Numerical Simulation of Tail Flow Field of Four- Nozzle Solid Rocket Engine

Bailin Zha, Zhe Hui*, Qingdong Su
Rocket Force University of Engineering, Xi’an, P. R. China

*Corresponding author e-mail: HJJhuizhe@163.com

Abstract. A solid rocket motor with four nozzles is taken as the research object. According to its actual flight environment and the simulation boundary conditions are given. Then, the integrated simulation of nozzle-flow field in exhaust plume of solid rocket motor is carried out by means of FLUENT. The velocity of flow field in exhaust plume, temperature, and pressure are obtained. The simulation results are in good agreement with the actual flow conditions and the existence of Mach disk is proved.

1. Introduction
Multi-nozzle rocket engine is widely used in high thrust rocket and variable thrust rocket. The characteristics of wake flow field of multi-nozzle rocket engine have great influence on the overall design of rocket, the selection of materials and the design of launching device. Therefore, it is necessary to study the characteristics of wake flow field. With the rapid development of CFD technology, the study cost and studying time cycle of the wake flow field characteristics can be greatly reduced, and the details of the flow field can be comprehensively analyzed. In this paper, the characteristics of wake flow field of multi-nozzle rocket engine are numerically simulated and analyzed based on FLUENT, the commercial software of computational fluid mechanics.

Hideyo et. [1] used Fluent commercial software and coupled radiative heat transfer calculation model to carry out the three-dimensional simulation calculation of the H-IIA rocket and its wake flame flow field, and the heat effect of complex wake flame flow field structure and the backflow on the tail of the rocket were obtained. Xiao et. [2] took the space multi-nozzle engine as the model, the numerical simulation of aero-engine space wake flame was carried out using computational fluid dynamics (CFD) and Monte Carlo (DSMC) method, and the characteristics of the exhaust wake flame flow field of the space multi nozzle engine were obtained. Houshang et. [3] used GIFS solver to carry out the three-dimensional numerical simulation of the wake flame flow field of the first and third-level dynamical system of the militia III, certified the simulation validity of the GIFS solver, and obtained the parameters distribution of the wake flame flow field. Hu Shengchaoet et. [4] carried out the numerical simulation study for the mechanism and radiation characteristics of multi nozzle gas jet noise, and put forward a scheme for reducing noise by using multiple nozzles instead of single nozzle, and verified the feasibility of the scheme. Sun Ping [5] carried out a three-dimensional numerical simulation of 4 nozzle engine, and got the effects of different altitude and nozzle layout on wake flame flow and radiation characteristics. Wang Yanming et. [6] obtained infrared characteristics of the 2~5μm-band spectra in the wake flame flow field of a multi nozzle engine by studying the wake flame...
flow field and infrared radiation characteristics of a multi nozzle engine at low altitude. Equipment College Nie Wansheng, et [7~9] started from the internal combustion mechanism of liquid rocket engine to build a set of internal and external integrated simulation model of engine combustion wake flame, and carried out the simulation calculation of wake flame radiation characteristic accordingly. The above documents studied the wake flame of a multi nozzle rocket engine from different angles, and provided an important computational model and simulation method for the related field work. However, with the increasing demand of carrying capacity, the number of engines in parallel work is increasing, which leads to more complicated structure of wake flame. At the same time, non-toxic non-polluting liquid propellant and gradually favored by the world, for example, the rocket engine with liquid hydrogen liquid oxygen and liquid oxygen kerosene as propellant is not only low cost, non-toxic and environmental protection, but also has the advantages of high density ratio, higher performance reliability, convenient use and maintenance, and is the best power choice for launch vehicle booster and first level. It is also the object of study in the world of various countries [10~12]. In this paper, the flow analysis of a rocket engine with 4 nozzles is carried out, and the distribution of the velocity, temperature and pressure of the wake field is studied.

2. Physical model and numerical methods

2.1. Physical model
Taking a solid rocket engine as the research object, the wake field distribution of its multi-nozzle during flight is studied. The structure diagram of the simulation object is shown in figure 1.

![Figure 1. Multi-nozzle structure diagram](image)

According to the actual flight environment, on condition that the following simulation boundary conditions, firebox pressure is 8.5 MPA, and the temperature of the firebox is 3200K, assuming that the gas is a single-phase ideal gas, and the specific heat at constant pressure is 1750J / kg K, and the gas law constant is 302J / kg K, the flying altitude is 5000m, and the flight Mach number is 2.

2.2. Governing equations
The mass, momentum, energy and component equations can be expressed in the following general form:

$$\frac{\partial (\rho \phi)}{\partial t} + \text{div}(\rho U \phi) = \text{div}(\Gamma \text{grad} \phi) + S_{\phi}$$  \hspace{1cm} (1)

The formula is a general variable, which can represent solving variables such as $u$, $v$, $T$, $\Gamma$ is a generalized diffusion coefficient; $S_{\phi}$ is a generalized source term. For a specific equation, $\phi$, $\Gamma$,
$S_\phi$ has a specific form, table 1 shows the correspondence between the three symbols and the specific equation:

| Symbol | Mass equation | Momentum equation | Energy equation | Composition equation |
|--------|---------------|------------------|----------------|---------------------|
| $\phi$ | 1             | $u_i$            | $T$            | $m_i$              |
| $\Gamma$ | 0          | $\mu$            | $\frac{\lambda}{\epsilon_p}$ | $\Gamma_i$ |
| $S_\phi$ | 0          | $-\frac{\partial p}{\partial x_i} + S_i$ | $S_T$          | 0                   |

*Table 1. Universal variable meaning*

Standard model $k - \varepsilon$ for turbulence equation selection:

$$\frac{\partial (\rho k)}{\partial t} + \text{div}(\rho u k) = \text{div} \left[(\mu + \frac{\mu_t}{\sigma_k})\text{grad} k \right] + G_k + G_b - \rho \varepsilon - Y_m + S_k \quad (2)$$

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \text{div}(\rho u \varepsilon) = \text{div} \left[(\mu + \frac{\mu_t}{\sigma_\varepsilon})\text{grad} \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{\alpha_s} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (3)$$

Where: $G_k$ is turbulence kinetic energy due to laminar flow velocity gradient; $G_b$ is turbulence kinetic energy due to buoyancy; $Y_m$ is the wave generated by transient diffusion in compressible turbulent flow. $\sigma_k$ and $\sigma_\varepsilon$ are turbulent Prandtl number respectively representing $k$ equations and $\varepsilon$ equations; $S_k$ and $S_\varepsilon$ are source items respectively defined for the user.

2.3. Meshing

Use ICEM CFD to mesh the above policy geometry model. The existence of boundary layer is considered when setting the size. Considering the requirements of computer resources and calculation accuracy, the thickness of the first layer grid at the wall surface of the throat is 0.083 mm, and the boundary layer has 20 layers, with an increase ratio of 1.2. The dimensions of the remaining positions are set appropriately on this basis, and finally a grid is generated, as shown in fig. 2.
3. Calculation and analysis
Firstly, the distribution diagram of the nozzle wall $y^+$ during the simulation process is given, as shown in figure 3.

![Figure 3. Nozzle wall $Y^+$ position nephogram](image)

The above figure shows that the wall $y^+$ value is between 6.5 and 91.6 in the simulation process, and the largest $y^+$ value is located at the throat, which is basically consistent with the selected turbulence model and wall function. In addition, it also shows that the number of the first layer grids taken in the grid division process is good, and the simulation results basically reflect the flow law. In addition, it also shows that in order to obtain better flow simulation results, the first layer grid of the throat can be further encrypted and a better turbulence model can be selected.

3.1. Velocity distribution

![Figure 4. Velocity nephogram](image)
Figure 4 shows the final Mach number cloud picture. From the figure, it can be seen that the simulation results conform to the actual flow situation. The air flow enters the far field after expanding and accelerating through the laval nozzle, and forms a relatively obvious Mach disk through continuous refraction and intersection in the far field (shown in figure 5).

Figure 5. Velocity nephogram along flowing section

Figure 6 shows the velocity distribution diagram of the far field exit section. From this, it can be seen that the engine wake is uniformly distributed in the radial direction and the mach number of the axial flow velocity is about 3.6 on the far field exit section.

Figure 6. Velocity nephogram on far-field exit
3.2. Temperature distribution

Figure 7. Temperature nephogram

Figure 8. Temperature nephogram along flowing section

Figure 7 shows the temperature cloud picture obtained by simulation. As can be seen from the figure, the gas temperature gradually decreased from the initial 3200 k to about 2400 k after expansion of the laval nozzle. In the cloud picture, there is no obvious sign of Mach disk, but from figure 8, we can see that there is a small temperature fluctuation at the front end of the far field and there are a certain number of shock waves and expansion waves. The overall temperature is relatively uniform.
Figure 9. Temperature nephogram on far-field exit section

From the temperature distribution diagram of the far field exit section, it can be concluded that the temperature of the engine wake in the far field exit section is uniformly distributed in the radial direction, and the static temperature of the nozzle axis is about 1360 K.

3.3. Pressure distribution

Figure 10. Pressure nephogram

Figure 10 shows the cloud picture of the simulation results of pressure. From the cloud picture, it can be seen that in the convergent section of the nozzle, the pressure is still relatively large and the decrease is not obvious. When the flow passes through the throat, the pressure of the gas begins to drop sharply, with larger amplitude. After this sharp drop, a relatively stable pressure distribution will follow. This result is consistent with the actual working condition.
Since the pressure in the convergent section is far greater than the pressure in the far field, it is difficult to see the pressure change in the far field in the cloud picture. Figure 11 shows the pressure cloud picture in the cross section along the flow direction in the far field. The obvious pressure fluctuation can be seen in this figure, and the supersonic flow in this region indicates that there is a shock wave at the fluctuation position, which causes the pressure change.

It can be seen from the pressure distribution diagram of the far-field exit section that the engine wake is uniformly distributed in the radial direction on the far-field exit section, and the pressure coupling between different nozzle wake is strong.
3.4. Mach disk

![Velocity curve graph of the axis of a certain nozzle](image1)

**Figure 13.** Velocity curve graph of the axis of a certain nozzle

From the velocity distribution map on a certain nozzle axis, it can be concluded that there is velocity fluctuation after the gas exits from the nozzle outlet. These areas prove the existence of Mach disk, but with the increase of axial length, the fluctuation tends to be gentle.

![Temperature curve graph of the axis of a certain nozzle](image2)

**Figure 14.** Temperature curve graph of the axis of a certain nozzle

From the temperature distribution map on a certain nozzle axis, it can be concluded that after the gas flows out from the nozzle outlet (the nozzle outlet cross section is $x = 170$ mm), it continues to expand and do work, causing the static temperature to drop.

Figure 15 shows a comparison diagram of the velocity and temperature curves on a certain nozzle axis. From the diagram, it can be seen that the change trend of the two physical quantities is completely opposite. There is a Mach disk at the location where there is fluctuation, and there is a change in the physical quantities.
4. Conclusion

(1) The $y +$ values obtained from the simulation show that the grid division is better and can better reflect the real flow situation;

(2) The simulation results are authentic, and the distribution and variation characteristics of the main physical quantities are consistent with the actual situation;

(3) Mach inventory lies in the far field, causing the fluctuation of velocity, temperature and pressure along the flow direction, and the fluctuation of velocity and temperature can correspond to each other one by one.

References

[1] Hideyo N, Nobuhiro Y, Makoto A, et al. Numerical Analysis of Plume Heating Environment for H- IIA Launch Vehicle during Powered Ascent [R]. AIAA 2007-5505.
[2] Ze-Juan Xiao, Hui-Er Cheng. Plume Interaction in Parallel Multi-Thrusters Propulsion System and the Effect on Backflow [R]. AIAA 2006-3599.
[3] Houshang B. E. Numerical Investigation of Multi-Plume Rocket Phenomenology [R]. AIAA 1997-3622.
[4] Shengchao Hu, Ang Li, Futing Bao, etc. multi-nozzle fuel gas denoise scheme feasibility data study [J]. JTHJ, 2012, 35(2): 198-202.
[5] Ping Sun, Xinyun Fan, multi-nozzle rocket engine Wakefield flow and radiation characteristic numerical simulation [J]. Mechanical, 2008, 14(7): 41-44.
[6] Yanming Wang, Heping Tan, Shikui Dong, etc. Low-altitude multi-nozzle engine flare research on infrared characteristics [J]. Solid rocket propulsion technology, 2009, 32(6): 634-637.
[7] Wansheng Nie, Songjiang Feng. Liquid motor kinetics of combustion numerical model [M]. BJ: National Defense Industry Publishing House, 2011.
[8] Wansheng Nie, Junhui Yang, Haobo He, etc. Liquid rocket engine tail flame infrared radiation [J]. Journal of Defence Science University of PRC, 2005, 27(5): 91-94.
[9] Songjing Feng, Wansheng Nie, Qingfen Xie, etc. Combustor combustion model affect tail flame flow field and its radiation [J]. Rocket-propelled, 2006, 32(2): 6-10.
[10] Zhaobo Ding, Jiguo Sun, Xiaohong Lu. Foreign representative high thrust hydrogen oxygen engine thrust chamber technical schemes summary [J]. Missile and Space Carrier technology, 2012(4): 27-30, 38.
[11] Yonghua Tan, High thrust liquid rocket engine Research [J]. Acta Astronautica, 2013, 34(10): 1303-1308.
[12] Dong Li, Tangming Cheng. Prospects for the Development of China's New Generation launch vehicle [J]. China Aerospace, 2008(2): 7-10.