A NUMERICALLY GENERATED TORNADO-LIKE VORTEX
BY LARGE EDDY SIMULATION

Takashi Maruyama
Associate professor, Disaster Prevention Research Institute, Kyoto University
Ujishi Gokasyo, Kyoto 611-0011, Japan, maruyama@scl.kyoto-u.ac.jp

ABSTRACT

A tornado-like vortex was generated numerically by Large Eddy Simulation. The calculation region was designed to reproduce the wind flow in a tornado simulator. The tornado simulator consists of a convection region and a convergence region. A series of unsteady flow fields of vortex were generated and examined the flow characteristics with varying the inflow conditions, the boundary conditions, and the configuration of calculating domain.

KEYWORDS: LARGE EDDY SIMULATION, TORNADO-LIKE VORTEX, NUMERICAL SIMULATION

Introduction

The study on tornado has been carried out with a number of laboratory tornado simulators such as Ward (1972) and Church et al. (1979) and so on. The field measurement has been also improved by the development of observing devices such as portable Doppler radars. The third trend of tornado study is base on the numerical simulation. For example, Kuai et al. (2008) studied the laboratory-simulated tornado by k-ε model. They did parameter sensitivity tests and examined the flow characteristics with varying calculating conditions, that is mesh size, inflow and outflow conditions, boundary conditions and surface roughness. They described various aspects of calculated tornado. But the results came from mean values because the k-ε turbulence model calculates ensemble or time averaged value of wind velocity, pressure and so on. We sometimes need the time series fluctuation of wind velocity to examine the unsteady flow field or to evaluate the extreme quantities such as maximum wind speed or minimum pressure. To obtain the unsteady turbulent wind field in the tornado we arranged the numerical calculation by Large Eddy Simulation (LES).

Calculating Method

The calculating scheme we used here is based on the RIAM-COMPACT (Uchida et al. 2003) developed in Kyusyu University and modified so as to calculate with multi grid system. The governing equations consist of filtered mass and momentum conservation equations are given by equations (1) and (2).

\[ \frac{\partial \bar{u}_j}{\partial x_j} = 0 \]  

(1)
\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial^2 \nu e}{\partial x_j} + \frac{\partial^2 J_{ij}}{\partial x_j} \quad \text{(2)}
\]

where the over bar, \(\bar{}\), denotes the spatial averaged quantity. \(i, j = 1, 2, 3\); \(x, y\) and \(z\), respectively. \(u_i\) is the \(x_i\)-component of velocity; \(\rho\) is the density; \(p\) is the pressure; \(\nu e\) is the effective viscosity which is sum of the viscosity \(\nu\) and the sub-grid scale viscosity \(\nu_{SGS}\). \(J_{ij}\) and \(\nu_{SGS}\) are defined as

\[
J_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad \text{(3)}
\]

\[
\nu_{SGS} = (C_s L)^2 |D| \quad |D| = \sqrt{2\bar{D}_y \bar{D}_y} \quad \text{(4)}
\]

where \(C_s = 0.1\) following standard Smagorinsky turbulent model. We conducted the numerical calculation with a finite difference method and related the averaging volume to the mesh volume \(\Delta x_1 \Delta x_2 \Delta x_3\), where \(\Delta x_i\) is the \(x_i\)-directional width of mesh discretization. Additionally, we related the filter width \(L\) to

\[
L \equiv \sqrt{\frac{\Delta x_1 \Delta x_2 \Delta x_3}{3}} \quad \text{(5)}
\]

The orthogonal grid with staggered mesh system was used for the arrangement of parameters. A second-order centered difference scheme by Kajishima (1999) was adopted for spatial derivatives. The Adams-Bashforth scheme was employed for time-marching. Numerical integrations were conducted according to the fractional step method for mass flux and pressure coupling. The calculation region was designed to reproduce the tornado-like vortex in a laboratory tornado simulator as shown in Figure 1. The simulator consists of a convection region and a convergence region. The equivalent configuration to the laboratory experiment of Monji et al. (1985) was arranged in the calculating domain as shown in Figure 2. The dimensions of calculating domain are summarized in Table 1.

The horizontal shear was supplied by the in flows on the side walls of convergence region. Figure 3 presents an example of inlet boundary condition. The Dirichlet condition areas where certain velocities were distributed, the free condition areas where the normal gradient of normal velocity and the tangential velocity were set to be zero and the non-slip condition areas where the velocity was set to be zero were laid out on the side wall of convergence region. The strength of vortices was controlled by the wind speed and the distribution of inflow conditions. The inflow velocities were arranged so as to conserve the total mass flux over the calculation region. We could generate no vortex without horizontal shear at the inlet boundary. Sensitivity tests for the geometric dimensions of the calculating domain and the inflow conditions on the side wall of convergence region were carried out. A series of unsteady flow fields of vortex were generated and examined the flow characteristics with varying the calculating conditions.

**Generated Tornado-like Vortices**

Figure 4 shows a snapshot of generated tornado-like vortex. The flows come from the inlet boundaries to the center of the convergence region and spin up to the convection region through the updraft hole. The calculated flow fields were turbulent and varied with the calculating conditions as shown in Figure 5. Figure 6 shows the variation of time averaged flow characteristics with the diameter of convection region. The pressure difference in the
Table 1: Dimensions of Calculating Domain

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius of exhaust outlet, $R_e$</td>
<td>0.05 m</td>
</tr>
<tr>
<td>Radius of convection region, $R_{conv}$</td>
<td>0.15 - 0.6 m</td>
</tr>
<tr>
<td>Radius of updraft hole, $R_{ud}$</td>
<td>0.15 - 0.45 m</td>
</tr>
<tr>
<td>Width of convergence region, $2D$</td>
<td>1.2 m</td>
</tr>
<tr>
<td>Height of convection region, $H_c$</td>
<td>0.2 - 0.6 m</td>
</tr>
<tr>
<td>Height of convergence region, $H_d$</td>
<td>0.2 m</td>
</tr>
<tr>
<td>Exhaust outlet wind speed, $U_0$</td>
<td>1.0 m/s</td>
</tr>
<tr>
<td>Mesh size</td>
<td></td>
</tr>
<tr>
<td>horizontal direction</td>
<td>0.006 - 0.03 m</td>
</tr>
<tr>
<td>vertical direction</td>
<td>0.006 - 0.016 m</td>
</tr>
<tr>
<td>Mesh number</td>
<td></td>
</tr>
<tr>
<td>horizontal direction</td>
<td>90</td>
</tr>
<tr>
<td>vertical direction</td>
<td>33 - 53 (convection region)</td>
</tr>
<tr>
<td></td>
<td>35 (convergence region)</td>
</tr>
</tbody>
</table>
vortex core and the tangential velocity decreases as the diameter of convection region increases. The downward velocity appears in the vortex core and the vortex transforms from one-cell tornado to two-cell tornado as the diameter increases. The region with large magnitude of down draft in the vortex core also comes down with the diameter.

Figure 7 shows distributions of the time averaged velocity and pressure averaged azimuthally as a function of radial distance, $r$, from the center of the vortex at which the time averaged pressure become minimum. Strong radial speed directing to the center of vortex is observed near the ground. Above it, a strong tangential speed area locates. A strong upward speed region is observed inside of the strong tangential speed area. The wind directs outward
The Seventh Asia-Pacific Conference on Wind Engineering, November 8-12, 2009, Taipei, Taiwan

Figure 6: Variation of Flow Characteristics with the Width of Convection Region: $R_{cvt}$

Time Averaged Values of 50 Second after Spin up in Vertical Plane; $\gamma = 0.0$,

$U_0 = 1\text{m/s}$, $R_i = 0.05\text{m}$, $R_{ad} = 0.15\text{m}$, $H_c = 0.6\text{m}$, $H_d = 0.2\text{m}$
The Seventh Asia-Pacific Conference on Wind Engineering, November 8-12, 2009, Taipei, Taiwan

Figure 7: Distributions of Azimuthally Averaged Velocity and Pressure Field

Radius of Convection Region: \( R_{cvt} = 0.15 \) m

Figure 8: Profiles of Azimuthally Averaged Values of Swirl Ratio, \( S \), Maximum Tangential Speed Radius, \( R_{cvt} \), and Velocities, \( V_{t-max} \), \( V_r \), \( W \) as a Function of Height, \( z \)

Figure 9: Profiles of Azimuthally Averaged Tangential Speed, \( V_t \), Radial Speed, \( V_r \), Vertical Speed, \( W \) and Pressure at \( z = 0.015 \) m as a Function of Radial Distance, \( r \)

above the strong upward wind region. Profiles of maximum tangential speed, \( V_{t-max} \) vary with the radius of convection region as shown in Figure 8. Profiles of the radius, \( R_{cvt} \), at which \( V_{t-max} \) occurs, the radial speed, \( V_r \), and the vertical speed, \( W \), at \( R_c \) and Swirl ratio defined as \( S = \pi R_c^2 V_{t-max}/Q \) where \( Q \) is the total outflow rate, \( \pi R_c^2 U_0 \) at the top exhaust are also presented in the figures. \( R_c \) and Swirl ratio increase with the height near the ground and become constant above the height of \( z = 0.05 \) m. They also increase with the radius of convection region. Figure 9 shows the profiles of velocity and pressure along the radial direction at a height where the maximum value of the maximum tangential speed is observed.

We also calculated with other conditions. Figure 10 shows the result with multi-cell type vortex. Figure 12 shows an instantaneous wind field of vortex near a building and a pressure distribution on the surface of the building. The wind flow was forced to direct upward and the region with strong updraft was generated near the building. The positive pressure and strong negative pressure were observed on the upwind wall and the roof edge of the building, respectively.
Figure 10: Instantaneous view of calculated vortex with multi cell

\[ U_0 = 1 \text{m/s}, \ H_c = 0.6 \text{m}, \ 2D = 1.2 \text{m}, \ R_{cvr} = 0.6 \text{m}, \ R_{wd} = 0.15 \text{m}, \ H_d = 0.2 \text{m}. \]

Figure 11: Instantaneous wind velocity and pressure distribution around and on a building
Blue and Light Purple Surfaces Show the Regions with Strong Updraft.

Conclusions

A numerical tornado simulator was constructed by using the unsteady flow calculation method with Large Eddy Simulation. The configuration of calculating domain was designed to reproduce a tornado-like vortex following the laboratory tornado simulator. The simulator consists of a convection region and a convergence region. The horizontal shear was supplied by the inflows on the side walls of convergence region. The strength of vortices was controlled by the wind speed and the distribution of inflow conditions. Series of tornado-like vortices were generated with varying the calculating conditions. The characteristics of unsteady flow field in the vortex were examined. The variation of the velocity and the pressure in the vortex with the dimension of the convection region, a vortex with multi cell and a wind field of vortex near a building were presented.
References


