Numerical study on the effect of rocket plume to the jet flow deflector in liquid rocket engine test stage

Jiawei Ding*, Yongwei Liu, Hong He
Xi'an Aerospace Propulsion Test Technique Institute, Xi'an 710100, China

*Corresponding author e-mail: dingjw163@163.com

Abstract. Jet flow deflector is one of the most important facility in liquid rocket engine test stage. It is necessary to investigate the effect of rocket plume to the jet flow deflector. Based on the theory of computational fluid dynamics, the flow field of a rocket engine plume is simulated and analysed by numerical method. The velocity field and temperature field of the plume in rocket engine test are obtained. The structural characteristics of engine plume flow field and the impact effect on flow deflector are analysed. Finally, results are applied to the thermal protection design of the test stage.

1. Introduction
During the process of liquid rocket engine test, high-temperature and high-speed plume is created by the rocket nozzle. Jet flow deflector is one of the most important facility in liquid rocket engine test stage. As shown in Figure 1, its function is to quickly and smoothly divert the plume from the liquid rocket engine test stage, to prevent the frontal reflection of the shock wave, the return of the plume and the impact of the plume on the ground, which could result in splashes that endanger the safety of the rocket engine, test stage and ground facilities [1]. Therefore, it is necessary to investigate the effect of rocket plume to the jet flow deflector.

Figure 1. Diagrammatic sketch of jet flow deflector in liquid rocket engine test stage.

In the early research, the impact characteristics of engine plume are preliminarily studied by experimental means [2], and then the numerical simulation technology is used to simulate the flow field of exhaust plume impact, so as to improve the calculation accuracy [3]. Accurate flow field such as pressure and temperature are obtained so as to better utilize and control the impact characteristics of
rocket plume. In recent years, the flow field simulation technology has been applied to practical engineering problems, to predict and guide the engineering practice [4-6]. Allgood et al. [7] investigated the plume impact characteristics of ARES V propulsion system by numerical method, and the impact shape, surface pressure, temperature and other parameters of B-2 flow deflector were obtained, which can help the practical construction of flow deflector and improve the safety and reliability of space launch. Giordan et al. [8] simulated the cooling process of water injection into a rocket engine plume. The pressure distribution of the flow field was obtained and the analysis shows that the pressure is the main cause of the deflector damage. Li et al. [9] conducted the numerical simulation of a large launch vehicle stage test with CFD technique. The flow field of the rocket engine plume was obtained and the safety of the test stage was analysed.

With the increasing demand for space missions, the thrust of the rocket engine is increasing and the structure of engine plume flow field becomes more complex. Based on the theory of computational fluid dynamics, this paper simulates and validates the model of a large thrust rocket engine plume by numerical simulation method, and applies it to the thermal protection design of the test stage.

2. Numerical methods

2.1. Simulation Set-up
In the present paper, the engine test stage is vertical which means that the plume is sprayed downward. The flow field in engine nozzle and exhaust plume are simulated numerically. As shown in Figure 2, The calculating area is simplified to a two-dimensional axisymmetric area. The center axis of the engine combustor is taken as the symmetric axis. The radius of the plume area under the nozzle is 5 m and the axial length is 20 m. Structured grids are used in the computational area. As shown in Figure 3, Local refinement is used in the nozzle area of the engine. At the same time, adaptive grids are used to optimize the computations.

![Figure 2. Simulation zone of two-dimensional axisymmetric area.](image1)

![Figure 3. Local computational mesh near the nozzle.](image2)

2.2. Numerical Methodology
Turbulent N-S equations in cylindrical coordinate system are used to describe the flow field the combustion chamber and the plume flow field. A standard k-ε model is used for the turbulence model. For the time discretion of governing equations, Crank–Nicholson method with second-order accuracy is applied. A linear form of the central differencing scheme is used for general field interpolations. The discretization of convective fluxes is carried out using Gauss linear scheme. The pressure implicit split operator (PISO) algorithm is used for the pressure velocity coupling.
2.3. Boundary Condition
The nozzle inlet is set as pressure-inlet boundary, and parameters such as inlet pressure, gas composition and temperature are given, as shown in Table 1. Pressure-outlet boundary is adopted at the outlet of the jet flow field, and the outlet pressure is 1 atmospheric pressure. The far-field condition near the exit of the engine nozzle is set to ambient pressure and temperature. Adiabatic wall condition without slip is adopted for nozzle wall condition.

| Pressure | Temperature | H2O   | N2   | CO2  | H2   | CO   |
|----------|-------------|-------|------|------|------|------|
| 7.5MPa   | 2700K       | 0.34  | 0.30 | 0.14 | 0.12 | 0.10 |

Table 1. Boundary conditions of inlet.

3. Results and Discussion

3.1. Analysis of Plume Velocity Field
For liquid rocket engines, there is a core flame area in the exhaust plume. In the core region, with the increase of the axial distance, the temperature and pressure drop of the exhaust plume is not obvious, but the velocity, temperature and pressure drop of the exhaust plume decreases rapidly beyond the core region of the flame [10].

The distribution of the plume velocity field is shown in Figure 4, and the Mach number distribution curve along the axis of the jet is shown in Figure 5. The calculated Mach number of the nozzle outlet is 3.5, which is close to the theoretical calculation of Ma=3.45, thus verifying the applicability of the simulation model.

![Velocity](image1)
![Mach Number](image2)

(a) Velocity
(b) Mach Number

Figure 4. The velocity field of rocket engine plume.
Figure 5. The Mach number distribution curve along the axis of the plume.

From Figure 4, it can be seen that the engine plume expands after ejected outward the nozzle. Because of the atmospheric compression, Barrel shock wave is generated and Mach disk appears at the tail of the shock wave. When the axial distance reaches 13 m, Mach number decreases obviously, and the sudden change of dynamic pressure disappears basically. This indicates that the region is separated from the core region of exhaust flame combustion, and the length of the flame core region for the engine is about 13m. In the design of the flow deflector, considering the test of this type of rocket engine, the impact height of the exhaust plume should be greater than 13m, so as to avoid the direct impact of the flame core on the surface of the flow deflector.

3.2. Analysis of Plume Temperature Field

As shown in Figure 6, Mach disk appears at the tail of Barrel shock wave, followed by a small high temperature zone. The high temperature region in the plume mainly concentrates on the edge of the plume and a small section behind the Mach disk. The impact surface of the flow deflector of a rocket engine test bed is 20m away from the nozzle exit. According to the simulation results, the average gas temperature on the impact surface is about 1500K, and the gas temperature will cause the steel plate to be damaged.

Figure 6. The temperature field of rocket engine plume.

Therefore, in order to solve the problem of high temperature gas scouring the surface of flow deflector for a long time during the test process, the flow deflector is cooled by water film. A large number of water spray holes are set on the surface of the flow deflector, and a large amount of water flows through the surface of the flow deflector during the test process. In this way, the working
temperature of the flow deflector is reduced, the ablation resistance of the flow deflector is increased, the service life is prolonged and the noise is reduced.

3.3. Analysis of gas Impact characteristics
The physical process of plume jet impinging on the flow deflector is simplified and mechanically analysed. As shown in Figure 7, the high temperature gas is impacted vertically downward with velocity $v_1$ and flow rate $Q_1$. The impact area is $S$, the Impact angle $\alpha_1$ of impact point is $30^\circ$. Cooling water is ejected horizontally with injection pressure $P_1$. After the impact, it is assumed that the high temperature gas flows completely along the surface of the flow deflector. The departure angle $\alpha_2$ is $40^\circ$.

![Figure 7. The simplified physical process of plume jet impinging on the flow deflector.](image)

According to the temperature distribution, the flame radius at the impact point is about 0.8m and the impact area is 1.54$m^2$. The vertical downward force of high temperature gas on the flow deflector in unit time (1s) is as follows:

$$F = Q_1 v_1 (\sin \alpha_1 - \sin (\alpha_1 - \alpha_2))$$  \(1\)

Gas velocity is 2300 m/s based on simulation results. Thus, the impact force is $5.9 \times 10^5$ kg$m/s$. Finally, the impact pressure $P = F/S$ is about 0.38MPa.

Therefore, in order to ensure the smooth ejection of cooling water from the flow deflector nozzle, it is necessary to ensure that the spray pressure is greater than the impact pressure of 0.38MPa, thus forming an effective liquid film for cooling.

4. Conclusion
In this paper, the flow field of a rocket engine plume is simulated and analyzed by numerical method. The velocity field and temperature field of the plume in rocket engine test are obtained. The structural characteristics of engine plume flow field and the impact effect on flow deflector are analyzed. The results are applied to the thermal protection design of the test stage. The following conclusions are drawn:

1. At the exit of the engine nozzle, Barrel shock wave is generated and Mach disk appears at the tail of the shock wave. There is a 13m long core flame area in the exhaust plume. In the design of the flow deflector, the impact height of the exhaust plume should be greater than 13m, so as to avoid the direct impact of the flame core on the surface of the flow deflector.

2. The average gas temperature on the impact surface is about 1500K, and the gas temperature will cause the steel plate to be damaged. The flow deflector should be cooled by water film.

3. When the plume jet impinging on the flow deflector, the impact pressure is about 0.38MPa. To ensure the forming of an effective liquid film for cooling, the spray pressure should be greater than the impact pressure of 0.38MPa.
References

[1] GREENWOOD T, Prozan R, Ratliff A, et al. Analysis of liquid rocket propellant engine exhaust plumes [C]// 5th Thermophysics Conference. 1970: 844.

[2] Dosanjh D S, Sheeran W J. Observations on jet flows from a two-dimensional, underexpanded, sonic nozzle [J]. AIAA Journal, 1968, 6 (3): 540-542.

[3] Cain T, MORRIS N, ROBERTS T, et al. A comparison of numerical predictions and experiments on under-expanded jets [C]// 17th Aerospace Ground Testing Conference. 1992: 4027.

[4] Rodionov A. New space-marching technique for exhaust plume simulation[C]//36th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit. 2000: 3390.

[5] Zhou S, Guo Z, Gao S, et al. Numerical simulation for unsteady flow field induced by rocket engine jet and shield board [J]. Journal of Propulsion Technology, 2007, 2.

[6] Lee K S, Hong S K, Park S O. Supersonic jet impingement Navier-Stokes computations for vertical launching system design applications [J]. Journal of spacecraft and rockets, 2004, 41(5): 735-744.

[7] Allgood D, Ahuja V. Computational plume modeling of conceptual ARES vehicle stage tests[C]//43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit. 2007: 5708.

[8] Giordan P, Fleury P, Guidon L. Simulation of water injection into a rocket motor plume[C]//35th Joint Propulsion Conference and Exhibit. 1999: 2517.

[9] Li M, Chen S Z, Chen C F. Simulation analysis on wake flow temperature field of rocket engine in horizontal ground test[J]. Journal of Rocket Propulsion, 2012, 38(6): 29-34.

[10] Rodionov A. New space-marching technique for exhaust plume simulation [C]//36th AIAA /ASME/ SAE/ ASEE Joint Propulsion Conference and Exhibit. 2000: 3390.