Experimental Investigation of Blade Leading Edge Jet Impingement with Varying Jet Nozzle Positions

Liangliang Liu  
School of Mechanical Engineering, Shanghai Jiaotong University  
liusoux@163.com  
Shanghai, 200240, China

Hao Liu  
School of Mechanical Engineering, Shanghai Jiaotong University  
lihuao@sjtu.edu.cn  
Shanghai, 200240, China

Xiaocheng Zhu  
School of Mechanical Engineering, Shanghai Jiaotong University  
zhxc@sjtu.edu.cn  
Shanghai, 200240, China

ZhaoHui Du  
School of Mechanical Engineering, Shanghai Jiaotong University  
zhdu@sjtu.edu.cn  
Shanghai, 200240, China

ABSTRACT

In this present an experimental investigation employs a transient thermos-chromic liquid crystals (TLC) technique to obtain detailed Nusselt number distributions on a concave surface that models the leading edge of a turbine blade. The jet impinges on the concave target channel by a single row of 5 aligned impinging jets, and it is extracted from one end of the cavity. The relative jet-to-target surface distance (Y/d) is 5.0, the jet-to-jet spacing (S/d) is 5.0, and the jet-to-target surface curvature (D/d) is 3.6. The impingement Reynolds number (Re) based on the jet nozzle diameter varies from 12000 to 20000. The effect of the jet nozzle position (E/d) on the impingement heat transfer on the target wall is also investigated. Experimental data show that, the Nusselt number on the target surface increases with the increase of the impingement Reynolds number, and the spanwise-average Nusselt number also increases with the increase of the impingement Reynolds number. For the same impingement Reynolds number, the heat transfer on the target surface enhances when the jet nozzle position is offset the center. The best heat transfer is obtained when the E/d is 0.5.

INTRODUCTION

Jet impingement cooling has been widely used in the internal cooling of turbine blades (especially blade leading edge regions) because of its advantages in the effective removal of locally concentrated heat and the easy adjustment to the location where the cooling is needed [1].

Impingement cooling has been studied mainly on flat plate, while relatively a few studies focus on curved surface. Extensive researches have been conducted on the effects of various basic parameters on the impingement heat transfer. Han and Goldstein [2] summarized the works in the 20th century and reported that the effects of jet Reynolds number, jet to target distance, jet nozzle spacing, the target surface curvature, rotation and jet angle etc. on the impingement heat transfer. Zuckerman et al.[3] summarized the experimental correlations of impingement cooling before 2005, including stagnation point and average heat transfer of single and array of jets. Taslim et al. [4-8] performed a series of experimental and numerical investigations of jet impingement in leading edge cooling channel for many years. They found that the racetrack shaped jet crossover holes, target surface roughness and showerhead film holes extraction can enhance the impingement heat transfer.

Since impingement heat transfer is quite non-uniform, contour measurement such as transient liquid crystal (TLC) and IR are applied in experiments on impingement cooling. Application of liquid crystals in transient heat transfer experiments is discussed by Jones et al.[9, 10]. Wagner el al.[11] analyzed the influence of surface curvature and finite wall thickness on TLC measurement. Approximated solution of curved surface was derived which helps leading edge heat transfer measurement. Yang et al.[12] used transient liquid crystal method to investigate the effect of film cooling hole position on the impingement heat transfer. They found that low film hole pitch could produce high heat transfer in the stagnation region. Jordan et al.[13-16] used TLC technique to obtain detailed Nusselt number distributions on a concave surface that models the leading edge of a turbine blade. The effect of hole shape, varying edge conditions at the jet nozzle, as well as varying inlet cross-flow conditions are investigated.
Results showed that racetrack shaped jets can enhance the impingement heat transfer. When introducing edge fillets and inlet cross-flow, the heat transfer on the target surface would degrade. Same test method was also used by Facchini et al. [17, 18] and Ricklick [19] to study the impingement heat transfer of a leading edge cooling.

Due to the limit of jet flow structure and stagnation region area, impinging jet velocity and jet nozzle location have bigger influence on impingement cooling. Therefore, it is necessary for turbine designers to investigate impingement cooling in turbine blade leading edge regions, and to obtain the flow and heat transfer characteristics in different jet nozzle location and impinging jet velocity. However, there are few studies on the effect of jet nozzle location on the impingement heat transfer of blade leading edge. Liu et al. [20] numerically compared the flow and heat transfer characteristics in a blade leading-edge cooling channel at five different jet nozzle position. It is found that both the streamwise length weighted average Nusselt number and the spanwise length weighted average Nusselt number increase with the decrease of the distance between jet nozzle center and pressure side. In addition, the departure of both the stagnation region and the area at the apex where the impinging air change direction decrease as the spacing between the jet nozzle and the pressure side decreases.

In this paper, a transient TLC method is performed to study the heat transfer of impinging cooling in the leading edge of a turbine blade. The impingement channel with a circular surface and two tapered side walls, which is supplied by 5 impinging jets, with exit flow in the axial direction, at one end of the passage. The purpose of the present paper is to study the effects of the jet Reynolds number and the relative position of jet nozzle on heat transfer on the target surface.

**EXPERIMENTAL SETUP**

**Experimental Apparatus**

Figure 1 and Figure 2 shows a sketch of the experimental setup and the test section. A centrifugal fan is used to generate the desired air flow in the test channel. The air enters a heater to heat the air from ambient temperature up to the desired temperature. Downstream of the heater, a triple value is employed to divert heated air so that it suddenly passes through the test model, in order to provide an abrupt temperature step along the surfaces. Before the test section, the honeycomb and screens are installed to ensure uniform incoming flow.

The test model is scaled model of the cooling passage within the leading edge of a turbine blade, which consists of a rectangular supply channel and a concave impingement cavity connected by 5 impinging holes. As the air leaves the supply duct, the portion of air that is used for impingement must turn 90° as it flows through each jet and into the impingement cavity. The target surface with 12mm in thickness is fabricated from plexi-glass to guarantee transparency and low thermal conduction coefficient. With the arrangement employed, the non-dimensional distance between the nozzle exit and the target surface \( Y/d \) is 5. The non-dimensional spanwise hole spacing \( S/d \) is 5, and the non-dimensional target surface curvature \( D/d \) is 3.6. The jet nozzle position \( E \) is varied to investigate the effect of the \( E \) on the impingement heat transfer. Three different ratios of \( E/d \) (\( E/d = 0, 0.5, 1.0 \)) are used in the present.

![Figure 1 Overview of the experimental facility](image)

![Figure 2 Details of test section](image)

**Measurement technique**

A transient method using thermochromic liquid crystals (TLC) is applied for measurement of heat transfer. Narrow bandwidth liquid crystals (R35C1W from LCR-Hallcrest) are used in the present work. The transient method for detailed heat transfer coefficient distribution is based on the assumption of the one-dimensional transient conduction over a semi-infinite wall, as described by the following equation.

\[
\frac{\partial^2 T}{\partial z^2} = \frac{1}{\alpha} \frac{\partial T}{\partial t}
\]  

(1)
Several boundary conditions are required for the solution:
\[ t = 0, T = T_i, \]
\[ z = 0, -k_i \frac{\partial T}{\partial z} = h(T_w - T_f) \]  
\[ z \to \infty, T = T_i \]

Within these equations, \( z \) denotes distance into solid material. Assuming that a purely convective boundary exists at the target wall (\( z=0 \)), and the thermal wave formed in the material throughout the course of a test does not propagate through the thickness of the material, the solution to the one-dimensional heat conduction equation is obtained:
\[ \frac{T_w - T_i}{T_f - T_i} = 1 - \exp\left(-\frac{h^2 \alpha t}{k_i^2}\right) \operatorname{erfc}\left(\frac{h\sqrt{\alpha t}}{k_i}\right) \]  

However, it is difficult to maintain a real temperature step for the mainstream flow as shown in our experiment in Figure 3. Therefore, the Eq.(3) must be modified with Duhamel’s theorem of superposition to take into account the time-dependent temperature history. Thus the solution can be written as follows,
\[ \frac{T_w - T_i}{T_f - T_i} = \sum_{n=0}^{\infty} \left[1 - \exp\left(-\frac{h^2 \alpha (\tau - \tau_n)}{k_i^2}\right) \operatorname{erfc}\left(\frac{h\sqrt{\alpha (\tau - \tau_n)}}{k_i}\right)\right] \Delta T_{f(n+1)} \]  

The heat transfer coefficient could be determined based on the value of \( T_w \) and \( T_f(t) \). In order to satisfy the assumption of one-dimensional transient conduction over a semi-infinite solid wall, the entire experimental time should meet the requirement of \( t \approx \frac{HF}{16\alpha} \), according to Schultz and Jones[21]. The calculated time is 106 s, and the entire experimental time \( t \) is about 100 s, so the assumption of one-dimensional transient conduction could be used.

In Eq.(4), the target surface temperature, \( T_w \), is a constant obtained through calibration of the liquid crystals (the current conditions yield a maximum green color corresponding to approximately 35.3). The initial temperature, \( T_i \), is measured with K-type thermocouples placed within the test section. The time, \( t \), is obtained from the 3CCD camera positioned above the test section. The flow temperature, \( T_f \) is also recorded at the same frequency as the camera’s frame rate, so the flow temperature change, \( \Delta T_f \), as well as the time step change, \( \tau \), are also known. Thus, detailed heat transfer coefficient distributions are obtained by iteratively solving Eq.4. The results are then non-dimensionalized to form the Nusselt number distributions, as shown in Eq.5.
\[ Nu = \frac{hd}{k_f} \]  

It should be noted that the Eq.(4) is obtained for convection on a flat surface, so the local heat transfer coefficient on the concave surface should be modified by the suggestion of Buttsworth and Jones[22]. Due to the low material thermal conductivity, the result calculated with Eq.(6) is less 1% higher than the result calculated with Eq.(4).
\[ h' = h + \frac{k_s}{D} \]  

**Measurement uncertainties**

The experimental error analysis used in here is based on the description by Moffat[23]. The accuracy of the measured heat transfer coefficient depended mainly on the accuracy of the thermocouples, the calibration of the liquid crystals and the measurement of the time.

For narrow band TLC, the typical uncertainty in measuring the wall temperature \( T_w \) is about 0.1K[24]. The error on \( T_i \) and \( T_f \) measurement with thermocouples is below 0.5K. The time resolution associated with the CCD camera is approximately 0.04s. Therefore, the overall experimental uncertainty for the Nusselt number on the target surface is below 12.0%.

**RESULTS AND DISCUSSION**

**Experimental verification**

To validate the measurement method used in this experiment, the Nu data is compared with the results of Chupp et al[25], as shown in Fig.4. In the figure, the \( Nu(z)/Nu(z=0) \) represents the Nusselt number ratios, and the \( z/d \) represents the non-dimensional distance away from the stagnation line. It is seen from Fig.3 that the measurements are match well with the data of Chupp et al. The maximum relative error is less than 5.0%.

![Figure 4 Comparison of spatially-averaged surface Nusselt number ratios between present investigation and data of Chupp et al. for Re=15000](image_url)
Effect of jet Reynolds number on impingement heat transfer

In order to analyze the effects of the jet Reynolds number on impingement heat transfer of turbine leading edge, three different jet Reynolds numbers (Re=12000, 15000 and 20000) at a constant $E/d=0$ are compared.

The local Nusselt number distributions on the target surface are presented in Figure 5 for the Re of 12000, 15000 and 20000. The $X$ axis indicates the streamwise direction, and the $Y$ axis indicates the spanwise direction. One region of augmented Nusselt numbers that is associated with the impact stagnation region beneath each impingement jet is found. The confluence of the jet flow and cross flow also has a region with relatively high Nu. For the same jet Reynolds number, the highest heat transfer is located in the impingement region of the first jet hole. As $Z/d$ increases, each Nusselt number distribution associated with an individual jet becomes smaller in size with lower Nusselt number values. This is because that the cross-flow generated by the accumulation of the upstream spent flow after impinging causes the jet deflection and degrades the local heat transfer. As the increase of the jet Reynolds number, the jet velocity increases, which resulting in enhance the impingement heat transfer on the target surface. It should be noted that the areas of white evident in Figure 5 are areas where the liquid crystals don’t reach a maximum green hue, so no heat transfer coefficients are calculated in these areas.

![Figure 5 Experimentally measured target surface](image)

**Figure 5 Experimentally measured target surface**

_Nu contours for Re of 12000, 15000 and 20000_

Figure 6 shows the spanwise-averaged Nusselt number $N_u$ distributions along the streamwise direction for different Re. The dash lines are the center of jet nozzles. It is noted that there is no heat transfer data in some regions as shown in Figure 5, so the curves in the Figure 6 are not continuous.

For each Reynolds number, the $N_u$ is higher in the stagnation region of each jet hole, and the confluence of jets also has a relatively higher $N_u$ peak value. The largest peak values of the $N_u$ is located in the stagnation region of the first jet. Along the impingement passage as $Z/d$ increases, the peak value of the $N_u$ decreases due to the effect of the cross flow, and the position of the stagnation point is also shifted to the downstream. The closer the peak value of $N_u$ to the outlet, the farther it deviates from the vertical place of jet nozzle along with the stagnation region. With the increase of the jet Reynolds number, both $N_u$ and its peak-to-peak value increase. It means that the $N_u$ increases with the increase of Re, which will lead to an increase to the thermal stress on the turbine leading edge. In addition, larger jet velocity will reduce gas turbine efficiency. Therefore, the reasonable jet velocity should be neither too large nor too small.

![Figure 6 Effects of Re on $N_u$ distribution along streamwise direction](image)

**Figure 6 Effects of Re on $N_u$ distribution along streamwise direction**

Effect of position of jet nozzle on impingement heat transfer

In order to investigate the effects of the jet nozzle position on impingement heat transfer of turbine leading edge, three different $E/d$ (0, 0.5 and 1.0) is used to compare at a constant Re=15000 in this section.

Figure 7 shows the Nu contours on the target surface at different $E/d$ when Re=15000. The areas of white evident in Figure 7 still represents the regions where no heat transfer data due to the test time restrictions. It is seen that the high Nu region is shifted to one side with the deviation of the jet nozzle. The Nu of the stagnation point increases with the increase of the $E/d$ due to the decrease of the jet distance, especially the stagnation heat transfer in the downstream. In addition, the Nu of the confluence region of the jet flow and cross flow also increases when the jet nozzle position is offset from the center of the channel. It is noted that the overall heat transfer decreases when the $E/d$ increases from 0.5 to 1.0. This is because the curvature of the target surface is not so big, some jet flow impinges on the side wall.
In order to conduct quantitative analysis, Figure 8 presents the spanwise-averaged Nusselt number \(\text{Nu}_{sp}\) distributions along the streamwise direction for different \(E/d\) at Re=15000. The dash lines are the centre of jet channel, and the curves in the Figure 8 are also not continuous. The \(\text{Nu}_{sp}\) is higher in the stagnation region of impinging jets, and the confluence of jets also has a relatively higher \(\text{Nu}_{sp}\) peak value. The peaks of \(\text{Nu}_{sp}\) move to the downstream along with the stagnation regions of jets. The closer it is to the outlet, the farther it deviates from the vertical place of jet nozzle along with the stagnation region. When the position of the jet nozzle is offset the centre of the channel, the \(\text{Nu}_{sp}\) increases and the deviation distance of the peaks of the \(\text{Nu}_{sp}\) decreases. The high \(\text{Nu}_{sp}\) region becomes wider when the jet nozzle is offset. In addition, it is also found that the best \(\text{Nu}_{sp}\) distribution is obtained at \(E/d=0.5\).

Figure 7 Nu contours of the target surface for different \(E/d\) at Re=15000

CONCLUSIONS

A transient TLC method has been performed to study the impingement heat transfer in the turbine blade leading edge, the influence of the jet Reynolds number and the jet nozzle position has been analyzed. From this study, the following conclusions can be drawn:

1. Both the \(\text{Nu}_{sp}\) and the overall Nu on the target surface increase with the increase of the jet Reynolds number. For the same jet Reynolds number, high single peak values of \(\text{Nu}_{sp}\) appear at the stagnation point of each jet hole and smaller peaks of \(\text{Nu}_{sp}\) appear at the confluence of the jet and the cross-flow. Along the impingement passage as \(Z/d\) increases, the peaks of \(\text{Nu}_{sp}\) decreases and deviates from the vertical place of jet nozzle.

2. When the jet nozzle position is shifted to one side wall, both the \(\text{Nu}_{sp}\) and the overall Nu on the target surface increase. The high \(\text{Nu}_{sp}\) region also becomes wider and the deviation distance of the peaks of the \(\text{Nu}_{sp}\) decreases. The best heat transfer on the target surface is obtained when the \(E/d\) is equal to 0.5.

NOMENCLATURE

- \(d\): diameter of the jet nozzle (m)
- \(D\): target surface curvature (m)
- \(E\): position of jet nozzle (m)
- \(h\): convective heat transfer coefficient (Wm\(^{-2}\)K\(^{-1}\))
- \(H\): thickness of the target surface (m)
- \(k_f\): thermal conductivity of air (Wm\(^{-1}\)K\(^{-1}\))
- \(k_s\): thermal conductivity of test section material (Wm\(^{-1}\)K\(^{-1}\))
- \(L\): impingement hole length (m)
- \(\text{Nu}\): Nusselt number (\(hd/k_f\))
- \(\text{Nu}(z=0)\): Nusselt number of the stagnation point
- \(\text{Nu}_{sp}\): spanwise-average Nusselt number
- \(\text{Re}\): Reynolds number of the impinging jet (\(U_jd/\nu\))
- \(S\): distance between the jets (m)
- \(t\): time (s)
- \(T\): temperature (°C)
- \(T_f\): temperature of jet nozzle exit (°C)
- \(T_i\): initial temperature of the test section (°C)
- \(T_w\): wall temperature of the test section (°C)
- \(U_j\): velocity of the impinging jet (ms\(^{-1}\))
- \(X\): spanwise direction
- \(Y\): wall-normal direction
- \(Z\): streamwise direction

Greek symbols

- \(\nu\): kinematic viscosity of air (m\(^2\)s\(^{-1}\))
- \(\tau\): time step change of bulk fluid temperature (s)
Subscripts

\[ j \quad \text{jet} \]

REFERENCES


