2022 ACWE
14th Americas Conference on Wind Engineering
May 17-19, 2022 – Lubbock, TX
National Wind Institute – Texas Tech University

Numerical Simulation of Tornado-Like Vortices generated in a Tornado Simulator Using Large Eddy Simulation

Hui Zhang [a], Zhuo Tang [b], Delong Zuo [c], Darryl James [d]

[a-d] Texas Tech University, Lubbock, Texas, USA, [a] hui.zhang@ttu.edu, [b] zhuo.tang@ttu.edu, [c] delong.zuo@ttu.edu, [d] darryl.james@ttu.edu

KEYWORDS: Tornado-like vortex, Large Eddy Simulation (LES), Boundary conditions

1. INTRODUCTION

Due to continued devastations caused by tornado hazards, extensive studies have been conducted to investigate tornadoes and their loading on structures. Many previous studies were based on field measurements and laboratory experiments. In recent years, numerical simulations utilizing Computational Fluid Dynamics have also gained much momentum. Many of these simulations aimed at producing tornado-like flows based on the principles employed for physical tornado simulators, and some representative characteristics of the numerically simulated flows were compared with the corresponding characteristics of either full-scale tornadoes or tornado-like flows generated by tornado simulators to demonstrate the effectiveness of the numerical simulations. Some simulations were also used to investigate the effects of various factors, such as the ground roughness and the translation speed, on the characteristics of the vortices. In some other studies, numerical simulations were also used to investigate the loading on structures, such as low-rise buildings and cooling towers, by tornado-like vortices.

This paper presents a numerical simulation aimed at reproducing a tornado-like vortex generated by a physical simulator. Major characteristics of the numerically simulated vortex and those of the pressures beath this vortex are compared with the corresponding characteristics observed in physical experiments. In addition, the effects of a number of factors, such as the boundary conditions and the type of outlet, on the effectiveness of the numerical simulation are investigated.

2. METHODOLOGY

The prototype of the numerical simulation is VorTECH (Fig. 1 (a)), a Ward-type tornado simulator at Texas Tech University. The simulation utilizes the technique of large eddy simulation (LES) with the adoption of the standard Smagorinsky subgrid turbulence model. It is carried out using the OpenFOAM platform with the pisoFOAM as the solver. The computational domain (Fig. 1 (b)) replicates most of the components of the physical simulator. It includes the testing chamber with the turning vanes at the periphery, the convergent region inside the turning vanes, and the convective region above the updraft hole. The orientations of the vanes are set to be 25° relative to the radial lines at their locations, the same as those of the vanes when the simulator was used to generate a two-celled vortex with a swirl ratio of 0.65. The velocity inlet of the model is placed outside the turning vanes, and the pressure outlet is set at the bottom of the honeycomb in VorTECH. Two types of pressure outlet are used, one allows reverse flow, and the other does not.

The boundaries of the convergent zone, turning vanes and updraft hole are set to be no-slip wall. Wall functions are used at the upper surface of convergent zone and the surface of turning vanes but not at the other surfaces. As shown in the grid distribution on the meridional plane (Fig. 1(c)), the resolution of the mesh is the highest in regions close to the walls as well as regions surrounding the axis of the simulator. The maximum dimensionless wall distance, y^+, of the first grids on the bottom of the convergent zone is less than 1, while those at the upper surface of the convergent zone and the surfaces of the turning vanes are less than 30 and 180, respectively. The time step of the simulation is set to be 2×10^{-5} seconds, which is small enough to ensure that the CFL number over the whole simulation domain is less than 1.

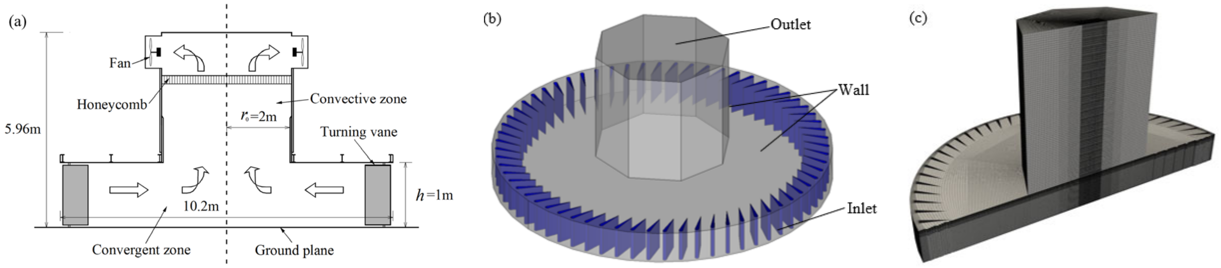

Fig. 1. (a) Schematic illustration of VorTECH (b) Computational domain (c) Grid Distribution

3. ILLUSTRATIVE RESULTS

The numerical simulation successfully reproduced the major characteristics of the prototype tornado-like flow generated by the physical simulator as well as the characteristics of the surface pressure. As an illustration, Fig. 2 (a) shows a comparison between the radial profiles of the mean tangential velocity at three heights based on data from the numerical simulation and physical experiment. Here $V_{\theta z}$ is the mean tangential velocity at heigh z above floor, $V_{\theta z\,\max}$ is the maximum mean tangential velocity at height z, r is the radial position and r_{cz} is the radial position at which the mean tangential velocity at height z reaches the maximum magnitude, and z_c is the height at which the overall maximum mean tangential velocity is reached. It is seen that the radial profiles resulting from the numerical simulation matches those based on the experimental data well. It is also apparent that the profiles based on both numerical and experimental data also agree well with that prescribed the Sullivan model for two-celled vortices. Fig. 2 (b) shows the radial profiles of the mean surface pressure based on the numerical and experimental data. Here $C_P = (p - p_\infty)/0.5\rho V_{\theta\max}^2$, with p being the surface pressure, p_∞ being the reference pressure, ρ being air density, and $V_{\theta\max}$ being the maximum mean tangential velocity. It is clear that the numerical profile agrees well with that based on the experimental data.

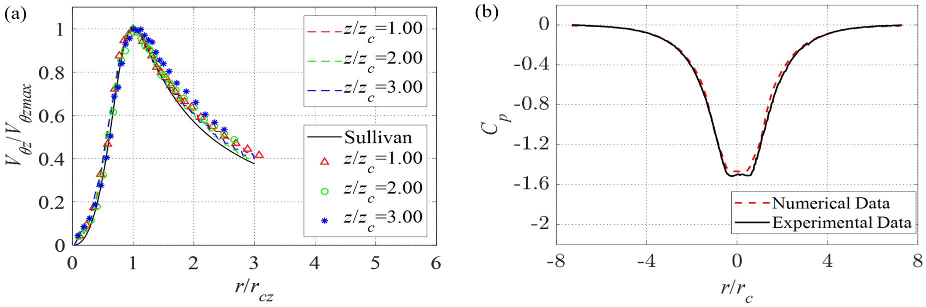

Fig. 2. (a) Radial profiles of mean tangential velocity component. Solid line: Sullivan model; Dashed lines: numerical data; Symbols: experimental data; (b) Radial profiles of mean surface-pressure coefficient.

More results from the numerical simulation as well as comparisons between these results with corresponding results from the experiments will be presented in the full paper along with discussions on the factors that affect the quality of the numerical simulation.