shallow hills.

Figure 4-1. (a) Laboratory and (b) simplified numerical model of Ward's tornado vortex chamber.
Figure 4-2. Determining scaling factor for small-scale numerical depicting Purdue’s TVC ............................................................................................................................................... 51

Figure 4-3. Dimensions for (a) Computational domain and (b) Bluff-body; (c) Boundary conditions for simulation ............................................................................................................................................... 52

Figure 4-4. Location of study building ................................................................................................................................................................. 54

Figure 4-5. Comparison of flow-building interaction streamlines for locations 1, 2 and 3 for S=0.4 ............................................................................................................................................... 55

Figure 4-6. Cp distributions a) along the entire computational domain, and surfaces of the building located b) at the core center (i.e. location 1), c) at the core radius (i.e. location 2) and d) outside the core (i.e. location 3) (S = 0.4) ............................................................................................................................................... 56

Figure 4-7. Ground Cp distribution ........................................................................................................................................................................... 57

Figure 4-8. Computational domain for synoptic flow-building interaction. Finer mesh (b) at the ground surface and around building (c) elevation view, and (d) plan view ......... 58

Figure 4-9. Cp distributions for: (a), (b) synoptic flow and for tornadic flow for locations at (c), (d) core center; (e), (f) core radius and (g), (h) outside core (S=0.4) .................. 60

Figure 4-10. (a)Cross-section of WindEEE Simulator; (a) 3-Dimensional, (b) Plan and (c) Front view of WindEEE Simulator Numerical model ................................................................................................... 62

Figure 4-11. Flow-structure for Tornadic flow (S = 0.5) ............................................................................................................................................... 63

Figure 4-12. Simplified Numerical Model ......................................................................................................................................................... 64

Figure 4-13. Flow-structure inside WindEEE Simulator ........................................................................................................................................... 64

Figure 4-14. Vertical velocity distribution ......................................................................................................................................................... 65

Figure 4-15. Ground Cp Comparison ................................................................................................................................................................. 66

Figure 4-16. Location of tornado center with respect to the center of the building ........ 66
Figure 4-17. Comparing External Cp using (a) WindEEE Numerical and (b) Simplified Numerical model and (c) WindEEE Simulator for location 1 .......................................... 67

Figure 4-18. Comparing External Cp using (a) WindEEE Numerical and (b) Simplified Numerical model and (c) WindEEE Simulator for location 2; Orientation 1 .................... 68

Figure 4-19. Comparing External Cp using (a) WindEEE Numerical and (b) Simplified Numerical model and (c) WindEEE Simulator for location 3; Orientation 2 ............... 69

Figure 5-1. Ward tornado chamber (a) Laboratory and (b) simplified numerical model of Ward's tornado vortex chamber, (c) cross-section of WindEEE simulator, (d) isometric, (e) plan and (f) front views of the WindEEE inspired numerical model ...................... 76

Figure 5-2. Dimensions of CAARC model ................................................................. 77

Figure 5-3. Determining scaling factor for small-scale numerical depicting Purdue tornado simulator ................................................................. 78

Figure 5-4. Mesh distribution: finer mesh close to the ground (a), and around study building plan view (b) and elevation view (c) ......................................................... 79

Figure 5-5. External Cp distribution (Orientation 1) .................................................... 81

Figure 5-6. External Cp distribution (Orientation 2) .................................................... 82

Figure 5-7. Ground Cp distribution .............................................................................. 83

Figure 5-8. Flow-structure at Location 1 .................................................................. 83

Figure 5-9. Flow-structure at Location 2 ................................................................. 84

Figure 5-10. Flow-structure at Location 3 ............................................................... 86

Figure 5-11. Simplified Numerical Model based on WindEEE Simulator ............... 87
Figure 5-12. Comparing Cp along (a, a’, a’’; e, e’, e’’) northward; (b, b’, b’’; f, f’, f’’) westward, (c, c’, c’’; g, g’, g’’)) southward and (d, d’, d’’; h, h’, h’’) eastward walls for location 1 and 2 ........................................................................................................ 88

Figure 5-13. Determining Force and Moment Coefficient .................................. 89

Figure 5-14. Comparison of Force and Moment Coefficient ............................. 90

Figure 6-1. (a) Laboratory and (b) simplified numerical model of Ward's tornado vortex chamber.................................................................................................................. 96

Figure 6-2. (a) Geometric scaling factor and (b) the computational domain, and the study bluff body for (c) sealed and (d) unsealed cases.................................................... 97

Figure 6-3. Bluff-body shape and openings.......................................................... 98

Figure 6-4. Location and orientation of the bluff-body in relation to the tornado center. 98

Figure 6-5. Employed mesh (a) computational domain, and around the study bluff-body (b) elevation view and (c) plan view................................................................. 99

Figure 6-6. Vertical flow structure of tornadoes having swirl ratio 0.4 (one-celled vortex) .......................................................................................................................... 101

Figure 6-7. Comparison of (a) ground Cp’ for S = 0.40 and (b) radial velocity along R/R₀ = 0.1025 for S = 0.28 .................................................................................. 102

Figure 6-8. Internal pressure distribution................................................................ 103

Figure 6-9. Flow structure at location 2................................................................. 104

Figure 6-10. Location of opening/s on the bluff-body for ABL-flow ..................... 105

Figure 6-11. Internal Pressure comparison between Tornadic and ABL-flow......... 106

Figure 6-12. (a) Comparison of mean internal Cp between stationary and translation tornado .................................................................................................................. 107
Figure 6-13. Flow structure at location 1 and 2 for stationary and translation tornado.. 108

Figure 6-14. Variation of Internal pressure with respect to the direction of translation. 109

Figure 6-15. Variation of minimum internal pressure coefficient with radial location.. 110

Figure 6-16. External Cp comparison................................................................. 111

Figure 6-17. Net Pressure Distribution............................................................. 112

Figure 6-18. External Cp comparison between tornadic and ABL-flow over bluff-body
 ........................................................................................................................................ 113
Chapter 1

1 Introduction

1.1 General Introduction

Our communities are exposed to various types of natural disasters such as extreme climate induced events. Based on the geographical location and local climatic behavior, different types of atmospheric instabilities can be caused. For instance, tornadoes are one of the most devastating and persistent atmospheric instabilities, that occur in the North America region. Canada is ranked second in the world in terms of number of tornadoes hit per year. One devastating tornado example in Canada is the F-5 scale tornado that touched down in Manitoba in 2007. Although, no fatalities were reported, it caused a very significant property damage. High speed swirling wind sandblasted the bark of the trees and utility poles (Canada's Top Ten Weather Stories for 2007). In Canada, most of the tornadoes occur in southern Ontario, the southern Prairies and southern Quebec (see Fig. 1-1). Historically, highly populated Canadian cities were also hit by significant strength tornadoes (F-3 or higher) causing large-scale damage and fatalities: in Regina (1912), Windsor (1946 and 1974), Sarnia (1953), Sudbury (1970), Woodstock (1979), London (1984), Barrie (1985), and Edmonton (1987).
Based on the damages caused by tornadoes, Fujita scale was introduced by Tetsuya Fujita in 1971. Although it was operational for 33 years, it had some limitations which led to underrating of tornado intensity in some instances and overrating in others. McDonald and Mehta (2006) modified the Fujita scale to introduce new damage indicators to overcome these limitations and introduced a new scaling technique- Enhanced Fujita scale (EF-scale), McDonald and Mehta (2006). A comparison of wind velocity for F and EF-scales are shown below, (see Table 1.1)

Table 1.1. Enhanced Fujita Scale (Retrieved from Environment Canada)

<table>
<thead>
<tr>
<th>F/EF Rating</th>
<th>F-Scale Wind Speed Rounded to 10 km/h (3s-gust)</th>
<th>EF-Scale Wind Speed Rounded to 5 km/h (3s-gust)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>60-110</td>
<td>90-130</td>
</tr>
<tr>
<td>1</td>
<td>120-170</td>
<td>135-175</td>
</tr>
<tr>
<td>2</td>
<td>180-240</td>
<td>180-220</td>
</tr>
<tr>
<td>3</td>
<td>250-320</td>
<td>225-265</td>
</tr>
<tr>
<td>4</td>
<td>330-410</td>
<td>270-310</td>
</tr>
<tr>
<td>5</td>
<td>420-510</td>
<td>315 or more</td>
</tr>
</tbody>
</table>
More recently Canada has adopted the modified Enhanced Fujita (EF) Scale in 2013. EF-scales in Canada has 31 tornado indicators.

1.2 Motivation and Objective

The engineering research for tornadic effect on structure entails wide range focus starting from characterization of tornadic flow field, its interaction with structure and surrounding (and topography) and tornado induced responses. It also extends up to the post-tornado effects and consequences.

The first group of studies focus on tornado flow characterization using either numerical or experimental approaches. The experimental flow characterization studies include Ward (1972); Davies Jones (1973, 1979, 1986); Mitsuta & Monji (1984); Haan et al (2008, 2010); Matsui and Tamura (2009). Ward (1972) built a tornado vortex chamber (TVC) with geometric and dynamic similarity to a real tornado. Subsequently, several laboratories have been built by various researchers to analyze different aerodynamic properties, such as those developed by Chang (1972) to analyze the velocity field in a simulated tornado using three-dimensional velocity probe for two different swirl ratios. During the same period, Davies Jones (1972) investigated the dependency of core radius over swirl ratio using Ward’s tornado simulator. Church et al. (1979) used a tornado simulator like the Ward’s simulator and identified the important transition points in a tornadic flow structure. Mitsuta and Monji (1984) used laboratory scaled model to simulate one and two-celled tornadoes and reported that the maximum horizontal velocity occurs near the ground surface and the height of this maximum velocity is insensitive to the swirl ratio. Diamond and Wilkins (1984) analyzed the impact of translation using modified Ward’s tornado simulator and translation causes a local increase in the swirl ratio and increase the size of the core radius compared to stationary vortex. Matsui and Tamura (2009) simulated tornadoes of different intensities for different floor roughness conditions and found that floor roughness is more effective for low swirl ratios than higher ones. Zhang and Sarkar (2009) used the ISU Tornado Simulator to analyze the flow structure near the ground and found that the tangential velocity is the dominant component of flow and its peak value is three times higher than radial velocity component. Tari et al. (2010) also simulated tornadoes of different swirl ratios and found
the radial and tangential velocity components of flow as well as the core radius increase with the higher swirl ratio values. Refan et al. (2013) used Mini WindEEE Dome at the University of Western Ontario to simulate tornadoes of different swirl ratio and compared the location of the maximum tangential velocity point with the actual scale tornado to develop consistent geometric scaling approach for tornadic flows.

Some of the numerical studies include Harlow and Stein (1974), Rotunno (1977), Church et al (1993), Noland and Ferrell (1999), Lewellen and Lewellen (1997, 1999, 2007), Hangan and Kim (2008); Natarajan and Hangan (2012 etc.). Harlow and Stein (1974) simulated tornadoes of different intensities and analyzed various flow related parameters. Rotunno (1977, 1979) numerically modeled Ward’s tornado simulator and reported that core radius is independent of the Reynolds number, and analyzed the flow structure for different swirl ratio values. Church et al. (1993) numerically simulated the tornado and reported that as the swirl ratio increases the altitude of the vortex breakdown decreases until swirl ratio, S = 0.45. Nolan and Ferrell (1999) indicated that vortex Reynolds number controls the flow structure and maximum wind speed of the tornado flow. Lewellen and Lewellen (1997, 2007) numerically simulated a three-dimensional tornado and analyzed the flow structure near the ground. Kuai et al. (2008) replicated the ISU Tornado Simulator numerically and compared their results with the laboratory model. Hangan and Kim (2008) used their simulation to analyse the dependency of flow dynamics on swirl ratio and its relation with the Fujita scale. Ishihara et al. (2011) simulated tornadic flow using LES turbulence for two swirl ratios which represented one and two-celled tornadoes and obtained from their study that, for one-celled type vortex peak vertical velocity occurs at the center, however for two-celled vortex, it occurs near the radius of the maximum tangential wind. Hangan and Natarajan (2012) used LES simulation to analyse the impact of ground surface roughness and translation and obtained from their study that translation reduces the maximum mean tangential velocity for low swirl ratio, however for high swirl ratio it increases slightly. Ground roughness was also reported to decrease the mean tangential velocity at all swirl ratios.

The second group of studies focused on simulating tornadic like vortex-structure interaction. Some of the experimental and numerical studies include, Chang (1971);
Mehta et al. (1976); Jischke and Light (1983); Bienkiewicz and Dudhia (1993); Mishra et al. (2003); Sarkar et al. (2006); Sengupta et al., 2006; Haan et al., 2010; Sabareesh et al. 2012 etc.). Chang (1971) reported that the pressure distribution on the model is a combined effect of both suction and dynamic pressure. Mehta et al. (1976) estimated tornado wind speeds from the analysis of structural failure. Jischke and Light (1983) and Bienkiewicz and Dudhia (1993) modified Ward's laboratory model to study the effects of the location of object with respect to the tornado vortex. Wang et al. (2001) and Fouts et al. (2003) both reported that the pressure distribution changes rapidly with change in location of the model with respect to the tornado core. Mishra et al. (2003) compared the pressure distribution due to tornadic vortex and atmospheric boundary layer flow in a cubic model. Sarkar et al. (2006) and Sengupta et al. (2006) used numerical (LES) and experimental methods to simulate transient loading due to tornado on a cubic building and reported that tornadoes produce higher peak loads than microburst and slow moving and smaller tornadoes produce higher peak loads. Their model was similar to the model presented in Fouts (2003). Also, Mishra et al. (2003) simulated a single celled tornado-like vortex and compared its flow field with an actual scale tornado of F4 intensity. Haan et al. (2010) analyzed the impact of tornadic load and atmospheric boundary layer in a low-rise building with gable roof and reported that the peak values of side forces exceeded the standard by up to a factor of 1.5 while the uplift exceeded the ASCE 7-05 provisions by a factor as high as 3.2, indicating that buildings designed based on ASCE-7-05 were possibly under designed. Some of the numerical studies on tornado-building interaction include the following. Selvam and Millet (2003) modeled the interaction between a tornadic vortex and cubic building using a large eddy simulation. They reported that a translating tornado produces higher overall forces on the side walls and roof of the building than quasi-steady wind.

The third group of researchers include post damage assessment after tornadic event. This includes the continuous field campaign reports that come out from University of Florida team (Wind Hazard Damage Assessment). Recently, they published several reports regarding the damage caused by tornadoes and the types of damage. Their recent reports include, southeastern US tornado outbreak (2017), Christmas tornado outbreak (2015), Johns Island tornado (2015) etc. One of their important observation was, besides the
intensity of tornado and flow-structure, is the tornado pathway. Where they observed some strong tornado did not cause severe damage because of offset pathway from the community.

While congregating adequate knowledge for further research in tornado-like vortex, it has been observed that compared to synoptic wind, tornadic flows are the least studied. Based on the limited studies, the following gaps were identified. Regarding flow structure, most of the tornado-like vortices were simulated over smooth flat ground and some studies were conducted with surface roughness into consideration. However, it is also important to assess the impact of topographical changes over tornado flow structure. For example, transmission towers are often installed over hilly areas and are exposed to tornadic damage. The state of the art in numerical tornadic tornado-like vortex structure interaction is limited to mean pressure evaluation on scaled model and there is no report on design load (peak) evaluations. Considering the computational resources required for peak load calculation may not be feasible any time soon. However, with regard to buildings with porosity (openings) the ability to do full-scale numerical studies needs to be emphasized. This is because for internal pressure studies, maintaining the volume or developing volume consistent scaling is important. To this effort, the work by Refan et al. (2014) on scaling tornadic flow shall be further examined and adopted appropriately. Due to the finite size and three-dimensional nature of tornadic like vortex, it is also important to study the effect of the size of the tornado on building aerodynamics parameters. This point does not seem to be addressed in the current literature. The various experimental facilities seem to adopt different ways of tornado-like vortex generation leading to multiple interpretation of the aerodynamic data. This problem will continue to persist as many tornado simulators following different concepts. This is aggravated further with lack of target field tornado flow data close to the ground. For example, the synoptic flow has benefitted a wealth of a strong historic meteorological database. This will remain a challenge in developing consistent tornado load evaluation methods.

Based on the observed limitations in tornado research discussed above, it is important that further studies both in field data collection and proper method development to
analyses the tornado-structure interactions required. Some of the overarching problems identified are listed below.

![Figure 1-2. Speed-up for Synoptic flow](image)

Previous studies show that, when a synoptic flow approaches a hill flow-structure changes over the surface of the hill and maximum speed-up occurs at the crest of the hill (see Fig. 1-2). However, tornado has a very complex flow and its flow structure changes at every location from tornado center. Thus, it requires different approach for obtaining speed-up ratios and identifying the location of maximum speed-up.

![Figure 1-3. Impact of tornado wind profile over the different types of building](image)

In actual tornadic flow, maximum tangential velocity lies within 20 to 50 m height from the ground level. So, size (height) of the building is an important aspect. Most of the low-rise buildings usually fall within the maximum wind speed height, while tall-buildings
(high rise) fall beyond this height (see Fig 1-3). Thus, it is anticipated that due to the complexity of the tornadic flow, low and high-rise buildings aerodynamics to be very different and the differences be examined carefully.

![Diagram of flow and opening](image)

**Figure 1-4. Internal volume correction for opening on the building**

Generally, impact of synoptic flow over openings on a building is evaluated using scaled laboratory models and applying an internal volume correction (see Fig. 1-4). However, tornadic flow is complex and it is cumbersome to apply internal volume correction and therefore full-scale simulation is warranted. But a balance with computational cost and large scale simulation need shall be investigated.

Based on the literature review and gathered knowledge, the main objective of the proposed study is to develop a numerical model to represent tornado flow-structure and to assess its interaction with bluff bodies. The numerical model is also validated in comparison with WindEEE experimental measurements.

To achieve this objective, the following tasks are pursued by numerically simulating tornado vortices and its interaction with engineering structures (representing simple building shapes):

- Developing a simplified numerical model inspired by Purdue tornado simulator to simulate tornado-like vortex for different swirl ratio values that realistically represents flow structure of mechanically produced tornadic flows (flow-
structure, ground pressure distribution, location of maximum tangential velocity etc.).

- Developing new methods to assess the impact of different types of topographic features on tornadic flow structure commonly referred as “speed-ups”.

- Numerically estimating the tornadic load on a low-rise building for different locations of tornado with respect to the location of the building.

- Numerically estimating the tornadic load on a standard tall building placed at different locations with respect to the tornado center.

- Numerically estimating the tornadic load on a low-rise building with opening’s, placed at different locations with respect to the tornado center. Assessing the internal pressure for various opening configurations and orientations.

1.3 Thesis Layout

The layout for this dissertation is “integrated article” format as specified by the faculty of graduate studies in Western University, Canada.

Chapter one is an introduction about tornado and its impact on our communities in recent years. It also presents the motivation behind the present study and the objectives of this study. Chapter two presents the simulation of tornado on an empty domain using a mechanically produced vortex. It discusses the development of simplified numerical model depicting the modified version of the Ward’s Tornado Simulator (simply named modified numerical model hereafter) and the WindEEE Dome. Description of tornadic flow structure depending on the swirl ratio is also included in this chapter. This chapter ends with a comparison of ground pressure distribution between numerical model of WindEEE simulator and modified numerical model for identical swirl ratio value to build a confidence on the numerical modeling of the tornado-like vortex. Chapter three presents the study of topographic effect on tornado-like vortex. Two different types of hills based on their slopes are considered. Three tornado locations with respect to the crest of the hill are also considered. Chapter four presents a numerical evaluation of
tornado-like vortex induced pressure on low-rise building. A comparison of pressure
distribution along the surface of a low-rise building is conducted among (i) simplified
numerical model, (ii) numerical model mimicking WindEEE Dome and (iii) experimental
measurements at WindEEE Dome for various locations and orientations of the study low-
rise building. A comparison of external surface distribution between ABL-flow and
tornadic flow cases is presented. Chapter five presents tornado-like vortex induced
pressure on a high rise building A comparison of pressure distribution along the surface
of a low-rise building is conducted among (i) modified numerical model and (ii)
numerical model mimicking WindEEE Dome. Chapter six presents tornado induced
internal pressure for the low-rise building for stationary and translating tornado. The
effects of building location, orientation and number of openings are investigated. Chapter
seven presents the conclusion based on the present work and scope of future studies.
References


Chapter 2

2 Tornado Wind-field Simulation

2.1 Experimental and Numerical Modeling of Tornado Simulation

Various scale and numerical tornado models exist in literature. The use of numerical method can generally be divided into two categories: thunderstorm scale simulations and tornado scale simulations. In the former category, Klemp and Whilhelsom (1978), three-dimensional cloud models are used to numerically simulate the formation and dynamics of the thunderstorms that are responsible for tornado formation. On the other hand, tornado scale models, pioneered by Rotunno (1977), assume a particular environment of rotation coupled with convection to create an intense vortex near the surface of earth. These models are intended to provide the details of the wind field in a tornado and an understanding of the dynamics that lead to tornadic flow structure. This research focusses on the second category to facilitate a better understanding of the interaction between tornado-like vortices and built environment.

Several laboratory scale models and numerical models have been used to analyze the flow structure. In 1972, Ward was the first to build a tornado vortex chamber with geometric and dynamic similarity to depict real scale tornadoes. In his design, at the outflow, a fan was provided to generate updraft and guide vanes near the floor to generate angular momentum. Wan and Chang (1972) replaced the guide vanes with the rotating screen and was able to measure the radial, tangential and axial velocities with the three dimensional velocity probes. From their analysis, they found that for low swirl ratio, vertical velocities were all positive and quite strong; however, for high swirl ratio vertical velocities were negative at the core of the tornado. Mitsuta and Monji (1984) modified the simulator and provided the rotation at the top. From their analysis they found maximum tangential velocity occurs near the ground surface for two celled type vortex. Recently, Haan et al. (2008) also used the same simulator with guide vanes at the top to generate the tornado vortex. Matsui and Tamura (2009) have conducted researches using Ward-type simulator with laser Doppler Velocimetry (LDV). Although laboratory
simulation provides a controlled axis symmetric tornado vortex, it still has some limitations. In laboratory simulations, it is difficult to obtain detailed three dimensional velocity and pressure fields due to the strong turbulence motion near the surface and the center of the vortex.

Numerical simulation is more efficient and economical to analyze the three-dimensional velocities and pressure fields due to the strong turbulence motion near the surface and the center of the vortex. Using laboratory model, based on Ward Tornado Vortex Chamber, Harlow and Stein (1974) was the first to simulate a tornado numerically. They were able to simulate one celled and two celled tornado vortices using free slip boundary condition and compared their results with laboratory scale model. Rotunno (1977, 1979) used the same model used by Harlow and Stein (1974), and was able to observe a tornado with Vortex Break Down. He also showed in his study that, core size of a tornado is a function of Swirl ratio.

Recently, Howells et al. (1988) and Nolan and Ferrell (1999) used axis-symmetric Navier-Stokes equation with constant viscosity in cylindrical coordinates to investigate the dynamics of tornado vortex. Nolan and Ferrell (1999) were able to simulate a low swirl one cell type vortex and drowned two celled type vortex. Lewellen and Lewellen (1997, 2007) used LES to analyze the impact of swirl ratio and transition. Lewellen et al. also analyzed the interaction of swirl ratio with the surface roughness. Hangan and Kim (2008) attempted to establish a relation between swirl ratio in laboratory or numerical simulations and the Fujita scale in actual tornadoes using RANS (Reynolds Averaged Navier-Stokes) equation for the simulation. Recently, Hangan and Natarajan (2012) used LES (Large Eddy Simulation) to analyze the impact of translation and surface roughness on tornado-like vortices. Moving wall condition was used to achieve the translation and surface roughness was achieved by using roughness blocks. From their analysis, they observed that for low swirl ratio, the maximum mean tangential velocity increases with the increase in translation while for the high swirl ratio the maximum mean tangential velocity decreases with the increase in the translation. Maryam et al. (2013) were able to establish a scaling ratio for the simulated tornadoes and full scale tornadoes using GBVTD (Ground-Based Velocity Track Display) analysis of full-scale data.
With regard to the tornado/structure interaction, Chang (1971) was the first to use his own simulator to produce a single celled tornado and analyze the pressure data on a cube model for two different locations. He obtained from his study that, the pressure distribution on the model is a combined effect of both suction and dynamic pressure. Most of the laboratory simulators were based on the pioneering work of Ward (1972). Mehta et al. (1976) were able to estimate tornado wind speeds from the analysis of structural failure. Jischke and Light (1983) and Bienkiewicz and Dudhia (1993) modified the Ward's laboratory model and concluded the damage related with tornado does not only depend upon the maximum velocity but also upon the location with respect to the tornado vortex. Wang (2001) and Fouts (2003) both found from their study that pressure distribution changes rapidly with change in location of the model with respect to the core position. Mishra et al. (2003) compared the pressure distribution due to tornadic vortex and atmospheric boundary layer flow in a cubic model. Sarkar et al. (2006) and Sengupta et al. (2006) used numerical (LES) and experimental methods to simulate transient loading due to tornado on a cubic building and reported that tornadoes produce higher peak loads than microburst and slow moving and smaller tornadoes produce higher peak loads. Recently, Mishra et al. (2008) was able to obtain a single celled tornado-like vortex and compared its flow field with actual scale tornado of F4 intensity. Haan et al. (2010) also analyzed the impact of tornadic load and atmospheric boundary layer in a low-rise building with gable roof and reported that the peak values of side forces exceeded the standard by up to a factor of 1.5 while the uplift exceeded the ASCE 7-05 provisions by a factor as high as 3.2, indicating that buildings designed based on ASCE-7-05 were possibly under designed.

Selvam and Millet (2003) were able to analyze the interaction between a tornadic vortex and cubic building developing a three-dimensional turbulence model based on a large eddy simulation. From their study, they found that a translating tornado produces higher overall forces on the side walls and roof of building than quasi-steady wind.
2.2 Simplified Numerical Model

Figure 2-1. (a) Purdue Tornado Simulator and (b) Simplified Numerical Model

For the present study, a numerical model depicting Purdue Tornado Simulator is used (see Fig. 2-1a). This Purdue simulator is an upgraded version of the Ward’s Tornado Vortex Chamber. In Purdue simulator, flow enters the simulator from the bottom and directional vanes provide the required flow direction angle to create a circular motion in the flow and exhaust fan sucks the flow out from the simulator. Similar concept is used for the present CFD based numerical modeling after some modifications, that is enabled due to the ease of boundary condition application in the numerical domain as shown in Fig. 2-1b. For example, the guide vanes can be simply replaced with an inclined velocity inflow boundary condition etc.

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:

\[ V_r = V_1 \times \left( \frac{z}{z_1} \right)^{1/7} \]
\[ V_t = \frac{2H_0}{R_0} \times S \times V_r \]

where, \( V_r \) and \( V_t \) are the radial and tangential component of velocity at \( z \) height from ground surface respectively. \( V_1 \) and \( z_1 \) are the reference velocity and height respectively. Shear free sidewall is used. For numerical simulation, RSM (Reynolds Stress Model) turbulence model is used.

### 2.3 Governing Parameters

To simulate a tornado properly in laboratory scaled modeled or numerical model, it is important to consider several parameters which controls the flow structure of tornadic flow. They are as follows,

Swirl ratio (\( S \)) determines the helicity of a tornado-like vortex (Lewellen (1962) and David Jones (1973)). It is the ratio of the ambient vertical vorticity to the ambient horizontal convergence. It defines as follows,

\[ S = \frac{\tan \theta}{2a} \]

Here, \( \theta \) is the inflow angle (see Fig. 2-2) at the inlet and \( a \) is the aspect ratio.

![Figure 2-2. Computational Domain for Simulating Tornado](image)
Aspect ratio \((a)\) is the ratio of depth of inflow \((H_0)\) in a tornado vortex chamber to the radius of updraft \((R_0)\).

In addition to these, another important parameter is the Radial Reynolds Number.

\[
Re_r = \frac{Q}{2\pi \vartheta}
\]

\(Q\) is the volumetric flow rate per axial length for flow inside the chamber (see Fig. 2-2) and \(\vartheta\) is the kinematic viscosity of air.

## 2.4 Tornado Flow-structure for different Swirl-ratios

![Figure 2-3. Comparison of Tornadic flow-structure](image)

Using the modified numerical model discussed above, several tornadoes are simulated by altering the swirl ratio values (see Fig. 2-3).

For low swirl ratio, swirling flow approaches toward the center of tornado and then moves upward. The swirling is not strong enough to create vertical pressure gradient which causes downward flow. As the swirl ratio value increases \((S = 0.25)\) a downward flow breaks the vortex at the center and this downward flow moves further toward the
ground as swirl increases. At around swirl ratio value 0.5 the downward flow touches the
ground and divides the one-celled vortex into two. This phenomenon is called vortex
touch down. For the present arrangement (aspect ratio = 1.025), vortex touch down
happens at swirl ratio 0.5, however, this phenomenon may change by changing the aspect
ratio value.

References

   University.

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

3. Chang, C.C., 1971: Tornado effects on building and structures with laboratory
   simulation. Proc. 3rd Int. Conf. on Wind Effects on Buildings and Structures,
   Saikon, Tokyo, 231-240.

dynamics, predictions and hazards. Washington, D.C., USA, American Geophysical
   union.

tornado-like vortices as a function of swirl ratio: A laboratory investigation. Journal
   of the Atmospheric Science.36, 1755-1776.

   Univ., Lubbock, Tex.

   Atmos. Sci., 31, 2081-2091.


3 Topographic Effect on Tornado-like Vortex

3.1 Introduction

Previous studies have used both laboratory and numerical scaled models to analyze the tornadic flow and its interaction with structures. Ward (1972) built a tornado vortex chamber (TVC) with geometric and dynamic similarity to a real tornado. Subsequently, several laboratories have been built by various researchers to analyze different aerodynamic properties, such as those developed by Chang (1972) to analyze the velocity field in a simulated tornado using three-dimensional velocity probe for two different swirl ratios. During the same period, Davies Jones (1972) investigated the dependency of core radius over swirl ratio using Ward’s tornado simulator. Church et al. (1979) used a tornado simulator like the Ward’s simulator and identified the important transition points in a tornadic flow structure. Mitsuta and Monji (1984) used laboratory scaled model to simulate one and two-celled tornadoes and reported that the maximum horizontal velocity occurs near the ground surface and the height of this maximum velocity is insensitive to the swirl ratio. Diamond and Wilkins (1984) analyzed the impact of translation using modified Ward’s tornado simulator and translation causes a local increase in the swirl ratio and increase the size of the core radius over stationary vortex. Haan et al. (2008, 2010) used the Iowa State University (ISU) Tornado Simulator to simulate tornadoes of different swirl ratios, they also compared peak load from the impact of tornadic load on a model low-rise building with those prescribed by the ASCE 7-05 for straight wind over open terrain. Matsui and Tamura (2009) simulated tornadoes of different intensity for different floor roughness conditions and found that floor roughness is more effective for low swirl ratios than higher ones. Zhang and Sarkar (2009) used the ISU Tornado Simulator to analyze the flow structure near the ground and found that the tangential velocity is the dominant component of flow and its peak value is three times higher than radial velocity component. Tari et al. (2010) also simulated tornadoes of different swirl ratios and found the radial and tangential velocity components of flow as well as the core radius increase with the higher swirl ratio values. Refan et al. (2013) used Mini WindEEE Dome at the University of Western Ontario to simulate tornadoes of different swirl ratio...
and compared the location of the maximum tangential velocity point with the actual scale tornado to develop consistent geometric scaling approach for tornadic flows.

In parallel with these experimental efforts, several numerical analyses have also been performed such as those by Harlow and Stein (1974) to simulate tornadoes of different intensities and analyse various flow related parameters. Rotunno (1977, 1979) numerically modeled Ward’s tornado simulator and reported that core radius is independent of the Reynolds number, and analysed the flow structure for different swirl ratio values. Church et al. (1993) numerically simulated the tornado and reported that as the swirl ratio increases the altitude of the vortex breakdown decreases until swirl ratio, $S = 0.45$. Nolan and Ferrell (1999) simulation indicated that vortex Reynolds number controls the flow structure and maximum wind speed of the tornado flow. Lewellen and Lewellen (1997, 2007) numerically simulated a three-dimensional tornado and analysed the flow structure near the ground. Kuai et al. (2008) replicated the ISU Tornado Simulator numerically and compared their results with the laboratory model. Hangan and Kim (2008) used their simulation to analyse the dependency of flow dynamics on swirl ratio and its relation with the Fujita scale. Ishihara et al. (2011) simulated tornadic flow using LES turbulence for two swirl ratios which represented one and two-celled tornadoes and obtained from their study that, for one-celled type vortex peak vertical velocity occurs at the center, however for two-celled vortex the peak it occurs near the radius of the maximum tangential wind. Hangan and Natarajan (2012) used LES simulation to analyse the impact of ground surface roughness and translation and obtained from their study that translation reduces the maximum mean tangential velocity for low swirl ratio, however for high swirl ratio it increases slightly. Ground roughness was also reported to decrease the mean tangential velocity at all swirl ratios.

Although a number of studies exist on topography effect on synoptic flow (Bitsuamlak et al. 2004, 2006, 2007; Abdi and Bitsuamlak, 2014 and many others), studies on topographic effect on non-synoptic with the objective of evaluating speed up are very limited. More recently, some studies have considered the effect of topography over the path of tornado. For example, Karstens et al. (2012) simulated an experimental model in ISU tornado simulator to determine the effect of topography and obtained that tornado
path deviated from its original direction as it climbs up and down a hill. Lewellen (2012), used immersed boundary method and large eddy simulation (LES) to simulate tornadoes of different intensities over different topographical changes in 3-dimensional domains. They analyzed the change in tornado path, structure and intensity over different topographical changes. The main objective of these studies is to analyze the change in flow structure in the presence of topographical changes. The present study focuses in generating flow speed-up parameters useful for engineering design. Due to the complexity of the problem, the scope of the present study is limited to a steady state numerical approach.

![Diagram](image)

(a) Laboratory, (b) simplified numerical model of Ward's tornado vortex chamber, and (c) boundary conditions

### 3.2 Methodology

#### 3.2.1 Numerical Model

For the present study, a numerical model mimicking the Purdue tornado simulator is used (see Fig. 3-1a). The actual Purdue tornado simulator is cylindrical in shape, where flow enters the simulator from the bottom part near the ground. Guide vanes are provided at the inlet to provide the 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 boost the vertical movement of swirling flow, an exhaust fan is installed at the outlet sucking out the air from the simulator. While keeping similar principle of operation, some modification is made to the simplified numerical model (see Fig. 3-1b). The simplified numerical model is also kept cylindrical in shape. Instead of providing the guide vanes, a computer code has been implemented to achieve the desired angle in the inflow. A “shear free” sidewall is provided and at the outlet “outflow” boundary condition is provided. The original laboratory simulator is about 2m in height which can produce scaled tornado-like vortex. However, in the present study, a full-scale simulation is targeted where the original simulator is scaled up following the procedure discussed in Refan et al. (2013) keeping the aspect ratio similar to the laboratory scaled model.

![Diagram showing dimensions](image)

**Figure 3-2. Dimensions for (a) full computational domain and (b), (c) hills**

<table>
<thead>
<tr>
<th>Location</th>
<th>Tornado center</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>At hill crest</td>
</tr>
<tr>
<td>2</td>
<td>At half-height of the hill</td>
</tr>
<tr>
<td>3</td>
<td>At foot of the hill</td>
</tr>
</tbody>
</table>
Numerical simulator used for present tornado simulation has the following dimensions $H_0=1730$ m and $R_0=1700$ m (See Fig. 3-1c and Fig 3-2a). The ground surface of the simulator is altered using a sinusoidal function (see Fig. 3-2b) to create the hills. Two different types of hills, steep and shallow, respectively are considered (see Figs 3-2b and 2c). Both stationary and translation motion of tornadoes are considered. For the stationary case, three different simulations corresponding to three different locations of tornado center with respect to the crest of the hill are considered by moving the tornado center along the $X$-axis. (see Fig. 3-3). More specifically the three locations represent cases when the tornado center coincide at the crest of the hill (Location 1), at the half height of the hill (Location 2) and at the foot of the hill (Location 3), respectively.

However, for the translation case, a single simulation where the tornado translates from Location 3 to Location 1 is conducted. In all simulation cases a commercial software STAR-CCM+ is used. 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 hills are kept very fine (wall $Y$-plus $< 2$) to capture the sharp velocity changes near the wall region (see Fig. 3-4a, b and c). Reynolds Average Navier-Stokes (RANS) equations are used together with the Reynolds Stress Model (RSM) for steady state simulation. RSM turbulence model is chosen over $k$-epsilon and other two equation models because of its better
accuracy to model 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 flows, such as tornadic flow, this particular assumption does not work because of the curvature effect.

Figure 3-4. Mesh distribution - finer mesh close to the ground (a), and around study

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 has 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:

\[ V_r = V_1 \times \left( \frac{z}{z_1} \right)^{1/7} \]  \hspace{1cm} 3-1

\[ V_t = \frac{2H_o}{R_o} \times S \times V_r \]  \hspace{1cm} 3-2

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 (See Fig 3-1b). \( H_o \) and \( R_o \) are the inlet height and radius of inlet respectively.
Reference velocity, $V_1$ and height, $z_1$ are chosen at 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.

### 3.2.2 Fractional Speed-up Ratio (FSUR) definition

Fractional Speed-Up Ratio (FSUR) is used typically to represent the flow change due to topography (Bitsuamlak 2004). For synoptic flow-field, it is defined as a $U(z)/U_o(z)$ ratio, where $U(z)$ is the velocity at height ‘$z$’ above the hill surface and $U_o(z)$ is the upstream velocity at the same height from the flat ground (see Fig. 3-5). Unlike synoptic flow, tornado has a very complex flow structure which consists of tangential, radial and vertical components. As a result, a new FSUR calculation method is proposed in a slightly different manner. For tornado, FSUR is obtained by $U'(z)/U_o'(z)$, where $U'(z)$ is the net velocity at height ‘$z$’ above the hill surface and $U_o'(z)$ is the net velocity at the same height at the same location inside the tornado but in the absence of the hill (see Figs. 3-6 and 7).

![Figure 3-5. FSUR calculation for synoptic flow](image1)

![Figure 3-6. FSUR calculation for tornadic flow over a hill](image2)
From preliminary analysis it was observed that there is more complexity that need to be addressed for tornadic flow. For example, in Figure 3-7 two horizontal line-probes are placed at same height (50m from the ground surface) for tornado on flat ground and on the hill. In Fig. 3-7b, downward vortex flow at the center of the tornado touches at the half height of the hill. Due to this, the symmetric distribution of the flow near the center of the tornado is disrupted. However, the tornado is formed quite symmetrically for flat ground (see Fig. 3-7a). As shown in Fig. 3-8, it is quite obvious that the overall flow distribution follows the Rankine Vortex Model, which is common with the rotating flows. In the presence of the hill, location of the maximum velocity occurs much closer to the center of the tornado (‘0’ in the figure). Keeping the location same, the velocity for the other case (tornado in a flat ground) is very small. As a result, evaluating FSUR using current methodology can produce a very high value due to a division by a small number issue. This is not representative of what is happening. Hence defining the FSUR by direct comparison of symmetric and unsymmetrical tornado-like vortex may suffer either division by a small number issues resulting in unrealistic high FSUR numbers. To avoid this issue, a new approach is adopted where the velocity profile for flat ground is intentionally off-set from its original position (Fig. 3-8a) while evaluating the FSUR ratio, so that the location of the maximum velocity for both the cases coincides as shown in Fig. 3-8b. The evaluation of the modified FSUR value starts from this maximum velocity location and continue along the line-probes. In this manner, a more representative FSUR value is generated for tornadic flows. This type of FSUR is used in the remaining of the study.
3.2.3 Grid Independence Test

Grid independency test is carried out for a single case of swirl ratio 0.4 and with building located at the core center. Two different grid densities are used. The coarse grid has a total number of grids 1.5 million while the fine grid had 4 million respectively. The maximum error maximum Cp was measured on the faces of the building was obtained under 2% indicating the simulations were independent of the grid size. The coarser of the two grids was adopted for the remaining of the study.

3.3 Results and Discussion

3.3.1 Tornado field validation

As already mentioned in this literature, for tornado simulation swirl ratio, $S = 0.4$ is used which represents an EF-2 scale tornado in nature (see Fig. 3-9). The ground pressure distribution, $C_p'$ is compared with numerical work of Natarajan et al. (2012) for swirl ratio $S = 0.4$ using following equation:

$$C_p' = \frac{P - P_0}{0.5\rho U_0^2}$$

Where $P$ is the pressure of the surface, $P_0$ is the maximum static pressure at the ground, $\rho$ is the density of air and $U_0$ is the reference velocity, which is average radial velocity at the inlet.
Figure 3-9. Vertical flow structure of tornado-like vortex having swirl ratio 0.4 (one-celled vortex)

Figure 3-10. Comparison of (a) ground Cp’ for S = 0.40 and (b) radial velocity along R/R₀ = 0.1025 for S = 0.28

Here, a different intensified tornado with a swirl ratio S = 0.28 is simulated in order to compare with experimental results of Baker (1981) and numerical results of Natarajan et al. (2012) as initial validation of the tornado field. The radial velocity is measured along
the normalized vertical line (normalized by the inlet depth $H_0$) at a radial distance where $R/R_0 = 0.1025$, where $R$ is radial distance from the core and $R_0$ is the radius of the computational domain as shown in Fig. 1b. The comparison shows a good match with the present study (see Fig. 3-10).

![Steep hill and Shallow hill comparison](image)

**Figure 3-11. Comparison of FSUR at tornado at Location 1 along X-axis**

![Steep hill and Shallow hill comparison](image)

**Figure 3-12. Comparison of Max. FSUR along X-axis**

### 3.3.2 FSUR comparisons

The FSUR values for tornado at Location 1 are shown in Fig. 3-11. Irrespective of the horizontal distance along the $X$-direction from the tornado center, the maximum FSUR occurs at around 20 m high from the ground for both the steep and shallow hills (see Fig. 3-11). From Fig. 3-12, it is can also be seen that the maximum FSUR is higher near the
crest of the hill irrespective of the type of the hill and then speed-up dies out for locations at the foot of the hill for both steep and shallow hill cases.

Figure 3-13. Comparison of different velocity components of tornadic flow, for tornado at Location 1 along X-direction

As tornado is a complex flow structure, it is important to analyze the variations of each component of the flow structure to understand the distributions of FSURs better. For the radial component of the flow, positive value indicates radially outward flow and negative value indicates radially inward flow. For the flat ground case, the inward radial flow becomes stronger as it moves closer to the core region and then near the vicinity of the core it changes its direction upward (Fig. 3-13b). However, inside the core region due to the vortex break down, downward flow closer to the ground changes its direction radially outward from the center. In the presence of the hill, this radially outward flow becomes
more prominent as slope of the hill enhances the transformation of vertical downward flow to radially outward flow (Fig. 3-13a). For the tangential direction of flow, the presence of the hill does not affect it significantly (Figs 3-13c and d). However, for vertical direction of flow, the presence of the hill, enhance both upward and downward direction of flow (Figs 3-13 e and f).

**Figure 3-14. Comparison of FSUR at tornado at Location 2 along X-axis**

**Figure 3-15. Comparison of FSUR and Max. FSUR at Location 2 (along X-axis)**

Likewise, the FSUR values for tornado at Location 2 are shown in Fig. 3-14. Overall, the speed-ups are higher along positive X-axis (which is the uphill zone) compared to negative X-axis (which is the downhill slope). Fig. 3-15 shows the maximum FSUR
distribution. Generally, the speed up values are in comparable order of magnitude with those seen in synoptic wind flows.

**Figure 3-16. Comparison of different velocity components of tornadic flow at Location 2 along X-axis**

The variations of each component of the tornado flow structure for tornado Location 2 are provided in Fig. 3-16. Along the negative X-axis, it is observed that the direction of flow is radially outward (see Fig. 3-16). This is due to the inclined surface of the hill, which actually disrupts the circular distribution of the flow. In addition, this also relocates the tornado center (see Fig 3-17).
Figure 3-17. Tornado center shift at Location 2

Figure 3-18. Comparison of FSUR and Max. FSUR for tornado at Location 3 along X-axis
The FSUR distribution for tornado at Location 3 is similar to the one at Location 2, except in this case, the difference in maximum FSUR distribution along uphill (positive X-axis) and downhill (negative X-axis) slope is less than when compared with tornado at Location 2 (Fig. 3-19).

![Figure 3-19. Comparison of FSUR and Max. FSUR at tornado Location 2 along Y-axis](image)

The FSUR values are higher along the positive Y-axis compared to the negative Y-axis. Again, this can be determined by analyzing different components of flow. From the radial velocity distribution, it is can be observed that the along negative Y-axis radial flow is inward, however along positive Y-axis the flow is outward (Fig. 3-20).
Figure 3-20. Comparison of different components of tornadic flow at tornado at Location 2 along Y-axis

Similar FSUR distribution is also observed for tornado at Location 3 (along Y-axis). However, for steep hill, speed-up increases around 2.5 times near the tornado center. This is due to the increase in upward velocity (see Fig. 3-21). Here upward velocity increases due to the presence of the downhill slope which is concave in shape and enhances the updraft strength (Fig. 3-22).
Figure 3-21. Comparison of FSUR and Max. FSUR for tornado at Location 3 along Y-axis

Figure 3-22. Vertical components of flow distribution for tornado at Location 3 along Y-axis
3.3.3 Comparison of Cp at the ground level

Pressure coefficient, $C_p$, on the ground is obtained using equation 3-3. Only in this case, the reference velocity is used as the maximum tangential velocity at the crest height of the hill in the absence of the hill. In Figure 3-23, for all the cases the plots are cropped to the size of the radius of the shallow hill to facilitate the visual comparison. For all the cases, the pressure distribution is similar where maximum suction (minimum $C_p$) occurs at the center of the tornado and decreases as it moves away from the center (see Fig. 3-23). This indicates, the slope of the hill does affect the overall pressure distribution on the ground slightly as shown in Figure 3-23.

![Image of pressure distribution](image)

**Figure 3-23. Ground Cp comparison between steep and shallow hills**
3.4 Conclusion

Based on the present numerical arrangements and selected governing parameters for
tornadic flow (swirl ratio 0.4 and aspect ratio = 1.025) following conclusion can be made,

- A pragmatic FSUR evaluation method has been developed
- With regards to FSUR values, the location of maximum velocity moves much closer
to the tornado center irrespective of the location of tornado with respect to the crest of
the hill with the presence of the hill (both steep and shallow).
- Irrespective of the location of the tornado with respect to the hill, speed-up occurs for
  all the cases.
- As tornado center coincides with the crest of the hill, the region of downward flow
  expands near the hill surface which increases the radial component of the flow.
- As tornado center coincides with the inclined surface of the hill, the downward flow
  becomes tilted which in turn agitates the symmetrical distribution of radial and
  vertical distribution of the flow.
- The slope of the hill does not affect the overall pressure distribution on the ground
  significantly. Only the location of maximum suction changes depending upon the
  location of tornado center.

3.5 Acknowledgement

The authors would like to acknowledge the financial support from the National Research
Council of Canada (NSERC), Ontario Center of Excellence, Hydro One, and Canada
Research Chair (for the second author). The authors are grateful for access to
SHARCNET (a high-performance computing facility) and support received from their
excellent technical support team.

References

1. Abdi, D. and Bitsuamlak, G.T. 2014: Wind flow simulations on idealized and real
   complex terrain using various turbulence models, Advances in Engineering Software,
   75, 30–41.
   University.


25. Lewellen, D.C., 2012: Effects of Topography on Tornado Dynamics. 26th Conference on Severe Local Storms, November 5-8, Nashville, TN, USA.

International Conference on Wind Engineering, June 2-5, Lubbock, Texas, USA, 2, 1149-1156.


Chapter 4

4 Computational Modeling of Tornado-like Vortex Induced Mean Pressure Distribution on a Typical Building with Flat Roof

4.1 Introduction

According to the National Centers for Environmental Information, in 2016 more than 320 confirmed tornadoes were reported in North America in 2015. The total cost of damage was estimated to be around a billion US dollars during the same year in North America alone. Over the years, researchers and scientists have been developing various methods to analyze the flow structure of tornadoes. For example, several laboratory scale models have been used to analyse the flow structure of tornado-like vortices. In 1972, Ward built a tornado vortex chamber (TVC) with geometric and dynamic similarity to a real scale tornado (Ward, 1972). Following this study, several laboratories scaled models have been built by various researchers to analyse different aerodynamic properties, such as those developed by Wan and Chang (1972), Davies Jones (1973, 1979, 1986), Mitsuta and Monji (1984), Haan et al. (2008), Haan et al. (2010); Matsui and Tamura (2009); and Refan et al. (2013). In parallel with these experimental efforts several numerical analyses have also been performed such as those by Harlow and Stein (1974), Rotunno (1977), Church et al. (1993), Nolan and Ferrell (1999), Lewellen and Lewellen (1997), Lewellen and Lewellen (2007), Hangan and Kim (2008), Hangan and Natarajan (2012), Nasir et al. (2014), and Nasir and Bitsuamlak (2014). However, there are limited studies reported in literature on tornado/building interaction. For example, Chang (1971) simulated a single celled tornado experimentally and analyzed the tornado pressure on a cube model for two different locations. Chang (1971) reported that the pressure distribution on the model is a combined effect of both suction and dynamic pressure. Mehta et al. (1976) estimated tornado wind speeds from the analysis of structural failure. Jischke and Light (1983) and Bienkiewicz and Dudhia (1993) modified Ward's laboratory model to study the effects of the location of object with respect to the tornado vortex. Wang et al. (2001) and Fouts et al. (2003) both reported that the pressure distribution changes rapidly with change in location of the model with respect to the tornado core. Mishra et al. (2003) compared the
pressure distribution due to tornadic vortex and atmospheric boundary layer flow in a cubic model. Sarkar et al. (2006) and Sengupta et al. (2006) used numerical (LES) and experimental methods to simulate transient loading due to tornado on a cubic building and reported that tornadoes produce higher peak loads than microburst and slow moving and smaller tornadoes produce higher peak loads. Their model was similar to the model presented in Fouts (2003). Also, Mishra et al. (2003) simulated a single celled tornado-like vortex and compared its flow field with an actual scale tornado of F4 intensity. Haan et al. (2010) analyzed the impact of tornadic load and atmospheric boundary layer in a low-rise building with gable roof and reported that the peak values of side forces exceeded the standard by up to a factor of 1.5 while the uplift exceeded the ASCE 7-05 provisions by a factor as high as 3.2, indicating that buildings designed based on ASCE-7-05 were possibly under designed. Some of the numerical studies on tornado-building interaction include the following. Selvam and Millet (2003) modeled the interaction between a tornadic vortex and cubic building using a large eddy simulation. They reported that a translating tornado produces higher overall forces on the side walls and roof of the building than quasi-steady wind.

One of the observations made based on these studies is that a scale factor is required to scale up a model scale tornado to full scale based on the flow characteristics. In the present study, a small- and full-scale investigation are carried out in parallel using the scaling-up method developed by Refan et al. (2013) to assess its efficacy. Aerodynamic parameters, such as surface and field pressure coefficients, are generated under different locations of building with respect to the core center. The empty domain and building located at the core center, core radius and outside the core are examined and compared with each other and with the synoptic flow case. Details about the computational domain, meshing and boundary condition are also provided.
4.2 Methodology

4.2.1 Numerical Model for Tornado Simulator and Bluff-body

![Diagram](image)

**Figure 4-1.** (a) Laboratory and (b) simplified numerical model of Ward's tornado vortex chamber

For numerical modeling, it is quite certain to depict a well-established experimental work. Although WindEEE Dome facility is currently available in Western University now, however, during the time of this study, WindEEE Dome was not fully prepared. As a result, for the present study, a simplified numerical model depicting Purdue Tornado Vortex Chamber (TVC) is used (see Fig. 4-1a). However, the numerical model is scaled-up using a scaling system developed by Refan et al (2013) for full scale simulation (see Fig. 4-1a). Besides, a numerical model depicting an exact scale of WindEEE Dome is used for validation purpose.

In Purdue’s TVC, an exhaust is provided at the top section (collection area) for exhaust the air from the simulator and directional vanes are provided near the floor for providing desired angle in the flow, so that it can create a swirling flow inside the simulator (convergent and collection area) (see Fig. 4-1a). Similar concept is used for the present CFD based numerical modeling after considering only convergent and convection area, that is enabled due to the ease of boundary condition application in the numerical domain.
as shown in Fig. 4-1b. For example, the guide vanes can be simply replaced with an inclined velocity inflow boundary condition.

4.2.2 Tornado Geometric Scaling Factor

![Figure 4-2. Determining scaling factor for small-scale numerical depicting Purdue’s TVC](image)

For the present simulations, a full scale numerical model is being used. As a result, a proper scale factor, to scale-up the modified numerical model, is required. The scale factor for the tornado simulation is chosen using the scaling technique developed by Refan et al (2013) based on experiments carried out at the mini WindEEE Dome. First, the maximum tangential velocity of simulated tornadoes is determined for various swirl ratio values. Then, the core radius and height corresponding to the location of maximum tangential velocities for simulated tornadoes at various swirl ratios are compared with actual tornado field measurements. Since there must be only one length scale ratio in fluid dynamics simulations, it is expected that both, the core radius 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 represents the target tornado is identified.

The target tornado data chosen is a one-celled tornadic event that took place in Happy, Texas in 2007. 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 Fig. 4-2
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 4200 (see Fig. 4-2).

4.2.3 Computational Domain and Bluff-body dimensions

Figure 4-3. Dimensions for (a) Computational domain and (b) Bluff-body; (c) Boundary conditions for simulation

The full-scale simulator is 8740m in height and the radius is 1700m (see Fig. 4-3a). A 20m by 20m bluff-body with 10m height is used to represent low-rise building (see Fig. 4-3b).

For meshing, octagonal mesh elements are used using commercial software STAR-CCM+. The total number of cells is around 4 million and finer meshes are provided at the ground surface (wall Y-plus < 2) of the domain and also around the surface of the building to capture the sharp changes in the velocity.
4.2.4 Boundary Conditions for Simulation

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:

\[ V_r = V_1 \times \left( \frac{z}{z_1} \right)^{1/7} \]  
\[ V_t = \frac{2H_0}{R_o} \times S \times V_r \]

where, \( V_r \) and \( V_t \) are the radial and tangential component of velocity at \( z \) height from ground surface respectively. \( V_1 \) and \( z_1 \) are the reference velocity and height respectively. Shear free sidewall is used. The boundary condition used for the small- and full-scale are similar, with the exception of the geometric size and magnitude of flow variables. Wall boundary conditions for all the sidewalls of the study building are considered (see Fig. 4-3c).

For the 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 (Refan et al., 2013). For both the cases, 10% of the turbulent intensity is used and the Radial Reynold number is of order of \( 10^5 \).

4.2.5 Turbulence Model Used for Simulation

For the numerical simulation, RSM (Reynolds Stress Model) turbulence model was used. This model is selected due to its better accuracy compared to the k-epsilon or other two equation models 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 flows like tornadic flow, this assumption is no more valid because of the strong curvature effect. On the other hand, Large Eddy Simulation (LES) model is not considered because of the parametric nature of the present study that makes it computationally costly for LES use.
4.2.6 Location of the study building with respect to the tornado core center

The tangential velocity starts from zero at the core center and peaks to some highest value and thereafter starts to decrease with the increase in the radial distance from the core center following Rankine’s Vortex Theory (Giaiotti and Stel, 2006) (see Fig. 4-4). The radial distance at which it reaches the maximum value is the core radius/core radius. For this research, three different study building locations with respect to the tornado core are considered. The first location is at the core center where the maximum tangential velocity is zero, the next location is at the core radius where the tangential velocity is the largest and the third location is at a distance 7 times larger than the distance from the core center to core radius where the maximum tangential velocity becomes more or less constant with the respect to the radial distance from the core center as shown in Fig. 4-4.

4.2.7 Grid Independence Test

Grid independency test is carried out for a single case of swirl ratio 0.4 and with building located at the core center. Two different grid densities are used. The coarse grid has a total number of grids 1.5 million while the fine grid had 4 million respectively. The maximum error maximum Cp was measured on the faces of the building was obtained under 2% indicating the simulations were independent of the grid size. The coarser of the two grids was adopted for the remaining of the study.
4.3 Results and Discussion

4.3.1 Flow Field Analysis

The CFD simulation captured details of the tornado-like vortices and their interaction with the study buildings. Flow field represented through the streamlines are generated both for the small- and full-scale study building located at each of the three locations, respectively.

For building located at the tornado core center (location 1), the flow field is symmetrical for both sections (A-A and B-B) as shown in Fig. 4-5. The weak downward flow inside the core region interacts with the roof top of the building and results on a separated flow that created recirculation zones on the four side walls of the building (see Fig. 4-5). As can be seen later, these interactions have little effect on the $C_p$ magnitudes for this location. The high suction produced by the high angular momentum dictates the overall magnitudes of the $C_p$. For Location 2, due to the anti-clockwise rotation of the tornadic flow, flow approaches the southward and eastward wall and separates and creates recirculation on the northward and westward wall. For the location 3, the flow-building interaction characteristics resembles to the location 2.
4.3.2 Tornado induced pressure coefficients acting on the building surfaces

Figure 4-6. Cp distributions a) along the entire computational domain, and surfaces of the building located b) at the core center (i.e. location 1), c) at the core radius (i.e. location 2) and d) outside the core (i.e. location 3) (S = 0.4)

Pressure distributions along the surfaces of the building are analyzed by evaluating the pressure coefficients. In this context pressure coefficient \( C_p \) was defined as

\[
C_p = \frac{P - P_0}{0.5 \rho U_0'^2}
\]

where \( P \) is the pressure of the surface, \( P_0 \) is the maximum static pressure at the ground, \( \rho \) is the density of air and \( U_0' \) is the reference velocity, which is maximum velocity observed at the building height but in the absence of the building. For tornado flow it is common to use the maximum velocity anywhere in the computational domain as a reference velocity, thus enabling the establishment of a relationship with, for example, the wind speeds prescribed in Enhanced Fujita Scale. However, in the present study the maximum horizontal velocity at the building’s roof-height (flat roof) is adopted to enable qualitative comparison with straight-flow induced pressures.

For the location at the core center, the pressure coefficients do not show significant variation in all surfaces as it is governed by the high angular momentum primarily, the flow-structure interaction results in very minor mean \( C_p \) variations ranging between -3.04 and -2.96 (small scale) and -2.95 and -2.87 (full scale) anywhere on the surface of the
building (see Fig. 3-6 location 1). However, for the locations at the core radius a wide range of mean Cp distributions is observed similar to what one would expect with synoptic flow-structure interactions. For the building location at the center of the core, the overall negative pressure is very high compared to the other two locations. This can be explained with the help of the pressure coefficient distribution measured at the ground surface of an empty the computational domain (see Fig. 4-7).

![Ground Cp distribution](image)

**Figure 4-7. Ground Cp distribution**
4.3.3 Comparison of tornadic Cp distribution with those induced by synoptic flow

![Computational domain for synoptic flow-building interaction](image)

**Figure 4-8.** Computational domain for synoptic flow-building interaction. Finer mesh (b) at the ground surface and around building (c) elevation view, and (d) plan view

For synoptic flow field, a laboratory scaled model of a building immersed in an atmospheric boundary layer is used. The dimensions of the computational domain are made identical to those used by Bitsuamlak et al. (2010) (see Fig. 4-8) and following the guidelines discussed in Dagnew and Bitsuamlak (2013). The building is located at distance 3H ($H =$ building height) from the inlet to capture the flow-structure in front of the building. The height of the domain is far from the influence of the building enough to have a uniform boundary layer profile. Reynold’s Stress Model (RSM) is also used for synoptic flow similar to the tornadic flow cases. All side walls and the top wall are made with symmetry boundary conditions to avoid the effect of friction from the walls. For the
outlet, outflow boundary condition was used. A typical 1/7 power law boundary layer profile (eqn. 4-4) was used at the inlet boundary representing an open exposure condition.

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

where, \( V_i \) is the inlet velocity at height \( z \). \( V_1 \) (29.43 m/s) and \( z_1 \) (at building height; 0.1m) are the reference velocity and roof height, respectively. A similar building 20m by 20m by 10m used for the tornado flow case is modeled at a scale of scale of (1:100) (compatible with the typical wind tunnel scales) as mentioned earlier. Around 3 million cells are provided for computation and with finer meshes provided near the ground surface and around the building. Likewise, tornado 10% turbulent intensity is used for synoptic flow case as well.
Figure 4-9. Cp distributions for: (a), (b) synoptic flow and for tornadic flow for locations at (c), (d) core center; (e), (f) core radius and (g), (h) outside core (S=0.4)

Tornado induced pressure coefficient magnitude and distributions on the building are compared with those induced by the synoptic flow qualitatively. For the ABL-flow, pressure coefficients along vertical center plane parallel to the flow and a vertical center plane perpendicular to the flow are extracted, i.e. Plane I-II-III-IV and Plane V-VI-VII-VIII shown in Fig. 4-9a and b, respectively. To compare the results qualitatively, pressure coefficients from tornadic flow are plotted along two planes, Plane 1-2-3-4 and Plane 5-
6-7-8, respectively as defined in Fig. 4-9c to h under tornado case. Tornado produces larger suction coefficients compared to ABL flow. Also, the distribution of the pressure coefficients on the surfaces of the building for tornado cases are quite different compared to ABL flow. Pressure coefficients produced by tornado do not show significant variations for the near core locations of study building when compared to the ABL induced pressure coefficients. The latter shows significant variation among the different surfaces and within each surface of the building. This difference has significant implication on loading cases for roofs or large walls. These differences also highlight the shortcomings in common practice where pressure coefficients developed under synoptic wind are used together with the tornado wind speeds to assess the tornadic performance. This approach will underestimate the tornadic loads, particularity for buildings located near the core regions of the tornado. A numeric example to the underestimation of the tornadic loads with these mixed synoptic \( Cp \) and tornado wind speed approach, the \( Cp \) values for a building located at the core region is provided as follows. The tornado \( Cp \) values both at the walls and roofs of the building are four times larger than the maximum synoptic wall and roof \( Cp \) values and their variation over their respected surface are also very different. Hence one cannot use the \( Cp \)’s interchangeably as it is often observed in common practice driven by the fact that very limited information is available for tornado aerodynamics.

The order of magnitude of the tornado \( Cp \) becomes comparable with those produced with synoptic wind for core radius region and beyond. Particularly, the tornado \( Cp \) magnitude and distribution shows high resemblance to the ABL flow for the building positioned at the core radius for the west-east flow. For this case, the effects of the horizontal tangential velocity are superimposed on the general tornado suction and the magnitude of the \( Cp \) in Fig. 4-9e is shifted towards negative pressures when compared to Fig. 4-9a. For building location outside the core, irrespective of the location of the wall or roof, the \( Cp \) remains quite similar and the suction on the walls are quite similar, as here the ground suction due to the angular momentum of the flow becomes very small.
4.4 Validating with WindEEE

To best of our knowledge, WindEEE Dome is the most advanced tornado simulator. Both stationary and translating tornadoes can be produce in this facility. A maximum of 2 m/s translation speed can be obtained and roughness can be added to the experiment by raising the metal blocks on the ground.

4.4.1 Numerical Model for WindEEE Simulator

![Diagram of WindEEE Simulator](image)

**Figure 4-10.** (a)Cross-section of WindEEE Simulator; (a) 3-Dimensional, (b) Plan and (c) Front view of WindEEE Simulator Numerical model

In WindEEE simulator, for producing tornadic flow, 6 exhaust fans located over the bell-mouth opening area operated simultaneously at the same speed to exhaust the air from the inside of the dome and to redirect the flow towards the inflow region where guide vanes are installed to provide the desired angle in the flow before entering the internal hexagonal zone. The flow passes from lower region to upper from through the bell mouth. (see Fig. 4-10).
For the numerical model, a close circuit computational domain is used for duplicating the WindEEE simulator exactly. All the boundary conditions are kept identical with the WindEEE simulator except at the inflow region a directional vector boundary condition is used instead of guide vanes. For the present simulation, a swirl ratio of 0.50 is used which is achieved by changing the angle of directional vector by $15^\circ$ with respect to the radial direction of the flow at the inflow. The Radial Reynolds Number, $Re$, is of the order $10^5$ for keeping the flow turbulent so the radius of tornado core remains independent of the Radial Reynolds Number. Flow-structure of such tornadic flow in WindEEE Dome is shown in Figure 4-11.
4.4.2 Simplified Numerical Model

![Simplified Numerical Model Diagram]

**Figure 4-12. Simplified Numerical Model**

Similar to the previous case (Sec. 3.2.1), a cylindrical shape simplified model is considered here (see Fig. 4-12). Although, the dimensions are changed as it is simplified over the WindEEE Dome Tornado Simulator.

![Flow-structure inside WindEEE Simulator Diagram]

**Figure 4-13. Flow-structure inside WindEEE Simulator**

In order to simplify the actual WindEEE simulator it is important to identify the convergent region because, in WindEEE simulator, flow enters inside the simulator.
distant away from the convection region. In order to correctly obtained the depth of the convergent region, vertical velocity is measured along the vertical line (red line) from the ground to the bell-mouth at the location of the outer edge of the bell-mouth (see Fig. 4-13).

![Figure 4-14. Vertical velocity distribution](image)

It is observed that, up to around 2m from the ground, change in vertical velocity is very small (see Fig. 3-14). Based on this it can be seen up to this height, the dominating flow is horizontal flow which in turn is considered as the convergent region. Over this height, the region is convection region. Due to this, where WindEEE simulator has the aspect ratio of 0.35 while for the simplified version it is 1.

### 4.4.3 Comparing the Ground Pressure Distribution

In order to validate the modified numerical model numerical model of WindEEE, ground pressure distribution is compared. For the present purpose, pressure coefficients are measured using the same equation (see Eqn. 4-3), only in this reference velocity is the average tangential velocity at the location of core radius considered.
The ground pressure distribution obtained from the two numerical models is very similar. The peak ground pressure is also identical.

4.4.4 Dimension and Location of low-rise building

For both the experimental and Numerical studies, a bluff-body of cross-section 40m by 30m and height 34m is considered. A 1:200 scale factor is obtained for both experimental and numerical modeling. Two different locations of tornado are considered as shown in
the figure 4-16. At location two, again two different orientation of the bluff-body is considered due to the rectangular cross-section of the bluff-body.

4.4.5 Comparing the Pressure Coefficient Distribution

![Figure 4-17. Comparing External Cp using (a) WindEEE Numerical and (b) Simplified Numerical model and (c) WindEEE Simulator for location 1](image)

From the overall pressure distribution, it can be observed that, higher suction is obtained for numerical model when compared to experimental model. This is due to the flexibility of capturing more intense data due to the computation advancement. However, in experimental it is only limited to provide pressure tabs on some specific spots. Another important observation is that, for numerical models, overall pressure distribution is almost identical (high suction). Now for, experimental case, on two of the sidewalls suction is lower than top and other two sidewalls. Also, these two walls are located side by side. The reason for such asymmetric distribution could be due to some mechanical errors. At WindEEE, simulator, the six exhaust fans in the upper plenum are expected to operate at same speed, however this is hard to achieve in reality due to mechanical errors. For this experiment, WindEEE was used in one of the two modes, where the fans at the inflow are not operated and only the angle of flow is controlled by the louvers at the inflow. However, the flow tends to rotate the inlet fans (that are not supposed to be rotating in this mode of operation) causing an undesired vorticity in the inflow. In addition to this, since the rotation of the inlet fans is caused by the flow itself, every fan
rotates at different speeds in an uncontrolled manner, leading to an inherent asymmetry in the inflow. These factors lead to the movement of the vortex back and forth about a mean position (even in the stationary vortex case) and offsets it from the center of the building, causing a slight discrepancy in the computational and experimental results (see Fig. 4-17).

Figure 4-18. Comparing External $C_p$ using (a) WindEEE Numerical and (b) Simplified Numerical model and (c) WindEEE Simulator for location 2; Orientation 1

Unlike, location 1, at location 2; Orientation 1 (L2O1) the pressure distribution is quite similar between numerical models and experimental model. Along southward and eastward walls suction is lower due to the anticlockwise rotation of the flow which provides an incoming angular flow towards these walls. This incoming flow gets separated and creates recirculation on the north and westward walls, as a result, suction is higher on these walls (see Fig. 4-18).
Likewise, location 2, the pressure distribution is also identical between experimental and numerical models for location 3. That is justified, as only the orientation of the building is changed.

4.5 Conclusion

The following observations are made as a result of the present comparative CFD based study between tornado-like vortex and synoptic wind and then interaction with a building with flat roof.

(i) The variation on the global tornado pressure field near the ground surface dominated the $C_p$ distribution patterns and their magnitudes compared to those induced through wind-building interaction, particularly near the core region.

(ii) For the building location at the core of the tornado, suction pressure coefficient is highest with a factor of 4 when compared to those $C_p$ produced for synoptic wind.

(iii) The tornado pressure field is not affected significantly by the presence of the building

(iv) For building location at the core radius location and beyond, the flow structure shows significant similarities with ABL flow case but are shifted by the tornado

Figure 4-19. Comparing External $C_p$ using (a) WindEEE Numerical and (b) Simplified Numerical model and (c) WindEEE Simulator for location 3; Orientation 2
pressure differential. The order of magnitude of $C_p$ for these regions (i.e. beyond the core radius) is comparable with synoptic wind induced $C_p$. 
References


Chapter 5

5 Numerical Modeling of Tornadic Load on a High-rise building

5.1 Introduction

Tornado is one of the most devastating natural peril. Each year 800-1000 tornadoes form on average, in North America alone, causing significant fatalities, injuries, and property damages worth of billions. Various studies are reported in literature that attempt to analyse the flow structure of tornadoes of different intensities. Most of these studies are limited to scaled-down laboratory based and numerical modeling. Some of the previous works 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 significantly 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. Selvam and Millet (2003) analyzed 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 quasi steady wind. Sarkar et al. (2006) made a comparison of tornadic loads with ASCE 7-02 and found that tornadoes of intensity F2 or higher exceeded the ASCE 7-02 design load. Sengupta et al. (2006) used numerical (LES) and experimental methods to simulate transient loading due to tornado on a cubic building and reported that tornadoes produce higher peak loads than microburst and slow moving and smaller tornadoes produce higher peak loads. Hangan and Kim (2008) simulated tornado of different intensities to find the dependency of flow dynamics on swirl ratio and relation between swirl ratio and Fujita Scale. Haan et al. (2010) analyzed the impact of tornadic load and atmospheric boundary layer in a low-rise building with the gable roof and reported that the peak values of side forces exceeded the standard by up to a factor of 1.5 while the uplift exceeded the ASCE 7-05 provisions by a factor as high as
3.2, indicating that buildings designed based on ASCE-7-05 were possibly under designed. Yang et al. (2011) analyzed the impact of tornadic load over a tall building for a laboratory based model and reported that wind loads were maximum when the building was placed in the outer boundary of the tornado-like vortex. They also observed that wind loading on the high-rise building test model due to tornadic flow was dependent on the orientation of the building and reached a maximum value at an orientation angle of about 30° and 45°. Natarajan and Hangan (2012) analyzed the impact of translation and surface roughness over a tornadic structure and concluded that translation reduces the maximum mean tangential velocity for lower swirl ratios and slightly increases the maximum mean tangential velocity for higher swirl ratios. Furthermore, surface roughness was reported to cause an effect similar to reducing swirl ratio. More recently Refan et al. (2014) developed a consistent scaling method through the mini WindEEE Dome experiments and field measurements. Building up on these prior studies, the current investigation presents a numerical assessment of tornadic loads over a tall building.
Figure 5-1. Ward tornado chamber (a) Laboratory and (b) simplified numerical model of Ward's tornado vortex chamber, (c) cross-section of WindEEE simulator, (d) isometric, (e) plan and (f) front views of the WindEEE inspired numerical model

5.2 Methodology

5.2.1 Numerical Model for Tornado Simulator

For the present study, two numerical models are used. The first one depicts the Purdue tornado simulator is used (see Fig 5-1a-b) and the second depicts the WindEEE wind testing chamber at Western (see Fig 5-1c-f). The actual Purdue tornado simulator is cylindrical in shape, where flow enters 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 exhausts out the air from the simulator. Keeping the main operational mechanism in mind, some modification is made into the simplified numerical model (see Fig 5-1b). For WindEEE simulator numerical model, all aerodynamically
influential elements of the WindEEE Dome are models to define the computational domain as shown in Figure 5-1-d-f.

![Diagram of CAARC model dimensions]

**Figure 5-2. Dimensions of CAARC model**

The study building is a standard high-rise building based on the Commonwealth Advisory Aeronautical Research Council (CAARC). The CAARC building is rectangular with dimensions 30.48m (width) by 45.72m (length) by 188.82m (height).

### 5.2.2 Geometric Scaling for Full-Scale Simulation

As already mentioned in the introduction that one of the main features of this current study is the full-scale numerical modeling. Thus, 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 Fujita scale-2 (F-2) tornado is chosen that took place in Happy, Texas in 2007. The reason for choosing such an F-2 scale tornado is that, in nature about 90% tornadoes are F-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 5-3 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 0.5 at scale factor of 4200 (see Fig. 5-3) for the target tornado case.

![Figure 5-3. Determining scaling factor for small-scale numerical depicting Purdue tornado simulator](image)

5.2.3 Meshing for Tornado Simulation

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 Fig. 5-4a, b and c).

![Mesh distribution: finer mesh close to the ground (a), and around study building plan view (b) and elevation view (c)](image)

**Figure 5-4.** Mesh distribution: finer mesh close to the ground (a), and around study building plan view (b) and elevation view (c)

### 5.2.4 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:

\[
V_r = V_1 \times \left( \frac{z}{z_1} \right)^{1/7} \tag{5-1}
\]

\[
V_t = \frac{2H_0}{R_o} \times S \times V_r \tag{5-2}
\]

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 and defined by \( S = \tan \theta / 2a \); ‘\( \theta \)’ is the inflow angle at the inlet and ‘\( a \)’ is the aspect ratio. For the present simulations, swirl ratio is being fixed to 0.5 to resemble an actual tornado of EF-2 scale. ‘\( H_o \)’ and ‘\( R_o \)’ are the inlet height and radius of inlet respectively. Reference velocity, ‘\( V_1 \)’ and height, ‘\( z_1 \)’ were chosen 0.3m/sec and 0.025m respectively for the laboratory scale model in line with an experimental research by Baker (1981). However, for the current full-scale numerical model, reference ‘\( V_1 \)’ velocity, and ‘\( z_1 \)’ height, 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.

5.2.5 Turbulence Model for Simulations

A 3D computational modeling is adopted for numerical simulation using Reynolds Averaged Navier-Stokes (RANS) equations and Reynolds Stress Model (RSM) for turbulence closure 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 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.

5.2.6 Grid Independence Test

Grid independency test is carried out for a single case of swirl ratio 0.4 and with building located at the core center. Two different grid densities are used. The coarse grid has a total number of grids 1.5 million while the fine grid had 4 million respectively. The maximum error maximum \( C_p \) was measured on the faces of the building was obtained under 2% indicating the simulations were independent of the grid size. The coarser of the two grids was adopted for the remaining of the study.
5.3 Results and Discussion

5.3.1 External Pressure Distribution

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

$$ C_p = \frac{P - P_0}{0.5 \rho U_0^2} $$

Here, $P_0$ is the maximum pressure at the ground, $\rho$ is the density of air and $U_0$ 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.

Figure 5-5. External Cp distribution (Orientation 1)
As the building is rectangular, 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 5-4 and 5-5).

Irrespective of the orientation of the building and the flow structure around the building, suction dominates the pressure field along the outer surface of the building for all three locations. Also, overall suction along the outer surface of the building is maximum for location 1, and this suction decreases as the tornado center moves away from the building (Location 2 and 3). Based on this pressure type and overall pressure distribution, it is quite obvious that, ground suction due to the angular momentum of the tornadic flow dominates the overall pressure distribution along the outer surface of the buildings for any location considered here (see Fig. 5-6)
At location 1, along the outer surface of the building, suction is almost identical. This is due to the very little dominancy of the flow-structure on the vicinity of the building. At tornado center, the only dominating flow is the very weak downward flow (see Fig. 5-8),
Figure 5-9. Flow-structure at Location 2
At location 2, where center of the building coincides with the core radius (location of maximum tangential component of the tornadic flow), approaching flow is more oblique towards the building near the ground and more directed towards a particular wall as it moves away from the ground (see Fig. 5-10). As a result, suction is lowest on the southward eastward face of the building near the ground and near the roof it is lowest on the southward face of the building for both orientations considered here. (see Fig. 5-7).

At location 3, where tornado core center is some distance away from the center of the building, as a result overall suction on all the surfaces are very small. In comparison with location 2, the approaching flow towards the building is steeper (almost 45°). Similar to location 2, recirculating flow occurs on the northward and westward force (see Fig. 5-11).
Figure 5-10. Flow-structure at Location 3
5.3.2 Comparing Pressure Distribution Using Numerical Model of WindEEE Dome Simulator

In order to validate the simplification process used for obtaining the simplified numerical model, results from actual WindEEE numerical model and simplified numerical model is compared. To do so, WindEEE numerical model, a close circuit computational domain is used for duplicating the WindEEE simulator exactly. All the boundary conditions are kept identical with the WindEEE simulator except at the inflow region a directional vector boundary condition is used instead of guide vanes. For the present simulation, a swirl ratio of 0.50 is used which is achieved by changing the angle of directional vector by 15° with respect to the radial direction of the flow at the inflow. The Radial Reynolds Number, $Re_r$ is of the order $10^5$ for keeping the flow turbulent so the radius of tornado core remains independent of the Radial Reynolds Number.

![Simplified Numerical Model based on WindEEE Simulator](image)

**Figure 5-11. Simplified Numerical Model based on WindEEE Simulator**

Based on the dimension of the WindEEE numerical model a simplified numerical is designed of the same geometrical scale factor.
Figure 5-12. Comparing $C_p$ along $\text{(a, a', a''); e, e', e'')$ northward; $\text{(b, b', b''); f, f', f'')$ westward, $\text{(c, c', c''); g, g', g'')$ southward and $\text{(d, d', d''); h, h', h'')$ eastward walls for location 1 and 2

Same CAARC building model of 1:200 scale is used to represent a high-rise building. Two different locations are considered here. At location 1, center of tornado coincides with the center of the building and for location 2, center of the building is at the core radius of tornado. In addition to this, two different orientation of the building are also considered at location 2 (see Fig. 5-12).
Likewise, the case of simplified numerical model, overall pressure distribution along the wall surfaces are dominated by the ground Cp distribution and the location of tornado. Higher suction is obtained for location 1 and decreases as it moves further away from tornado center. On contrary to the previous comparison of Cp at location 1, in present case the pressure distribution is less identical over the surface due to the relative size of the building with respect to the simulator size. In this case building is relatively bigger compare to the simulator size, as a result the presence of the building is distracting the pressure distribution more. Now at the Location 2, irrespective of the orientation of the building, the along the southward and eastward walls suction is lower than other walls due to the anti-clockwise rotation of the flow. This phenomenon is similar to the previous case.

5.3.3 Comparing the Force and Moment Coefficients

![Diagram of forces and moments](image)

**Figure 5-13. Determining Force and Moment Coefficient**

Both Force and Moment coefficient are measured along all three directions (X, Y and Z axis) for all the locations of tornado and orientation of the tall building (see Fig. 5-13).
Irrespective of the orientation of the building, it is quite obvious that, the anti-clockwise rotation of tornado intends to push the building toward tornado core and in the tangential direction of flow. In addition to this it also tries to lifts off the building from the ground. This lift becomes more prominent as the tornado center moves closer to the building, which also indicates the dominancy of the ground suction. For location 1 and 3, the orientation of the building does not affect the force and moment coefficient. In addition to this at location 1 only coefficient along Z direction which indicates lift is more dominant. However, at location 2 the variation is quite notable due to the presence of high tangential velocity in the tornadic flow. At this location, for orientation 1, where the longer edge of the building is along the X-axis, the force coefficient along X-axis is smaller compare to that for orientation 2. This clearly indicates, longer edge of the
building clearly lessens the effects of the tornadic flow. In addition to this, the building is more intended to bend along Y-direction than X-direction. This is due to the more dominancy of tangential direction of flow over radial direction of flow (see Fig. 5-14).

5.4 Conclusion

Based on current study, following remarks can be obtained,

- Irrespective of the location of tornado center with respect to the building and also the orientation of building, ground suction dominates the overall pressure distribution along the outer surface of the building.
- At location 1, suction is quite identical along all the surface of the building due to the weak downward flow at the tornado center which does not disrupts even slightly the pressure distribution due to ground suction.
- At location 2, higher anti-clockwise rotation of the flow decreases the suction on the windward wall (southward) and creates recirculation on the leeward walls (northward and westward) which in turn increases the suction.
- At location 3, the approaching flow is steeper than location 2. As a result, the southward and eastward walls have less suction than northward and westward walls due to the inward flow.
- The variation in overall suction with respect to the location of tornado is identical between WindEEE and Modified Numerical model. However, the pressure distribution along the surface of the body is quite different due to the geometrical scaling difference between the two models.
References


Chapter 6

6  Tornado-like vortex induced pressure on a bluff-body with and without openings

6.1 Introduction

Tornadoes are one of the violent natural disasters that cause fatalities and losses in millions of dollars in property damages every year in North America. More than 320 confirmed tornadoes were reported in North America in 2015. The cost of damages has been estimated to be close to a billion US dollars during the same year in North America alone. Because of the extent of tornado induced fatalities and damages, it is important to develop methods to model the structure of tornado-like vortex and its impact over engineering structures. Several scaled laboratory model versions were developed and used by researchers for analyzing the flow structure of tornadoes (Davies-Jones, 1972; Ward, 1972; Wan and Chang, 1972; Mitsuta and Monji, 1984; Diamond and Wilkins, 1984; Haan et al., 2008, 2010; Matsui and Tamura, 2009; Tari et al., 2010; Refan et al., 2013). In parallel to laboratory scaled modeling, several numerical studies are carried out (Rotunno, 1977; Church et al., 1993; Lewellen and Lewellen, 2007; Hangan and Kim, 2008; Hangan and Natarajan, 2012). Although most of the analyses were focused on studying the tornado flow structure, there are only few studies on impact of tornadic loads over engineering structures (i.e. tornado/structure interaction). Chang (1971) simulated a tornado experimentally and assessed the pressure distribution over a cubic building for different locations. This study reported that pressure distribution on the faces of the building is a combined effect of the near ground pressure field and dynamic pressure due to the tornadic flow interaction with the building. Wang et al. (2001) and Fouts et al. (2003) both reported that tornadic pressure distribution on an object changes rapidly for different locations with respect to the core center. Sengupta et al. (2006) used numerical (LES) and experimental methods to simulate transient loading due to tornado on a cubic building and reported that tornadoes produce higher peak loads than microburst and slow moving and smaller tornadoes produce higher peak loads. Mishra et al. (2003) simulated a single celled tornado for different locations of a cubical model and compared its results with an F4 scale tornado. Others used post damage assessments to
estimate wind loads. For example, Mehta et al. (1976) estimated the wind speed from a structural failure due to tornadoes.

Some of the numerical studies on tornado-structure interactions include the following: Selvam and Millet (2003) compared the results between translating and stationary tornadoes using LES (Large Eddy Simulation) turbulence model and observed that all the faces and roof of the building experience higher pressure for translating tornado than stationary one. Nasir et al. (2014), used RANS turbulence model for analyzing the pressure distribution along the faces and roof of a rectangular bluff-body for different location with respect to the core center and observed that the suction was higher for core center location. They also obtained that the pressure distribution was dominated by the tornado pressure field close to the ground near the core center.

In parallel to these studies, some efforts have been made in recent years to analyze the impact of tornadic load on a building with opening/openings. Thampi et al. (2011) studied tornado-structure interaction for a one storey gable roof building using finite element analysis to predict the successive damage stages caused by a translating tornado and concluded that leakage and openings affect the net load and can be crucial for alleviating tornado induced damage. However, openings are path ways to water intrusion and needs to be avoided. Kikitsu et al. (2011) analyzed the internal pressure of a building under the influence of a stationary tornado in a large tornado simulator. Sabareesh et al. (2012) analyzed the internal pressure due to tornadic winds using the experimental set up at Tokyo Polytechnic University. They concluded that mean and minimum internal pressure were higher for a building with single opening placed within vortex when compared to a building with multiple openings of the same opening ratio, placed at the same location and that this behavior is reversed when the building is placed outside the vortex. Letchford et al. (2015) analyzed the internal pressures which were obtained from the external pressures measurements using the analytical model proposed by Holmes. Building up on these earlier experiences, the current study presents a numerical assessment of tornadic load over a low-rise building for sealed and unsealed cases, and two opening types (single and cross-opening).
6.2 Methodology

6.2.1 Numerical model for tornado simulator

In this study, a simplified cylindrical shape computational domain at full-scale dimensions is used which mimics the scaled-up version of the Purdue Tornado Vortex Simulator (TVC), (Church et al. 1993) (see Fig. 4-1a). This Purdue TVC has been originally modified from the Ward’s Tornado Vortex Chamber (TVC) which has already been validated for simulating aerodynamically stable scaled tornadoes. The actual Purdue TVC is cylindrical in shape, where flow enters the simulator from the bottom. Guide vanes are implemented at the inlet to provide the 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 facilitate this vertical movement of swirling flow, an exhaust fan is installed at the outlet to exhaust out the air from the simulator. Keeping the main operational mechanism in mind, some modification is made in the numerical model (see Fig. 4-1b). The modified model is also cylindrical in shape. Instead of simulating the guide vanes, a suitable inflow boundary condition is implemented to achieve the desired flow angle in the inflow. A “shear free” sidewall is provided and at the outlet, “outflow” boundary condition is provided.

![Figure 6-1. (a) Laboratory and (b) simplified numerical model of Ward's tornado vortex chamber](image-url)
For bluff-body, a short square cross-sectional building is chosen. Besides, three different opening cases are considered; (i) sealed case (no-opening), (ii) unsealed case with single-opening, and (iii) unsealed case with cross-openings.

6.2.2 Tornado simulator and bluff body dimensions

The full-scale computational domain (CD) is shown in Figure 4-2a. The size of the CD is approximately 4200 times larger than the original laboratory tornado simulator. This scaling relationship can be obtained by comparing the location (core-radius and core-height) of the maximum tangential velocity point in an actual tornado and simulated tornado in a laboratory simulator as discussed in more detail by Refan et al. (2013). For the present study, Geometrical scaling ratios are obtained by comparing the core-radius and core-height between simulated tornado and an actual tornado which took place in Happy County, Texas (see Fig. 4-2a).

![Diagram showing geometric scaling factor and computational domain](image)

**Figure 6-2.** (a) Geometric scaling factor and (b) the computational domain, and the study bluff body for (c) sealed and (d) unsealed cases.

The full-scale simulator for the present study is 8740m in height and 1700m in radius. The depth of inlet flow is 1730m (see Fig. 4-2b). The study bluff-body is a rectangular building with a 20m by 20m by10m dimension (see Fig. 4-2c).
Opening configurations, a porosity of 3% of the wall area is considered. More specifically, the dimension of the opening is 3.46m by 1.73m. The opening is provided on the center of upper portion of the wall located 6.64m above the ground level (see Fig. 4-2d). Four different orientations of the bluff-body with respect to the tangential direction of the flow for single, and two different orientations for cross-opening cases have been considered respectively (see Fig. 4-4).

<table>
<thead>
<tr>
<th>Sealed Case</th>
<th>Single-Opening Case</th>
<th>Cross-Opening Case</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Sealed Case Diagram" /></td>
<td><img src="image2" alt="Single-Opening Case Diagram" /></td>
<td><img src="image3" alt="Cross-Opening Case Diagram" /></td>
</tr>
<tr>
<td>$O_1$: Orientation 1</td>
<td>$O_2$: Orientation 2</td>
<td>$O_1$: Orientation 1</td>
</tr>
<tr>
<td>$O_2$: Orientation 3</td>
<td>$O_3$: Orientation 4</td>
<td>$O_2$: Orientation 3</td>
</tr>
</tbody>
</table>

- $L_1$: Location 1; center of bluff-body is at the tornado core center
- $L_2$: Location 2; center of bluff-body is at the core radius
- $L_3$: Location 3; center of bluff-body is outside the tornado core
- $O_1$: Orientation 1; Opening is perpendicular to the tangential velocity
- $O_2$: Orientation 2; Opening is parallel to the tangential velocity
- $O_3$: Orientation 3; Opening is perpendicular to the tangential velocity (opposite to $O_1$)
- $O_4$: Orientation 4; Opening is parallel to the tangential velocity (opposite to $O_2$)
For the present simulations, commercial software STAR-CCM+ is used. Although, the base mesh size is kept comparatively coarse in most of the CD because of the large domain size, the mesh size near the ground and around the bluff-body are kept very fine (wall Y-plus < 2) to capture the sharp velocity gradient changes near the solid region (see Fig. 4-5a, b and c).

**Figure 6-5. Employed mesh (a) computational domain, and around the study bluff-body (b) elevation view and (c) plan view.**

### 6.2.3 Boundary conditions for tornado simulator and bluff-body

At the inlet, the flow velocity has two components (radial and tangential) in order to produce the swirling flow field. The radial and tangential velocity components are given by equations (1) and (2), respectively.

\[
V_r = V_1 \times \left( \frac{z}{2l} \right)^{1/7} \\
V_t = \frac{2H_0}{R_0} S \times V_r
\]
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 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 at the inlet boundary are chosen respectively so that they produce velocity similar to those obtained from the tornadic event that took place in Happy, Texas in 2007 (39 m/s) close to the study building. An exhaust fan at the outlet of the laboratory model is replaced by the outflow boundary condition in the numerical model.

### 6.2.4 Turbulence Modeling

Due to the limitation in computational power and a large number of cases to be considered, steady state condition of the flow is opted. Therefore, for stationary tornado simulations Reynolds Averaged Navier-Stokes (RANS) equations are used along with Reynolds Stress Model (RSM) turbulence model for all the current simulations. RSM model is chosen over k-epsilon and other two equation models because of its better accuracy for rotating flows. However, for translating tornado LES (Large Eddy Simulation) is used. For LES, a second order implicit time stepping with delta time, $\Delta t = 0.0001s$ is used and the translational speed of the tornado is 15 m/s (in actual tornadoes translational speed varies between 10-20 m/s) (Natarajan and Hangan, 2012).

### 6.2.5 Grid Independence Test

Grid independency test is carried out for a single case of swirl ratio 0.4 and with building located at the core center. Two different grid densities are used. The coarse grid has a total number of grids 1.5 million while the fine grid had 4 million respectively. The maximum error maximum $C_p$ was measured on the faces of the building was obtained under 2% indicating the simulations were independent of the grid size. The coarser of the two grids was adopted for the remaining of the study.
6.3 Results and Discussion

6.3.1 Tornado Field Validation

For tornado simulation, a swirl ratio, $S = 0.4$ is used that represents an EF-2 scale tornado (see Fig. 4-6). The ground pressure distribution, $C_p'$ is compared with numerical work of Natarajan et al. (2012) for swirl ratio $S = 0.4$ using equation (4-3).

$$C_p' = \frac{P - P_0}{0.5\rho U_0^2}$$

Figure 6-6. Vertical flow structure of tornadoes having swirl ratio 0.4 (one-celled vortex)

where $P$ is the pressure of the surface, $P_0$ is the maximum static pressure at the ground, $\rho$ is the density of air and $U_0$ is the reference velocity, which is average radial velocity at the inlet. This reference velocity has been used only to compare with literature. For the rest of the paper a different type of reference velocity is used as discussed in the next section. The ground $Cp$ shows a good match with the present study (see Fig. 4-7a). In addition, a different tornado with a swirl ratio $S = 0.28$ is simulated to allow direction comparison with the experimental results of Baker (1981) and the numerical results of Natarajan et al. (2012) as initial validation of the tornado field.
Figure 6-7. Comparison of (a) ground $C_p$ for $S = 0.40$ and (b) radial velocity along $R/R_0 = 0.1025$ for $S = 0.28$

The radial velocity is measured along the normalized vertical line (normalized by the inlet depth $H_0$) at a radial distance where $R/R_0 = 0.1025$, where $R$ is radial distance from the core and $R_0$ is the radius of the domain is shown in Figure 4-7b.

6.3.2 Pressure Coefficient Distribution

Pressure distributions along the surfaces of the bluff-body are analyzed by evaluating the pressure coefficients. In this context pressure coefficient $C_p$ is evaluate by using equation 4-3. However, in this case the reference velocity is taken as the maximum velocity anywhere on a horizontal plane located at the bluff-body height but in the absence of the bluff-body. For tornado flow, it is common to use the maximum velocity anywhere in the computational domain as a reference velocity, thus enabling the establishment of a relationship with, for example, the wind speeds prescribed in Enhanced Fujita Scale. However, in the present study the maximum velocity at the roof-height of the bluff-body (flat roof) was adopted to be able to qualitatively compare with synoptic wind loads.
6.3.3 Internal Pressure Distribution

Internal pressures are monitored along lines inside the bluff body and running parallel to the walls but located at a distance equal to the quarter of the dimension of the bluff-body away from the walls (see Fig. 4-8). Overall, suction develops for all the considered location of tornado center with respect to the bluff-body irrespective of the orientation and number of the opening on the bluff-body. The high suction effect occurs due to the angular momentum of the tornadic flow near the ground surface. This dominates the overall pressure distribution on the surface of the bluff-bodies that drives the internal pressure. Though this effect still remains dominant at all locations, its effect reduces as

Figure 6-8. Internal pressure distribution
the bluff body is located further from the tornado center. For location 1, suction is higher for cross-opening case than single-opening case. At this location downward flow moves towards the roof of the building and creates recirculation zones around the sidewalls of the bluff-body. This recirculation increases the internal pressure (i.e. suction) more in cross-opening case compared to single-opening case due to the higher number of openings subjected to a similar suction phenomenon. For location 3, suction is almost identical for all the cases and orientations of the bluff-body. However, for location 2, where the center of the bluff-body coincides with the core radius (location of maximum tangential velocity), the radial component of the flow is also very strong. As a result, the overall horizontal flow is strong enough to disrupt the overall internal pressure distribution based on the orientation and the number of openings on the bluff-body.

<table>
<thead>
<tr>
<th>Single-opening</th>
<th>Cross-opening</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
</tr>
<tr>
<td><img src="image3.png" alt="Image" /></td>
<td><img src="image4.png" alt="Image" /></td>
</tr>
</tbody>
</table>

**Figure 6-9. Flow structure at location 2**

From the figure, it is observed that suction is highest for orientation O₃ and O₄ and suction lowest for orientation O₁ and O₂ for single opening case as these openings were
located on the “leeward” side of the local flow. (see Fig. 4-8 and 9). Internal suction for both cross-opening cases (Orientations O₁ and O₂) falls in an intermediate position between the lowest and highest suctions of the single-opening cases.

For single-opening case and orientations O₁ and O₂, strong anti-clockwise swirling flow approaches toward the bluff-body at an angle where the opening is aligned on the one of the “windward” walls with respect to the local flow. As already obtained from the previous section that ground suction is the dominant pressure field inside the bluff-body which is opposite to the positive pressure generated by the anti-clockwise rotating flow. As a result, the suction gets slightly suppressed. However, for orientations O₃ and O₄, the openings fall in the wake of the flow. As a result, the suction due to the recirculation on the wake of the bluff-body enhances the suction inside the bluff-body. For both cross-opening cases, the location of one of the opening is on the “windward” and the other opening on the “leeward” walls of the bluff-body with respect to the direction of the local flow. As a result, the internal pressure is the resultant of the effect of the inward flow (that minimizes suction) and recirculating flow (that maximizes suction) (see Fig. 4-9).

6.3.4 Internal Pressure Comparison

<table>
<thead>
<tr>
<th>Direction of flow</th>
<th>Single-opening</th>
<th>Cross-opening</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SO₁</td>
<td>SO₂</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SO₃</td>
</tr>
</tbody>
</table>

SO₁: Opening on the windward wall
SO₂: Opening on the leeward wall
SO₃: Openings are on the windward and leeward walls

Figure 6-10. Location of opening/s on the bluff-body for ABL-flow

Internal pressure coefficients are compared between tornado center at location 2 and ABL-flow over a bluff-body, where the dimension of the bluff-body is identical and
rotated at an angle 45°. The reason for such rotation is to resemble the approaching flow towards bluff-body for location 2 (see Fig. 4-10).

<table>
<thead>
<tr>
<th>Single-opening</th>
<th>Cross-opening</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Image 1]</td>
<td>![Image 2]</td>
</tr>
<tr>
<td>![Image 3]</td>
<td>![Image 4]</td>
</tr>
<tr>
<td>![Image 5]</td>
<td>![Image 6]</td>
</tr>
</tbody>
</table>

**Figure 6-11. Internal Pressure comparison between Tornadic and ABL-flow**

For obtaining the internal pressure coefficient, identical reference velocity (maximum resultant velocity anywhere on a horizontal plane located at the bluff-body height but in
the absence of the bluff-body) is used for both tornadic and ABL-flow cases. Irrespective of the location and number of openings on the bluff-body, suction occurs inside the bluff-body for tornadic cases (see Fig. 4-11). However, for the ABL-flow case, whenever the opening is on the windward side, positive pressure occurs inside the building. When the opening is on the leeward wall, recirculation on the wake of the bluff-body creates suction which increased the suction inside the bluff-body. Thus it can be concluded that for tornadic flow, the internal pressure is primarily dominated by the tornado pressure field (suction) near the ground due to the high angular momentum particularly for core center and core radius locations. At these locations, orientations and number of openings may increase or decrease the internal suction depending on the orientation and configuration due to local wind/bluff body interaction but only slightly. Whereas, for the ABL-flow, the wind/structure interaction dominates the internal pressure and thus the location and size of the opening significantly determines the type of pressure inside the bluff-body.

6.3.5 Comparison of the mean internal pressure coefficient for stationary and translating tornado

![Graph showing comparison of mean internal Cp between stationary and translation tornado.](image)

Figure 6-12. (a) Comparison of mean internal Cp between stationary and translation tornado.
For translating tornado case, a bluff-body with single-opening having orientation, $O_1$ is chosen. Based on the translation speed reported earlier tornado translates from location 3 to location 1. For comparison, mean internal pressure coefficients are compared between locations 1 and 2 (see Fig. 4-12).

At location 1, internal suction is higher for translating than stationary tornado. However, the internal suction is slightly higher for stationary case compared to the translating at location 2. At location 1, ground suction is highest and the only dominating flow is weak downward flow which when comes on the vicinity of the bluff-body, gets separated over the roof and creates recirculation around the sidewalls. Presence of the translational movement of tornado increases this recirculation by enhancing the entrainment of the flow inside the bluff-body which in turn increases the recirculation of flow as well as internal suction. At location 2, the dominating parameter is the strong tangential flow for stationary tornado. As a result, the flow enters inside the bluff-body through the opening at an angle for stationary case.

Figure 6-13. Flow structure at location 1 and 2 for stationary and translation tornado
However, the vector direction of this incoming angled flow and the translational movement of tornado is opposite (see Fig. 4-13). As a result, the resultant angle of the incoming flow becomes less and thus decreases the recirculation inside the building which in turn decreases the suction.

From the above discussion, it can be obtained that, the addition of translation in a tornadic case variates the internal pressure with respect to the location of tornado center and direction of the translation. For location 1, the impact of translation upon the pressure inside the building is independent of the direction of rotation and translation of tornado. This is due to the collinear presence of the tornado center and the center of the bluff-body. As stated earlier, for the present condition, suction decreases for translation tornado than stationary tornado at location 2. This phenomenon could have been paradoxical if the direction of translation would have been opposite to what stated, keeping the anti-clockwise rotation identical (see Fig. 4-14).
6.3.6 Validating with Experimental Results

Minimum internal pressures (maximum suction) inside the bluff-body (Orientation 1) are measured along different radial locations of the tornado center with respect to the center of the bluff-body (see Fig. 15). These radial locations are normalized by the core radius ($r_c$). Irrespective of the three different studies, higher suctions are obtained for such location where center of the tornado coincides with the center of the bluff-body. From the figure, it can be obtained that better agreement is obtained with the observation of Sabareesh et al., (2012); because the size of the opening was 3.9%, where for the current study the size is 3%. However, the opening size is 11% in the paper of Letchford et al., 2015 which is much higher than the two cases discussed above with 3.9% and 3% opening.

![Figure 6-15. Variation of minimum internal pressure coefficient with radial location](image)

6.3.7 External Pressure Distribution

External pressure coefficients are measured along center line of the outer faces of the bluff-body for both sealed and unsealed cases (see Fig. 4-16). Similar to the internal pressure case, ground suction dominates the external pressure distribution. As a result, higher suction is obtained for location 1, where center of the tornado coincides with the center of the bluff-body. As the tornado moves away from the bluff-body, overall suction along the outer faces of the bluff-body decreases.
At location 1, suction is higher for sealed case than both the unsealed (single and cross-opening) cases. From the previous study, it is observed that when the center of the bluff-body coincides with the center of the tornado, downward flow at the center of the tornado flows down and gets separated at the roof and creates recirculation along the side faces of the bluff-body (Nasir et al., 2014). However, for unsealed cases, a portion of the recirculating flow enters inside the building through the opening or openings. As a result, suction, which occurs due to the recirculation, decreases.

For location 2, where center of the bluff-body is at the core radius of the tornado, the pressure distribution is quite identical for all the cases irrespective of the orientation and number of opening or openings. As already mentioned previously, the resultant horizontal flow approaches the bluff-body at an angle which gets separated on the southward and eastward walls (windward) and creates recirculation on the northward and westward walls.
(leeward). Because of this, suction is lower on the southward and eastward faces and higher suction on the northward and westward faces.

Lastly for location 3, where the center of the bluff-body is at quite a distance away from the center of the tornado, overall suction is very low and also similar to the location 1 and 2, hardly any differences can be obtained among all the cases. Also in this location, both the tangential and radial direction of flow are not strong enough to create any impact over the suction on the outer faces, as a result the pressure distribution is very uniform.

6.3.8 Net Pressure Distribution

![Net Pressure Distribution Diagrams]

Figure 6-17. Net Pressure Distribution

From internal pressure distribution, it is obvious that, pressure remains constant inside the building for each individual case (based upon location of the building and the orientation and number of openings). As a result, it is the external pressure distribution, which dictates the characteristics of the net pressure distribution. However, net pressure offsets
the external pressure distribution towards the higher suction due to the presence of the internal pressure.

6.3.9 Comparison of External Cp between Tornadic and ABL-flow over Bluff-body

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Location 1</th>
<th>Location 2</th>
<th>Location 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_T = 71.1\text{m/s}$</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

$U_T = \text{maximum velocity anywhere on a horizontal plane located at the bluff-body height but in the absence of the bluff-body for tornadic case.}$

<table>
<thead>
<tr>
<th>Parameter</th>
<th>ABL - flow</th>
<th>Parameter</th>
<th>ABL - flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_A = 68.5\text{m/s}$</td>
<td></td>
<td>$U_A = 68.5\text{m/s}$</td>
<td></td>
</tr>
</tbody>
</table>

$U_A = \text{maximum velocity anywhere on a horizontal plane located at the bluff-body height but in the absence of the bluff-body for ABL-flow case.}$

**Figure 6-18. External Cp comparison between tornadic and ABL-flow over bluff-body**

External Cp’s along the surface of the bluff-body are measured for three different locations of tornadic flow and ABL-flow over the bluff-body. Generally, reference velocity for measuring Cp for a ABL-flow over a bluff-body is chosen as $U_A$ in the figure. The advantage of choosing such is that, the pressure coefficient, Cp values will be true for any wind velocity and air density unless the size of the bluff-body and direction of flow has changed. Now obtaining external Cp values for ABL-flow over the bluff-body using the reference velocity, which was used previously for obtaining Cp for tornadic flow case, under predicts the Cp. So vice-versa, if reference velocity for ABL-flow would have been used for obtaining Cp’s for tornadic flow case, then it could have over predicted the values. So from the above discussion, it is quite obvious that, reference
velocity is a very sensitive issue for obtaining pressure coefficient accurately, which depends on the nature of the flow.

6.4 Conclusion

Based on the present analysis, the following conclusions can be made,

- Tornado induced pressure field (suction) near the ground dominates the overall pressure distribution for all the locations of tornado. As a result, irrespective of the number and location of openings, suction is higher both inside and outside the bluff-body at location 1 and decreases as tornado moves away from the bluff-body (locations 2 and 3).

- For stationary tornado, at location 2, suction inside the bluff-body is maximum for single opening on the leeward side of the wall and minimum for opening on the windward wall. For cross-opening case, internal suction falls within the maximum and minimum values. This is due to the resultant action of having openings on both the windward and leeward walls.

- At location 1, translation movement of the tornado increases the suction inside the bluff-body compared to the stationary case.

- At location 2, suction inside the bluff-body may increase or decrease, compared to the stationary case, depending upon the resultant flow from the rotation of tornadic flow and translation movement of the tornado.

- At location 1, suction on the outer faces of the bluff-body decreases due to the presence of the openings.

References

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

3. Chang, C.C., 1971: Tornado effects on building and structures with laboratory
simulation. Proc. 3rd Int. Conf. on Wind Effects on Buildings and Structures, Saikon,
Tokyo, 231-240.


structure, dynamics, predictions and hazards. Washington, D.C., USA, American
Geophysical union.


Lubbock, Texas.

Atmos. Sci., 31, 2081-2091.


Performance of a Large Tornado Simulator for Wind Engineering Applications.
Engineering Structures, 30, 1146-1159


14. Holmes, J. D., 1979: Mean and fluctuating internal pressure induced by wind. 5th Int. Conf. on Wind Engineering. Fort Collins, Colo., 435–450.


Chapter 7

7 Conclusion

In this dissertation, at first numerical simulations are carried out to simulate tornadoes of laboratory scale in order to validate the present numerical approach with the experimental findings. Secondly, a similar numerical approach is used to simulate full scale tornado-like vortex using a geometrical scale-up process based upon the location of maximum tangential velocity point. Once a confidence level is reached in simulating full scale numerical model, tornado interaction with buildings of different heights (low or high-rise) and types (sealed or unsealed) for different locations of tornado with respect to the location of the building is studied. In addition to this, tornado is simulated over two different types of hill (steep and shallow) to obtain the effects of the hill over the tornado flow-structure. Findings from these each research topic are summarized as follows,

7.1 Summary of the Findings

While developing flow-structure for different swirl ratio values, it is obtained that for low swirl ratios angular flow converges toward the center and reroutes its way vertically upward. As the swirl ratio increases, this angular flow becomes stronger and Vortex Break Down (VBD) occurs and with more increment in the swirl ratio this VBD moves toward the ground and at some point touch the ground. This particular phenomenon is called Vortex Touch Down (VTD) and this transforms a one-celled tornado to a two-celled tornado.

To assess the effect of hill over tornado flow-structure, tornado is placed in three different locations with respect to the crest of the hill (at hill crest, at the half height of the hill and at the foot of the hill). It is observed that; presence of the hill mostly effects the radial and vertical components of the flow. Also, for offset locations of the tornado with respect to the crest of the hill, tornado center relocates from its original location after the completion of the simulations due to the presence of the inclined ground surface. In addition to this, for offset locations of tornado center, speed-up is higher along the uphill slope compare to the downhill slope.
For simulating tornado over a low-rise building, it is observed that ground suction due to the angular momentum of the tornadic flow dictates the overall pressure distribution over the surfaces of the building. As a result, higher suction is obtained for such location of tornado where center of the tornado coincides with the center of the building. This suction decreases as tornado moves further away from the location of the building. Besides, for building location at the tornado core radius, due to the presence of the stronger tangential component of the flow, the pressure distribution follows a similar trend like that of a straightline wind (ABL flow) flows toward the building with an oblique wind direction. However, still the ground suction dictates the magnitude of the pressure distribution.

Likewise, low-rise building, similar pattern of pressure distribution is obtained for high-rise building except for building location at the core radius. It is observed that, the incident wind is more angular near the ground and it becomes less angular (i.e. becomes more parallel to the tangential component of the flow) for flow near the roof of the building. This particular change in incident flow dictates a different type of pressure distribution along the surface of the building. Besides, it is also observed that, for tornado location at the building center lift coefficient is highest.

Lastly, for low-rise building with opening’s, overall external and intel pressure distribution along the surfaces of the building again dictated by the ground suction of the tornadic flow. Internal suction increases with the increase in the number of openings for such location where tornado center coincides with the center of the tornado. In addition to this, higher internal suction is obtained for translating tornado over stationary one. Besides, for building location at the core radius, higher internal suction is obtained for opening on the leeward wall of the building compare to the opening on the windward wall. In addition to this, translation may or may not increase the internal suction for this particular location of the building depending upon the direction of translation and rotation of the tornado (clockwise or anti-clockwise).
7.2 Future Recommendation

In this study, an effort has been made to model steady tornado-structure interaction and it can be expanded in the future in many ways such as,

- By changing the aspect ratio value, Modified Numerical Simulator has been used to compare the results between two physical simulators (WindEEE and Purdue Tornado Simulator). This technique can be used in the future to create a simulator to compare the results among them and to come up with a tornado test guideline.
- For most of the simulation Reynolds Averaged Navier Stokes equations are used to keep the flow steady and main concentration was on the mean properties of flow. In the future, Large Eddy Simulations can be used to gather time series data analyze the peak or other statistical properties of flow.
- For off-set location of tornado center with respect to the building location, different angle of attack can be simulated to find a relation between angle of the tornado pathway and the structure.
- For the present study, only one tornado intensity was considered. In the future, higher intensity tornado can be considered.
- Current techniques for simulation can be used to, translate a tornado over a city or a cluster of buildings and then compare with the current results.
Curriculum Vitae

Name: Zoheb Nasir

Post-secondary Education and Degrees:

- Bangladesh University of Engineering and Technology
  Dhaka, Bangladesh
  2004-2008 B.A.

- Western University
  London, Ontario, Canada
  2010-2012 M.A.

- Western University
  London, Ontario, Canada
  2012-2016 Ph.D.

Related Work Experience

Teaching Assistant

The University of Western Ontario

2010-2016

Publications

- Nasir, Z., Bitsuamlak, G.T., Hangan, H., 2016: Computational modeling of tornado-like vortex induced mean pressure distribution on a typical building with flat roof. Submitted for publication.

- Nasir, Z., Bitsuamlak, G.T., Tornado-like vortex induced pressure on a bluff-body with and without openings. Submitted for publication.

- Nasir, Z., Bitsuamlak, G.T., Topographic Effect on Tornado-like Vortex. Submitted for publication.


- Nasir, Z., Bitsuamlak, G.T., 2014: Tornado induced load on a bluff-body with and without openings. 14th International Conference on Wind Engineering, Porto Alegre, Brazil.


• Nasir, Z., Bitsuamlak, G.T., 2016: Tornado-like vortex induced load on a bluff-body with and without openings, 8th International Colloquium on Bluff-Body Aerodynamics and Applications, Boston, USA.