HiSST: International Conference on
High-Speed Vehicle Science Technology
26–29 November 2018, Moscow, Russia

Research of Flow Control on Supersonic Inlet Diffuser

Zhang Jinsheng[1], *Yuan Huacheng*[2] ,*Wang Yunfei*[3], *Huang Guoping*[4]

Abstract

Design of a two-dimension variable inlet operating from Mach 0 to 4 was developed in this paper to meet the specific request. Numerical simulation was carried out to investigate the aerodynamic performance and variable geometric rules of the inlet. In addition, passive control methods were used to improve the performance. The result indicated that the initial inlet basically met the design request over a wide speed range expect Ma3 and Ma3.5.Using geometric modification and suction case could improve inlet performance. In this article, the total pressure recovery σ_e at Ma3 increased about 9% and distortion $\Delta\sigma_0$ decreased 80% at Ma3.5.The thicker boundary layer on the sidewall layer is the reason for the low performance of the original configuration.

Keywords: *Supersonic Inlet; Diffuser; Aerodynamic performance; Numerical simulation*

Nomenclature

D – Diameter of the exit
P_m –Control point of the diffuser
X_m –X-coordinate of the control point
Y_m –Y-coordinate of the control point
σ_e–Total pressure recovery coefficient of the exit
$\Delta\sigma_0$ –Distortion of the exit
Ma–Mach Number
Subscripts
m–Control point
e-Exit of the diffuser

---

[1] *College of Energy and Power Engineering, Nanjing University of Aeronautics and Astronautics ,Nanjing People' Republic of China, E-mail :1021965168@qq.com*
[2] *College of Energy and Power Engineering, Nanjing University of Aeronautics and Astronautics ,Nanjing People' Republic of China, E-mail :yuanhuacheng@nuaa.edu.cn*
[3] *College of Energy and Power Engineering, Nanjing University of Aeronautics and Astronautics ,Nanjing People' Republic of China, E-mail :nuaawangyunfei@126.com*
[4] *College of Energy and Power Engineering, Nanjing University of Aeronautics and Astronautics ,Nanjing People' Republic of China, E-mail :hgp@nuaa.edu.cn*

1. Introduction

Hypersonic vehicle has draw an ever increasing attention in the world[1]-[3]. Inlet is the key aerodynamic component of the hypersonic propulsion system and it has been studied extensively.

As the connection of the inlet and the combustor, diffuser is designed to contain the high pressure rise generated by heat release within the combustor to ensure the normal starting work of the inlet. Usually the thrust of the engine will be larger if the backpressure capability of the diffuser is higher because of more fuel injection. For better performance of the engine, it's necessary to take some control methods to improve the maximum backpressure of the supersonic inlet.

In recent years, a great deal of research has been done on the control methods by experimental and theoretical measures. Passive control methods, such as suction, blowing, and geometric modifications [4]-[9]can affect the boundary layer so that pressure loss associated with shock/ boundary layer interaction could been reduced. Among them, the inlet suction is widely concerned because of simple implementation and obvious effect. Haberle [4], Chang Juntao [5], Yuan Huacheng [6] set suction in the internal contraction section and found that this method could effectively eliminate the flow separation caused by shock/boundary layer interaction which improved the start performance and backpressure capability of the inlet . Tam et al. [7] studied various kinds and positions of suction slot of the isolator, and found that slot suction in the corner of the isolator is conducive to improve the backpressure capability of the inlet. Yue Lianjie et al. [8]found that set suction on the side wall is conducive to improve the backpressure capability of the inlet because the flow vortex of the isolator generated from the boundary layer of the side wall by experiment; White [9] used the method of blowing jet to study the separation of the internal contraction section and get obvious control effect.

Design of a two-dimension variable inlet operating from Mach 0 to 4 was developed in this paper to meet the specific request. Numerical simulation was carried out to investigate the aerodynamic performance and variable geometric rules of the inlet. In addition, passive control methods were utilized to improve the performance.

2. Inlet Diffuser Design

The design of subsonic diffuser is an important factor in the design process of the inlet. The change of geometric shape directly affects the viscous loss and diffuser loss. Inappropriate shape may even cause local flow separation in the internal path, resulting in the decrease of total pressure and increase of distortion at the exit of the inlet. Therefore, under the premise of meeting the overall request of the engine, the design parameters and rules should be reasonably selected to avoid the occurrence of flow separation for higher total pressure recovery coefficient and flow field quality at the exit of inlet.

In this article, the entrance cross section of the diffuser is rectangle while the exit is circle, so conventional rectangle-to-circle S-bend approach is adopted. The main parameters include center line distribution, area distribution and area shape distribution et al. A good combination of these parameters could maximize diffuser performance.

The deflection of the flow in the diffuser is determined by center line distribution which also control the transverse pressure gradient and the secondary flow. A center line distribution with changing starting angle is given which was designed according to the mathematic method presented in the reference [10]-[11].

$$y / Y_s = f(x / L_s) = A(x / L_s)^4 + B(x / L_s)^3 + C(x / L_s)^2 + D(x / L_s) + E$$

Where Ls and Ys are the length and offset distance of the diffuser respectively. The coefficients in the equation are determined by the following constraints: (a)The center line crosses three points(0,0), (1,1), (X_m ,Y_m) ; (b) The initial slope is given and the termination slope is zero so that point Pm (X_m,

Y_m) determines the shape of the line. Selecting the appropriate combination of A, B and C could determine the center line distribution of the diffuser.

The expansion of the flow in the diffuser is determined by area distribution which also control the streamwise pressure gradient and the separation. An area distribution is also given which was designed according to the mathematic method presented in the reference [12].

$$(A_i - A_1)/(A_2 - A_1) = f(x/L_s) = A(x/L_s)^4 + B(x/L_s)^3 + C(x/L_s)^2$$

Where A_1 is the area of the entrance,A_2 is the area of the exit while A_i is the area of arbitrary cross section along the diffuser. Both initial and termination slope are zero. Selecting the appropriate combination of A, B and C could determine the area distribution of the diffuser.

3. Inlet Model and Numerical Simulation

According to the specific request , a two-dimension variable inlet was developed. The inlet is devised to work within Mach 0-4 and cruise at Mach 4.0.

It's difficult to realize starting performance at low Mach number and aerodynamic performance at high Mach number at the same time. To solve the contradictory, a variable geometry plan was put forward as shown in Fig1. Curved compression ramp was used as the first ramp for better compression efficiency while wedge compression ramp used as the second ramp for variable geometry adjustment. By rotating the second compression ramp, the second compression angle and the height of the throat could be controlled. Meanwhile the suction below the throat was applied to match the mass flow between the inlet and the engine. Another function of the variable ramp was controlling the Mach number of the throat while lower Mach number could improve inlet performance. As the flying Mach number changed, the second ramp rotated different angles. The height of the throat increased at low Mach number and decreased at high Mach number so that the inlet could work normally over a wide range flight. The variable geometry plan is similar to the ATREX [13] and it has been verified by many numerical simulations and experiments [14].

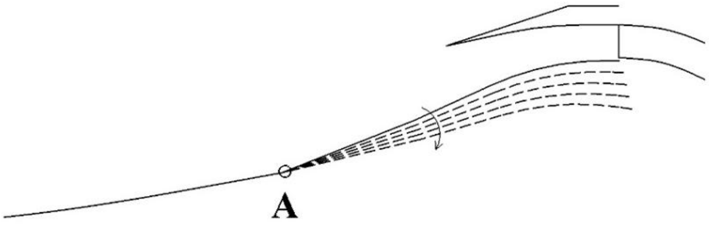


Fig 1. Variable Geometry Plan

Diffuser was designed using conventional rectangle-to-circle S-bend. Centre line distribution and area distribution affect the transverse and streamwise pressure gradient which would cause secondary flow and separation. Centre line starting angle of the S-bend in this paper was controlled and X-coordinate of the control point P_m was 0.5 while Y-coordinate was 0.8. Area distribution we used here allowed a rapid turning at the exit.

Fig.2 shows the sketch and computational mesh of this inlet. The inlet geometry size was dimensionless treated using exit diameter D of the diffuser. The starting and ending angle of the curved compression ramp are 6° and 10°,and the secondary compression ramp degree is 11.5° .The overall length of the inlet is 10.17D while the width of the half-model is 0.5D. The length of the external compression ramp、the internal compression ramp and the diffuser are 4.04D、2D and 4.67D respectively .The capture height is 1.56D and the throat height is 0.3D.

A numerical simulation of the steady flow field in inlet was performed with FLUENT solver. The Reynolds averaged Navier-Stokes equations in three dimensions were solved with a finite volume spatial discretization method. The turbulent flow was modeled by using the one-equation SA. The fluid was treated as compressible ideal gas.

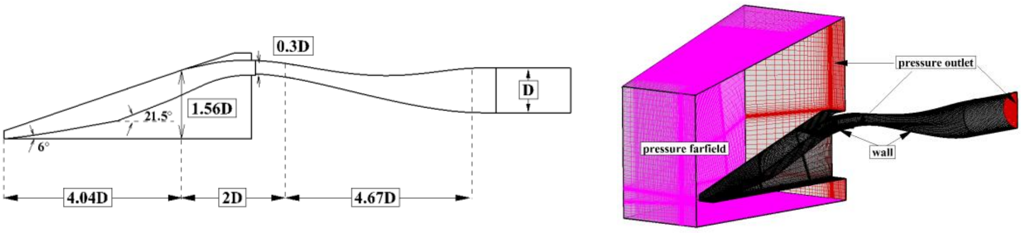
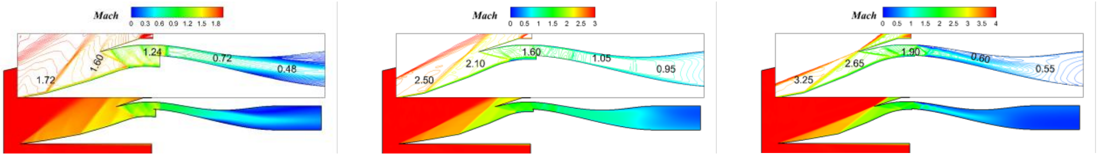


Fig 2. Sketch of Initial Inlet and Computational Mesh

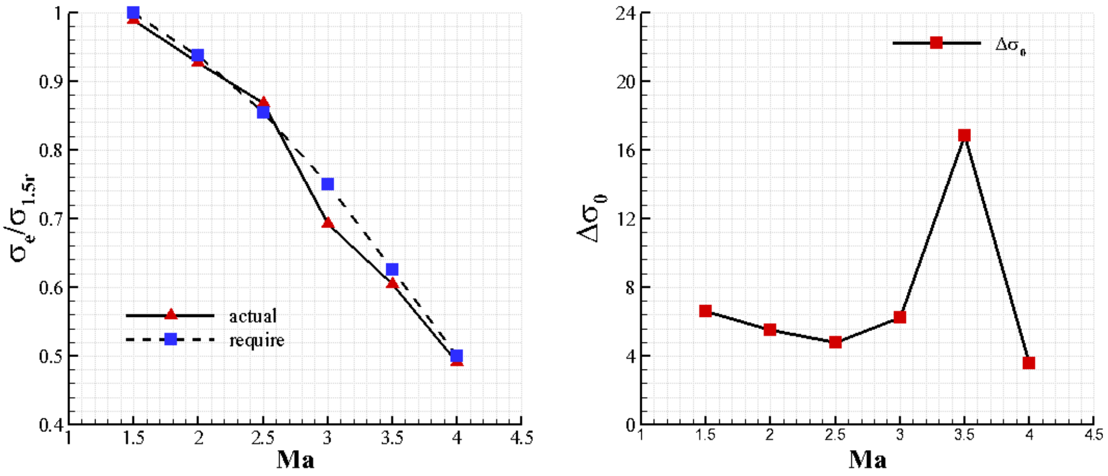
Fig.3 shows the Mach Number contour of the initial inlet in typical conditions (Ma2, 3, 4). It can be seen from the picture that as the Mach number increases, the second compression ramp rotates around the fixed hinge and the second compression angle increases gradually. Meanwhile, the curved shock wave and oblique shock wave move toward the cowl lip and attach on the lip at Ma4.0. The inlet works normally at all flight condition and there is no obvious flow separation in the internal flow path.



Ma=2 Ma=3 Ma=4

Fig 3. Mach Number Contour of the Initial Inlet at Different Flight Mach Number

Also, the aerodynamic performance of the inlet is given in Fig.4(a) and Fig.4(b) where X-coordinate represents fly Mach number, Y-coordinate represents total pressure recovery coefficient σ_e and distortion $\Delta\sigma_0$ of the inlet exit. Different lines represent numerical result and design request respectively. The blue curves in Figure (a) represent the request and the red curve represents the actual performance. And in Figure (b), the red curve represents the actual distortion $\Delta\sigma_0$ of the inlet exit. The result indicates that the initial inlet basically met the design request over a wide speed range expect Ma3 and Ma3.5. The total pressure recovery coefficient on Ma3 is slightly low and the distortion of M3.5 is large. In order to ensure the good aerodynamic performance of the inlet under various operating conditions, it is necessary to carry out the study on the performance optimization scheme of Ma3 and Ma3.5.



(a) total pressure recovery coefficient (b) distortion

Fig 4. Aerodynamic Performance of the Initial Inlet at Different Flight Condition

4. Optimized Case

4.1. Geometric Modification Case

The first method was geometric modification, in this case, the diffuser was redesigned .Following the design method of S-bending diffuser, the center line starting angle was adjustable, Y-coordinate of the control point P_m changed from 0.8 to 2.0 so that the diffuser became more winding. Area distribution we used here still allowed a rapid turning at the exit.

Fig.5 shows the sketch and computational mesh of this modified inlet. Compared with Fig.2 above, it can be seen that the diffuser changes from single S-bend to double S-bend. There are sharp bends in the diffuser, and the pressure gradient in streamwise and transverse direction are strong, so it is easy to form complex secondary flow. The total pressure recovery coefficient and the distortion would be poor when there is no backpressure. However, under the working conditions with backpressure, the actual length of the diffuser becomes longer and the area expansion rate decreases compared with the single S-bend, so the actual backpressure capability of the inlet would enhance slightly.

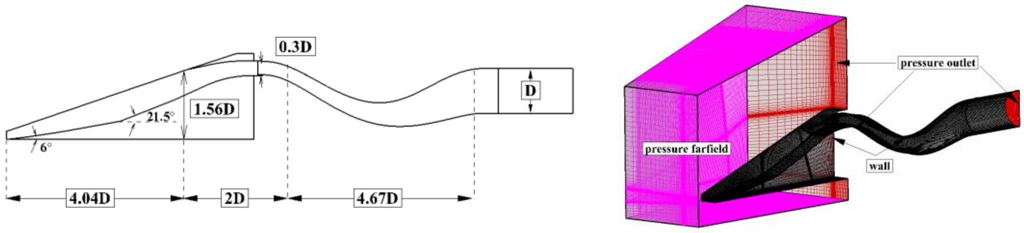
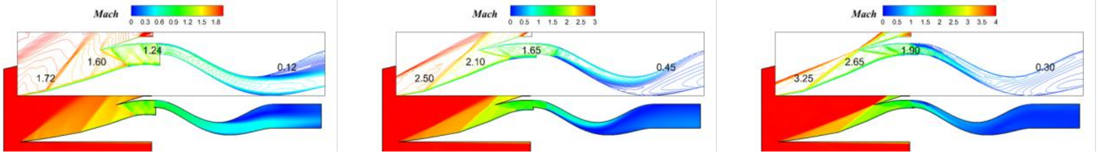


Fig 5. Skech of Geometric Modification Inlet and Computational Mesh

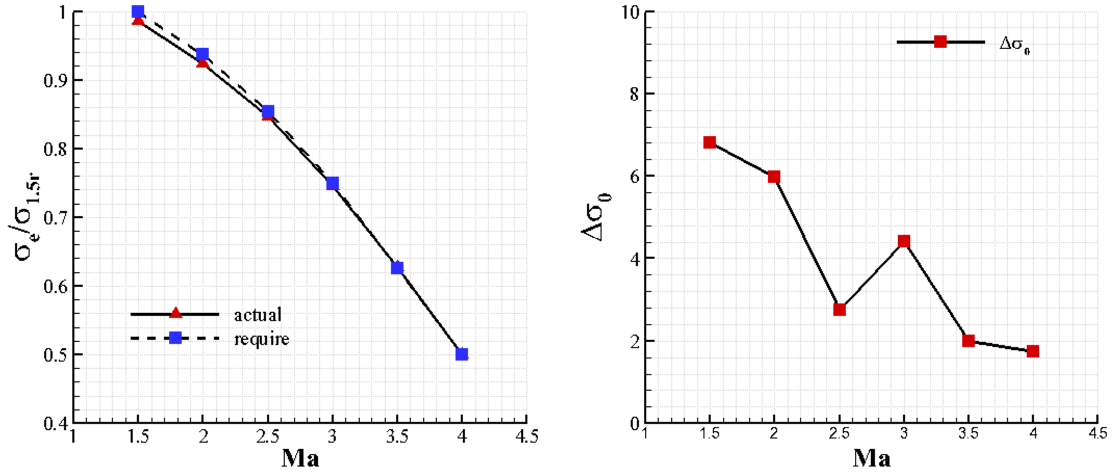
Fig.6 shows the Mach Number contour of the geometric modification inlet in typical conditions (Ma2, 3, 4). It can be seen from the picture that the upstream flow field has not been changed after geometric modification of the diffuser. The shock train at Ma3.0 becomes obvious after the optimization, and the flow field structure at Ma2.0 and Ma4 almost remain unchanged.



Ma=2 Ma=3 Ma=4

Fig 6. Mach Number Contour of Geometric Modification Inlet at Different Flight Condition

Also, the aerodynamic performance of the inlet was given in Fig7(a) and Fig7(b) where X-coordinate represents fly Mach number, Y-coordinate represents total pressure recovery coefficient σ_e and distortion $\Delta\sigma_0$ of the inlet exit. Different lines represent numerical result and design request respectively. The result indicates that after geometric modification the inlet basically met the design request over a wide speed range. Compared with the initial inlet ,the total pressure recovery σ_e at Ma3 increased 7.6% while distortion $\Delta\sigma_0$ at Ma3.5 decreased 88%.

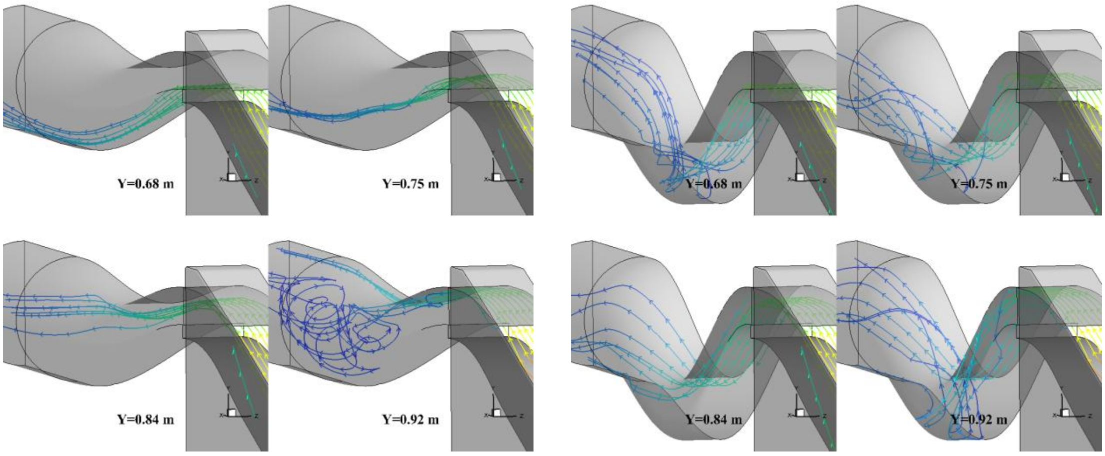


(a) total pressure recovery coefficient

(b) distortion

Fig 7. Aerodynamic Performance of Geometric Modification Inlet at Different Flight Condition

Fig. 8 shows the flow streamline at different heights of the diffuser before and after the geometric modification. Fig.8(a) shows the initial configuration and Fig.8(b) shows the optimized one. It can be seen from the picture that the inner streamline distribution of the diffuser is different. Compared with the initial diffuser, the inner streamline of optimized one is more smooth with stable flow field. In addition, the streamline from the cowl side is more disordered than compression side which contribute to reflux in the diffuser. Double S-bend is conducive to guide the flow along the specified path because of the smaller expansion rate. The generation of the reflux has been suppressed so that maximum backpressure of the inlet enhanced.



Y_m=0.8(initial)

Y_m=2.0(optimized)

Fig 8. Flow Streamline in the Diffuser before and after Geometric Modification

4.2. Suction Case

The second method was suction, in this case, five vertical bleed slots were set on the side wall at the entrance of diffuser as shown in Fig.9. The slots were opened only at Ma3.0 and Ma3.5 because of the bad performance.

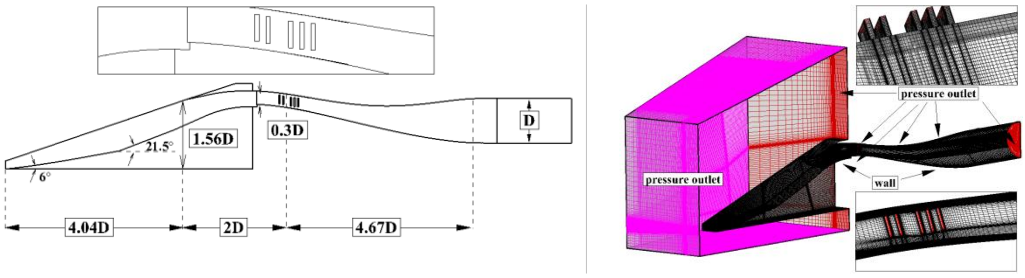
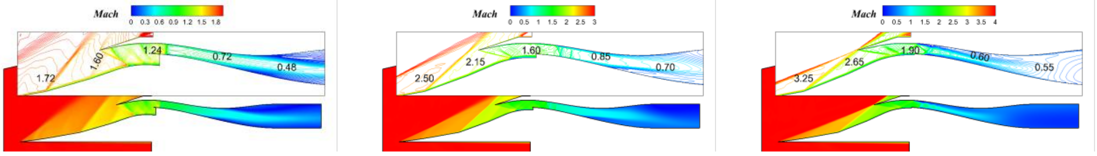


Fig 9. Skech of Suction Inlet and Computational Mesh

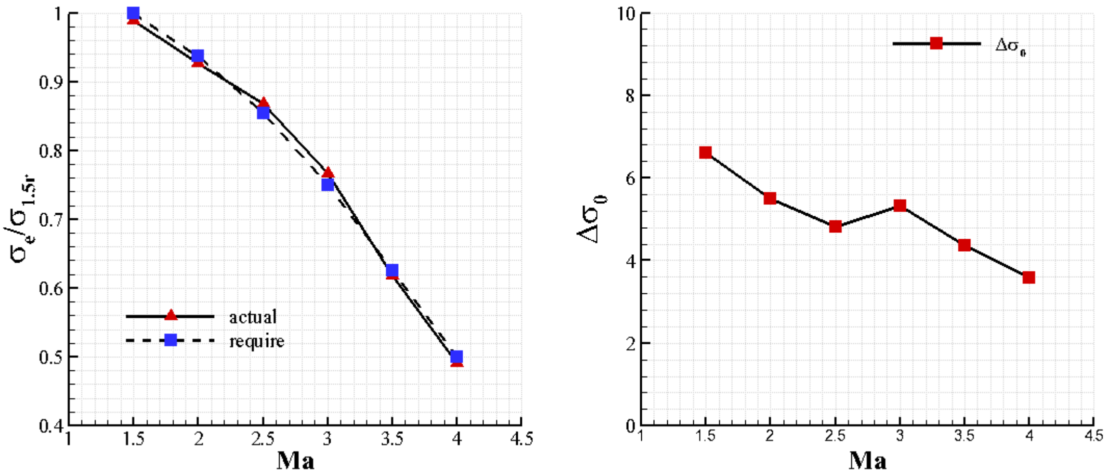
Fig. 10 shows the Mach Number contour of the inlet with suction in typical conditions (Ma2, 3, 4). It can be seen from the picture that the upstream flow field has not been changed using suction case. Similar to the geometric modification case ,the shock train at Ma3.0 becomes obvious after the optimization, and the flow field structure at Ma2.0 and Ma4.0 almost remain unchanged.



Ma=2 Ma=3 Ma=4

Fig 10. Mach Number Contour of Suction Inlet at Different Flight Condition

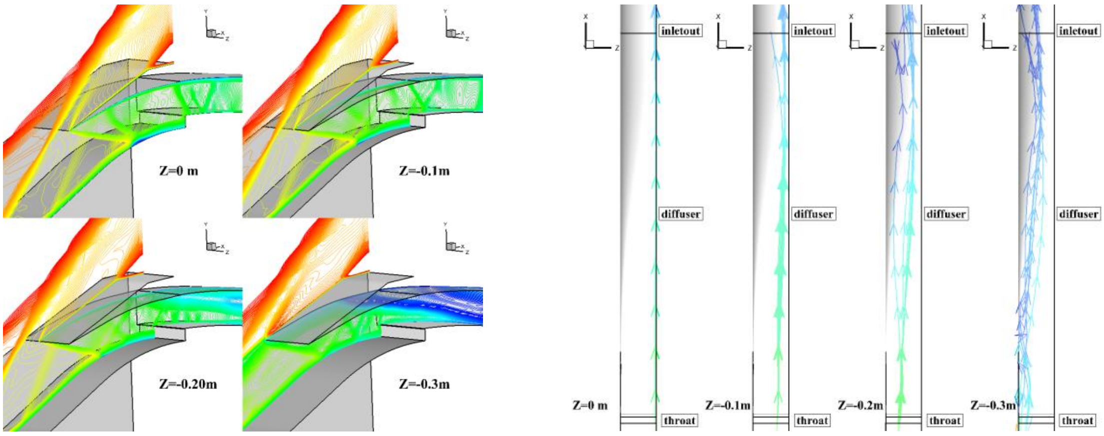
Also, the aerodynamic performance of the inlet was given in Fig.11(a) and Fig11.(b) where X-coordinate represents fly Mach number, Y-coordinate represents total pressure recovery coefficient σ_e and distortion $\Delta\sigma_0$ of the inlet exit. Different lines represent numerical result and design request respectively. The result indicates that after suction modification, the inlet basically met the design request over a wide speed range. Compared with the initial inlet ,the total pressure recovery σ_e at Ma3 increased 10.7% while distortion $\Delta\sigma_0$ at Ma3.5 decreased 74% .



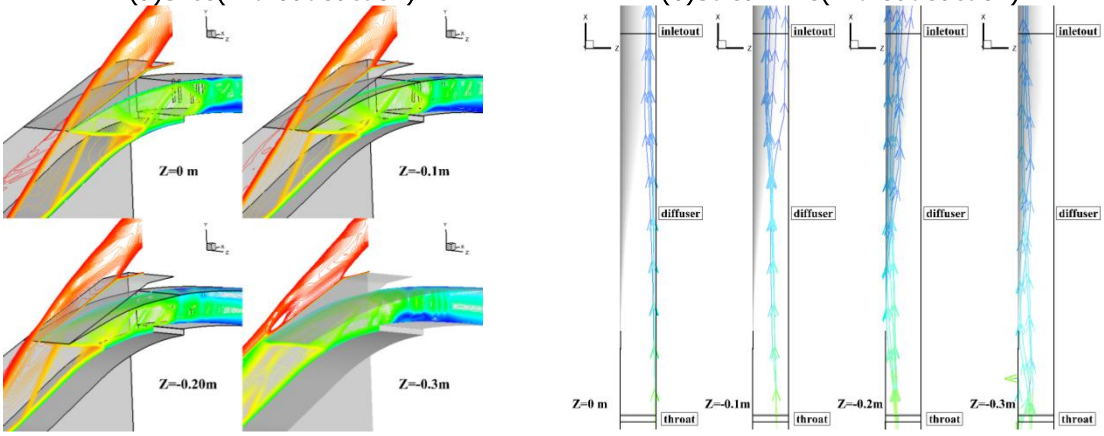
(a) total pressure recovery coefficient (b) distortion

Fig 11. Aerodynamic Performance of Geometric Modification Inlet at Different Flight Condition

Fig. 12 shows the slice and streamline of the flow field at different spanwise positions of the inlet before and after suction. It can be seen from the picture that the boundary layer on the sidewall is thicker than that on the upper and lower wall because of the window of bleed region blew the throat. Although the shock train on the symmetrical plane have not yet reached the throat, the shock train on the sidewall plane has already reached the throat so that the backpressure has affected the upstream flow field (see Fig.12(a)).In addition, airflow along the side wall move to the middle which cause local separation(see Fig12.(b)).After suction modification, airflow moving towards the middle is driven back to the sides because of differential pressure which weaken the flow separation. Thus the streamline near the symmetrical surface does not deflect(see Fig.12(c) and Fig.12(d)).



(a)Slice(without suction) (b)Streamline(without suction)



(c)Slice(with suction) (d)Streamline (with suction)

Fig 12. Slice and Streamline of the inlet before and after Suction

5. Conclusions

Design of a two-dimension variable inlet operating from Mach 0 to 4 was developed to meet the specific request. Numerical simulation was carried out to investigate the aerodynamic performance and variable geometric rules of the inlet. The result indicated that the initial inlet basically met the design request over a wide speed range.

Geometric Modification changing diffuser from single S-bend to double S-bend improves inlet performance. In this article ,the total pressure recovery σ_e at Ma3.0 increased 7.6% and distortion $\Delta\sigma_0$ decreased 88% at Ma3.5.

Setting suction slot on the sidewall of the diffuser improves inlet performance. In this article ,the total pressure recovery σ_e increased 10.7% at Ma3.0 and distortion $\Delta\sigma_0$ decreased 74% at Ma3.5.

The two methods proposed in this paper provide some reference for this kind of problem and also have certain vales for theory study and application research.

6. Acknowledgments

This work is supported by the National Natural Science Foundation(No.11772155) and Postgraduate Research & Practice Innovation Program of Jiangsu Province(No.KYCX18_0313).

References

1. Walker B，Kennedy K，Mikkelson C: US army hypersonic scramjet propelled missile technology program. AIAA Paper 2006-7927(2006).
2. Heiser W H，Pratt D T，Daley D，et al: Hypersonic airbreathing propulsion. Washington DC: AIAA Paper Education Series(1994).
3. Cai Guobiao, Xu Dajun :Hypersonic Vehicle Technology. Beijing: Science Press(2012)
4. Haberle J,Gulhn A: Investigation of the performance of a Scramjet Inlet at Mach 6 with Boundary Layer Bleed. AIAA-2006-8139(2006).
5. Chang Juntao, Bao wen,Cui Tao: Effect of suctions on maximum backpressure ratios of hypersonic inlets. Journal of Aerospace Power,23(3):505-509(2008)
6. Yuan Huacheng ,Liang Dewang: Effect of suction on starting of hyperson ic inlet . Journal of Propulsion Technology,27(6):525-528(2006).
7. Tam C, Eklund D, Behdadnia R, et al: Investigation of Boundary Layer Bleed for Improving Scramjet Isolator Performance. AIAA-2005-3286(2005).
8. Yue Lianjie,,Ye Qing,Xu Xiankun:Boundary layer bleeding of three-dimensional compression hypersonic inlet. Jouranl of Aerospace Power,27(2):372-378(2012).
9. White M, Lee R, Thompson M, et al: Tangential mass addition for shock/boundary-layer interaction control in scramjet inlets. Journal of Propulsion, 7(6): 1023-1029(1991).
10. Wang Weixing, Gu Qiang, Guo Rongwei.: Study of Flow Control of Inward Turning Inlet. Journal of Propulsion Technology,38(5):961-967(2017).
11. Sun shu: Design and Investigation for Unconventional Inlets. Nanjing University of Aeronautics and Astronautics(2006)
12. Li Jianhua, Bao Xiaoxiang, Liu Kai: Investigation on the design and flow characteristics of diffusing double-S inlet. Advances in Aeronautical Science and Engineering,8(2):219-225(2017)
13. Lee C C, Boedicker C: Subsonic diffuser design and performance foe advanced fighter aircraft. AIAA-85-3073(1985).
14. Kojima T, Taguchi T H，Aoki K,et al: Development study of the air-intake of the ATREX engine.AIAA-2003-7042(2003).
15. Liu Jun, Yuan Huacheng, Ge Ning: Design and flow characteristics analysis of mode transition simulator for tandem type TBCC inlet. Acta Aeronautica et Astronautica Sinica,37(12):3675-3684(2016).