Simulation Calculation and Analysis of the Influence of Wake Jet on Aircraft Tail Flow Field

Wenlin Liu¹, Pengyu Zhou¹,²*, Xiongfei Yu³, Zhijie Song¹ and Jinku Wang³

¹Naval Aviation University, Yantai, China
²Unit 92326 of PLA, Zhanjiang, China
³Unit 93279 of PLA, Dandong, China

*Corresponding author email: ytuuszb@ytu.edu.cn

Abstract. In order to analyze the influence of the wake jet on the tail flow field during the take-off and landing phase of the aircraft, this paper uses Computational Fluid Dynamics technology, takes an aircraft as the research object, and uses the RANS equation, Realizable k-ε turbulence model and the Delaunay unstructured meshing method. The Roe discrete scheme is used to numerically simulate the wake field of the aircraft using the coupled implicit algorithm, which is divided into two conditions: wake jet and no wake jet. The simulation results show that there is a certain back pressure zone in the wake field of the aircraft, the gauge pressure range is -272~1067 Pa, the pressure difference at the same position in the wake field area is small, the range is -240~60 Pa, and the wake field pressure has little correlation with the wake jet, and there is no separation vortex under the two conditions. The simulation results verify the rationality and effectiveness of the simulation calculation, and provide a certain theoretical reference for the safe take-off and landing of the aircraft.

Keywords: Aircraft; Wake jet; Tail flow field; Meshing; Numerical simulation.

1. Introduction
Aircraft wakes are mainly divided into the following types: wingtip vortexes generated at the wingtips of the wings, turbulence generated by the airflow on the surface of the aircraft, high temperature and high-speed jets generated by the tail nozzle of the engine, and slippage caused by the high-speed rotation of the propeller of a propeller aircraft flow. A series of possible effects on aircraft wake fields have been studied at home and abroad for many years. However, there are few studies on the wake fields of aircraft on the ground taxiing state, and the data is difficult to consult. In this paper, using computational fluid dynamics technology, taking the digital model of a high-speed aircraft as the research object, based on the Delaunay unstructured meshing method, the coupled implicit algorithm is used to divide the jet flow and the non-tail jet flow into the tail flow field. The numerical simulation and analysis are compared. The results verify the rationality of grid generation and simulation calculation, and the visualization of the flow field provides a theoretical reference for pilots to safely control aircraft take-off and landing, and scientific researchers to carry out related simulation research.

2. Geometric Modeling and Meshing

2.1. Geometric Modeling
During take-off and landing, the aircraft increases the wing area, changes the wing camber, and delays the airflow separation of the wing during take-off and landing, thereby increasing the lift at low speeds, which is conducive to the rapid take-off and slow landing of the aircraft, and ensures flight safety.
Whether the flaps are lowered or not will have a certain impact on the outer flow field of the aircraft. To improve the simulation accuracy and simulate the real taxiing state of the aircraft, the front and rear edges of the wing are divided into the leading edge flaps and flaperons according to the actual ratio. Decrease the specifications and deflect downwards respectively. The digital model of the aircraft before and after the flaps is placed is shown in Figure 1.

![Aircraft digital model](image)

**Figure 1.** Aircraft digital model.

### 2.2. Meshing

The grid is a series of discrete points in the simulation calculation domain\(^3\). Computational Fluid Dynamics uses discrete control equations and numerical calculation methods to obtain physical characteristic data on each grid node. The speed, pressure, temperature and other data obtained are the numerical solutions of the simulation calculations we need\(^4\). The model has a complex shape and structure, and uses Delaunay unstructured triangular grid generation technology, which has better adaptability to irregular and complex configurations. The significant advantage of Delaunay triangulation is that it can make the minimum angle of each triangle mesh as large as possible, which can significantly improve the efficiency of unstructured mesh generation\(^5\).

The surface of the aircraft is divided into a triangular unstructured grid, the grid size is about 0.05% of the fuselage size. The tail part of the aircraft is the position of the parachute cabin of the deceleration parachute, which is the key area of the simulation in this paper. It needs to be encrypted separately to ensure that the flow field simulation results are more accurate in the later stage. The size of the far field part is divided according to the overall size of the fuselage. Figure 2 shows the grid distribution of the fuselage and the far field.

![Grid distribution](image)

(a) Aircraft surface (b) Far field

**Figure 2.** Grid distribution

Table 1 shows the grid size of each area.

| Area                        | Grid size |
|-----------------------------|-----------|
| Aircraft surface            | 0.1       |
| Caudal surface              | 0.05      |
| Wake field encryption area  | 0.1       |
| Ground effect encryption area| 1         |
| Far field boundary          | 10        |

| Table 1. Grid size.        |
|---------------------------|

Because the turbulence model needs to be used in the later numerical calculation of the solver, the turbulence model referred to here is usually a fully developed turbulence, which is mostly used in high Reynolds number flows\(^6\). In the near-wall area, due to the effect of viscosity, the Reynolds number is not large and the development of turbulence is insufficient. Therefore, special boundary layer processing is required in the near-wall area, and this work must be completed when dividing the mesh.
The thickness of the first layer of the boundary layer of the simulation grid is 0.001 m, the maximum number of boundary layers is set to 100, the growth rate is 1.3, the boundary attenuation coefficient is 0.9, and the spatial volume grid types are all categories, including tetrahedron, pyramid, prism and hexahedron. The grid inspection has no negative volume and the quality meets the calculation requirements. Figure 3 shows the visualization of the Z=0 vertical section volume grid at an angle of attack of 7°.

![Figure 3. Z=0 longitudinal section volume mesh.](image)

3. Theoretical Model and Numerical Method

3.1. Theoretical Model

In the field of modern aeronautical engineering applications, accurate numerical methods are required to simulate the viscous flow around complex-shaped aircraft in order to predict aerodynamic characteristics\[7\]. The N-S equation (RANS) of the Reynolds average equation (RANS) introduced into the turbulence model can accurately calculate the aerodynamic force of the aircraft, and it is the mainstream method of numerical calculation of the current flow field\[8\]. The governing equation adopts the conservative form, and the RANS equation is as follows:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0
\]

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu + \frac{\partial u_i}{\partial x_j} \right] + \frac{2}{3} \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]
\]

\[
\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_i} (\rho u_i E) = -\frac{\partial p u_i}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \frac{2}{3} \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]
\]

Among them, \( \mu_{eff} = \mu + \mu_t \), \( \mu_{eff} \) is the effective viscosity, \( \mu \) is the molecular viscosity, \( \mu_t \) is the turbulent viscosity.

3.2. Numerical Method

In terms of numerical methods, a density-based steady-state solver, coupled implicit algorithm, and Roe discrete scheme, which is conducive to solving compressible flow, is adopted. Another outstanding advantage of the algorithm is that it can solve the whole velocity range, that is, the solution range is from low-speed flow to high-speed flow\[9\]. In terms of spatial discretization, the cell-based least squares method is used for the gradient term, the second-order upwind style is used for the flow term, and the first-order upturn style is used for both the turbulent kinetic energy term and the turbulent dissipation term\[10\]. After the equation is discretized, it is coupled and solved. The Coulomb number is 5, the sub-relaxation factor of the turbulent kinetic energy term and the turbulent dissipation term are both 0.8, and the sub-relaxation factor of the turbulent viscosity term and the solid term is 1. In terms of boundary conditions, when there is a tail jet, the cross section of the tail nozzle is set to the parameter value of high temperature, high pressure and high speed; when there is no tail jet, the cross section of the tail nozzle is set to the standard atmospheric pressure and the ambient temperature. The specific parameters of the tail jet boundary conditions are shown in Table 2.
Table 2. Boundary condition parameters.

|              | Wake jet condition | Far field inlet | Far field outlet | Jet inlet | Jet outlet |
|--------------|--------------------|----------------|-----------------|-----------|-----------|
| With jet condition | Total pressure: 105216 Pa | Static pressure: 101325 Pa | Static pressure: 101325 Pa | Total temperature: 291.3 K | Total temperature: 291.3 K |
|               | Static pressure: 101325 Pa | Total temperature: 291.3 K | Total temperature: 291.3 K | Total temperature: 291.3 K | Total temperature: 291.3 K |
| Without jet condition | Total pressure: 105216 Pa | Static pressure: 101325 Pa | Static pressure: 101325 Pa | Total temperature: 291.3 K | Total temperature: 291.3 K |
|               | Static pressure: 101325 Pa | Total temperature: 291.3 K | Total temperature: 291.3 K | Total temperature: 291.3 K | Total temperature: 291.3 K |

4. Simulation Result Analysis

4.1. Axial Analysis

In the axial direction, the characteristic within 4 m of the positive X-axis direction after the tail nozzle section is given. Figure 4 shows the pressure contrast cloud diagram of the Z=0 m longitudinal section with and without the tail jet.

![Pressure contrast cloud map with and without jet condition](image)

(a) With jet condition  
(b) Without jet condition

**Figure 4.** Z=0 longitudinal section pressure contrast cloud map

Figure 5 shows the comparison curve of gauge pressure characteristics with or without tail jets, starting from the intersection of the Y symmetry axis of the outer cross-section of the two tail nozzles and the surface of the tail cone, along the positive X-axis at intervals of 0.15 m, where X=1.65 m is the point taken on the surface of the tail cone, after X=1.65 m is the point taken in the horizontal direction of the X axis.

![Gauge pressure characteristic comparison curve](image)

**Figure 5.** Axial gauge pressure characteristic comparison curve.

From the image and curve analysis, it can be seen that with or without the wake jet, the pressure is based on the end point of the tail vertebra as the peak point of back pressure, and there is a downward trend on both sides of the axis; in the area of X=0.6 m, there are two conditions The bottom is a negative pressure vacuum zone lower than the standard atmospheric pressure, and the negative pressure value first drops and then rises; the pressure curve fits well under the two conditions, and the
pressure difference at the same position is -240~50 Pa. The wake jet has little effect on the size of the axial backpressure zone of the wake field.

4.2. Horizontal Analysis
In the lateral aspect, the characteristics are given in the range of 1.2 m in both positive and negative directions of the Z axis centered on the end point of the tail vertebra. Figure 6 shows the pressure contrast cloud diagram of the cross-section through the end point of the tail vertebra with and without the tail jet.

![Figure 6](image)

(a) With jet condition  (b) Without jet condition

**Figure 6.** Cross-section pressure contrast cloud map.

Figure 7 shows the comparison curve of the gauge pressure characteristics in the Z-axis direction with the tail point as the center at intervals of 0.15 m.

![Figure 7](image)

**Figure 7.** Horizontal gauge pressure characteristic comparison curve.

From the image and curve analysis, it can be seen that with or without the tail jet, the pressure takes the end point of the tail vertebra as the peak point of the back pressure, and shows a downward trend along the two sides of the Z axis, and the downward curve on both sides is more symmetrical; The pressure curve has a strong fit, and the pressure difference at the same site is -50~60 Pa. The wake jet has little effect on the size of the lateral back pressure zone of the wake field.

4.3. Longitudinal Analysis
In the longitudinal direction, the characteristics are given in the range of 1.2 m in the upper and lower directions of the Y axis centered on the end point of the tail vertebra. Figure 8 shows the pressure cloud diagram of the longitudinal section through the end point of the tail vertebra under the condition of wake jet.

![Figure 8](image)

(a) With jet condition  (b) Without jet condition

**Figure 8.** Vertical section pressure contrast cloud map.

Figure 9 shows the comparison curve of the gauge pressure characteristic every 0.15 m in the Y axis direction with the end point of the tail vertebra as the center.
It can be seen from the image and curve analysis that with or without the tail jet, the pressure is based on the end of the tail vertebra as the peak point of back pressure, and there is a downward trend along both sides of the Y axis; the pressure above the end of the tail vertebra continues to decrease until it approaches the outside Atmospheric pressure tends to be flat after 0.6 m below, and it becomes a stable high-pressure area due to ground effect. The pressure curves under the two conditions have a strong fit, and the pressure difference at the same site is -70~30 Pa. The wake jet has little effect on the longitudinal backpressure zone of the wake field.

5. Conclusion
In summary, there is a certain back pressure area in the wake field of the aircraft, the gauge pressure range is -272~1067 Pa, the pressure difference at the same position in the wake field area is small, the range is -240~60 Pa, because of this model The tail cone is close to 3 m from the tail nozzle, and the distance is far. The high pressure gas generated by the tail jet dissipates quickly when leaving the tail nozzle, and it is close to the standard atmospheric pressure near the tail cone. The wake field characteristics are related to the tail jet flow. The simulation results verify the rationality and effectiveness of the simulation calculation, and provide a certain theoretical reference for the safe take-off and landing of the aircraft.

References
[1] L. S. Ko, V. Tremblay-Dionne, T. Lee. Impact of Ground Proximity on an Inverted Delta Wing [J]. Journal of Aerospace Engineering, 2020, 33(5): 22-23.
[2] AHMED M R, TAKASAKI T, KOHAMA Y. Aerodynamics of a NACA4412 airfoil in ground effect [J]. AIAA Journal, 2007, 45(1): 37-47.
[3] HITZEL S M, WEIDE E, TREMEL U. X-31A vector high angle of attack descent Euler and Navier-Stokes simulations of unsteady maneuvers [C] // Proceedings of 23rd International Congress of Aeronautical Sciences. Toronto: ICAS Press, 2002, 243(6): 1-9.
[4] C. Deiler, T. Kilian. Dynamic aircraft simulation model covering local icing effects [J]. CEAS Aeronautical Journal, 2018, 9(3): 56-58.
[5] Yicong Gao, Zixian Zhang, Yixiong Feng, Maria Savchenko, Ichiro Hagiwara, Hao Zheng. Flexible mesh morphing in sustainable design using data mining and mesh subdivision [J]. Future Generation Computer Systems, 2020, 108(5): 113-114.
[6] Rajesh Kumar, Nirmal Kant Singh. Large eddy simulation of flow over square cylinder arrays in an octagonal configuration at subcritical Reynolds numbers [J]. Heat Transfer, 2020, 49(6): 56-58.
[7] Chandrareshkar R. Jadhav, Rashmi P. Chorage. Modification in commercial bus model to overcome aerodynamic drag effect by using CFD analysis [J]. Results in Engineering, 2020, 6(3): 19-21.
[8] Can Cao, Matthias Kraume. Hysteresis effect of propeller jet flows in viscoelastic fluids: Steady state flow patterns [J]. Chemical Engineering Science, 2020, 223(9): 24-25.
[9] Norum T D. Supersonic rectangular jet impingement noise experiments [J]. Aiaa Journal, 2015, 29(7): 1051-1055.
[10] Goraj Z. Design and optimisation of fuel tanks for BWB configurations [J]. Archive of Mechanical Engineering, 2016, 63(4): 605-617.