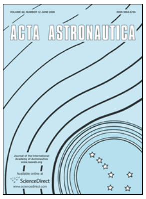
Acta Astronautica 105 (2014) 17–23

Contents lists available at ScienceDirect

Acta Astronautica

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



Numerical investigation of the nonreacting and reacting flow fields in a transverse gaseous injection channel with different species

Li Yan, Wei Huang*, Tian-tian Zhang, Hao Li, Xiao-ting Yan

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

ARTICLE INFO

Article history:
Received 23 June 2014
Received in revised form
23 July 2014
Accepted 20 August 2014
Available online 28 August 2014

Keywords:
Aerospace propulsion system
Transverse jet
Supersonic crossflow
Molecular weight
Combustion

ABSTRACT

The mixing and combustion process has an important impact on the engineering realization of the scramjet engine. The nonreacting and reacting flow fields in a transverse injection channel have been investigated numerically, and the predicted results have been compared with the available experimental data in the open literature, the wall pressure distributions, the separation length, as well as the penetration height. Further, the influences of the molecular weight of the fuel and the jet-to-crossflow pressure ratio on the wall pressure distribution have been studied. The obtained results show that the predicted results show reasonable agreement with the experimental data, and the variable trends of the penetration height and the separation distance are almost the same as those obtained in the experiment. The vapor pressure model is suitable to fit the relationship between the penetration height, the separation distance and the jet-to-crossflow pressure ratio. The combustion process mainly occurs upstream of the injection port, and it makes a great difference to the wall pressure distribution upstream of the injection port, especially when the jet-to-crossflow pressure ratio is large enough, namely 17.72 and 25.15 in the range considered in the current study. For hydrogen, the combustion downstream of the injection port occurs more intensively, and this may be induced by its smaller molecular weight.

1. Introduction

With the breakthrough of the sonic barrier, the hypersonic vehicle technique has attracted considerable attention worldwide [1–3]. The mixing and combustion process between a sonic jet and a supersonic crossflow has been the subject of interest in the aerospace engineering [4–9], and it makes a great difference to the engineering realization of the scramjet engine [10]. This is due to the extremely low fuel residence times in the combustor [11]. In order to prolong the time for the mixing process, a special fuel injection strategy has been proposed by the researchers, namely the inlet injection scheme [12–14]. Huang and Yan [15] have provided a detailed review on mixing techniques for transverse injection flow fields from four aspects, namely the jet-to-crossflow pressure ratio, the injector configuration, the number of injectors and the injection angle.

Lee [16,17] has investigated the nonreacting and reacting properties of a dual transverse injection system numerically, and the influences of the jet-to-crossflow momentum flux ratio and the distance between injection ports

---

* Corresponding author. Tel.: +86 731 84576447; fax: +86 731 84576449.
E-mail address: gladrain2001@163.com (W. Huang).

http://dx.doi.org/10.1016/j.actaastro.2014.08.018
0094-5765/© 2014 IAA. Published by Elsevier Ltd. All rights reserved.

on the mixing and combustion characteristics have been discussed comprehensively. The obtained results have shown that the dual injection system would bring better mixing and combustion performance but more total pressure loss than the single injection system, and there exists an optimal distance between injection ports for the dual injection system.

Gao and Lee [18] have compared the mixing characteristics of different injection schemes for supersonic transverse jet, namely the slot, circular-hole and two-stage injections, and the influences of the injection angle, the injector diameter and number on the mixing performance have been discussed numerically. It is found that the circular-hole injection can induce higher mixing efficiency, and the two-stage injection is superior to the single-stage one.

In addition, the influence of injectant species on the mixing properties in the transverse injection system has been carried out numerically by Watanabe et al. [19], and four different kinds of fuel have been taken into account, namely hydrogen, helium, nitrogen and ethylene. However, the combustion properties have not been considered.

To the authors' best knowledge, the nonreacting and reacting flow properties in the transverse injection system is not clear for the design of the scramjet engine, and comprehensive investigations still need to be performed to determine the mixing and combustion properties with different types of fuel in a relatively accurate supersonic combustor environment.

In the current study, the nonreacting and reacting flow fields in a transverse injection channel have been investigated numerically, and the numerical results have been compared with the available experimental data in the open literature. At the same time, the influences of the molecular weight of the fuel and the jet-to-crossflow pressure ratio on the wall pressure distribution have been discussed as well, and the jet-to-crossflow pressure ratio has been set to be 4.86, 10.29, 17.72 and 25.15.

2. Physical model and numerical method

2.1. Physical model

The experimental model, as studied by Aso et al. [20] is employed as the physical model to provide data for validation of the numerical method, and this is due to its good two-dimensional flow field structure. The distance from the plate leading edge to the centerline of the injection port $l=330.5$ mm, slot width $w=1$ mm, are taken according to the experimental conditions employed, and the distance from the centerline of the injection port to the exit boundary of the computational domain is prescribed as 221.5 mm.

We deal with the gas phase only, and the processes of atomization and spray formation are not taken into consideration in this study because all the materials are injected preheated to elevated temperatures [21]. The supersonic airstream flows from left to right. At the same time, the air properties are set to be a Mach number M_∞ of 3.75, a static pressure P_∞ of 11,090 Pa and a static temperature T_∞ of 78.43 K. The jet flow Mach number M_j is set to be 1.0, with a static temperature $T_j=249$ K and a jet-to-crossflow pressure ratio $P_j/P_\infty=4.86$, 10.29, 17.72 and 25.15.

2.2. Numerical method

In the current study, the two-dimensional Reynolds-averaged Navier–Stokes (RANS) equations are solved along with density based (coupled) double precision solver of FLUENT [22], and the SST k-ω turbulence model has been employed to simulate the transverse injection flow field for its good prediction of mixing layers and jet flows [23,24]. The SST k-ω turbulence model is a combination of the Wilcox 1988 k-ω model in the near wall region and the standard k-ε model in the detached regions [25]. At the same time, the one-step hydrogen-air mechanism and the Finite-rate/Eddy-dissipation reaction model have been used to simulate the reacting flow field, and the rate parameters for the one-step hydrogen-air mechanism can be referred to Ref. [26]. The hydrogen-air mechanism has only a slight impact on the mean flow properties in the reacting flow field [27], especially the wall pressure distribution [28], and the Finite-rate/Eddy-dissipation model has been proved to be more accurate for the simulation of the reaction flow field than the Eddy-dissipation model [29].

The second order spatially accurate upwind scheme (SOU) with the advection upstream splitting method (AUSM) flux vector splitting is utilized in the numerical process [30], and the Courant–Friedrichs–Levy (CFL) number is kept at 0.5 with proper under-relaxation factors to ensure stability. The standard wall functions are used 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. The air is assumed to be a thermally and calorically perfect gas, and the mass-weighted-mixing-law of viscosity is used. The convergence criterion is the same as that stated clearly in Ref. [31].

Meanwhile, the computational domain is structured by the commercial software Gambit, and the computational domain is multi-blocked in order to cluster the grid around the injection port. The computational domain is a rectangle surrounded by adiabatic walls at the bottom, supersonic inlet at the left boundary and sonic inlet at the injection port, an outlet at the right boundary and finally a symmetric condition on the upper boundary. The turbulent intensities for the air and fuel are both set to be 10%. The number of cells is 114,111, and the height of the first row of cells is set at a distance to the wall of 0.001 mm, which results in a value of $y^+ < 2.0$ for all the flow field. y^+ is a non-dimensional parameter defined by:

$$y^+ = \frac{\rho u_\tau y_P}{\mu} \tag{1}$$

here in, u_τ is the friction velocity, y_P is the distance from point P to the wall, ρ is the fluid density, and μ is the fluid viscosity at point P.

3. Code validation

3.1. Case 1

Fig. 1 displays the wall pressure distribution comparisons for the single transverse injection system employed