Applied Thermal Engineering 89 (2015) 883–896

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research paper

Numerical study of combustion and convective heat transfer of a Mach 2.5 supersonic combustor

Xi Wang, Fengquan Zhong*, Hongbin Gu, Xinyu Zhang

State Key Laboratory of High Temperature Gas Dynamics, Institute of Mechanics, Chinese Academy of Sciences, China

HIGHLIGHTS

- Convective heat transfer of supersonic combustor was numerically studied.
- Peaks of wall heat flux at varied fuel/air ratios are identified.
- Shock structures and vortices are related to flow separation caused by combustion.

ARTICLE INFO

Article history:
Received 26 December 2014
Accepted 25 June 2015
Available online 6 July 2015

Keywords:
Supersonic combustor
Convective heat transfer
Numerical study
Wall heat flux

ABSTRACT

In this paper, characteristics of combustion and convective heat transfer of a supersonic combustor at two fuel/air equivalence ratios of 0.9 and 0.46 were numerically studied. The numerical method of Favre averaged Navier–Stokes simulation with SST k-ω turbulence model and a multiple-step reaction mechanism of ethylene is introduced. The inlet Mach number of the combustor is 2.5 and inlet total temperature is 1650K, corresponding to Mach 6 flight conditions. Ethylene is injected at two locations upstream of a flame-holding cavity. The numerical method was validated by comparing the present results of wall pressures and heat fluxes to experiments and theoretical analysis. It is found that, due to injection of fuel at the bottom wall, fuel/air mixing and combustion occurs mainly in the vicinity of the bottom wall. High non-symmetry in distributions of the bottom and the top wall heat fluxes is observed. Peaks of wall heat flux at different locations and at varied fuel/air equivalence ratios are identified, which are caused respectively by effect of cavity and by shock structure formed upstream of the injection points. It is also found that heat flux peaks are strongly related to the reaction step of $CO \rightarrow CO_2$, contributing to major heat releasing.

1. Introduction

Thermal protection is one of the key technologies for successful scramjet operations. The combustor of scramjet has the most severe thermal environment. For example, the maximum total temperature of combustor may exceed 3000K at a flight Mach number of 6, and the wall heat flux would exceed 3 MW/m^2 [1].

It is known that convective heat transfer and heat loading on the combustor wall are mainly determined by flow field and combustion properties. Many physical processes including fuel injection and mixing, chemical reaction and heat releasing, shock train structure and its interaction with turbulent boundary layer, may affect wall heat flux, leading to highly non-uniform distributions along the main flow and the circumferential directions. The high non-uniformity in the wall heat flux imposes difficulties in design and optimization of thermal protection such as active cooling for supersonic combustor [2,3]. Therefore, it is imperative to study characteristic of convective heat transfer and wall heat flux at typical flow conditions for scramjet applications.

The direct and conventional measurement of wall heat flux is using high-temperature heat flux gage [4,5]. Another method is to measure time evolution of wall temperatures and interpret the temperature data to the wall heat flux via unsteady heat analysis [6,7]. Besides experiment works about heat flux measurement as mentioned in Refs. [4–7], other related works about wall heat flux of supersonic combustor are reported. Sanderson et al. [8] measured wall heat flux using surface thermocouple sensor.

* Corresponding author. No.15 Beisihuanxi Road, 100190, Beijing, China. Fax: +86 10 82544041.
E-mail address: fzhong@imech.ac.cn (F. Zhong).

http://dx.doi.org/10.1016/j.applthermaleng.2015.06.071
1359-4311/

Vincent et al. [9] obtained surface heat flux and temperature of Zirconia-coated copper wall via water-cooled heat flux gage and sub-surface temperature measurement for HIFiRE Direct Connect Rig. Kennedy et al. [10] applied Direct Write Technology for the measurement of heat flux of a direct-connect hydrocarbon-fueled scramjet combustor.

Those methods can only obtain heat fluxes at several points on the wall and lack of fine spacing resolution because of relatively large size of gages. However, numerical study with high accuracy is capable to obtain distributions of wall heat flux and to identify clearly local heat flux peaks. The numerical simulation also provides details of reacting flow field for better understanding of convective heat transfer of supersonic combustor.

In this paper, combustion and convective heat transfer characteristics of a supersonic combustor with inlet Mach number of 2.5 and inlet total temperature of 1650K were numerically studied. The computational method is to solve Favre averaged Navier–Stokes equations (FANS) with SST k-ω turbulence model and a multiple-step reaction model of ethylene. In the next section, numerical method and computational domain are introduced, followed by numerical validations. Results of the combustor flow and wall heat flux at two typical fuel/air equivalence ratios are then presented. Finally, conclusions based on the present results are given.

2. Numerical methods

The configuration of Mach 2.5 supersonic combustor is shown in Fig. 1. The height and width of the combustor inlet cross-section are 40 mm and 85 mm respectively. The total length of the combustor is 1419 mm including an isolator with constant cross section and a length of 395 mm, three divergent sections with angles of 1.5°, 2.0°, 5° at the bottom wall. As shown in the figure, there are two injection locations (Φ1, Φ2), of which, the upstream one is the main injection point. A cavity is installed downstream of the injections, which has a length-to-height ratio of 5.5. The fuel is ethylene and as shown in Table 1, the fuel/air equivalence ratios at two injections are 0.36/0.1 or 0.8/0.1. The total temperature of air at the inlet is 1650K, and the inlet Mach number is 2.5. The thermal boundary of the combustor wall is set to be a constant temperature of 1000K. The reason is that the long run combustor usually operates under regenerative cooling conditions at which the wall temperature is kept to be approximately 1000K [5,11]. The non-reflecting boundary condition is a commonly used outlet B.C. for supersonic flow based on characteristic analysis as described in the literature [12].

The computational domain is half of the combustor due to symmetry in the spanwise direction (z direction in Fig. 1) and the total mesh for computational domain are 3.3 million. The gird numbers in the normal and streamwise directions are changed to study the grid independence. It is found that 90 grids in the normal direction, 500 grids in the streamwise direction, 60 girds in the spanwise direction and 700,000 grids in the cavity are sufficient to obtain an accurate result. Parallel computation based on MPI algorithm is applied to accelerate the computation. It is noted that the first grid point from the wall is at $\Delta y^+ \leq 1$ and there are at least 10 grid points below $y^+ = 10$ for a good mesh resolution to simulate near-wall turbulent flow. The grids are structured except that in the vicinity of the fuel injectors with circular injection holes. The number of unstructured girds is 160,000 and very small compared to the total grids.

Table 1
Boundary conditions for computational domain.

Mach number at combustor inlet	2.5
Total temperature of air at combustor inlet	1650K
Wall temperature	1000K
Fuel/air equivalence ratio at injector Φ1	0.36/0.8
Fuel/air equivalence ratio at injector Φ2	0.1

It is worthy noticing that the Favre averaged wall heat flux with low frequency properties can be obtained by FANS method and fluctuations of wall heat flux caused by turbulence small scales with high frequencies would be lost. However, it is known that for metallic wall of the combustor, fluctuations of wall heat flux is not a critical issue as regarded in the applications of ceramic wall. Besides, FANS has advantages of good computational stability and high efficiency in solving engineering problems. Therefore, in this paper, FANS is adopted to simulate reacting flow field to investigate the spatial distribution of wall heat flux of supersonic combustor. The governing equations including continuity, momentum and energy equations are averaged to obtain FANS equations:

Continuity equation:

$$\frac{\partial \overline{\rho}}{\partial t} + \frac{\partial}{\partial x_i}\left(\overline{\rho}\widetilde{u}_i\right) = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial t}\left(\overline{\rho}\widetilde{u}_i\right) + \frac{\partial}{\partial x_j}\left(\overline{\rho}\widetilde{u}_j\widetilde{u}_i\right) = -\frac{\partial}{\partial x_j}\left(\overline{p}\delta_{ij}\right) + \frac{\partial}{\partial x_j}\left(\widetilde{\tau_{ji}^{tot}}\right) \tag{2}$$

Energy equation:

$$\frac{\partial}{\partial t}\left(\overline{\rho}\widetilde{e}_0\right) + \frac{\partial}{\partial x_j}\left(\overline{\rho}\widetilde{u}_j\widetilde{e}_0\right) = -\frac{\partial}{\partial x_j}\left(\widetilde{u_j p}\right) - \frac{\partial}{\partial x_j}\left(\widetilde{q_j^{tot}}\right) + \frac{\partial}{\partial x_j}\left(\widetilde{u}_i\widetilde{\tau_{ij}^{tot}}\right) + \widetilde{S}_J + \widetilde{S}_h \tag{3}$$

where, density weighted time averaging (Favre averaging) is defined as follows:

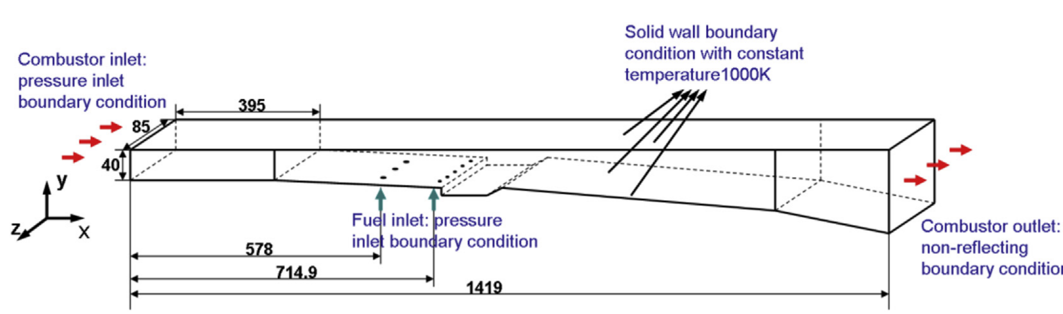

Fig. 1. Schematic diagram of a Mach 2.5 combustor with boundary conditions (Unit: mm).