On heat transfer and flow characteristics of jets impinging onto concave surface with varying bleeding arrangements

Dandan Qiu
Harbin Institute of Technology, Harbin, China

Lei Luo
School of Energy Science and Engineering, Harbin Institute of Technology, Harbin, China

Zhiqi Zhao and Songtao Wang
Harbin Institute of Technology, Harbin, China

Zhongqi Wang
School of Energy Science and Engineering, Harbin Institute of Technology, Harbin, China, and

Bengt Ake Sunden
Department of Energy Sciences, Lund University, Lund, Sweden

Abstract

Purpose – The purpose of this study is to investigate the effects of film holes' arrangements and jet Reynolds number on flow structure and heat transfer characteristics of jet impingement conjugated with film cooling in a semicylinder double wall channel.

Design/methodology/approach – Numerical simulations are used in this research. Streamlines on different sections, skin-friction lines, velocity, wall shear stress and turbulent kinetic energy contours near the concave target wall and vortices in the double channel are presented. Local Nusselt number contours and surface averaged Nusselt numbers are also obtained. Topology analysis is applied to further understand the fluid flow and is used in analyzing the heat transfer characteristics.

Conflict of interest: None declared.
Findings – It is found that the arrangement of side films positioned far from the center jets helps to enhance the flow disturbance and heat transfer behind the film holes. The heat transfer uniformity for the case of 55° films arrangement angle is most improved and the thermal performance is the highest in this study.

Originality/value – The film holes’ arrangements effects on fluid flow and heat transfer in an impingement cooled concave channel are conducted. The flow structures in the channel and flow characteristics near target by topology pictures are first obtained for the confined film cooled impingement cases. The heat transfer distributions are analyzed with the flow characteristics. The highest heat transfer uniformity and thermal performance situation is obtained in present work.

Keywords Heat transfer, Film cooling, Jet impingement, Flow structure, Topology analysis

Paper type Research paper

Nomenclature
\(d_f\) = film diameter [mm];
\(d_j\) = jet diameter [mm];
\(D\) = target wall diameter [mm];
\(h\) = Heat transfer coefficient [W/(m².K)];
\(M\) = Mass flow rate at inlet [kg/s];
\(Nu\) = Nusselt number [–];
\(Nu_{ave}\) = Area-averaged Nusselt number [–];
\(Nu_{ave,b}\) = Area-averaged Nusselt number for baseline [–];
\(P\) = Jet-to-jet/film-to-film distance for the same line jets/film holes at Y direction [mm];
\(P_i\) = Inlet mass flow average pressure [Pa];
\(P_o\) = Outlet mass flow average pressure [Pa];
\(q\) = Heat flux [W/m²];
\(Re\) = Jet Reynolds number [–];
\(S\) = Stream-wise along the concave target surface [–];
\(TKE\) = Relative turbulence kinetic energy, nondimensionalized by \(u_i^2\) [–];
\(T_i\) = Jet inlet temperature [K];
\(T_{w}\) = Impingement wall temperature [K];
\(u_i\) = Jet inlet velocity [m/s];
\(Z\) = Jet-to-target distance [mm];
\(\theta\) = Degree between the middle array jets and side film arrays [°];
\(\lambda\) = Fluid thermal conductivity [W/(m.K)];
\(\mu\) = Fluid dynamic viscosity [Pa.s];
\(\rho\) = Fluid density [kg/m³]; and
\(\tau_w\) = Wall shear stress [Pa].

1. Introduction
Due to its high potential in enhancing local heat transfer, jet impingement has been widely used in turbine thermal load components, especially for the highest heat load portion – the blade leading edge, which directly suffers hot gas from combustor (Kwak and Han, 2002).

Many significant works spent much efforts on the effects of jet Reynolds number, jet nozzle configuration, jet-to-target spacing, target arrangement and jet inclination angle on single jet impinging onto a confined or an unconfined concave or convex surface (Yang et al., 1999; Rahman et al., 2010; Qiu et al., 2019; Cornaro et al., 1999; Luo et al., 2016; Lee et al., 2007; Yang and Hwang, 2004).

In real applications, multiple impingement jets are commonly used. Concerning the parameters mentioned above, the jet arrangement plays an important role of a single row or
arrays of jets situations. Concerning the single case, the jets are often arranged at the center of the surface. Chupp et al. (1969) performed an experiment to investigate how the heat transfer is influenced by jet Reynolds number, jet-to-target/jet-to-jet spacing and target curvature for an array of round jets impinging onto a confined concave surface. They presented dimensionless correlations of the area-averaged and stagnation point Nusselt numbers. Metzger et al. (1969) experimentally summarized the correlations of maximum and local Stanton number of a confined concave channel. Efforts on effects of jet Reynolds number, jet-to-target/jet-to-jet spacing and target curvature on local Nusselt number distributions of a single array of jets impinging onto a confined or an unconfined concave surface were conducted experimentally (Bunker and Metzger, 1990; Patil and Vedula, 2018) and numerically (Kumar and Prasad, 2008). Yang et al. (2014) numerically conducted unsteady predictions of flow characteristics in a cylinder channel with a single row impingement jets at a constant jet Reynolds number 15,000. For arrays of jet arrangements, the Nusselt number distributions of two staggered arrays of impingement jets on a confined concave channel with changing jet Reynolds number were experimentally investigated with the transient liquid crystal technique by Calzada and Alvarez (2012). Jung et al. (2018) performed an investigation numerically and experimentally about the influence of injection angle on fluid flow and heat transfer for three staggered arrays of impingement jets. More recently, Qiu et al. (2020) predicted the vortical structure of jets impinging onto a confined curved surface for three arrays of jets arranged inline and staggered and further analyzed the local heat transfer characteristics with the fluid flow identified by numerical simulations.

At the blade leading edge of a high-pressure turbine blade, impingement is often applied conjugated with film cooling. Metzger and Bunker (1990) designed an experiment to measure the local heat transfer characteristics of blade impingement-cooled leading edge with bleedings and found that the change of the jet-to-film position can cause significant variations of the leading-edge temperature. Taslim et al. (2003) performed a study to compare the effects of the target wall roughening conditions on an impingement-cooled blade leading edge with film holes numerically and experimentally. They claimed that the numerical works presented good agreement with the experimental results. It was found that impingement on the smooth target produced the highest overall heat transfer coefficients followed by the notched-horseshoe ribs and horseshoe geometries. Taslim and Khanicheh (2006) conducted numerical and experimental works on the influence of varying film extraction patterns on the heat transfer coefficients of an airfoil leading edge with impingement cooling. The results inferred that the presence of film holes along the leading edge enhanced the internal impingement heat transfer coefficients significantly compared with the case without film holes. Yang et al. (2013) measured the heat transfer coefficient on a 2/3 cylinder leading edge model with impingement cooling and varying film cooling arrangements by the transient liquid crystal technique and compared with numerical simulation works. The results recommended the hole position for the best cooling performance. Andrei et al. (2013) evaluated the heat transfer coefficients on a real-engine leading edge cooling system characterized by racetrack-shaped crossover holes, showerhead and film cooling extraction holes and large fins both experimentally and numerically. They highlighted that asymmetric mass flow extraction and variable crossflow conditions slightly influence the Nusselt number. Wang et al. (2018) investigated the effect of jet impinging position on the heat transfer of a leading edge with extractions and showed that the tangential jet provided more uniform heat transfer distribution than the normal jet with similar area averaged Nusselt numbers. Luo et al. (2020) presented a numerical analysis of the pin-fins’ arrangement in a Lamilloy cooling system. The results showed the optimum
location and diameter of pin-fin with high heat transfer enhancement and relatively low pressure loss.

Impingement-cooled blade leading edge with extractions has been extensively investigated but mainly with the flow and geometry effects on the local or averaged heat transfer coefficient. Very limited published work indicates the relationship between flow structure and heat transfer characteristics of impinging-cooled leading edge conjugated with varying film arrangement. In this study, the relationship between flow structure and heat transfer characteristics of a single array of jets impingement conjugated with two staggered arrays of film holes of a semicylinder double wall channel are numerically investigated. The effects of film hole arrangements and jet Reynolds number (10,000–40,000) on the fluid flow and heat transfer are also presented. The characteristics of the flow structure (vortices, streamlines and skin-friction lines) and heat transfer (local Nusselt number contours and surface averaged Nusselt numbers) for different cases are obtained. Topology analysis is also applied to understand the fluid flow which is then applied to analyze the local heat transfer characteristics.

2. Problem description

2.1 Physical model

According to the previous discussions, jet impingement conjugated with film cooling is usually applied at the blade leading edge due to its high heat transfer enhancement. As shown in Figure 1(a), the blade leading edge cooling structure is simplified to a semicylinder double wall channel with a center array of jets and two side arrays of film cooling holes. The degree \( \theta \) between the center jets and side films is changing from 0° to 55° [Figure 1(a)]. The case of \( \theta = 0° \) is set as the baseline, in which only one array of film holes is arranged. From the top view [Figure 1(b)], the side arrays of film holes (black circles) are staggered in relation to the center array of jets (shadow circles) with jet-to-jet/film-to-film spacing \( P/d_j \) = 4. All parameters presented are normalized by the jet diameter \( d_j \). The whole area of film cooling holes is constant for all cases and the film hole diameter for the baseline \( d_f \) is equal to the jet diameter \( d_j \). The jet-to-target distance \( Z/d_j \) and relative surface curvature \( D/d_j \) are 1 and 10, respectively. The detailed information of the cases is shown in Table 1. The jet Reynolds number varies from 10,000 to 40,000 (Patil and Vedula, 2018).

2.2 Boundary conditions

This study reveals the effects of jet Reynolds number \( Re \) and film holes arrangement on fluid flow and heat transfer characteristics of a semicylinder double wall jet impingement cooling system conjugated with film cooling. As presented in Figure 1, the target wall is set as no-slip and an isothermal wall condition. No-slip and adiabatic wall boundary condition are applied to the other surfaces. A mass flow rate at the inlet and isothermal boundary condition with 5% turbulence intensity are used at the jet inlet. A pressure outlet boundary is set at the film hole outlet. The temperature difference between the jet inlet and target wall is 41 K. The fluid is considered as an ideal gas.

2.3 Topology of skin-friction fields

Topology analysis is a method to understand the three-dimensional separated flows. It uses skin-friction lines coupled with the critical point theory (Legendre, 1956; Poincaré, 1882; Hirschel et al., 2014) and has been applied in some research and engineering works (Gbadebo et al., 2005; Zhao et al., 2020; Kan et al., 2016).
Four main critical points (spiral node, attachment node, separation node and saddle) and two lines (attachment line and separation line) are used in the present work. The detailed definitions are presented in Figure 2 (Délery, 2001).

3. Computational solution method
Three-dimensional, nonrotating, steady and compressible simulations have been performed with the commercial software FLUENT (2018). The second-order accuracy option and phase

<table>
<thead>
<tr>
<th>Cases</th>
<th>Film arrangement</th>
<th>$P/d_j$</th>
<th>$\theta$ (°)</th>
<th>$d/d_j$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>1 row/same line</td>
<td>4</td>
<td>–</td>
<td>1</td>
</tr>
<tr>
<td>Case 1</td>
<td>2 rows/staggered</td>
<td>4</td>
<td>15</td>
<td>0.707</td>
</tr>
<tr>
<td>Case 2</td>
<td>2 rows/staggered</td>
<td>4</td>
<td>25</td>
<td>0.707</td>
</tr>
<tr>
<td>Case 3</td>
<td>2 rows/staggered</td>
<td>4</td>
<td>35</td>
<td>0.707</td>
</tr>
<tr>
<td>Case 4</td>
<td>2 rows/staggered</td>
<td>4</td>
<td>45</td>
<td>0.707</td>
</tr>
<tr>
<td>Case 5</td>
<td>2 rows/staggered</td>
<td>4</td>
<td>55</td>
<td>0.707</td>
</tr>
</tbody>
</table>

Table 1. Detailed information of the cases
coupled semi implicit pressure linked equations algorithm are applied. The computation is regarded to be converged as the residuals below $1 \times 10^{-5}$ for the continuity equation, velocity, turbulence and energy equations.

### 3.1 Governing equations

The steady viscous fluid flow motion is governed by the following equations (Versteeg and Malalasekera, 2007):

**Continuity equation:**

$$
\frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0
$$

**Momentum equation:**

$$
\frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i} \left( \mu + \mu_t \right) \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)
$$

**Energy equation for fluid:**

$$
C_p \bar{u}_i \frac{\partial \bar{T}}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \lambda \frac{\partial \bar{T}}{\partial x_j} \right) - C_p \frac{\partial}{\partial x_i} \left( \frac{\mu_t}{Pr_t} \frac{\partial \bar{T}}{\partial x_i} \right)
$$

The computations were conducted by using a Reynolds-averaged Navier–Stokes model combined with the $k\omega$-baseline turbulence model (BSL). The $k\omega$-BSL is believed to improve the predictive capability for complex turbulent flows with flow swirling and separation and
it is also claimed to offer the best trade-off between accuracy and computational cost for the parameters considered (Versteeg and Malalasekera, 2007; Menter, 1994; Wallin and Johansson, 2000). Therefore, the $k\omega$-BSL turbulence model is used and the following equations are applied.

The equation of the turbulent kinetic energy $k$ reads as follows:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_l}{\sigma_{k3}} \right) \frac{\partial k}{\partial x_i} \right] + P_k - \beta^* \rho \omega k \quad (4)$$

The equation of the dissipation rate $\omega$ reads as follows:

$$\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_l}{\sigma_{\omega3}} \right) \frac{\partial \omega}{\partial x_i} \right] + \left( 1 - F_1 \right) \frac{2\rho}{\sigma_{\omega2} \omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} + \alpha_3 \frac{\omega}{k} P_k - \beta_3 \rho \omega^2 \quad (5)$$

where

$$\sigma_{k3} = F_1 \sigma_{k1} + (1 - F_1) \sigma_{k2} \quad (6)$$

$$\sigma_{\omega3} = F_1 \sigma_{\omega1} + (1 - F_1) \sigma_{\omega2} \quad (7)$$

$$\beta_3 = F_1 \beta_1 + (1 - F_1) \beta_2 \quad (8)$$

$$\alpha_3 = F_1 \alpha_1 + (1 - F_1) \alpha_2 \quad (9)$$

$$\mu_l = \rho \frac{k}{\omega} \quad (10)$$

$$P_k = \mu_l \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \left( 3 \mu_l \frac{\partial u_k}{\partial x_k} + \rho k \right) \quad (11)$$

$$F_1 = \tanh \left( \arg_1^{\frac{1}{4}} \right) \quad (12)$$

$$\arg_1 = \min \left[ \max \left( \frac{\sqrt{k}}{\beta^* \omega d}, \frac{500v}{\omega d^2} \right) \frac{4 \rho \sigma_{\omega2} k}{CD_{k\omega} d^2} \right] \quad (13)$$

$$CD_{k\omega} = \max \left( 2 \rho \sigma_{\omega2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-20} \right) \quad (14)$$

The constants have the values $\sigma_{k1} = 2, \sigma_{k2} = 1, \sigma_{\omega1} = 2, \sigma_{\omega2} = 1/0.856, \beta_1 = 0.075, \beta_2 = 0.0828, \beta^* = 0.09, \alpha_1 = 5/9, \alpha_2 = 0.44, A_1 = 1.245, A_2 = 0, A_3 = 1.8, A_4 = 2.25, C_\mu = 0.09.$
3.2 Mesh information
As shown in Figure 3, a structured mesh is generated by the ICEM computational fluid dynamics (CFD) (2018) with quality above 0.6. Near-wall meshes with \( y^+ \sim 1 \) are applied. Grid independence validation has been performed for the baseline to make certain that the simulations are reasonable accurate with low computational efforts. As presented in Figure 4, the laterally averaged Nusselt number on the target wall and pressure along the stream-wise direction for changing mesh numbers are provided. It is found that there are no significant differences between the meshes 6.5 and 12 million. To save the computational demands, the grid configurations with 6.5 million cells are applied in this work.

3.3 Data reduction
The Reynolds number is defined as:

\[
Re = \frac{\rho u_c d_j}{\mu}
\]  

(15)
The heat transfer coefficient $h$ is defined as:

$$h = \frac{q}{T_i - T_w}$$  \hspace{1cm} (16)

The Nusselt number is defined as:

$$Nu = h \times \frac{d_j}{\lambda}$$  \hspace{1cm} (17)

The pumping power (Bergman et al., 2011) is defined as:

$$P = (P_i - P_o) \times \frac{M}{\rho}$$  \hspace{1cm} (18)

where $u_i$ is the mean velocity at the jet inlet, $d_j$ is the hydraulic diameter of the jet and $\rho$ and $\mu$ are the fluid density and dynamic viscosity, respectively. $q$ is the heat flux, $\lambda$ is the fluid thermal conductivity, $T_i$ and $T_w$ are the temperature of the jet inlet and target wall, respectively. $P_i$ and $P_o$ are the mass flow average total pressures at the jet inlet and outlet, respectively. $M$ is mass flow rate at the inlet.

3.4 CFD validation

In this work, the laterally local Nusselt number and averaged Nusselt number are compared with the experimental data by Patil and Vedula (2018) to verify the accuracy of the computations. The case with $D/d_j = 10$, $P/d_j = 4$, $Z/d_j = 2$, $Re = 50,000$ is considered for comparison. As presented in Figure 5, the numerical validations are conducted by comparing the laterally variations for some turbulence models in FLUENT with the experimental data by Patil and Vedula (2018). The error bar of the experimental data is set as 10% which is the maximum uncertainty of the experiment. It is obvious that the trends of the $k\omega$-BSL model are very similar with the experiments and the maximum inaccuracy is satisfactory. Concerning the average parameters, the average Nusselt numbers are presented in Table 2. The maximum difference between the numerical and experimental

![Figure 5](image-url)
data is 5.4% which is lower than the experimental uncertainty, i.e. 10%. Therefore, the $k\omega$-BSL turbulence model is considered acceptable in this work.

4. Results and discussions

4.1 Fluid flow characteristics

The overall heat transfer performance is highly related to the local heat transfer characteristics which are significantly affected by the fluid flow. To understand the effect of the conjugated film holes’ arrangement on fluid flow characteristics for jets impinging onto a confined concave surface, streamlines at different sections, skin-friction lines near the target wall, topology pictures and vortices in the channel of the baseline and Case 5 are introduced. Due to the symmetry characteristics of the geometry and fluid flow, half part ($X/d_j > 0$) is presented for the near-wall surface skin-friction lines, vortex figures and Nusselt number plots. To reveal the similarity of the fluid characteristics, figures of the vortices around the middle jet are presented.

Figure 6 provides the vortices with the $\lambda^2$-(a-b) criterion, streamlines at (c) different sections of the baseline. Due to the symmetry of the flow and to avoid confusion, only the

<table>
<thead>
<tr>
<th>Table 2. Different included angle averaged Nusselt number comparison between $k\omega$-BSL turbulence model results and experimental work</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\theta$ = $-25^\circ$ - $25^\circ$</td>
</tr>
<tr>
<td>Experiment [12]</td>
</tr>
<tr>
<td>Present</td>
</tr>
<tr>
<td>Difference</td>
</tr>
</tbody>
</table>

Figure 6. Fluid flow of middle jet for baseline at $Re = 20,000$: (a-b) vortices colored with velocity (M – primary vortices; Se – separation vortices; $S_1$, $S_2$, $S_3$ – secondary vortices); (c) streamlines at different sections; (d) topology picture

$Sp_a$ – spiral node
$a'$ – half attachment node
$La_1$, $La_2$ – attachment line
$Sa_1$, $Sa_2$, $Sa_3$, $Sa_4$, $Sa_5$ – saddle
$a_1$, $a_2$, $a_3$, $a_4$ – attachment node
$Ls_1$, $Ls_2$, $Ls_3$, $Ls_4$, $Ls_5$ – attachment line
system of \(2 \geq Y/d_j \geq 0\) is discussed. As shown in Figure 6(a), the main vortex (marked as M) can be observed, which coincides with an outward clockwise circulation of the streamlines \([\theta \text{ from } 0^\circ \text{ to } 30^\circ \text{ in Figure 6(c)}]\). This unsteady flow can be attributed to the interactions of adjacent jets and the suction effect of the film hole. Below the main vortex M, a secondary vortex \(S_1\) [Figure 6(a)] and inward anticlockwise rotation \([\theta = 20^\circ, 30^\circ \text{ in Figure 6(c)}]\) are found near \(Y/d_j = 2\), which can be related to the restriction effect of the up-wash main flow (main vortex M) and target wall. Other secondary flows, marked as \(S_2, S_3\) [Figure 6(a, b)] and parallel recirculation \([\theta = 55^\circ, 70^\circ \text{ in Figure 5(c)}]\) are also found near \(Y/d_j = 0\). A steady separated flow occurs which can be observed in the pictures of the vortices as the vortex \(S_e\) marked in Figure 6(b) and the internal clockwise circulations shown in Figure 6(c) \((\theta = 55^\circ, 50^\circ, 40^\circ)\). Concerning the topology picture [Figure 6(d)], the attachment node \(a_1\) can be related to the jet impinging. With the interaction of the main flow (clockwise vortex M) and secondary flow (counterclockwise vortex \(S_1\)), the separation line \(L_{s1}\) is formed. The appearance of the attachment lines \(L_{a1}\) and \(L_{a2}\) can be attributed to the interactions of adjacent vortices M around \(Y/d_j = 0\) and \(S_1\) around \(Y/d_j = 2\), respectively. Along the stream-wise direction, the position of the spiral node \(S_{sp}\) coincides with the appearance of the separation vortex \(S_e\) which is spiraled by the separation lines \(L_{s1}\) (from saddle \(s_{a1}\)), \(L_{s2}\) (from saddle \(s_{a2}\)), \(L_{s3}\) (from saddle \(s_{a3}\)) and \(L_{s4}\) (from saddle \(s_{a4}\)). The separation line \(L_{s5}\) shows similar trends with the secondary vortex \(S_1\). The region between separation line \(L_{s4}\) and center red attachment line \((Y/d_j = 0)\) coincides with the secondary vortex \(S_3\). Between the separation lines \(L_{s2}\) and \(L_{s5}\), another secondary flow region might be detected by the topology analysis.

As film holes positioned farther from the center array jets, the flow characteristics change. Figure 7 presents pictures of vortices with \(\lambda^2\) - (a-b) criterion, streamlines at (c) different sections of Case 5. Compared with the baseline, a similar main vortex M and secondary vortex \(S_{12}\) with paralleling unsteady outward clockwise circulation in streamlines \([\theta = 0^\circ - 25^\circ, \text{ Figure 7(c)}]\) and inward anticlockwise circulation around \(Y/d_j = 2\) \([\theta = 0^\circ - 45^\circ \text{ in Figure 7(c)}]\) also occur. The corresponding attachment lines \(L_{a1}, L_{a2}\) and the separation line \(L_{s1}\) due to the interaction of adjacent main vortices M, between main vortex M and secondary vortex \(S_1\) and neighboring secondary vortices \(S_1\) are clearly shown in Figure 7(d). The most different styles are the appearance of secondary vortex \(S_4\), vortex S and the absence of the separation vortex \(S_e\) and secondary vortices \(S_2, S_3\). The vortex \(S_1\) around \(Y/d_j = 0\) is related to the interaction between vortex M and the upper wall. The corresponding circulation in the streamlines is observed \([\theta = 15^\circ - 40^\circ, \text{ Figure 7(c)}]\). The L type vortex S moves against the stream-wise direction and toward the \(Y/d_j = 0\) section \([\text{Figure 7(c)}, \text{ from } \theta = 70^\circ - 40^\circ]\) which might be attributed to the film suction effect.

4.2 Local heat transfer characteristics

On the basis of the discussion, the local wall heat transfer characteristics are analyzed with the wall skin-friction lines, topology pictures, wall dimensionless velocity, wall shear stress and turbulence kinetic energy \((TKE)\) contours. The quarter plots of wall distributions are provided in Figures 8 and 9 for the baseline and Case 5, respectively, due to the symmetry of geometry and fluid flow. Generally, the Nusselt number distributions show higher values around the attachment lines [red lines presented in Figures 8(a) and 9(a)]. As the flow attaches to the target wall, the velocity, wall shear stress and \(TKE\) present high levels. In contrast, the values near the separation lines [blue lines presented in Figures 8(a) and 9(a)] present low levels.

For the baseline, it is obvious that relative high velocity, wall shear stress and turbulence kinetic energy values occur around the impingement region (marked as A) and then
decrease along the stream-wise direction [Figures 8(c-e)]. This can be attributed to the main flow impinging effect of jets which can be the reason of the significantly enhanced heat transfer region A marked in Nusselt number contours [Figure 8(b)]. Along the stream-wise direction, a low value region (marked as B) coincides with the region enclosed by some separation lines [Figure 8(a)], which can be related to the secondary flow mentioned above. Around the spiral node, the values of the Nusselt number, velocity, wall shear stress and TKE are also very low. This can be attributed to the movement of separation vortices. As the flow separates from the target wall and generates the separation vortices Se with low velocity, wall shear stress and TKE distributions, which may lead to a thinner flow boundary layer and thus result in a low heat transfer enhancement. Further away from the center jet positions, the values are also very low near the end of the target wall in the stream-wise direction. This can be related to the low energy secondary vortices S2 discussed in Figure 6. Between the center jets, the heat transfer near the film holes is significantly enhanced and the flow parameters also show high values, which can be related to the suction effect. In Figure 8, another high value region marked as C can be observed around the attachment lines of $Y/d_j = 2, 6, 10$. This appearance of region C coincides with the secondary vortex $S_5$.

Comparing the topology pictures of Case 5 and the baseline, the disappearance of vortex $S_e$, $S_3$, $S_2$ can also be clearly observed. Concerning the flow characteristics and heat transfer distributions (Figure 9), high values of region A and around the film holes, low values (region B) at the recircled separation lines and relative high values (region C) around the attachment line La2 are also observed. Not like the baseline, at the backward of the film holes, the Nusselt number contours present higher levels coinciding with a relative high
level in the distributions of the dimensionless velocity and wall shear stress. The flow
disturbance behind the holes is stronger. This can be related to the appearance of vortex S
which may be attributed to the strong suction effect of the film holes.

4.3 Overall discussion

Figure 10 provides the relative area averaged Nusselt numbers for different cases and
different Surfaces 1–3 at Re = 20,000. Surface 1 is set as the region of \( \theta = -25^\circ \) to \( 25^\circ \);
Surface 2 is defined the region of \( \theta = -45^\circ \) to \( 45^\circ \); and \( \theta = -90^\circ \) to \( 90^\circ \) is for the region of
Surface 3. \( Nu_{ave,b} \) is the averaged Nusselt number of the baseline. The detailed surface
Nusselt number distributions with skin-friction lines of quarter part are provided in
Figure 11. Surface 1 is circled by a red rectangle and the blue one is for Surface 2.

As presented in Figure 10, \( Nu_{ave}/Nu_{ave,b} \) for Surface 1 shows an increase as the film
arrays are positioned at \( \theta = 15^\circ \), whereas a decrease appears from \( \theta = 15^\circ \)–\( 35^\circ \) and finally a
relative constant level is found up to \( \theta = 55^\circ \). By comparing baseline and Case 1, it is
observed that the high values of region A and the film region are roughly the same. The

Figure 8.
(a) Skin-friction lines
with topology picture,
(b) Nusselt number,
(c) dimensionless
velocity, (d) wall
shear stress and
relative turbulence
kinetic energy (TKE)
near target wall of
baseline. Red line-
attachment line; blue
line-separation line
Figure 9.
(a) Skin-friction lines with topology picture, (b) Nusselt number, (c) dimensionless velocity, (d) wall shear stress and relative turbulence kinetic energy (TKE) near target wall of Case 5. Red line-attachment line; blue line-separation line.

Figure 10.
Relative area averaged Nusselt number averaged from Surfaces 1–3 at $Re = 20,000$. $\text{Nu}_{\text{ave},b}$ is the number of baseline.
appearance of the relative high value region C might be due to the secondary vortex $S_{12}$. The following decrease as $\theta$ changes from 15° to 35° can be related to the absence of a high region around the film holes due to the suction effect. The constant value of $\frac{Nu_{ave}}{Nu_{ave,b}}$ for Cases 3–5 can be due to the relative similar distributions for Surface 1. Thus, it can be confirmed that the arrangement of side array film holes provides little influence on the heat transfer enhancement of Surface 1 as $\theta > 35°$. For Surface 2, the averaged values, $\frac{Nu_{ave}}{Nu_{ave,b}}$, increase from 1 to 1.05 as $\theta$ changes from 0° to 35°, then decrease to 0.999 as $\theta$ changes from 35° to 55°. As shown in Figure 10, this increasing trend can be attributed to the absence of a low heat transfer region coinciding with the low energy region C [marked in Figures 8(c-e) and 9(c-e)] and enlarging the relatively high Nusselt number region due to the secondary vortex $S_{12}$. The value is less than unity for Case 5. This can be related to the appearance of the low heat transfer region due to the low energy region C [marked in Figures 9(c-e)] and the absence of high Nusselt numbers around the film holes. Concerning the whole surface averaged parameter, as shown in Figure 10, $\frac{Nu_{ave}}{Nu_{ave,b}}$ increases with increasing $\theta$. This can be related to the strong suction effect of the film holes on the flow behind the holes along the stream-wise direction. The farther the film holes are arranged from the center jets, the stronger is the flow disturbance generated behind the film holes’ region, where the heat transfer is more enhanced (Figure 11). For a specific case, $\frac{Nu_{ave}}{Nu_{ave,b}}$ increases by enlarging the evaluated surface (Surface 1 to Surface 3) for Cases 3–5 ($\theta$ from 35° to 55°). It can be inferred that the far away side film holes play an effective part for the heat transfer uniformity.
Figure 12(a) illustrates the averaged Nusselt number for Surfaces 1–3 and the relative pressure with varying Reynolds number and film holes arrangement. Generally, $Re$ plays an unquestionable positive role in enhancing the heat transfer for all arrangements. The effect of the film holes arrangement on the heat transfer enhancement is independent of the Reynolds number. In Figure 12(b), a comparison of the area-averaged heat transfer coefficients of the target wall subject to the pumping power is provided. As illustrated by the curves, the heat transfer coefficient is slightly increased with an increasing $\theta$ from $0^\circ$ to $55^\circ$ at a constant pumping power. It could be inferred that the far away arrangement of the side film holes to the center jets shows a positive influence in increasing the thermal performance in this work.

5. Conclusions
In this study, the effects of arrangement of film holes on fluid flow and heat transfer characteristics are provided for jet impingement system conjugated with film cooling in a semicylinder double wall channel by numerically simulations. The detailed flow structures, flow characteristics influence on the local heat transfer distributions for two typical cases are discussed. The results show that the far away staggered film holes arrangement plays a...
positive role in heat transfer uniformity and thermal performance. The heat transfer uniformity and thermal performance are significantly improved for the cases of 35°–55° film holes’ arrangement angle compared with the baseline due to the flow disturbance enhanced by the films suction effect. The case of 55° shows the most improved heat transfer uniformity and the thermal performance is the highest in this study.

References


Further reading

Corresponding author
Bengt Ake Sundén can be contacted at: bengt.sunden@energy.lth.se