

文章编号:1000-8055(2013)02-0459-07

Numerical simulation of axial liquid film cooling in rocket combustor

YANG Wei, SUN Bing, ZHENG Li-ming

(School of Astronautics,
Beijing University of Aeronautics and Astronautics, Beijing 100191, China)

Abstract: Numerical simulation has been done for liquid film cooling in liquid rocket combustor. Multiple species of axial Navier-Stokes equations have been solved for liquid-film/hot-gas flow field, and $k-\epsilon$ equations have been used for compressible turbulent flow. The results of the model agree well with the results of software FLUENT. The results show that: (1) Liquid film can decrease the wall heat flux and temperature effectively, and the cold border area formed by the film covers the whole combustor and nozzle wall. (2) The turbulent viscosity is higher than the physical viscosity, and its biggest value is in the border area of the convergent area in nozzle. The effect of turbulent flow on the whole simulation field can not be ignored. (3) The mass fraction of kerosene at the film inlet is 1, but it decreases along the nozzle wall and achieves its lowest value at the outlet. However, the mass fraction of kerosene near the wall is the biggest at any axial location.

Key words: liquid rocket engine; liquid film cooling; heat flux; numerical simulation; turbulent flow

CLC number: V231.3

Document code: A

Introduction

The temperature in liquid rocket engine is always more than 3 000 K, and the liquid film cooling can decrease the wall temperature and wall flux effectively. Numerical simulation method has been developed rapidly with the development of computer science. Many people have done computational fluid dynamics (CFD) analysis in liquid film cooling^[1-5]. However, a good method to simulate the liquid film cooling effectively and quickly is still in research.

Numerical study has been done for slot inlet film cooling in Kiran and his partners' study^[6]. The software of Loci-CHEM was used in turbulent subsonic film cooling simulation as CFD tool. Zhang et al.^[7] did axial numerical simulation for film cooling and separated the whole simulation field into liquid flow field and gas flow field. Chen et al.^[8] did CFD analysis for

supersonic turbulent film flow and they used different turbulent models to simulate film cooling near wall. Ohminami et al.^[9] did three-dimension CFD analysis for film cooled N_2H_4 /NTO liquid rocket, in which chemical reactions were included. In the study of Lu et al.^[10], cooling films coming from different kinds of slots were simulated by FLUENT. Kumar et al.^[11] did numerical analysis for film cooling with different geometry structures and the simulation results have been compared with the experimental results in their work followed. Li et al.^[12] simulated film cooling for single inclined slot structure based on L-B (Lattice-Boltzmann) method. Cruz et al.^[13] used LES (large eddy simulation) in film cooling numerical simulation. Further more, many studies have been done on film cooling by the commercial FLUENT^[14-16], and they showed that the simulation results are consistent with experimental results.

Received: 2012-02-16

URL: http://www.cnki.net/kcms/detail/11.2297.V.20130131.0829.201302.459_030.html

E-mail: maggieyangxie@sa.buaa.edu.cn

Numerical simulations have been done for liquid film cooling in LOx/kerosene liquid rocket combustor in this paper. Multi-species axial N-S (Navier-Stokes) equations for liquid film cooling have been solved. The main contents of this paper include the following items: establishing simulation models for liquid-film/hot-gas field in liquid rocket combustor, programming for the models based on C# codes and obtaining results of the flow field parameters, comparing the results of this method and that of FLUENT and analyzing them.

1 Calculation models

Fig. 1 shows the sketch map of liquid film cooling in liquid rocket combustor. The liquid film comes into combustor from the film inlet next to the wall. A cold border area is formed by the liquid film to protect the whole combustor wall.

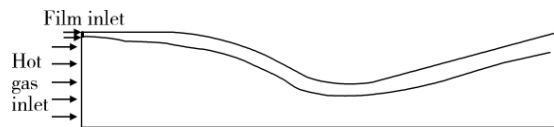


Fig. 1 Sketch map of liquid film cooling

1.1 Assumptions

To simplify the calculation models of the liquid-film/hot-gas flow field, the following assumptions are made.

1) The main stream gas consists of the combustion products of oxygen/kerosene, and the combustion process is completed instantaneously and sufficiently at the mainstream inlet.

2) The Reynolds number of the film flow is small and the gravity is neglected.

3) The axial conductivity of metal wall is neglected, and the wall is adiabatic.

4) Kerosene cracking and reactions between the film and the hot gas are neglected.

5) The radiation between the hot gas and the wall is neglected.

1.2 Equations

N-S equations for multiple species axial unsteady flow field are solved for the liquid-film/

hot-gas simulation area, which is shown in equation (1).

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial r} + \mathbf{G} = \frac{1}{Re} \left(\frac{\partial \mathbf{E}_v}{\partial x} + \frac{\partial \mathbf{F}_v}{\partial r} + \mathbf{G}_v \right) + \mathbf{S} \quad (1)$$

In equation (1), momentum equation and energy equation are

$$\mathbf{U} = \begin{Bmatrix} \rho \\ \rho u \\ \rho v \\ e \end{Bmatrix} \quad \mathbf{E} = \begin{Bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ (e + p)u \end{Bmatrix}$$

$$\mathbf{F} = \begin{Bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (e + p)v \end{Bmatrix} \quad \mathbf{G} = \frac{1}{r} \begin{Bmatrix} \rho v \\ \rho uv \\ \rho v^2 \\ (e + p)v \end{Bmatrix}$$

$$\mathbf{E}_v = \begin{Bmatrix} 0 \\ \tau_{xx} \\ \tau_{xr} \\ u\tau_{xx} + v\tau_{xr} + \dot{q}_x \end{Bmatrix} \quad \mathbf{F}_v = \begin{Bmatrix} 0 \\ \tau_{xr} \\ \tau_{rr} \\ u\tau_{xr} + v\tau_{rr} + \dot{q}_r \end{Bmatrix}$$

$$\mathbf{G}_v = \frac{1}{r} \begin{Bmatrix} 0 \\ \tau_{xr} \\ \tau_{rr} - \tau_{\theta\theta} \\ u\tau_{xr} + v\tau_{rr} + \dot{q}_r \end{Bmatrix} \quad \mathbf{S} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ \dot{q} - \dot{q}_{rad} \end{Bmatrix}$$

where e is the total energy of the fluid.

$$e = \frac{p}{(\gamma - 1)} + \frac{1}{2} \rho (u^2 + v^2)$$

$\dot{q}_{rad}$ is the radiation loss which equals to 0 here.

For viscosity items, there are

$$\tau_{xx} = -\frac{2}{3} \mu_t (\nabla \cdot \mathbf{V}) + 2\mu_t \frac{\partial u}{\partial x}$$

$$\tau_{rr} = -\frac{2}{3} \mu_t (\nabla \cdot \mathbf{V}) + 2\mu_t \frac{\partial v}{\partial r}$$

$$\tau_{xr} = \mu_t \left(\frac{\partial u}{\partial r} + \frac{\partial v}{\partial x} \right)$$

$$\tau_{rr} - \tau_{\theta\theta} = 2\mu_t \left(\frac{\partial v}{\partial r} - \frac{v}{r} \right)$$

$$\nabla \cdot \mathbf{V} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial r} + \frac{v}{r}$$

For heat transfer items, there are

$$\dot{q}_x = \beta k \frac{\partial T}{\partial x} \quad \dot{q}_r = \beta k \frac{\partial T}{\partial r}$$

where $\beta = \frac{k_\infty T_\infty}{\mu_\infty V_\infty^2}$, and the Reynolds number is

$$Re = \frac{\rho_{\infty} V_{\infty} L_{\infty}}{\mu_{\infty}}.$$

Equation (2) is for the k - ϵ equations and the species equations which only include Fick diffusion.

$$\begin{aligned} \frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial r} + \mathbf{G} = \\ \frac{\partial \mathbf{E}_v}{\partial x} + \frac{\partial \mathbf{F}_v}{\partial r} + \mathbf{G}_v + \mathbf{S} \end{aligned} \quad (2)$$

$$\mathbf{U} = \begin{Bmatrix} \rho k \\ \rho \epsilon \\ \rho Y_i \end{Bmatrix} \quad \mathbf{E} = \begin{Bmatrix} \rho u k \\ \rho u \epsilon \\ \rho u Y_i \end{Bmatrix} \quad \mathbf{F} = \begin{Bmatrix} \rho v k \\ \rho v \epsilon \\ \rho v Y_i \end{Bmatrix}$$

$$\mathbf{G} = \frac{1}{r} \begin{Bmatrix} \rho v k \\ \rho v \epsilon \\ \rho v Y_i \end{Bmatrix} \quad \mathbf{E}_v = \begin{Bmatrix} \mu_k k_x \\ \mu_{\epsilon} \epsilon_x \\ \rho D_{im} Y_{ix} \end{Bmatrix}$$

$$\mathbf{F}_v = \begin{Bmatrix} \mu_k k_y \\ \mu_{\epsilon} \epsilon_y \\ \rho D_{im} Y_{iy} \end{Bmatrix} \quad \mathbf{G}_v = \frac{1}{r} \begin{Bmatrix} \mu_k k_y \\ \mu_{\epsilon} \epsilon_y \\ \rho D_{im} Y_{iy} \end{Bmatrix}$$

$$\mathbf{S} = \begin{Bmatrix} G_k - \rho \epsilon \\ \frac{\epsilon}{k} (C_1 G_k - C_2 \rho \epsilon) \\ 0 \end{Bmatrix}$$

The turbulent viscosity is $\mu_t = \rho C_{\mu} \frac{k^2}{\epsilon}$, in which $C_{\mu} = 0.09$.

1.3 Numerical methods

MacCormack format is used in the equation (2), in which “predicting step, correcting step, average step” are selected to achieve the second order accuracy on time and space. Furthermore, certain artificial viscosity has been added, and the time step is restricted strictly by the condition of CFL (Courant, Friedrichs, Lewy) number. Stable questions are solved by these equations here.

Coordinate changing has been done for the governing equations of the simulation field. Then continuity equation, momentum equation and energy equation, k - ϵ equations and the specie equation of kerosene are solved. Circular simulations are continuing until the biggest error between the results of the nearest simulations is no more than 10^{-6} .

The whole program is realized by C# codes.

1.4 Boundary and initial conditions

In the liquid film inlet and the hot gas inlet, $u = u_0, v = v_0, \rho = \rho_0, e = e_0, Y_i = Y_{i0}, L = D_h, l = 0.07L, k = \frac{3}{2}(uI)^2, \epsilon = C_{\mu}^{3/4} \frac{k^{3/2}}{l}$, where u_0, v_0, ρ_0, e_0 and Y_{i0} are the given axial velocity, radial velocity, density, total energy and mass fraction, which have different values at different inlets; D_h is the hydraulic diameter of the inlet, and I is the turbulent intensity which is in 2% ~ 7% generally.

Outlet boundary conditions: the diffusion coefficients of all the parameters calculated are set to 0 because of the supersonic outlet.

Axis boundary conditions: the diffusion coefficients of all the parameters in the equations are 0.

Wall boundary conditions: adiabatic wall; $u = 0, v = 0, k = 0, \epsilon = 0$; the diffusion coefficients of e, ρ and Y_i are 0.

The initial conditions are settled the same as the hot gas inlet parameters.

2 Results verification and analysis

A liquid film cooled LOx/RP-1 rocket combustor is considered in this paper. The results are compared with that of FLUENT. The calculation conditions are listed in Table 1.

Table 1 Calculation conditions

Item	Hot gas	Cooling film
Inlet pressure/MPa	5	5
Inlet temperature/K	1000	300
Inlet mass flowrate/(kg/s)	25.43	2.32

2.1 Grids generation

Fig. 2 shows the grids of the liquid-film/hot-gas simulation field, in which there is $30 < y^+ < 300$ because of the request on y^+ in k - ϵ model. Further more, three types of grids, $131 \times 61, 151 \times 81$ and 201×101 , have been tested on grid independently. The biggest error between the wall temperatures is no more than 2.3%, so the 131×61 grids are chosen for sav-

ing calculation time.

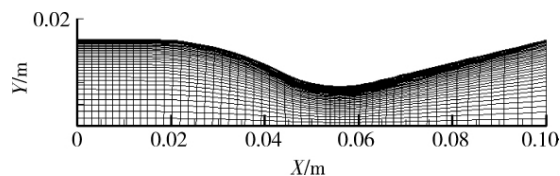


Fig. 2 Grids in simulation

2.2 Results verification

Wall temperature results from the simulation of the model in this paper and that of FLUENT are shown in Fig. 3, in which result 1 is obtained from the model in this paper and result 2 is obtained from FLUENT. These two curves are agreed well with each other, and the biggest error is no more than 6.8%.

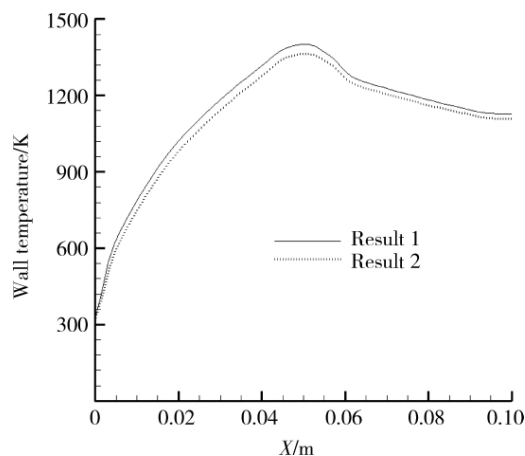


Fig. 3 Wall temperature results of two methods

Fig. 4 and Fig. 5 show the pressure distribution and temperature distribution of the two simulation methods. And the distributions are in ac-

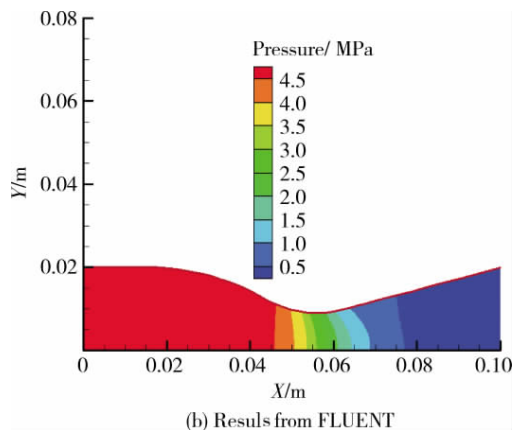
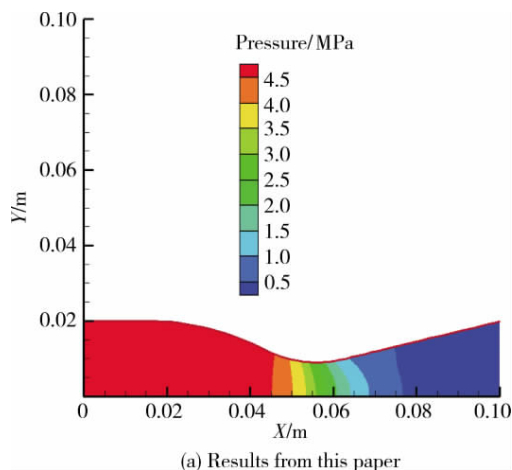


Fig. 4 Pressure distributions of two methods

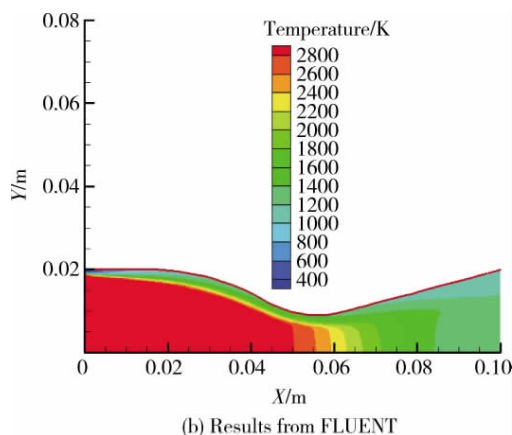
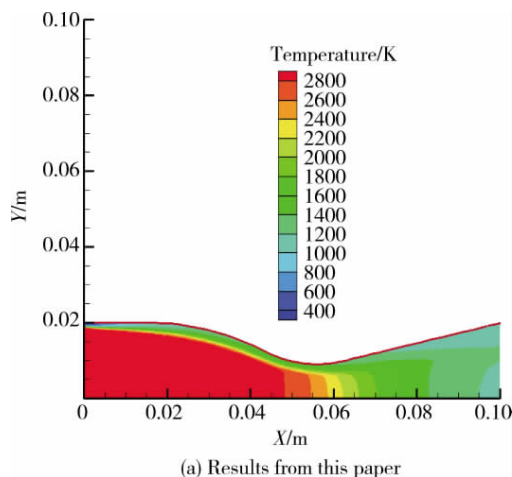


Fig. 5 Temperature distribution of two methods

cordance with each other except that the values are a little different. That verifies the accuracy of the method in this paper.

Just as Fig. 4 shown, the maximum pressure is at the cylindrical area of the combustor, and the combustor pressure decreases gradually along the nozzle. In Fig. 5, the maximum temperature is also at the cylindrical area of the

combustor and the minimum temperature is at the film inlet area. The main stream temperature decreases gradually along the whole nozzle behind the convergence area. It is obvious that the cold border area expands along the nozzle and it covers the whole wall until the outlet. Accordingly, the liquid film not only influences the combustor temperature distribution in the film outlet area, but also protects the whole nozzle wall.

2.3 Results analysis

Fig. 6 and Fig. 7 show the axial velocity and radial velocity distributions of the flow field. They shows that the axial velocity in the combustor increases along the axis; furthermore the smallest and the biggest axial velocity are at the film inlet and the center of the outlet area respectively. The velocity at the film inlet area is very small because of kerosene's big density, and the velocity in the front area of combustor is bigger. The radial velocity is below 0 in the convergence area of the nozzle and greater than 0 at the expanse area of the nozzle, however it is almost 0 in the other areas.

Fig. 8 shows the energy distribution of the flow field. It can be seen that the total energy of the combustor gas increases along the nozzle. The biggest value of it is at the centre position of the outlet area, and the smallest value in this area is at wall.

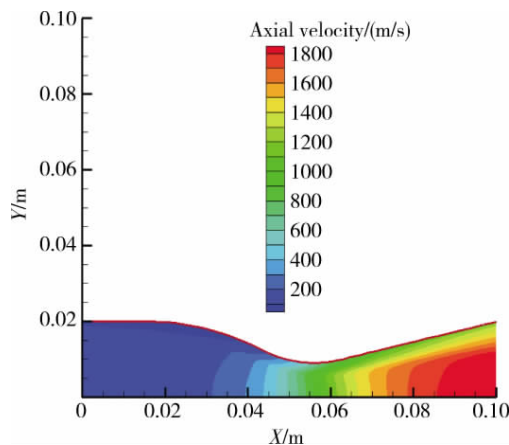


Fig. 6 Axial velocity distribution

Turbulent viscosity distribution of the flow field is shown in Fig. 9, from which it can be seen

that the turbulent viscosity is greater than the physical viscosity which is less than 10^{-4} Pa · s in most areas. The biggest value of the turbulent viscosity is in the border area of convergence area of the nozzle

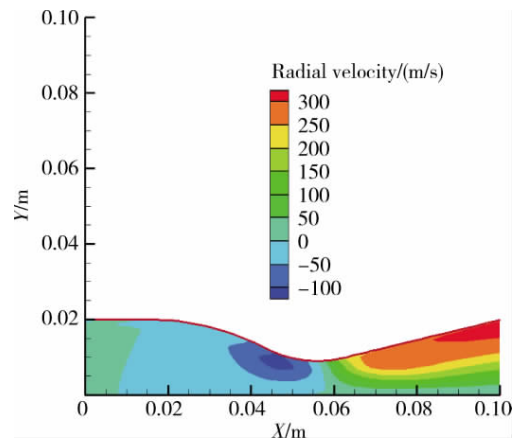


Fig. 7 Radial velocity distribution

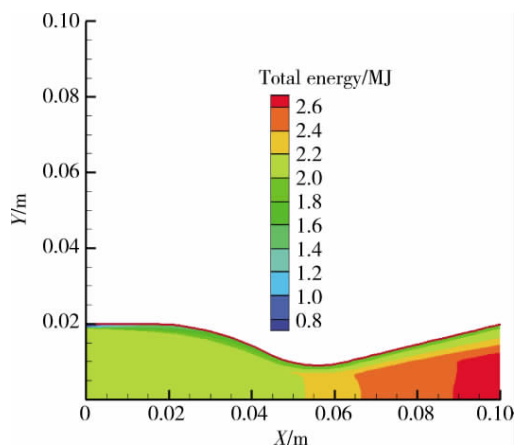


Fig. 8 Energy distribution

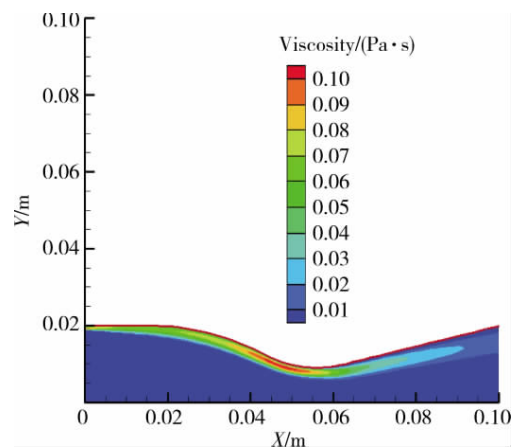


Fig. 9 Turbulent viscosity distribution

because the temperature of the kerosene film rises and the velocity of it increases quickly in that area.

Fig. 10 and Fig. 11 show the mass fraction of hot gas and kerosene distributions of the flow field, in which kerosene only distributes near the wall and the mass fraction of hot gas near the axis is almost 1. The mass fraction of kerosene at the film inlet is 1. However, with the diffusion from kerosene to the hot gas, the mass fraction of kerosene decreases and gets the minimum value at the outlet. But the biggest value of it at any settled axial location is at the wall.

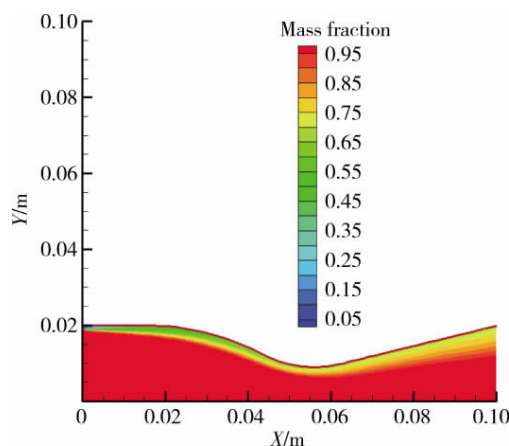


Fig. 10 Mass fraction of hot gas distribution

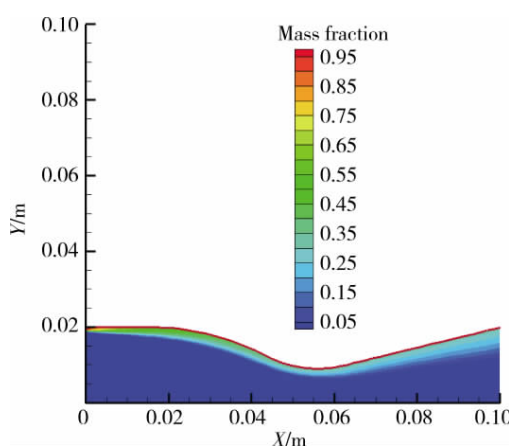


Fig. 11 Mass fraction of kerosene distribution

3 Conclusion

A calculation model has been established for liquid film cooling in liquid rocket combustor.

Numerical simulation has been done for LOx/kerosene rocket based on C# codes. The results show the following conclusions by studying this method.

1) The results of the method in this paper are consistent well with the results of FLUENT. This method can simulate the liquid film cooling in the liquid rocket combustor well.

2) Liquid film cooling is effective in decreasing the wall temperature, and the protection covers from the film inlet to the outlet of the nozzle.

3) The influence of turbulent flow to the whole flow field can not be neglected in film cooling simulation.

References:

- [1] Choi D. A Navier-Stokes analysis of film cooling in a turbine blade[R]. AIAA-93-0158, 1993.
- [2] Matesanz A, Velazquez A, Rodrlguez M. Numerical simulation of slot film cooling in convergent-divergent nozzles [R]. AIAA-93-1977, 1993.
- [3] James L R, Paul I K. Net heat flux reduction for pulsed film cooling on a turbine blade leading edge[R]. AIAA 2009-5103, 2009.
- [4] Gerald A, Pitalo P E, Joseph K. Numerical modeling of altitude diffusers[R]. AIAA-96-2280, 1996.
- [5] Inamura T, Amagasaki S, Yanaoka H. Thickness of liquid film formed by impinging jets on a concave wall[J]. Journal of Propulsion and Power, 2007, 23(3): 612-617.
- [6] Kiran H D, Andre W M, Arnaud T, et al. Numerical simulation of subsonic slot-jet film cooling of an adiabatic wall [R]. AIAA 2009-1577, 2009.
- [7] Zhang H W, Tao W Q, He Y L, et al. Numerical study of liquid film cooling in a rocket combustion chamber[J]. International Journal of Heat and Mass Transfer, 2006, 49 (1): 349-358.
- [8] Chen Y S, Chen C P, Wei H. Numerical analysis of hypersonic turbulent film cooling flows[R]. AIAA-92-2767, 1992.
- [9] Ohminami K, Uesugi K T, Ogawa H. Performance prediction of film-cooled bipropellant thruster[R]. AIAA 2006-4533, 2006.
- [10] Lu Y P, Ekkad S V. Predictions of film cooling from cylindrical holes embedded in trenches [R]. AIAA 2006-3401, 2006.
- [11] Kumar G N, Duncan B S, Mongia H. Results of a DOE on the film cooling effectiveness of a modern combustor with machined ring liners[R]. AIAA 2000-3354, 2000.
- [12] Li Y B, Zhou Y, Shock R, et al. Simulation of film cooling flow from one row of inclined cylindrical jets by using Lat-

- tice-Boltzmann method[R]. AIAA 2007-127, 2007.
- [13] Cruz C A, Raffan F, Cadou C, et al. Slot film cooling: LES and heat transfer measurements[R]. AIAA 2007-926, 2007.
- [14] Sung I K, Ibrahim G H. Unsteady heat transfer analysis of a film cooling flow[R]. AIAA 2008-1287, 2008.
- [15] Yang H T, Chen H C, Han J C. Film-cooling prediction on turbine blade tip with various film hole configurations[J]. Journal of Thermophysics and Heat Transfer, 2006, 20 (3): 558-568.
- [16] Thornburg H, Sekar B, Zelina J, et al. Design studies of turbine blade film cooling with unburned fuel in cross stream flow[R]. AIAA 2009-510, 2009.