NUMERICAL SIMULATION OF IMPINGING COOLING ON THE LEADING EDGE OF A TURBINE BLADE

Keyong Cheng
Institute of Engineering Thermophysics, Chinese Academy of Sciences
Graduate University of Chinese Academy of Sciences
Beijing 100190
China
chengkeyong@iet.cn

Xiulan Huai
Institute of Engineering Thermophysics, Chinese Academy of Sciences
Beijing 100190
China
hxl@mail.etp.ac.cn

Jun Cai
Institute of Engineering Thermophysics, Chinese Academy of Sciences
Beijing 100190
China
sc_caijun@163.com

Zhixiong Guo
Department of Mechanical and Aerospace Engineering, Rutgers, The State University of New Jersey
NJ 08854
USA

ABSTRACT

In the present study, numerical simulation is carried out for impingement/effusion cooling on the leading edge of a turbine blade similar to an experimental model tested previously. The $k$-$\varepsilon$ turbulence model is used, and simulation parameters are set in accordance with the experimental conditions, including temperature ratio, blowing ratio, and Reynolds number of the main stream. The accuracy and reliability of the simulation is verified by the experimental data, and the influence of various factors on fluid flow and heat transfer is analyzed in detail. The results indicate that the blowing ratio is one critical factor which affects the cooling effectiveness. The greater the blowing ratio is, the higher the cooling effectiveness is. In addition, a staggered-holes arrangement is numerically studied and compared with a line-holes arrangement. The results show that the staggered-holes arrangement has a lower temperature on the outer surface of the leading edge and has improved the cooling effectiveness.

INTRODUCTION

A thermodynamic analysis on gas turbines shows that thermal efficiency and power output can be increased with increasing turbine inlet temperature [1] far above the permissible metal temperature [2]. At present, such a high operating temperature has made the cooling of blades indispensable to extend the component life and reduce relevant maintenance costs. Currently, turbine blades are cooled both internally and externally, including film cooling, impingement cooling, rib-turbulated cooling and pin-fin cooling [3]. Film cooling and impingement cooling are utilized together as impingement/effusion cooling, which is also one of the cooling techniques used to protect the blade surface from external hot gas stream.

Film cooling represents one of the few changing technologies that have allowed the achievement of today’s high firing temperature, high-efficiency gas turbine engines [4]. Therefore, many investigations have been conducted to improve the performance of film cooling, focusing on injection angles of cooling stream, injection hole configuration, blowing ratios and so on. Kercher [5] presented an exhaustive list of film cooling literatures which collectively described the continual emerging of numerical film cooling as a viable design tool for high temperature components. Bunker [6] provided an excellent overview of literatures involving the shaped film cooling holes, and examined the origins of shaped film cooling and summarized the extant literature knowledge concerning the performance of such film cooling holes. Kusterer [7] investigated high film cooling effectiveness as a result of a well-designed cooling hole arrangement for interaction of two neighboring cooling jets. The results showed that improved film-cooling effectiveness can be reached because an anti-
kidney vortex pair is established in the double-jet. Heidmann [8] proposed an anti-vortex turbine film cooling hole concept. Unlike the shaped film cooling holes and other cooling structure, it only requires easily machinable round holes. Detailed flow visualization showed that as expected, the design could counteract the detrimental vorticity of the round holes flow, allowing it to remain attached to the surface. Sargison [9] experimentally studied the performance of the converging slot-hole or console film-cooling hole geometry. The console film-cooling hole geometry offers advantages to the engine designer due to a superior aerodynamic efficiency over the fan-shaped hole geometry. Jia et al. [10] conducted numerical simulations and experiments on a slot film cooling configuration with various blowing ratios and angles. Their results showed that the 30-deg jet could provide the highest film cooling effectiveness in all the tested blowing angles, and the appropriate blowing ratio was around 1.0. Bunker [11] conducted experiments to examine two film cooling geometries which were formed by the combination of internal discrete film holes feeding continuous 2D surface slots. The holes-within-slot film effectiveness data were compared with both the axial and radial film effectiveness data obtained in the same test section.

Impingement cooling is a technique commonly used to increase the internal heat transfer. In particular, it is suitable for the leading edge of a stator/rotor airfoil where the thermal load is high and a thicker cross-section enables accommodation of a coolant plenum and impingement holes. Many experimental and theoretical investigations have been carried out to better understand the flow and heat transfer characteristics of impingement cooling. Downs and James [12], Jambunathan et al. [13] and Viskanta [14] reviewed studies on jet impingement heat transfer. The effects of jet-jet spacing, low nozzle-plate spacing and spent air exits located between the jet orifices were studied by Huber and Viskanta [15] on the magnitude and uniformity of heat transfer coefficients (HTC) for air jets impinging normally to a heated surface. The results showed that the jet-jet spacing affected HTC by varying the influence of the adjacent jet interference and the fraction of the impingement surface covered by the wall jet; and the addition of spent air exits increased HTC and influenced the location of the optimum separation distance. Cho and Goldstein [16, 17] presented the heat and mass transfer characteristics of the inner surface of a circular film cooling hole and external wall. Complex heat and mass transfer was observed inside the film cooling hole due to the separation and reattachment of the coolant flow. Ekkad et al. [18] proposed detailed heat transfer distributions for an array of jets impinging on a target plate with a staggered array of film cooling holes. The impingement/effusion cooling experiment between two parallel perforated plates were investigated by Cho et al. [19] for varying gap distance between the perforated plates of 0.33 to 10 hole diameters. Higher heat and mass transfer was obtained with a smaller gap distance. Rhee et al. [20] and Cho et al. [21] conducted the experiments for three different hole arrangements, i.e. staggered, square, and hexagonal, with various gap distances, and further investigated the local heat

(mass) transfer characteristics of flow through perforated plates. The results indicated that the staggered arrangement had higher heat transfer rates than other cases. Lee et al. [22] measured the cooling effectiveness on full-coverage film cooled surface with and without array jet impingement cooling by using an infrared thermography technique. The results showed that the effect of H/D was not significant, and the cooling effectiveness with the impingement jets increased significantly for the stainless steel plate.

Many previous studies on impingement/effusion cooling have been focused on the solid plates. Quite few investigations have been carried out on turbine blades. No literature was found in systematic investigation of both the internal impingement cooling and external film cooling by using experimental and numerical approaches. Recently, You et al. [23] conducted an impingement/effusion cooling experiment on the leading edge of a turbine blade. As an extended work, the present study focuses on numerical simulation. The accuracy and reliability of the simulation are verified by the experimental data, and the influences of various factors on the fluid flow and heat transfer are analyzed. In addition, the effect of a staggered-holes arrangement on the cooling effectiveness is numerically studied; and a comparison with a line-holes arrangement is also conducted.

**NUMERICAL SIMULATIONS**

**Geometry**

As shown in Fig.1, the simulated section is a rectangular channel, with the dimensions of 100mm × 80mm × 80mm (in X, Y, and Z directions). Within the channel, there are a half-cylinder and a jet pipe.
Figure 2 (a) shows the cross section of the half-cylinder. There are four rows of φ3×6 cylindrical film cooling holes with symmetrically distributed. With regard to the edge stagnation line (symmetrical axis), the film hole rows are in 20°, 60°, -20° and -60° directions, and defined as the first, second, third, and fourth row, respectively. There is a jet pipe with jet holes in the leading edge’s inner cavity, as shown in Fig.2 (b). On the wall of the jet tube, there are three rows of jet holes (φ2.5×8). High-pressure cooling air is jetted through these jet holes to impinge the inner surface of the cavity, and then outflows from the film cooling holes and mixes with the main stream. In Fig.2 (a), the stagnation line of the leading edge is defined as “Location 1”, and the regions between the first and second hole rows and between the third and fourth hole rows are defined as “Location 2”, and the regions on the edge of the leading edge are defined as “Location 3”.

To characterize the effect of different film-hole arrangements on heat transfer, two different cases are considered in the present study. In Case 1, a line-holes arrangement is used in accordance with the work done in the literature [23], as shown in Fig.3 (a). Case 2 simulates a staggered-holes arrangement. As shown in Fig.3 (b), the first and second hole rows are staggered, and so are the second and fourth hole rows.

Data treatment and simulation conditions

The blowing ratio is expressed as

\[
m = \frac{\rho_2 u_2}{\rho_1 u_1}
\]

Here, \( \rho \) and \( u \) are the density and velocity, respectively; and subscripts 1 and 2 represent the main stream and cooling air, respectively.
The cooling effectiveness is an important parameter to evaluate cooling effect. Because the simulations involve the composite effect of multiple cooling events, the composite cooling effectiveness is defined as follows:

\[
\eta = \frac{T_g - \overline{T_w}}{T_g - T_i}
\]  

(2)

Here, \(\overline{T_w} = (T_{wo} + T_{wi}) / 2\), and \(T_{wo}\) is the temperature of outer wall of turbine blade and \(T_{wi}\) is the temperature of inner wall. \(T_g\) and \(T_c\) are the temperatures of main stream and cooling air, respectively.

The conditions for numerical simulations are listed in Table 1.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Ranges</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reynolds number of the main stream, (Re)</td>
<td>24000 ~ 39353</td>
</tr>
<tr>
<td>Blowing ratio, (m)</td>
<td>0.5 ~ 2.0</td>
</tr>
<tr>
<td>Temperature of the main stream, (T_g)</td>
<td>324.5 K ~ 529.875 K</td>
</tr>
<tr>
<td>Temperature of the cooling air, (T_c)</td>
<td>290 K ~ 295 K</td>
</tr>
<tr>
<td>(T_g/T_c)</td>
<td>1.1 ~ 1.8</td>
</tr>
<tr>
<td>Flow rate of the main stream</td>
<td>0.02604 kg/s ~ 0.042 kg/s</td>
</tr>
<tr>
<td>Flow rate of the cooling air</td>
<td>0.00036 kg/s ~ 0.00275 kg/s</td>
</tr>
</tbody>
</table>

The leading edge used in the present study is axisymmetric, and therefore, only half of the geometry structure is simulated. Periodic and symmetry conditions are applied to minimize the computational effort. Unstructured tetrahedral meshes are used to perform the numerical simulation and a prism layer is constructed to attach to the leading edge. Adiabatic conditions are usually adopted as boundary conditions on blade surfaces in many previous simulations studies. In reality, however, there exists heat conduction from the outer to the inner wall of the turbine blades. Therefore, a conjugate heat transfer solver method has been developed for the simulation on the leading edge in the present study. Adiabatic and no-slip conditions are used for the other walls. A pressure boundary condition is specified at the exit of the main stream flow channel. A wall-function method is taken in the near wall region. The equations are solved by adopting the first-order resolution method. The convergence criterion of the solutions is that the relative residuals of the physical quantities are less than 10^{-6}. All the predictions are done with the \(k-\varepsilon\) turbulence model.

**Grid independency**

Figure 4 shows the meshes which are made up of unstructured grids. From the figure, it is obviously seen that prism layers are constructed to attach to the leading edge. Prior to the actual numerical simulation, a grid independency test for the \(k-\varepsilon\) model is performed by using different grid numbers with 1034425, 1440123, 2001060, 2615692 and 2961502 cells, respectively. The simulation parameters are: \(Re = 26338\), \(T_c/T_g = 1.1\), and \(m = 0.8\). The cooling effectiveness is calculated by averaging the temperature of location.

As shown in Fig.5, the maximum variety of predicted film cooling effectiveness is 2.18% as the grid number increases from 1034425 to 1440123, and 2.14% from 1440123 to 2001060, and 1.89% from 2001060 to 2615692, while a minimum of 0.82% is observed as the grid number increases from 2615692 to 2961502. Therefore, a compromise between computation accuracy and computing capability leads to the use of 2615692 grids. It is noted that the first layer grid points adjacent to all the bounding wall surfaces are spaced within the
value of y+ less than 11 in order to meet the requirements for the k-ε turbulence model.

RESULTS AND DISCUSSION

Distributions of temperature and velocity

Figure 6 shows the temperature distributions on the X-Y plane (Z/D = 7.6) with different blowing ratios. In Fig. 6(a), the temperatures for the solid wall and gas in the cavity are about 420 K and 350 K at $m = 0.5$, respectively. However, the temperatures are about 385 K and 315 K at $m = 0.8$ and approximately decreased 35 K, as shown in Fig. 6(b). The blowing ratio is the main reason causing the different temperatures. According to Eq. (1), the velocity of the cooling air is improved with the increase of the blowing ratio. If the velocity of the cooling air is too low, the main stream will suppress cooling air and flow into the inner cavity through the film holes, which cause the rising of the temperature in the leading edge. This situation can be demonstrated by Fig. 6(a) and Fig. 7(a). On the other hand, if the velocity is high enough, cooling air can fight against main stream and form a protective layer on the outer surface of the leading edge. This point can be also demonstrated by Fig. 6(b) and Fig. 7(b). Based on the analysis above, it can be concluded that the cooling air takes effect on protecting the outer surface of the leading edge when the blowing ratio is larger than 0.8. Therefore, the blowing ratio is taken in the range of 0.8 ~ 2.0 in the present study.

Effect of temperature ratio on cooling effectiveness

On the basis of Eq. (1), the cooling effectiveness can be also expressed as

$$
\eta = \frac{T_g - T_w}{T_g - T_c} = \frac{T_g}{T_c} - 1
$$

(3)

By taking a derivation for $\eta$, it can be found that only

$$
\frac{d(T_g)}{T_c} > \frac{T_g}{T_c} - 1
$$

when

$$
\frac{d(T_g)}{T_c} \frac{T_g}{T_c} - 1
$$

when $\frac{d(T_g)}{T_c} < 0$, the cooling effectiveness decreases with increasing $T_c/T_g$, which means that $T_w$ increases faster than $T_g$. An analysis from the physical angle shows that, with the increase of the cooling air temperature, the density of the cooling air decreases and the velocity increases, which causes the impingement velocity to increase and the wall temperature to decrease (the cooling effectiveness increases).

Hu et al. [24] concluded the conclusion that the cooling effectiveness slightly decreases with increasing $T_c/T_g$ when they studied the heat transfer of guide vane. Actually, the change of the cooling effectiveness with the temperature ratio is associated with heat transfer intensity of the internal and
external flows. When the external flow predominates as compared to the internal flow, the same conclusion can be drawn in the present study.

Figure 8 shows the variation of the cooling effectiveness with $T_g/T_c$, when Reynolds number of the main stream is 24000. The cooling effectiveness is calculated from the average temperature of the outer and inner surfaces. From the figure, it is seen that the predicted cooling effectiveness has a slight decrease with increasing $T_g/T_c$. The predicted curves are more flat than the experimental ones. When $T_g/T_c$ increases from 1.1 to 1.8, the predicted cooling effectiveness drops only about 6.2% at $m = 0.8$, and about 2.4% at $m = 1.1$. Therefore, it can be concluded that $T_g/T_c$ is not a main factor affecting the cooling effectiveness. In Fig.8, there are obvious differences between the simulation results and the experimental ones. The reason is that, in the previous experiment [23], the thermocouples were welded onto the inner surface point by point and could not accurately indicate the temperature of the leading edge inner surface. Therefore, the experimental results are greater than the numerical results.

![Fig.8 Effect of temperature ratio on cooling effectiveness](image)

**Cooling effectiveness at different locations**

Figure 9 shows the variation of the cooling effectiveness with the blowing ratio at different locations, when $T_g/T_c$ is 1.67 and $Re$ is 39353. The cooling effectiveness is calculated by averaging the location temperature. Compared with the experimental results, the simulation ones are underestimated 12% approximately. From the figure, it can be clearly seen that a higher cooling effectiveness is obtained with a larger blowing ratio. A larger blowing ratio means increased flow rate of the cooling air. Therefore, more heat can be taken away, and the cooling effect is improved. The blowing ratio is a significant parameter which affects the cooling effectiveness, as demonstrated by the simulation results. In addition, Figure 9 also indicates that when $m$ is below 1.2, the slope of the curve is higher, and the cooling effectiveness rapidly increases with the increase of the blowing ratio. The cooling effectiveness increases about 8% when $m$ increases from 1.2 to 2.0. However, it increases about 22% when $m$ rises from 0.8 to 1.2.

![Fig.9 The cooling effectiveness at different locations](image)

**Effect of different hole arrangements**

Figure 10 shows the cross-section temperature distributions on the Y-Z plane ($X/D = 1.8$). The simulation parameters are: $Re = 26338$, $m = 2.0$ and $T_g/T_c = 1.67$. For a line-holes arrangement, the film from the first hole row near the stagnation line overlaps with that from the second hole row on the outer surface of the leading edge downstream. But for a staggered-holes arrangement, the film completely covers the outer surface of the leading edge. Therefore, the temperature of the outer surface is lower, and the cooling effectiveness is improved.
seen in the figure, there is a clear temperature gradient along the blade-high direction for the two cases studied. The reason is probably that the velocity distribution of the cooling air flowing out of the film holes is non-uniform. The temperature distribution for a line-holes arrangement is shown in Fig. 11 (a). It is seen that the temperature is about 363 K in the top half of the surface, and about 373 K in the bottom half of the surface. However, the temperature is about 358 K and 366 K for a staggered-holes arrangement. The temperature of the outer surface for a staggered-holes arrangement is lower than that for a line-holes arrangement, which verifies the analysis before.

The average cooling effectiveness on the outer surface of the leading edge is shown in Fig. 12 for different hole arrangements at \( Re = 26338, m = 2.0 \) and \( Tg/Tc = 1.67 \). The cooling effectiveness is calculated by averaging the temperatures of the outer and inner surfaces. Compared with the average cooling effectiveness for the line-holes arrangement, the cooling effectiveness is improved about 4.3% for the staggered-holes arrangement.

**CONCLUSIONS**

In the present study, numerical simulations on the impingement/effusion cooling for the leading edge of a turbine blade are carried out. The accuracy and reliability of the simulation is validated through comparison with the experimental data available in the literature, and the influences of various factors on the fluid flow and heat transfer are analyzed in detail. The following conclusions can be drawn:

1. When the blowing ratio is less than 0.8, the cooling air can’t flow out of the film holes, and forms a protective layer which reduces the temperature of the outer surface of the leading edge.

2. The blowing ratio is the most important factor affecting the cooling effectiveness. The cooling effectiveness increases with the increase of the blowing ratio, and increases faster under low blowing ratio.

3. \( Tg/Tc \) has little effect on the cooling effectiveness. The cooling effectiveness slightly decreases with the increase of \( Tg/Tc \). Therefore, \( Tg/Tc \) is not a significant factor affecting the cooling effectiveness. However, the mechanism about the effect
of $T_g/T_c$ on the cooling effectiveness needs to be further investigated.

(4) Compared with the corresponding experiment, the numerical simulations underestimate the overall cooling effectiveness about 10% ~ 20%. The main reason lies in the temperature-measured method used in the experiment.

**NOMENCLATURE**

- $Re$: Reynolds number
- $m$: Blowing ratio
- $T$: Temperature (K)
- $T_a$: Average blade temperature (K)
- $\eta$: Cooling effectiveness
- $\rho$: Density (kg/m$^3$)
- $g$: Main stream
- $c$: Cooling stream
- $u$: Velocity (m/s)
- $w_i$: inner wall
- $w_o$: outer wall
- $\phi$: Diameter of holes (mm)

**ACKNOWLEDGMENTS**

This work was supported by the National Natural Science Foundation of China under Grant No.51076151 and the National Basic Research Program of China (973 Program) under Grant No. G2010CB227302. The authors also gratefully acknowledge the support of K.C. Wong Education Foundation, Hong Kong.

**REFERENCES**


Cooled Surface”, Heat Transfer Summer Conference collocated with the InterPACK09 and 3rd Energy Sustainability Conferences, pp:565-572.
