Aerospace Science and Technology 56 (2016) 1–13

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Investigation of boundary-layer ejecting for resistance to back pressure in an isolator

Yubao He, Hongyan Huang, Daren Yu *

School of Energy Science and Engineering, Harbin Institute of Technology, Harbin, 150001, PR China

A R T I C L E   I N F O

Article history:
Received 22 October 2015
Received in revised form 27 March 2016
Accepted 18 April 2016
Available online 2 May 2016

Keywords:
Shock/boundary layer interaction
Inlet unstart
Flow control
Ejecting
Cracking gas

A B S T R A C T

In order to increase resistance to back pressure in an inlet isolator, an ejecting flow control method is applied in this paper. For the sake of checking out the control effect, test cases with air and cracking gas ejecting are completed in $M = 2.41$ inlet. According to the numerical results, the resistance to back pressure with air and cracking gas ejecting is increased by 15% and 11.76% at $P_{\mathrm{t,eje}} = 1.07 \times 10^6$ Pa and $P_{\mathrm{t,eje}} = 4 \times 10^6$ Pa, respectively, which indicates that the control method is effective. The flow field characteristic with air and cracking gas ejecting is compared to reveal the difference of shock/boundary layer interaction and the propagating path of adverse pressure gradient. As the back pressure increased, the adverse pressure gradient can propagate upstream along the wide range of aerodynamic subsonic bands far from the wall, which are formed by the large-scale Mach stem of Mach reflection. Furthermore, the influence of ejecting total pressure on flow field is further analyzed to understand the physical mechanism of the resistance to back pressure. The increase of the ejecting total pressure can indirectly increase the ejecting momentum and decrease the ejecting dynamic viscosity, which prompts the random motion of molecules in the shear layer between ejecting flow and core flow, thereby increasing the mixing of the momentum, mass and energy to narrow the subsonic band and suppress the adverse pressure gradient.

1. Introduction

The inlet and isolator are important aerodynamic components in an air-breathing engine [1,2]. In these compression systems, shock-wave/boundary-layer interaction (SWBLI) is a complex physical phenomenon of compressible viscous flow. After a type of pre-combustion shock train inside the inlet undergoes multiple reflections, it can thicken boundary layer [3], cause flow distortions [4] and even induce flow separation [5]. At this point, if the combustor back pressure is too high for the shock-train length to match, unstart can occur. It is therefore critical to effectively control the boundary-layer flow to improve the performance of propulsion system and increase the stability margins.

Methods to control or minimize shock-induced flow separation have been proposed and are mainly classified as passive and active control. Control nature, whether passive or active, manages to increase the momentum of fluid near the wall so that the boundary layer can withstand the adverse pressure gradient imposed by incident shock or heat release at high equivalence ratio. Of these methods, since the suction control was first brought forward, its application has lasted to the present time. It can remove the low energy boundary-layer flow and reduce the size of the separation through a perforated domain [6–8]. Although suction provides many benefits to the propulsion system, its use often comes at a cost of increased drag and weight of the aircraft, thereby increasing the system complexity [9]. Another attractive control technique is micro vortex generators such as the micro-ramps. Saad et al. [10] performed the wind tunnel experiments at Mach 5 with two micro-ramps of different sizes to investigate the control of shock-wave/boundary-layer interaction and revealed the mechanism of the flow control device through schlieren visualization technique, surface flow visualization, and a new type of luminescent measurement technique such as infrared thermography [11]. Oorebeek et al. [12] devised a normal shock experiment with a supersonic inflow of Mach 1.35 to investigate the vortex generator and bleed effectiveness suppressing flow separation. They found that the vortex generator is similar to the bleed and can considerably reduce the separation bubble size, thereby improving the diffuser performance. Martis et al. [13] conducted a three dimensional numerical investigation to analyze the effect of micro-ramps on the separated swept shock-wave/boundary layer interactions. They analyzed the parametric influence of the height, width, and spacing

---

* Corresponding author. Address for correspondence: No. 92, West Da-Zhi Street, Harbin, Heilongjiang, 150001, PR China.
E-mail address: yudaren@hit.edu.cn (D. Yu).

http://dx.doi.org/10.1016/j.ast.2016.04.016
1270-9638/© 2016 Elsevier Masson SAS. All rights reserved.

Nomenclature

H_{ne}	height of the Aft-Facing Step Nozzle entrance
h_{iso}	height of the isolator
I	turbulent intensity
Imp_{bp}	the increased backpressure ratio
k	turbulent kinetic energy
L_{total}	total length of inlet model
L_1	the horizontal length of the first ramp in Aft-Facing Step Nozzle
L_2	the horizontal length of the second ramp in Aft-Facing Step Nozzle
M	Mach number
m	mass flow rate
PR	ratio of backpressure to freestream static pressure
P	pressure
R_{e}	Reynolds number
T	temperature
WE	with ejecting
WOE	without ejecting
v	mass-weighted average velocity at the entrance of AFST
x	X-axis coordinate
y	Y-axis coordinate
α	angle of attack
μ	dynamic viscosity
δ_1	first ramp angle
δ_2	second ramp angle
δ_3	cowl angle
δ_4	divergence angle
ω	turbulent dissipation rate

Subscripts

b	backpressure
c	cowl wall
cap	capture of inlet model
eje	ejecting condition
iso	isolator
r	ramp wall
s	static condition
t	total condition
∞	freestream condition

of the micro-ramps and drew a conclusion that the micro-ramps can significantly delay the boundary-layer separation. Alternatively, the larger height of micro-ramps can be more conductive to delaying the flow separation. Although the vortex generator can induce pairs of counter-rotating streamwise vortices to mix the high momentum fluid of core flow with the near-wall low momentum boundary layer flow, there is still a potential hazard of engine damage if the structure is destroyed [14]. Recently, an array of continuous air jet vortex generators (AJVGs) on upstream surface of separation bubble is used to successfully reduce the separation bubble size through inducing the periodical change of velocity which can redistribute the boundary-layer momentum, but the effectiveness is reduced at off design conditions [15]. However, the most of research mainly focuses on a single point control of separation bubble. Fewer investigations on the back pressure control can be seen. It is thus necessary to utilize a control method in inlet model to restrain the upstream propagation of adverse pressure gradient and simultaneously avoid several disadvantages of above methods.

The motivation for this work presented in this paper is derived from the idea that the turbopump system inside the supersonic or hypersonic aircraft has the ability to compress the cracking gas in the cooling channel of scramjet, a portion of high temperature cracking gas is expected to eject into the isolator to increase resistance to back pressure. Therefore, in this paper, test cases with air and cracking gas ejecting are conducted in $M = 2.41$ inlet to check out the control effect and initially grasp the flow field characteristic. The flow field characteristic with air and cracking gas ejecting is compared to reveal the difference of shock/boundary layer interaction and the propagating path of adverse pressure gradient. Furthermore, on that basis, the influence of ejecting total pressure on flow field is further analyzed to understand the physical mechanism of the resistance to back pressure.

2. Numerical approach

In the current investigation, the flow simulations are performed by ANSYS® Fluent 14.5. It uses a finite-volume technique with second-order upwind discretization to solve the two-dimensional compressible Reynolds-averaged Navier–Stokes equations and species transport equations without reactions. Flux vector splitting is done using advection upstream splitting method for approximation of convective flux functions. Implicit residuals smoothing, a multiple-grid method and full multigrid (FMG) initialization are applied to accelerate convergence. Additionally, the separation prediction is very important in many compression systems both for internal and external flows. Currently, the most prominent two-equation models in aerodynamic area are the k–ω based models of Menter [16]. The k–ω based Shear-Stress-Transport (SST) model is designed to give highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients by the introducing the transport effects into the formulation of the eddy-viscosity. The superior performance of this model has been demonstrated in a considerable number of validation studies [17]. So, the turbulent velocity profile in the paper is modeled by a two-equation k–ω SST turbulence model.

Since the discretization error and rounding error greatly depend on the grid resolution, a sequence of multiblock structured grids is tested to determine the grid sensitivity and validate the numerical approach according to the aerodynamic experiment by Reinartz at the Aachen Jet Propulsion Laboratory [18]. The sketch of the similar inlet model is shown in Fig. 1. Fig. 2 gives computational domain discretized by using a structured grid with regularly shaped cells and corresponding set of boundary conditions. Since the horizontal incoming freestream has already been compressed by the first ramp which has been completely neglected according to Ref. [18], the flow condition in Table 1 is applied to the far field at the left boundary of the domain while the pressure outlet is set by using a characteristic boundary condition, i.e. defining the static pressure equal to the incoming static pressure. The no-slip condition, adiabatic walls and zero normal-pressure gradients are imposed on all solid walls.

As reported in Ref. [18], the three dimensional effect of isolator almost had no effect on the pressure distribution on a symmetric surface in the numerical algorithm validation process. Therefore, a two-dimensional model is used for the numerical validation, and the impact of a small amount of flow separation on the side wall on the formation of shock wave is then ignored for the symmetric surface. The coarse grid, fine grid, and dense grid which clusters near the solid wall contains 610×65, 1117×130, and 2234×260 cells in the x and y directions, respectively. For the fine grid and