EFFECT OF VELOCITY RATIO ON FLOW AND HEAT TRANSFER CHARACTERISTICS OF AN IMPINGING JET IN CROSSFLOW

M. Wae-hayee 1, C. Nuntadusit 1*, P. Tekasakul 1
1. Department of Mechanical Engineering, Faculty of Engineering, Prince of Songkla University, Hatyai, Songkhla, Thailand
*Authors to correspondence should be addressed via email: chayut@me.psu.ac.th

Abstract: The effect of velocity ratio (VR, ratio of jet velocity to crossflow velocity) on flow and heat transfer characteristics on an impinging jet with crossflow was investigated. The air jet issued from circular orifice and then impinged normal to heated surface in wind tunnel. The jet-to-plate distance (H, wind tunnel height) was fixed at 2D (D is orifice diameter). The velocity ratios were varied from VR=3, 5 and 7. The temperature distribution on an impinged surface was visualized by using thermochromic liquid crystal sheet (TLCs) and Nusselt number distribution was evaluated with image processing technique. The flow patterns on impinged surface were visualized by using oil film technique. The numerical simulation was also employed to gain insight into the fluid flow of jet impingement in the crossflow. The results show that the averaged Nusselt number was highest in case of VR=5. The heat transfer in jet impingement region was enhanced when increasing VR from 0 to 5 and the heat transfer was decreased again when increasing VR from 5 to 7. The locations of maximum Nusselt number were shifted to downstream direction due to crossflow and the interaction between impinged jet and cross flow near the wall was revealed.

Key Words: Impinging jet, Crossflow, Thermochromic liquid crystal sheet, CFDs

1. INTRODUCTION
Jet impingement is a high-performance technique for heat transfer enhancement in thermal equipment. It has been also used in industrial process for a heating, cooling or drying. It provides for rapid cooling or heating on local heat transfer area. However, the heat transfer rate is very high on jet directly impinged area. Many of thermal industrial applications have large heat transfer area such as, a combustor wall chamber and gas turbine blade cooling, steel and glass quenching, textile and paper drying. A high and uniform heat transfer rate is required over entire that areas. Hence, multiple of impinging jets are applied.

Multiple of impinging jets are formed by number of individual jet impingements. The flow and heat transfer characteristics of multiple impinging jets are influenced by two interactions. First is an interaction between adjacent before impingement wall and secondly, the wall jets formed by the adjacent jets collide on the target surface [1]. Another parameter that influences multiple jet impingements in a confined space is the crossflow. The crossflow is defined as the fluid flow in the direction perpendicular to the jet impingement flow. The crossflow can be either due to external flow resource or due to accumulated spent jet fluid flow. The crossflow has been found to significantly reduce the heat transfer of impinging jet in downstream [2, 3].

Goldstein and Behbahani [4] studied the single jet impingement from a circular pipe orifice for case of with and without crossflow. The results show that a maximum Nusselt number decreases with increasing flux momentum of crossflow for jet-to-plate distance H=12D (D is orifice diameter) and the maximum Nusselt number increases, when decreased the jet-to-plate distance to H=6D with moderate flux momentum of crossflow. Bouchez and Goldstein [5] experimentally studied the local heat transfer on the impinged surface and flow visualization of the jet in the crossflow, the experiment carried out of the single jet impingement from a circular pipe orifice normal to a surface, the results show that the low velocity of crossflow can created a recirculation zone upstream of the stagnation point and the heat transfer coefficient for jet-to-plate distance H=6D has higher than H=12D for all flux momentum ratios.

Barata and Durao [6] investigated the flow characteristic of an impinging jet in crossflow. They found that the upstream side of wall jet interacted with the crossflow and formed a vertex close to the ground target plate which flow was similar to the horseshoe structure. Nakabe et al. [7] experimentally studied a single inclined impinging jet in the crossflow with jet to crossflow velocity VR=3, 5 and 7 and showed the increasing heat transfer on the target surface in case of
high crossflow velocity VR=7. Yang and Wang [8] conducted the numerical simulation of an inclined impinging jet in crossflow with same experimental condition of Nakabe et al. [7]. The results indicated that for the case of the low velocity ratio (VR=3), there appeared very strong circulation flow near the stagnation region when compare with other velocity ratio (VR=5 and 7).

All literature reviews have been briefly discussed in above. It was found that the crossflow significantly reduced or enhanced the heat transfer of impinging jet according to the ratio of jet to crossflow velocity and jet-to-plate distance. Most of the previous works on impinging heat transfer under the crossflow are concerned with jet from a pipe nozzle [4, 5, and 6]. In practical industrial applications, the heat transfer surface is large and the multiple impinging jets were must applied with orifice type nozzle. Hence, the interacted characteristic between the jet and the crossflow are unlike for case of single and multiple impingements. Also, the flow and heat transfer characteristics in case of jet impingement from the pipe and orifice nozzle are difference.

In case of multiple of jet impingements, the maximum heat transfer on the target surface was 2D-3D of jet-to-plate spacing (according on the jet-to-jet spacing) [2, 3 and 9]. While, the maximum heat transfer on the target surface in case of single jet impingement was 5D-8D of the jet-to-plate distance, according on the nozzle type and confined or unconfined of the test section [10, 11 and 12]. From these reasons, the effect of crossflow on jet impingement from orifice nozzle with low jet-to-plate distance (H=2D-3D) should be concerned than the jet impingement from the pipe nozzle with high jet-to-plate distance (H>6D).

The aim of this research was to study the effect of velocity ratio on flow and heat transfer in case of impinging jet from orifice nozzle with low jet-to-plate distance H=2D. The experimental investigation was carried out of the jet to the crossflow velocity ratios VR=V_/V_j =3, 5 and 7. The temperature distribution on the impinged surface was investigated by using TLCs and Nusselt number distribution was evaluated by using image processing technique. The flow characteristics on the impinged surface were visualized by using oil film technique. The numerical simulation was employed to gain insight into the fluid flow of jet impingement in the crossflow by using commercial CFD software (ANSYS ver. 12.0).

2. EXPERIMENTAL MODEL AND PARAMETERS

The experimental model in this study, the jet was discharged from a circular orifice and then impinged normal to opposite heated surface in wind tunnel with rectangular cross section as show in Fig.1. The crossflow was generated by sucking air pass through the test section with centrifugal blower located downstream of wind tunnel. To consider the effect of jet to crossflow velocity (VR) on flow and heat transfer characteristic on the impinged surface, the jet flow was fixed at a constant flow rate and the crossflow velocity was varied. An origin of the Cartesian coordinates was located on the impinged surface as shows in Fig.1. The X-axis, Y-axis and Z-axis are the streamwise of crossflow, normal to streamwise and spanwise direction of wind tunnel, respectively.

The experiment was carried out at orifice diameter D=13.2 mm and jet-to-plate distance H=2D. The comparisons for flow and heat transfer characteristics on the impinged surface were based on the constant jet velocity (at Re=V_D/ν=12,700) and varied crossflow velocity, corresponding to velocity ratio between the jet and the crossflow VR=3, 5 and 7.

![Fig. 1. Experimental model of an impinging jet in a crossflow](image)

3. EXPERIMENTAL SETUP AND METHOD

3.1. Experimental setup

Fig.3. shows a schematic view of the experimental apparatus. The experimental apparatus composed of two parts: jet flow supplied part and crossflow supplied part. For the jet flow supplied part, the centrifugal blower (3HP) accelerates the air which then flows through a temperature controlled chamber and towards the orifice flow meter. The air subsequently passes through a jet chamber with constant cross-section 360-mm-wide, 360-mm-long and 850-mm-high. The jet chamber was equipped with two layers of perforated plates and two layers of mesh plates to ensure that a uniform flow field approaches to the orifice plate.

The crossflow in the wind tunnel was sucked through the inlet chamber, a flow straightener, two of mesh plates, the test section and an outlet chamber with centrifugal blower (3HP) located at downstream of wind tunnel. The wind tunnel has rectangular cross-section 300-mm-width and the height is 2D (Aspect ratio is 11.4.). The wind tunnel has sufficient length to ensure that the flow passes through the test section with a fully developed velocity profile. In addition, the inlet chamber and wind tunnel was assembled by convergent connection to reduce the effect from wind tunnel inlet.

The test section was mounted upon the jet chamber and its dimension was 139-mm-wide and 26.4-mm-high.
The surface of heat transfer measurement (Opposite site of jet plate) of test section was designed for replaceable with transparent acrylic plate for the flow visualization technique. As well as, the Pitot-static tube was mounted before test section for the crossflow velocity profile measurement. For all experimental conditions, the jet and crossflow were controlled with constant temperature at 27°C and temperature variation of the jet and the crossflow was controlled within 0.2°C.

3.2. Heat transfer measurement

Fig. 3 shows the detail of test section for heat transfer measurement. The air with constant temperature discharged from an orifice plate and impinged upon the heat transfer surface. The heat transfer surface was made of stainless steel foil (30-µm-thicknesses) which attached with TLC sheet on the rear side of jet impinged surface. The stainless steel foil was stretched between couple of copper bus bars. The heat transfer surface was heated by DC power supply that can supply current up to 40A passes through copper bus bars. An amount of electrical energy was dissipated in the stainless steel foil and it can be calculated from equation

\[ \dot{Q}_{\text{input}} = I^2 \cdot R \]  

where here, \( I \) is the electrical current and \( R \) is the electric resistance of stainless steel foil.

The heated impinged surface was cooled with impinging jet. Hence, the local values of heat transfer coefficient (h) by force convection of jet can be evaluated from equation

\[ h = \frac{\dot{Q}_{\text{input}} - \dot{Q}_{\text{losses}}}{\dot{Q}_{\text{input}} - \dot{q}_c} = \frac{\dot{Q}_{\text{input}} - \dot{q}_c}{(T_w - T_j)} \]  

where \( \dot{q}_c = \sigma \varepsilon_{TLC}(T_w - T_j) \) and \( \dot{q}_c = h_c(T_w - T_j) \) are the heat loss transferred to the environment by radiation and convection, respectively. The \( T_w \) and \( T_j \) are the wall and jet temperature, the \( \sigma \) is a Stefan-Boltzman constant, the \( \varepsilon_{TLC} \) is a emissive coefficient of the black background paint and the TLCs that has been given in [9], \( T_r \) is a surrounding temperature and \( h_c \) is a natural heat transfer coefficient that was calculated from natural convective heat transfer from the horizontal plate orientation to the surrounding.

The wall temperature \( (T_w) \) on the impinged surface was measured by using TLCs that attached on the rear side of jet impinged surface. The CCD camera was used to capture colour on TLCs. The images of colour pattern on TLCs were then converted from the RGB (Red, Green and Blue) colour system to the HSI (Hue, Saturation and Intensity) colour system. The \( H \) value provides a convenient way to correlate the colour of TLCs to their
temperature in range of 28-40°C. The TLCs was calibrated with same location on the test section to keep all external factors same with the heat transfer experiment. The local Nusselt number was calculated from

\[ Nu = \frac{hD}{k} \]  

(3)

where \( D \) is the diameter of orifice and \( k \) is a conductivity of air jet. An average Nusselt number was calculated from

\[ \overline{Nu} = \frac{\overline{hD}}{\overline{k}} \]  

(4)

where here, the average heat transfer coefficient \( \overline{h} \) was calculated from Eq.(2) by replacing \( T_w \) to \( \overline{T_w} \) that is a averaged temperature on the impinged surface.

3.3. Flow visualization on the impinged surface

The flow visualization on the impinged surface was illustrated by using oil film technique. The oil film was prepared by liquid paraffin, titanium dioxide and oleic acid. A transparent plastic plate was replaced to the impinged surface and oil filme was painted uniformly on jet impinged surface. The CCD camera was captured the oil film flow on the impinged surface at each different time (30 second/frame).

4. NUMERICAL SIMULATION (CFD)

The flow characteristic was illustrated by using 3-D numerical simulation (ANSYS ver.12.0). The model of numerical simulation is same with the experimental model; dimension, mass flow rate of jet and crossflow, boundary and experimental condition. The standard k-\( \varepsilon \) turbulent model with general wall-function mode was used for solving numerical simulation problems.

5. RESULTS AND DISCUSSION

5.1 Velocity profile of crossflow

Fig.4 shows the velocity profile of crossflow in Y-axis pass the center of wind tunnel(Y is wind tunnel height). The velocity profile shows a good agreement between the CFD and experimental data. The trend of both data illustrate that the flow before enter to the test section with fully developed flow. From this result, the flow characteristic of crossflow before enter to the test section of both experiment and CFD are almost matched.

5.2 Flow characteristic of jet and crossflow

Fig.5 shows the velocity distribution in ZX-plane through the center of jet (Y=0). The jet discharged from the nozzle and impinged upon the target surface. Before jet impinging on the wall, the crossflow deflected the jet to downstream side of crossflow. The tendency of deflection of jet depends on the crossflow velocity or VR. The jet was more deflected to the downstream of crossflow when the VR decreases.

In case of VR=5 and 7, the wall jet was appeared on both side of upstream and downstream of jet impingement region. This condition was different from the VR=3 which the wall jet was appeared only downstream.

Fig. 5 (b) and (c) show the point of velocity approach to zero which nearly located to the target surface in middle of jet impingement region for VR=3 and 7. This represented the stagnation point. It was found that the stagnation point was shifted to the downstream when the crossflow velocity was increased. However, the stagnation point in case of VR=3 was obscured.

![Fig. 5. Velocity contour in ZX-plane (Y=0) at different VR](image)

Fig.6 shows streamline in ZX-plane at Y=0. The results obviously illustrated the interaction between jet and crossflow. In case of VR=5 and 7 shows the ground vortex in the upstream of jet impingement region. The ground vortex of VR=7 is larger than VR=3, because of the wall jet can be penetrated in the upstream of crossflow about 3.3D then the wall jet collided with crossflow and turn to downstream as shows in Fig. 6 (c). Difference from in case of VR=5, the wall jet can penetrate in the crossflow about 2.5D as shows in Fig. 6 (b). The dimension of ground vortex depends on the distance of wall jet that can be penetrated as much as in the upstream direction.

In case of higher velocity of crossflow VR=3 (Fig. 6 (a)), the ground vortex was disappeared. But, the wall jet can be penetrated in the crossflow about 1.2D in the upstream and then rapidly turn to the downstream. In this case, the velocity of crossflow dominated the velocity of wall jet.
Fig. 7 shows the streamline in ZX-plane at Y=-6 mm from the jet exit. The result shows that the crossflow passed through the jet flow with difference flow characteristics according to the VR. In case of VR=3, the circulation flow was appeared in downstream side of jet. This circulation flow can be promoted the turbulent flow inside the jet before impingement.

Fig. 6. Streamline in ZX-plane (Y=0) at different VR

Fig. 7. Streamline in ZX-plane at Y=-6 mm from the jet exit (Solid circle are the position of orifice).

5.3 Flow and heat transfer on the impinged surface

Fig. 8 shows the flow visualization on the impinged surface by using Oil film technique. The black area represented oil film completely removed wall region and white area represented area of oil film. The impingement region has black area due to high shear stress on the surface. The oil film was removed from this region. Small white area in the middle of black area represents the stagnation point of jet with velocity almost zero. In case of without crossflow and case of VR=7 with lowest crossflow velocity (Fig. 8 (a) and (d)), the stagnation point was clearly expressed. For case of VR=3, the stagnation point unclearly expressed as shown in Fig. 8 (b), because of highest velocity of crossflow (jet impinged on surface weakly).

From the Fig.8, the distance of stagnation point shifted far away from the center of orifice 0.15D, 0.25D and 0.5D for VR=7, 5 and 3, respectively. The distance from the stagnation point to the central orifice increased with decreasing the VR, corresponding to the numerical simulation model that shows in Fig.5 as has been disscused in above.

Fig. 8. Oil film patterns on an impinged surface at different VR

Fig. 9 shows the local Nusselt number distribution on the impinged surface. The area of high heat transfer in jet impingement region became smaller in spanwise direction for VR=3 and become larger for VR=5 and 7. This result is consist with the area of stagnation region that has been shonw in Fig. 8. The distance of stagnation region in centerline in spanwise are 3D, 5D, and 5.6D for VR=3, 5 and 7, respectively. This results illustreted the area of stagnation region become smaller as the crossflow velocity increasing.

The heat transfer peak in stagnation region for case jet without crossflow is lower than other case of jet with crossflow. For case jet with crossflow, the peak of heat transfer increased with increasing velocity of crossflow. The variation of heat transfer peak was obviously as shown in Fig.10. This heat transfer enhancement is attributed to the interaction between the jet and the crossflow which increased turbulent intensity in the jet before impingement, corresponding to the numerical simulation model that shows in Fig.7.

Table 1 shows the variation of averaged Nusselt number on the impinged surface that calculated from the equation (4). The VR=5 has highest averaged Nusselt number, because of the high local Nusselt number are appropriated between stagnation region and around it. So, difference from the VR=5 that shows the peak of heat transfer only at stagnation point.
(2) The higher velocity crossflow can increase the peak of heat transfer in stagnation region. This is attributed to the interaction between the jet and the crossflow which increased turbulent intensity in jet before impingement.

ACKNOWLEDGEMENT
This research was sponsored by Faculty of Engineering, Prince of Songkla University through grant No. ENG-53-2-7-02-0070-S.

REFERENCES

Table 1. Averaged Nusselt number

<table>
<thead>
<tr>
<th>VR</th>
<th>3</th>
<th>5</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Averaged Nusselt number</td>
<td>72.7</td>
<td>78.1</td>
<td>72.9</td>
</tr>
</tbody>
</table>

5. CONCLUSION
The main results are shown as follows;
(1) The jet was more deflected to the downstream of crossflow when the crossflow velocity increasing and the ground vortex in upstream of VR=7 larger than in case of VR=3.