Analysis of Droplet Motion in Air Engine Washing Technology

Liwen Wang¹, Zhiqi Yan², Feiqi Long¹, Jie Tang¹, Tao Wang¹, Xudong Shi³

1. School of Avition Automation, Civil Aviation University of China, Tianjin 300300, China
2. College of Mechanical Engineering, Yanshan University, Qinhuangdao 066004, China
3. Ground Special Equipment Science and Research Base, Civil Aviation University of China, Tianjin 300300, China

Abstract: In the process of air engine water washing, the effect of which would be influenced by the characterization of liquid sheet. A numerical method is put forward for the research of motion of liquid drop in the aero-engine to simulate the movement law when the particle enters into the front of the engine from nozzle. Firstly, the mathematical model of movement is established to describe droplet moving behavior locating the engine intakes, which is three-dimensional, based on movements of a small sphere subject the forces on the droplet and the droplet physical properties. Then, the model is solved using the four order Runge-Kutta method, considering the coupling effect of the multi force field including gravity, drag force and so on. Finally, the motion velocity and the trajectory of the drop-lets are obtained. And the distribution law of the droplet is calculated. Compared with the calculation results of other mathematical models, the results prove the correctness of the model.

Keywords: engine; water wash; jet; droplet; swirling turbulent; kinetic characteristic

1. Introduction

Online aircraft-Engine washing is one of the important ways to removing dirt from the interior road described in the Aircraft Maintenance Manual (AMM), which can effectively recover the exhaust gas temperature margin caused by the pollution of the aero-engine (Exhaust Gas Temperature Margin, EGTM), decreasing fuel consumption rate, raising propulsion efficiency, and solving the problem of the shortening of life span of air path components[1].

With the world's growing awareness of environmental protection, in 2012 the European Union began to impose aviation carbon tax on all incoming airlines. This makes China recognize that Airlines need to reduce operating costs and enhance flight safety by improving the engine cleaning technology higher. The process of ordinary cleaning is shown in Figure 1. In this process, a plurality of nozzles are injecting the cleaning liquid into both the interior and the outer duct of the engine which is turning cold in the low speed of 2000rpm. Engine duct will be clean in the process of the cleaning fluid is rotated by the engine blades

![Figure 1 Schematic diagram of aircraft engine washing](image)

Online Aero-engine cleaning is defined as cleaning the engine duct without disassembling the engine. Aero-engine cleaning is described as a process of removing the contaminants from the surface of leaves through a series of processes of being stretched, slid and rolled, etc. it is also called fluid - solid coupling interaction. Influence the quality of the cleaning effect is influenced by some factors like the initial movement conditions, including velocity, trajectory and distribution of liquid droplets.

Many parameters such as force situation, the droplet distribution and the velocity of jet are affected by The geometry of the nozzle, the initial conditions of the liquid flow[2][3], and the gas flow field. Many research has been done on the motion characteristics of jet droplets at home and abroad. K. A. Sallam et al. measured the droplet volume on a vertical jet surface, proposing a statistical method for the average diameter of the droplets - the SWD method[4]. S.S.Yoon[5] and others developed the SWD method to predict the average droplet size, velocity and flow direction more accurately. In the rotating flow, the droplet will be affected by a series of forces, in which the Stokes drag force is considered as the main factor affecting the droplet movement[6]. Debo Li et al. have shown that additional mass forces can be induced when the gas around the droplet accelerates motion[7]. The additional mass force (Mirko Salewski et al. ) in the flow field is much smaller than the drag force and negligible in the droplet diameter of 1-100 microns. Additionally, the dynamic Reynolds number determines the size of the drag force, and the Reynolds number is related to the velocity of both droplet and flow[8]. Sung Wook Park et al. establish a mathematical model of droplet movement, validating the model by measuring the size and velocity of the droplet with means of spark lamp, laser, long-distance microscope and CCD camera[9].

To study the movement of liquid droplets jetting into engine duct, A mathematical model is established to describe jet droplet movement under the action of multi-force field. Further exploring to characteristics of droplets Has important significance for improving the cleaning effect.

In this paper, a three-dimensional model of droplet flow is established, researching the characteristics of jet droplet under rotating flow field.

2. Mathematical model

The aircraft engine washed in this study is SAFENG's CFM56, which is used in the mainstream engine A320. Pure water is used as the cleaning medium in the cleaning process when the aircraft engine is in cold turn at outdoors apron. Based on this working condition, the mechanical model of the droplet is established, and then the equation of the droplet movement is obtained.

2.1 Physical hypothesis
The motion droplets in the jet field are the object of this study. Hypotheses are put forward, taking into account the complexity of fluid movement.

The rotating air flow and droplets are thermodynamically balanced. Because the effect of thermophoretic force on droplet velocity is not significant in the process of droplet movement[17]. The assumption is proposed for simplifying the operation

The droplets are spherical without being deformed in motion. So when being studied in terms of movement and force, droplets can be taken as particles in the air flow, whose shape is not influenced by rotating flow.

The complex force of the droplet can be decomposed into the superposition of the force in the jet and the force in the rotating flow field.

The velocity of the rotating air flow and droplets are independent of each other, because of ignoring the coupling effect. The pure rotation of the droplet does not affect the pure translation. The rotation of the droplet produces Magnus lift. But only in the case of \(Re <1\)[18]. In this paper, the droplets are in a high Reynolds number turbulence environment. So the theory of Magnus effect is not applicable, pure rotational motion is not considered.

2.2 Physical model of droplet

The movement of the cleaning fluid is the result of the coupling of the jet and the rotating air flow. Some research on gas - liquid two-phase flow, such as micro bubble plume[10], jet in shear flow[11-12], particle motion in low Reynolds number[9] and three-dimensional droplet movement[13], Which provides a reference for the study of the movement mechanism of the liquid flow into the engine duct. Many particles motion mathematical modeling, such as the S-Z & B model[13], the D & A model and the B & B model[14-16] are limited to the infinite space and ideal uniform force solution, not fitting the condition of the engine inlet Therefore, it is necessary to construct a flow kinematics model suitable for the complex rotating gas - liquid two-phase flow.

The velocity of liquid droplet in cleaner jet is given by:

\[
V = u_x i + v_x j + w_x k
\]  
(1)

The velocity of air flow is

\[
U = u_y i + v_y j + w_y k
\]  
(2)

And the equation of droplet motion is used as follows

\[
m_d \frac{\partial V}{\partial t} = F_D + F_A + F_B + F_G
\]  
(3)

Where the terms on the right-hand side of Eq.(3) are the drag force, the virtual mass force, the buoyancy force and the gravity force. Expression respectively is

\[
F_D = \rho_g |U - V|(U - V)C_D \frac{\pi d^3}{6}, F_A = \pi d^3 \rho_g \frac{(U - V)^2}{2}, F_B = \pi d^3 \frac{[\frac{\partial (U - V) - \frac{\partial U}{\partial t}}{\partial t}]^2}{2}, F_G = \frac{\pi d^3}{6} \rho_d g
\]

The drag coefficient \(C_D\) stands for drag coefficient, \(\rho_g\) stands for air density.

The equation of droplet motion is described as:

\[
m_d \frac{\partial V}{\partial t} = \rho_g |U - V|(U - V)C_D \frac{\pi d^2}{8} + \pi d^3 \rho_g \left[\frac{\partial (U - V) - \frac{\partial U}{\partial t}}{\partial t}\right] + \pi d^3 \left(\rho_d - \rho_g\right) g
\]  
(4)

The drag coefficient \(C_D\) is taken as

\[
C_D = C_D^{oo} \times \frac{2 \mu_g + 3 \mu_d}{3 \mu_g + 5 \mu_d} \times h
\]  
(5)

\[
C_D = \begin{cases} 
\frac{24}{Re_d} & Re_d \leq 1 \\
\frac{24}{Re_d} \left(1 + \frac{1}{6} Re_d^{2/3}\right) & 1 < Re_d \leq 1000 \\
0.424 & 1000 \leq Re_d < 2 \times 10^5 \\
0.1 & Re_d \geq 2 \times 10^5 
\end{cases}
\]  
(6)

Where \(Re_d\) is Reynolds number when droplets moving, which is
\[ Re_d = \left| \frac{U - V}{d / \mu_g} \right| \tag{7} \]

In the equation, \((2 \mu_g + 3 \mu_d) / (3 \mu_g + 3 \mu_d)\) stands for the comprehensive effect of inner loop flow of liquid drops. \(\mu_g\) and \(\mu_d\) are gas dynamic viscosity and droplet kinetic viscosity respectively. The parameter \(d\) is droplet diameter, while droplet deformation is \(h = [1, \infty]\), which is equal to 1 confirmed in this paper[19].

2.3 Model solving

Droplet movement can be considered as particle motion, when a research is carried out to study the movement of a single droplets in front of the engine. That is, a particle motion model can be used to describe droplet motion. Figure 2 show that the process of the droplet move from time \(t_i\) to \(t_{i+1}\). Set that the speed of droplet will from \(V(r_i, t_i)\) to \(V(r_i, t_{i+1})\), with its position form point \(r\) to point \(r_i\). The velocity and location of the droplets can be deduced as followed:

At the moment of \(t = t_i\), The position vector of the droplet is \(r = x_i i + y_i j + z_i k\); The droplet velocity is \(V_i = u_{di} i + v_{di} j + w_{di} k\); the velocity of flow where droplets locate is \(U_i = u_g (r_i) i + v_g (r_i) j + w_g (r_i) k\).

![Figure 2: The Droplet motion in rotating flow field](image)

Set the step size from \(t_i\) to \(t_{i+1}\) is \(\Delta t\), and the initial time is \(i = 0\), that is, the time \(t_0\)

At time of \(t_i\) and the location of \(r = t_i\), the equation can be written as:

\[ \frac{\partial V(r_i, t_i)}{\partial t} = a_1 \left| U_i - V(r_i, t_i) \right| \left| [U_i - V(r_i, t_i)] \right| + a_2 \tag{8} \]

\[ a_1 = (\rho_g C_D \pi d^2 / 8) / (m_d + \pi d^3 \rho_g / 12) \tag{9} \]

\[ a_2 = (\rho_d - \rho_g) C_D \pi d^2 / 8) / (m_d + \pi d^3 \rho_g / 12) \tag{10} \]

So at the same time and location, The solution of the fourth - order standard Runge - Kutta method for droplet velocity as followed:

\[ K_1 = a_1 \left| U_i - V(r_i, t_i) \right| \left| [U_i - V(r_i, t_i)] \right| + a_2 \]
\[ K_2 = a_1 \left| U_i - V(r_i, t_i) - K_1(t_i - t_{i+1}) / 2 \right| / [U_i - V(r_i, t_i) - K_1(t_i - t_{i+1}) / 2] + a_2 \]
\[ K_3 = a_1 \left| U_i - V(r_i, t_i) - K_2(t_i - t_{i+1}) / 2 \right| / [U_i - V(r_i, t_i) - K_2(t_i - t_{i+1}) / 2] + a_2 \]
\[ K_4 = a_1 \left| U_i - V(r_i, t_i) - K_3(t_i - t_{i+1}) \right| / [U_i - V(r_i, t_i) - K_3(t_i - t_{i+1})] + a_2 \]

(11)

When the droplet is approaching the aero-engine blade, the rotation of flow field where droplet locates resembles that of the rigid body, which described as \(U = \omega r\). The \(\theta\) is angular velocity of swirl. The rotation center is \(r_0(x_0, y_0, z_0)\). The velocity of the local flow field where droplets locate in the x-axis and y-axis in the xy plane is:

\[ u_{x_i} = \omega (x_i - x_0) \tag{12} \]
\[ u_{x_i} = \omega(y_i - x_0) \]  

(13)

At the point of \( r = r_i \), the blade tangential velocity is \( u_i \) when the blade angle of the engine blade is \( \theta_i \) on the XY projection of the droplet. There is velocity \( w \) along the direction into the impeller in Figure 3, called “air inlet angle”, the value of which is \( u_i = u_i \cos \theta_i \). The velocity component in the z-axis is \( u_i \cos \theta_i \cos \theta_i \cos \theta_i \) As the c-direction velocity component, the velocity \( u_i = u_i \sin \theta_i \) produce z-axis component \( u_i \cos \theta_i \cos \theta_i \cos \theta_i \), the velocity in the z-direction of the flow field at position \( r \) is the algebraic sum of the two sub-velocities.

\[ u_{z_i} = u_i \cos \theta_i \sin \theta_i + u_i \cos \theta_i \sin \theta_i = u_i \sin 2\theta_i \]  

(14)

\( u_{x_i} \) and \( u_{y_i} \) are the speed component of \( u_i \), so it can also be written as:

\[ u_{z_i} = \omega \sqrt{(x_i - x_0)^2 + (y_i - x_0)^2} \sin 2\theta_i \]  

(15)

When the droplets collide with the inner wall of the engine duct, a rebound effect occurs.

\[ \vec{V}_n \] for rebound velocity, Then there \( \vec{V}_n - \vec{V} / \vec{F} \) and \( \vec{F} \). The \( a \) and \( b \) stand for velocity components in the xy axis. Related equations are established:

\[ \begin{cases} a - u_d/\|\vec{u}_d\| = b - v_d/\|\vec{v}_d\| \\ \sqrt{a^2 + b^2} = \sqrt{u_d^2 + v_d^2} \end{cases} \]  

(16)

Then \( a \) and \( b \) are solved:

\[ \begin{cases} a = \frac{u_d v_r^2 + 2v_d u_r v_r - u_d u_r^2}{u_r^2 + v_r^2} \\ b = \frac{v_d u_r^2 - 2u_d u_r v_r - v_d v_r^2}{u_r^2 + v_r^2} \end{cases} \]  

(17)

2.4 Parameter determination

The air temperature at the local airport is 26℃. The air density is 1.18Kg/m3 and the aerodynamic viscosity is 1.85×10^{-5}Pa•s according to the table. The pre-set water temperature is heated to 68℃. The water density is 0.978×10^{3}Kg/m3 when be heated to the preset temperature, and the dynamic viscosity of water drop is 1.01×10^{-3}Pa•s.

Aircraft engine cleaning equipment is set inside the engine in operation, the front of the which is the fan-shaped nozzle. To have a better effect, the nozzles should be aligned with the middle of the blades clearance. CFM56 engine radius is about 1500mm. Set the rotation center of the blade as the mathematical model coordinate zero point (0,0), then the coordinate of the center nozzle is (0, -0.375). During the cleaning process, the airplane engine is turning cold in speed of 4000 rpm.

In order to get the initial parameter, the situation of this airport has been studied in this paper with the establishment of the nozzle model shown in Figure 4.
The nozzle parameters are given in Table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value/mm</th>
<th>Parameter</th>
<th>Value/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>D1</td>
<td>13</td>
<td>L1</td>
<td>18.5</td>
</tr>
<tr>
<td>D2</td>
<td>8</td>
<td>L2</td>
<td>2.5</td>
</tr>
<tr>
<td>D3</td>
<td>16</td>
<td>R</td>
<td>1</td>
</tr>
<tr>
<td>D4</td>
<td>2</td>
<td>θ</td>
<td>50.19°</td>
</tr>
<tr>
<td>H</td>
<td>2</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The rated inlet pressure during purging ranges from 20 psi to 100 psi. The internal velocity profile of the nozzle under different inlet pressures is shown in Figure 7.

The velocity profiles are shown along the nozzle centerline at input pressures of 100 psi and 20 psi in Fig7. The droplet slowly accelerates in the cavity of nozzle, accelerating to the highest peak at the nozzle tip, attenuating at the nozzle outlet. The velocities are shown in Table 2 at the nozzle outlet center at different inlet pressures.

<table>
<thead>
<tr>
<th>Pressure PSI</th>
<th>Velocity m/s</th>
<th>Pressure PSI</th>
<th>Velocity m/s</th>
<th>Pressure PSI</th>
<th>Velocity m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>26.43</td>
<td>70</td>
<td>22.12</td>
<td>40</td>
<td>16.60</td>
</tr>
<tr>
<td>90</td>
<td>25.00</td>
<td>60</td>
<td>20.46</td>
<td>30</td>
<td>14.39</td>
</tr>
<tr>
<td>80</td>
<td>23.56</td>
<td>50</td>
<td>18.62</td>
<td>20</td>
<td>11.67</td>
</tr>
</tbody>
</table>

3. Model validation
In the swirling air flow, the drag force of the air to the droplet is $F_D$. The force of the swirl on the droplets includes the Basset force due to the accelerated motion of the droplet, but the Basset force is much smaller than the drag force. When the droplets are small enough ($\mu m \leq d \leq 10 \mu m$), the additional mass $F_A$, buoyancy $F_B$, and gravity $F_G$ of the droplets are not considered [20].

According to Newton's second law, the differential equation of droplet motion can be written:

$$m_d \frac{\partial V}{\partial t} = \rho_g |U - V| |(U - V)| C_D \frac{\pi d^2}{8}$$

(18)

In the Stokes flow of spherical droplets ($Re_d < 6.2$), the drag coefficient is described as:

$$C_D = \frac{24}{Re_d} = \frac{24 \mu_g}{\rho_g |U - V| d}$$

(19)

Substituting $m_d = \rho_g d^3 \frac{\pi}{6}$:

$$\frac{\partial V}{\partial t} = \frac{18 \mu_g}{ \rho_g d^2} (U - V) = \frac{1}{\tau} (U - V)$$

(20)

Where the parameter $\tau = \rho_g d^2 \sqrt{18 \mu_g}$ stands for particle relaxation time.

Define the velocity ratio between the state of a droplet and its terminal state as the terminal velocity ratio[16]. The speed of the terminal state is defined as the resultant force of the droplet is 0, that is:

$$\frac{\partial V}{\partial t} = \frac{1}{\tau} (U_{max} - V_{max}) = 0$$

(21)

The droplet terminal state $V_{max} = U_{max}$. The droplet is at the edge of the engine outer duct when $U$ is the maximum. At this time $|V_{max}| = |U_{max}| = r \omega$, and the terminal velocity ratio of droplet is $|V_{max}| / |V| = |r \omega | / |V|$. Contradistinction of results of droplet motion model and other particle motion models is shown in Figure 6. The abscissa is dimensionless time, defined as $\tau^* = t / \tau$. The ordinate is the speed terminal ratio. The initial state is taken from the water velocity.

It can be seen from the figure that the results of this model (T & Y model) are in agreement with the S-Z & B model with slower convergence than the D & A model. At the later stage, the motion state of the S-Z & B model and the D & A model become stable in the infinite environment because of the constant force action. In this study, droplets are in a very limited environment within the aircraft engine, which will accelerate the movement and then collide with the duct. In this process, liquid droplets will quickly close to the cutoff speed and stop, instead of closing to the final value. Revealing in the picture, the curve of the S-Z & B model and that of the D & A model on the image are bifurcated at a later stage, T & Y model curves continue upward, while the curves of the S-Z & B model and the D & A model gradually smooth when $V_{max} / V$ close to 1.

The pressure from 20 psi to 100 psi is input into the mathematical model, and the trajectories of the droplets are shown in Figure 7.
The five curves in Figure 7 are the trajectories of the droplets at 100psi to 20psi, respectively, from top to bottom. With the influence of the drag force, the droplet will be deflected and revolved around the center of the blade, ending of the collision wall. When the input pressure is increased, the trajectory of the droplet is elongated in the positive direction of the Z-axis.

If the x-axis is taken as the vertical axis, the z-axis as the horizontal axis, the trajectory of droplets can be obtained and the linear processing curve shown in Figure 8.

In the figure, slightly amplitude offset can be found from the trajectory of droplets, which coming from nozzle not more than 1m away under the input pressure of 100psi, without any obstruction. Obvious amplitude offset is found in the range of 1 m to 1.5 m, larger one will happen more than 1.5m away, droplets will reach to the surface of engine at about 2.3 m. Under the input pressure of 20psi, Obvious amplitude offset is found in the range of 0.5 m to 1.2 m, larger one will happen more than 1.2m away, droplets will reach to the surface of engine at about 1.7 m.

In table 3, the end value in the Z direction of each curve in Figure 9 are shown

<table>
<thead>
<tr>
<th>Pressure Input/Psi</th>
<th>Displacement /m</th>
<th>Pressure Input/Psi</th>
<th>Displacement /m</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>1.7638</td>
<td>70</td>
<td>2.1787</td>
</tr>
<tr>
<td>30</td>
<td>1.9112</td>
<td>80</td>
<td>2.2137</td>
</tr>
<tr>
<td>40</td>
<td>2.0050</td>
<td>90</td>
<td>2.2455</td>
</tr>
<tr>
<td>50</td>
<td>2.0769</td>
<td>100</td>
<td>2.2771</td>
</tr>
<tr>
<td>60</td>
<td>2.1322</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

In the table3, the radial displacement of the droplets increases with growing input pressure, while reducing the amplitude. According to the Lagrange interpolation method, the data in Fig. 8 is substituted into the formula

$$\frac{fu}{u} = \sum_{j=3} y_j \prod_{r=0}^{j} \frac{u-x_r}{x_j-x_r}$$

The fitting formula is giving by:

$$f(x) = -0.00007x^2 + 0.01446x + 1.5256$$

The numerical results are compared with those of the fitting formula, as shown in Fig.9 By comparison, the maximum error is 1.3%. The result of fitting result is very close to that of the original numerical.
4. Conclusion

In this paper, motion characteristics of jet droplets in the aircraft inlet rotational aerodynamic flow are studied in the process of CFM56 aircraft engine online cleaning. Conclusions are followed:

(1) The S-Z&B model is used to reveal the particle motion in a rotating flow field. The velocity of particle described in the S-Z&B model will be closed to the limit in an Infinite rotation space. Additionally, forces of particle is variable at different instantaneous positions in the rotating air flow. Therefore, particles tend to keep infinitely accelerating motion by the trajectory of the spiral, with the force of them increasing gradually and the terminal velocity approaching infinity.

The D&A model and the B&B model is study particle motion systems under constant force. The velocity of the particle can reach the limit for a sufficient time, and it will slowly approach a stable terminal velocity under constant force.

T&Y model is improved by above model. In the T & Y model, droplets accelerate near the terminal velocity. In the engine duct, the droplets will follow the blades rotating by the spiral track. Before colliding with the inner wall of the engine, the droplet continues to accelerate and move rapidly towards the terminal velocity.

(2) When injected into the duct of the aircraft engine at a certain initial velocity, the droplet gradually changes from a linear motion to a circular motion around the center of the blade due to the rotating flow field of the blade. Droplets will be more susceptible to the influence of rotating flow field with low initial velocity.

(3) Before entering the blade, the trajectories of droplets moving at a faster speed approach a straight line. From the z-direction, the radial distribution of the droplets and the input pressure show a quadratic function. And the results show that the droplets will collide with the engine wall at about 2.3 meters. If the distance between the nozzle and the blade exceeds 2.3 m, most of the droplets in the jet do not reach the blade. So as to achieve the purpose of cleaning. At an input pressure of 20 psi, droplets from the nozzle after the movement to the z-direction 0.5 m, by the rotating flow field, began to occur in different degrees of deflection. In order to achieve better cleaning results, the nozzle and the blade should be controlled at a distance of 0.5 meters.

Conflict of interest

The authors confirm that this article content has no conflicts of interest.

Acknowledgment

This work is supported by science and technology project of Civil Aviation, China (20150218), the Fundamental Research Funds for the Central Universities, China (3122016D017), and Central University of China Civil Aviation University Project, China (3122014D020)

References