Numerical Investigation of the Aerodynamic Characteristics of a Missile

Yu-shan Meng¹, Li Yan¹, Wei Huang¹† and Xin-yu Tong²

¹College of Aerospace Science and Engineering, National University of Defense Technology, Changsha, Hunan 410073, China
²Northwestern Polytechnical University, Xian, Shanxi 710072, China

†E-mail: gladrain2001@163.com

Abstract: During supersonic flight of supersonic missile, the missile configuration has great influence on the aerodynamic characteristics of the missile. This article evaluates the variation of the flow field results from the change of missile configuration by changing the shape of the head and the slenderness ratio of the body. The study is carried out by the CFD numerical method, and the structured grids are generated by ANSYS CFD software. We evaluate the aerodynamic characteristics of two-dimensional model and three-dimensional model by the flow solver FLUENT, and the standard Spalart-Allmaras one-equation is selected to simulate the flow around the missile, the results indicate that the drag coefficient of the oval head is less than parabolic head case and conical head case, and drag coefficient decreases with the increase of Mach number and slenderness ratio. With the increase of Mach number, the stagnation temperature increases correspondingly.

Keywords: Supersonic missile; Head shape; Drag coefficient; Aerodynamic characteristics

1. Introduction
The aerodynamic characteristic of missile is of great significance for the design and usage of missile. During the supersonic flight of missile, it encounters strong shock wave such as the bow shock which enables the rise of wave drag[1], especially for the supersonic aircraft with blunt body[2]. And there exists serious aerodynamic heating and large entropy gradient in the zoon close to the leading edge of the aircraft, the air around the warhead is strongly compressed and is subjected to severe friction. And the degree of air compression increases as the Mach number increases, meanwhile, the temperature around the missile increases correspondingly.

Numerical simulation is a branch of CFD technology, with the improvement of the computer performance, a series of solution algorithm based on Euler method were proposed, such as explicit Runge-Kutta method[3]. In recent years, numerical investigations of the vehicles flying at supersonic and hypersonic speeds have been widely carried out[4-8]. Qin et al. [9] adopted a fluid-thermal coupled strategy to study the aerodynamic characteristics of different flow fields and various body configurations. The results show that the shock intensity and the drag coefficient increase correspondingly as the free stream Mach number increases. Yue et al.[10] in virtue of the CFD software to carry out the numerical computations of aerodynamics of a missile with the Mach numbers and attack angle being different. The results show that the numerical simulation have special reference value for the design of a missile. Hao et al. [11] studied the transonic-supersonic wind tunnel
experiment on the aerodynamics of the rockets and missiles with different fins. The results indicate that adopting more than six fins is beneficial for increasing the stability for low speed missiles while it is adverse for high speed missiles. Besides, Zhang et al. [12] has summarized parametric approaches specific to aerodynamic design process, and Huang et al. [13-16] numerically studied the performance of different thermal protection in details.

In this paper, we analyze the effect of the missile configuration on the aerodynamic characteristic, compare the impact of Mach number and discuss the flow characters around the missile. In Section 1, we briefly introduce the researches on the blunt body. Section 2 is an introduction to the governing equations, turbulence model and solving settings. The discussions and results are introduced in Section 3. And some remarks and conclusions are drawn in the last section.

2. Physical and numerical models

2.1 Governing equations and Turbulence model

The basic governing equations are continuity equation, momentum equation and energy equation, here we start from the three-dimensional steady NS equations to carry out the numerical simulation[17-18].

\[
\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = \frac{\partial F_v}{\partial x} + \frac{\partial G_v}{\partial y} + \frac{\partial H_v}{\partial z}
\]

Where \( U \) is a conserved quantity, \( F,G,H \) are inviscid convective fluxes , and \( F_v, G_v, H_v \) are sticky fluxes.

\[
U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ E \end{bmatrix}, \quad F = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uw \\ (E + p)u \end{bmatrix}, \quad G = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho vw \\ (E + p)v \end{bmatrix}, \quad H = \begin{bmatrix} \rho w \\ \rho uw \\ \rho vw \\ \rho w^2 + p \\ (E + p)w \end{bmatrix}
\]

Where \( \rho \) is the density, \( u,v,w \) are used to measure the X-component, Y-component and Z-component of the velocity. Also, \( p \) is the static pressure, \( E \) is the total energy. The state equation is

\[
P = (\gamma - 1)[e - \frac{1}{2} \rho (u^2 + v^2 + w^2)]
\]

The governing equations are discretized and solved by finite volume method. We adopt the standard Spalart-Allmaras one-equation to carry out the numerical simulation based on \( \bar{v} \), the kinematic coefficient of viscosity, and the follow is the transport equation

\[
\rho \frac{D \bar{v}}{Dt} = G_v + \frac{1}{\sigma_v} \frac{\partial}{\partial x_i} \left( \mu + \rho \bar{v} \frac{\partial \bar{v}}{\partial x_j} \right) + C_{b2} \rho \left( \frac{\partial \bar{v}}{\partial x_j} \right) - Y_v
\]

Where \( G_v \) denotes the production of the turbulent viscosity, \( \sigma_v, C_{b2} \) are the constant term, and \( Y_v \) denotes the decrease of the turbulent viscosity.

2.2 Geometry

In this section we mainly study the impacts that the head shape has on the aerodynamic characteristic of missile. We select a long-range air defense missile without main wings as the basic model, it has a cylindrical body with an oval head, and there exists a spherical transition from head to body. The total length of the body is 5.3 m, the length of the head is 0.98794 m, of which the radius of curvature is 10mm, and the diameter of the cylindrical body is 0.41 m, see Fig.1. In this paper, the aerodynamic interference among the structure would not be considered.

**Figure 1.** The geometry model of the missile.

**Figure 2.** Structured grids of the model.
2.3 Solver setting
We establish a rectangular far field with the dimension of twenty times as large as the basic model in ICEM-CFD. This paper uses the local grid refinement approach to refine the structured grids of the head and due to the assumption that the model is symmetrical, only half of the mesh needs to be generated. The Reynolds number $3.31472 \times 10^6$ is based on the diameter of the body[19]. We set the desired $Y+$ value as 30, which is suitable to calculate the first cell thickness, the first layer grid height of the boundary is obtained as $4.59703 \times 10^{-5}$ m, and the numerical simulation is carried out using the medium size of grids(1346860 cells), the accuracy of which is sufficient, see Fig. 2. After importing the mesh into FLUENT, we choose two-dimensional accuracy solver to carry out the calculation. Table 1 presents the parameters of the free stream. The analysis assumes that the fluid is idea-gas, there is no slip for velocity and adopts S-A turbulent model and second-order accuracy AUSM + spatial discretization scheme[20]. And the Courant number changes as the calculation goes on to accelerate the first layer grid height convergence. The monitors of drag coefficient and lift coefficient are defined and we assume the calculation is finished when the residuals decrease to five orders of magnitude.

3. Results and Discussions

3.1 Effect of Mach number
We obtain the aerodynamic contours of the missile with the angle of attack being 0° and the Mach number being 4 Ma. See Figs 3-4. Fig.5 depicts that the drag coefficient decreases as the free stream Mach number increases, and the slope decreases gradually as well. After carrying out the analysis, it is of great significance to learn the flow characteristic around the missile. It can be observed that when the free stream flows around the missile, it encounters a strong compression which lead to the shock wave. The basic model has an oval head, and the radius of curvature becomes larger as the stream keep moving backward, thus a series of weak shock waves are formed. The velocity of the airflow encounters a sharply decline while the pressure of the warhead encounters a sharply increase. At the transition from head to body, a series of continuous sectorial expansion waves occur, so it is concluded that the distribution of the flow field is affected by a series of shock waves and expansion waves.

| Table 1. Parameters of free stream. |
|------------------------------------|
| Mach number | $M_{\infty}$ | 4 |
| Static pressure | $p_{\infty}$ | $p_a$ | 12111.8 |
| Static Temperature | $T_{\infty}$ | $K$ | 216.65 |
| Reynolds No. | $Re_{\infty}$ | million/m | 3.31472 |
| Density | $\rho_{\infty}$ | kg/m$^3$ | 0.194755 |
| Velocity | $u_{\infty}$ | m/s | 1180.28 |

Figure 3. Mach cloud of reference model.

Figure 4. Pressure cloud of reference model.

Figure 5. Effect of Mach number on Drag coefficient.

3.2 Effects of the head shape
In this section, we discuss the aerodynamic characteristics of missiles with different head shapes, see Fig.6. During the numerical simulation, Mach number is changed as 2, 2.5, 3, 3.5, 4, 4.5 respectively.
Fig. 7 presents that the variation trend of drag coefficient is similar with the basic model. The drag coefficient of the missile with Parabolic head is the largest while it of the missile with oval head (basic model) is the smallest. The strength of the shock wave increases as the radius of curvature of head decreases, it means that the blunter the head is, the stronger the shock waves are. The radius of curvature of parabolic head is the largest among these configurations, and the intensity increases correspondingly.

![Figures](image1)

**Figure 6.** Geometry of the models with different head shape.

![Figures](image2)

**Figure 7.** Comparison of Drag coefficient with respect to head shape.

![Figures](image3)

**Figure 8.** Temperature distribution on the surface of the missile with the Mach number being 3.5 Ma.

![Figures](image4)

**Figure 9.** Comparison of stagnation temperature with respect to head shape.

![Figures](image5)

**Figure 10.** Effect of Slenderness ratio on Drag coefficient.

Fig. 8 presents the aerodynamic heating characteristics along the missile, and the variation trend from stagnation point to the junction of head and body is worthy of attention. We can observe that the variation trend of oval head tends to be linear, which, results in the linear variation of the thermal stress. Fig.9 presents that with the increase of Mach number, the stagnation temperature increases correspondingly. And the curves of different head shapes basically agree with the theoretical curve. It is concluded that the radius of curvature of missile head has little effect on the stagnation temperature. Fig.10 presents the effect of slenderness ratio on drag coefficient in MATLAB.

3.3 Effects of the slenderness ratio
Take the missile with parabolic head as the object of study, remain the Mach number being 4 Ma unchanged, and change the slenderness ratio as 2, 3, 4, 5 respectively. The comparison of Mach number contour is presented in Fig.11. The results show that there exists strong detached bow shock wave in front of the head and a zone with high temperature and high-pressure area is formed, which, is the reason for the inevitable rise of drag coefficient, and the drag coefficient decreases as the slenderness ratio increases. In addition, the pressure and temperature increase significantly while the velocity decreases when the stream pass through the shock wave.

![Fig 11. Comparison of Mach number contour with different Slenderness ratio.](image)

When there exists a hemispherical head, the aerodynamic feature is more remarkable, see Fig.12. The radius of curvature of the blunt head is large and it has big angle with the air flow. The head hinders the free stream from moving forward and there is an obvious detached bow shock wave, which, creates stronger energy consumption than the attached shock wave.

3.4 Y + Self-adaptive grid technique
FLUENT software has the self-adaptive grid technique, such as boundary self-adaptive technique, isoline self-adaptive technique and Y + self-adaptive technique, which, can improve the computational accuracy by adjusting the degree of grid density and refining the grid in the area with large gradient. In this paper, we use Y + self-adaptive grid technique to prove the accuracy of our calculation.
Fig. 13 presents the Y+ distribution before and after the self-adaption, we choose the basic oval head model to carry out the self-adaptive calculation with the Mach number being 4 Ma and the angle of attack being 15°. It is obtained that Y+ value before self-adaption is higher than Y+ value after self-adaption, which is mainly because that we set the first cell thickness as a fixed value and the head of missile suffers large pressure. After self-adaption, the Y+ value is distributed around 1. Table 2 shows that before and after the self-adaption, there is little difference among the results, which, further verifies the accuracy of the analysis above.

| Self-adaption times | Lift coefficient | Drag coefficient | Lift-drag ratio |
|---------------------|------------------|------------------|-----------------|
| 0                   | 0.14744          | 0.059625         | 2.473           |
| 1                   | 0.14793          | 0.059763         | 2.475           |
| 2                   | 0.14832          | 0.059804         | 2.480           |
| 3                   | 0.14869          | 0.059853         | 2.484           |

3.5 Simulations of the three-dimensional missile
In order to further study the aerodynamics of the missile, based on the above analysis, in this section, simulations of the three-dimensional missile are done. The flow field is investigated with the angle of attack being 0° and 30°. Mach number contour with different angles of attack are presented in Fig.14, and the position of the stagnation point gradually moves down. When there is an angle of attack being 30°, a series of oblique shocks are generated, which are related to the deflection angle of the free stream. In addition, the lower surface is the windward side, which is the reason for the separation of airflow occurs on the upper surface of the missile.
Fig. 14. Mach number cloud with different angles of attack.

Fig. 15 presents the streamline in the rear of missile with angle of attack being 30°, a clockwise vortex can be seen clearly, which, may be the reason for the temperature decrease. There exists a large velocity gradient within the boundary layer region, and the temperature of boundary layer is high, which, is related to the viscous friction. We can observe the temperature distribution of the rear of missile with different angle of attack clearly in Fig. 16. Temperature curves of upper and lower surfaces with different angle of attack are obtained in Fig. 17. We can obtain that upper and lower surfaces are at the same temperature when the angle of attack is 0°, and the temperature of lower surface is higher than it of upper surface. The temperature of lower surface increases as the angle of attack increases, which, is related to the increase of windward area, and there is a downward trend of the curve as the lengthwise position changes and finally comes closer to balance since the distance from the stagnation point increases.

Figure 15. The streamline in the rear of missile with angle of attack being 30°.

Figure 16. Temperature distribution of the rear with different cases. (Left: 0°, Right: 30°)
In order to study the process of aerodynamic heating more clearly, we use Fluent transient calculation to solve the external flow field of the basic model when the Mach number is 4 Ma and the angle of attack is 15°. The time step is set to 0.0001s, and the maximum iteration time is 50 times in each time step. We get temperature curves every 0.12 seconds as shown in Fig.18. It can be found that the temperature distribution on the upper and lower surface changes rapidly before 0.3s, but gradually slows down after 0.3s, and the whole flow field tends to be stable.
4. Conclusions

In this paper, an investigation on the aerodynamic characteristic of the missile has been discussed in detail. The flow distribution is studied and the drag coefficient for different configuration is compared by the CFD numerical method. Based on the above discussion the conclusions are obtained as follows.

(1) During the supersonic flight, the missile encounters strong shock waves, and the velocity of the airflow has a sharp decline while the pressure of the warhead has a sharp increase. The drag coefficient decreases as the freestream Mach number increases from 2 to 4.5, and the shock wave strength increases correspondingly.

(2) The missile configuration has significant impact on the aerodynamic characteristics. The strength of the shock wave increases as the radius of curvature of head decreases, and as the slenderness ratio increases, the drag coefficient presents a gradually trend of decrease.

(3) The position of the stagnation point changes as the angle of attack changes. The air around the warhead is strongly compressed and is subjected to severe friction. Therefore, the temperature of the missile is influenced by shock waves as well as the viscous friction, and a vortex can be seen clearly in the rear of the missile.

(4) The temperature of lower surface is higher than the temperature of upper surface when there exists an angle of attack and it increases as the angle of attack increases. And curves obtained by transient calculation presents that temperature distribution on the upper and lower surface changes rapidly before 0.3s, but gradually slows down after 0.3s, and the whole flow field tends to be stable. Further investigation will mainly focus on the aerodynamic heating of the hypersonic missile.
Conflict of interest statement
The authors declare there is no conflict of interest in relation to this manuscript.

References
[1] Narayana G, Selvaraj S. Experimental investigation of heat transfer over double disk spike-blunt body at Mach 5.7. Experimental Thermal and Fluid Science, 2019, 102: 452-466
[2] Pezzella G. Aerodynamic and aerothermodynamic design of future launchers preparatory program concepts. Aerospace Science and Technology, 2012, 23(1): 233–249
[3] Jameson A. Numerical Simulation of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Marching Schemes. AIAA paper, 1981, 81-1295
[4] Persova M G, Soloveichik Y G, Belov V K, et al. Modeling of aerodynamic heat flux and thermo elastic behavior of nose caps of hypersonic vehicles. Acta Astronaut, 2017, 136: 312–331
[5] Huang W, Zhao Z T, Yan L, et al. Parametric study on the drag and heat flux reduction mechanism of forward-facing cavity on a blunt body in supersonic flows. Aerospace Science and Technology, 2017, 71: 619–626
[6] Suzuki T, Aoki T, Ogasawara T, et al. Nonablative lightweight thermal protection system for mars aeroflyby sample collection mission. Acta Astronaut, 2017, 136: 407–420
[7] Shoev G, Oblapenko G, Kunova O, et al. Validation of vibration-dissociation coupling models in hypersonic non-equilibrium separated flows. Acta Astronaut, 2018, 144: 147–159
[8] Belov I A, Isaev, Ju Mitin A. Calculation of separated flows within the framework of ideal fluid with allowance for a turbulent shear layer on the boundary of the separation region. Journal of Engineering Physics and Thermophysics, 1986, 51 (4): 1159–1164
[9] Qin Q H, Xu J L, Guo S. Fluid-thermal analysis of aerodynamic heating over spiked blunt body configurations. Acta Astronautica, 2017, 132: 230-242
[10] Yue C G, Chang X L, Zhang Y H, Yang S J. Numerical Calculation of a Missile's Aerodynamic Characteristic. Advanced Materials Research, 2011, 1157
[11] Hao L, Wu J S. Wind Tunnel Experimental Investigation on the Aerodynamic Characteristics of the Multifin Rockets and Missiles. Journal of Beijing Institute of Technology (English Edition), 2005(03):293-296
[12] Zhang T T, Wang Z G, Huang W, Yan L. A review of parametric approaches specific to aerodynamic design process. Acta Astronautica, 2018, 145: 319-331
[13] Huang W. A survey of drag and heat reduction in supersonic flows by a counterflowing jet and its combinations. Journal of Zhejiang University-Science A, 2015, 16(7): 551–561
[14] Huang W, Jiang Y, Yan L, Liu J. Heat flux reduction mechanism induced by a combinational opposing jet and cavity concept in supersonic flows, Acta Astronaut, 2016, 121: 164-171
[15] Sun X W, Guo Z Y, Huang W, Li S B, Yan L. A study of performance parameters on drag and heat flux reduction efficiency of combinational novel cavity and opposing jet concept in hypersonic flows. Acta Astronaut, 2017, 131: 204-225
[16] Huang W, Chen Z, Yan L, Yan B B, Du Z B. Drag and heat flux reduction mechanism induced by the spike and its combinations in supersonic flows: A review. Progress in Aerospace Sciences, 2019, 105: 31-39
[17] Qu F, Sun D, Bai J Q, Zuo G, Yan C. Numerical investigation of blunt body’s heating load reduction with combination of spike and opposing jet. International Journal of Heat and Mass Transfer, 2018, 127: 7-15
[18] Zhao R, Wen C Y, Tian X D, et al. Numerical simulation of local Wall heating and cooling effect on the stability of a hypersonic boundary layer. International Journal of Heat and Mass Transfer, 2018, 121: 986-998
[19] Crawford D H, Investigation of the flow over spiked-nose hemisphere-cylinder at Mach Number of 6.8. NACA, 1959, 6 (1): 112-118
[20] Liou M S. A sequel to AUSM: AUSM +. Journal of Computational Physics, 1996, 129(2): 364-382