DOI: 10.13009/EUCASS2017-55

Preliminary Investigation on the Starting Characteristics and Flow Control of an Inward-turning Inlet

Anyuan YU, Dawei YANG**, Pengyu GUO**, Yingchuan WU*, Feng WEI**, and Jialing LE**
**Science and Technology on Scramjet Laboratory of CARDC, Mianyang, China, 621000*
***Airbreathing Hypersonic Technology Research Center of CARDC, Mianyang, China,621000*

Abstract
An inward-turning inlet is designed and its starting characteristics, as well as the flow control, are numerically researched first, then a test model of the researched inlet is manufactured and some verification experiments are launched in the current paper. The main designed Mach number of the inlet is 6.5. The diameter of the inlet throat is 50mm. The total contraction of the inlet is 5.6, and the internal contraction from the end section of the entrance to the throat is about 1.7. The nominal Mach number of the experiments is 5, at which the numerical research shows that the inlet can start but can't self-start. In order to make the inlet self-start, some flow-control with bleeding are investigated and the reasonable position of the bleeding zone, the area of the zone, and the orifice ratio of the zone are obtained from numerical simulation. On the base of these numerical results, some replacement parts with bleeding are designed for the test model. The experiments are carried out in a routine cold-flow blow-down hypersonic wind tunnel. The outlet section of the round-cross-section nozzle used in the current experiments has a diameter of 0.5m, and the nominal Mach number of the nozzle is Ma5. The total pressure of the tests is about 10atm, and the total temperature is about 360K. In the tests, silk threads and shock wave schlieren video are used as flow visualization methods to show the structure of the flow field, by which to judge the status of the inlet. The bleeding parts can be replaced to conduct the research of the flow control at typical unstarted status. The tests get the typical starting characteristics of the inward-turning inlet with and without the flow control. All the results prove that the flow control is effective and that the design method and the numerical simulation of the inward-turning inlet are soundly reliable.

1 Introduction

The research of inward-turning inlet [1],[2] is a hot research field comparing with those of planar compressional ones and axisymmetric compressional ones for hypersonic frame/propulsion integrated vehicles, because an inward-turning inlet has many advantages such as strong compression, low drag, high integration and etc. In fact, the inward-turning inlet has some similarities to side-wall compressional inlet, but the compression of the former is better-proportioned than that of the latter.
The design method of typical inward-turning inlet [3] is totally different from other inlets. Generally speaking, there are three main processes to design an inward-turning inlet. Firstly, a basic flow field adopting an inner conical compression [4],[5],[6],[7] is designed often by using MOC. Secondly, a series of streamlines are extracted from the basic flow field designed in the previous process for the use in the next process. Lastly, the captured curve or the outlet outline should be specified so that those streamlines extracted in the previous process can be put in the proper positions on the curve, thus a flow tube or an inward-turning inlet is constructed.
But the aero-performances of such an inward-turning inlet will always decline when the leading edge of the inlet is blunted or the sharp corner of the entrance is blended, or the surface of the inlet is reshaped to satisfy the overall need of the frame/propulsion integration. In order not to decline the aero-performances, some special handling should be done for the design. These handlings will be introduced in the current paper.
The inward-turning inlets has particular features [8] that makes its aerodynamic performances different from other inlets. For example, on the one hand, three dimensional compression leads to strong compression and high flow capturing, on the other hand, the starting problem may be serious for there is no natural window for air flow to overflow. Meanwhile, three spatial compressions will also form a thick boundary layer which comes from the upstream and accumulates at the shoulder of the inlet, where there always is strong interaction of shock/boundary layer when an incident shock generated from the inlet cowl gets to the same place. Moreover, the flow field becomes particularly complex after this interaction and the interferences between streamlines enhance the complex, thus the

DOI: 10.13009/EUCASS2017-55

separation in the internal flow increases so that the ability of back-pressure-resistance decreases greatly. In this case, in order to let the inlet start or restart, and to let the inlet resist the high back-pressure from the combustor, flow control must be brought to the inlet[1][9].

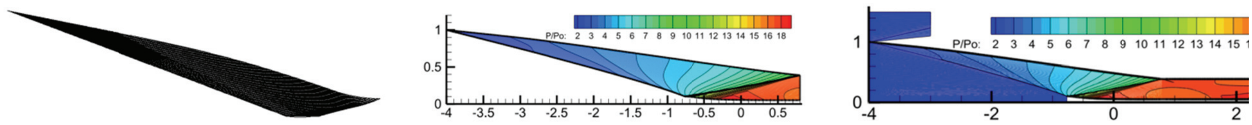
On the basis of the qualitative knowledge above about the inward-turning inlet, a typical inward-turning inlet is designed, and the aero-performances including the starting characteristics are obtained by numerical simulation [12], and some flow control [10] is also launched in the paper. At last, using the same flow visualization as used by Li Zhufei, etc. [13], some experiments with bleeding flow control [1] are conducted to check the starting characteristics at Ma5 in a blow-down hypersonic wind tunnel. The results show that the flow control is effective and that the design method and the numerical simulation of the inward-turning inlet are soundly reliable.

2 Design Processes

The design method of an inward-turning inlet includes three main processes [2],[3],[4],[7],[11]. The first process is to design a basic flow field. The second process is to extract streamlines from the basic flow field. The third process is to construct flow tube by osculating streamlines sheets to special position. The main processes are simply described as follows.

2. 1 The Basic Flow Field

For the basic flow field, a configuration of double-shock compressional system [7] is adopted in the current paper. The compressional system is designed by means of MOC (method of characteristics). Figure 1a gives the grids of MOC, and Figure 1b gives an typical basic flow field from MOC. With the comparison to inviscid CFD result in Figure 1c, it can be seen that the design of MOC is credible.



(a) The grids of MOC (b) The field from MOC (c) The field from inviscid CFD

Figure 1 Basic flow flied design

2. 2 Streamlines Extracted from the Basic Flow Field

From the basic flow field calculated by MOC, a series streamlines are extracted, each with a specific position. That is to say, each of these streamlines has a position which can be marked by the distance from the axis of the inner cone zone. Figure 2 gives some typical streamlines extracted from the basic flow field.

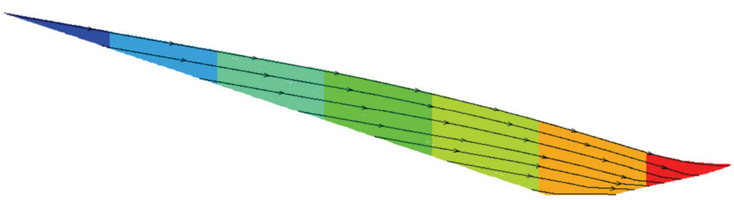


Figure 2 Typical streamlines extracted from the basic flow field

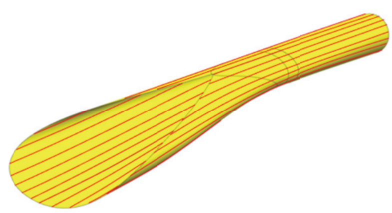


Figure 3 Flow tube made of streamlines

2. 3 Flow Tube Constructed through osculating with streamlines

The key process of designing an inward-turning inlet is to construct osculating sheet with streamlines. Since each streamlines has a specific position, when an inlet captured curve(ICC) is given, every streamline can be set at its point on the ICC just corresponding to the distance from the axis of the inner cone zone. If an inlet outlet curve(IOC) is given, the process is almost the same. Thus the flow tube can be constructed and an inward-turning inlet can be designed. Figure 3 gives an inlet with some streamlines shown on the surface of the inlet.

DOI: 10.13009/EUCASS2017-55

3 Special Handling for the Design

Theoretical design for an inward-turning inlet will has a geometry that always can't satisfy the overall needs, so some special handling for the design should be done to enhance the performances of the inlet. Among these handlings in current design are the blunting of the leading edge, the blending of the entrance corner, and the reshaping of the flow path.

3. 1 The leading edge bluntness of the basic flow field

If a basic flow field has no bluntness for the leading edge, there will be no bluntness for the inward-turning inlet either. But the practical inlet always has a bluntness[11]. This bluntness always changes the external shock to deviate the cowl tip, resulting directly in a reduction of the air captured ratio. To solve the problem, a method used in the paper is that the bluntness should be considered before basic flow field design. That is to say, before the basic flow field is designed, the flow field with a specific bluntness is obtained first. By analysing the flow field, the average Mach Number after the bow shock of the blunt leading edge is achieved and is used to design the basic flow field. This correction method always guarantees the aero-performances of the inlet at design Mach Number. Figure 4 gives this effect of Leading edge bluntness on the basic flow field.

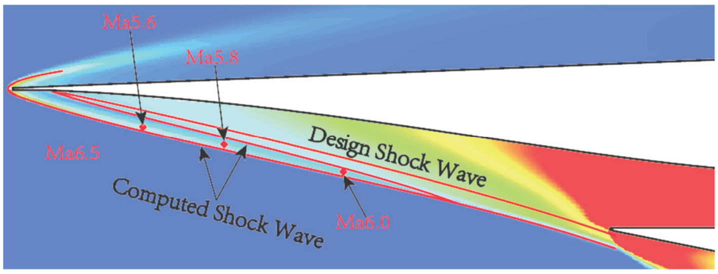


Figure 4 The effect of Leading edge bluntness on flow field

3. 2 The sharp corner blending of the entrance

For an inward-turning inlet, there often is a sharp corner at the rear part of the inlet entrance. As is known, the sharp corner will has much more heat flux than blunted edge. Usually the sharp corner can be handled by geometric blending. Yet such handling will decrease the captured mass flow rate. In the current research, a special handling to blend the inlet entrance is not geometrical, but aerodynamic. The detailed practice is just set the specific streamlines along the curved edge which has been blended already. Figure 5 gives the contrast between the original corner and the blending edge of the inlet entrance.

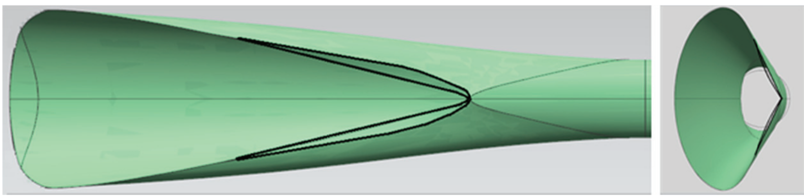
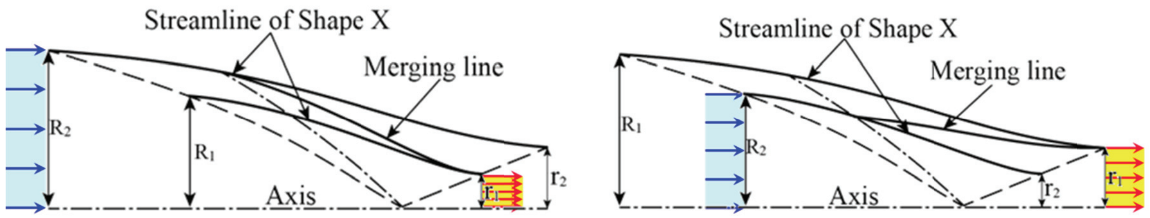


Figure 5 The sharp corner blending of the inlet entrance

3. 3 The shape transition for the treatment of streamlines

It's true for a basic flow field that the outline of the throat of the inlet matches the aero-captured curve of the inlet one-to-one. When used in vehicles with frame/propulsion integration, the inlet needs to match not only the inlet of the engine, but also the fore-body of the vehicles. This often results in a disagree with geometry. One solution used in current paper is to transit streamlines to specific position by mathematical merging, which is shown in Figure 6. These treatments don't change the external shock system and have no influence on the capturing flow mass rate, and they always satisfy the need of the frame/propulsion integration.

DOI: 10.13009/EUCASS2017-55



(a)When R2>R1 and r1<r2 (b)When R2<R1 and r1>r2

Figure 6 Two typical treatments to transit the streamlines

3. 4 The design result

The design parameters chosen for the inward-turning inlet are as follows. The design Mach Number is 6.5, the angle of attack is 0 Deg. The bluntness of the leading edge is 0.5mm. The radius of the blending round at the corner of the entrance is 6mm. The throat of the inlet is a circle with a diameter of 50mm. Considering the leading edge bluntness, the design method of inward-turning inlet, the shape transition and double-shock system, with the help of limited optimization[11], an inward-turning inlet is designed in the current paper. Figure 7 gives the design result.

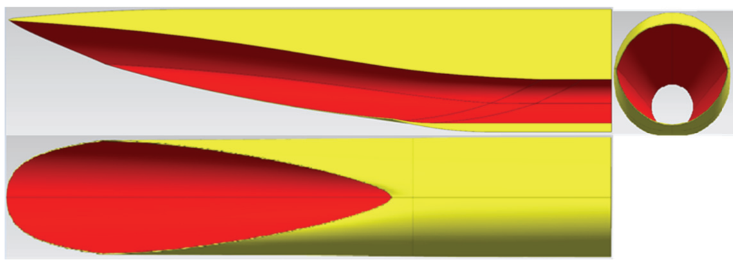


Figure 7 The designed inward-turning inlet

The inlet has a total length of 950mm, including an equal-area isolator with a cross section of circle just the same as the throat section. The total geometric contraction ratio of the inlet is 5.6. The entrance of the external compression has a length of 450mm. The length of the internal compression path from the cowl to the throat is 150mm, and the length of isolator is 350mm. This inlet will be used in the next numerical and experimental research.

4 The mesh and the numerical method

The mesh is composed of multi-block structured grids, generated through mesh-generation software. A half model with half calculation domain is adopted because the inlet is left-right symmetric. The outside boundary of the calculation zone is a half ellipse-sphere, of which the shape is well corresponding to the inlet model. The mesh of the domain has about 15 million hexahedral grids.

In order to simulate the effect of wall boundary layer, the first layer of the grids is set to 0.001mm, to ensure the parameter of Y+ is about O(1). The grid size increases according to geometric series with a coefficient of 1.1 along the wall normal direction. Figure 8 shows the structured blocks of the inlet domain.

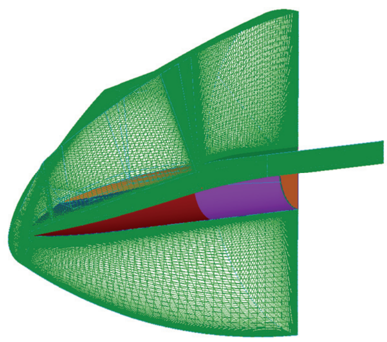


Figure 8 The calculation domain and structured multi-blocks

DOI: 10.13009/EUCASS2017-55

The numerical method used in the paper is AHL3D, a parallel CFD software self-developed by CARDC independently. AHL3D solves the time-dependent, Reynolds-averaged Navier-Stokes equations for turbulent, compressible flows using a finite volume, time-marching approach on multi-zone, structured grids. Spatial accuracy is formally second order using the steger-warmming flux-difference splitting upwind formulation for inviscid flux. Steady flows are simulated through an iterative process using local time stepping. Unsteady flows are simulated through a second-order marching in time. Turbulence models can be chosen from S-A one-equation model, or k-e/k-ω two-equation models. AHL3D is capable of solving for flows of speeds ranging from low subsonic to hypersonic, especially for supersonic combustion flow. At high temperature AHL3D has models to simulate high temperature effects in air with specific heat as a function of temperature.
For the current calculation, the solver used is based on Navier-Stokes coupled solution algorithm, adopting time-dependent method to march to a steady state with a turbulent model of kw-sst and a material of ideal gas. The calculation domain to be discretized is also shown here. The boundary conditions include free-flow (incoming flow) boundary, non-slip adiabatic wall boundary, free-pressure outlet boundary and symmetry boundary conditions. Table 1 gives the free incoming flow boundary conditions used in current numerical simulation.

Table 1 The free incoming flow boundary conditions

Ma0 (Nominal)	**Total pressure (Nominal)**	**Static pressure**	**Total Temperature**	**Static Temperature**	**AOA**	**Yaw angle**
5.0	10atm	1890Pa	360K	60K	0Deg	0Deg

The calculation adopts steady-calculation. The criterion of convergence is that the mass flow rate and other parameters of the inlet at any internal flow path are the same as a constant value respectively and don't change any more, and that the residual such as pressure, velocity, mass flow rate, etc. decrease under 10^{-4}. The incoming flow is set as ideal gas, the specific heat is fixed to be 1.4, and the coefficient of the viscous is calculated by equation of Sutherland.

5 The CFD results

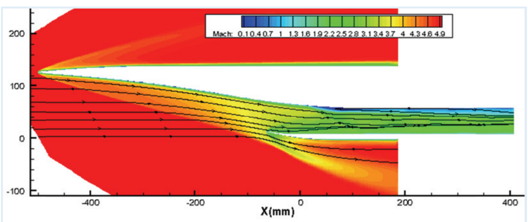
5. 1 The starting and self-starting characteristics

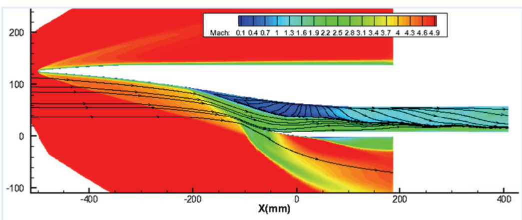
In order to obtain the starting and self-starting characteristics by CFD, many ways with different tactics are used in current research, among them are Zero-Init, Far-Init methods[12]. The results obtained by means of these tactics are shown in Table 2.

Table 2 The starting and self-starting characteristics at throat section from CFD

No.	**Ma0**	**Calculation tactic**	**Mass flow ratio**	**Ma ratio**	**Total pressure Ratio**	**results**
1	Ma5	Far-Init	0.887	47.1%	0.638	Started
2	Ma5	Zero-Init	0.726	37.7%	0.372	Unstarted

From the table, it can be seen that Zero-Init calculation for inward-turning inlet will lead to unstarted solution, which is like the calculation for inlets under lift-bodies(see examples in document[12]). The result shows that at Ma5 the inlet doesn't self-start yet.
Figure 9 gives the flow field result calculated at Ma5 by using Far-Init and Zero-Init methods[12]. So one can see that at Ma5, the inlet has two states, one is started and the other is unstarted. In other words, there are double-solutions at Ma5 for the current inward-turning inlet.



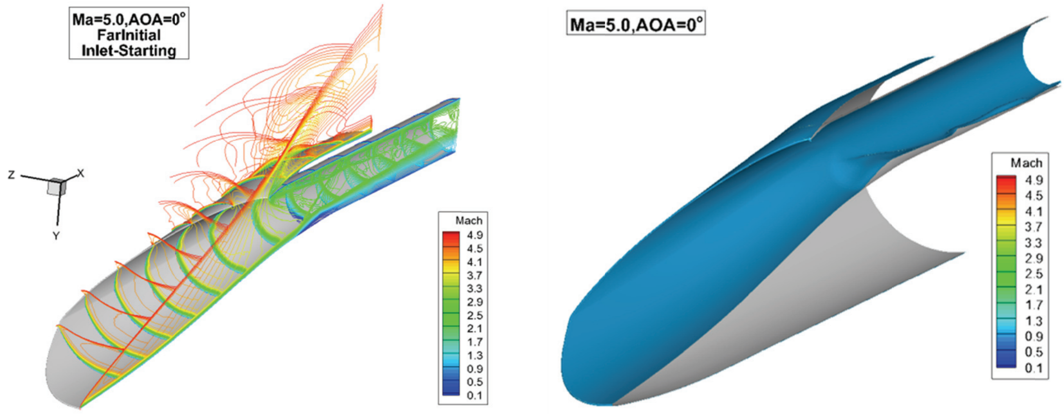


DOI: 10.13009/EUCASS2017-55

(a)Started(from Far-Init method) (b)Unstarted(from Zero-Init method)

Figure 9 Mach Number contours at the meridian plane at Ma5

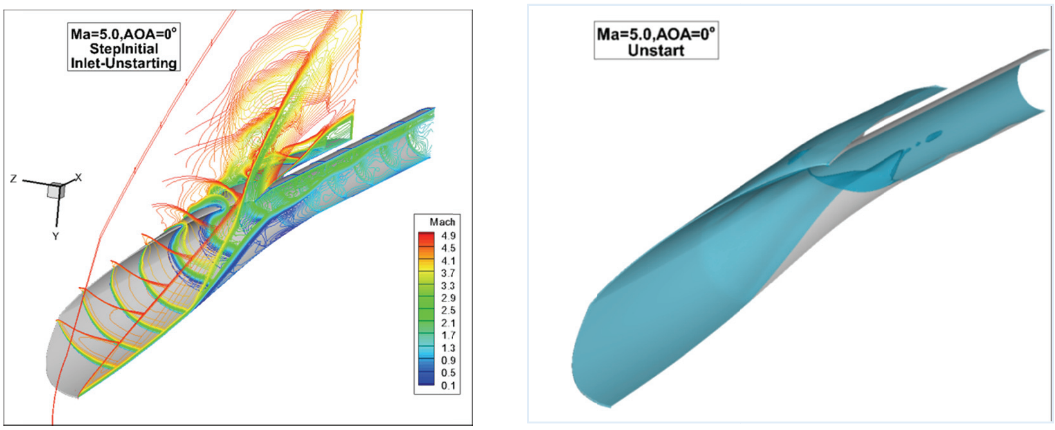
Figure 10 gives the result of inlet starting characteristics through the method of Far-Init[12]. Through analysing the boundary layer of the inlet, it can be seen that the boundary layer gets thicker and thicker along the internal flow path downstream of the inlet shoulder till the outlet of the inlet. It is obvious that the reason for the boundary layer getting thick abruptly at inlet shoulder is that shock/boundary layer interaction(the incident shock is generated by cowl) happens at the shoulder, while the reason for the boundary layer getting thick continuously after inlet shoulder is that the three dimensional compression in the internal flow path leads inverse pressure gradient to be increasing continuously.



(a) The flow field (b) Isosurface of Ma=1

Figure 10 The flow field and an iso-surface at Ma5 for the started state

The phenomenon that the boundary layer at inlet shoulder connects to the boundary layer after shoulder indicates that the internal flow field of the inward-turning inlet is very complex. It's obvious that the development of the boundary layer is also influenced by the three dimensional compression and the flow swirl in the flow tube, so the boundary layer is interacted in a limited zone just near the meridian plane. This means that the thick boundary layer has a limited effect on the ability of back-pressure resistance.



(a) The flow field (b) Isosurface of Ma=1

Figure 11 The flow field and an iso-surface at Ma5 for the unstarted state

Figure 11 gives the result of inlet unstarting characteristics through the method of Zero-Init[12]. From the figure it can be seen that the inlet is unstarted at Ma5 now. Associating this unstarted result with the started result (see Figure 10), one can see that both started and unstarted solutions are the possible solutions for the current inward-turning inlet at Ma5.

Comparing unstarted flow field in Figure 11 with started flow field in Figure 10, it can be seen that there is almost no changes of the external wave system, so it is difficult to judge the state of the inward-turning inlet only by depending on the external shock wave system. This is not the same as planar or axisymmetric inlets. Since the external shock

DOI: 10.13009/EUCASS2017-55

wave can't be used to judge the states of the inward-turning inlet, the internal flow field have to be considered in wall pressure distribution along the flow direction, or the statistic aero-parameters such as Mach Number, total pressure at throat section or outlet section, instead of the external flow field to be considered. For example, Figure 12 gives the wall static pressure distribution in the flow direction. There are two curves in the figure, one is corresponding to started state (solid curve), and the other is corresponding to unstarted state (dashed curve). It is distinct that the wall pressure distributions are different from each other, and that the pressure rises suddenly just at the beginning of the separation on external compression surface. So one conclusion can be drawn that when it isn't convenient for researchers to observe the flow inside, the pressure distribution along the wall can be used to be a criteria to judge whether an inward-turning inlet is started or not.

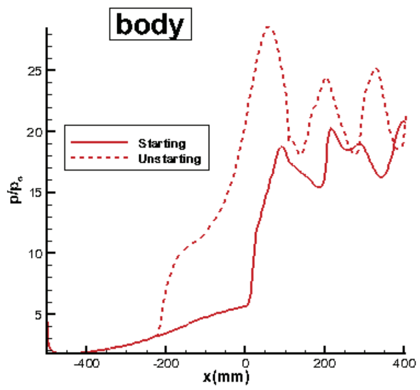
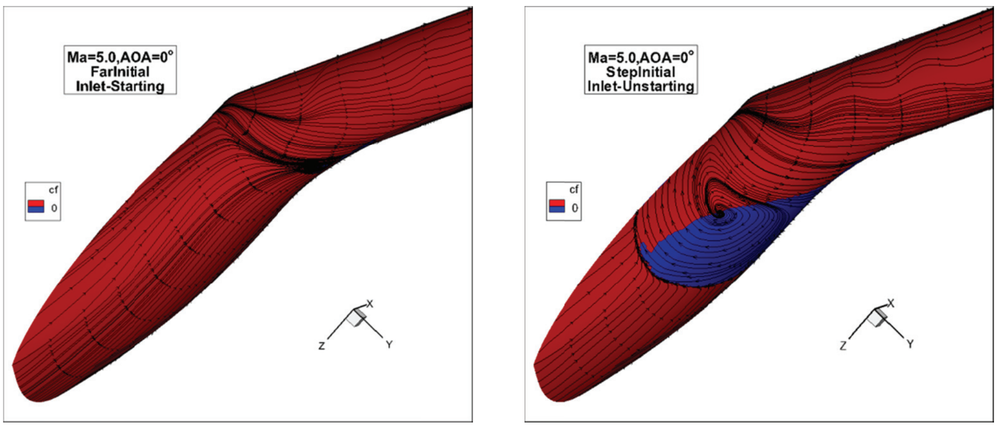


Figure 12 The wall pressure distribution at Ma5

Figure 13 gives the friction lines attached inlet wall of started and unstarted inlet at Ma5. From the figures it can be seen that there are apparent transverse flow and reverse flow on the external compressional surface when the inward-turning inlet is unstarted. It's obvious that the transverse flow and reverse flow are the proofs of large separation for an unstarted inlet. So if flow visualization, for example, tracer particles, silk-threads, oil-flow or pressure-sensitive paint, etc. is used on external compressional surface, it is sure that whether the inward-turning inlet is started or unstarted can be easily judged. Such an effective way can be put forward to judge the inlet states in wind tunnel [13].



(a) started state (b) unstarted state

Figure 13 The wall friction lines at Ma5

5. 2 Flow control

In order to turn an unstarted inward-turning inlet to restart, flow control [14] with bleeding is launched in the numerical simulation.

Based on the flow structures under unstarted state of the inward-turning inlet at Ma5, some bleeding holes are set in the internal flow wall. This is similar to the actual operation in document [1]. In the current research, a mathematical model in AHL3D fluid software is used to control the flow instead of real perforation [9]. There are three main

DOI: 10.13009/EUCASS2017-55

parameters to set in the software. The first parameter is bleeding zone, that is to say, a certain zone is specified to be a porous zone or a boundary condition of bleeding. The second parameter is orifice ratio (the area of all orifices to the area of the throat section). The third parameter is back-pressure ratio (the back-pressure to the incoming flow pressure). An unstarted flow field at Ma5 is used as initial flow field (Zero-Init). A series of these parameters are researched in the paper.
The results obtained are: (1) The position near the shoulder should be chose to be bleeding zone, otherwise the inlet is still difficult to restart. (2) An orifice ratio of 42% is enough for the inlet to restart. (3) The back-pressure ratio should be no greater than 8 to ensure the bleeding effect.

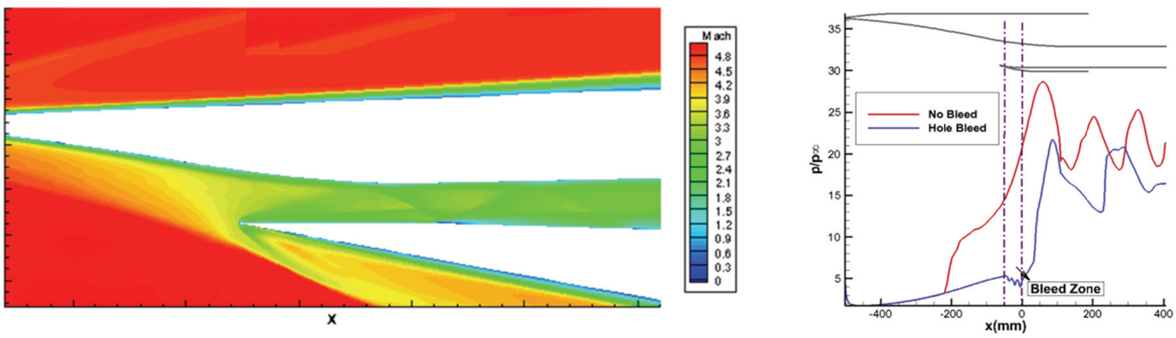


Figure 14 The flow field after flow controlling at Ma5

Figure 14 gives the result of the inlet after flow control at Ma5. Here, the bleeding zone is near the shoulder of the inlet, the orifice ratio is about 42%, and the back-pressure ratio is 6. From the figure, it can be seen that the unstarted inlet has return to started state, and that the bleeding model with the bleeding boundary is very convenient to control the flow field, and that the inward-turning inlet can be restarted by bleeding.

6 The preliminary experiment

A model of the same inlet is manufactured and some experiments are launched to test the starting characteristics and the control effection.

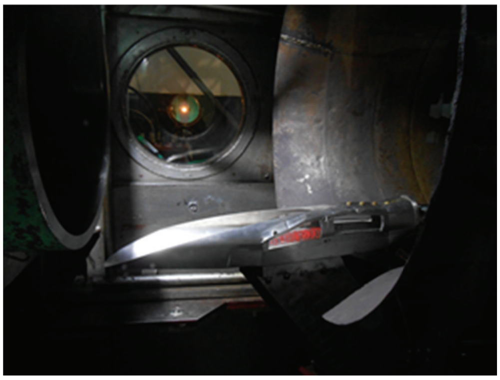
6. 1 The wind tunnel

The experiments have been done in the blow-down hypersonic wind tunnel. The nominal Mach Number of the wind tunnel is 5. The nominal total pressure at the nozzle exit is 10atm. The nominal total temperature at the nozzle exit is 360K.
Two high speed cameras are used, one(marked as Camera1) for recording external compressional shock schlieren photos, with a frequency of 1000 frame/s, the other(marked as Camera2) for recording external compressional surface silk-threads to monitor flow patterns, with a frequency of 1000frame/s.

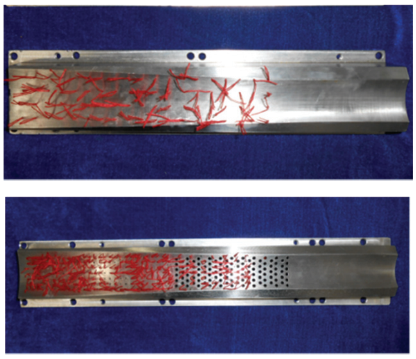
6. 2 The test model

The test model is the same as the one in the numerical simulation above. The proportion of the test model to numerical model is 1:1. Figure 15 gives the photo of the inlet model(see Figure 15a) and the replacement parts of the inlet(see Figure 15b) for the flow visualization (with silk threads) and flow control (with bleeding orifices). The threads (see the top figure of Figure 15b) planted in the replacement parts are red silk threads that can work on a temperature more than 500K. The diameter of the silk threads is about 0.1mm, the length of the silk threads extending from the surface of the inlet is about 10mm. And the interval distance between silk threads is about 5mm. The bleeding orifices (see the bottom figure of Figure 15b) drilled into the replacement parts are round orifices, all with a diameter of 3mm. These orifices open to the inlet flow path and all connect to a common cavity(in the inlet model) which is connecting to the outside of the inlet mode. The number of these orifices are nearly 120, and the radio of the total area of these orifices to the area of the inlet throat is about 42%, which is the same as that in the previous numerical research.
The two high-speed cameras are installed outside the test segment and all face to the model. Camera1 faces to the meridian plane of the model (i.e., from front to back), and Camera2 faces to the external compressional surface of the model (i.e., from top to bottom).

DOI: 10.13009/EUCASS2017-55



(a)The test mode in the wind tunnel



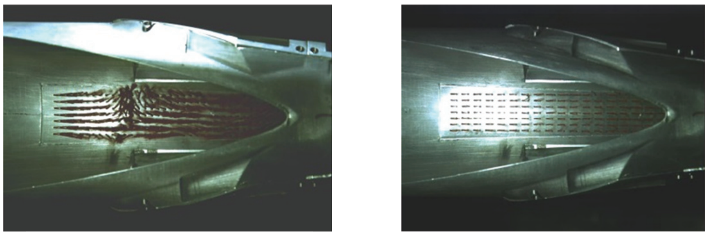
(b)Silk threads and bleeding orifices

Figure 15 The test model and the replacement parts

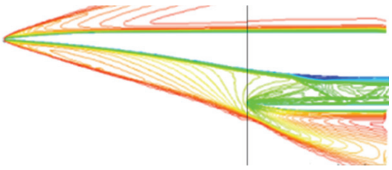
6. 3 The test results

The main result obtained from the experiment is shown in Figure 16.

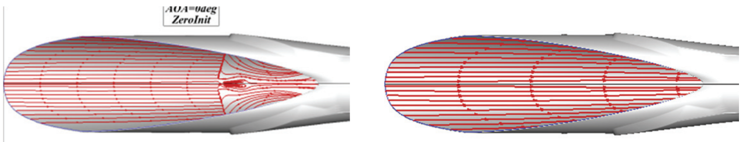
(a)The schlieren photo of the external wave



(b)The silk thread photos(Left: unstarted, Right: started)



(c)The contours of Mach Numbers



(d)The streamlines along the surface(Left: unstarted, Right: started)

Figure 16 The typical photos from the test and the some results from CFD

Figure 16 (a) is a typical schlieren photo of the external shock system which is recorded by Camera1. Figure 16 (b) are two typical silk-threads photos that the trace of the silk-threads are recorded by Camera2. The left photo is corresponding to the unstarted state of the inlet without bleeding, while the right photo is corresponding to the started state of the inlet with bleeding. In Figure 16c and Figure 16d, some results from CFD are given for the comparison of the CFD and the experiments.
From Figure 16 (a), it can be observed that the external shock from the external compressional leading edge and the external shock from cowl leading edge can be seen clearly. But it is difficult to judge whether the inlet is started or not. From the figures, it can be seen that when the inlet is unstarted, the silk-threads near the inlet shoulder are a chaos, which means the streamlines go transverse or reverse(see Figure 16b), and that when the inlet is started, the silk-threads near the inlet shoulder all extend downstream, which means the streamlines near the wall go downstream too(see Figure 16c). Obviously, it is not difficult at all to judge the state of the inward-turning inlet by observing the silk-threads before the shoulder of the inlet. Also, from the comparison of the results from CFD(see Figure 16c and Figure 16d) and the result from the experiments(see Figure 16a and Figure 16b) it can be seen that the forma results agree to the latter very well.

DOI: 10.13009/EUCASS2017-55

7 Conclusion

An inward-turning inlet is designed and its aero-performances are researched in this paper. Through the research, some conclusions can be drawn as follows.
(1) In the design, some special handlings considered include blunting the leading edge for a basic flow field, blending the sharp corner aerodynamically, and transiting the flow tube surface to satisfy the overall need. All these handlings can increase the aero-performances of inward-turning inlets.
(2) The starting and self-starting characteristics are the key techniques for inward-turning inlets. Far-Init and Zero-Init numerical techniques can get the starting and self-starting characteristics of the inward-turning inlet. In order to judge the state of the inlet, such methods should be used as the pressure distribution, internal cross section aero-parameters, as well as wall tracing method.
(3) Bleeding in the shoulder of the inlet is an appropriate and convenient way to control the flow field to improve the starting and self-starting characteristics of the inward-turning inlet.
(4) The test results in the blow-down hypersonic wind tunnel show that the silk-threads method is a good way to judge the state of the inward-turning inlet. The results also show that the flow control is effective and that the design method and the numerical simulation of the inward-turning inlet are soundly reliable.

References

[1] Michael K. Smart and Carl A. Trexler，Mach 4 Performance of a Fixed-Geometry Hypersonic Inlet with Rectangular-to-Elliptical Shape Transition[C]. 41st Aerospace Sciences Meeting and Exhibit AIAA2003-0012, 2003.
[2] Capt. Barry A. Croker, ON THE DESIGN OF HYPERSONIC INWARD-TURNING INLETS, AFRL-RB-WP-TP-2009-3016
[3] Daniel E. F. Barkmeyer, Ryan P. Starkey, and Mark J. Lewis. Inverse Waverider Design for Inward Turning Inlets, AIAA 2005-3915
[4] Li Yongzhou, Zhang Kunyuan, Nan Xiangjun, Design of Hypersonic inward turning inlets with controllable Mach number distribution, AIAA 2012-4151
[5] Sannu Molder，Internal,Axisymmetric, Conical Flow，AIAA Journal,VOL.5,NO.7,1967.07
[6] Lance S. Jacobsen，Chung-Jen Tam，Robert Behdadnia，Frederick S. Billig，Starting and Operation of a Streamline-Traced Busemann Inlet at Mach 4[C].42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, AIAA 2006-4508, 2006
[7] Wei Fen, He Xuzhao, He Yuanyuan, Wu Yingchuan, the Design Method of Double Shock Basic Flow Field for Three-dimensional Inward-turning Inlet. Journal of Propulsion Technology(in Chinese)，2015.03
[8] Faure J. Malo-Molina, Datta V. Gaitonde, Paul H. Kutschenreuter. Numerical Investigation of an Innovative Inward Turning Inlet. AIAA 2005-4871
[9] Dawei Yang, Anyuan YU, ant etc., RESEARCH ON THE SELF-START ABILITY OF A HYPERSONIC INWARD TURNING INLET WITH BLEEDING[C], The 8th National Hypersonic Science Conference(in Chinese) , CSTAM2015-A35-B0129, 2015.12
[10] John D.Mach, Ensign, USNR. An Investigation Of Starting Techniques For Inward Turning Inlets At Flight Speeds Below The On-Design Mach Number. Department of the air force air university. AFIT/GAE/ENY/05-J07
[11] Travis W. Drayna, Ioannis Nompelis, and Graham V. Candler. Hypersonic Inward Turning Inlets: Design and Optimization, AIAA 2006-297
[12] Anyuan YU, Dawei YANG, Jie WU, Hongli NI, Jialing LE. The Numerical Simulation Techniques Research and Preliminary Experimental Validation of the Start Characteristics for a Two-Dimensional Hypersonic Inlet.6TH EUCASS, 2015.07
[13] Li Zhufei, Huang Rong, Guo Shuaitao, Zhan Dongwen, Wu Yingchuan, Yu Anyuan, Yang Jiming,. Application Of Surface Tuft Flow Visualization In Hypersonic Inward Turining Inlet. Journal of Propulsion Technology(in Chinese) (Has Been Recorded to Be Published in 2016.05).
[14] W. Flaherty and J.M. Austin, Hypervelocity Boundary Layer Studies for Inward Turning Inlets, AIAA 2013-0017