The Effects of Compression Direction on the Performance of a Two-Dimensional Inlet Mounted on the Aft Body

Jingfan Chen, Xiaqiang Fan * and Bing Xiong

Hypersonic Technology Laboratory, National University of Defense Technology, Changsha 410003, China; chenjingfan14@nudt.edu.cn (J.C.); xiongbing10@nudt.edu.cn (B.X.)
* Correspondence: xqfan@nudt.edu.cn

Abstract: The aft-body mounted inlet of a hypersonic vehicle has garnered attention for its potential to shorten the propulsion system. This study aims to explore the influence of compression direction on the performance of the hypersonic inlet attached to the vehicle’s aft body. To achieve this objective, a simplified, integrated model of the body and two-dimensional inlet was developed and evaluated using numerical simulation techniques. The study conducted a comparative analysis of the overall and starting performance between inverted and normal inlet layouts while ensuring uniformity in inlet configuration and installation location. The results indicated that the inverted layout surpassed the normal layout in terms of airflow capture capabilities, with an 8.24% higher mass flow rate. However, the inverted inlet layout exhibited an 11.46% reduction in total pressure recovery performance compared to the normal layout. Additionally, the study found that the inverted inlet layout demonstrated a self-starting Mach number 1.62 lower than that of the normal inlet layout. This difference stemmed primarily from the pressure gradient on the body surface induced by the incident shock wave of the inverted inlet, which enhanced starting performance by eliminating low-energy flow near the wall.

Keywords: inlet layout; boundary layer; shock boundary layer interference; aerodynamic design

1. Introduction

Designing hypersonic aircraft is inherently complex, owing to a multitude of factors, including mission diversity, power discrepancies, and variations in engine layout [1]. Consequently, designing the inlet layout presents a significant challenge. The design process necessitates a comprehensive consideration of various factors, such as external airflow, airframe structure, and propulsion system requirements [2], to ensure optimal and consistent inlet performance across diverse mission profiles. In recent years, numerous aerodynamic configurations have emerged for high-speed aircraft, some sharing similar characteristics. Notably, hypersonic aircraft like the Quarterhorse by Hermeus [3] and the SR-72 by Lockheed Martin [4] feature the ventral inlet mounted on the aft body of the hypersonic aircraft fuselage.

The ventral inlet has garnered popularity in hypersonic vehicle design. This layout not only enhances the hypersonic vehicle’s lift-to-drag ratio and pitch balance but also facilitates precompression of incoming airflow by utilizing the forebody to provide denser airflow for the engine [5]. Extensively studied by researchers globally, this layout has been investigated in various contexts, such as the X-43 [6] and X-51 [7] hypersonic verification vehicles in the United States. Furthermore, Gollan et al. [8] integrated the (REST) inlet design method into the integrated design of inward turning inlet and vehicle fuselage. You [9] employed streamlined tracing techniques to design multiple 3D integrated configurations of inward-turning inlet and fuselage, conducting comprehensive comparative analyses of single- and dual-module inlets. Xiong [10] also proposed a novel method for designing a hypersonic inlet, analyzing three inlet layouts for an axisymmetric vehicle and delving...
into the performance characteristics of each layout and its impact on the vehicle's overall aerodynamic properties.

However, current research predominantly concentrates on the inlet mounted on the front and middle of the vehicle, with insufficient attention to aft-body inlet layouts. Xu [11] highlighted the significant advantages of aft-body inlet layouts, including the substantial reduction in engine length and efficient utilization of fuselage internal space, for achieving compact and efficient designs. Nonetheless, this type of layout challenges engineers, particularly regarding the thick boundary layer and the non-uniformity airflow captured by the inlet. Wind tunnel tests conducted by Lawing [12] revealed that under certain flight conditions, the boundary layer thickness at the inlet entrance of the aft-body inlet may reach 30%-50% of the inlet height. Lewis [13] utilized mixed compressible flow theory to analyze the impact of non-uniform flow generated by the attached surface layer on inlet and combustion chamber performance, underscoring the significant effect even a thin attached surface layer can have on free-flow characteristics. Pan [14] investigated the effect of non-uniform incoming airflow on side-pressure inlet. The result demonstrated that non-uniform incoming flow significantly reduces the starting performance and total pressure recovery of the side-compression inlet.

However, the influence of compression direction on the performance of an inlet mounted on the aft body lacks extensive research. To address this gap, this study investigates the specific effects of compression direction on the performance of a 2D inlet. A simplified, integrated model of the body and 2D inlet, customized for this layout, will be developed. Numerical simulations will be employed to comprehensively analyze the significant impact of compression direction. Ultimately, this study aims to provide valuable insights into the design and optimization of inlet systems for hypersonic vehicles with aft-body inlet layouts.

2. Computation Method and Case Validation

2.1. Computation Method

This study employs numerical computation as the primary research method, extensively utilizing commercial computational fluid dynamics (CFD) software Ansys Fluent 2021 for numerical simulation. The software utilizes the finite volume method (FVM) to discretize the Navier–Stokes (N-S) equations [15], providing a robust foundation for subsequent viscous flow analysis. The FVM is employed to discretize the Navier–Stokes equations, transforming the continuous equations into discrete algebraic equations suitable for numerical solution. The density-based solver in Ansys Fluent [16] is used for the simulations, which is particularly well suited for high-speed compressible flows such as those encountered in hypersonic flight. The Roe-FDS flux difference splitting method is used for flux calculation. This method provides robust and accurate flux evaluation for shock capturing and high-speed flow simulations. A second-order upwind scheme is used for spatial discretization, which improves the accuracy of the solution by considering the gradients of the variables. The least squares cell-based method is employed for gradient calculation, enhancing the accuracy of gradient reconstruction in the computational domain. In terms of turbulence models, the present study selects the shear stress transport (SST) model, coupled with the Sutherland formula, for precise calculation of fluid viscosity. To address the high-temperature effects generated during hypersonic flight, the research employs a thermally complete gas model with variable specific heat ratio to more accurately reflect gas properties in real flight environments, thereby enhancing the simulation’s realism and reliability. This computation method has been validated by Xiong [17].

In terms of boundary condition settings, the wall is defined as adiabatic with no-slip conditions to ensure no relative motion between the fluid and the wall while prohibiting heat exchange between the wall and the environment. Pressure far-field boundary conditions are applied at the domain’s inlet or far-field, with values precisely determined based on simulated flight altitude and conditions. Pressure outlet boundary conditions
are implemented at the computational domain’s exit to simulate a realistic exit pressure environment. For models with symmetry planes, in the absence of considering side slip angles, a half-mode mesh is utilized, and symmetry plane boundary conditions are set. This measure effectively reduces computational volume and enhances research efficiency.

2.2. Case Validation

The case validation employs a model consisting of a symmetrical double wedge positioned behind a flat plate, as depicted in Figure 1. This experiment was conducted by Kussoy [18]. The dimensions of the flat plate are 2200 mm in length and 760 mm in width, designed to provide a stable inflow boundary layer for the rear double wedge. The double wedge is configured with two adjoining 15° angles, forming a distinctive geometry with a leading edge spacing of 152 mm, gradually decreasing along the flow direction to 430 mm at the trailing edge. Additionally, each wedge unit has a length and height of 406 mm and 200 mm, respectively.

![Geometry of the three-dimensional double wedge](image_url)

Figure 1. Geometry of the three-dimensional double wedge [18].

To comprehensively evaluate the performance of the 3D symmetric double wedge in the wind tunnel, computational freestream parameters were established based on experimental conditions: an incoming Mach number of 8.28, static pressure set at 430 Pa, and static temperature maintained at 80 K, ensuring a unit Reynolds number of $5.3 \times 10^6$. Regarding computational methodology, the aforementioned numerical scheme was employed, and specific computational grids were utilized. Concerning boundary conditions, the following details were specified: the outer boundary was subjected to pressure far-field conditions, the wedge and the flat plate were designated as solid-wall boundaries, pressure outlet conditions were applied at the exit, and the remaining portions were set as symmetric surface boundary conditions. Further details regarding the boundary conditions can be found in Figure 2. This study adopts boundary condition settings similar to those used by Chen [19], which are widely recognized and validated in the field of hypersonic flow simulations. These settings are selected based on their proven effectiveness in accurately modeling the thermal and flow characteristics of hypersonic vehicles as documented in previous research and standard CFD practices.
Firstly, the parametric design methodology for the simplified, integrated model is shown in Figure 4. Drawing from practical experience, enhancing the two-dimensional characteristics of the under-surface of the hypersonic aircraft can contribute to improving its wide speed adaptability. It is noteworthy that this paper primarily focuses on the inlet performance and does not delve into the overall aerodynamic forces and load analysis of the vehicle. In the case of a hypersonic vehicle equipped with a ventral inlet, the impact of its leeward surface is deemed negligible. The forebody can be treated as a flat plate with a specific length $L_1$ and half-width $W_1$. The length $L_1$ of the forebody is a key indicator of the install location of the inlet. In order to prevent the detached shock waves at the tip of the forebody, the model head is sharpened, and a specific angle $\theta$ is given.

Figure 2. Schematic diagram of computational grid and boundary conditions for the model.

By comparing the pressure experiment data with the numerical simulation results of the centerline of the flat plate, a strong agreement between the two is observed. The detailed comparison relationship is illustrated in Figure 3, further validating the capability of the numerical method employed in this study to accurately simulate the interactions between shock waves and the complex flow phenomena within the boundary layer.

Figure 3. Comparison of flat plate centerline pressure [18].

3. Results and Discussion
3.1. Design and Performance Evaluation Methods

Firstly, the parametric design methodology for the simplified, integrated model is shown in Figure 4. Drawing from practical experience, enhancing the two-dimensional characteristics of the under-surface of the hypersonic aircraft can contribute to improving its wide speed adaptability. It is noteworthy that this paper primarily focuses on the inlet performance and does not delve into the overall aerodynamic forces and load analysis of the vehicle. In the case of a hypersonic vehicle equipped with a ventral inlet, the impact of its leeward surface is deemed negligible. The forebody can be treated as a flat plate with a specific length $L_1$ and half-width $W_1$. The length $L_1$ of the forebody is a key indicator of the install location of the inlet. In order to prevent the detached shock waves at the tip of the forebody, the model head is sharpened, and a specific angle $\theta$ is given.
In the design of the inlet, a mixed-compress 2D inlet and an isostraight isolator was selected. This design incorporates various key parameters, including the external compression angle \( \alpha \), inlet height \( H \), width \( W_2 \), inlet length \( L_2 \), and throat height \( h \). The isolator is configured as an isostraight shape, aligned with the inlet exit, with a designated length \( L_3 \). The external compression angle \( \alpha \) significantly impacts the compression strength of the inlet. The shock-on-lip condition and reflected shock-on-shoulder are adopted to achieve a better total pressure recovery. An oblique excitation relationship is utilized to derive the intrinsic connection among the design Mach number \( Ma \), inlet compression angle \( \alpha \), inlet height \( H \), inlet length \( L_2 \), isolator length \( L_3 \), and throat height \( h \), as depicted in Equation (1).

\[
\begin{align*}
\tan(\alpha - \beta_1) &= \frac{1}{Ma^2} \cdot \frac{\sin(2\beta_1)}{\cos^2(\beta_1) - Ma^2} \\
ma_2 &= \sqrt{\frac{2+(\gamma-1)Ma^2\sin^2\beta_1}{2\gamma Ma^2\sin^2\beta_1 - (\gamma-1)}} \cdot \frac{1}{\cos(\beta_1 - \beta_2)} \\
\tan(\alpha - \beta_2) &= \frac{1}{ma_2^2} \cdot \frac{\sin(2\beta_1)}{\cos^2(\beta_1) - ma_2^2} \\
\frac{H}{\tan(\beta_1)} + \frac{h}{\tan(\beta_2 - \alpha)} &= \frac{H-h}{\tan \alpha} \\
L_2 &= H - h \\
L_3 &= 6 \cdot \frac{h}{\tan \alpha}
\end{align*}
\]  

where \( \beta_1 \) and \( \beta_2 \) represent the incident excitation wave and throat excitation wave excitation angles, respectively, \( \gamma \) is the specific heat ratio, and \( ma_2 \) represents the post-incident shock wave Mach number.

To evaluate the overall performance of the inlet, a 3D unstructured mesh was established for numerical simulation. The boundary conditions and grid configuration are illustrated in Figure 5. The height of the first-layer near-wall grid was set to 0.1 mm.

![Figure 4. Parameterized method of the integrated model.](image)

Figure 4. Parameterized method of the integrated model.

![Figure 5. Schematic diagram of boundary conditions and 3D mesh CFD model.](image)
The mass flow coefficient $\phi$ is introduced as an important parameter to evaluate the airflow capture capability of the inlet. This parameter has a direct impact on the engine thrust. The computing formula of the mass flow coefficient is established as Equation (2). Specifically, where $m_{\text{air}}$ is the actual captured mass flow rate of the inlet, $\rho$ is the incoming flow density, $u$ is the incoming flow velocity, and $A_{\text{FCT}}$ is the maximum captured area of the inlet discussed above.

$$\phi = \frac{m_{\text{air}}}{\rho u A_{\text{FCT}}} \tag{2}$$

In order to quantify the energy loss in the airflow compression process, the total pressure recovery coefficient $\sigma_{\text{inlet}}$ is introduced. It is defined as the ratio of the total pressure at the throat of the inlet to the total pressure of the free-flowing stream and is calculated as shown in Equation (3), where $p_{0t}$ and $p_{0\infty}$ is the total pressures at the throat of the inlet and the free-flowing stream, respectively.

$$\sigma_{\text{inlet}} = \frac{p_{0t}}{p_{0\infty}} \tag{3}$$

Three mesh scales of the inlet were generated, with a total grid number of 0.46, 1.03, and 2.11 million. Table 1 lists the inlet throat flow coefficients $\phi$ and mass-averaged total pressure recovery coefficients $\sigma_{\text{inlet}}$ for the three grid sizes. The total pressure recovery coefficients and the inlet throat flow coefficients of the fine grids are observed to be 0.75% and 0.82% higher, respectively, than those of the medium grids.

<table>
<thead>
<tr>
<th>Mesh Scale</th>
<th>Cell Count (Million)</th>
<th>$\sigma_{\text{inlet}}$</th>
<th>$\phi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>0.46</td>
<td>0.5071</td>
<td>0.8563</td>
</tr>
<tr>
<td>Medium</td>
<td>1.03</td>
<td>0.5442</td>
<td>0.8831</td>
</tr>
<tr>
<td>Fine</td>
<td>2.11</td>
<td>0.5483</td>
<td>0.8903</td>
</tr>
</tbody>
</table>

The surface pressure distributions on the symmetry plane of the three grids are compared in Figure 6. The results reveal that the pressure distributions with medium and fine grids are almost identical. Therefore, the reasonable grid size for the CFD model is approximately 1.03 million.

Figure 6. Surface pressure distributions along the bottom wall.

3.2. Effect of Compression Direction on Overall Performance of Inlet

In hypersonic vehicle design, the compression direction of the inlet significantly affects internal airflow characteristics and total pressure recovery, particularly as the airflow interacts with the vehicle body. To simulate the inlet mounted on the aft body, the inlet was
positioned at \( L/H = 12 \). The design parameters for the 2D inlet mentioned are a height \( H \) of 100mm, a half-width \( W_2 \) of 100mm, and a compression angle of 12°.

In this paper, two compression directions are chosen. They are normal and inverted layouts, respectively. Normal layouts typically direct airflow away from the vehicle body upon compression, while inverted layouts bring the compressed airflow closer to the body. For specific comparisons of the external designs of both layouts, please refer to the accompanying Figure 7. Through numerical simulation and comparative analysis, significant differences in inlet performance between normal and inverted layouts emerged. Specifically, the inverted layout exhibited an approximately 8.24% increase in flow coefficient \( \phi \) compared to the normal layout. However, it is worth noting that the inverted layout demonstrated inferior total pressure recovery \( \sigma_{inlet} \) performance, with a reduction of 11.46% compared to the normal layout. Table 2 provides a detailed description. Therefore, a comprehensive consideration of the balance between flow coefficient rate and total pressure recovery is crucial in determining the inlet compression direction during the design process.

![Comparision of normal and inverted inlet layout](image)

**Figure 7.** Comparison of normal and inverted inlet layout.

**Table 2.** Overall performance comparison.

<table>
<thead>
<tr>
<th>Type of Inlet</th>
<th>( \phi )</th>
<th>( \sigma_{inlet} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal inlet</td>
<td>0.818</td>
<td>0.543</td>
</tr>
<tr>
<td>Inverted inlet</td>
<td>0.886</td>
<td>0.481</td>
</tr>
<tr>
<td>Difference (%)</td>
<td>8.24%</td>
<td>−11.46%</td>
</tr>
</tbody>
</table>

An in-depth study of the symmetry plane contours in Figure 8 reveals significant differences in the flow field structure between the inverted and normal layouts. Specifically, the streamlines in the direction of airflow compression vary between the two layouts: In the normal inlet layout, the streamlines noticeably incline towards the fuselage side. Additionally, the forms of incident shocks differ significantly between these two inlet configurations. In the normal inlet layout, low-energy flow tends to enter the external compression ramp. Due to the outward shift of the equivalent wall formed by the boundary layer, the incident shock generated in the external compression ramp also shifts outward, away from the lip, which may lead to flow spillage and affect inlet performance. In contrast, in the inverted layout, the boundary layer resides on the fuselage side, keeping the external compression ramp unaffected. This allows the oblique shock to maintain a stable shape, effectively eliminating spillage issues, thereby enabling the inlet to more efficiently capture and guide airflow. The compression surface in the normal inlet layout is covered by the boundary layer, resulting in a lower airflow Mach number sensed by the compression surface compared to the actual incoming Mach number. Typically, for the same inlet, the lower the incoming Mach number, the higher the total pressure recovery coefficient. This phenomenon is particularly pronounced in the normal inlet layout and requires careful consideration in the design process. It is also observed that streamlines originating from the same distant position are captured within the inlet in the inverted intake layout, whereas they overflow from the lip in the normal inlet layout. This results in a lower flow rate for the normal inlet layout compared to the inverted inlet layout.
Figure 8. Comparison of Mach contours and streamlines on symmetry.

Furthermore, the inlet layout significantly influences the flow structure at the throat. To investigate the differences in flow characteristics between the normal and inverted inlet layouts, we meticulously plotted the flow field Mach number contours for a comparative analysis, as shown in Figure 9. In Figure 9a,b, the left side shows a normal inlet layout, and the right side shows an inverted intake layout. The analysis reveals that in the throat region, the mainstream area of the normal inlet layout is broader, while the low-speed region near its lower surface is relatively smaller, resulting in a more uniform overall velocity distribution. However, from the total pressure contours, it is observed that although the mainstream total pressure absolute value is higher in the normal inlet layout, its mainstream position is relatively lower compared to the outlet center. Notably, in the nonmainstream region of the normal inlet layout, the interaction between the lip shock and the boundary layer intensifies, leading to significant flow distortion and energy loss. This phenomenon is particularly severe in the corner region.

Figure 9. Comparison of flow regimes at the throat of the inlet for different inlet layouts.

3.3. Effect of Compression Direction on Start Performance of Inlet

Additionally, the self-starting performance of a hypersonic inlet is crucial for ensuring the stable operation of hypersonic vehicles. The self-start Mach number reflects the inlet's ability to transition from an unstart state to a started state. This section aims to reveal the differences in starting characteristics between normal and inverted layouts. To achieve this, we compared the acceleration self-starting Mach numbers of these two inlet layouts at a 0° angle of attack. Initially, a low Mach number of three was set to establish an unstart
state for the inlet. Subsequently, the inflow Mach number was incrementally increased by ∆Ma = 0.02 until the inlet successfully transitioned to a started state. Figures 10 and 11 detail the specific self-starting processes of the two layouts, indicating that the normal and inverted configurations achieve self-starting at Mach numbers of 5.52 and 3.90, respectively. Notably, the self-starting Mach number of the inverted inlet layout is lower, indicating its superior self-starting capability.

Figure 10. Self-starting process of the normal inlet layout.

Figure 11. Self-starting process of the inverted inlet layout.

Figure 12 also describes the changes in the flow coefficient and total pressure recovery at the throat of the inlet during acceleration. Although the flow rate did not change significantly from Mach 5.50 to 5.52, there was a sudden change in the total pressure recovery coefficient. Similarly, for the normal inlet layout, while the flow rate did not change significantly from Mach 5.50 to 5.52, there was a sudden change in the total pressure recovery coefficient. This also confirms that the normal and inverted configurations achieve self-starting at Mach numbers of 5.52 and 3.90, respectively.

(a) Normal inlet layout

(b) Inverted inlet layout

Figure 12. Comparison of the variation curve of aerodynamic parameters during the starting process.
Figure 13 depicts the non-starting flow field states of the normal and inverted inlet layouts at Ma = 3.88. Diverging blue and red color maps illustrate pressure distribution on the fuselage surface, while a smaller rainbow-colored map indicates Mach number distribution on the symmetry plane. Streamlines near the wall surface and lip are also depicted in Figure 13a,b. In the normal inlet layout, the spanwise wall pressure distribution on the fuselage side is relatively uniform, with no pressure gradient, hindering the effective sweep of low-energy flow. The lip shock wave forms a separation zone in the internal contraction section, obstructing the overflow of low-energy flow and making self-starting challenging. Conversely, the inverted inlet layout exhibits unique advantages. Here, the incident internal cone shock wave is projected onto the fuselage, forming a local high-pressure region. Together with low-pressure regions on both sides unaffected by the shock wave, a significant transverse pressure gradient is formed, effectively discharging low-energy flow near the wall through overflow windows, reducing the scale of the separation region and enhancing self-starting performance.

![Figure 13](image)

**Figure 13.** Comparison of unstart flow regimes for different inlet layouts (Ma = 3.88).

We have conducted an analysis to assess the effect of spillage on Mach number (Ma) and pressure recovery. The results of the calculations, including the mass-weighted average total pressure recovery $\sigma_{inlet}$, mass flow rate $\dot{m}_{air}$, and mass-weighted average Mach number Ma for both normal and inverted inlet layouts, are presented in Table 3 below. From the table, we can observe that the inverted inlet layout achieves a significantly higher mass-weighted average total pressure (0.802) compared to the normal inlet layout (0.296). This suggests that the inverted configuration is more effective in maintaining higher total pressure, which is beneficial for pressure recovery. The inverted inlet layout also shows a higher mass flow rate (0.270 kg/s) than the normal inlet layout (0.155 kg/s). This indicates that the inverted configuration is capable of capturing a greater amount of airflow, which can be attributed to reduced high-energy flow spillage. Both configurations have the same mass-weighted average Mach number (1.66). This implies that the Mach number is unaffected by the configuration.

<table>
<thead>
<tr>
<th>Type of Inlet</th>
<th>$\sigma_{inlet}$</th>
<th>$\dot{m}_{air}$</th>
<th>Ma</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal inlet</td>
<td>0.296</td>
<td>0.155</td>
<td>1.66</td>
</tr>
<tr>
<td>Inverted inlet</td>
<td>0.802</td>
<td>0.270</td>
<td>1.66</td>
</tr>
<tr>
<td>Difference (%)</td>
<td>+171.62%</td>
<td>+74.19%</td>
<td>0%</td>
</tr>
</tbody>
</table>

**Table 3.** Overall inlet unstart performance comparison at Ma = 3.88.
4. Conclusions

This paper provides an in-depth analysis of a simplified model representing a hypersonic vehicle integrated with a 2D inlet. Normal and inverted inlet layouts are examined to assess the impact of the inlet compression direction on the inlet’s performance. The following key conclusions emerge:

(1) The parametrization method proposed in this paper is both simple and efficient. It allows for the parametrization of the inlet model with several parameters, each of which holds significant physical meaning in the design process. It can be utilized to quickly obtain the 3D shape of the model under different geometric parameters without relying on CAD software such as Solidworks 2016 and offers a valuable tool for further exploration and optimization of inlet designs in future studies.

(2) A comparison between normal and inverted inlet layouts demonstrates that the inverted layout surpassed the normal layout in terms of airflow capture capabilities, with an 8.24% higher mass flow rate. The inverted inlet layout exhibited an 11.46% reduction in total pressure recovery performance compared to the normal layout. These insights provide valuable guidance for further choosing inlet layouts.

(3) In the context of the specific conditions outlined in this paper, the self-starting Mach number for the inverted inlet is 1.68 lower than that of the normal inlet. For the inverted inlet layout, the transverse pressure gradient induced by the inlet incident shock wave can result in the low-energy flow near the forebody being diverted in time, preventing the accumulation of the separation zone and causing the non-starting.

Future considerations involve exploring how the interaction of inlet incident shock waves with different intensities and boundary layers with different thicknesses influence the overall aerodynamic and starting performance of the hypersonic inlet.

Author Contributions: Conceptualization, J.C.; Funding acquisition, X.F. and B.X.; Methodology, J.C.; Software, J.C.; Supervision, X.F. and B.X.; Writing—Original draft, J.C.; Writing—Review and editing, B.X. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the National Natural Science Foundation of China (12102470 and 12372298).

Data Availability Statement: The original contributions presented in the study are included in the article, further inquiries can be directed to the corresponding author.

Acknowledgments: The authors thank the reviewers for their valuable advice on this study.

Conflicts of Interest: The authors declare no conflicts of interest.

References


Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.