Aerospace Science and Technology 28 (2013) 91–99

Contents lists available at SciVerse ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Numerical prediction on the interaction between the incident shock wave and the transverse slot injection in supersonic flows

Wei Huang *,1, Zhen-guo Wang *,2, Ji-ping Wu, Shi-bin Li

Science and Technology on Scramjet Laboratory, College of Aerospace Science and Engineering, National University of Defense Technology, Changsha, Hunan 410073, People's Republic of China

ARTICLE INFO

Article history:
Received 6 April 2012
Received in revised form 27 September 2012
Accepted 19 October 2012
Available online 30 October 2012

Keywords:
Aerospace propulsion system
Transverse slot injection
Incident shock wave
Ramp
Supersonic flow

ABSTRACT

The incident shock wave has a great impact on the transverse injection flow field in supersonic flows, and it can improve the mixing efficiency between the incoming flow and the fuel. In the current study, the interaction between the incident shock wave and the transverse slot injection has been investigated numerically, and the influences of the size and the location of the ramp on the wall pressure profiles in the transverse slot injection flow field have been performed. Additionally, the swept angle of the ramp is set to be 15.784°, 28.158° and 35.974°, respectively. The obtained results show that the size of the ramp has a great impact on the transverse injection flow field, and the shock wave formed upstream of the injection slot moves towards the entrance of the channel with the increase of the swept angle of the ramp and the jet-to-crossflow pressure ratio. The transverse injection flow field is disturbed when the incident shock wave formed at the leading edge of the ramp is strong enough, and the separation regions both upstream and downstream of the injection slot increase with the increase of the intensity of the incident shock wave. The incident shock wave has a large impact on the mixing improvement between the incoming supersonic flow and the fuel.

1. Introduction

Transverse injection strategy for the fuel has been widely employed in the scramjet engine, and it provides good fuel penetration, mixing, and heat release but at the expense of a larger pressure loss [6,8,19,21]. Efficient mixing, ignition, and combustion are necessary for the successful operation of any air-breathing system [7,9].

Abdelhafez et al. [1] investigated the oblique and traverse configurations of injecting gaseous fuel in a low-aspect-ratio supersonic combustor numerically, and they found that the injecting fuel obliquely results in higher efficiency as well as effectiveness.

Huang et al. [12] assessed the impacts of angles of both the incoming air stream and the helium injection under various pressure conditions on the mixing process in the transverse injection flow field, and recently, they investigated the influences of the turbulence model and the slot width on the transverse injection flow field [11]. It is found that the RNG k–ε turbulence model is better for the wall pressure prediction of the transverse slot injection with the low jet-to-crossflow pressure ratio, and the SST k–ω turbulence model is better for the prediction of the transverse injection flow field with high jet-to-crossflow pressure ratio.

Erdem et al. [4] studied the influence of incoming flow and jet turbulence levels on jet interaction phenomenon, and a wide range of pressure ratios have been employed. Additionally, the transition locations have been captured and compared with the experimental data of Spaid and Zukoski [23]. Aso et al. [2] investigated the effect of injection angle for supersonic mixing by circular nozzle experimentally and numerically, and they found that large injection angle causes large total pressure loss ratio.

Further, Mai et al. [18] performed an investigation on the interaction between an incident shock wave and a transverse jet flow for mixing and combustion experimentally, and the effect of the incident shock wave on the three-dimensional flow structure and mixing mechanism between the airstream and the injected gas downstream of the injection slot has been conducted. In their study, the NO planar laser induced fluorescence and particle imaging velocimetry have been adopted for non-reactive flows. At the same time, Kim et al. [15] conducted an investigation on the effect of shock waves on the supersonic jet flame, and the interaction was classified according to the increasing tendency of the growth rate of the mixing layer.

In addition, the effect of the incident shock wave on the turbulent boundary layer was assessed numerically by Liou et al. [17], and the shock strength was changed by adjusting the angle of the shock wave generator. The turbulence models included a realizable

---

* Corresponding author.
E-mail addresses: gladrain2001@yahoo.com.cn (W. Huang), zgwang@nudt.edu.cn (Z.-g. Wang).
[1] Lecturer. Tel.: +86 731 84576452; fax: +86 731 84576449.
[2] Professor.

1270-9638/$ – see front matter © 2012 Elsevier Masson SAS. All rights reserved.
http://dx.doi.org/10.1016/j.ast.2012.10.007

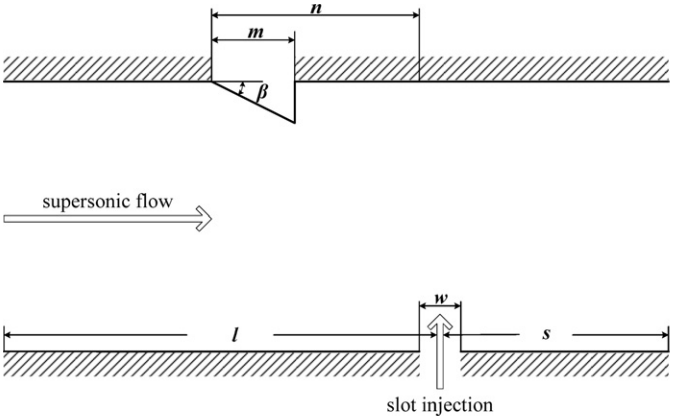

Fig. 1. A sketch of the interaction between the incident shock wave and the transverse slot injection flow field.

k–ε model, a k–ε/k–ω hybrid model, and a one-equation ν_t model. At the same time, Panaras [20] gave a brief review on the swept-shock/boundary layer interaction mechanisms of a high-speed vehicle, and the shock wave boundary layer interactions in non-equilibrium laminar flows were performed numerically by the Computational Fluid Dynamics (CFD) software [16].

However, the influences of the size, the location and the swept angle of the ramp on the interaction flow field have been performed rarely in the open literature, and this has a large impact on the mixing improvement between the incoming air and the fuel in supersonic flows. The incident shock wave can reduce penetration and increase mixing for injectants of all molecular weights, and the effects of the shock impingement are strongest when this occurs downstream of, and close to, the injection slot [22]. The investigation on the corner effect is out of the scope of the paper, namely the three-dimensional effects, and this is clearly stated in Ref. [3].

In this paper, the influence of the incident shock wave generated by the ramp on the transverse slot injection flow field has been performed numerically, and the size, the location and the swept angle of the ramp have been varied accordingly. In addition, the numerical code has been validated by the test results without the ramp, and the interaction mechanism between the incident shock wave and the transverse slot injection has been discussed in detail.

2. Physical model and numerical method

2.1. Physical model

In order to perform the investigation on the interaction between the incident shock wave and the transverse slot injection in supersonic flows, the experimental model, as studied by Spaid and Zukoski [23], is employed as the baseline configuration, and a ramp is located on the top wall of the channel to generate the incident shock wave, see Fig. 1. At the same time, the test results, without the ramp, have been used to validate the code predictions, and the validation process has been comprehensively described in Ref. [11]. The physical model can be considered to be two-dimensional for its small aspect ratio.

The distance from the plate leading edge to the centerline of the slot $l = 228.6$ mm, slot width $w = 0.2667$ mm, is taken according to the experimental conditions, and the distance from the centerline of the slot to the exit boundary of the computational domain is prescribed as 68.58 mm, namely $s = 68.58$ mm in Fig. 1. Fig. 1 shows the sketch of the interaction between the incident shock wave and the transverse slot injection in supersonic flows, and the incident shock wave is generated by the ramp. At the same time, the incoming air properties are set to be a Mach number M_∞ of 3.5, a static pressure P_∞ of 3145 Pa and a static temperature T_∞ of 86.5 K. The nitrogen jet flow Mach number M_j is set to be 1.0, with a static temperature $T_j = 298$ K and a jet-to-crossflow pressure ratio $P_j/P_\infty = 8.74$ and 63.5.

Table 1
Sizes and locations of the ramp for three cases when the swept angle is constant.

Items	m (mm)	n (mm)
Case 1	20	114.3
Case 2	40	114.3
Case 3	20	64.3

The dimensions of the ramp are set according to the relationship between the shock wave angle and the swept angle of the ramp, as shown in

$$\tan\beta = 2\cot\theta\,\frac{M_\infty^2\sin^2\theta - 1}{M_\infty^2(\gamma + \cos 2\theta) + 2} \tag{1}$$

where γ, θ, β and M_∞ represent ratio of specific heat, shock wave angle, swept angle of the ramp and the freestream Mach number, respectively.

In the current study, the shock wave angle is set to be 30°, 45° and 60°, and according to Eq. (1), the swept angle of the ramp is calculated to be 15.784°, 28.158° and 35.974°, respectively. At the same time, the length of the ramp (m) is set to be 20 mm and 40 mm, and the leading edge of the ramp is located at $x = 114.3$ mm and 164.3 mm, namely $n = 114.3$ mm and 64.3 mm, respectively. When the swept angle of the ramp is set to be constant, three cases are tested, and they are clearly illustrated in Table 1. Cases 1–2 are used to investigate the influence of the size of the ramp on the interaction between the shock wave and the transverse slot injection, and Cases 1 and 3 are employed to discuss the effect of the location of the ramp.

2.2. Numerical method

In the current study, the Reynolds Averaged Navier–Stokes (RANS) equations are solved with density based (coupled) double precision solver of FLUENT [5]. The RNG k–ε and SST k–ω turbulence models are employed to simulate the interaction between the incident shock wave and the transverse slot injection with the low and high jet-to-crossflow pressure ratios ($P_j/P_\infty = 8.74$ and 63.5), respectively. This is because the RNG k–ε turbulence model can predict accurately the wall pressure profile of the transverse slot injection with the low jet-to-crossflow pressure ratio ($P_j/P_\infty = 8.74$), and the SST k–ω turbulence model for the high jet-to-crossflow pressure ratio ($P_j/P_\infty = 63.5$) [11], see Fig. 2.

The second order spatially accurate upwind scheme (SOU) with the advection upstream spitting method (AUSM) flux vector splitting is utilized, and the Courant–Friedrichs–Levy (CFL) number is kept at 0.5 with proper under-relaxation factors to ensure stability [4]. This is because that the larger value causes the numerical results to diverge, while the smaller value slows down the numerical speeds [10]. The standard wall functions are introduced to model the near-wall region flow, and the no-slip conditions are assumed for the walls of the channel. At the outflow, all the physical variables are extrapolated from the internal cells due to the flow being supersonic [12–14]. The air is assumed to be a thermally and calorically perfect gas, and the mass-weighted-mixing-law of viscosity is employed in the current study. The solutions can be considered as converged when all of the residuals reach their minimum values after falling for more than three orders of magnitude, and the difference between the computed inflow and the outflow mass flux is required to drop below 0.005 kg/s.