Applied Thermal Engineering 98 (2016) 628–635

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

# Numerical investigation of flow distribution and heat transfer of hydrocarbon fuel in regenerative cooling panel

Yu Chen, Yu Wang, Zewei Bao, Qiyi Zhang *, Xiang-Yuan Li

*School of Chemical Engineering, Sichuan University, Chengdu 610064, Sichuan, China*

## HIGHLIGHTS

- A three dimensional CFD simulation of regenerative cooling panel was conducted.
- Pressure field in manifold was analyzed to explain the flow distribution.
- Branch throttle was adopted to eliminate the difference of flow rate.
- Subcritical and supercritical flow distribution of hydrocarbon fuel was simulated.
- Temperature difference on the surface of cooling panel was analyzed.

## ARTICLE INFO

*Article history:*
Received 20 September 2015
Accepted 23 December 2015
Available online 30 December 2015

*Keywords:*
Regenerative cooling panel
Hydrocarbon fuel
Flow distribution
Heat transfer
Numerical simulation
Branch throttle

## ABSTRACT

To study the flow and heat transfer characteristics of hydrocarbon fuel in parallel branch channels of regenerate cooling panel in scramjet engine, a three-dimensional model with a blockage structure set in downstream channel was investigated numerically. The influences of downstream blockage on pressure field and flow distribution were analyzed in detail. The results indicated that transverse pressure gradient and difference of pressure drop at inlet of branch channels directly caused non-uniform distribution of mass flow rate. To eliminate the difference of flow distribution, branch throttle was carried out, and the characteristics of pressure drop produced by throttle were discussed. Simultaneously, flow distributions with branch throttle at subcritical and supercritical temperature region were presented. The changes of wall temperature as a result of flow rate distribution were described. The results demonstrated that throttle structure at inlet of each branch channel was able to restrain the difference of mass flow rate at different temperature region effectively and avoid local overheating.



## 1. Introduction

Thermal protection is critical for hypersonic-flight vehicles. The scramjet (supersonic combustion ramjet engine) uses fixed components to compress and accelerate intake air by ram effect. The air enters the inlet and diffuser, which serve the same purpose as a compressor. The air delivered to a combustion chamber is mixed with injected fuel. This mixture is ignited and burns in the combustor. When scramjet works at the condition of flight speed over Mach 5, the walls of internal inlet, isolator and combustor are required to endure extremely high heat fluxes from the high-temperature (3000 K) burning gases [1–3]. Regenerative cooling by using hydrocarbon fuel on board has been considered to be the most effective solution so far [4]. In the cooling process, hydrocarbon fuel flows in several parallel cooling channels inside the wall of combustor as coolant, and may be heated even up to 1000 K as shown in Fig. 1. In order to avoid the instability of system caused by phase change of fuel, the operational pressure is maintained supercritical condition generally [5,6]. The chemical reactions of fuel in high temperature could contribute extra chemical heat sink for active cooling. Meanwhile, carbon deposition and coking will inevitably occur at high-temperature due to the aggregation of small molecules [7].

To improve heat transfer, the cooling channels were designed as many parallel small rectangular channels but with a single inlet manifold. There are many factors which lead to the difference of flow rate distribution: (1) Nonuniform flow in inlet manifold; (2) local flow resistances; (3) differences of friction resistance in channels. Nonuniform effusion caused by structure of inlet manifold is a traditional problem in fields of energy and chemical engineering. Local flow resistances in channels refer to corner, groove, nozzle that threaded channels and blockages produced by the coke adhered to inner wall. Meanwhile, the coke will lead to the difference of friction resistance among parallel channels. Furthermore, the asymmetric heat fluxes and complicated variations of thermodynamic physical properties result in difference of friction resistance as well. Seriously, the flow rate will reduce in channels with high

* Corresponding author. Tel.: +86 2885401846; fax: +86 2885407797.
*E-mail address:* qyzhang@scu.edu.cn (Q. Zhang).

http://dx.doi.org/10.1016/j.applthermaleng.2015.12.088
1359-4311/

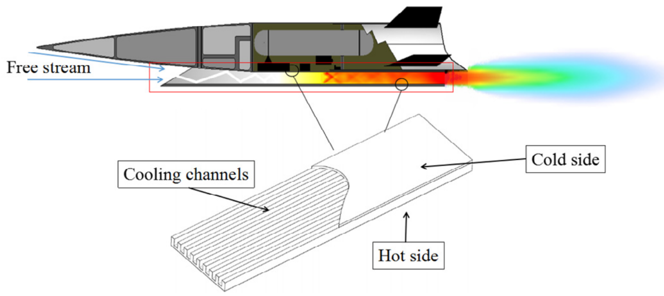


**Fig. 1.** Schematic diagram of cooling channels.

resistance, then the temperature will rise greatly, and finally the coking will be severe. Consequently the deviation of flow rate will be intensified sharply, and the heat transfer will be deteriorative obviously. It will lead to overheating in local region easily, and burning down of the whole system at last.

The flow distribution in parallel tubes had been studied by several investigators. Sparrow et al. [8] developed quasi-analytical method for determining the fluid flow in a multi-inlet collection manifold. Tong et al. [9] conducted numerical investigation to identify perfect strategy to achieve the same rate of mass outflow of a distribution manifold; C.C. Wang, et al. [10,11] studied characteristics of flow distribution in Z and U-type manifold experimentally and numerically, and discussed influences of gravity and jet flow on flow distribution. These investigations mainly considered structure of manifold, and drew quite significant conclusion. However, the complex properties of hydrocarbon fuel were not involved in these studies. Y. Feng et al. [12] conducted a numerical study focused on flow and convective heat transfer characteristics of hydrocarbon fuel in cooling panel with a local blockage structure (nozzle). By optimizing the shape of blockage structure, flow distribution became uniform only in subcritical temperature region, but the possible flow resistances in the downstream were not discussed in his work.

This paper is to investigate the flow distribution of hydrocarbon fuel with the flow resistance in the downstream of parallel channels. A three-dimensional CFD model was built and validated. The non-uniformity of flow distribution caused by blockage resistance in downstream was analyzed. Then a strategy was carried out at inlets of branch channels to eliminate difference of flow distribution. Furthermore the effect of convective heat transfer between fluid and solid was illuminated.

## 2. Physical and numerical model

### 2.1. Geometry description

As shown in Fig. 2a, there are several cooling channels inside the wall of combustor. The fuel is pumped to the single inlet of distribution manifold from tank, and then distributed to parallel channels. When fuel flows through parallel channels, it absorbs heat and injected into combustor to combust at last. As shown in Fig. 2b, the investigated model is composed of five parallel channels with length of 150 mm (No. 1, 2, 3, 4, 5) which are connected by one distribution inlet. For obtaining a full-developed flow-field, the inlet section is lengthened to 100 mm.

The cross-section dimension of channel is shown in Fig. 2c. The width and height are set to 2 mm and 1.5 mm respectively. The thickness of top and bottom plate both are 1.5 mm. The thickness of ribs among channels is 2 mm and that of outside is 1 mm. The bottom heated surface is named as "hot side".

A critical factor influenced on flow distribution is the pressure field in distribution chamber [13]. Resistances in downstream caused

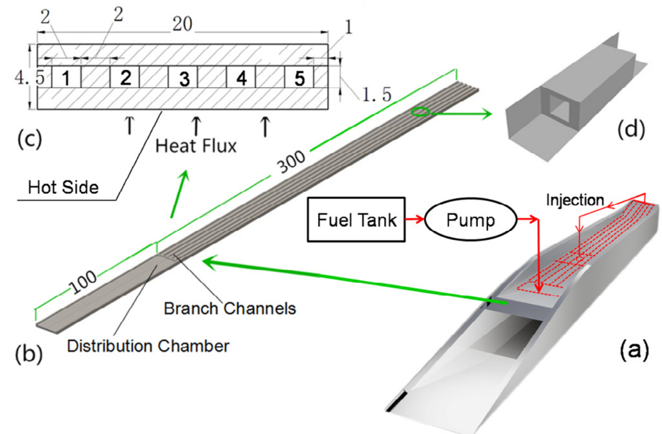


**Fig. 2.** Schematic configuration (Unit: mm).

by asymmetric thermal loading, structure differences and coking will all result in the changes of the pressure field in distribution chamber in upstream. Hence aforementioned resistances have the same property to impact flow distribution when flow reaches steady. Therefore, a blockage structure, as shown in Fig. 2d, was set up in the No. 3 channel, which represented all possible non-uniform restrain in downstream. It can observably effect the flow rate distribution.

### 2.2. Governing equations and boundary condition

A three-dimensional numerical simulation was conducted to investigate flow and heat transfer behavior of fuel in the cooling panel. The following conservation equations of mass, momentum, and energy were numerically solved in the fluid phase:

$$\frac{\partial \rho}{\partial t}+\nabla\cdot(\rho\vec{v})=0 \tag{1}$$

$$\frac{\partial(\rho\vec{v})}{\partial t}+\nabla\cdot(\rho\vec{v}\vec{v})=-\nabla p+\nabla\cdot(\bar{\bar{\tau}}) \tag{2}$$

$$\frac{\partial(\rho H)}{\partial t}+\nabla\cdot(\vec{v}(\rho h+p))=-\nabla\cdot\left(k_{eff}\nabla T+\left(\bar{\bar{\tau}}\cdot\vec{v}\right)\right) \tag{3}$$

Simultaneously, the three-dimensional governing equation of thermal conduction without internal heat source was solved in the solid region:

$$\frac{\partial}{\partial t}(\rho c\cdot T)-\nabla\cdot(k_s\nabla T)=0 \tag{4}$$

Because the hydraulic diameter of channel was very small, and the flow velocity in the channel was sufficiently high, the buoyancy effect was small and neglectable [12]. RNG $k-\varepsilon$ two-equation turbulence model was employed with standard wall functions. The SIMPLEC algorithm was adopted in simulation.

A commercial CFD package, Fluent 15.0, was used for the solution of the governing equations. The inlet boundary conditions were the constant mass flow rate and static temperature. The total mass flow rate was 10 g/s. There was a single inlet in the model, the static temperatures of the inlet were set as 400 K or 750 K in different calculation. The pressure of all outlets of 5 branch channels was 3 MPa. One side heating condition with constant heat flux 800 kW/m$^2$ was used on the one outside wall (hot side) from 70 mm after inlet to end, and the other outside surfaces were all treated as adiabatic.