Research on coupled heat transfer of film cooling in LOX/GH₂ thrust chambers

Jixin XIANG* and Bing SUN*

*School of Astronautics, Beihang University, Beijing 100191, China
E-mail: sunbing@buaa.edu.cn

Received: 15 May 2018; Revised: 6 September 2018; Accepted: 19 November 2018

Abstract
To better understand the performance of gaseous film cooling near the injector region in the LOX/GH₂ thrust chambers, a methodology is employed to simulate the coupled heat transfer of the film cooling in thrust chambers with regenerative cooling. The conjugated flow and heat transfer behaviors of film coolant, hot gas, cooling channels and regenerative coolant are numerically investigated. A three-dimensional non-adiabatic flamelet model using real gas equation of state is developed to solve the combustion and validated against the experimental data. Film cooling performance is predicted for the conditions with different geometrical parameters and mass flow rates of the film coolant. The result shows that the reverse flow zones are formed and developed in the region near the injectors. The occurrence of those zones is responsible for the significant reduction of the hot-gas-side wall temperature near the head plate of the injector. The coolant mass flow rate has a great influence on the film cooling performance due to the variety of coolant momentum in the exit of orifices and vorticity in the near-injector region. An optimum mass flow rate for maximizing the averaged effectiveness exists for a given film orifice geometric configuration. The high averaged effectiveness and the uniform flux distribution of hot-gas-side wall are observed at small orifices spacing. The film cooling effectiveness in the orifice exit region is obviously enhanced when the diameter of the orifice increased in the front part of the combustion chamber. The results would be useful for the analysis and optimization design of the straight cylindrical coolant orifices in the LOX/GH₂ thrust chamber.

Keywords: Liquid rocket engines, Gaseous film cooling, Turbulent combustion, Coupled heat transfer, Cooling effectiveness

1. Introduction
For the design of a new generation of liquid rocket engines (LRE), the performance in the thrust and the reliability for the propulsion system of LRE should be considerably improved, which puts forward higher requirements on the design of the engines. To provide the enormous energy for the thrust chamber in a typical liquid rocket engine, highly energetic and cryogenic propellants such as liquid oxygen and liquid hydrogen are used (Staschus and Frederick, 2016). Herewith, an effective thermal protection technology must be adopted to ensure the reliability of the thrust chamber working under harsh thermal conditions. Film cooling is an effective cooling technique widely used in combustors, turbines blades, afterburners, and nozzles (Arnold et al., 2009; Kim and Kim, 2016; Martelli et al., 2009). In the film cooling, coolant is directly introduced into the main stream through slots or orifices close to the protected surface. This is performed by injecting a layer of coolant fluid formed between the surface and the hot gas stream, which forms a barrier to protect the high temperature components from damage. This cooling method is typically used in liquid rocket thrust chambers in combination with the regenerative cooling.

Studies on film cooling over the years have been carried out in many investigations by the experimental and numerical approaches, which mainly refer to the effectiveness affected by various parameters such as blowing ratio, curvature of the wall and geometry of the coolant orifice. Bladauf et al. (1999) investigated the local heat transfer coefficients on a flat plate surface downstream a row of cylindrical ejection holes. The results showed that the insufficient coolant was not enough to protect the wall surface, but elevated blow ratio could lead to low effectiveness resulting from the intense mixing between the mainstream and the jet. Koç et al. (2006) did a numerical investigation of
the film cooling effectiveness on five different curved surfaces and a flat surface. The results revealed that the performance depended on the geometry, the shape of the curved surface and the blowing ratio. A study made by Yao et al. (2013) focused on the film cooling mechanism and characteristics of a row of cylindrical holes with branched jet injections by numerical simulation. Their findings revealed that the branched jet injection design produced a clearer improvement in film effectiveness at increased blowing ratios. These studies aimed to understanding the complex gas-film flow and heat transfer processes and to design unconventional holes with improved performance, but the simplified geometrical model was dominantly employed in these studies. Film-cooling over flat surfaces with streamwise coolant injection is similar to a row of discrete coolant holes along the surface of the blades, which is different from the film orifices configuration used in the liquid rocket thrust chamber.

It is a necessary process for the design of film cooling in a LOX/GH₂ thrust chamber to deeply understand the mechanisms about film cooling and injector-wall interaction in the axial and circumferential directions under different conditions (Suslov et al., 2009; Wang et al., 2018). Some works have been done by published experiments on the investigation of film cooling in a LOX/GH₂ thrust chamber. Arnold et al. (2010) found that the application of film cooling on a high performance rocket engine combustion chamber can effectively reduce the thermodynamic and structural loads of the combustion chamber within an acceptable range of engine loss. Suslov et al. (2011) used a new measurement method to obtain the axial and circumferential distribution of the cooling efficiency in LOX/GH₂ combustion chamber. But the experiments in Arnold et al. (2010) and Suslov et al. (2011) were performed on a simplified engine operating condition. In order to provide insight into the detailed and comprehensive flow field characteristic and film cooling effectiveness, the computational fluid dynamics (CFD) method have been employed in the investigation of the gaseous film cooling. Shine et al. (2012) computationally investigated coupled heat transfer performance of gaseous film-cooling inside a cylindrical test section similar to a rocket combustion chamber. Although their findings about the conjugate heat transfer model could be more accurate in predicting the film cooling effectiveness and could show significant difference with the adiabatic model, they used hot air as the core gas and gaseous nitrogen as the film coolant which were inconsistent with practical conditions. Miranda et al. (2011) used a CFD model for the calculation of gas-film cooling. This model was carried out to analyze the characteristics of gas film applied in Space Shuttle Main Engine (SSME), but the regenerative cooling channel was not included in their model. Amato et al. (2016). also presented a CFD method considering chemical reacting flow for film cooling in SSME. The results showed that the heat flux obtained by applying chemical reactions is different from that predicted by non-reacting model. Actually the mixing and combustion processes should be included to further improve computing precision at representative engine-like conditions.

This paper puts forward a numerical model including the combustion process to investigate gaseous film in the near injector region for LOX/GH₂ rocket engines. The Soave-Redlich-Kwong (SRK) equation of state is used to predict the real gas thermophysical properties for various species. The evaluation of the LOX/GH₂ turbulent combustion is based on a non-adiabatic flamelet model. Both Two separated parts are conducted in this methodology: the heat transfer and combustion analysis in the hot-gas region and the coupled fluid-solid calculation in the regenerative region. Furthermore, the combustion flow and heat transfer processes of a LOX/GH₂ thrust chamber are analyzed to explain the effects of various geometric parameters and film mass flow rate on the film cooling behavior in supercritical pressure. Finally, the investigation in this paper may be expected to optimize the design of film cooling in LOX/GH₂ thrust chamber in technological application.

2. Computational methodology

2.1 System of governing equations

In this study, the simulation is conducted under the assumption that the LOX/GH₂ thrust chamber operates in the stable condition to predict the steady coupled flow and heat transfer. The respective Reynolds-Averaged Navier-Stokes (RANS) equations are solved for the compressible steady 3-D flow by the finite volume method. The RANS equations are closed by the standard \( k-\varepsilon \) model which is more accurate than others in studying the boundary layer with the inverse pressure gradient and the near-wall flow for the film cooling (Kusterer et al., 2009; Lu et al., 2006). The RANS equations can be written respectively in a conservative form as:

1) Continuity equation
\[
\frac{\partial}{\partial x_j} (\rho u_j) = 0
\]

(1)

2) Momentum equation

\[
\frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \tau_{ij} \right) \quad (i, j = 1, 2, 3)
\]

(2)

Where stress tensor: \( \tau_{ij} = (\mu + \mu_i) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - 2/3(\mu + \mu) \frac{\partial u_k}{\partial x_k} \delta_{ij} \), \( \delta_{ij} = \begin{cases} 1 & (j = i) \\ 0 & (j \neq i) \end{cases} \).

3) Energy equation

\[
\frac{\partial}{\partial x_j} (\rho h u_j) = \frac{\partial}{\partial x_j} \left[ \frac{1}{\rho} \frac{\partial h}{\partial x_j} \right] + \frac{1}{\Pr} \frac{1}{\Pr} \sum_{i=1}^{N} \frac{\partial}{\partial x_j} \left( \frac{p}{c_{pi}} \right)
\]

(3)

Besides, the Fourier’s equation is used to the heat conduction of the cooling channel:

\[
\frac{\partial}{\partial x_j} \left( \lambda \frac{\partial T}{\partial x_j} \right) = 0
\]

(4)

2.2 Real-fluid equation of state and real-fluid transport properties

In most hydrogen/oxygen thrust chamber, hydrogen is injected at high temperatures which can be treated as an ideal gas. However, the temperature of the cryogenic liquid oxygen is below the critical temperature (Kim et al., 2011; Park and Kim, 2015). In the vicinity of the critical point, the thermophysical properties of fluid are dramatically varied owning to the modification of the relationship between pressure, density, and temperature caused by the intermolecular repulsive forces. Therefore, the real gas state of fluid should be taken into account for the cryogenic oxygen. The properties of fluid mixtures at critical state are difficult to be obtained due to the lack of the experimental data on multicomponent mixtures. To describe the real gas behavior, some analytic equations of state such as BWR were proposed in Cox and Chapman (2001). But the complexity of the equation may lead to high computational capacity. In this study, SRK equation of state (EOS) was used because of its high accurate and effective calculation process for hydrogen and oxygen (Cox and Chapman, 2001; Soave, 1972). The SRK equation of state takes the form:

\[
p = \frac{RT}{V-b} - \frac{a(T)}{V(V+b)}
\]

(5)

where \( a(T) \) is a temperature dependence parameter, which can be written as \( a(T) = \alpha \left[ 1 + n \left( 1 - \frac{T}{T_c} \right)^{0.5} \right]^{2} \), \( n = 0.48 + 1.574 \alpha - 0.176 \alpha^2 \). Terms \( \alpha \) and \( b \) which take into account attraction and repulsion effects among molecules are calculated by the following one-fluid van der Waals mixing rules (Redlich and Kwong, 1949).

Specific heat \( c_p \) was computed in terms of \( c_{p,\text{ideal}} \) and \( c_{p,\text{dep}} \) as follows:

\[
c_p = c_{p,\text{ideal}} - c_{p,\text{dep}} \frac{M_{\text{at}}}{M}
\]

(6)

\( c_{p,\text{dep}} \) is computed by differentiating departure internal energy with respect to \( T \), and the partial derivatives of the specific volume are computed by differentiating Eq. 5 appropriately.
Transport property such as the dynamic viscosity $\eta$ and the thermal conductivity $\lambda$ were calculated by using a formalism established by Chung et al. (1988), which was based on a formulation for dilute gas with a multiplication of the high pressure correction. For both $\eta$ and $\lambda$ for high pressure dense fluid, the appropriate mixing rules recommended by Chung et al. (1988), were applied.

### 2.3 Meshing and boundary conditions

A schematic of the LOX/GH$_2$ thrust chamber with film cooling in this study is shown in Fig. 1, which consists of a sector domain for combustion gas flow, injector elements, film cooling orifices, regenerative cooling channel, coolant, cooper alloy inners and nickel alloy outer jacket. There are 18 shear-coaxial injector elements in the injector head plate. The diameter of the central oxidizer inlet is 3.8mm, while the inner and outer diameters of the annular fuel inlet are 5.8mm and 7.9mm respectively. The length from the head plate to the throat is 394.2mm and the diameter of the cylinder part is 300mm. The nozzle was designed by dual arc method to ensure airflow acceleration and uniform flow field at the throat. The chamber contraction ratio and the expansion ratio are 4 and 7.5 respectively. As shown in Table 1, eight cases have been calculated in this paper in which geometry parameters film orifice diameter $D$, orifice spacing $\beta$ and the mass fraction of the film cooling orifice $m_{film}$ are varied.

![Fig. 1 Schematic of the LOX/GH$_2$ thrust chamber.](image)

<table>
<thead>
<tr>
<th>Case No.</th>
<th>$m_{film}$/%</th>
<th>$D$/mm</th>
<th>$\beta$/°</th>
<th>No. of holes around circumference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 0</td>
<td>10</td>
<td>1.2</td>
<td>10</td>
<td>36</td>
</tr>
<tr>
<td>Case 1</td>
<td>10</td>
<td>1.2</td>
<td>15</td>
<td>24</td>
</tr>
<tr>
<td>Case 2</td>
<td>10</td>
<td>1.2</td>
<td>30</td>
<td>12</td>
</tr>
<tr>
<td>Case 3</td>
<td>0</td>
<td>1.2</td>
<td>10</td>
<td>36</td>
</tr>
<tr>
<td>Case 4</td>
<td>7.5</td>
<td>1.2</td>
<td>10</td>
<td>36</td>
</tr>
<tr>
<td>Case 5</td>
<td>12.5</td>
<td>1.2</td>
<td>10</td>
<td>36</td>
</tr>
<tr>
<td>Case 6</td>
<td>10</td>
<td>0.8</td>
<td>10</td>
<td>36</td>
</tr>
<tr>
<td>Case 7</td>
<td>10</td>
<td>1.6</td>
<td>10</td>
<td>36</td>
</tr>
</tbody>
</table>

The computation domains and boundary conditions are shown in Fig. 2. In this study, in order to decrease the time of numerical calculation, only a 30° region of the chamber is simulated because the thrust chamber is axisymmetric in the circumferential direction. The designed combustion chamber pressure is 6MPa, which is above the critical pressure of oxygen (5.04MPa). The overall oxidizer-to-fuel mixture ratio $O/F$ is 6.5, and the total propellant mass flow rate is 5.9 kg/s. For the boundary conditions, the injection temperature of LOX and GH$_2$ is 94 K and 300 K, respectively. The inlet temperature of the regenerative coolant is 57K, and the inlet pressure is 8 MPa. The no-slip and iso-thermal wall is given on interface wall in the film/hot gas domain, while the no-slip and heat flux wall are given on the same wall of the coolant/cooling channel domain. For the other wall, no-slip and adiabatic condition are imposed.

The partial views of the computational grid around the injector and regenerative coolant/cooling channel domain are presented in Fig. 3. The software Gambit was used for the grid generation. The hexahedral mesh was used as grid topology in the computation domain so that the detail characteristics of the flow field could be efficiently captured by it. The grid was concentrated at complex flow regions, such as the regions around the injector, sudden contraction/expansion locations and wall boundaries. Meanwhile, the $y^+$ of the near-wall grid was 30-300 along with the entire length. The non-conformal grid system was adopted to the interfaces between the film/hot gas and the coolant/cooling channel domain, and the minimum grid size close to the interface in the film/hot gas domain is 10 times larger than that in the fluid of coolant/cooling channel domain.

2.4 Numerical method and coupled strategy

The respective RANS equations were solved by pressure-based solver considering the real gas property. The COUPLED algorithm (Ghobadian and Vasquez, 2007) was used for the pressure-velocity coupling in the turbulent flow fields of the hot gas and regenerative coolant. This algorithm offers strong robustness and more efficiency than the pressure-based segregated algorithm for steady-state flows. The standard wall function was chosen as the near-wall treatment because this function gives accurate results for boundary flow in very high Reynolds numbers at $y^+=30$–300. To reduce computational spending and get stable numerical results, the calculation is divided into two parts: regenerative coolant/cooling channel coupled flow and film/hot-gas heat transfer analysis. The manual iterative procedure for the couple heat transfer between the two parts was performed which has been used in our previous work to calculate the similar coupling heat transfer problems (Song and Sun, 2016; Yang and Sun, 2013).

For the film/hot-gas region, non-adiabatic flamelet model was used to simulate the mixing and combustion process in the LOX/GH$_2$ thrust chamber. This model was adopted in this study because of its accuracy and economy in the simulation of turbulent combustion (Song and Sun, 2017, 2016). The flamelet concept is based on the assumption...
that the turbulent flame is regarded as an ensemble of laminar, locally one-dimensional flamelet structures within the turbulent flow field. With this assumption, the species and energy transport equations can be transformed from the physical space to mixture fraction space. Therefore, the solution of the transportation equation for the mixture fractions is involved. Equations for individual species are not solved directly, and the species concentrations are derived from the predicted mixture fraction fields. The thermochemistry calculations are preprocessed and then tabulated for look-up in FLUENT. Interaction of turbulence and chemistry is accounted for with an assumed-shape Probability Density Function (PDF).

It is highly sensitive to the initial numerical calculation because the non-premixed combustion model is conjugated with SRK real gas equation. In this paper, the initialization process was completed by two steps. The first step was to use the first-order upwind scheme to discretize the density equation with the ideal gas model. The second-order upwind scheme and SRK real gas equation were not employed until the flow and reaction became stable in the second step. Each simulation was performed using FLUENT, and the solution was iterated unless the convergence criteria was satisfied. In this study, the convergence criteria was that the scaled residual for each governing equation was less than $10^{-3}$ and the net mass flow rate in the whole domain was within 1%.

3. Reference certification

3.1 Validation of thermophysical properties

In this paper, the chamber pressure 6MPa is higher than the critical pressure of oxygen, and the injection temperature of 85K is lower than the critical value. The hydrogen is injected in the supercritical state, while both of the chamber pressure and injected temperature are greater than the critical value of hydrogen. The thermodynamic and transport properties of gaseous supercritical hydrogen may be correctly modeled using the ideal gas EOS model. Therefore, we concentrated on verifying the thermodynamics and transport properties of oxygen estimated by the real gas EOS model.

![Fig. 4 Oxygen thermophysical properties](image)

Figure 4 shows the comparison of the data in National Institute of Standards and Technology (NIST) and predicted...
thermodynamic and transport properties of oxygen. The NIST data shows a sharply discontinuous transformation from subcritical condition to supercritical condition and pseudo-boiling with a drastic change in constant pressure specific heat is quite apparent. In terms of the density and constant pressure heat, real fluid SRK EOS fits the NIST data very well for a wide range of pressure, while the ideal fluid EOS is unable to predict the values on the subcritical and transcritical conditions accurately. For the transport properties, the effects of pressure as well as the transition from subcritical to supercritical temperatures are very closely predicted to the NIST data by Chung model. However, neither for dynamic viscosity nor for the thermal conductivity, the pressure influence cannot be considered by the ideal EOS.

3.2 Validation of CFD methodology

To verify the applicability of the selected model, the CFD methodology was validated prior to the numerical simulation in the investigated LOX/GH\textsubscript{2} thrust chamber. The experiment used for the validation of CFD methodology is the RCM-3 test case was performed at MASCOTTE cryogenic combustion test facility (Juniper et al., 2000). The operating conditions of the RCM-3 case were summarized in Pohl et al., (2013). The chamber pressure is 6MPa which is above the critical pressure of oxygen. Oxygen is injected in a subcritical state, while hydrogen is injected in a supercritical state which is similar to the jet conditions of LOX/GH\textsubscript{2} thrust chamber in this paper.

As shown in Fig. 5, the two-dimensional model is used for the numerical investigation which is also used in Cheng and Farmer (2006). The length and width of the square combustion chamber is 487mm and 50mm respectively. At the downstream thrust chamber, the nozzle has a convergent length of 20mm and a throat diameter of 15mm. More other geometric parameters are the same as the model parameters in Pohl et al. (2013). A three-zone mesh system (60×54, 322×54, 24×54) was employed to model the injector section, the combustion chamber and the nozzle, respectively. In the LOX post region and GH\textsubscript{2} post region, a grid-sensitivity study was accomplished with fine grids.

![Fig. 5 Calculation fields and grids for the MASCOTTE RCM-3 test case.](image)

![Fig. 6 Comparison of calculated temperature contour and Abel transformed OH intensities from the experiment for RCM-3 test case.](image)

![Fig. 7 Comparison of experimental and calculated axial temperature profiles for the RCM-3 test case.](image)
Banuti et al. (2014) and Pfitzner et al. (2008) used the distribution of Abel transformed OH emission intensities $I_{\text{OH}}$ from chemiluminescence measurement technique in RCM-3 test to qualitatively validate the numerical flame shape. The comparison between the experimental data and simulated results is shown in Fig. 6. In this figure, the reaction zone and temperature contours for the simulation with non-premixed combustion model give a good match with RCM-3 test data. The length of the simulated flame is correctly reproduced, and the radial extension of the flame seems same as that in the experiment. A quantitative comparison of the axial temperature distribution of the flame is presented in Fig. 7. The position of maximum temperature indicates the place of flame tip. It can be recognized that axial place predicted in this paper is closer to the experimental data than that predicted in Juniper et al. (2000). However, both combustion models underestimate the temperature in the flame tip. Meanwhile, the predicted temperature in the downstream region after 0.2 m is lower than the experimental result by about 200-300 K, which could be acceptable in the engineering application. This mismatch could be explained by the neglect of some exothermic reactions resulting from the simplified chemical reaction mechanism. The LES method may be better than RANS method for the heat transfer analysis of the rocket engine combustion chamber, but the demand for computing resources of LES is too large to meet.

4. Grid independent test

A grid convergence analysis was carried out for the case 0 with 10% GH$_2$ film. Table 2 shows the three different grid resolutions used for different regions. Independence of adopted grid was verified by comparing the hot-gas-side wall temperature with different resolution. Fig. 8 illustrates the hot-gas-side wall temperature of varied grid resolutions. The largest difference between resolution type 2 and type 3 is about 3%, which is much less than those between resolution type 1 and type 2. Thus, the completed grid tests show that resolution type 2 is applicable to accurately and effectively describe flow and heat transfer of regenerative cooled thrust chamber with the gas film cooling. However, in order to reduce the computation quantities, the resolution type 2 is adopted in the following research.

<table>
<thead>
<tr>
<th>Region</th>
<th>Type 1</th>
<th>Type 2</th>
<th>Type 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hot gas</td>
<td>166x65x17</td>
<td>270x117x35</td>
<td>370x155x50</td>
</tr>
<tr>
<td>Film orifice</td>
<td>10x4x4</td>
<td>20x8x6</td>
<td>30x12x8</td>
</tr>
<tr>
<td>LOX post</td>
<td>20x6x6</td>
<td>40x12x12</td>
<td>60x18x18</td>
</tr>
<tr>
<td>GH$_2$ post</td>
<td>20x6x2</td>
<td>40x12x4</td>
<td>60x18x6</td>
</tr>
<tr>
<td>Inner wall</td>
<td>150x50x3</td>
<td>253x101x4</td>
<td>350x150x5</td>
</tr>
<tr>
<td>Rib</td>
<td>150x25x4</td>
<td>253x45x8</td>
<td>350x65x12</td>
</tr>
<tr>
<td>Nickel jacket</td>
<td>90x30x4</td>
<td>180x60x8</td>
<td>240x90x12</td>
</tr>
<tr>
<td>Coolant</td>
<td>290x72x10</td>
<td>327x144x20</td>
<td>360x210x30</td>
</tr>
</tbody>
</table>

![Grid resolution](image)

Fig. 8 Hot-gas-side wall temperature for various grid arrangements.
5. Results and discussion

To evaluate the efficiency of the film cooling, the non-dimension film cooling effectiveness $\eta$ is defined by

$$\eta = \frac{T_{wg,0}(x) - T_{wg,\text{film}}(x)}{T_{wg,0}(x) - T_c}$$  \hspace{1cm} (7)

where $T_{wg,0}$ is the wall temperature without film cooling and $T_{wg,\text{film}}$ is the wall temperature with film cooling.

5.1 Velocity field

The contour of velocity and streamlines in a plane paralleled to the hot gas flow and passing through the center of the middle coolant holes is shown in Fig. 9. It can be note that the film coolant is injected into the combustion chamber and diffuses in the circumferential and axial direction changing the original flow field structure in the head region. Because of the shear force caused by differential velocity between the main stream and the film, the counter rotation vortex pairs (CRVP) are created on both sides of the hole. The CRVP impedes the flow of coolant in the circumferential direction and enhances the mixing strength between the coolant and the mainstream which result in a reduction film cooling capacity.

![Fig. 9 Contour of axial velocity and streamlines in a parallel plane.](image)

![Fig. 10 Contour of axial velocity and 3D streamlines near the injector plate.](image)

Streamlines demonstrating the downstream flow features of the film are shown in Fig. 10. The reverse flow region is developed in three regions: (1) the region adjacent to the jet exit and close to the wall; (2) the region between the film exit and the injector exit; (3) the region exists between different injector exits. It can be observed that the reverse flow region close to the wall results from the rebound generated after the film coolant impacts the wall. In the region between the film and injector, a vast eddy is also developed and is responsible for the higher mixing of the coolant and the mainstream. The recirculation zone between the injectors is due to the mixing and shearing between the injected propellants. The reverse flow can also result in the increasing heat transfer coefficient in these regions.

5.2 Temperature of film/hot-gas region

Figure 11 depicts the static temperature contours of the film/hot-gas region in the LOX/\(\text{GH}_2\) thrust chamber for case 0 and case 3. In the near injection region, the existence of film cooling significantly reduces the temperature in the injector plate, and forms a cold border area near the wall. The intense mixing and reaction between cryogenic oxygen and gaseous hydrogen result in rapid temperature increase in the recirculation zone. It can be note that the temperature
of the recirculation zone in case 0 is lower than that in case 3. In the middle cross section of the cylindrical combustion chamber, the coolant unmixed with the hot gas leads to the low hot gas temperature near the wall in case 0. In the nozzle, the temperature distribution is consistent with that in a typical Laval nozzle, and the effects of film cooling are not obvious. For both case 0 and case 3, the development of the chemical reaction is complete and the hot gas has a tendency to become more homogeneous in the exit of the chamber.

![Temperature contour of film/hot-gas region](image)

Fig. 11 Temperature contour of film/hot-gas region

The streamwise distributions of the hot-gas-side wall temperature are compared to illustrate the film cooling effect along the thrust chamber wall. Fig. 12 shows the difference between the case 0 without the film cooling and case 3 with 10% film. The hot-gas-side wall temperature is averaged in circumferential direction. After the film cooling is adopted, the local maximum hot-gas-side temperature is almost vanished in the near injection region because the film coolant absorbs more heat from the hot gas. The influence of film cooling on the temperature of the hot-gas-side wall is remarkable in the axial distance of -0.3m to 0m. However, the difference of the wall temperature decreases along the hot gas flow direction in the nozzle due to the mixing and reacting of the mainstream fluid and the coolant, and slightly lower temperature of the throat is observed in case 0.

![Streamwise distributions of hot-gas side wall temperature with case 0 and case 3](image)

Fig. 12 Streamwise distributions of hot-gas side wall temperature with case 0 and case 3

Figure 13 shows the radial profiles of the constant-pressure specific heat and gas temperature in the symmetric plane including the central injector and one half of the outer ring injector at x=0.015m. These numerical results clearly reveal the abrupt increase of the constant-pressure specific heat near the oxidizer post for the cryogenic propellant reacting flows. This data suggest that the effects of the pseudo-boiling behavior in the near-injector region are relatively strong. The location of the pseudo-boiling is evidently close to the oxidizer post, where chemical reaction of the oxidizer becomes significant.

![Radial profiles of the constant-pressure specific heat and temperature at axial distance x=0.015m](image)

Fig. 13 Radial profiles of the constant-pressure specific heat and temperature at axial distance x=0.015m
propellant is not intense. In the oxidizer post, the level of the constant-pressure specific heat reaches local minimum because the oxidizer remains a relatively low temperature state. Meanwhile, the local maximum temperature of the gas is observed in the region close to fuel post and behind the lip separating the oxidizer and fuel post, which leads to the local minimum of the constant-pressure specific heat near the same radial location because of the strong chemical reactions.

5.3 Effects of mass flow rate

Figure 14 shows the variation of the cooling effectiveness and gas-side-wall heat flux with normalized axial distance for different mass flow rate. As shown in Fig. 14 (a), a large drop in effectiveness near the film injection along the direction in all cases appears which can be explained by the effects that intense expansion of the flame impacts the wall. In the neighborhood of film exit, cooling efficiency increases at first and then decreases as coolant flow rate increases. That is because the intense mixing and higher shear interaction with the mainstream at high mass flow rate brings about the increase in heat transfer from hot gas to coolant. However, the cooling effectiveness is the highest at the larger mass flow rate behind the injection zone. Because of the existence of the film coolant near the boundary layer, the heat flux is prevented from being transmitted to the metal surface. Thus the high cooling efficiency corresponds to a low hot-gas side wall heat flux value (Fig. 14 (b)). The dynamical balance between hot-gas-side temperature and heat flux confirms that the coupling effects obviously influence the characteristics of the heat transfer in thrust chambers.

![Figure 14: Streamwise variations of cooling effectiveness and wall heat flux for three mass flow rate](image)

Table 3 shows the normalized maximum turbulence intensity level at various downstream locations of the injector. The comparison is done with the maximum turbulence intensity in the 2 D downstream plane of cooling injection in case 5 (12.5% film). The turbulence is generated by the difference in velocity between the main stream and the coolant jet and also generated by the expansion of the hot gas flow near the exit plane of the orifice. The maximum turbulence intensity of in the downstream section increases with the mass flow rate. At high turbulence intensity condition, the hot gas is dragged toward the wall and the high convection heat transfer coefficient is produced. Therefore a decline in effectiveness in the vicinity of injection appears when the mass flow rate of coolant is sufficiently large.

<table>
<thead>
<tr>
<th>Mass flow rate</th>
<th>normalized maximum turbulence intensity level</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0 D downstream of cooling injection</td>
</tr>
<tr>
<td>7.5%</td>
<td>0.25</td>
</tr>
<tr>
<td>10%</td>
<td>0.32</td>
</tr>
<tr>
<td>12.5%</td>
<td>0.39</td>
</tr>
</tbody>
</table>

5.4 Effects of cooling orifice spacing
The effects of three film orifice arrangement patterns on film cooling are simulated while the film orifice diameter, coolant flow rates and mainstream gas conditions keep constant. Fig. 15 presents the effect of changes in orifice spacing on film cooling effectiveness. The overall low cooling effectiveness of case 2 is due to the fact that only 12 orifices are arranged, which results in inhomogeneous and inadequacy cooling on the region between the injectors. The result of Case 0 shows high cooling efficiency in the region very close to the injection region drops dramatically. The cooling effectiveness in the region slightly far from the injection of case 0 is less than that in case 1. When the orifices are closer, the generated vortices in the adjacent holes resulted from the changed velocity of the coolant interact each other and prevent the hot gas from the chamber wall. The main characteristics observed for the closer orifices are the increase in vortices in the near-injection region before 2D (about 30% increase for case 0). When the orifice spacing increase, more and more hot gas reaches the surface of the chamber wall and the effectiveness is low. Meanwhile, the vortices are quickly destroyed away from the injection zone due to the mixing and combustion process between the hot gas and coolant film.

![Effectiveness variations along the axial direction at different orifice spacing](image)

Figure 16 shows the variations of the normalized heat flux around the circumference for three different arrangements of film cooling orifices in the plane at 1D. The normalized heat flux is defined as the ratio of the circumferential heat flux in different cases to the circumferential heat flux in case 3. The circumferential heat flux distribution of case 3 at the same location is also presented. The heat flux profile of case 3 shows a non-uniformity and the peaks of the heat flux is located in the position where the injector is arranged, which is caused by the impingement of the near-wall flame (see Fig. 11). The circumferential heat flux distribution is more uniform in case 0 than that in case 1 and case 2. The obvious “wavy shape” heat flux profile is observed in case 1 and case 2 due to the large gap between the film orifices so that the coolant cannot cover the inner wall as case 0. However, the more mass flow rate of coolant in a single film orifice makes the less wall heat flux in large orifice spacing where the injector is arranged.
5.5 Effects of diameter

Figure 17 shows the variations in the axial direction for the cases with different coolant orifice diameters. As shown in Fig. 17, the coolant orifice with the largest diameter (case 7) shows the best upstream cooling effectiveness and the worst downstream cooling effectiveness. The variations in turbulence intensity along the axial direction of three different diameter conditions are compared to demonstrate this trend (see Fig. 18). Because the flow velocity of film is slower and the turbulence intensity decreases sharply in the large diameter case, the higher cooling performance is observed for case 7 in the near injection region. The effectiveness in case 6 is better than those in case 7 and case 0 in the downstream cylinder section of the combustion chamber, because the increased initial momentum and the long film protection distance are developed in the condition of small diameter.

The H\(_2\) mole fraction distribution in different cross sections for case 6 and case 7 are shown in Fig. 19. The adopted film cooling induces the hydrogen concentrated layer near wall, and the high H\(_2\) mole fraction illustrates the obvious effect of film cooling flow in the neighborhood of head plate. The H\(_2\) mole fractions in case 7 (\(D=1.6\)mm) throughout the cross section \(x=12\)mm and \(x=50\)mm are higher than that in case 6 (\(D=0.8\) mm), but this magnitude relation is reversed from \(x =200\) to 300 mm. The high H\(_2\) mole fraction at those cross sections is the explanation for the higher downstream effectiveness in case 6. For all cases, the H\(_2\) mole fraction decreases along the hot gas flow direction because of the mixing and reaction. At the cross section in the nozzle, the distribution of H\(_2\) becomes to be almost uniform, while the H\(_2\) mole fraction in case 6 is higher than that in case 7.
6. Conclusion

The film cooling in a LOX/GH$_2$ thrust chamber has been simulated numerically with real gas EOS model. The flow and coupled heat transfer between the hot gas, film coolant, cooling channels and regenerative coolant are considered. The numerical methodology adopted was validated using experimental data. The effects of coolant flow rate, film orifice spacing and diameter on cooling performance and heat loads are compared and analyzed. The results show the following conclusions:

1. The reverse flow regions are found between the exits of the injectors and at the coolant jet exits adjacent to the wall and center. These structures enhance the mixing and coupled heat transfer between the hot gas and the film coolant in the combustion chamber.

2. The introduction of film cooling in the near injector region reduces the hot-gas-side wall temperature substantially. The effects of the pseudo-boiling behavior in the near-injector region are obvious. The film cooling has a great influence on the wall in the entire combustion chamber other than in the nozzle. The sufficiently high coolant concentration near the wall would extremely reduce the local maximum temperature of the hot-gas-side wall close to coolant exits.

3. There is an appropriate mass flow rate that makes the overall cooling performance optimal. More coolant is available at the downstream under the condition with the large coolant flow mass rate which causes higher cooling effectiveness. However, the heat exchange between the coolant and the hot gas is increased and cooling effectiveness is reduced near the head plate when the mass flow rate is increased.

4. The advantage of uniform distribution of heat flux at small orifices spacing is observed, which results in a better overall cooling performance. However, the downstream cooling effectiveness in the case with 15° orifices spacing is the highest. For the multi-injector plate widely used in the thrust chamber of the LRE, the uniform distributions temperature may lead to the increase in the operational life because of the decrease in the thermal tress.

5. In the vicinity of injectors, the case with large diameter orifices is characterized by high H$_2$ mole fraction and low turbulence intensity which brings about the improved cooling performance. During the design of film cooling, large diameter could be considered to compensate for the lack of cooling capacity of regenerative coolant in the near injection region.

6. The methodology presented in this paper could accurately and comprehensively obtain three dimension coupled flow and temperature fields in the LOX/GH$_2$ thrust chamber, which has the potential to support the optimal design of straight cylindrical coolant orifice configuration in the entire thrust chamber of the LRE.

Nomenclature

- $a$ $b$: coefficients of SRK EOS
- $c_p$: constant pressure heat capacity, J/(kg K)
\[D \quad \text{film cooling orifice diameter, m}\]

\[h \quad \text{enthalpy, J/kg}\]

\[k \quad \text{enthalpy, J/kg}\]

\[m \quad \text{Mass flow rate, kg/s}\]

\[M_w \quad \text{turbulence kinetic energy, m}^2/\text{s}^2\]

\[p \quad \text{pressure, Pa}\]

\[Pr \quad \text{Prandtl number}\]

\[q \quad \text{heat flux, W/m}^2\]

\[R \quad \text{universal gas constant}\]

\[S \quad \text{source term}\]

\[S_c \quad \text{Schmidt number}\]

\[T \quad \text{temperature, K}\]

\[u \quad \text{velocity, m/s}\]

\[V \quad \text{specific molar volume, m}^3/\text{kmol}\]

\[x \quad \text{axial distance from head plate, m}\]

\[\beta \quad \text{orifice spacing angle, } ^\circ\]

\[\mu \quad \text{dynamic viscosity, Pa s}\]

\[\rho \quad \text{density, kg/m}^3\]

\[\omega \quad \text{acentric factor}\]

\[\lambda \quad \text{thermal conductivity, W/(m K)}\]

\[\epsilon \quad \text{turbulence energy dissipation rate, } 1/\text{s}\]

\[\eta \quad \text{film cooling effectiveness}\]

\[\tau \quad \text{stress tensor, Pa}\]

Subscripts

\[0 \quad \text{without film}\]

\[c \quad \text{critical}\]

\[\text{dep} \quad \text{departure value}\]

\[\text{coolerant} \quad \text{film cooler}\]

\[\text{film} \quad \text{film cooling}\]

\[\text{ideal} \quad \text{ideal gas}\]

\[t \quad \text{turbulence}\]

\[\text{wg} \quad \text{hot-gas-side wall}\]

References


Soave, G., Equilibrium constants from a modified Redlich-Kwong equation of state, Chemical Engineering Science, Vol. 27, No. 6 (1972), pp.1197-1203.