Acta Astronautica 139 (2017) 67–76

Contents lists available at ScienceDirect

Acta Astronautica

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

Effects of injection pressure variation on mixing in a cold supersonic combustor with kerosene fuel

Wei-Lai Liu [a], Lin Zhu [b,*], Yin-Yin Qi [b], Jia-Ru Ge [b], Feng Luo [b], Hao-Ran Zou [b], Min Wei [b], Tien-Chien Jen [c]

[a] *Department of Precision Machinery & Precision Instrumentation, University of Science & Technology of China, Hefei, 230027, China*
[b] *Lab of Mechanical Structure & Biomechanics, Anhui Agricultural University, Hefei, 230036, China*
[c] *Mechanical Engineering Department, University of Johannesburg, Johannesburg, 2006, South Africa*

ARTICLE INFO

Keywords:
Injection pressure
Transversal cavity injection
Cold supersonic flow
Couple Level Set & Volume of Fluids (CLSVOF)
Improved Kelvin-Helmholtz (K-H) & Rayleigh-Taylor (R-T) model

ABSTRACT

Spray jet in cold kerosene-fueled supersonic flow has been characterized under different injection pressures to assess the effects of the pressure variation on the mixing between incident shock wave and transverse cavity injection. Based on the real scramjet combustor, a detailed computational fluid dynamics model is developed. The injection pressures are specified as 0.5, 1.0, 2.0, 3.0 and 4.0 MPa, respectively, with the other constant operation parameters (such as the injection diameter, angle and velocity). A three dimensional Couple Level Set & Volume of Fluids approach incorporating an improved Kelvin-Helmholtz & Rayleigh-Taylor model is used to investigate the interaction between kerosene and supersonic air. The numerical simulations primarily concentrate on penetration depth, span expansion area, angle of shock wave and sauter mean diameter distribution of the kerosene droplets with/without evaporation. Validation has been implemented by comparing the calculated against the measured in literature with good qualitative agreement. Results show that the penetration depth, span-wise angle and expansion area of the transverse cavity jet are all increased with the injection pressure. However, when the injection pressure is further increased, the value in either penetration depth or expansion area increases appreciably. This study demonstrates the feasibility and effectiveness of the combination of Couple Level Set & Volume of Fluids approach and an improved Kelvin-Helmholtz & Rayleigh-Taylor model, in turn providing insights into scramjet design improvement.

1. Introduction

The jet-in-crossflow process is of particular importance in hypersonic vehicles because of its operations at high Mach number, e.g. at Mach 8. Sufficiently high speed generally leads to significantly short residence time of the free-stream within the combustion chamber, which in turn incurs incomplete mixing and strongly affects the combustion efficiencies [1]. Therefore, the study of the flow topology related to the liquid jet injecting into supersonic flow is intriguing and meaningful. Current design of the optimum injection system with great performance capabilities is really a challenge [2]. In this regard, the fundamental mechanism of the liquid jet in the high speed crossflows must be understood.

Currently, studies have already focused on the design of injector systems for improving the fuel-air mixing characteristics. For example, transversal fuel injection through a wall orifice is considered to be one of the simplest and most conventional approaches for the scramjet engine, possibly due to its better fuel penetration, interaction, and mixing [3–6]. Abdelhafz et al. [7] numerically investigated oblique and transverse configurations as gaseous fuel was injected into a low-aspect-ratio supersonic combustor. They claimed that injecting fuel obliquely can result in higher mixing efficiencies. Huang et al. [8] addressed the effect of injection angle of helium, under various pressure conditions, on the interaction between fuel and incoming air in the transversal injection flow field. They concluded that Re-normalization group (RNG) $k-\varepsilon$ turbulence model is better in predicting the wall pressures under low jet-to-crossflow pressure ratios, whilst shear stress transport (SST) $k-\varepsilon$ is more appropriate for high jet-to-crossflow pressure ratios. Hu et al. [9] studied the performance of a scramjet combustor with kerosene fuel injected from a strut under different equivalence ratios and observed that the mixing characteristics greatly vary with the strut fuel feeding ratio, especially when this ratio is close to the lowest and the highest limits. However, to the authors' knowledge, there are few studies in literature

---

* Corresponding author.
E-mail address: zl009@mail.ustc.edu.cn (L. Zhu).

http://dx.doi.org/10.1016/j.actaastro.2017.06.031
Received 19 June 2017; Received in revised form 26 June 2017; Accepted 28 June 2017

for the injection-pressure effects on fuel-air mixing characteristics in a cold kerosene-fueled supersonic air-stream. This emphasis is on the mixing characteristics in the kerosene-fueled supersonic flow field, and especially on the effects of the injection pressure on the interaction between incident shock wave and transversal cavity injection.

2. CFD model and simulation approach

2.1. Model geometry

Fig. 1 shows the real prototype and the corresponding three-dimensional (3D) model of the scramjet combustor developed by Liu [10]. The real scramjet combustor contains flange 1 and 6, pressure pad of upper glass window 3 and upper glass window 4. However, to simply the CFD model in this study, we have ignored the above-mentioned components due to no effects on the kerosene-air interaction and due to the reduced calculation time. Meanwhile, although strut 8 affects the flow behavior of the mixture in the combustor, it is not considered either because no related component exits in the present experimental setup of Liu's. Thus, the scramjet combustor herein only consists of rear cover 2, upper cover 5, lower cover 7 and cavity 9. Fig. 2 shows the simplified 3D geometry of the scramjet combustor by using feature-based modeling [11]. Table 1 shows the primary specifications of the combustor for calculations. Note that the incident shock wave generated by kerosene is injected from the orifice at the center of the cavity, as shown in Fig. 2.

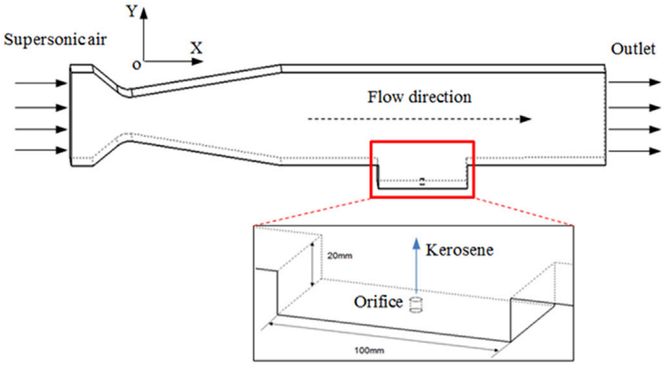

Fig. 2. Schematic of the scram jet combustor geometry.

Table 1
Specifications of the scramjet combustor.

Item	Dimension
Scramjet combustor	$0.6 \times 0.05 \times 0.08\text{m}^3$ (length × width × height)
Cavity	$0.1 \times 0.05 \times 0.02\text{m}^3$ (length × width × height)
Orifice	$1.0 \times 10^{-3}\text{m}(\Phi)$,0.02 m long
Inlet & outlet	$0.05 \times 0.08\text{m}^2$ (width × height)

2.2. CFD modeling

The meshed CFD model of the scramjet combustor is shown in Fig. 3, where the total of 330,000 hexahedron cells is used. As observed, the mesh cells are concentrated around either the walls or the region near the cavity due to the strong interaction between incident shock wave and transverse cavity injection.

Currently, the supersonic air-fuel interaction is predicted by using commercial CFD software ANSYS Fluent 14.0, in which volume of fluids (VOF) model is generally used to calculate the interface breakup and coalesce with another interface [12]. But, when a large deformation of the gas-liquid interface is involved additional re-meshing is greatly necessitated [13]. The mixing process of incident shock wave and transverse cavity injection in supersonic flows facilitates the more complex turbulent structures. Hence, in this study three dimensional Couple Level Set & Volume of Fluids (CLSVOF) model is proposed to investigate the effects of the injection pressure on the mixing characteristics between supersonic flow and kerosene in the cold scramjet combustor. This may be explained as due to the fact that the CLSVOF model, coupling the LS (Level Set) and VOF, not only contains their common advantages but also overcomes their disadvantages [14]. For CLSVOF, a single set of the conservation equation is used for the whole domain and there are no separate gas-liquid velocities. Consequently, the Navier-Stokes has the following form [14]:

$$\int_v \frac{\partial(\rho u)}{\partial t} + \int_v \nabla\cdot(\rho u \otimes u) = -\int_v \nabla p + \int_v \nabla\cdot(2\mu D) + \int_v F_{st} + \int_v \rho g \tag{1}$$

and the continuity equation is

$$\int_v \frac{\partial \rho}{\partial t} + \int_v \nabla\cdot(\rho u) = 0 \tag{2}$$

where $\vec{V}$ is a velocity vector, ρ–fluid density, t–time, μ–fluid viscosity, D–viscous deformation tensor, g–gravity vector and F_{st}–body force due to the surface tension.

In addition, such the other governing equations as the energy equation, the state equation of gaseous mixture and the turbulent model are also taken into account in this study. The detailed formulas are presented below, respectively.

The energy equation for a droplet is [15]:

$$\mathrm{m}\frac{de}{dt} = q + Q_s \ \ , \text{ where } e = c_{vs}T_s + h_f^0 \ \ , \ Q_s = \frac{dm}{dt}h_L \tag{3}$$

where Q_s is the energy of phase transitions, the heat flux q to a single droplet from the surrounding gas flow is determined by Ref. [16].

$$q = \begin{cases} \pi d\lambda\cdot Nu\cdot(T - T_s) & \text{Re} < 1000 \\ \pi d^2\rho|\vec{v} - \vec{u}|\cdot St\cdot(H_r - H_w) & \text{Re} \geq 1000 \end{cases} \tag{4}$$

where $Nu = 2 + 0.16\cdot\text{Re}^{2/3}\cdot\text{Pr}^{1/3}$, $St = \frac{C_d}{2}\text{Pr}^{-2/3}$, $\vec{v}$–droplet velocity,

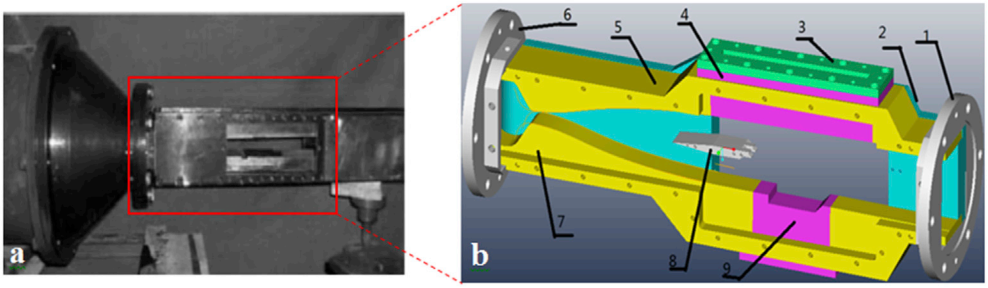

Fig. 1. (a) A real prototype and (b) A 3D model of the scram jet combustor: 1. flange, 2. rear cover, 3. pressure pad of upper glass window, 4. upper glass window, 5. upper cover, 6. flange, 7. lower cover, 8. strut, 9. cavity.