Modeling and Simulation of Bow Wave in Aerial Refueling

Kunxian Zhai, Suiping Zhou*, Xiaoping Li and Yanming Liu
School of Aerospace Science and Technology, Xidian University, 710126, China
*Email: spzhou@xidian.edu.cn

Abstract. In a probe-and-drogue aerial refueling system, the tanker docking system consists of a hose and a drogue. When the receiver approaches the drogue, the dynamics of the drogue may be seriously disturbed by the aerodynamic effect from the receiver which is commonly known as the bow wave effect. In this paper, a numerical simulation method based on potential flow theory is proposed to investigate the bow wave effect. The proposed method is efficient and can be conveniently used in the modelling and simulation of autonomous aerial refuelling system. The results of this method are highly consistent with the CFD results, which demonstrates the effectiveness of the proposed method.

1. Introduction

The aerial refueling technology can greatly improve the endurance of the aircraft and is highly valued in the airline industry. Although the probe-and-drogue aerial refueling technology has made significant progress during the past few decades, it still remains as a challenging task for most pilots. In addition, as more and more UAVs (Unmanned Aerial Vehicles) will be used in the aviation industry, the autonomous refueling of the UAVs has attracted many researchers in recent years, which has a very high requirement on the accuracy of the refueling system.

In the probe-and-drogue aerial refueling process, the bow wave effect is one of the main factors that contribute to the docking failure. Due to the complex aerodynamics involved, modeling the bow wave effect is still a challenging research problem. In [1], the effects of the bow wave is analyzed based on look-up tables obtained from CFD simulations. Dai and Wei used a series of line doublets to simulate the nose and cockpit of the receiver aircraft, and developed a correction function to reduce the modeling error [2]. In 2018, Zhang and Hu proposed a modeling method based on potential flow theory, where the fuselage is treated as a Rankine body [3]. Zhong and Li select several velocity field data and model the bow wave by fitting and interpolation [4].

Considering the modeling convenience and accuracy, this paper proposes a method based on potential flow theory to model the bow wave effect. The main contributions of this paper are: (1) the method considers the geometrical characteristics of the head of receiver, so it is suitable for different types of aircraft (2) the obtained model can be easily applied to the controller design and the real-time simulations due to its computational efficiency and modeling accuracy.

The remainder of this paper is organized as follows. Section 2 presents the bow wave effect modeling assumptions and major methods involved. Section 3 presents the results of the proposed method and also compares the results the proposed method and CFD simulation, which indicates that the modeling method is effective. Conclusions and future work are summarized in Section 4.
2. Bow wave effect modeling

2.1. Modeling assumptions
Figure 1 (a) shows the flow field around the receiver. Many parts of the receiver, such as the head, the cockpit, the wing, the probe, may affect the bow wave effect. For some aircraft with special configurations, the wings, air inlet or the propeller may also have an effect on aerodynamics of the drogue [2]. In general, the bow wave effect can be approximated based on the principle of flow field superimposition as shown in (1):

\[ W_{\text{bow}} = W_{\text{nose}} + W_{\text{cockpit}} + W_{\text{wings}} + W_{\text{probe}} + W_{\text{others}} \]

\[ (1) \]

\[ (a) \quad (b) \]

Figure 1. CFD modeling of velocity distribution of F-16 in flow field.

Since the flow field attenuates rapidly with increasing distance from the drogue [2], the nose of receiver and the cockpit are the main causes of the bow wave, as shown in figure 1(b). In this paper the equation (1) is simplified as

\[ W_{\text{bow}} = W_{\text{nose}} + W_{\text{cockpit}} \]

\[ (2) \]

2.2. 2-D Bow wave effect modeling method
First, we present the panel method to model the 2-D bow wave effect. To solve aerodynamic problems, the panel method discretizes the surface of an object, and places singularities (such as source, vortex, doublets or their combinations) on the panel.

Assuming that a body is placed in a uniform flow of speed U [5], the velocity potential of the flow field is superimposed by the uniform flow and the source which placed on panels to obtain a total velocity potential of the form

\[ \phi = Ux + \varphi \]

\[ (3) \]

\[ \varphi \] represents the velocity potential due to the sources on the panels.
According to the panel method, the potential flow around a body of any given shape can be modeled by a distribution of sources over its surface. As shown in Figure 2, the first step is to discretize the 2D contour and divide it into N panels. The velocity potential at P due to sources on panel i is given by

\[ \Delta \Phi_{pi} = \sigma_i \int_{-\Delta s/2}^{\Delta s/2} \ln R_{pi} ds \]  

(4)

\[ R_{pi} = \sqrt{(x_p - x_i)^2 + (y_p - y_i)^2} \]  

(5)

where \( \sigma_i \) is the source strength, \( R_{pi} \) is the distance from P to center of the panel i, \( (x_p, y_p) \) is the coordinates of P; \( (x_i, y_i) \) is the coordinates of the center of panel i, ds is the length of the panel.

The velocity potential at P due to all sources is

\[ \Phi_P = \sum_{i=1}^{N} \Delta \Phi_i = \sum_{i=1}^{N} \sigma_i \int_{-\Delta s/2}^{\Delta s/2} \ln R_{pi} ds \]  

(6)

Thus, following equation (3), the total velocity potential at P can be written as

\[ \phi_P = U_x + \Phi_P = U_x + \sum_{i=1}^{N} \sigma_i \int_{-\Delta s/2}^{\Delta s/2} \ln R_{pi} ds \]  

(7)

In order to use equation (7) for numerical modeling, the source strength on panels is required. Next, we discuss how to obtain source strength.

Similar to equation (6), the velocity potential at panel i due to sources on other panels is given by

\[ \Phi_i = \sum_{j=1}^{N} \sigma_j \int_{-\Delta s/2}^{\Delta s/2} \ln R_{ij} ds \]  

(8)

The normal velocity of the source panel is

\[ V_{ni} = \frac{\partial \Phi_i}{\partial n_i} \left( \text{when } i = j, \ V_n = \pi \sigma_i \right) \]  

(9)

Superimposed with the uniform, and because of the boundary condition which the normal velocity is 0, we can get a set of equations with N unknowns and N equations as shown in (10). By solving these equations, the source strength of each panel can be obtained.

\[ V_n = U \cdot \hat{n}_i + \pi \sigma_i + \sum_{j=1}^{N} \sigma_j \int_{-\Delta s/2}^{\Delta s/2} \frac{\partial \ln R_{ij}}{\partial n_i} ds = 0 \]  

(10)

Once the source strength of each panel is obtained, the velocity potential at P can be obtained through equation (7). The velocity at P is then given by

\[ \begin{align*}
  u &= \frac{\partial \phi_P}{\partial x} = U + \sum_{i=1}^{N} \sigma_j \int_{-\Delta s/2}^{\Delta s/2} \frac{\partial \ln R_{pi}}{\partial x} ds \\
  v &= \frac{\partial \phi_P}{\partial y} = \sum_{i=1}^{N} \sigma_j \int_{-\Delta s/2}^{\Delta s/2} \frac{\partial \ln R_{pi}}{\partial y} ds
\end{align*} \]  

(11)
where \( u \) and \( v \) are the velocity components in the \( x \) direction and \( y \) direction.

### 2.3. 3-D Bow wave effect modeling method

For 3-D objects, the solution method is similar to the 2-D case, where the 3-D surface is discretized into a number of 2-D planes. The distance in equation (5) is changed to

\[
R_{p_i} = \sqrt{(x_p - x_i)^2 + (y_p - y_i)^2 + (z_p - z_i)^2}
\]  

(12)

where \((x_p, y_p, z_p)\) is the coordinates of \( P \), \((x_i, y_i, z_i)\) is the coordinates of the center of panel \( i \). The major process involved is similar to the 2-D case, however, the difficulty in the 3-D case lies in how to discretize the 3-D surface.

As most aircraft has a symmetric body along the vertical plane of the body coordinate system, we can simplify the computation procedure as described below.

\[
x_p = x, y_p = \sqrt{y^2 + z^2}
\]  

(13)

By substituting \((x_p, y_p)\) into equation (8), the 2-D velocity can be obtained, and the 3-D velocity vector can be obtained by

\[
\begin{align*}
    v_x' &= u(x_p, y_p) \\
    v_y' &= \frac{y}{\sqrt{y^2 + z^2}} v(x_p, y_p) \\
    v_z' &= \frac{z}{\sqrt{y^2 + z^2}} v(x_p, y_p)
\end{align*}
\]  

(14)

However, since the cross section of most receivers is not circular but elliptic, coordinate transformation is required, as shown in Figure 3, and the new velocity component is closer to the actual situation.

\[
v_x = v_x', v_y = \frac{b}{a} v_y', v_z = v_z'
\]  

(15)

![Figure 3. Coordinate transformation.](image)

### 3. Simulation results and analysis

To analyze the effectiveness of the proposed modeling and computation method as described in Section 2, we have implemented the proposed method in the MATLAB on a laptop computer. For comparison, we generate the flow field results using the commercial software Fluent. We use Tecplot360 to display the results.

#### 3.1. 2-D bow wave simulation

Figure 4 shows the contour plots of the velocity field. To generate the results, we discretize the 2-D model into 364 panels. In Figure 4, (a) and (c) are the \( x \)-direction and \( y \)-direction velocity distribution generated from our proposed method, (b) and (d) are the \( x \)-direction and \( y \)-direction velocity distribution.
generated from the CFD method. During the docking process, we pay more attention to the flow field near the receiver nose and the cockpit. The obtained velocity distribution characteristics are consistent well with the CFD results, the error is less than 3%. The average calculation time of our method is 0.1s.

![Generated from CFD method](image1)

![Flow field near receiver nose and cockpit](image2)

**Figure 4.** 2-D bow wave model velocity distribution from proposed method((a), (c)) and CFD ((b), (d)).

Next, we observe the velocity characteristics with the change of the angle between the axis of the fuselage and the relative airflow, and observe the cases where the angle is 5°, 10°, and 15°, as shown in Figure 5, 6, and 7. In the figures, (a) and (c) are the x-direction and y-direction velocity distribution generated from our proposed method, and (b) and (d) are the x-direction and y-direction velocity distribution generated from the CFD method. It can be seen that the velocity distribution of our method and the CFD result are consistent. As the angle increases, the accuracy decreases, and the error increases to about 10%. The average calculation time of our method is 0.1s.

![Velocity distribution at 5°](image3)

![Velocity distribution at 10°](image4)

![Velocity distribution at 15°](image5)

**Figure 5.** Angle = 5°, velocity distribution from proposed method((a), (c)) and CFD ((b), (d)).
Figure 6. Angle = 10°, velocity distribution from proposed method((a), (c)) and CFD ((b), (d)).

Figure 7. Angle = 15°, velocity distribution from proposed method((a), (c)) and CFD ((b), (d)).

The increase in the error as the angle can be explained as follows. In our propose method, we ignore the air viscosity and the air friction, therefore when the angle of attack increases, the increase in the lift is simply ignored. This error can be reduced if the effect of air viscosity, friction, and attack of angles is
included in our model by introducing a correction factor based on some CFD results or wind tunnel test results.

3.2. 3-D bow wave model simulation

The 3D receiver head model as shown in Figure 8. Figure 9 and Figure 10 show the bow wave velocity distribution. In the figures, (a)(c)(d) are the x-velocity, y-velocity, z-velocity distribution generated from the proposed method, and (b)(d)(f) are the x-velocity, y-velocity, z-velocity distribution generated from CFD method. It can be seen that in the important place near the aircraft nose and the cockpit, the simulation results and the CFD results are still consistent with each other. The average calculation time of our method is 0.13s.

Figure 8. The 3D receiver head model.

(a) (b)

(c) (d)

(e) (f)

Figure 9. 3-D bow wave model velocity distribution from proposed method and CFD at panel Z=0 (note that at panel Z=0, the z-direction velocity is almost equal to 0 everywhere, so in figure (e) and (f), the results shown are on the panel Z = -0.25).
The small computation error of our method is mainly due to:

- the inherent defects of the panel method lead to some errors in the 2-D model, and the 3-D model is based on the 2-D model, which result in the accumulation of errors;
- the nose and cockpit are simulated by two ellipsoids. The overlap between them leads to a superposition of calculations, thus the result is slightly larger than the CFD result;
- the entirety of object geometry in the panel method. The nose and the cockpit can only be deformed at the same time, although the eccentricity of the two cross sections is different.

4. Conclusions and future work

This paper proposes a method based on potential flow theory to model the bow wave. The method considers the geometrical characteristics of the head of receiver, which can be used for different types of aircrafts. Experimental results from the proposed method are consistent with the CFD results, and the
average computation time for the 3-D cases used in the simulation experiments is around 0.13s using MATLAB on a laptop computer. Therefore, the propose method could be used in the controller design for UAVs autonomous aerial refueling and some real-time simulations.

Our future work will includes: (1) refining the model to improve the accuracy; and (2) applying the proposed model to the autonomous air refueling system.

References
[1] Bhandari U, Thomas P R, Bullock S and Richardson T S 2013 Bow wave effect in probe and drogue aerial refueling AIAA 2013-4695
[2] Dai X H, Wei Z B and Quan Q 2016 Modeling and simulation of bow wave effect in probe and drogue aerial refueling Chinese Journal of Aeronautics 29(2) pp 448-461
[3] Zhang X F et al 2018 J. Phys.: Conf. Ser. 1064 012045
[4] Zhong Z W, Li D W, Wang H L and Su Z K. Modeling and analysis of receiver-aircraft bow wave based on fitting and interpolation. Electronics Optics & Control 2018 25(2) pp 69-73.
[5] Houghton E L et al 2012 Aerodynamics for Engineering Students 6th ed (Waltham: Butterworth-Heinemann) chapter 3 pp 195-203