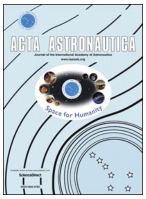
ARTICLE IN PRESS

Acta Astronautica xx (xxxx) xxxx–xxxx

Contents lists available at ScienceDirect

Acta Astronautica

journal homepage: www.elsevier.com/locate/aa



Numerical study on hypersonic nozzle-inlet starting characteristics in a shock tunnel

Xiaoliang Jiao[a,b], Juntao Chang[a,c,*], Zhongqi Wang[a,b], Daren Yu[a,b]

[a] Harbin Institute of Technology, 150001 Heilongjiang, People's Republic of China
[b] School of Energy Science and Engineering, People's Republic of China
[c] Academy of Fundamental and Interdisciplinary Sciences, People's Republic of China

ARTICLE INFO

Keywords:
Hypersonic inlet
Nozzle
Shock tunnel
Pulse-starting
High Mach number
Local unstart

ABSTRACT

An unsteady viscous numerical simulation is performed to study the starting process of a hypersonic nozzle coupled with a simplified inlet model in a shock tunnel under the condition of inflow Mach number higher than the design value. And the effect of the initial backpressure of test section in shock tunnel on the pulse-starting characteristics of the hypersonic nozzle-inlet is studied. It is found that the operation mode of inlet changes from local unstart to start with the initial backpressure decreasing. At high initial backpressure, two large separation bubbles are induced by the secondary shock at the nozzle walls, the separation induced shock waves repeat reflections between the nozzle walls and a shock train structure appears. The Mach number of the flow into the test section rises and falls repeatedly as a result of the presence of the successive shocks in the shock train structure. During the hypersonic nozzle-inlet starting process, a large separation bubble occurs at the forebody side and the inlet is choked. After the forebody shock moving close to the cowl lip, a bow shock is generated and moves before the cowl lip subsequently. Finally, the quasi-steady flowfield of the inlet is established with a bow shock before the cow lip and a subsonic region at the cowl side, namely the inlet is local unstarted. At low initial backpressure, the separation bubbles at the nozzle walls vanish and no shock train structure appears. Furthermore, an unsteady expansion wave may appear upstream of the secondary shock and the Mach number of the flow following the secondary shock is higher than that of the nozzle quasi-steady flow. The Mach number of the flow into the inlet increases to the maximum value quickly, the inlet starts successfully. The numerical results show that the inlet starts easier and faster with lower initial backpressure.

1. Introduction

As the main compression component of a hypersonic propulsion system, the inlet affects the performance of scramjet greatly. The starting problem is the most critical item in the design of the hypersonic inlet. If an inlet is not started, the mass capture will be greatly reduced, and the spillage drag will be excessive. For hypersonic air-breathing engines, inlet unstart will cause a large drop of both engine thrust and specific impulse, and it may cause a catastrophic damage during hypersonic flight [1].

Over the past few decades, the subject of hypersonic inlet start/unstart have been studied extensively [2–16]. Many experimental investigations are carried out in ground test facilities, which are capable of simulating hypervelocity flow. The pulse facility is one of the most popular types of facilities as it offers many advantages. The pulse facility is capable of producing high total pressure and total enthalpy flows required at the hypersonic flight conditions, and the cost of construction and operation are lower than in other types of facilities. It has been shown that the inlet starts easier and faster with lower backpressure of test section in pulse facilities [2,16]. The inlet with larger internal contraction ratio (ICR), which can not be started in general facilities, could be started in pulse facilities [2,17], because of the unsteady effects in flow establishment of the facility [18] which have strong capability of helping inlet to start. The experimental investigations mainly focus on the flow evolution of inlet, while neglect the nozzle. The nozzle starting process in a shock tunnel is extensively studied by numerical simulation [19–24]. The flow in a hypersonic nozzle starts when a strong planar incident shock reaches the nozzle entrance, the shock in front of it enters the nozzle and evolves as the primary shock, while the part in front of the tube end wall is reflected. Test gas, which accelerates through the nozzle throat following the primary shock, expands to a high Mach number. Simultaneously, the secondary shock is generated and the flowfield increases its complexity. At the wall, the boundary layer strongly interacts with the secondary

---

* Corresponding author at: Harbin Institute of Technology, 150001 Heilongjiang, People's Republic of China.
E-mail addresses: jiao_xiaoliang@126.com (X. Jiao), changjuntao@hit.edu.cn (J. Chang), wang_zhongqi@yeah.net (Z. Wang), yudaren@hit.edu.cn (D. Yu).

http://dx.doi.org/10.1016/j.actaastro.2016.10.027
Received 6 August 2016; Received in revised form 12 September 2016; Accepted 22 October 2016

Please cite this article as: Jiao, X., Acta Astronautica (2016), http://dx.doi.org/10.1016/j.actaastro.2016.10.027

Nomenclature

c	speed of sound
D	diameter of the parabolic reflector
k	order of accuracy of numerical scheme
L_1	domain size in the direction of integration
M_s	incident shock Mach number
N_1	number of cells in the direction of integration
n	number of time steps
$n_{\max}$	maximal allowable number of time steps
R_S	reliability of results
S_i	relative error in the direction of integration
S_{err}	relative error in several directions of integration
$S^{\max}$	allowable value of total error
t	time
X	X-axis coordinate
Y	Y-axis coordinate
γ	specific heat ratio
ΔL	cell size in the direction of integration

shock and gives rise to multiple transverse waves. The complex structure of the shock waves evolves in the nozzle. The process, starting with the incident shock wave entering the nozzle until a quasi-steady flow is achieved, is called the starting process of the nozzle.

Hypersonic inlet works in a wide range of Mach numbers and the design Mach number is generally a middle value. So, it may work at a Mach number higher than the design value [25,26]. When hypersonic inlet works at a high Mach number, the forebody shock wave impinges on the cowl at a position downstream of the lip and reflects. The reflection of shock wave result in two types of wave configurations: regular reflection (RR) and Mach reflection (MR) [27]. The hypersonic inlet is started with a regular reflection of the forebody shock wave at the cowl. Whereas a Mach reflection will result in the shock propagating forwards to cause a shock detachment at the cowl lip, which is called “local unstart of inlet” by Jiao et al. [28]. Mahapatra and Jagadeesh [29] experimentally observed the local unstart of a two-dimensional simplified inlet model in a hypersonic shock tunnel. And they studied the effect of cowl length on the pulse-starting characteristics of hypersonic inlet at high Mach number. However, the pulse-starting characteristic of a fixed-geometry hypersonic inlet at high Mach number was not carefully considered. The evolution of flowfield in the nozzle, which directly affects the starting characteristics of hypersonic inlet, is critical. But it is difficult to record in detail by experiment. Wang et al. [30] numerically studied the unsteady starting of a hypersonic inlet with the simplified nozzle startup wave structure, which is modeled with the help of inviscid simulation. Fan [31] studied the starting process of a simple internal-compression inlet couple with nozzle by inviscid numerical simulation. The viscous effect is neglected in these works while it is impractical. During the nozzle starting process, the boundary layer strongly interacts with the secondary shock and gives rise to multiple transverse waves. The complex structure of the shock waves evolves in the nozzle, and the effect of the boundary-layer becomes larger on the transmitted shock, the contact surface, and the secondary shock system.

This paper focuses on the pulse-starting characteristics of a fixed-geometry hypersonic inlet at high Mach number. Unsteady viscous numerical simulation about a hypersonic nozzle coupled with a simplified inlet model pulse-starting in a shock tunnel is performed. And the effect of the initial backpressure of test section in shock tunnel on the pulse-starting characteristics of the hypersonic nozzle-inlet is studied.

2. Numerical methodology

2.1. Shock tunnel configuration

The simulation model in this paper is a two-dimensional shock tunnel as shown in Fig. 1a. A shock tube of 0.1 m height is connected to a convergent-divergent nozzle of 1.28 m length and 0.18 m exit height. A simplified inlet model was placed in the middle of the test section. As shown in Fig. 1b, the model has a ramp angle of 27° and a cowl angle of 11°. A circular arc is used to connect the cowl to the duct. The present nozzle is designed to simulate hypersonic flow of Mach number 7. The static pressure and static temperature of the freestream is 1845 Pa and 223.7 K, respectively, which simulates the flight altitude of 27 km. The freestream Mach number is higher than the inlet design value, and therefore the compression ramp shock impinges on the cowl under the condition.

2.2. Numerical method

The numerical simulation is accomplished by the commercial computational fluid dynamics (CFD) software ANSYS Fluent 14, which is based on a finite-volume method. The two-dimensional compressible Reynolds-averaged Navier-Stokes (RANS) equations are adopted as the governing equations, and the shear stress transport (SST) k-ω two-equation model developed by Menter et al. [32] is employed as the turbulent model because it lead higher accuracy of the simulation of the whole flowfield by treating the high-speed flow in the main stream with the k-epsilon model and treating the low velocity flow in the near wall region with the standard k-omega model. An implicit algorithm with second-order spatial accuracy and dual time-stepping is used for the computation of transient flowfield. The global time step is set as 1e−7 s, which is much lower than the flow timescale of the nozzle startup wave system propagation. The transient flowfield is converged at every global time step. The fluid is modeled as a single-specie, thermally perfect air. The piecewise-polynomial method is selected to compute specific heat ratio while viscosity is solved using Sutherland's formula. The advection upstream splitting method (AUSM) flux vector splitting is applied for the approximation of the convective flux functions. The second order upwind scheme is used to discretize the convective terms. The numerical method has been applied in [33–36].

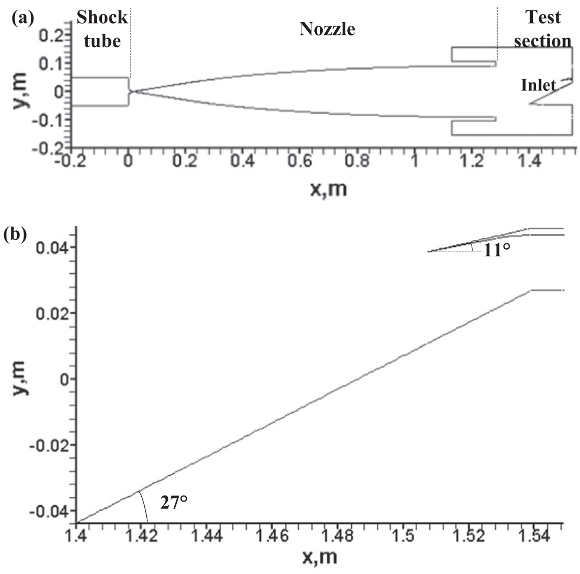


Fig. 1. Schematic of a) shock tunnel and b) inlet model.