LARGE EDDY SIMULATION OF TRANSONIC FILM COOLING WITH INJECTION THROUGH CYLINDRICAL ORIFICES INCLUDING SHOCK-FLOW INTERACTION

Hao Guo  Peixue Jiang  Yinhai Zhu*
(Key Laboratory for Thermal Science and Power Engineering of Ministry of Education, Key Laboratory of CO2 Utilization and Reduction Technology, Department of Energy and Power Engineering, Tsinghua University, Beijing 100084, China)
(*Corresponding author; Email: yinhai.zhu@tsinghua.edu.cn)

ABSTRACT
Laminar air coolant injected into a fully turbulent preliminary stream is analyzed using Large Eddy Simulation. Three freestream Mach numbers, 0.8, 1.2 and 1.4, are investigated. Regions of typical shock-flow interaction in supersonic film cooling are distinguished into 3 types, including shock generation, shock reflection and shock absorption. Numerical results show that the separation/bow shock promote the secondary flow to adhere to the protected wall and widen its transverse coverage. The mechanism of near field crossflow is different from that of far field, which requires disparate forms of corresponding empirical correlations.

INTRODUCTION
The development of advanced engines with higher thrust-to-weight ratio has been the main object of aeronautics and astronautics for decades. The high-pressure-stage configuration of advanced aero-engines has evolved from the subsonic stationary and moving blades to the single supersonic/transonic row of stationary or moving blades, and is developing towards all supersonic/transonic blades. The shock-cooling-structure interaction is ubiquitous in the flow field due to complicated flow behaviour arranged on blades. It is of great significance for guiding the design of aero-engine to study the flow and heat transfer mechanism of film cooling in transonic/supersonic freestream with shock wave interference.

Many experimental and numerical studies have been conducted on the supersonic film cooling of a two-dimensional-slot structures. Seban et al. (Seban and Back, 1962) divided the mixing of slit tangential jet and mainstream into potential core region, wall jet region and boundary layer region. Juhany's research team (Juhany et al., 1994) carried out detailed experiments on supersonic film cooling with a mainstream Mach number of 2.44 and studied the effects of gas type, blowing ratio and jet Mach number on cooling characteristics. They found that the cooling efficiency could be improved by increasing the blowing ratio of both air and helium jets. The effective cooling length of the gas film can be slightly increased by lifting the jet Mach number except for high-temperature helium jets. Juhany et al. (Juhany and Hunt, 1994) also studied the interference of oblique shock wave on gas film under the same condition. They used hanging wedges in wind tunnels to generate oblique shock waves that incident in the boundary layer region. The results show that the separation region of helium film induced by shock wave is smaller than that without jet, while the separation region of air film is slightly larger. Kanda et al. focused on the study of supersonic gas film cooling with shock wave interference. They explored the influence of the shock strength (Kanda et al., 1996a) and the incident position (Kanda et al., 1996b). Various types of RANS model are widely used in the simulation of slotted structures (Auipoix et al., 1998; O'Connor and Haji-Sheikh, 1992; Peng and Jiang, 2009; Peng and Jiang, 2014; Peng et al., 2017). Konopka et al. carried out the corresponding LES study for the experimental conditions of Juhany's work. They discussed the effects of oblique shock waves incident in the potential core region or the boundary layer region respectively (Konopka et al., 2012), and the cooling characteristics of helium and hydrogen jets with or without shock wave interference (Konopka et al., 2013).

Although many studies have been carried out on the two-dimensional-slot structure, film cooling technology is applied to the turbine blades through the arrangement of orifices rows in the application. Fric et al. (Fric and Roshko, 1994) distinguished several typical vortex structures in detail through flow field visualization techniques: count-rotating vortex pairs (CVPs), horseshoe vortices, shear layer vortices and wake vortices. Kelso et al. (Kelso et al., 1996) combined flow visualization techniques with flying hot wire to explain how the jet shear layer affects the CVPS...
circulation through bending and folding. Peterson et al. (Peterson and Plesniak, 2004) adopted particle image velocimetry (PIV) to study the complete evolution process of the secondary flow from the supply channel to the through-hole and finally to the main flow. They found that different gas supply directions can strengthen or weaken the CVPs in the crossflow.

As long as either jet or freestream reaches supersonic state, shock waves will appear in the flow field. Gruber et al. (Gruber et al., 1997) experimentally studied the jets in crossflow (JICF) problem of sonic jet and supersonic freestream. They found that there was no significant difference in the transverse jet penetration among different gas types, which, however, would produce completely different compressible levels that greatly affect the characteristics of large scale structures in the shear layer as well as the entrainment and mixing between jet and freestream. By comparing the work of Gruber et al. and Juhany et al., it was noted that results of two-dimensional parallel jets and three-dimensional transverse jets are lack of comparability even though both studies are carried out on air and helium. Chenault et al. (Chenault et al., 1999) found that RANS models using Boussinesq assumption had significant errors in the calculation of turbulence parameters such as Reynolds stress. Many researchers began to use LES or mixed LES-RANS methods for numerical research (Chai and Mahesh, 2011; Peterson et al., 2006), which can provide more accurate qualitative and quantitative characterization of complex coherent structures. Most of the current studies focus on describing the flow field structure and the shock wave shape itself but few among them investigate the shock influence mechanism on cooling film in detail.

In this study, Large Eddy Simulation (LES) was carried out on the film cooling of transonic freestream through cylindrical orifices to explore the different influence mechanism of compressible effect and shock wave system, and to study the interaction law of various types of shock wave-cooling structure. The second section first introduces the physical model and numerical method used in this work. Then in the first part of the third section, various kinds of coherent structures and wave systems in transonic film cooling are studied; in the second part, the influence of main flow transonic condition on the film cooling characteristics is discussed in detail.

**METHODOLOGY**

**PHYSICAL MODEL**

The physical model adopted in this paper is a rectangular channel containing an inclined circular through hole, as shown in Figure 1. The size of the mainstream domain \((L \times H \times W)\) is \(30D \times 10D \times 3D\), where \(D\) is the diameter of cylindrical orifice, of which the inclined angle is 37°. The computational domain can be reduced based on orifice spacing since the orifice row are periodic structures along the spanwise direction. The final effective domain is marked of grey in Figure 1.

Three Mach number of high-temperature crossflow is selected, namely, 0.8, 1.2 and 1.4, to represent the high Mach subsonic flow and the supersonic flow with weak/strong shock, respectively. See Table 1 for details of other parameters. Using the blowing ratio \(BR\) for analysis, the jet Mach number can be calculated according to the blowing ratio by the following formula:

\[
Ma_i^2 = \left[1 + 2(C(\gamma - 1)Ma_i^2 - \frac{\gamma - 1}{2}Ma_i^2)\right]^{-\frac{\gamma}{\gamma - 1}} - 1
\]

(1)

![Figure 1 Schematic of Computational Geometry.](image)

**Table 1** Detailed Parameters of Transonic Film Cooling

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Values</th>
<th>Parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>(p_{i,\infty})</td>
<td>3.35MPa</td>
<td>(T_{i,\infty}/T_{i,\infty})</td>
<td>0.47</td>
</tr>
<tr>
<td>(T_{i,\infty})</td>
<td>1848K</td>
<td>(BR=\rho u_i/\rho u_{\infty})</td>
<td>1.0</td>
</tr>
</tbody>
</table>
NUMERICAL METHOD

Favre filtered Navier-Stokes equations describing the conservation of mass, momentum and energy of the compressible fluid are

\[ \partial_t \rho + (\rho u_j)_j = 0, \quad (2) \]

\[ \partial_t (\rho u_j) + (\rho u_i u_j)_j + \partial_j p - \partial_i \sigma_{ij} = -(\rho u_i u_j - \rho \bar{u}_i \bar{u}_j)_j - (\sigma_{ij} - \bar{\sigma}_{ij})_j, \quad (3) \]

\[ \partial_t (\rho h) + (\rho h u_j)_j - \partial_j \bar{p} - \partial_i \bar{\sigma}_{ij} + \partial_j \hat{\Phi} = -(\rho u_i h - \rho \bar{u}_i \bar{h})_j + (u_i u_j - \bar{u}_i \bar{u}_j)_j - (\bar{h}_j - \hat{\phi}_j)_j + (\hat{\Phi} - \hat{\phi}), \quad (4) \]

where \( \phi \) and \( \hat{\phi} \) are the physical quantities after direct filtering and Favre filtering respectively. In order to accurately capture the transition process in the flow field and the separation and re-adhesion process of the boundary layer in the near-wall region, the WALE model (Nicoud and Ducros, 1999) is adopted to model the subgrid stress provided that Boussinesq hypothesis is valid (Boussinesq, 1877).

Non-slip adiabatic boundary condition (B.C.) is used on the upper and lower walls. Periodic B.C. is adopted in the spanwise direction to reduce computational domain. The upcoming high temperature freestream is fully turbulent boundary layer flow while the air jet is laminar. The Generation of inlet turbulence pulsation adopts the Random Flow Generation (RFG) method that first proposed by Kraichnan (Kraichnan, 1970) and then improved by Smirnov et al (Smirnov et al., 2001).

The LES method used in this paper adopts a classical box filter to filter the physical space implicitly through a computational grid. The NVD-based Gamma differencing scheme, which based on the normalized variable diagram (NVD) approach (Leonard, 1991) together with the convection boundedness criterion (CBC) (Leonard, 1991), is used to discretize advection term. Spatial discretization of diffusion term uses a second-order-accurate central-differencing discretization scheme. Temporal discretization adopts a bounded version of Second Order Implicit Euler scheme. The time step selected meets the requirement that the CFL number of the whole field is less than 1. A converged high-precision RANS solution with the same B.C. is generated as initial condition for computation. The statistical average was carried out with a duration about 10 characteristic times after the calculation reached the quasi-stationary state.

MESH TOPOLOGY

The O-block strategy is adopted to ensure a good transition between the boundary layer inside the hole and the grid of the core region. Local mesh encryption was applied in the area close to the wall and with large difference in topological structure, as shown in Figure 2(a) and Figure 2(b). The LES used in this paper directly resolve the flow in the near-wall region through the filtered governing equations without using any wall functions, so the \( y^+ < 1 \) condition is strictly guaranteed on both the upper and lower walls.

RESULTS AND DISCUSSION

FLOW BEHAVIOUR AND SHOCK SYSTEM OF TRANSONIC FILM COOLING

In incompressible JICF researches, it is generally believed that there exist shear layer vortices on the windward side, counter-rotating vortex pairs (CVPs), horseshoe vortices and wake vortices in transverse jets when the velocity ratio VR
is below 1. However, the shear layer vortices and the CVPs are not independent vortex structures but the projections of three-dimensional hairpin vortices on different cross sections. Figure 3 shows flow behaviours after the secondary flow enters the main stream and the characteristics of jet mixing and spreading. Horseshoe vortices and wake vortices can still be observed clearly in the strongly compressible crossflow, which is basically the same as in incompressible cases. The wake vortex is a tentacle-like structure derived from the hairpin vortex in the Figure 3, which is most significant in the downstream of jet incidence, and gradually breaks up and disappears in the process of secondary flow reattachment. Further study shows that the projected position of the hairpin vortex in Figure 3 on the spanwise section is basically the same as that of the shear layer vortex.

Figure 3  Coherent Structures in JICF of 0.8 Mach

The adiabatic cooling effectiveness shown in the contour of protected wall in Figure 3 is defined by the following equation:

$$\eta = \frac{T_{r,i} - \langle T_{sw} \rangle}{T_{r,i} - T_{r,i}},$$

(6)

where $T_{r,i}$ and $T_{r,i}$ are the recovery temperature of the inlet of the main stream and secondary flow, respectively. The recovery temperature is defined as:

$$\frac{T_r}{T} = 1 + r \frac{\gamma - 1}{2} \text{Ma}^2,$$

(7)

where $r$ is the recovery factor, which is generally considered as $Pr^{1/3}$ and $Pr^{1/2}$ in turbulent and laminar flow respectively. The wall cooling effectiveness distribution will be further discussed in the next section.

Figure 4  Numerical Schlieren of Shock System at Different Mach Numbers

Figure 4(a) and Figure 4(b) show the calculation results of two supersonic cases respectively. 1.4 Mach case demonstrates general shock structures in supersonic film cooling: the Mach disk in the downstream of jet overexpansion, separation shock and bow shock because of flow bend and separation, and an oblique shock reflected by channel at the top wall. These shock systems occurring in supersonic film cooling can be divided into three categories: shock generation, shock reflection and shock absorption, which are shown as region (a), (b) and (c) in Figure 4 respectively. The secondary flow injection causes the main flow to deflect and produces the bow shock wave with three-dimensional structure around the film hole. At the same time, the upstream of the injection induces the flow separation and generates the separation shock in front of the bow shock. The intensity of the separation shock is weaker than that of the bow shock wave, and it can be regarded as isentropic when the mainstream Mach number is close to the speed of sound.
COOLING CHARACTERISTICS OF INJECTED FILM IN TRANSONIC FLOW

Calculation results of the momentum and energy transfer on each flow section are shown in Figure 5. Because of the symmetry of statistical mean field, only one side of the result is shown in the figure. The vector arrow length only represents the relative size of the physical quantity on each sub-graph. The left part of single figure shows the contour of dimensionless statistical average velocity in the x direction together with arrows representing the average velocity vector components in the cross section; the right part shows the contour of dimensionless recovery temperature together with arrows representing turbulent temperature transport vector components in the cross section. As shown in Figure 5, the energy transport between mainstream and secondary flow is stronger than the momentum transport, and it is turbulent motion rather than average motion that dominates energy transport in the crossflow. The development of the effective cooling range of the gas film defined by the recovery temperature is obviously faster than the development of the CVPS: when the secondary flow in the left figure is still sucking the mainstream fluid on both sides through CVPS or horseshoe vortices at some locations, the recovery temperature on the right side has basically completed the heat transfer, as shown in Figure 5(a), (c) and (e).

Figure 5  Momentum and Energy Transport at Cross Plane in Flow Direction: (a) 0.8 Mach, x/D=2; (b) 0.8 Mach, x/D=10; (c) 1.2 Mach, x/D=2; (d) 1.2 Mach, x/D=10; (e) 1.4 Mach, x/D=2; (f) 1.4 Mach, x/D=10.

Shock wave generation mainly affects the coherent structure in the near field. Compared with Figure 5(a), jets in the near field in Figure 5(c) and (e) have a lower core and a wider spreading, which indicates that the emergence of separate shock and bow shock make the jet spreading uniformity under supersonic condition significantly stronger than that under subsonic condition as well as reducing the height of jets. By comparing Figure 5(c) and (e), it can be found that the compressible effect enhanced by the increase of Mach number is shown as the overall strengthening of CVPS. As the effective cooling range and effective cooling length of the gas film in the far field area expand, it also intensifies the lifting and weakens the adhesion ability. This shows that shock wave and compressibility effect have different mechanism influencing the wall protection characteristics of gas film.

The whole cooling process of jets contains two stages: the free shear mixing between mainstream and secondary flow, the separation re-attachment of secondary flow. The former influences physical parameters of film while the latter directly determines the wall cooling effectiveness of the distribution. Figure 6 shows the distribution of adiabatic wall
cooling effectiveness calculated by Equation (6) under three working conditions. As mentioned above, the jet-wall interaction region in the figure can be clearly distinguished into the near field and the far field.

The cooling characteristics in the near field are dominated by shocks and coherent structure. When both the main flow and secondary flow are subsonic, there is no separation shock wave or bow shock wave in the upstream of the film hole. At this point, the backflow area caused by the upstream flow separation is large, and the entrapping and mixing effects of horseshoe vorticity are strong, so that the cooling airflow forms an arrow-shaped distribution on the wall surface as shown in Figure 6(a). The upstream of the through hole sucks the low-temperature fluid in the downstream to expand the effective cooling area, while the high-temperature main stream rapidly heats the jets on both sides of the orifice. When the mainstream become supersonic flow, the emergence of separate and bow shock limits the area and intensity of horseshoe vorticity (see Figure 5). In Figure 7(a), the variation law of peak cooling efficiency at different mainstream Mach numbers in the near field region is consistent, while the average cooling efficiency in the supersonic case is significantly higher than that in the subsonic case, which indicates that shocks do promote the spreading of the air film along the spanwise direction.

**CONCLUSIONS**

In this paper, transonic film cooling under the influence of shock waves was studied by Large Eddy Simulation. The main conclusions are as follows:

1) The temperature transport and momentum transport in the strongly compressible crossflow are bidirectional coupled, and the heat transfer of the jet mainly depends on the turbulent temperature transport rather than the average motion advection. When the flow mixing is still consistent with the large-scale vortex structure, the temperature boundary is significantly different from that of the flow dominant vortex and is basically perpendicular to the turbulence temperature transport vector.

2) The mechanism of transonic film cooling is different in the near field and the far field. The former that influenced by multiple types of coherent structures is close to the orifice outlet and is susceptible to the separation/bow shock. The
latter is mainly marked by flow mixing dominated by CVPs. At this time, the mixing range begins to shrink gradually and the cooling efficiency decreases monotonously.

3) When the crossflow is in supersonic condition, the flow deflection and backflow caused by the jet will lead to the separation shock and the bow shock near the orifice. The shock waves generated by the flow fields themselves will inhibit the lifting of the air film and promote its spreading, so that the downstream separation area of the jet decreases and both peak and average cooling effectiveness are improved.

**NOMENCLATURE**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Subscripts</th>
<th>Superscripts</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D$</td>
<td>orifice diameter</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$h$</td>
<td>enthalpy</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$Ma$</td>
<td>Mach number</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$MR$</td>
<td>blowing ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$p$</td>
<td>pressure</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$r$</td>
<td>recovery factor</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$u, v, w$</td>
<td>streamwise, wall-normal, and spanwise velocity components</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$VR$</td>
<td>velocity ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\eta$</td>
<td>cooling effectiveness</td>
<td></td>
<td>direct filtered</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
<td></td>
<td>~ Favre filtered</td>
</tr>
</tbody>
</table>

**APPENDIX A - COPYRIGHT/OPEN ACCESS**

The GPPS policy is that all articles will be Open Source accessible. This article will be published using the Creative Commons Attribution commercial rights licence [CC-BY-NC-ND 4.0](https://creativecommons.org/faq/), thus allowing the author(s) to retain their copyright.

For answers to frequently asked questions about Creative Commons Licences, please see [https://creativecommons.org/faq/](https://creativecommons.org/faq/).

**APPENDIX B - GPPS PRESENTER POLICY AND PAPER ACCEPTANCE**

According to GPPS's presenter attendance policy, a paper cannot be published or be indexed and may not be cited as a published paper until at least one author pays the registration fee and attends the conference. The GPPS reserves the right to withdraw from its publications any paper that is not presented by an author of the paper at the appropriate conference. Any paper that is withdrawn may not be cited as a published paper.

**REFERENCES**


