Near Ground Aircraft Wake Dissipation with Obstacles

C.H. John Wang* and J. Schlüter†

Nanyang Technological University, Singapore

The current study is a collaboration between the German Aerospace Center (DLR), the Nanyang Technological University, and the Civil Aviation Authority of Singapore to explore the effects of different ground based obstacles on the evolution of near ground wake vortex. Simulations were conducted using OpenFOAM solver, and validated against the DLR water tow-tank data collected using PIV measurements. The simulation domain was scaled to match the tow tank dimensions, and the vortex pair were initialized using Lamb-Oseen vortex equation using \( \Gamma_0 \) and \( r_c \) values from the PIV data. Simulations were conducted in the tow tank simulation domain for flat ground without obstacles and with obstacles of various configuration and design.

I. Introduction

The studying of aircraft wake vortex decay near ground has long been a focus of civil aviation research, with experimental LIDAR study as early as the 1970s to the more recent European WakeNet3 project. The dissipation rate of these vortex behind large aircraft has long been a limiting factor for aircraft separation both at cruise and takeoff/landing leg of the flight, and is a major factor dictating the maximum air traffic volume at an airport. With the recent increase in the acquirement of Airbus A380, classified as “super” by International Civil Aviation Organization, by more airlines operating at a growing number of airports world wide, research into methods for improving the near ground dissipation rate of aircraft wake vortex became more important both from the aviation safety stand point and the airport operational efficiency stand point. With the expected increase in the number of A380 transiting Singapore’s Changi Airport, a joint study by Nanyang Technological University, German Aerospace Center (DLR), and the Civil Aviation Authority of Singapore (CAAS) was proposed to investigate the possibility

*Research Fellow, School of Mechanical and Aerospace Engineering.
†Assistant Professor, School of Mechanical and Aerospace Engineering
of implementing ground based obstacles to encourage dissipation of aircraft wake vortex. The study was conducted with large eddy simulation (LES) carried out using the open source solver OpenFOAM, covering various configurations of ground obstacle in addition to the obstacles proposed in previous DLR studies.\textsuperscript{1–3}

II. Previous Studies

The study into aircraft wake vortex has a long history: one of the earlier study was the remote sensing study using continuous wave LIDAR in 1970,\textsuperscript{4} which saw extensive use in the following years studying aircraft wake vortex of various aircraft types at airports throughout the U.S. \textsuperscript{5–8} Additional studies during the 1990s saw the integration of atmospheric condition and air traffic data into wake vortex study\textsuperscript{9} and the modification of the Terminal Area Simulation System (TASS) at NASA Langley for the simulation of aircraft wake vortex.\textsuperscript{10} The WakeNet project was funded by the European commission in 1998 with the aim of consolidating the knowledge among European countries which, along with its successor programs, lead to the series of research conducted at DLR\textsuperscript{1–3,11,12} on the fundamental of vortex pair interaction with each other, with ground, and with ground obstacles; these studies served as the bases of the collaborative study described in the present paper.

The idea of introducing ground obstacles to reduce the effect of aircraft wake vortex on subsequent aircraft is not new, as previous wind tunnel study by NASA Langley Research Center analyzed the effect of different plate type obstacles and suction/blowing devices on top of or along the runway;\textsuperscript{13} they reported that barriers placed paralleled to the runway is effective in containing the wake vortex with the fastest dissipation rate among the device tested, while barrier normal to runway direction showed only localized disturbance. The earlier DLR study on the vortex pair interaction and dissipation,\textsuperscript{11} on the other hand, lead to the proposal of using ground based obstacle to trigger short wave instability (as opposed to the long-wave Crow instability), which is hypothesized to disrupt the integrity of overall vortex structure. Initial experimental studies\textsuperscript{1} were conducted using the DLR water towing tank at their Göttingen facility using the DLR F13 aircraft model; the experimental data were recorded using stereo PIV and visualization of vortex structure using black ink injected into the water from the wing tip of their aircraft model. Based on the recorded initial circulation of 0.052\textit{m}^2/\textit{s}, the Reynolds number was calculated to be $Re_\Gamma = \frac{\Gamma_0}{\mu} = 52,000$. The basic ground obstacle used in the study is modeled using a square cylinder spanning across the bottom of the tank (referred to as “standard obstacle” in this paper), perpendicular to the direction of travel, with a $2b_0 \times 2b_0$ cross sectional area; the other obstacle configuration tested placed a second identical square cylinder $7.2b_0$ downstream from the initial obstacle.

LES simulations by Gerz et al.\textsuperscript{2} using the DLR’s MGLET code, which utilized La-
grangian dynamic sub-grid scale turbulence model, were conducted in concert with the experimental study using the geometry of a full scale mid-size aircraft, with a vortex separation of 47.1m and conducted at a Reynolds number of 23,130. Simulations were conducted within a \(4b_0 \times 8b_0 \times 3b_0\) domain with 256, 512, and 256 mesh points respectively along each edge. The wake vortex were initialized using the Lamb-Oseen vortex model and super imposed into the domain that contained a fully developed boundary layer flow along the floor.

III. Simulation Setup

III.A. Solver

The current study aims to validate the OpenFOAM solver and obstacle model we have access to against the circulation data and vortex trajectories from the DLR study; once the simulation method has been validated, additional ground obstacles with varying shape and form is introduced to assess their impact on vortex decay near ground. Simulations were conducted using the NTU High Power Computing facility running OpenFOAM version 2.2.x. The solver used utilizes a hybrid PISO-SIMPLE algorithm to solve the Navier-Stoke equation, which introduced an additional loop which recycles the previous PISO loop solution within the same time step; this would allow the usage of larger time step while maintaining stability even with \(CFL > 1\).

Several turbulence model were available within stock OpenFOAM package. However, due to the domain averaged implementation of the dynamic Smagorinsky model in OpenFOAM, the current simulation uses the version modified by Alberto Passalacqua to calculated spatially and temporally localized constants.

III.B. Simulation Domain

Two separate domains were setup for our simulations: Case 1 has the same dimension as the measurement section of the DLR towing tank at \(0.918m \times 0.306m \times 1.224m\) \((w \times h \times l)\) with mesh setup to have three mesh cells across \(r_c\), which gave us a resolution on vortex center similar to what was in the prior full scale simulations; Case 2 was conducted using the same dimension and similar grid setup as the LES simulation case documented in Gerz et al.\(^2\)

Wall boundary mesh were introduced onto the ground boundary patch in simulations where crosswind condition were introduced; boundary mesh were retained for cases without crosswind, and the near wall flow field is resolved as best as possible, with computational resource being the limiting factor: The current number of mesh points in Case 1 is the maximum number of cells, at around 28 million, that could be loaded into the solver with the available computational resource. In order to maintain consistency between the two simulation cases, mesh resolution across the vortex and \(y^+\) value for
wall boundary mesh are set to be identical even though Case 2 only contain around 18.5 million cells.

The vortex were initialized via superposition using Lamb-Oseen equation:

$$V_{\theta,0}(r) = \frac{\Gamma_0}{2\pi r} \left(1 - \exp\left(-\frac{r^2}{r_{c,0}^2}\right)\right)$$  \hspace{1cm} (1)

the initial values $\Gamma_0$ and $r_{c,0}$ are listed in Table 1 below.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Gamma_0$ ($m^2/s$)</td>
<td>0.052</td>
<td>530</td>
</tr>
<tr>
<td>$r_{c,0}$ ($m$)</td>
<td>0.009</td>
<td>3</td>
</tr>
</tbody>
</table>

III.C. Ground Obstacles

The Stephan et al. towing tank study investigated the effect of single square cylindrical shaped obstacle on the dissipation rate of wake vortex. For our study, three additional obstacles were proposed, the dimensions of which were shown in Figure 1 below.

Obstacle 1 is a simplified version of the plate-line design proposed, and investigated, by Holzäpfel et al., and served as our proof of concept case to ensure our ability to model baffle type obstacles. Obstacle 2 and 3 were proposed based on existing vortex generator designs; the assumption was that secondary vortex generated would pull the wake vortex downward, reducing vertical displacement of the vortex post first contact with the ground. The three dimensional view of the Obstacle 1 and Obstacle 2 are shown below in Figure 2.
IV. Validation Results

The velocity data were extracted using “slice” filter in paraview, and circulation data were calculated using a Octave/Matlab function. Several turbulence models were assessed in our simulations, including standard and dynamic version of Smagorinsky and One Equation Eddy models, the dynamic Lagrangian model, and Spalart-Allmaras DDES model. These simulations were conducted using baseline Case 1 setup, i.e. towing tank without obstacles, with the initial vortex height set to $b_0$. The vortex center was tracked using the point of minimal pressure within the vortex structure, and the results were analyzed using $\Gamma_{\text{max}} = \max_r \{\Gamma(r)\}$ to obtain the circulation about the vortex core. Data were taken at the center of the domain ($z = 0.612m$) and $\Gamma_{\text{max}}$ from the two vortices were averaged to obtain a single value.

Mesh convergence studies were conducted in two parts: the first regarding the mesh’s ability to resolve the vortex structure as it descend from its initial height within the atmosphere; the second regarding the mesh’s ability to capture the interaction between the vortex structure and the ground. For the first part of the study the simulation domain was discretized into 5.6, 18.5, and 36.6 million cells for the coarse, baseline, and dense mesh setup, respectively, where the baseline setup provided identical mesh resolution across the vortex core as the DLR LES simulation. The mesh convergence study results are shown in Figure 3 below:
Figure 3. Comparison of circulation data from simulation domain with various mesh density, as represented by the total mesh count.

The results showed very little difference in circulation measured between the time the vortex began descending from the initial position and the formation of secondary vortex due to vortex-ground interaction at $t^* \sim 1$ for the three mesh cases. However, circulation results from post vortex-ground contact showed significant dissipation of energy from the vortex structure, as shown in Figure 4. The $y_1$ values of 0.0019$m$, 0.0015$m$, and 0.0003$m$ corresponds to the first cell distance from the ground for the three difference mesh density from Figure 3, which was obtained by maintaining the mesh growth rate while reducing the vertical mesh count; on the other hand, the $y_1$ value of 0.0007$m$ were obtained by calculating the $y^+ = 1$ value based on maximum tangential velocity$^b$ and substituted the new boundary layer mesh into the baseline domain.

Figure 4. Comparison of circulation data among various boundary mesh size (in meters) along the ground.

$^b$instead of the “cross-wind” velocity from the original Case 1 setup
While Figure 4 illustrated that sufficiently fine boundary layer mesh would lead to a predicted vortex dissipation rate that corresponds well with the experimental data, we would need to look at the velocity contour of the vortex-ground interaction to find out the cause of higher dissipation rate shown in the results from the coarser wall mesh.

![Velocity contour of the vortex-ground interaction with various wall-boundary setup](image)

(a) $y_1 = 0.0019$
(b) $y_1 = 0.0015$
(c) $y_1 = 0.0007$
(d) $y_1 = 0.0003$

Figure 5. Velocity contour of the vortex-ground interaction with various wall-boundary setup

As seen in Figure 5, the velocity contour for the two coarsest boundary mesh setup is insufficient in resolving the vortex-ground flow interaction, and a sharp drop in velocity magnitude can be observed at the wall boundary. On the other hand, simulation using the two denser wall mesh setup showed a smoother transition from the no slip boundary patch. Simulation results above suggested that the baseline setup with wall boundary mesh is sufficient in capturing the change in $\Gamma_{max}$ about the vortex structure during the course of its interaction with the ground, and will be use as bases for the construction of simulation domain for the subsequent study on the effect of obstacles.

In addition to vortex circulation, the normalized vertical position of vortex center were also traced over time as part of the comparison with available experimental data. The trajectory data is compared with the DLR data in Figure 6 below.
Apart from the higher initial bounce, the vortex trajectory from the simulation result match fairly closely with the experimental data. The greater bounce amplitude could be due to the width difference between the towing tank (1.1m) and the simulation domain (0.918m) where the side wall forces the vortex to raise higher.

Apart from the mesh convergence study, a turbulence model study was carried out using all other available turbulence model from stock OpenFOAM installation: Standard Smagorinsky Mode, dynamic Lagrangian model, standard one equation eddy model, dynamic one equation eddy model, and Spalart-Allmaras DDES model. The simulations were carried out using the baseline model with wall mesh implemented, and the results from the turbulence model study is shown below in Figure 7.

The simulation results have shown that, at least for our setup, minimal difference among results from various sub-grid scale models, and even between models with constant and dynamic coefficients.
V. Obstacle Results

Circulation measurements from the simulation results were presented below. Data extraction were performed at location specified in the experimental study as $x^* = 0, 1.05, 3.6$, which corresponds to the $z$-location of $z = 0.612m, 0.77265m$, and $1.1628m$ in our simulation domain, respectively. Circulation data from the square cylinder obstacle simulation is presented below in Figure 8, while the simulation from the three obstacles introduced in the present simulation is presented in Figure 9.

![Figure 8](image1.png)

(a) $x^* = 0$

![Figure 8](image2.png)

(b) $x^* = 1.05$

![Figure 8](image3.png)

(c) $x^* = 3.6$

Figure 8. Normalized circulation from the square cylinder obstacle simulation comparing to experimental data.
While the circulation data at the position of the obstacle match closely with the experimental data, the results showed that the disruption to vortex structure is passed along the axis at a faster rate than observed in the experiments; similar trend is also observed in the Figure 9.

![Figure 9. Normalized circulation from different obstacle configurations comparing to experimental data.](image)

The increased dissipation further from the obstacle could be due to the periodic boundary condition implemented at the end wall, with the instability traveling away from the obstacle position compounded the rate of dissipation closer to the boundary. Thus the vortex is influenced by multiple sets of obstacles along the axial direction instead...
of experiencing the effect of a single set of obstacle. It should be noted that the result is similar to the LES results from the previous study\(^1\), which indicated that the increased in vortex dissipation could be due to the end-wall effect, insufficient mesh resolution for secondary vortex structures, or additional high frequency instability filtered out by the LES simulations.

Additional visualization of the vortex-obstacle interaction were conducted using the square cylinder obstacle setup in the domain described in Case 1 and is presented in Figure 10.

![Visualization from the LES simulation, with iso-surface of \(\|\omega^*\| = 31.4\).](image)

The visualization showed similar secondary vortex feature created by the interaction between the wake vortex and ground obstacle as described in Gerz et al.\(^1\); these secondary vortex subsequently wrapped around the primary vortex structure, disrupting the integrity of the vortex core and leading to an increased dissipation rate.

**VI. Conclusion**

While the current simulation setup provided results in good agreement with the benchmark data for vortex dissipation without obstacles, further adjustment to the numerical setup is needed to account for the end wall effect to better simulate for the case with obstacle. However, the results still allowed us to compare the effectiveness of different obstacle for the purpose of increasing vortex dissipation. The three obstacle setup investigated in this report showed very similar effect on vortex dissipation, which lead to the hypothesis that the projected area as observed by the vortical flow plays a larger role in determining the strength of disturbance to vortex structure than the shape. Additional
simulation is also planned to investigate the usage of vortex generator that could produce a stronger secondary vortex loop, which in turn could increase the propagation of the disturbance caused by the obstacle.

Acknowledgments

We thank the Civil Aviation Authority Singapore (CAAS) for the financial support through the Air Traffic Management Research Institute (ATMRI).

References


