CFD Simulation of Aerial Crop Spraying

Zamri Omar #1, Kua Yong Qiang #2, Sofian Mohd #3, Nurhayati Rosly #4

# Aeronautical Engineering Department, Faculty of Mechanical & Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia
1 zamri@uthm.edu.my
2 ad120170@siswa.uthm.edu.my
3 sofian@uthm.edu.my
4 nurhayati@uthm.edu.my

Abstract—Aerial crop spraying, also known as crop dusting, is made for aerial application of pesticides or fertilizer. An agricultural aircraft which is converted from an aircraft has been built to combine with the aerial crop spraying for the purpose. In recent years, many studies on the aerial crop spraying were conducted because aerial application is the most economical, large and rapid treatment for the crops. The main objective of this research is to study the airflow of aerial crop spraying system using Computational Fluid Dynamics. This paper is focus on the effect of aircraft speed and nozzle orientation on the distribution of spray droplet at a certain height. Successful and accurate of CFD simulation will improve the quality of spray during the real situation and reduce the spray drift. The spray characteristics and efficiency are determined from the calculated results of CFD. Turbulence Model (k-ε Model) is used for the airflow in the fluid domain to achieve a more accurate simulation. Furthermore, spray simulation is done by setting the Flat-fan Atomizer Model of Discrete Phase Model (DPM) at the nozzle exit. The interaction of spray from each flat-fan atomizer can also be observed from the simulation. The evaluation of this study is validation and grid dependency study using field data from industry.

Keywords: Aerial application, CFD, DPM, and k-ε Model.

1. Introduction

This aerial crop spraying, also known as crop dusting, is made for aerial application of pesticides or fertilizer. An agricultural aircraft which is converted from an aircraft has been built to combine with the aerial crop spraying for the purpose. In recent years, many studies on the aerial crop spraying were conducted because aerial application is the most economical, large and rapid treatment for the crops. In other words, its development can result in greater harvest yield of crops. The research on the atomization characteristics of the spray nozzles used on the agricultural aircraft has continuously been carried out by the agricultural aviation industry since long time ago. At early stage, the droplet size and droplet density were found to influence the efficacy of crop production and protection material. However, most recently the issues of concern are about spray drift and environmental law. Spray drift, which is the off target of spray, causes lower application rates, damage to non-target organisms, and pollution. To control spray drift, the most governing factors are droplet size and spray droplet spectra. Spray drift causes the higher tendency of small droplets to drift from the application zone than large droplets. [1]
Aerial spray material is transported in a combined wind field of aircraft wingtip vortices and atmospheric turbulent wind. The transport processes is manipulated by the vortices when the spray is released, however turbulent wind becomes dominant after the wingtip vortices decay the atmospheric. Extensive studies has been done on the effects of vortices induced by the aircraft the theory has been widely applied in agriculture spray models. [2]

The objective of the present work paper is to study the airflow simulation of aerial crop spraying system. In this research, a computational simulation approach is proposed by using Computational Fluid Dynamics (CFD), which is ANSYS Fluent. The aim of this paper is to examine the spray characteristics at different aircraft speed and height. The study also focuses on the effect of nozzle orientation on the spray droplets features.

2. Methodology

2.1 Aerial Crop Spraying System Schematic Diagram

The schematic diagram of the aerial crop spraying system is shown in Figure-1. The spacing between each nozzle is 0.5m and the distance of nozzle from airfoil is 0.5m. There are 13 nozzles on the spray boom and the length of spray boom is 6m. The length of spray boom is designed to be about 75% of the wingspan in order to minimize the drift. The longer spray boom will feed the spray into the vortices near the wing tip and increase drift.

![Figure-1: Schematic diagram of aerial crop spraying system: (a) Top view and (b) Side view.](image)

2.2 Nozzle Selection

The nozzle used in the aerial application has the specification as shown in Figure-2. The highlighted nozzle type is flat-fan nozzle (632.334). The mass flow rate, \( m \), is obtained by using Eqn. (1) where \( \rho \) is the density of water (spray liquid) and \( V \) is the nozzle volume flow rates:

\[
m = \rho \cdot V
\]
2.2 Computational Geometry and Grid
The computational model created by SolidWorks is the combination of the wing and the rectangular fluid domain with the dimension of 32m x 32m x 200m as shown in Figure-3 and Figure-4 respectively. The combination of both model is illustrated in Figure-5.

The study of [3] states that it is a good practice to do CFD simulations by using at least three different levels of grid refinement in order to assess the sensitivity of the CFD predictions to the number of grid nodes. ‘Grid-independence’ is a condition which the grid should have a sufficient number of nodes so that further grid refinement has no effect on results. The course, medium, fine and finer computational grid is shown in Figure-6.

**Figure-2**: Specifications of nozzle (Lehler)

**Figure-3**: Wing.

**Figure-4**: Fluid domain.

**Figure-5**: Combined fluid domain.

The study of [3] states that it is a good practice to do CFD simulations by using at least three different levels of grid refinement in order to assess the sensitivity of the CFD predictions to the number of grid nodes. ‘Grid-independence’ is a condition which the grid should have a sufficient number of nodes so that further grid refinement has no effect on results. The course, medium, fine and finer computational grid is shown in Figure-6.
2.3 Realizable K-ε Model
The selection of model depends on the Reynolds number which is a dimensionless value that measures the ratio of inertial forces to viscous forces and describes the degree of laminar or turbulent flow. The calculation of Reynolds Number is based on Eqn. (2).

\[ RE = \frac{\rho vl}{\mu} = \frac{vl}{\nu} \]  

In this equation, the velocity, \( v \) is given as 25m/s, 30m/s, 35m/s for three certain velocities for the aircraft, the chord width, \( l \) is given as 1 m and the kinematic viscosity, \( \nu \) is given as \( 1.7894 \times 10^{-5} \) m\(^2\)/s. The Reynolds number for three cases are 1,397,116, 1,676,540 and 1,995 963 which are all above \( 5 \times 10^5 \). This means the flow can be assumed as turbulence. In real life, most of the flow are turbulent.

2.4 Discrete Phase Model
Discrete Phase Model (DPM) in FLUENT performs Lagrangian trajectory simulation by applying Euler-Lagrange approach. When solving using DPM, the fluid phase is treated as continuous phase using Navier-Stokes equations, while the discrete phase is solved by tracking a fixed amount of
particles or droplets. According to the number of continuous phase iterations per DPM iteration being set, the frequency of the discrete phase model source terms are updated after each continuous phase iteration. The automated tracking scheme, is selected to enhance the accuracy of DPM simulations, is divided into higher order scheme (trapezoidal and Runge-Kutta) and low order scheme (implicit and analytic).

The atomizer that has been chosen is flat-fan atomizer which is usually used in agriculture spraying. The material used for atomizer is water-liquid. The properties of flat-fan atomizer (Figure-7) are arc position, virtual position, normal vector, temperature, mass flow rate, duration of injection, spray half angle, orifice width and sheet breakup. The value of the point properties of the flat-fan atomizer is also illustrated in Figure-8.

![Flat-fan atomizer parameters.](image)

**Figure-7:** Flat-fan atomizer parameters.

The spherical drag law is used to estimate the drag coefficients acting on droplets. It assumes that the surface tension on the dropfluid interface is strong enough to resist the tendency of the aerodynamic force to deform the drop, i.e the Webber number, \( \text{We} \ll 1 \). [4]

### 2.5 Velocity Inlet

The highlighted surface of fluid domain shown in Figure-9 is velocity inlet. The value for velocity inlet is the relative velocity of the aircraft to the fluid domain.

![Point properties for flat-fan atomizer.](image)

**Figure-8:** Point properties for flat-fan atomizer.
2.6 Wall
Solid wall is the most basic boundary condition in fluid domain. There are two types of wall which is stationary wall and moving wall. Figure-10 illustrates the stationary wall as blue surface. In Figure-10(a), the discrete phase boundary condition is set to reflect because the droplet is assumed to reflect when it hits airfoil. The temperature of the top (Figure-10(b)) and bottom (Figure-10(c)) is set as 299.8K and 300K respectively because the temperature drops when the altitude increases.

2.7 Pressure Outlet
The highlighted surface in Figure-11 is pressure outlet. The gauge pressure is set to be 0(atmospheric pressure) and the discrete phase boundary condition is escape type.
2.8 Solver Setting
In order to simulate the continuous phase flows, the 3D steady Reynolds-Average Navier-Stokes (RANS) equations for conservation of mass, momentum and energy are solved along with the realizable k-ε turbulence model by [5]. The Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm is used for pressure-velocity coupling, pressure interpolation is second-order and second-order discretisation schemes are used for both the convection terms and the viscous terms of the equations. After the solutions for the continuous phase are converged, the Lagrangian trajectory simulations are continued for the discrete phase. [4] The discrete phase interacts with the continuous phase, and the discrete phase model source terms are updated after each continuous phase iteration. The number of iteration is set in order to converge the solutions. The list of parameters of cases is shown in Table-1, where V is the airstream velocity, α is the nozzle orientation angle measured clockwise from horizontal, and h is the height of the nozzle from the ground. Case 1 is used for validation while case 2 is used for grid dependence study.

Table-1: List of parameters of cases.

| Case | V(m/s) | α (°) | Number of flat-fan nozzle | h(m) |
|------|--------|-------|---------------------------|------|
| 1    | 0      | 0     | 1                         | -    |
| 2    | 25     | 0     | 1                         | -    |
| 3    | 25     | 0     | 13                        | 2, 4, 6, 8, 10 |
| 4    | 25     | 0     | 13                        | 2, 4, 6, 8, 10 |
| 5    | 25     | 0     | 13                        | 2, 4, 6, 8, 10 |
| 6    | 25     | 0     | 13                        | 5,10 |
| 7    | 25     | 45    | 13                        | 5,10 |
| 8    | 25     | 90    | 13                        | 5,10 |

3. Results and Discussion
3.1 Spray Model Validation
In order to validate the CFD simulation models, the results from the first case of Table-1 is compared with the specification of flat-fan nozzle shown in Figure-2. Table-1 presents the field experiment data done by the industry such as spray volume flow rate and spray width at certain height. The result which is selected to validate is the spray width at height at 250mm and 500mm respectively. The spray width and spray angle is measured manually and put into Table-2.

Figure-12(a) and (b) illustrate the contour of particle mass concentration at h=250mm and h=500mm respectively. At h=250mm, the spray width is measured as 353mm while at h=500mm the spray width is 520mm. The contour of particle mass concentration parallel to the spray is shown in Figure-12(c). The simulated spray shows 51° of spray angle after being measured.
Figure 12: Contour of particle mass concentration: (a) h=250mm, (b) h=500mm, (c) Parallel to the spray.

The data obtained is put into Table 2 and the percentage error is calculated using Eqn. (3). The average percentage error is 31.12%.

### Table 2: Comparison between field data and simulated data.

| Case                        | Field data | Simulated data | Percentage error (%) |
|-----------------------------|------------|----------------|----------------------|
| Spray width at 250mm        | 220mm      | 353mm          | 60.45                |
| Spray width at 500mm        | 440mm      | 520mm          | 18.18                |
| Spray angle                 | 60°        | 51°            | 15.00                |

\[ \%Error = \frac{Data_{simulated} - Data_{field}}{Data_{field}} \]  \hspace{1cm} (3)

3.2 Grid Dependence Study

A grid dependence study was performed on three additional grids which are course grid, fine grid and finer grid. The basic or medium grid has 168,623 elements. On the other hand, the course grid has 80,455 elements, while the fine grid has 288,512 elements and the finer grid has 454,892 elements. The four types of grid are also illustrated in Figure 6. The scatter chart of DPM concentration against distance at height of 2m is plotted and shown in Figure 13.
Figure-13: Result for grid dependence study: (a) Comparison between course grid and medium grid, (b) Comparison between medium grid and fine grid, (c) Comparison between fine grid and finer grid.

As illustrated in Figure-13(a) and (b), there is obvious difference observed by comparison of different grids. Among the three figure, the deviation between fine grid and finer grid (Figure-13(c)) is negligible. Purpose of grid dependence study is achieved and it can be said as ‘Grid-independence’. Therefore, the fine grid was retained for further analysis.

3.3 Release Height of Spray

Release height of spray has significant effect on spray quality as well as spray drift level. It is investigated by comparing the contour of DPM concentration with height set to 2m, 4m, 6m, 8m and 10m as shown in Figure-14 (v=25m/s). It appears from Figure-14 that the downwind deposition gradually increases as the release height increases. The downwind deposition is almost 2.3 times greater for a release height of 10m (151.35m) compared with 2m (66.21m). However, the DPM concentration decreases as the release height increases. The maximum DPM concentration at release height of 2m is about 4 times bigger than at 10m. The present finding also supports [6]’s study which concluded that higher release heights tend to level out the downwind deposition pattern.

3.4 Aircraft Speed

The influence on the downwind deposition of varying aircraft speed is presented in Figure-14 (v=25m/s), Figure-15 (v=30m/s) and Figure-16 (v=35m/s). From this data we can see that the downwind deposition increases as the aircraft speed increases. However, as Figure-16 shows, there is a significant difference if compared to the other two cases. In Figure-16 there is a clear trend of decreasing in downwind deposition when the release height increases especially at 6-10m. What is interesting in this data is that no DPM concentration at release height of 10m (Figure-16) is detected. The finding is consistent with findings of past studies by [7] and [6], which provide evidence that increasing windspeed there must always be increasing downwind deposition. With
increasing windspeed there may be increased turbulence which acts to disperse the spray cloud more rapidly therefore reducing far downwind deposition levels.

**Figure-14:** Contour of DPM concentration at (a) h=2m, (b) h=4m, (c) h=6m, (d) h=8m, (e) h=10m at aircraft speed, v=25m/s.

**Figure-15:** Contour of DPM concentration at (a) h=2m, (b) h=4m, (c) h=6m, (d) h=8m, (e) h=10m at aircraft speed, v=30m/s.
3.5 Nozzle Orientation
Figure-17 illustrates the DPM concentration against deposition distance with nozzle orientation for 0°, 45° and 90°. From comparison between the data shown in Figure-17(a) and (b), we can see that the DPM concentration decreases when the nozzle orientation changes from 0° to 45°. This phenomena can be explained as the higher wind shear on spray when the orientation is increased. This result reflects the findings of [8]. The most striking result to emerge from the data is that the DPM concentration of nozzle orientation of 90° (Figure-17(c)) is higher compared with orientation of 45° (Figure-17(b)). This significant difference can be explained as the nozzle orientation of 90° (point perpendicular to ground) speeds up the spray droplets to reach the ground.

3.6 Spray Coverage Area
Table-3 presents the spray coverage area at different aircraft speed and spray release height. The method to measure the spray coverage area is by using AutoCAD software. The polyline is drawn on the contour and the area can be found (Figure-18).

| Aircraft speed, v(m/s) | Release height, h(m) | Spray coverage area, A(m²) |
|------------------------|----------------------|---------------------------|
| 25                     | 2                    | 447.4                     |
| 25                     | 4                    | 626.4                     |
| 25                     | 6                    | 1163.3                    |
| 25                     | 8                    | 1252.7                    |
| 25                     | 10                   | 1610.7                    |
4. Conclusion and Recommendation

A CFD method which is a Lagrangian Discrete Phase Model (DPM) which predicts and simulates the spray droplet transport by aerial crop spraying system, was studied in this paper. The purpose of the current study was to determine the spray characteristics like DPM concentration and spray coverage area under different parameters such as aircraft speed, spray release height as well as nozzle orientation.

This study set out to determine the factors that affect the spray quality. First of all, the result of this investigation concluded that the higher release heights tend to level out the downwind deposition pattern. Secondly, this study also found that increasing aircraft speed will increase downwind deposition. Furthermore, the unsatisfactory results were obtained on studying the effect of nozzle orientation. The reasonable results were presented at relatively low speed only.

The second major finding was the method to measure the spray coverage area. With the aid of CAD software, the irregular shape of spray area could be measured. This proposed a method to calculate the spray coverage area with reliable and accurate data. In addition, validation cases and grid dependence study were also presented in the research. Validation ensures the accuracy of the model prediction within an acceptable tolerance. Grid dependence study will find out the most suitable grid for preventing higher computational cost (finer grid) and inaccurate results (courser grid).

4.1 Recommendation

The trend of aerial crop spraying in agriculture area will be greater in the future. It requires advanced simulation setting to make prediction that close to the real situation. Hence, the mathematical model and formulation applied should be enhanced in further research. There are a few recommendations that could be considered:

1. Wind tunnel experiment would be great for further investigation and experimentation on the aerial crop spraying. The role of wind tunnel experiment is to study the real field data which is different from the prediction of CFD simulation. Besides that, the findings from simulation can be validated with the data obtained from wind tunnel experiment.

2. A further study investigating spray by transient solver in simulation would be very interesting. Transient solver will treat the spray particle in an unsteady fashion. The spray
particles can be tracked within a period. However, transient method spends longer time than steady method to reach solution convergence.

3. It is recommended that further research be done with using user-defined function or UDF. UDF is a C programming language that can be interpreted by ANSYS to enhance its standard features. For example, the injection of particle can be initialized with varying location which can demonstrate the moving atomizer.

Acknowledgments
This research has been funded by Universiti Tun Hussein Onn Malaysia and Ministry of Higher Education Malaysia, under Fundamental Research Grant Scheme (FRGS) 2015/1.

References
[1] Kirk, I.W. (2007). Measurement and prediction of atomization parameters from fixed-wing aircraft spray nozzles. Transactions of the ASABE, 50(3), pp. 693–703.

[2] Wang, Y., Miller, D.R., Anderson, D.E. & McManus, M.L. (1995). A lagrangian stochastic model for aerial spray transport above an oak forest. Agricultural and Forest Meteorology, 76, pp. 277–291.

[3] Gant, S. (2006). CFD modelling of water spray barriers. Health and Safety Laboratory.

[4] Montazeri, H., Blocken, B. & Hensen, J. (2015). Evaporative cooling by water spray systems: Cfd simulation, experimental validation and sensitivity analysis. Building and Environment 83 (2015) 129-141.

[5] Shih, T.H., Liou, W.W., Shabbir, A., Yang, Z. & Zhu, J. (1995). A new k-ε eddy viscosity model for high reynolds number turbulent flows. Computers & Fluids, 24(3), pp. 227–238.

[6] Teske, M.E. & Thistle, H.W. (2004). Aerial application model extension into the far field. Biosystems Engineering, 89(1), pp. 29 – 36.

[7] Craig, I., Woods, N. & Dorr, G. (1998). A simple guide to predicting aircraft spray drift. Crop Protection, 17(6), pp. 475–482.

[8] O’Connor-Marer, P.J. (2011). Aerial Applicator’s Manual: A National Pesticide Applicator Certification Study Guide. National Association of State Departments of Agriculture Research Foundation.

[9] Lehler International GmbH. (1998). Flat Fan Nozzles Product Catalogue, Lehler International GmbH.