August 27-31, 2023
FLORENCE - ITALY
16th International Conference on Wind Engineering
ICWE 16

An innovative computational approach to generate tornado-like vortices using large eddy simulation (LES)

Faiaz Khaled [1], Guangzhao Chen[1], Franklin Lombardo[1],

[1]University of Illinois at Urbana-Champaign, Urbana, United States, faiazk@illinois.edu

SUMMARY:
The investigation of tornado wind fields and induced loading using computational fluid dynamics (CFD) is attracting increasing attention within the structural wind engineering community. The majority of the CFD studies attempt to replicate the setup of physical tornado simulators. One of the key objectives of the current study is to propose an unconventional technique to generate the flow characteristics of a tornado-like vortex using large eddy simulation (LES). The proposed idea is based on the philosophy of employing opposing flow, boundary effects, and the pressure difference between the inlet and outlet of the computational domain. The current study serves as a foundation for the ongoing attempt to create flow features of a three-dimensional tornado-like vortex without the use of specialized tornado simulators. Besides, the future steps to accomplish the remaining tasks are outlined to present a complete picture. The proposed methodology will allow structural engineers to study tornadoes with limited CFD expertise. Moreover, researchers using physical straight-line wind simulators can extrapolate from the findings of this study to create a tornado-like vortex using such facilities.

Keywords: Computational Fluid Dynamics (CFD), tornado-like vortex, Large Eddy Simulation (LES), two-way flow.

1. INTRODUCTION

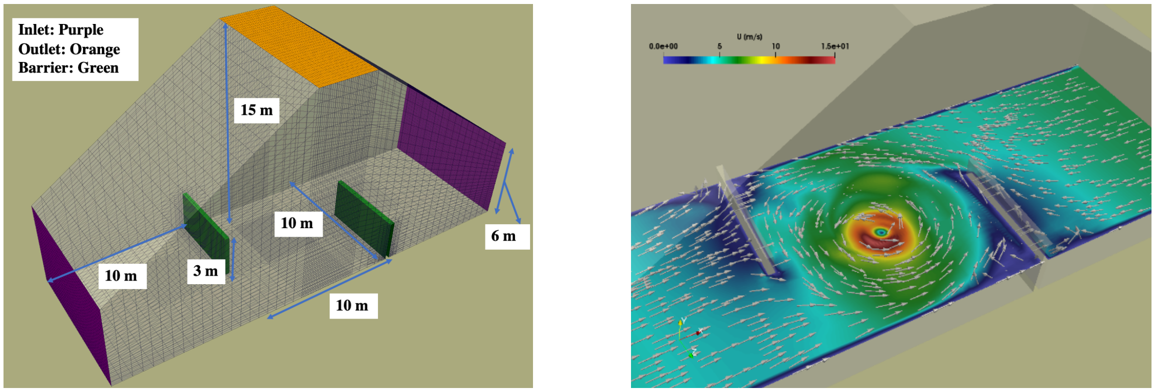
Tornadoes are responsible for considerable financial losses and a significant number of fatalities. The structural wind engineering community is expected to play a crucial role in protecting human lives, privately owned properties, and businesses from tornadoes. This amplifies the importance to investigate the tornado wind field and aerodynamic loads induced on structures by tornado-like winds. Field measurements (K. A. Kosiba and Wurman 2013) are considered the most accurate source of tornado wind-field data. However, the field velocity data are rare and typically collected at elevations much higher than those of the majority of buildings and often, not at a resolution required for wind engineering applications. This limitation instigated the generation of tornadoes in controlled environments using tornado simulators (Tang et al. 2018). Despite addressing some challenges encountered in field measurements, experimental techniques employed to generate tornado wind-field have their own limitations such as measuring velocity near the highly turbulent and three-dimensional core of the vortices with high fidelity (Tang et al. 2018). In addition, the considerable scale difference between generated vortices observed in the tornado simulators and real tornadoes warrants questions regarding the appropriateness of testing scaled aerodynamic models in such simulators (Fred L. Haan, Sarkar, and Gallus 2008; Tang et al. 2018).

Computational Fluid Dynamics (CFD) is considered a resourceful alternative to counteract the limitations of field measurements and laboratory-based tornado simulators. Employing CFD to investigate tornado-like vortices is a contemporary research area. In recent years, several research groups (Verma et al. 2022; Gairola and Bitsuamlak 2019) have attempted to produce tornado-like vortices using CFD models. However, most such studies either adopt an inflow generation mechanism identical to the well-known Ward-type laboratory tornado simulators, for instance, the VorTECH at Texas Tech University (TTU), or the CFD studies endorse some simplified models of physical tornado simulators. In this study, we attempt to produce a tornado-like vortex using a unique approach that does not involve replicating the characteristics of well-known physical tornado simulators. In the current approach, straight-line winds from opposite ends produce a tornado-like vortex by employing the necessary boundary effects of walls and barriers in the computational domain. This approach is proposed to generalize tornado-like vortex generation regardless of a certain type of physical tornado simulator and to make vortex generation easy for CFD practitioners dealing with tornadoes. In addition, such an approach can facilitate straight-line wind simulators to employ their resources for tornado-related research. The proposed method is an alternative computational approach to replicating the characteristics of specialized tornado simulators. The method is expected to benefit CFD practitioners studying tornado vortex. The authors aim to compare the flow characteristics of the proposed vortex with those from specialized tornado simulators or full-scale data.

2. METHODOLOGY AND SIMULATION SETTINGS

In the current study, an unconventional computational approach is proposed to generate a tornado-like vortex using a unique arrangement of two opposing inlets with corresponding barriers in their respective flow paths, and an outlet on the top of the domain. Figure 1 (a) presents the dimensions of the computational domain besides introducing the boundaries of the domain. The two identical purple faces denote the two opposing inlets of the same elevation, the orange face on the roof of the domain represents the outlet, and all other faces are modeled as a 'wall' with a 'no-slip' boundary condition. The 'no-slip' boundary condition indicates zero velocity on the surface of those faces. The core idea involves introducing the updraft, similar to a tornado-like vortex, using an upward suction generated by adjusting the pressure difference between the inlet and the outlet of the domain. In addition, the rotational motion is created by introducing two barriers in the flow path, which produces a steady vortex due to boundary effects and the mixing of the two opposing flows.

A barrier in the flow path induces flow separation, and because of a small wall thickness, the flow does not reattach onto the wall. Instead, the flow progresses past the wall and creates a rotational motion. Furthermore, the foundation of this technique is based on the boundary effects created by the barrier and the walls of the domain or virtual wind tunnel. Besides revealing the potential of this process to create a vortex, the ongoing study aims to examine the boundary effects by varying the aspect ratio of the inner chamber created between the two barriers. In this study, large eddy simulation (LES) was used in conjunction with the wall-adapting local eddy-viscosity (WALE) subgrid-scale (SGS) model because of its computational efficiency (M.F. Khaled, Aly, and Elshaer 2021). The cited paper details the numerical schemes and solvers used for the simulations. Before embarking on the investigation of the boundary effects, several trial runs are conducted to come up with a suitable pair of opposing velocities to generate a relatively steady vortex within the closed chamber. The preliminary findings were observed and, eventually, a uniform velocity of 5 m/s from the left and of 6 m/s from the right of the computational domain was introduced for the simulations of the current study.

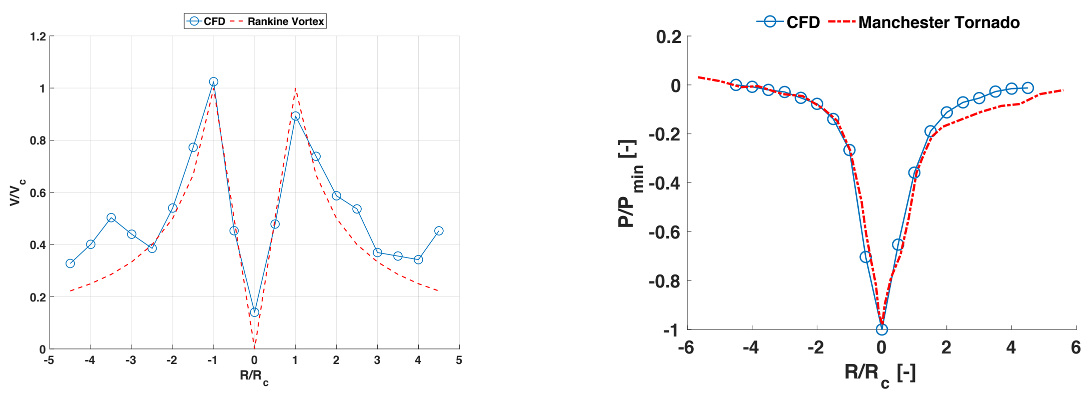


(a) (b)

Figure 1. (a) Dimensions of the proposed computational domain and location of the boundaries, and (b) instantaneous vortex field at t=14 seconds in the simulation.

3. RESULT AND COMPARISON

As for the CFD simulation, a stable single-cell vortex is created by the merger of the two-way flow. Based on Figure 1. (b), the minimum pressure point is assumed to be the vortex centre and the vortex is divided into a core region and an outside region. The boundary between the core and outside regions is assumed to be the maximum velocity envelope. The core region is dominated by the rotational flow and the outside region is restricted by the boundary effects of the no-slip walls. The zone within the two barriers is investigated by placing 551 probes at each elevation to analyze velocity and pressure. The simulation is continued for 50 seconds, and a relatively stable vortex is observed from the 11th second in the simulation. Beyond $t = 11\ sec$, the instantaneous and time-averaged plots look similar. This study is limited to the analysis of the two-dimensional vortex. For this purpose, probe data corresponding to 1 m elevation is chosen for further analysis.



(a) (b)

Figure 2. Comparison of vortex radial profiles

The radial profile of the velocity and pressure drop is generated from the data collected after the 11^{th} second in the simulation. The velocity profile from CFD-LES is compared with a numerical vortex model (Figure 2 (a)). The radial profile of pressure drop at $t = 11\ sec.$ is compared with those obtained from the Manchester, South Dakota tornado. The LES results of the two radial profiles agree well with those of the numerical model and full-scale data. As a continuation of this study a detailed analysis of vortex characteristics is planned, which will include, but not limited to, vertical profiles of velocity components and time-averaged plots.

4. CONCLUSION AND FUTURE WORK

This article demonstrates the ability of the proposed two-way flow approach employing LES to generate some fundamental characteristics of a tornado vortex. The current article is limited to the assessment of the two-dimensional vortex at a specific elevation in the domain. However, this is part of an ongoing study to reproduce and control a realistic three-dimensional tornado-like vortex using simple inflow. The influence of several aspects such as, but not limited to, the aspect ratio of the closed space, opening percentage of the barriers, height, etc. are being investigated. The technique to control the swirl ratio using this approach will be explored in the later stage. As for the vortex scale parameter, the boundary effect is explored by creating multiple cases with different aspect ratios of the closed chamber between the two barriers. Moreover, best practice simulation guidelines for creating reliable tornado-like vortex models with CFD will be introduced along this process.

ACKNOWLEDGEMENTS

The research was supported by the National Science Foundation (NSF). The NSF grant number is 2131961.

REFERENCES

Gairola, Anant, and Girma Bitsuamlak. 2019. "Numerical Tornado Modeling for Common Interpretation of Experimental Simulators." Journal of Wind Engineering and Industrial Aerodynamics 186 (March): 32–48. https://doi.org/10.1016/j.jweia.2018.12.013.

Haan, Fred L., Partha P. Sarkar, and William A. Gallus. 2008. "Design, Construction and Performance of a Large Tornado Simulator for Wind Engineering Applications." *Engineering Structures* 30 (4): 1146–59. https://doi.org/10.1016/j.engstruct.2007.07.010.

Khaled, M.F., A.M. Aly, and A. Elshaer. 2021. "Computational Efficiency of CFD Modeling for Building Engineering: An Empty Domain Study." *Journal of Building Engineering* 42 (October): 102792. https://doi.org/10.1016/j.jobe.2021.102792.

Kosiba, Karen A., and Joshua Wurman. 2013. "The Three-Dimensional Structure and Evolution of a Tornado Boundary Layer." *Weather and Forecasting* 28 (6): 1552–61. https://doi.org/10.1175/WAF-D-13-00070.1.

Tang, Zhuo, Changda Feng, Liang Wu, Delong Zuo, and Darryl L. James. 2018. "Characteristics of Tornado-Like Vortices Simulated in a Large-Scale Ward-Type Simulator." *Boundary-Layer Meteorology* 166 (2): 327–50. https://doi.org/10.1007/s10546-017-0305-7.

Verma, Sumit, R. Panneer Selvam, Zhuo Tang, and Delong Zuo. 2022. "Comparison of Tornado-Induced Pressures on Building from CFD Model with TTU Experimental Measurements." *Journal of Wind Engineering and Industrial Aerodynamics* 228 (September): 105076. https://doi.org/10.1016/j.jweia.2022.105076.