Numerical investigation of the unsteady behavior of a hypersonic inlet under throttling

Pei-Zhao Sun*, Hai-Tao Shi, Xi-Yun Lu

Department of Modern Mechanics, University of Science and Technology of China, Hefei, Anhui 230026, China

Abstract

Numerical investigation of the unsteady behaviors of the hypersonic inlet caused by throttling was carried out by means of large-eddy simulation for a free stream Mach number 5.9 and the unit Reynolds number approximately $5.2 \times 10^6 \, m^{-1}$. Results have been validated carefully against experimental data. The throttling was caused by the plug placed near the exit of isolator. Two typical cases with and without plug are investigated. Various fundamental mechanisms dictating the flow phenomena inside the isolator have been studied systematically. The results of this study provide physical insight into the flow behaviors caused by the throttling.

Keywords: Large-eddy simulation; inlet; hypersonic flow

1. Introduction

The inlet starting characteristics are vital for the realization of hypersonic flight. The hypersonic air breathing propulsion systems are highly susceptible to the flow-induced instabilities [1]. In the inlet, the incoming air is compressed through a series of shocks before entering the combustion chamber. To isolate the effect of flight conditions on the precombustion shock structure, a nearly parallel walled duct named an isolator is placed between the inlet and the combustor. The isolator contains a time-varying shock train system providing a stable flow to the combustion chamber. When the backpressure generated by the combustion process exceeds a particular value, the entire shock structure moves upstream, leading to engine unstart [2].

Some numerical simulations are carried out to investigate the relevant problem. Koo and Raman [2] performed a large-eddy simulation (LES) of the Mach 5 inlet-isolator system adopted by the experiment [3]. Two cases including a started isolator and transient unstart propagation were considered to investigate the characteristics of the unstart propagation. Ingenito et al. [4] performed a LES of the HyShot combustor system to analyze the effects of mixing and combustion. Cocks et al. [5] performed a detached-eddy simulation (DES) of a supersonic combustor to study the normal shock train. Krishnan et al. [6] taken a LES of a Mach 6 inlet system to reveal the transition mechanisms on the compression ramps.

* Corresponding author. Tel.: +86-551-6360-1237; fax: +86-551-6360-6459.
E-mail address: pzsun@mail.ustc.edu.cn
2. Numerical methods and inflow boundary condition

2.1. Numerical methods

To investigate the supersonic flow inside a isolator, the three-dimensional Favre-filtered compressible Navier-Stokes equations in generalized coordinates are employed. The equation of state for an ideal gas is used and the molecular viscosity is assumed to obey the Sutherland law. To non-dimensionalize the equations, we use the free stream variables including the density \( \rho_\infty \), temperature \( T_\infty \), speed of sound \( a_\infty \), and characteristic scale \( L \) based on the geometry of the configuration.

The LES is implemented in the present work for turbulence closure. Some terms in the Favre-filtered equations arise from unresolved scales and need to be modelled in terms of resolved scales. Then, dynamic subgrid-scale models for compressible flows are employed. A detailed description of the mathematical formulation of the non-dimensionalized equations and the subgrid-scale models can be found in our previous papers [7,8].

2.2. Inflow boundary condition

It is essential to generate the inflow turbulent boundary layer on the bottom wall to keep the inlet from unstarting. The simulation of turbulent boundary layer requires the prescription of three-dimensional, unsteady inflow boundary conditions to get fast transition into a fully turbulent state.

We have implemented the synthetic turbulence approach [9]. Time-dependent perturbations can be introduced into the inlet and superposed onto a mean turbulent boundary layer profiles, which simulate the coherent boundary layer structures. The mean velocity profiles are generated from the results of RANS. The synthetic velocity disturbances in the streamwise \((x)\) and wall-normal \((z)\) directions are specified as follows:

\[
\begin{align*}
  u' (x, y, z, t) &= \sqrt{\frac{\rho_\infty}{\overline{\rho}(x)u_\infty}} \sum_{j=1}^{5} a_j U_j(z) \sin \left[ \omega_j \left( \frac{y}{u_{c_j}} - t \right) \right] \cos \left( 2\pi \frac{y}{L_y} + \phi_j \right) \\
  w' (x, y, z, t) &= \sqrt{\frac{\rho_\infty}{\overline{\rho}(x)u_\infty}} \sum_{j=1}^{5} b_j W_j(z) \sin \left[ \omega_j \left( \frac{y}{u_{c_j}} - t \right) \right] \cos \left( 2\pi \frac{y}{L_y} + \phi_j \right)
\end{align*}
\]

with

\[
U_j(z) = \left( \frac{z}{z_j} \right) e^{-\frac{z}{z_j}}, W_j(z) = \left( \frac{z}{z_j} \right)^2 e^{-\frac{z}{z_j}}
\]

The form of the velocity disturbances in the spanwise direction \((y)\) can be derived from a divergence-free condition and presented as follows:

\[
\partial u'/\partial x + \partial v'/\partial y + \partial w'/\partial z = 0
\]

Some parameter values are listed in table 1. The mode \( j = 1 \) is associated with inner layer streaks and streamwise vortices travelling downstream at a convective velocity of \(10u_\infty\). The streaks have a spanwise spacing of \( \lambda_z = 128 \) and length of \( \lambda_x = 500 \), while the modes \( j = 2, \ldots, 5 \), correspond to large-scale streamwise vortices in the outer units travelling downstream at a velocity about \(0.75u_\infty\).

<table>
<thead>
<tr>
<th>( j )</th>
<th>( z_j )</th>
<th>( a_j )</th>
<th>( b_j )</th>
<th>( \omega_j )</th>
<th>( u_{c_j} )</th>
<th>( \lambda_j )</th>
<th>( \phi_j )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12.06 ( \delta_y )</td>
<td>1.33</td>
<td>-0.25</td>
<td>0.108a_\infty/\delta_y</td>
<td>10u_\infty</td>
<td>128\delta_y</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>0.296\delta_y</td>
<td>0.33</td>
<td>-0.07</td>
<td>0.839a_\infty/\delta_y</td>
<td>0.75u_\infty</td>
<td>L_\gamma/2</td>
<td>0.1</td>
</tr>
<tr>
<td>3</td>
<td>0.447\delta_y</td>
<td>0.33</td>
<td>-0.07</td>
<td>0.419a_\infty/\delta_y</td>
<td>0.75u_\infty</td>
<td>L_\gamma/4</td>
<td>0.2</td>
</tr>
<tr>
<td>4</td>
<td>0.596\delta_y</td>
<td>0.33</td>
<td>-0.07</td>
<td>0.210a_\infty/\delta_y</td>
<td>0.75u_\infty</td>
<td>L_\gamma/6</td>
<td>0.3</td>
</tr>
<tr>
<td>5</td>
<td>0.447\delta_y</td>
<td>0.33</td>
<td>-0.07</td>
<td>0.336a_\infty/\delta_y</td>
<td>0.75u_\infty</td>
<td>L_\gamma</td>
<td>0.4</td>
</tr>
</tbody>
</table>
A random perturbation, $f'$, with a maximum amplitude of 4% of the free-stream velocity, is used to break any remaining symmetries in the inflow condition, with $f' = c (r - 0.5) F(z)$, where $r$ is a random number varying between 0 and 1, $c = 0.08u_\infty$, and

$$F(z) = \begin{cases} 
\left(\frac{z}{\hat{z}_1}\right)^2, & z < \hat{z}_1 \\
\exp\left(-\frac{(z - \delta_0)/\delta_{vw}}{2}\right), & z > \delta_0 \\
1, & \text{otherwise}
\end{cases}$$

(4)

3. Computational overview and validation

3.1. Computational overview

An hypersonic inlet configuration is considered here with free-stream Mach number $M_\infty = 5.9$ [10]. The total pressure and total temperature at the inflow condition were $P_0 = 1.27MPa$ and $T_0 = 810K$, respectively, which correspond to a unit Reynolds number of approximately $5.2 \times 10^6 m^{-1}$.

The inlet/isolator model in the experiment was a generic two-dimensional mixed compression system with an ICR (internal contraction ratio) of 1.53 that was able to self-start in the current flow conditions. As shown in Fig. 1, the system consists of two compression ramps, a horizontal cowl, and a constant cross-sectional area isolator. A flow plug with a wedge angle of 20$^\circ$ and a length of 40mm was placed near the isolator exit to simulate the high backpressure induced by combustion.

The throttling ratio ($TR$) is defined as $TR = 1 - A_{th,plug}/A_{isolator}$, where $A_{th,plug}$ is the geometrical throat area at the plug and $A_{isolator}$ is the cross-section area of the isolator. Two $TR$ values are chosen as 0 and 0.3. The inlet keeps start at the two $TR$ values but has quite different characteristics because of the influence of the plug.

The flow domain is discretized using equally spaced grids along the streamwise ($x$) and spanwise ($y$) directions and a stretched grid in the wall-normal direction ($z$), so that there are enough points near the two walls to capture the boundary layer information. The characteristic scale $L$ is 120mm. As the width of the isolator is quintuple its height and confined by sidewalls in the experiment, thus twice the isolator’s height in the present computation. The isolator is assumed to be periodic in the spanwise direction. The two ramps are subtracted to save the cost of computation and simplify the application of the synthesis of the incoming turbulent boundary layer. As is shown in Table 2, three cases are considered. Case 1 and 2 have no plug with different grid resolutions to understand the influence of the grid resolution. Case 3 has plug and adopts the same grid resolution with case 2.

<table>
<thead>
<tr>
<th>Case</th>
<th>Grid ($N_x \times N_y \times N_z$)</th>
<th>$a_\infty \Delta t/D$</th>
<th>$\Delta z^*$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>951 x 81 x 201</td>
<td>0.0005</td>
<td>&lt; 1</td>
</tr>
<tr>
<td>2</td>
<td>647 x 61 x 141</td>
<td>0.0005</td>
<td>&lt; 1</td>
</tr>
<tr>
<td>3</td>
<td>647 x 61 x 141</td>
<td>0.0005</td>
<td>&lt; 1</td>
</tr>
</tbody>
</table>
3.2. Validation

To validate the present simulation, we compare numerical results with experimental data [10] in terms of the mean wall pressure. Fig. 2 shows the distributions of mean wall pressure along the lip side averaged in time and along the spanwise direction. The results compare well with the experimental data. Moreover, validation of the results predicted by different grid resolutions is also performed. It is shown that the results for the two cases are consistent well with the experiment data.

4. Results and discussion

A typical started flow without the plug is shown in Fig. 3 using numerical schlieren-like visualization of mean flow averaged in time and along the spanwise direction. By means of the synthetic turbulence method, the inflow boundary layer is well simulated and the inlet will keep start. As shown in Fig. 3, the main features of the internal flow field consist of a series of shock waves that are reflected between the ceiling and floor of the isolator. The cowl shock impinges near the shoulder of the inlet and causes separation. The induced separation shock and the reflected shock hit the ceiling of the isolator. A small separation bubble is generated by the reflected shock, which causes a weak separation shock and a second reflected shock. The shocks become weaker due to deceleration of the flow after shocks and lower Mach number.

When the plug is added to the isolator, the separation shocks will be caused by the plug. The inlet remains start with $TR = 0.3$ for case 3. The separation structures are pushed upstream by the high backpressure yielded by the plug. To illustrate the unsteady features of the flow, Fig. 4(a) shows the time development of flow structures in the mid-span $(x, z)$ plane by means of the density-gradient magnitude. The separation on the floor of the isolator caused by the plug is stronger than that on the ceiling. The upstream moving of the separation shocks could be observed obviously.

With the upstream moving of the separation flow, the flow field downstream the isolator becomes turbulence. Fig. 4(b) shows the pressure and Mach number histories averaged in the spanwise direction in three positions downstream the isolator. The Mach number on the center line of the isolator decreases with time. Finally the flow downstream the isolator becomes subsonic over a large region. The shock system of the internal flow limits the upstream moving of the separation shocks. The backpressure increases due to the choking effect caused by the throttling. The separation shocks slightly oscillate until the backpressure becomes high enough to break the limitation of the shocks ahead.
5. Concluding remarks

The flow characteristics of a two-dimensional hypersonic inlet/isolator model have been investigated numerically using LES approach. Two TR values are considered and flow features are discussed. The synthetic turbulence approach can keep the separation bubble at the shoulder from moving upstream and keeping the inlet remaining start. When the TR value changes to 0.3, the backpressure will increase and the separation flow caused by the plug will move upstream. The flow field downstream the isolator becomes subsonic.

Acknowledgements

This work was supported by the National Natural Science Foundation of China (Nos. 11132010 and 11072236).

References