Numerical Simulation Coupling on Fluid Flow and Heat Transfer in Nozzle and Equipment Cabin of Solid Rocket Motor during Operation Process

Zheng Ni 1, Yue Wu 1, Yang Liu 1, 2, *, Pengfei Zhu 2

1 Shaanxi Province Aerospace and Astronautics Propulsion Research Institute, Xi’an, 710003, China
2 Science and Technology on Combustion, Internal Flow and Thermo-Structure Laboratory, Northwestern Polytechnical University, Xi’an, Shaanxi, 710072, China

*Corresponding author e-mail: liuyang802@nwpu.edu.cn

Abstract. Driven by the goal of studying the temperature changes in the nozzle and equipment cabin of solid rocket motor during the whole process of ignition starting, working, cooling and decompression, this paper uses the fluid dynamics software which uses the RNG k-ε turbulence model to simulate the mixing of the fuel and oxidant and the Sutherland law to describe the viscosity of gas and air, and the controlling equation is the two-dimensional axisymmetric viscous compressible URANS equation. In this paper, the temperature field of the nozzle and equipment cabin in the working process of solid rocket motor during 0 ~ 60s is numerically simulated, and the ground experiment is carried out. The results show that (1) the numerical simulation results are in good agreement with the ground test results, and the absolute value of the maximum error appears at the time of 60s, which is 5.9k, indicating that the numerical calculation method and heat source equivalent method established in this paper are suitable for the temperature field simulation of composite structure nozzle and equipment cabin, (2) the unsmooth surface of the nozzle formed by high temperature ablation makes the flow situation in the divergent section of the nozzle complex and affects the heat transfer coefficient of the wall contacting with the equipment cabin, (3) due to the insulation design of the equipment cabin, only a small part of the heat is transferred to the equipment cabin. During the whole working process of solid rocket motor, the temperature field of equipment cabin increases sharply at both ends and gently in the middle.

Keywords: Solid rocket motor, Nozzle, Equipment Cabin, Flow-Heat transfer coupled simulation

1. Introduction
The nozzle is widely used in aerospace and other power fields. It is an important part of power plant for energy conversion. For solid rocket motors, the performance of the nozzle directly affects the general performance of the motor. In the thermal structural design of the solid rocket motor nozzle, due to the...
different functions of each component, the difference of gas environment, and the different performance requirements of ablation resistance and erosion resistance, generally, the nozzle has a type of composite structure composed of many kinds of materials [1, 2]. The commonly used nozzle thermal structures are made of composite materials such as carbon/carbon, high silica/phenolic, carbon/phenolic, etc. The physical properties, ablation resistance and erosion resistance of each material are different [3]. For example, the ablation of carbon-based materials has very complex physical and chemical processes, including thermochemical ablation, gas mechanical ablation, and condensed phase particle erosion.

The most influential factor is thermochemical ablation, which is the retreat of the inner wall surface caused by the chemical reaction between the carbon on the inner wall of the nozzle and the oxidizing components of the gas. The regression rate is determined by the diffusion rate and the reaction kinetic rate of the oxidizing components of the gas to the inner wall of the nozzle [4]. Keswani and Kuo [5,6] proposed a relatively complete theoretical model of thermochemical ablation, including conservation equations of gas phase, heat conduction equations of solid phase and gas-solid interface boundary conditions. Researchers have carried out a lot of research on the heat transfer and ablation of nozzles [7-11], and have obtained many empirical formulas and ablation prediction models, but the accuracy and rationality of the models are affected by the difference of materials, working environments and working processes.

In this paper, the nozzle of solid rocket motor with multiple composite structures is taken as the research object, combined with the experimental results of ground tests, establishing an equivalent source term method. The temperature change process of the nozzle and equipment cabin is studied in this paper from the ignition start, stable working and pressure relief, which provides some guidance for the thermal protection design of the solid rocket motor nozzle.

2. Description of the research object and the method of numerical simulation

2.1. Description of the research object
The calculation area is shown in Figure 1, including partial combustor and the flow area of the nozzle, partial combustor insulation layer and shell area, the entire nozzle area assembled with rear cover, the nozzle expansion section area, the flow area and the shell area of the equipment cabin. As the result of nozzle thermal protection is composite of many materials, the density nephogram is used to identify it. Two kinds of fluids (gas and air) and nine kinds of solid materials with different properties are involved in the whole calculation area.

![Figure 1. Diagram of the nozzle assembly and the equipment cabin.](image)

According to the characteristics of the research object, the axisymmetric calculating domain is selected, and the quadrilateral structured grid is used, with a total number of 174000 grids as shown in Figure 2. Considering that this study is a coupling problem of flow and heat transfer, in order to accurately calculate the heat flux of flow field and solid interface, the grid of the boundary layer was refined near the convection field and solid interface [12]. After the ignition and start-up of the motor, heat transfer and ablation occur when the high-temperature gas contacts with the rear cover and nozzle.
The internal temperature of the rear cover and nozzle assembly increases continuously, which causes air convection and temperature rise in the equipment cabin. After the motor stopped working, although the combustor was depressurized, the heat transfer and ablation of the rear cover and nozzle continued, and the air in the equipment cabin continues to heat up.

The entire flow field is a strong transient process involving the process of ignition, steady working and depressurization. However, considering that the time of depressurization is short during the ignition and the shutting down of the motor, the heat transfer of the rear cover and nozzle assembly can be ignored, the steady state is used to solve the flow field. In the calculation process, the second-order implicit coupling algorithm is used, and the calculation residual is less than 0.001. When the flow field is stable, the unsteady calculation of coupled heat transfer with the solid region is started (0-18s), the calculation step is 0.01s, and the second-order implicit coupling method is used.

When the motor stops working at 18s, because the calculation of the depressurization process requires a smaller timing step (< 0.0002s), it is not only a large amount of calculation, but also difficult to converge. Therefore, after the motor stops working, the flow field is manually patched, and the influence of the depressurization process on the heat transfer of the rear cover and nozzle is ignored. The same calculation method is used to obtain the temperature field distribution of the air in the solid domain and the equipment cabin during the 18-60s process.

![Figure 2. Diagram of mesh in flow and solid area.](image)

In addition, the ablation process is also involved in the calculation. During the ablation process, the heat is absorbed when the pyrolysis layer is formed, the heat is generated by chemical reaction during the formation of carbonization layer, and the heat will be taken away in the mass consumption process. Since it is difficult to calculate the actual physical and chemical process of ablation, the heat flow variation law in the insulation material during the process of 0 ~ 18s is obtained through the ablation calculation program according to the ablation results of ground experiments, and it is added into each carbonized ablation material as the source term in the calculation of heat transfer process.

The radiative heat transfer between the gas phase and the wall was considered in the calculation. Due to the large optical thickness in the combustion chamber, the P-1 model is selected to carry out the radiative heat transfer. In addition, due to the large space in the equipment cabin, considering that the air density changes with temperature after being heated and flows under the action of gravity, Boussinesq model is used to consider the temperature field changes caused by the air flow due to heating.

2.2. Numerical simulation method
The governing equation is the two-dimensional axisymmetric viscous compressible URANS equation [13], and the calculational model is RNG $k-\varepsilon$ turbulence model [14,15], compared with the standard $k-\varepsilon$ turbulence model, the RNG $k-\varepsilon$ turbulence model improves the accuracy of high-speed flow calculation by modifying the turbulent viscosity, and considers the influence of vortex on turbulence, which improves the accuracy of vortex calculation. The basic equations of RNG $k-\varepsilon$ turbulence model are as follows:
\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho u_i k) = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{1}
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho u_i \varepsilon) = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_3 \varepsilon G_b) - C_2 \rho \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \tag{2}
\]

\[C_{1\varepsilon} = 1.42, \quad C_{2\varepsilon} = 1.68, \quad G_k \quad \text{is the turbulent kinetic energy produced by the average velocity gradient,}\]
\[G_b \quad \text{is the turbulent kinetic energy produced by buoyancy,}\]
\[Y_M \quad \text{is the contribution of wave expansion to the total dissipation rate in compressible turbulence,}\]
\[\alpha_k \quad \text{and} \quad \alpha_\varepsilon \quad \text{is the reciprocal of} \quad \rho \quad \text{and} \quad \rho \quad \text{efficient Prandtl numbers,}\]
\[R_\varepsilon \quad \text{represents the effect of the average strain rate on} \quad \rho.\]

\[
\eta = \frac{S}{\varepsilon} \quad \text{is the ratio of mean flow time scale to turbulence time scale,} \quad S = \sqrt{2S_{ij}S_{ij}} \quad \text{is the norm of the strain rate tensor,} \quad \eta_0 = 4.38 \quad \text{is the typical value of} \quad \eta \quad \text{in uniform shear flow,} \quad C_\mu = 0.0845, \quad \beta = 0.012. \tag{3}
\]

The wall is processed by standard wall function, the discretization of convection term adopts the second-order upwind style, and the discretization of diffusion term adopts the central difference format. The specific heat, thermal conductivity and density of fluid and solid materials are all fitted with polynomials, and the viscosity of gas and air is described by Sutherland law.

As the ablation process of adiabatic materials is involved in solving the problem, it not only involves the process of heat conduction of the material, heat absorption of the material pyrolysis, chemical reaction in the carbonization layer, but also involves the coupling of flow, heat transfer and ablation between materials with different physical properties in a variety of regions. In this paper, the simplified equivalent method is used to deal with the inner boundary. The specific equivalent simplified method is as follows:

(1) According to the calculation results of the motor flow field under the ground test conditions, considering that the insulation surface and nozzle assembly surface will be ablated in the actual working process, and the ablated insulation surface has a great influence on the flow field, at the same time, the change of the flow field will also affect the heat transfer of the wall, since it is difficult to consider the surface movement in the calculation, the ablated surface is selected as the calculation area.

(2) The boundary of flow field and solid region is determined according to the morphology after ablation, and the flow boundary of the heat of the entire flow and solid region is extracted.

(3) According to the flux boundary condition of heat as the initial condition of ablation calculation, a mature one-dimensional thermochemical ablation program was used to calculate the ablation process of EPDM, and iterative adjustment was carried out according to the ablation rate of insulation material (0.316mm/s) under the ground test conditions to obtain the variation law of heat flux in the process of 0 ~ 18s.

(4) Without considering the boundary shift effect of the fluid and solid interface, only the unsteady coupled heat transfer process between the fluid and the solid area is considered. In the calculation process, the change law of the internal heat flux of the material during the 0-18s process obtained in the step is added as a source term to the solid area to obtain the internal temperature field distribution of the entire solid area and the air temperature distribution in the equipment cabin.

Figure 3 shows the change law of the internal heat flux density of the insulating material over time under the conditions of air and ground (different ablation rates) calculated through the above steps (2) and (4). It is obvious that the heat flux under different ablation rates varies greatly, but the change trend is the same, and tends to be stable with the increase of time.
3. Numerical simulation results and analysis under the conditions of the ground test

3.1. Numerical simulation results of the flow field under the conditions of the ground test

Figure 4 shows the Mach number distribution of the transonic flow field in the combustor and nozzle. The Mach number distribution of the flow field indicates that the uneven surface after ablation complicates the flow in the expansion section of the nozzle. At the same time, multiple reflected shock waves are formed, and the gradient of heat transfer coefficient of the wall surface of the shock wave attachment point changes greatly.

Figures 5-7 show the temperature field distribution at 3s, 12s, and 18s respectively.

Figure 3. Heat flux of thermal insulation material with time.

Figure 4. Mach number distribution contour.

Figure 5. Temperature distribution contour at t=3s.
Figures 8-9 show the temperature field distribution under the conditions of 30s and 60s, respectively. When the motor stops working after 18s, ignoring the depressurization process of the flow field in the calculation, it is considered that the flow field keeps the same state as the outside after the motor stops. It can be concluded from the calculation that after 18s the heat transfers to the equipment cabin and nozzle both ways.
3.2. Distribution of temperature field in equipment cabin under the conditions of the ground test

Figures 10 and 11 show the temperature field distribution in the equipment cabin under the conditions of the ground test at 18s and 60s, respectively. The maximum temperature at 18s is about 775K, which appears at the exit of the nozzle expansion section. The entire internal temperature field of equipment cabin increases sharply at both ends and gently in the middle. The maximum temperature appears at the throat at 60s, and the maximum temperature is about 940K, while the maximum temperature at the exit of nozzle divergent section is 721.2K, which is slightly lower than that at 18s.

![Figure 10. Temperature distribution contour of equipment cabin at t=18s.](image1)

![Figure 11. Temperature distribution contour of equipment cabin at t=60s.](image2)

3.3. Temperature of nozzle measuring points under the conditions of the ground test

The positions of temperature measuring points in nozzle expansion section and equipment cabin during ground test are shown in Figure 12.

![Figure 12. Temperature measuring position in nozzle and equipment cabin.](image3)
Figure 13. Comparison of measured temperature and numerical results.

Figure 13 shows the comparison results between the experimental and numerical values of the temperature measuring point 1 on the shell of the diffusion section. From the calculation results, it is obvious that the temperature variation law with time is basically consistent, and the absolute value of the maximum error is 5.9K, which appears at the time of 60s. From the distribution of temperature field in the equipment cabin under the conditions of the ground test, the internal temperature distribution is not uniform, and the air temperature is higher near the throat and the tail of nozzle expansion section.

The figure shows that the temperature rising trend is mainly divided into three stages. The first stage is about 6-18s. When the motor starts to work, due to the existence of the insulation layer, the heat will not be transferred to the measuring point, so the temperature does not rise in the first 6s. The temperature rises rapidly in 6-18s because of the high heat flux of the nozzle. The second stage is about 20-30s, when the motor stops working, the heat flow decreases, so the temperature rise is relatively stable. The third stage is about 30-60s, the temperature of the measuring point increases sharply again. Considering that there are two reasons, the first is that the entire temperature of the equipment cabin rises, and the equipment cabin also transfers heat to the measuring point. In addition, there is a certain ablation rate of the insulation layer. Due to the partial failure of the insulation layer, the temperature of the measuring point rises sharply.

Considering that there are many kinds of equipment in the equipment cabin, the heat capacity of the cabin will make the temperature in the equipment cabin drop to a certain extent.

4. Conclusion

(1) After the nozzle is ablated, an uneven surface is formed. This complicates the flow in the expansion section of the nozzle, forms multiple reflected shock waves, and causes greater fluctuations in the gradient of heat transfer coefficient of the wall surface of the shock wave attachment point changes greatly.

(2) The working phase of the motor is from 0 to 18s. During this process, the nozzle throat heats up more obviously due to the high heat flow density. In addition, due to particle deposition and eddy currents in the submerged section of the nozzle, the heat transfer in this part will increase, and the temperature rise at the bottom of the submerged section is also relatively severe. In addition, the weaker adiabatic design at the tail of the nozzle expansion section caused a significant increase in temperature in the tail equipment cabin.

(3) Motor work ends at 18s, after which the temperature of the motor gradually decreases, and the heat is transferred to the nozzle and equipment cabin in both directions. Due to the adiabatic design, only a small part of the heat is transferred to the equipment cabin, and most of the heat is transferred to the motor combustor, which plays a protective role for the equipment in the equipment cabin and avoids the failure caused by overheating of the equipment.
Acknowledgments
This work was financially supported by the National Natural Science Foundation of China under grant numbers 51876177.

References
[1] Ren Q B, He J X, Liu W K, et al. Technical basis for solid rocket engine design[M]. Xi’an: Northwesterly Polytechnical University Press, 2001: 83-120.
[2] Li J L, Zhao F, Zhang Y, et al. Application of Carbon Fiber and its Composites in Military Industry[J]. Synthetic Fiber in China, 2014, 43(3):33-35.
[3] Zhu Y, Meng X L, Cui H. Study on thermal structure material of solid liquid engine nozzle with long time resistance to strong oxidation and denudation[J]. Journal of Propulsion Technology, 2020, v.41;No.273(03):225-231.
[4] Thakre P,Yang V. Chemical Erosion of Carbon-Carbon/Graphite Nozzles in Solid-Propellant Rocket Motors[J]. Journal of Propulsion & Power, 2008, 24(4):822-833.
[5] Kuo K K,Keswani S T.A Comprehensive Theoretical Model for Carbon-Carbon Composite Nozzle Recession[J]. Combustion Science and Technology, 1985, 42( 3-4): 145-164.
[6] Keswani S T, Kuo K K. Validation of an Aerothermochemical Model for Graphite Nozzle Recession and Heat-Transfer Process[J]. Combustion Science and Technology, 1986, 47(3-4): 177-192.
[7] Chen J Q, Wang B, Tian X, et al. Stability Analysis of Solid Rocket Motor Combustion with Double Base Propellant by Computational Fluid Dynamics. Journal of Ordnance Equipment Engineering, 2017, 12(v.38;No.233):201-206.
[8] Xiong Y L, Yang D, Gao Y, et al. Numerical study of acoustic oscillations in solid rocket motors[J]. Journal of Propulsion Technology, 2006, 27(6):484-487.
[9] Sun N. Numerical Calculation of 3-D Internal Flow Field of Slotted Single-Chamber Dual-Thrust Solid Rocket Motor[D]. Harbin Engineering University, 2009:40-61.
[10] Tan Y. Combustion Stability Analysis for Solid Rocket Motor Under Overload[J]. Journal of Projectiles, Rockets, Missiles and Guidance, 2015, v.35;No.3:113-116.
[11] WU Y, HU B, et al. Analysis of Combustion Instability in Low-temperature Ground Test of A Solid Rocket Motor[J]. Journal of Projectiles, Rockets, Missiles and Guidance, 2016,36(4):67-69.
[12] Cui L Q, Wei Y. Numerical Simulation of the Wave Structures and Characteristics About Solid Rocket Motor Plume[J]. Journal of Ordnance Equipment Engineering, 2018, 39(12):46-50.
[13] Yu Y Q, Gu H Y, Yang Y H, et al. RANS and URANS Simulations of Turbulent Flow Inside Different Subchannels in Tight Lattice Bundle[J]. Atomic Energy Science & Technology, 2011, 45(10):1191-1197.
[14] Wang Y, Shi F. Numerical Simulation of Turbulent Separated Flow in 180° Bend with RNG k-ε mode. Chinese Journal of Theoretical and Applied Mechanics[J], 1996, 28(3):257-263.
[15] Lee J, Chen G Q. Numerical simulation of line puff via RNG k - ε model[J]. Communications in Nonlinearence & Numerical Simulation, 1996, 1(004):6-11.