Computer Simulation Research on Flow Field of SRM High Altitude Simulation Test

Dongyi Jia¹ ², Zhuo Li²,*

¹College of science, Inner Mongolia University of Technology, Hohhot, 010051, China
²Department of Mechanical and Electronic Engineering, Hetao College, Bayannur, 015000, China

*Corresponding author’s e-mail: lizhuoimcu@163.com

Abstract. The simulation test of passive ejection type high-altitude solid rocket motor (SRM) was simulated by computer. For SRM passive ejector type high-altitude test, the tempering in the process of engine ignition and flameout poses a serious threat to the nozzle of the engine test device and circuit arrangement in the upper compartment. Using the N-S equation and Spalart-Allmaras model: numerical simulation on the flow structure of engine pressurize, decompression and stratochamber air phase in the testing process, and flow field of the diffuser and the cabin altitude simulation, analyzing the flow field structure of high-altitude experiment stages. The results showed the rough coincidence between the simulation curve and the measured data, with slight difference at the initial stage of air supply, the measured data and the simulation data reach the overall high degree of coincidence. During the process of building a compression and decompression, the adiabatic measures should be taken to prevent high temperature gas from damaging the engine, especially the nozzle and the head ring, in addition, the reasonable design of the nozzle contour surface can control gas separation tempering, reducing the influence caused by tempering. Stability of the flow field in the high altitude chamber of the pressure will not affect the engine.

Keywords-computer simulation; solid rocket motor; high altitude simulation test

1. Introduction

Computational fluid dynamics, a branch of fluid mechanics, is the product of the combination of mathematics, computer and fluid mechanics, and has great applications in scientific research and engineering technology. High-altitude simulation test is a ground test equipment that can simulate the speed and altitude of a rocket engine in the air. It is important for assessing the performance and reliability of the high-altitude engine structure and accurately measuring the internal ballistic performance of the engine. Ground test. The high modulus test methods include active ejection high modulus test and passive ejection high modulus test. The active ejection high modulus test has high simulation height and good measurement accuracy, but the test equipment is extremely expensive.
[1-5]. Therefore, the passive ejection high modulus test is still the main method in China [4]. Passive ejection high modulus test has backfire phenomenon in the initial stage of engine ignition and flameout, which brings greater danger to the engine nozzle. At the same time, the high-temperature gas flowing back during the tempering poses a serious threat to the test equipment and lines arranged in the high-altitude cabin. At present, the research on the passive ejector high-mode test flow field by domestic research institutes is mainly focused on the steady-state flow field of the diffuser, and there is little research on the dynamic flow field structure of the high-altitude cabin and the diffuser [6-11].

In this paper, a transient numerical simulation of the high-modulus test flow field of a certain type of solid rocket motor is carried out. The flow field in the diffuser and the high-altitude cabin is analyzed and compared with the test data to help researchers better understand the test process. The flow field structure in the medium provides a reference for the design and research of solid rocket motors, especially nozzles.

2. **Principle and Model**

2.1 **Principle**

The solid rocket motor has the advantages of simple structure and high propellant density. Disadvantages: the specific impulse is small, about 250~300 seconds, the acceleration is large, and the working time is short, which causes the thrust to be difficult to control, and the repeated starting is difficult, which is not conducive to manned flight. Mainly used as engines for rockets, missiles and sounding rockets, as well as boost engines for spacecraft launches and aircraft takeoffs. Its working principle [10] is: the solid propellant is ignited and burned in the combustion chamber to produce high-temperature and high-pressure gas, which converts chemical energy into heat energy; the gas expands and accelerates through the nozzle, and the heat energy is converted into kinetic energy at a very high speed. It is discharged from the nozzle to generate thrust to propel the missile forward.

2.2 **High-modulus experimental geometric model**

The high-altitude simulation geometric model of passive ejection of a solid rocket motor is shown in Figure 1. The engine, diffuser, and high-altitude cabin are all rotating axisymmetric structures, so the model is simplified to two-dimensional axisymmetric structures.

![Figure 1. Computational domain geometry](image)

2.3 **Numerical model**

2.3.1 **Governing equation**

The calculation domain is two-dimensional axisymmetric, the governing equation also adopts a two-dimensional axisymmetric form. Navier-Stokes equation [7,11] (N-S) is also simplified to a two-dimensional axisymmetric.

If

\[ U = r(\rho, \rho u, \rho E)^T \]
\( F = r(\rho u, \rho u^2 - \tau_{xx}, \rho u v - \tau_{xy}, \rho Eu - \tau_{xz} u - \tau_{xy} v + q_x) \)

\( G = r(\rho v, \rho u v - \tau_{xy}, \rho v^2 - \tau_{yy}, \rho Ev - \tau_{yz} u - \tau_{xy} v + q_y) \)

\( H = (0,0,\tau_{xx},0)^T, E = (y-1)\frac{P}{\rho} + \frac{1}{2}(u^2 + v^2) \)

\[ \tau_{xx} = -p - \frac{2}{3} \mu (\frac{\partial u}{\partial r} + \frac{v}{r}) + \frac{4}{3} \mu \frac{\partial u}{\partial x} \]

\[ \tau_{xy} = -p - \frac{2}{3} \mu (\frac{\partial u}{\partial x} + \frac{v}{r}) + \frac{4}{3} \mu \frac{\partial v}{\partial r} \]

\[ \tau_{yy} = -p - \frac{2}{3} \mu (\frac{\partial v}{\partial x} + \frac{v}{r}) + \frac{4}{3} \mu \frac{\partial v}{\partial r} \]

\[ \tau_{yz} = \tau_{xy} = \mu (\frac{\partial u}{\partial r} - \frac{\partial v}{\partial x}) \]

\[ q_x = -k_{eff} (\frac{\partial T}{\partial x}), q_y = -k_{eff} (\frac{\partial T}{\partial r}) \]

\[ k_{eff} = k_e + k_T, \quad \mu = \mu_e + \mu_T \]

Then:

\[ \frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + H = 0 \quad (1) \]

Where: t-time; axial coordinate; radial coordinate; density; pressure; T-temperature; internal energy; axial velocity; radial velocity; viscosity coefficient; specific heat ratio; thermal conductivity.

### 2.3.2 Turbulence model

In the momentum equation [4], it must be obtained by the turbulence model, involving the turbulence viscosity coefficient \( \mu_i \). The widely used Spalart-Allmaras [8,12] model is used to solve only one transport equation related to eddy viscosity, which is relatively simple and relatively small amount of calculation. This model can give good simulation results for the boundary layer that is subjected to the back pressure gradient. The turbulent viscosity coefficient is calculated as follows:

\[ \mu_i = \rho \nu f_{\nu} \]

\[ \rho \frac{\partial \nu}{\partial t} = G_i + \frac{1}{\sigma_i} \left( \frac{\partial}{\partial x_j} (\mu + \rho \nu) \frac{\partial \nu}{\partial x_j} \right) + C_{\nu} \rho \left( \frac{\partial \nu}{\partial x_j} \right)^2 - Y_i \quad (3) \]

### 3. Result Analysis

#### 3.1 Distribution of measuring points

During the test, there are multiple measuring points on the high-altitude cabin and the engine, as shown in Figure 3. The temperature, pressure and other parameters during the test are tested separately. In order to compare with the test data, the data during the entire calculation process is monitored at the same location. Among them, measuring points 1, measuring points 2, and measuring points 3 are distributed in the high-altitude cabin to monitor the temperature and pressure in the cabin, and measuring points 4 and 5 are arranged on the outer wall of the nozzle to monitor the temperature. The distribution of measuring points is shown in Figure 2. The lower point is the symmetrical position of the measuring points.
3.2 Flow field analysis during ignition

0.4555 ms and 34.55 milliseconds after the nozzle opening, the Mach number distribution shown in Figure 3 and Figure 4.

Through analysis, it is found that about 0.6555 milliseconds after the nozzle plug is opened, flashback begins. At this time, the nozzle is not full, and the gas flows to the nozzle outlet and directly flows back into the high-altitude cabin. At 34.555 milliseconds, a jet flow field is established at the entrance of the diffuser, and the gas return to the high-altitude cabin ends. After about 20 milliseconds, the swirl structure caused by the gas reflux disappears. The entire tempering process lasts about 50 milliseconds.

During the tempering process, the high-temperature gas flows into the high-altitude cabin along the outer surface of the nozzle. The speed and temperature are high, which has a great impact on the nozzle and the surrounding test equipment.

Figure 5 shows the temperature fluctuation curve of observation point 4 and observation point 5. The highest temperature of measurement point 4 is 702K, which occurs at 38.56 milliseconds, and the highest temperature of measurement point 5 is 2734K, which occurs at 24.55 milliseconds.
3.3 Analysis of stable section

After the cover is opened, a stable flow field is quickly established in the entire calculation domain. Figure 6 shows the flow field streamline diagram of the stable section.

The pressure change in the cabin during ignition and stabilization stage is shown in Figure 7. It can be seen from the figure that during the ignition and engine stable working phases, the pressure at the three measuring points in the cabin does not change much, with only a small shock. The high-altitude cabin maintains a stable low air pressure, ensuring that the test can effectively assess the high-altitude performance of the engine.
4. Conclusion
In this paper, a numerical simulation of the flow field of the solid rocket motor during the high-altitude simulation test run is carried out, and the simulation results are compared with the test results. Based on this, the following conclusions are obtained:

1) Both the engine pressure build-up process and the decompression process will cause backfire, which will bring high-temperature gas into the cabin and affect the engine and high-altitude cabin. Thermal insulation measures should be taken to prevent the high-temperature gas from affecting the engine, especially the nozzle and rear head. Measuring equipment, etc. cause damage. At the same time, reasonable design of the nozzle profile and control of the separation point of the tempering gas is beneficial to reduce the impact of tempering.

2) The structure of the flow field in the high-altitude cabin of the stabilization section is stable. With the existence of a vortex structure of varying strength, the temperature and pressure fluctuations are small, which will not affect the engine.

References
[1] Bauer R C, and German R The effect of second throat geometry on the performance of ejectors without induced flow AEDC-TN 61 p 133
[2] Jones W L, Frice H G and Lorenzo C F Experimental study of zero-flow ejectors using gasous nitrogen NASA TN D- p 203
[3] Jiang Xiaorui, Li Zhuo, Cui Likun 2013 Numerical Analysis of the Diffuser in the High Altitude Simulation Test Platform Inner Mongolia Science Technology and Economy
[4] Cui Likun, Jiang Xiaorui, and Li Zhuo 2013 Numerical simulation of two-phase flow in diffuser of high altitude simulation test bed Solid Rocket Technology 04 pp 559-563+568.
[5] R. Spalart, and S. Allmaras A one-equation turbulence model for aerodynamic flows AIAA-92-0439
[6] Xu Wanwu, Tan Jianguo, and Wang Zhenguo 2003 Numerical study of supersonic ejector on high altitude simulation test bed solid rocket technology 26 (2) pp 71-74.
[7] Ding Xuejin, Wang Zhihao, and Liu Xiaoli 2008 Numerical analysis of diffuser in high altitude simulation test bed Journal of Chongqing University of Science and Technology (Natural Science Edition) 10 (1) pp 61-63.
[8] Xu Wanwu, Tan Jianguo, and Wang Zhenguo 2003 Numerical study of supersonic ejector on high altitude simulation test bed. Solid rocket technology 26 (2) pp 71-74.
[9] Zhu Ziyong, Li Peichang, and Qu Qian 2010 Numerical calculation and experimental comparison of diffuser in a certain type of rocket engine high-altitude simulation test Spacecraft Environmental Engineering 27 (2) pp 231-237.
[10] Wu Xiaosong, Chen Jun, et al. 2006 Numerical simulation of solid rocket motor working process Beijing: Higher Education Press.
[11] Jian Yu, and Chao Yan 2010 Discontinuous Galerkin finite element method for Navier-Stokes equations Chinese Journal of Theoretical and Applied Mechani 05 pp 962-970.
[12] Yang Xiaouquan, Yang Aiming, and Sun Gang 2013 An efficient numerical calculation method for the RANS equation of a strongly coupled Spalart-Allmaras turbulence model Acta Aeronautica Sinica 09 pp 2007-2018.