Performance Evaluation of Second-Throat Diffuser for High Altitude Test facility

Hengxiao Qian*, Wei Zhong, Zhao Li, Pan Li and Shilong He
Beijing Institute of Control Engineering, Beijing, China

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

Abstract. Aimed at the improvement of ability of the high altitude test (HAT) facility for rocket engine, a new designed secondary-throat diffuser with ejection of low total pressure and supersonic cold air is proposed. The main structural characteristic of the new-designed diffuser is that: a groove is added to the start of the secondary throat of diffuser, and the supersonic cold air with low total pressure is ejected into diffuser to form the gas film, which leads to a reduction of the wall temperature and thermal load of the new-designed diffuser, and an improvement of the ability of diffuser holding shock train. The effects of cold air on flow characteristics and performance of diffuser are numerically investigated in detail under different conditions including total pressure and Mach number. The results show that the wall temperature along the diffuser increases with an increase of Mach number, and decreases with an increase of total pressure; with an increase of Mach number, the starting position of pressure rise moves upstream to the inlet of diffuser; and with an increase of total pressure, the starting position of pressure rise moves downstream to the end of diffuser.

1. Introduction
To test and qualify the rocket engine before its flight, a ground testing of the engine is necessary, and the low pressure environment corresponding to the high altitude flight situation should be created in the ground testing facility. A typical high altitude test (HAT) facility is usually composed of a vacuum chamber, a secondary-throat diffuser and an ejector system. The secondary-throat diffuser is often employed in HAT facility to maintain the flow field at the exit of the rocket engine in the vacuum chamber. In the diffuser, the supersonic gas generated by rocket engine is decelerated, and the pressure of the gas increases.

Since of directly exposing to the high temperature gas generated by rocket engine, it is very difficult to design the thermal protection of the diffuser. According to the research of Manikanda Kumaran R et al. [1], the wall temperature of the diffuser is above 3000 K. To resolve these issues, theoretical researches and experimental measurements by Wang Yongzhong [2] have been performed to examine the internal spray of diffuser. It is found that an increase of the flow rate of the spray water and getting the spray position to the wall of diffuser with hot gas can effectively reduce the wall temperature. Based on the numerical simulations of the flow field of diffuser under spray water, Liu Bo et al. [3] token the mass flow rate of spray water, ejection speed and the axial positon of nozzle into consideration. The results of numerical simulation by Jiang Yitong et al. [4] show the relationship between mass flow rate of spray water, the wall temperature and the back pressure at the exhaust of diffuser. A further study by
Vubt et al. [5] obtained the relationship between the angle of water spray and the temperature of plume. Besides, Tian Ning et al. [6] found the distribution of wall temperature of diffuser through numerical simulation, and applied the theory of boiling heat transfer to the protection of diffuser.

As mentioned above, there are two main methods for the cooling of diffuser, one is internal cooling, such as spray water of gas at the start of diffuser; the others is external cooling, such as the application of water jacket around diffuser. However, the method of internal cooling often increases the burden of ejector downstream the diffuser; the method of external cooling usually needs the improvement of pressure and mass flow rate of spray water, which brings great difficulty of the design of water supply system and secondary-throat diffuser.

This paper describes our recent efforts on reducing the thermal load and the ability of holding shock train for secondary-throat diffuser by ejection of supersonic cold air with low total pressure. And numerical simulations were carried out to study effects of cold air on flow field structures and performance of the new-designed secondary-throat diffuser.

2. Modeling of Diffuser and Computational Methodology

2.1. Modeling of the new-designed diffuser

Figure 1 presents the structure diagram of high altitude test facility (excluding ejector) with low total pressure and supersonic cold air flow. Compared to conventional secondary-throat diffuser, the new-designed diffuser with a groove is shown in Fig.1 in detail. To seal the pressure environment in vacuum chamber from the cold air, the groove is designed at the start of the secondary throat of the new-designed diffuser. The exhaust-gas conditions of rocket engine employed in study is as follows: the stagnation pressure \( P_{t1} \) is 0.9MPa, the stagnation temperature \( T_{t2} \) is 2000K, and the Mach number \( M_{a1} \) is 4.3.

2.2. Computational Methodology

The Reynolds-average N-S equations for axisymmetric two-dimensional turbulent flow are employed in numerical simulations. Since that the supersonic flow field in diffuser involves in expansion wave/shock wave interaction, shock wave/boundary layer interaction, an emphasis is focusing on the selection of turbulence model for numerical simulation. According to the literatures [7-9], the SST k-\( \omega \) model has the advantages of both k-\( \varepsilon \) model and k-\( \omega \) model, including the options of transition and shear analysis, thus it is suitable for the numerical simulations in this paper.

Figure 2 presents the mesh of the secondary-throat diffuser with rocket engine in Section 2.1, with the number of grid cells about 60000. Finer grids have been used in the gap between the rocket engine nozzle and the convergent duct of the diffuser to accurately capture the flow field, as shown in Fig.2. Finer grids have also been employed in the entrance of cold air to provide tremendous information to understand the flow phenomenon. The height of first layer of grids near the wall is set to 1*10^{-6}m to
ensure that the $Y^+$ of wall is close to 1. The gas employed in numerical simulations meets the formula of ideal-gas, and the viscosity of ideal-gas is expressed by the three coefficient Sutherland formula. The boundary conditions, flow situations and the operational conditions considered for the numerical simulations are summarized in Tables 1 and 2. Simulations have been carried out until the residues fall below $1 \times 10^{-5}$ for all of the flow variables.

![Figure 2. Mesh arrangement of computation model](image)

![Figure 3. Wall pressure comparison of numerical and experimental data](image)

| Location          | Type of boundary condition     |
|-------------------|--------------------------------|
| Engine inlet      | Pressure-inlet                 |
| Inlet of cold air | Pressure-inlet                 |
| Exit of diffuser  | Pressure-outlet                |
| Wall              | No-slip, Adiabatic             |

| Table 2. Parameters varied for study of cold air |
|-----------------------------------------------|
| Parameter                      | Values   |
| Stagnation Pressure, $P_2/(MPa)$ | 0.01, 0.02, 0.03, 0.04 |
| Stagnation Temperature, $T_2/K$       | 300      |
| Mach number, $Ma_2$                 | 1.2, 1.6, 2.0 |

2.3. Verification of Computational Methodology
A comparison between numerical and experimental results was made to guarantee the credibility of computational methodology with the research of J. C. Dudek [10], as shown in Fig. 3. From Fig 3, it can be seen that numerical results are in agreement with the experimental data in spite of some scatter, which indicates that the computational methodology employed in this paper is reliable.
3. Results and Discussion

3.1. Characteristics of flow field of the new-designed diffuser

With the ejection of supersonic cold air with low total pressure into secondary-throat diffuser, a gas film is formed adjacent to the diffuser wall, which interacts with the high temperature gas generated by rocket engine. Thus there is an obviously difference between the flow field of the new-designed diffuser and the conventional diffuser.

Figures 4 presents the typical flow characteristics of secondary-throat diffuser with cold air flow at different Mach numbers (with the same total pressure: 0.03MPa), corresponding to the conventional diffuser at the same condition. Figs. 4 shows that with the interaction between the supersonic cold air and the original inner flow of diffuser, there exist separated regions upstream and downstream of the groove. On one hand, a similarity can be observed in the flow field near the groove with cold air at different Mach numbers. On the other hand, with an increase of the Mach number, the separated region increases gradually. It can be illuminated that with an increase of the Mach number, the static pressure of cold air decreases, thus the interference of cold air on the inlet of diffuser decreases gradually.

Figure 5 displays the wall temperature distribution along the diffuser at different Mach numbers. As shown in Fig.5, compared to the conventional diffuser, the wall temperature of diffuser with supersonic cold air dropped significantly, especially downstream of flow field of the groove. With an increase of Mach number, the wall temperature along the diffuser increases, and the length of film created by cold air reduces. The trends above indicates that the interaction between the cold air and the inner flow of diffuser becomes stronger, which weakened the effect of cold air on the wall of diffuser.

![Figure 4. Flow characteristics of secondary-throat diffuser with vary conditions](image)

(a) Diffuser without cold air
(b) Diffuser with $Ma_2=1.2$, $Pt_2=0.03$MPa
(c) Diffuser with $Ma_2=1.6$, $Pt_2=0.03$MPa
(d) Diffuser with $Ma_2=2.0$, $Pt_2=0.03$MPa
3.2. Influence of cold air on the wall temperature of diffuser

As shown in Figure 6, numerical simulation cases were carried out to investigate the influence of cold air with different total pressures and Mach numbers on the temperature variation of the wall of diffuser. From Fig 6, it can be observed that the wall temperature along the convergent section of diffuser increases gradually, reaching the peak at the secondary-throat section about 2000K. And it is necessary to protect the diffuser from high temperature by water cooling jacket. By contrast, the average wall temperature of the diffuser with cold air is below 1000K, and then the thermal load of the diffuser reduced by more than 50%, which greatly reduces the construction cost of the HAT facility.

From Fig. 6, it can be know that the wall temperature along the diffuser increases with an increase of Mach number at the same total pressure, and decreases with an increase of total pressure at the same Mach number. It’s worth noting that for Mach1.2 of cold air, its interference on the convergent section of diffuser increases with an increase of the total pressure. Therefore, it is necessary to carry out appropriate parameters of cold air for different rocket engines.

Figure 7 presents the variation of the mass flow rate ratio of cold air to rocket engine at different total pressures. From Fig. 7, it can be seen for the cold air with lower total pressure (Pr2=0.01MPa), the mass flow rate of cold air with different Mach numbers is almost the same, which attributes to the compression of inner flow of the diffuser. Beyond that, with an increase of the total pressure, the mass flow rate of cold air is proportional to the Mach number, which provides a reference for the design of groove with cold air.
For the further investigation of the effects of cold air on diffuser at vary conditions including different total pressure and Mach number, the variation of temperature with ratio of wall temperature of diffuser with cold air to conventional diffuser is presented in Fig. 8. As shown in Fig. 8, the wall temperature of the diffuser with cold air decreases gradually with an increase of total pressure, whereas the mass flow rate of cold air increases gradually.

According to Fig. 7 and Fig. 8, aimed at minimizing the load of the ejector downstream of secondary-throat diffuser and avoiding the interference of the cold air at the engine exhaust, the optimal solution for cold air among the cases is total pressure 0.02MPa and Ma number 2. At this condition, the thermal load of wall is reduced by more than 50%, and the flow rate of cold air is only about 12% of the rocket engine.

3.3. Influence of cold air on the ability of diffuser holding shock train

Figure 9 and figure 10 display the Mach counters and wall pressure distribution of the secondary-throat diffuser with cold air at different Mach numbers and total pressures. As shown in Fig. 9 and Fig. 10, for the diffuser without cold air, due to the back pressure at the outlet of diffuser, there exists Mach-disk in the center area of flow field and separated region near the wall of the end of diffuser. And the pressure along the wall of diffuser increases gradually until to the back pressure level. By comparison, for the diffuser with cold air, with a decrease of Mach number, the starting position of pressure rise moves downstream, and wall pressure around the starting position of pressure rise also decreases.

From Fig. 4 and Fig. 9, it can be inferred that a gas film formed by the ejection of cold air causes the compression of the flow field of diffuser, which improves the ability of diffuser holding shock train further. As can be seen from Fig. 10, with an increase of Mach number, the starting position of pressure rise moves upstream to the inlet of diffuser, which leading to a reduction in the ability of diffuser holding shock train; with an increase of total pressure, the starting position of pressure rise moves downstream.
to the end of diffuser, which leading to an improvement in the ability of diffuser holding shock train. This phenomenon illuminates that with the increase of total pressure and the decrease of Mach number, the static pressure of the axial interface of diffuser increases, which is beneficial to the improvement of diffuser holding shock train.

![Image](a) Diffuser without cold air          (b) Diffuser with Ma2=1.2, Pt2=0.02MPa
![Image](c) Diffuser with Ma2=1.6, Pt2=0.02MPa     (d) Diffuser with Ma2=2.0, Pt2=0.02MPa

**Figure 9.** Flow field of secondary-throat diffuser with vary conditions (with Pb=0.032MPa)

![Image](a) Pt2=0.02MPa                  (b) Pt2=0.03Mpa                  (c) Pt2=0.04MPa

**Figure 10.** Wall pressure comparison of diffuser with vary Pt2 conditions (Pb=0.032MPa)

### 4. Conclusion

Aimed at the improvement of capacity of the high altitude test (HAT) facility for rocket engine, a new designed secondary-throat diffuser with ejection of low total pressure and supersonic cold air was proposed. Numerical simulations were carried out to study effects of cold air on flow field structures and diffuser performance. The major conclusions were as follows:

1. The main structural characteristic of the new-designed diffuser is that: a groove is added to the start of the secondary-throat of diffuser, and the supersonic cold air with low total pressure is ejected into diffuser to form the gas film. Thus the wall temperature of diffuser with cold air dropped significantly.

2. The wall temperature of diffuser reduced with the ejection of cold air: the wall temperature along the diffuser increases with an increase of Mach number, and decreases with an increase of total pressure. Besides, the mass flow rate of cold air is proportional to the Mach number and total pressure.
(3) Due to the ejection of cold air and interaction between cold air and inner flow of diffuser, the static pressure of the axial interface of diffuser increases, which is beneficial to the improvement of diffuser holding shock train: with an increase of Mach number, the starting position of pressure rise moves upstream to the inlet of diffuser; and with an increase of total pressure, the starting position of pressure rise moves downstream to the end of diffuser.

References
[1] Manikanda Kumaran R, Vivekanand P K, Sundararajan T. Analysis of Diffuser and Ejector Performance in a High Altitude Test Facility. 45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit. AIAA 2009-5008.
[2] Wang Yong-zhong. Development of cooling system for diffuser [J]. Journal of Rocket Propulsion, 2001, (06): 19-23.
[3] Liu bo, Li jin-xian, Feng xi-ping, Tang jin-lan. An simulational study of supersonic diffuser spray cooling. Computer Simulation, 2010, 27(01):106-109.
[4] JIANG Yi-tong, ZHANG Li-song, ZOU Yang-hui, TIAN Ning, LI Yan-liang. Numerical Simulation Research on Cooling Effect of High-Temperature Wind Tunnel by Water Injection. Journal of Astronautics, 2018, 39(6):702-708.
[5] Vu B T, Bachchan N, Peroomian O, et al. Multiphase modeling of water injection on flame deflector. The 21st AIAA Computational Fluid Dynamics Conference, San Diego, USA, June 24-27, 2013.
[6] Tian ning, Qi bin, Zou yang-hui, Liu zhao-jun. Thermal Protection Design of Diffuser for High Temperature Supersonic Combustion Tunnel. Journal of Astronautics, 2016, 37(09):1129-1134.
[7] Menter F R. Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows. AIAA 93-2906, 1993.
[8] Menter F R. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal, 1994, 32(8):1598-1605.
[9] Bartosiewicz Y, Aidoun Zine, Desevaux P, et al. CFD Experiments Integration in the Evaluation of Six Turbulence Models for Supersonic Ejectors Modeling. Glasgow: Conference Proc. Integrating CFD and Experiments, 2003.
[10] J. C. Dudek, D. O. Davis and J. W. Slater. Validation and Verification of the WIND Code for Supersonic Diffuser Flo [J]. AIAA-2001-0224.