Study on heat transportation of an impinging jet interfering with a Couette flow
(comparison with Poiseuille flow)

Jumpei KITAYAMA* and Takashi KUBO**
*Department of Mechanical Engineering, Meijo University
1-501 Shiogamaguchi, Tempaku-ku, Nagoya 468-8502, Japan
E-mail: Jumpei_Kitayama@mazak.co.jp
** Department of Mechanical Engineering, Meijo University

Received: 6 February 2019; Revised: 14 April 2019; Accepted: 20 May 2019

Abstract
In this study, we compare the flow and temperature fields of a heated impinging jet interfering with a Couette flow and a Poiseuille flow through numerical simulations. We find that the instantaneous vortex structure in the Couette main flow has some periodicity and affects the heat transfer on the jet impinging surface. In time-averaged flow and temperature fields, the Nusselt number shows unique distributions. Further, a counter-rotating vortex pair is extracted, similar to the previously reported studies on a jet in a cross flow. It is found that this vortex pair involves the main flow fluid and causes a unique Nusselt number distribution. The overall heat transfer in the Couette main flow is lower than that in the Poiseuille main flow. From the mean velocity field, a back current occurs behind the jet inlet, which is similar to the separation in the wake flow of a cylinder crossing the main flow. In addition, the fluctuation velocity in the Couette main flow does not spread wider than that in the Poiseuille main flow, and high velocity fluctuations concentrate on the front edge of the jet flow. The distributions of both mean and fluctuation velocities of the Couette flow interfering with the jet are similar to those of the turbulent Couette flow.

Keywords: Impinging jet, Couette flow, Interfering flow, Heat transfer, Counter rotating vortex pair

1. Introduction

An impinging jet exhibits good heat and mass transfer characteristics, and thus, is widely used in industry for the cooling of heating devices. To improve these characteristics, studies such as the application of periodic fluctuation to the jet (Liu and Sullivan, 1996; Tsujimoto et al., 2009) or multiplexing and intermittently controlling the jets (Jinno et al., 2016) have been conducted. However, in an actual engineering application, the flow fields are often composed of not only the impinging jet, but other flows as well. Thus, several researchers have studied the impinging jet crossing the main flow. Bouchez and Goldstein (1975) conducted a visualizing experiment and compared the flow fields for the various velocity ratios between the main flow and jet or impinge height as the experimental parameters. Previous studies have compared the heat transfer with and without the main flow or impinge height (Sparrow and Goldstein, 1975; Goldstein and Behbahani, 1982). These studies explored the heat transfer in only one dimension – the direction of the main flow. In addition, Metzger and Korstad (1972) discussed the effect of the jet layout on heat transfer.

Despite a fundamental flow field found in the cooling of the rotary bearing (Onda, 2012) or carrier gas flow of additive manufacturing, no study has focused on the heat transfer field of impinging jets interfering with Couette flows. Benmouhoub and Mataoui (2014) studied jet impingements on a moving surface using two-dimensional Reynolds averaged Navier–Stokes simulation, defining the flow configuration and organizing the mean Nusselt number with the velocity ratio between the jet and the moving surface. However, they focused only on the moving surface, excluding the Couette flow. Therefore, in this study, we conducted numerical simulations of the impinging jet interfering with the Couette flow, or Poiseuille flow, and compared the flow fields and their heat transfers.
2. Numerical simulation method and its verification

2.1 Numerical simulation method

In this study, direct numerical simulation (DNS) was used to obtain the unsteady flow and temperature fields. The governing equations include the incompressible Navier–Stokes equation, equation of continuity, and heat transfer equation.

\[
\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 u_i}{\partial x_i \partial x_j}
\]  
\[
\frac{\partial u_i}{\partial x_i} = 0
\]  
\[
\frac{\partial T}{\partial t} + \frac{\partial u_i T}{\partial x_j} = \frac{1}{Re \cdot Pr} \frac{\partial^2 T}{\partial x_j \partial x_j}
\]

(1)  
(2)  
(3)

The order of the Richardson number was estimated at \(O(Ri) \sim 10^{-2}\), so the temperature was dealt with as a passive scalar to the flow field. These equations are non-dimensional owing to the characteristic values of the jet.

In this simulation, we discretized the equations on a staggered grid and solved them using a finite-difference method. The velocities and pressures were coupled using a marker-and-cell method, and a successive over-relaxation method was applied to solve the Poisson equation for pressure. We applied a secondary conservative second-order central-difference scheme to all terms of the equations and integrated time using the fourth-order Runge–Kutta method.

2.2 Simulation verification

As mentioned in the Introduction, there have been no previous studies focusing on a jet crossing the Couette flow. The order of the Reynolds number in a similar study (Goldstein and Behbahani, 1982) was too high, at \(10^5\), so it was difficult to virtually solve the same flow field through direct numerical simulation. To verify the present simulation method, we compared several physical quantities of normal impinging jets with those in previous studies. Figure 1 shows the simulation domain and boundary conditions, and Table 1 presents the simulation conditions. Using the inlet diameter of the jet as the representative length, the simulation domain lengths were found to be \(L_x \times L_y \times L_z = 10 \times 2.5 \times 10\). The spatial resolutions of the horizontal directions \(\Delta x/D\) and \(\Delta z/D\) were both 0.05. To confirm the lattice dependency, we simulated two cases with spatial resolutions of vertical direction \(\Delta y/D\) as 0.025 and 0.0125. The lower boundary, as the impingement surface, had a non-slip and constant temperature, and the free out boundary condition (FOBC) was applied to the four side boundaries. The jet blew from the center of the upper boundary, which was surrounded by a side flow. The velocity of this side flow was 7.5\% of the jet velocity \(V_{in}\), which is the same as Tsujimoto et al. (2009). Both these jets and side flows were heated, and this representative temperature difference \(\Delta T\) was 10 K from the lower constant-temperature boundary.

![Fig. 1 Schematic of a normal impinging jet simulation. The jet is heated and impinges to a lower surface, which has a non-slip, constant temperature boundary. All side boundaries exhibit FOBC. In addition, the jet is surrounded with a side flow, which has a velocity of 7.5% of the main jet, identical to the reference study by Tsujimoto et al. (2009).](image)
Figure 2 shows the mean flow velocities in the x-direction, $u_m$, along with the vertical direction $y$ at each radial location $x/D$. Figure 2(a) shows the results of the spatial resolution in the vertical direction with $\Delta y/D$ equal to 0.025, and Fig. 2(b) shows the results with $\Delta y/D$ equal to 0.0125. The abscissa is normalized with half width of $b_{0.5}$, and the ordinate is normalized with the maximum velocity $u_{max}$. The white circles indicate the DNS result of the impinging jet with Reynolds number $Re = 1500$ and an impinge height $H/D = 4$ obtained by Tsujimoto et al. (2009). In the wall jet region, the velocity distribution at each location shows self-similarity, and these are similar to Tsujimoto et al.'s (2009) results. Here, the distribution over $y/b_{0.5} = 1.25$ at $x/D = 4$ differs from the result obtained by Tsujimoto et al. (2009). We consider this difference to be caused by the difference in the boundary conditions; FOBC is applied to the side boundaries in this simulation, while Tsujimoto et al. (2009) used suction boundary conditions. Comparing these two graphs, despite the roughness in the distribution under $y/b_{0.5} = 0.5$ in Fig. 2(a), all distributions correspond to those in Fig. 2(b) quantitatively; therefore, the grid dependency is strictly limited. Omitted in this paper are the data showing that both velocity distributions, which are not normalized with $b_{0.5}$ and $u_{max}$, are completely in agreement.

The radial distributions of the time-averaged Nusselt numbers are shown in Fig. 3. Here, the heat transfer coefficient, which is a component of the Nusselt number, is calculated based on Newton’s law of cooling with linear approximation in the y-direction near the wall. The radial location in the abscissa is normalized by the jet inlet diameter $D$, the white circles indicate the experimental result of the impinging jet with Reynolds number $Re = 1000$, and the impingement height $H/D = 4.5$ obtained by Angioletti et al. (2005) is used for comparison. It is known that the Nusselt number has a peak of $x/D = 0.5$, slightly apart from the stagnation point (Shakouchi, 2004), and both results have a peak at $x/D = 0.6$. There is some quantitative difference caused by the Reynolds number in the area between the stagnation point and $x/D = 1.5$; however, the results are qualitatively similar.

Figure 4 shows the Nusselt number at the stagnation point arranged with the impinge height, Prandtl number, and...
Reynolds number. The abscissa is the impinge height, which is normalized with the jet inlet diameter $D$. The white circles indicate the experimental results obtained by Liu and Sullivan (1996), while the red triangles indicate the results obtained from the current simulation. Liu and Sullivan did not perform their experiment using the same conditions as those in this simulation, i.e., $H/D = 2.5$; however, the results of this report do not show a contradiction between $H/D = 2$ and $H/D = 3$. Shadlesky (1983), through theoretical analysis based on the potential flow, indicated that the Nusselt number at the stagnation point obeys the function $Nu_0 / (Pr^{0.4}Re^{0.5}) = 0.585$; as a result, the results obtained in this simulation, $Nu_0 / (Pr^{0.4}Re^{0.5}) = 0.63$, are considered appropriate. Here, we confirmed that the jet has a potential core just on the wall.

![Graph](image_url)

**Fig. 3** Radial distribution of local Nusselt number. The white circle indicates the experimental result ($Re = 1000$, $H/D = 4.5$) obtained by Angioletti et al. (2005). Both the present simulation and the experimental results show the maximum value at $x/D = 0.6$. The present simulation value is lower than the experimental value due to the difference in the Reynolds number; however, both results qualitatively agree approximately.

![Graph](image_url)

**Fig. 4** Nusselt number at stagnation point as a function of $H/D$. The white circle indicates the experimental result ($Re = 12271$) obtained by Liu and Sullivan (1996), and the broken blue line indicates the potential analysis conducted by Shadlesky (1983). The present result plotted by red triangle approximately locates the midpoint of the experimental results.

Based on these results, this simulation method is considered to have the ability to represent the flow and heat transfer fields of a normal impinging jet. This study focuses on an impinging jet crossing a Couette or Poiseuille main flow. Because the jet was advected downward, the vertical velocity gradient $\partial v/\partial y$ near the impinge wall was smaller than that of the normal impinging jet flow field. Therefore, the simulation accuracy of the impinging jet crossing a main flow is considered sufficient, given that the accuracy of the normal impinging jet is confirmed. Finally, the study simulated the interfering flows using a vertical spatial resolution $\Delta y/D = 0.025$.

### 3. Numerical simulation of interfering flow

#### 3.1 Simulation condition

The simulation domain and conditions are shown in Fig. 5 and Table 2. Figures 5(a) and 5(b) show the cases of Poiseuille main flow and Couette main flow, respectively. The jet blows vertically from the upper surface to the lower surface, and the origin is defined at the counter point of the jet inlet location. The Reynolds number of the jet is 926, based on the jet inlet diameter $D$ and jet flow velocity $V_m$ as the representative values. This jet has uniform velocity distribution. The simulation domain lengths are $x/D = 15$ (main flow direction), $y/D = 2.5$ (main flow height), and $z/D = 10$ (main flow spanwise). The spatial resolutions are $\Delta x/D = \Delta z/D = 0.05$ as the horizontal directions and $\Delta y/D = 0.025$ as the vertical direction. The simulation time that is non-dimensionalized by $D$ and $V_m$ is 350, and the time resolution is $7.0 \times 10^{-3}$. Here, all physical properties related to the fluid, such as kinematic viscosity or Prandtl number, are based on air at 1 atm and 293.15 K.

The upper surface except the jet inlet in both main flow and lower surface in case of the Poiseuille main flow are
applied the non-slip boundaries. The lower surface, in case of the Couette main flow, moves left to right in Fig. 5(b) at velocity $U_{\text{plate}} = V_{\text{in}}$. The main flow blows from the left boundary in the figures with the velocity distribution of Poiseuille flow or laminar Couette flow. The remaining three side boundaries are applied FOBC. It is confirmed that the error of the flow volume balance is slightly limited. In addition, the flow out volume passed through the both side boundaries is 1.4 % of the total incoming flow volume (the jet and the main flow) in the Couette flow case, while it is 13.4 % in the Poiseuille flow case. Here, the center velocity of the Poiseuille flow, $U_{\text{Poiseuille}}$, is set to $(3/4)V_{\text{in}}$ to have the same flow volume as that of the Couette flow. The Reynolds numbers of the Poiseuille and Couette flows are $Re_{\text{Poiseuille}} = 1736$ and $Re_{\text{Couette}} = 2315$, respectively.

Because the transitional Reynolds number of the Couette flow takes several values by different researchers (Tillmark and Alfredsson, 1992; Nakabayashi et al., 1988), there is no unified view on the Reynolds number. However, Nakabayashi (1988) reported that the Couette flow maintains its laminar velocity distribution up to $Re = 3150$. This simulation confirmed that a normal Couette flow, with $Re_{\text{Couette}} = 2315$, has a laminar velocity distribution.

Both the initial temperature and main flow inlet temperature are 293.15 K, and the jet is heated with a representative temperature difference of 10 K. The lower surface, which the jet impinges, is treated as a constant-temperature wall.

### Table 2 Simulation conditions of interfering flows

| Parameter                                      | Value          |
|------------------------------------------------|----------------|
| Jet inlet diameter $D$                         | (representative value) |
| Jet inlet velocity $V_{\text{in}}$            | (representative value) |
| Center velocity of Poiseuille flow $U_{\text{Poiseuille}}$ | $(3/4)V_{\text{in}}$ |
| Moving plate velocity $U_{\text{plate}}$      | $V_{\text{in}}$ |
| Jet Reynolds Number $Re_{J}$                  | $926$ |
| Poiseuille flow Reynolds Number               | $1736$ |
| Couette flow Reynolds Number                  | $2315$ |
| Richardson number $R_i$                       | $1.37 \times 10^2$ |
| Prandtl number $P_f$                          | $0.71$ |
| Simulational time $t/(D/V_{\text{in}})$       | $350$ |
| Time resolution $dt/(D/V_{\text{in}})$        | $7.0 \times 10^{-3}$ |
| Spatial resolution $(\Delta x/D \times \Delta y/D \times \Delta z/D)$ | $0.05 \times 0.025 \times 0.05$ |

### 3.2 Comparison of vortex structures and local Nusselt number distributions

Figure 6 shows the instantaneous vortex structures at a steady state after 280 non-dimensional time has passed. The vortex structures are represented by the isosurface of the second invariant of the velocity gradient tensor, $Q = 1.0$. Here, these figures are reversed about the $y$-direction in order to view the condition near the wall that the jet impinges. Figure 6(a) shows the case of the Poiseuille main flow and Fig. 6(b) indicates the Couette main flow. Upon initial observation, there is no noticeable difference between the two vortex structures. However, differences in detail, such as the angle of
the jet inclining in the Poiseuille flow, shows that it is larger than that of the Couette flow. The vortex structure in the Couette flow has some periodicity, and multiple longitudinal vortexes are stretching, whose axes are inclined with respect to the $x$-axis in the Couette flow. Moreover, this periodicity of the vortex structure affects the temperature distribution near the wall, as indicated in the instantaneous local Nusselt number distribution in Fig. 7(b). Regarding the stretching longitudinal vortexes, whose axes are inclined with respect to the $x$-axis; Fric and Roshko (1994) reported that there are similar vortexes in the wake region of a jet crossing a main flow.

Instantaneous local Nusselt number distributions are shown in Fig. 7. Unlike the vortex structure shown before, there is a large difference in the type of main flow. In the Poiseuille main flow shown in Fig. 7(a), a high heat transfer value concentrates around the jet impinge region. On the other hand, the periodic distribution of the Nusselt number is shown in the Couette main flow in Fig. 7(b), as marked with white arrows. This distribution corresponds to the periodicity of the vortex structure shown in Fig. 6(b). In addition, the maximum Nusselt number is located on $z/D = 0$ around the jet impinge region. However, the high Nusselt number distribution is split around $z/D = \pm 1.5$ like enclaves and the heat transfer on $z/D = 0$ is low in the wake region. The reason for this will be addressed later in this paper.

![Instantaneous vortex structures](image)

**Fig. 6** Instantaneous vortex structures. The vortex structure is extracted by the isosurface of the second invariant of the velocity gradient tensor $Q = 1$. Initially, both flow fields are similar. However, the angle of the jet axis in Poiseuille main flow is more inclined than that in Couette main flow. In addition, the vortex structures have some periodicity, and there are longitudinal vortexes similar to the wake vortex, which Fric and Roshko (1994) reported in their study on a jet crossing a main flow.
Addressing the time-averaged flow and temperature fields, Fig. 8 shows the local Nusselt number distributions, which are time-averaged during $10^5 - 300$ in non-dimensional time. Similar to those of instantaneous fields, there is a large difference in the type of main flow in the time-averaged Nusselt number distribution. In case of the Poiseuille main flow shown in Fig. 8(a), a high heat transfer value also concentrates around the jet impinge region, similar to that of the instantaneous field, and there are high heat transfer regions around $z/D = \pm 1.5$, similar to exclaves. In case of the Couette main flow shown in Fig. 8(b), the distribution has a unique feature, in that the heat transfer on $z/D = 0$ is low and takes its maximum value around $z/D = \pm 1$. The overall surface heat transfer of the Couette main flow is lowered by 20% as compared to the Poiseuille main flow.

Figure 9 shows the vortex structures of the time-averaged fields. The vortex structures are represented by the isosurface of $Q = 0.1$. Here, these figures are reversed along the $y$-direction in order to view the condition near the wall that the jet impinges. Unlike instantaneous fields, there is a large difference between both main flows. Figure 9(a) shows that the jet reaches the wall directly, and the hanging vortexes are derived. Because these hanging vortexes transfer the high temperature fluid to the near walls, the exclaves of the high heat transfer region shown in Fig. 8(a) is generated. On the other hand, in case of the Couette main flow in Fig. 9(b), there is a pair of large vortexes behind the jet around $z/D = \pm 1$, which take their axes to the main flow direction. In addition, some relation is suggested between this pair of vortexes and the local Nusselt number distribution shown in Fig. 8(b).

Fric and Roshko (1994) reported, in their experimental research on a jet crossing a main flow, that there is a vortex pair behind the jet in the main flow. Moreover, the same vortex pair is extracted in the time-averaged field in other numerical studies (Yuan et al., 1999; Tyagi and Acharya, 2003; Sakai et al., 2012), revealing that the heat transportation by this vortex pair aggravates the film cooling performance behind the jet inlet. The pair of vortexes shown in Fig. 9(b) is considered comparable to the vortexes identified in these previous studies.
3.3 Relation between vortex pair and heat transport mechanism

In this section, we discuss the effect of the vortex pair structure mentioned prior to the heat transport mechanism. Figure 10 shows the time-averaged vortex structures \((Q = 0.1)\) and the flow velocity vector in the cross section at \(z/D = 2.5\). In case of the Poiseuille main flow shown in Fig. 10(a), there is a pair of horn-like vortexes in the middle of the figure, as well as a circular flow, which is induced by the vortex pair. The vortex pair can be seen in the Poiseuille main flow, though it is weak. In case of the Couette main flow shown in Fig. 10(b), there is a large pair of vortexes, which is found from the flow velocity vector field that the strong circular flow takes as its axis in the center of this vortex pair. In addition, there is a pair of hanging vortexes in both flow fields, which are derived from the jet along the lower jet impinge wall.
Figure 11 shows the time-averaged velocity vector and temperature distributions at $x/D = 2.5$. In case of the Poiseuille main flow shown in Fig. 11(a), the high temperature fluid of the jet spreads in the vertical direction and the vortex pair circulates in this high temperature fluid. Furthermore, there is a relatively strong flow from the central area to the outer edge caused by a hanging vortex of around $z/D = \pm 1.5$. This flow transports the high temperature fluid from the central area to the lower wall. It generates the exclaves of the high heat transfer region shown in Fig. 11(a). In case of the Couette main flow shown in Fig. 11(b), the vortex pair involves the low temperature fluid of the main flow to the central area.
Figure 12 shows the $z$-direction distribution of the time-averaged temperature at several heights. The solid line indicates the distributions on $x/D = 2.5$, and the broken one does on $x/D = 3.5$. The temperature around $z/D = 0$ is low in the Couette main flow; the maximum temperature in the Couette main flow is located around $z/D = \pm 1.0$, while that in the Poiseuille main flow is located around $z/D = 0$.

The near-wall turbulent heat flux $(v' T')_m$ distributions in the $z$-direction are shown in Fig. 13. In the Couette main flow, $(v' T')_m$ has a positive or slightly negative value around $z/D = 0$, and a large negative value around $z/D = \pm 1.2$. A negative $(v' T')_m$ value indicates the transportation of high temperature fluid from the upper area to the bottom, or that of the low temperature fluid from the bottom to upper area, so the $(v' T')_m$ distribution consists with Nusselt number.
distribution. Therefore, this result indicates that the turbulent heat flux \((v' T')_m\) considerably affects heat transportation in this flow field.

Summarizing these results, the low temperature fluid is transferred to the lower wall, and drastically decreases the heat transfer on the \(x\)-axis shown in Fig. 8(b). Similar studies (Yuan et al., 1999; Tyagi and Acharya, 2003; Sakai et al., 2012) have reported that the heat transfer by this vortex pair drastically affects the wall heat transfer behind the jet inlet (corresponds to the upper surface in Fig. 11). According to the results of the present study, this pair vortex also affects the heat transfer on the wall that the jet impinges.

---

Fig. 12  The \(z\)-direction distribution of time-averaged temperature at several heights. The temperature around \(z/D = 0\) is low and the maximum temperature is located around \(z/D = \pm 1.0\) in the Couette main flow, while the maximum temperature is located around \(z/D = 0\) in the Poiseuille main flow.

Fig. 13  Near-wall turbulent heat flux \((v' T')_m\) distributions in the \(z\)-direction. In the Couette main flow, \((v' T')_m\) has a positive or slightly negative value around \(z/D = 0\), and a large negative value around \(z/D = \pm 1.2\). This means that the transportation of the high temperature fluid from the upper area to the near-bottom wall or of the low temperature fluid from the near-bottom wall to the upper area.
3.4 Comparison of mean velocity and fluctuation velocity rms value

Figure 14 shows the distributions of the mean velocity $U_m$ in the $x$-$y$ cross section at $z/D = 0$, and Fig. 15 identifies the vertical direction plots at each $x/D$ position. The upper image in Fig. 14 indicates that the case of the Poiseuille main flow, while the lower image is that of the Couette main flow. Here, the red arrow indicates the location of the heated jet inlet. The mean velocity distribution in wake of the jet is very different between both main flows. In the Poiseuille main flow, the impingement of the jet provides the momentum to the range of $x/D = 2$–4 near the lower wall, and generates a high heat transfer region, as shown in Fig. 8(a). In the Couette main flow, the flow field originally has a large momentum around the near lower wall, so the effect of the impinging jet to the lower region is limited. In addition, there is a strong back current region behind the jet. Fric and Roshko (1994) reported that there is a back-current region behind a jet crossing a main flow caused by a separate flow, similar to the one behind a cylinder. The results of this report have confirmed the same situation.

In Fig. 15, in the middle of the flow channel’s height, at $x/D = 0$, the jet affects each original velocity distribution, which is indicated by the broken line. At $x/D = 2$ of the Poiseuille main flow, a large momentum is supplied within the range of $y/L_y = 0$–0.3 by the jet. No effect of the jet is seen in the profiles at $x/D = 8$ in both main flows, so the interfering flows are already developed. Both velocity profiles are flat in the middle of the flow height, and the velocity gradients near the upper and lower walls are steep. Therefore, they are similar to the turbulent flow of each main flow.

![Contours of mean velocity $U_m$. In the Poiseuille main flow (upper), the momentum induced from the jet is observed. In the Couette main flow (lower), there is a strong back current behind the jet. Fric and Roshko (1994) reported, in their study on a jet crossing a main flow, that there is also a flow separation and roll up, which causes a back current behind the jet inlet.](image1)

![Velocity distributions in y-direction. At x/D = 0 and x/D = 2, the effects of the jet are observed clearly. However, at x/D = 8, no effect of the jet is observed and the profiles are similar to the turbulent flow of a Poiseuille or Couette flow.](image2)
Figure 16 shows the distributions of the fluctuation velocity’s rms value $u_{rms}$ in the $x$-$y$ cross section at $z/D = 0$, and Figure 17 shows their vertical direction plots at each $x/D$ position. The spreading of fluctuation velocity behind the jet in the Couette main flow is smaller than that in the Poiseuille main flow, and there is a high fluctuation velocity on the front side of the jet (the left side of the jet in the figure). It is considered that, while the jet is swept downward by the Poiseuille flow constantly, there is periodic velocity fluctuation owing to the vortex ring of the jet because of the Couette flow velocity near the jet inlet being almost quiescent. As shown in Fig. 17, the fluctuation velocity induced by the interference of the jet is shown around the middle of the flow channel height at $x/D = 0$ and near the lower wall region at $x/D = 2$. No effect of the jet is observed on the fluctuation velocity profile at $x/D = 8$ in both main flows; however, that of the Couette main flow is similar to the distribution of the turbulent Couette flow (Nakabayashi et al., 1988).

![Fig. 16 Contours of velocity fluctuation $u_{rms}$. The velocity fluctuation in the Poiseuille main flow (upper) spreads wider than that in the Couette main flow (lower). There is high velocity fluctuation on the front side of the jet. These phenomena are caused by the low velocity of the Couette main flow around the jet inlet.](image)

![Fig. 17 Distributions of velocity fluctuation in $y$-direction. At $x/D = 2$, the effects of the jet flow are observed near the lower surface. At $x/D = 8$, no effect of the jet is observed and the profile of the Couette main flow is similar to that of the turbulent Couette flow.](image)
4. Conclusion

This study evaluated the numerical simulation of an impinging jet interfering with the Poiseuille and Couette flows, and obtained following conclusions.

(1) Similar to previous research on interfering flow, a vortex pair is extracted in a time-averaged flow field.

(2) In case of the Poiseuille main flow, the impinging jet is dominant on the heat transfer and the hanging vortex generates exclaves of a high heat transfer region. In case of the Couette main flow, the heat transfer is predominantly caused by the vortex pair near the wall. In addition, this vortex pair involves and transports the low temperature fluid from the main flow to the near-wall region, thus decreasing the heat transfer on the \( x \)-axis.

(3) The distribution profiles of the mean and fluctuation velocities in wake of the jet interference are similar to those in each turbulent state.

References

Angioletti, M., Nico, E. and Ruocco, G., CFD turbulent modelling of jet impingement and its validation by particle image velocimetry and mass transfer measurements, International Journal of Thermal Sciences, Vol.44 (2005), pp.349–356.

Bouchez, J.-F. and Goldstein, R. J., Impingement cooling from a circular jet in a cross flow, International Journal of Heat and Mass Transfer, Vol.18, No.6 (1975), pp.719–730.

Fric, T. F. and Roshko, A., Vortical structure in the wake of a transverse jet, Journal of Fluid Mechanics, Vol.279 (1994), pp.1–47.

Goldstein, R. J. and Behbahani, A. I., Impingement of circular jet with and without cross flow, International Journal of Heat and Mass Transfer, Vol.25, No.9 (1982), pp.1377–1382.

Jinno, K., Tsujimoto, K., Shakouchi, T. and Ando, T., Direct numerical simulation of intermittent multiple impinging jets, Proceedings of the Mechanical Engineering Congress 2016 Japan (2016), S0520102 (in Japanese).

Liu, T. and Sullivan, J. P., Heat transfer and flow structures in an excited circular impinging jet, International Journal of Heat and Mass Transfer, Vol.39, No.17 (1996), pp.3695–3706.

Metzger, D. E. and Korstad, R. J., Effects of crossflow on impingement heat transfer, Journal of Engineering for Power, Vol.94, No.1 (1972), pp.35–42.

Nakabayashi, K., Kitoh, O., Iwata, H., Kato, E. and Uehara, T., Basic study on turbulent lubrication (1st report, turbulent plane Couette flow), Transactions of the Japan Society of Mechanical Engineers, Series B, Vol.54, No.499 (1988), pp.547–552 (in Japanese).

Onda, Y., Mizutani, M. and Mori, M., Machine tool main spindle bearings with air cooling spacer, NTN Technical Review, No.80 (2012), pp.38-41 (in Japanese).

Sakai, E., Takahashi, T. and Watanabe, H., Large eddy simulation of inclined round jet issuing into cross-flow, Transactions of the Japan Society of Mechanical Engineers, Series B, Vol.78, No.789 (2012), pp.1107–1117 (in Japanese).

Shadlesky, P. S., Stagnation point heat transfer for jet impingement to a plane surface, AIAA Journal, Vol.21, No.8 (1983), pp.1214–1215.

Shakouchi, T., Jet Flow Engineering – Fundamentals and Application –, Morikita Publishing Company (2004), pp.107 (in Japanese).

Sparrow, E. M. and Goldstein, R. J., Effect of nozzle – surface separation distance on impingement heat transfer for a jet in a crossflow, Transaction of the ASME Journal of Heat Transfer, Vol.97, No.4 (1975), pp.528–533.

Tillmark, N. and Alfredsson, P. H., Experiments on transition in plate Couette flow, Journal of Fluid Mechanics, Vol.235 (1992), pp.89–102.

Tsujimoto, K., Ishikura, T., Shakouchi, T. and Ando, T., Direct numerical simulation of active-controlled impinging jets, Journal of Fluid Science and Technology, Vol.4, No.2 (2009), pp.279–291.

Tyagi, M. and Acharya, S., Large eddy simulation of film cooling flow from an inclined cylindrical jet, Journal of Turbomachinery, Vol.125 (2003), pp.734–742.

Yuan, L. L., Street, R. L. and Ferziger, J. H., Large-eddy simulations of a round jet in crossflow, Journal of Fluid Mechanics, Vol.379 (1999), pp.74–104.