Flow Characteristics of Circular Liquid Jet Impinging on a Moving Surface Covered with a Water Film

Hitoshi FUJIMOTO,1) Yuhei SUZUKI,2) Takayuki HAMA1) and Hirohiko TAKUDA1)

1) Graduate School of Energy Science, Kyoto University, Kyoto, 606-8501 Japan.
2) Formerly Graduate Student, Kyoto University. Now at Sumitomo Metal Industries.
(Received on April 26, 2011; accepted on May 31, 2011)

The flow characteristics of a single circular water jet impinging on a moving surface covered with a water film have been investigated by means of experiments and three-dimensional computer simulations. The experiments were conducted by varying the jet velocity, nozzle-to-plate distance, and flow rate of the water film. It was found that the following three types of flow structures existed: an almost steady flow structure, an unsteady flow structure, and a transition flow structure between the steady and the unsteady flows. The critical boundary, at which the almost steady flow structure appears, is discussed using a simple potential flow theory. In the numerical simulation, the liquid flow was assumed to obey the Navier-Stokes equation in the three-dimensional Cartesian coordinate system. The effects of viscosity, gravity, and the presence of a free liquid surface were taken into account. The predictions were in reasonable agreement with the experimental results. Each flow structure has been studied in detail for a better understanding of the physics of the phenomena.

KEY WORDS: ROT cooling; impinging jet; free surface flow; numerical simulation; hydraulic jump.

1. Introduction

In hot rolling processes in steel-making industries, impinging liquid jets are widely employed to cool hot metal sheets from the final rolling temperature to a desired coiling temperature on a run-out table (ROT).1–4) Hot materials are cooled by the impingement of arrays of circular jets as illustrated in Fig. 1. A wide range of the rate of heat removal from the hot material to liquid jets can be obtained by adjusting the liquid flow rate and the number of impinging jets. Accurate temperature control and uniform cooling of the metal plates are required to produce the desired mechanical characteristics. The utilization of circular jets is one method that satisfies these requirements.

Figure 2 shows a schematic diagram of a single circular liquid jet impinging on a solid surface. The liquid jet issues from a circular nozzle into air, falls vertically downward, and then impinges on a horizontal solid surface. Radial film flow is formed from the impact point along the solid surface. Circular hydraulic jump occurs at a certain point where the liquid film increases steeply. There are many factors5–9) that affect the flow characteristics of a free surface impinging jet, including liquid flow rate, nozzle diameter, type of liquid, and roughness of solid surface.

In an actual run-out table cooling, a large amount of water impinges on a moving metal plate, resulting in the formation of a thick water film along the solid surface, as shown Fig. 1. As a result, most of the circular jets do not have a direct impact on the solid surface, but they have a direct impact on the preexisting liquid film. The interaction between the circular impinging jets and the preexisting film generates very complex flows, which are completely different from the flows for single jet impingement onto an immobile solid.
Despite many prior works on pipe cooling systems, which will be mentioned in a later subsection, a few studies took account of the presence of a liquid film and a moving plate. The present study is a fundamental study on ROT cooling and it addresses the above issue, although the moving plate is unheated.

The objective of this study is to investigate the flow characteristics of single circular water jet impingement on a moving surface covered with a water film, as illustrated in Fig. 3. Experiments and numerical simulations were conducted in order to achieve this objective. In the experiments, the nozzle diameter, jet velocity, nozzle-to-plate distance, flow rates of the water film, and thickness of the water film were varied systematically. It was found that three types of flow structures existed depending on the experimental conditions, namely, an almost steady flow structure, an unsteady flow structure, and a transition structure between the steady and the unsteady flows. The critical boundary, at which the almost steady flow structure appears, is discussed using a simple potential flow theory. In the numerical simulation, the system of Navier-Stokes equations for a three-dimensional unsteady fluid has been solved using finite difference method. The volume-of-fluid (VOF) method has been applied to track the time evolution of the free liquid surface. Detailed flow structures will be discussed for a better understanding of the physics of the phenomena.

2. Literature Survey

Previous studies on impinging jets are briefly discussed. Webb and Ma provided an excellent review of the studies of axisymmetric and planar impinging jets. They compiled many related studies and summarized the available analytical and experimental models for predicting heat transfer from a solid to the impinging jets. No data concerning a single circular impinging jet issued onto a moving solid were provided.

Several studies have addressed the impingement of liquid jets on a moving solid. Rahman et al. numerically characterized the convective heat transfer of a free liquid jet impinging on a rotating, uniformly heated solid disk with finite thickness and radius. Chen et al. studied the heat transfer between a circular impinging jet and a moving metal plate at temperatures above and below the boiling point of water by means of experiments and numerical simulations. Samaras and Simaan numerically studied the temperature control of a hot strip through a two-point boundary temperature control system approach. Xu and Gadala simulated the cooling process of steel strips on an ROT using a 1D model. Gradeck et al. experimentally as well as numerically investigated the flow field of an impinging water jet on a moving plate for various jet and plate velocities, nozzle diameters, and nozzle-to-plate spacings. They also empirically studied the quenching of a hot rotating cylinder at an initial temperature of 500–600°C using a subcooled planar water jet.

Studies on the interaction between the water jets and the residual fluid on the surface are relatively rare. Ishida et al. studied the basic characteristics of pipe nozzle cooling with water retention on a moving plate. Cho et al. numerically investigated the flow characteristics of a circular water jet array impinging onto a moving steel strip using a three-dimensional computational model. They considered the depth of the water accumulated in the water pool that forms on the surface of the moving steel strip. The VOF method with a high-resolution interface capturing scheme was implemented to handle the free surface flow. In these studies, multi-circular jets were used. However, the knowledge of detailed flows is still lacking because many other factors affect the physics of the phenomena.

3. Experimental Apparatus

A schematic diagram of the experimental apparatus is shown in Fig. 3. The setup is composed of a water reservoir, two pipelines for generating the circular jet and film flow, and a moving stainless steel belt attached to a pair of rolls. Purified water was used as the test liquid. Water temperature was maintained at 20 ± 0.1°C in a constant-temperature reservoir. Each pipeline consists of a mechanical pump, a flow meter, a regulating valve, a circular pipe or a slot nozzle unit. In the case of the pipeline with a circular pipe, for generating a circular jet, water flows to the top end of the circular pipe whose inner diameter is 5.0 or 7.0 mm. The pipe length (= 560 mm) is sufficiently long to achieve fully developed turbulent pipe flows at the exit. A circular jet issues vertically downward from the pipe exit at a mean velocity of \( V_0 \) and impacts the horizontal, moving solid surface. The mean velocity \( V_0 \) was varied from 0.3 to 2.5 m/s.

It should be noted that the surface tension acts on the free liquid surface of the falling jet, producing instability waves. The instability waves develop downwards and distort the shape of the jet. Eventually, the cylinder-shaped jet split into many small droplets. Thus, a very large nozzle-to-plate
spacings is unsuitable for obtaining a stable circular jet. In the present study, the nozzle-to-solid surface spacing \( H \) was set as \( H = 20–100 \) mm to avoid the appearance of any unstable wave.

A stainless steel belt with a thickness of 0.15 mm and a width of 50 mm was used as the moving solid surface. The stainless steel belt was tightly mounted on a pair of stainless steel rolls with a diameter of 50 mm. The distance between the two rolls was 500 mm. An electric motor drives one of the rolls at a preset rotation speed. The moving speed of the belt \( V_t \) can be varied from 0 m/s to 1.3 m/s. The upper limit of the belt speed is attributed to the specifications of the motor. Moreover, the belt slides on a rigid support plate so that the local bend of the belt due to the impingement of the circular jet is prevented (see Fig. 3(b)).

Figure 4 shows a schematic of the slot nozzle unit used to generate the film flow. The slot nozzle unit was composed of two sections, that is, the convergent section and the uniform section. In the convergent section, the cross-sectional area monotonically decreases away from the inlet of water. A sponge is inserted in this section to minimize flow disturbances. In the uniform section, the gap \( d_f \) between a pair of parallel plates is set to 2.0 or 4.0 mm. The flow length in the uniform section is 76 mm for \( d_f = 2.0 \) and 100 mm for \( d_f = 4.0 \) mm. The liquid film flow was created by issuing a slot jet parallel to the moving belt at a mean velocity \( V_t \), as shown in Fig. 3. The mean velocity of \( V_t \) can be varied from 0.2 to 1.0 m/s.

The distance between a jet impact point and the slot nozzle exit is set to 30 mm for \( d_f = 2.0 \) and 60 mm for \( d_f = 4.0 \) mm. If a slot nozzle-to-jet impact point is too large, it is experimentally difficult to ensure a uniform film flow over the moving belt. Some dry areas appear on the moving solid near the side edges owing to the effects of surface tension and wettability of water on the stainless steel belt. When the slot nozzle is set very close to the jet impact point, the flow on the slot nozzle wall is influenced. An appropriate nozzle-to-impact point distance, which depends on the flow conditions, was determined by a trial-and-error method. In addition, discharged water from the test section was returned to the reservoir through the return path.

Volume flow rates of water issued from the circular and slot nozzles were measured directly from the volumes of water discharged during a certain period (5–60 s). The measurement accuracy of volume flow rates was within \( \pm 0.025 \) m\(^3\)/s. The mean velocities of the jet issued from the circular and slot nozzles are determined from the discharged volume flow rates of water and the cross sections of the nozzles. Each experiment was conducted under the condition that the jet velocity, film flow velocity, and velocity of the moving belt were maintained at preset values. It should be noted that the mean velocity issued from the slot nozzle is equal to the moving velocity of the belt in the present study, although these velocities can be varied independently.

To gain an understanding of the unsteady flow characteristics of the impinging water jet on a moving solid surface, high-speed videography was adopted. A metal halide lamp with high brilliance was used as the light source. In most experiments, video images were captured at a frame rate of 1/600 s with a resolution of 432 \( \times \) 192 pixels. When the images near the jet impact point were captured at a higher time resolution, the frame rate was set to 1/1 200 s with a spatial resolution of 336 \( \times \) 96 pixels.

4. Numerical Simulation

4.1. Conservation Equations and Numerical Procedure

The flows are assumed to obey the Navier-Stokes equations for an incompressible fluid and the continuity equation in the three-dimensional Cartesian coordinate system. In the present model, the effects of gravity, viscosity, and the presence of a free liquid surface have been considered. The effects of surface tension and turbulence were neglected, although they would make our model more realistic. The conservation equations with constant thermo-fluid properties are as follows:

\[
\frac{\partial u_i}{\partial x_i} = 0 \quad \text{.................................. (1)}
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = F_i - \frac{1}{\rho} \frac{\partial p}{\partial x_j} + \nu \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad \text{.......................... (2)}
\]

where \( t, x, u_i \) are the time, coordinate, and velocity, respectively. \( p, \rho, \) and \( \nu \) denote the pressure, apparent fluid density, and apparent molecular kinematic viscosity, respectively. \( F_i \) denotes the volume force associated with gravitational acceleration.

The VOF method\(^{(10)}\) is applied to track the time evolution of the free liquid surface. A color function \( \phi \) is introduced to represent the volume fraction of liquid in a computational cell. The equation for the color function is given by

\[
\frac{\partial \phi}{\partial t} + u_i \frac{\partial \phi}{\partial x_i} = 0 \quad \text{.................................. (3)}
\]

The apparent density is given using the color function as follows:

\[
\rho = \rho_{\text{water}} \phi + \rho_{\text{air}} (1-\phi) \quad \text{.......................... (4)}
\]

The conservation equations are dispersed and solved by a finite difference method. A fractional step method is adopted to solve the conservation equations in accordance with the CIP combined and unified procedure developed by Yabe et al. (2001).\(^{(21)}\) Figure 5 shows a schematic of the computational domain and boundary conditions. The origin of the coordinate system is set at the jet impinging point on
the solid. The size of the domain is \(-0.03\, \text{m} \leq x \leq 0.05\, \text{m}, -0.035\, \text{m} \leq y \leq 0.035\, \text{m},\) and \(0\, \text{m} \leq z \leq H\). A non-uniform staggered mesh system focusing on the \(z\) direction was used. The mesh size in the \(z\) direction is minimal adjacent to the moving surface and increases in the upward direction. The number of cells is 160 \(\times\) 150 \(\times\) 60 in the \(x\), \(y\), and \(z\) directions.

The moving solid surface is set in the area \((-0.025\, \text{m} \leq y \leq 0.025\, \text{m}, z = 0\, \text{m})\), and a pair of outflow regions are introduced outside the moving solid \((0.025\, \text{m} \leq \left| y \right| \leq 0.035\, \text{m}, z = 0\, \text{m})\) (see Fig. 5). A constant velocity condition was maintained at the moving solid surface. In the outflow regions, the outflow condition was imposed. This condition allows the fluid to flow out through the boundary, but prohibits the inflow of fluid. At the boundaries corresponding to the circular and slot nozzles, the inflow condition was used. The inflow velocities of water were specified using the experimental mean velocities. The free-slip wall condition was employed at the upstream boundary in the \(x\) direction and the upper boundary in the \(z\) direction in order to prohibit unrealistic inflow of air. At the downstream boundary in the \(x\) direction, the zero gradient condition was used. At the side boundaries in the \(y\) direction, the outflow condition was employed. It should be noted that all the boundaries, except for nozzle exits and the plate surface, are virtual boundaries where the exact flow conditions cannot be determined. The boundary conditions would lead to some numerical errors because of the small computer domain. Although computer simulations should be conducted under a sufficiently large domain with a fine mesh system, they are impossible because of the limited computer resources.

5. Results and Discussion

5.1. Experimental Results

5.1.1. Basic Flow Structures of Free Surface Impinging Jets

First, the fundamental flow structures of free surface impinging jets are studied. Three types of experiments were conducted, namely, free surface jets impinging on a static solid, a dry moving solid, and a moving solid with a preexisting liquid film. Figure 6 shows the three experimental results under the conditions \(V_0 = 0.43\, \text{m/s}, H = 40.0\, \text{mm}\) and \(d = 7.0\, \text{mm}\). Figure 6(a) shows the radial film flow formed by the impingement of a circular water jet on a static solid. The liquid film is thin in the vicinity of the point of impact. The hydraulic jump occurs at \(r/d \approx 2.4\) where the liquid film thickness increases abruptly (also see Fig. 2). For convenience, the thin liquid layer inside the hydraulic jump is called the “thin film zone.” It should be noted that the radial distance of the hydraulic jump \(R_j\) increases with an increase in the volume flow rate of water, although the results are not shown. This is consistent with the results of prior works.23–26)

In Fig. 6(b), a circular water jet impinges on a moving surface without a preexisting film. Figure 6(b) shows the results of Gradeck et al.17) In Fig. 6(c), a circular water jet impinges on a moving surface with a preexisting water film at \(V_f = V_s = 0.2\, \text{m/s}\). Unlike the previous case, the shape of the free surface is no longer axisymmetric. On the upstream side, a bow-shaped hydraulic jump occurs, followed by an upstream thick liquid film zone. The moving solid surface is dry in the upstream of the thick film zone. The liquid film is thin on the downstream side. The flow structures show trends similar to the results of Gradeck et al.17)

In Fig. 6(c), a circular water jet impinges on a moving surface with a preexisting water film at \(V_f = V_s = 0.2\, \text{m/s}\) and \(d_f = 2.0\, \text{mm}\). A bow-shaped liquid swelling is seen on the upstream side, which is similar to the case with no liquid film in Fig. 6(b). The thin film zone on the upstream side is apparently smaller than that in the case shown in Fig. 6(b) owing to the presence of a liquid film flow issued from the slot nozzle. The liquid film is thin on the downstream side. The flow field is relatively steady over the entire region. As is obvious, both the moving solid surface and the preexisting liquid film flow is thin in the vicinity of the point of impact. The hydraulic jump occurs at \(r/d \approx 2.4\) where the liquid film thickness increases abruptly (also see Fig. 2). For convenience, the thin liquid layer inside the hydraulic jump is called the “thin film zone.” It should be noted that the radial distance of the hydraulic jump \(R_j\) increases with an increase in the volume flow rate of water, although the results are not shown. This is consistent with the results of prior works.23–26)
liquid film affect the flow profile in the vicinity of the jet impact point.

5.1.2. Effect of Varying the Velocity of Film Flow on the Flow

Figure 7 shows the experimental results for $V_0 = 0.43$ m/s, $H = 80.0$ mm, $d = 7.0$ mm, and $d_1 = 2.0$ mm: (a) $V_f = V_s = 0.2$ m/s, (b) $V_f = V_s = 0.3$ m/s, (c) $V_f = V_s = 0.4$ m/s, (d) $V_f = V_s = 0.5$ m/s, (e) $V_f = V_s = 0.7$, and (f) $V_f = V_s = 0.9$ m/s. As expected, the flow structure is strongly dependent on the velocity of film flow. When $V_f = 0.2$ m/s, a bow-shaped hydraulic jump occurs on the upstream side. A thin film zone is present not only on the downstream side but also on the upstream side. When $V_f = 0.3$ m/s, the upstream hydraulic jump is observed to be very close to the jet impact point. The upstream thin film zone is apparently smaller than that in the case of (a). Since the position of the hydraulic jump slightly shifts in the direction of the moving solid with time, the circular jet is often in direct contact with the hydraulic jump. This results in a wavy motion of the flow on the upstream side. In contrast, the liquid film is thin on the downstream side.

In the cases of (c) 0.4, (d) 0.5, (e) 0.7, and (f) 0.9 m/s, the upstream thin film zone completely disappears. In (c), the free liquid surface is wavy even in the thin film region. In (d)–(f), the hydraulic jump is formed in the shape of a wedge. The wedge angle of the hydraulic jump is smaller for a larger film flow velocity $V_f$. The downstream thin film zone becomes narrower, as well. The flow fluctuates a lot and is absolutely unsteady. It was revealed from naked-eye observations that some air is sometimes entrapped near the impact point owing to the interaction of the circular jet with the hydraulic jump.

It is found from Fig. 7 that the flows can be roughly categorized into three types depending on the shape of thin film zone near the jet impact point. In the first mode, as in the case of the results shown in Fig. 7(a), a stable thin film zone is formed not only on the downstream side but also on the upstream side. We call this flow “Mode 1.” In Fig. 7(b), the hydraulic jump sometimes contacts the vertical jet. Therefore, the free surface in the upstream hydraulic jump region becomes wavy. This flow is categorized as “Mode 2.” For larger film velocities, the upstream thin film zone completely vanishes, as shown by the results in Figs. 7(c)–7(f). The flow is absolutely unsteady in the entire region. Such a flow is called “Mode 3.” It should be noted that “Mode 2” is a transient category between Modes 1 and 3. The determination of “Mode 2” flow is somewhat subjective.

5.1.3. Effect of Varying Nozzle-to-plate Distance

The impact velocity and cross section of a circular jet are dependent on the nozzle-to-plate distance associated with the gravity effect. According to Bernoulli’s equation and the continuity equation, we have the following relations:

\begin{align}
V_j &= \sqrt{V_0^2 + 2gH} = V_0 \sqrt{1 + \frac{2}{Fr_H^2}}, \\
Fr_H &= \frac{V_0}{(Hg)^{1/2}} \\
r_j &= r_0 \sqrt{\frac{V_0}{V_j}}, \\
r_0 = d / 2
\end{align}

where $V_j$, $r_j$, $g$, and $Fr_H$ are the jet impinging velocity, estimated jet radius at the impact point, gravitational acceleration, and the Froude number, respectively. For a larger nozzle-to-plate distance $H$, the jet velocity $V_j$ is larger and the estimated jet radius $r_j$ is smaller. Moreover, the impact inertia of the jet becomes larger for a larger $H$ owing to an increase in the gravitational potential. It is noted that the flow conditions shown in Figs. 6(c) and 7(a) are identical except for the nozzle-to-plate distance. By comparing the two results, it is seen that the jet radius near the impact point for $H = 40$ mm in Fig. 6(c) is larger than that for $H = 80$ mm in Fig. 7(a). This is consistent with Eq. (6). In addition, we conducted similar observations for the Mode 3 flows. However, appreciable effects on the flows cannot be observed because the flows fluctuate greatly.
5.2. Critical Conditions for the Presence of Upstream Thin Film Zone

As mentioned in the previous subsection, three types of flows are observed. In Mode 1, an upstream thin film zone is present. In Mode 3, the upstream thin film zone completely disappears. The flow is essentially unsteady. Mode 2 is a transient category between Modes 1 and 3. **Figure 8** shows the flow patterns for various experimental conditions. The dashed lines represent the predicted critical boundary between Modes 1 and 3, as will be explained later. It is obvious that the Mode 3 flows occur at a large film velocity $V_f$. The critical boundary depends on the parameters $V_0$, $V_f$, and $H$.

A simple theoretical model for predicting the critical condition, at which the upstream thin film zone is present, is built on the basis of the steady, irrotational flow theory for an inviscid fluid in a two-dimensional coordinate system. We assumed that the thickness of film flow issued from the slot nozzle is comparable to the thickness of the radial flow formed by the impingement of the circular jet. The following mean velocities integrated across the liquid film are introduced:

$$
\overline{u}(x, y, z) = \frac{1}{\delta} \int_0^\delta u(x, y, z)dz
$$

$$
\overline{v}(x, y, z) = \frac{1}{\delta} \int_0^\delta v(x, y, z)dz
$$

$$
\overline{w}(x, y, z) = \frac{1}{\delta} \int_0^\delta w(x, y, z)dz
$$

where $\delta$ is the local film thickness. Since the liquid flows almost parallel to the solid surface, the mean velocity component in the $z$ direction can be regarded as $\overline{w} \sim 0$, except in the region of the circular jet impact. Thus, the flow, explained by the mean velocities, becomes two dimensional.

![Fig. 8. Comparison of observed flow patterns with the proposed correlation Eq. (14) for various experimental conditions.](image-url)
We introduce a model composed of two flow components, that is, a radial flow formed by jet impingement and a uniform flow issued from a slot nozzle, as illustrated in Fig. 9. According to the potential flow theory,\(^\text{27)}\) such a flow can be described by the following complex potential function:

\[
F(x + iy) = V_f (x + iy) + \frac{q}{2\pi} \ln(x + iy) \quad \text{......... (8)}
\]

The first term on the right-hand side represents a uniform flow parallel to the \(x\)-axis at a velocity \(V_f\). The second term is a point source term showing that the fluid flows outward from the origin with a strength \(q\). The mean velocity components can be determined as\(^\text{27)}\)

\[
(\overline{u}, \overline{v}) = \left( V_f + \frac{q}{2\pi} \frac{x}{(x^2 + y^2)^2}, \frac{q}{2\pi} \frac{y}{(x^2 + y^2)^2} \right) \quad \text{......... (9)}
\]

The velocity profile determined by Eq. (9) has a stagnation point \((\overline{u} = \overline{v} = 0)\) at the location \((x, y) = (-q / (2\pi V_f), 0)\), as illustrated in Fig. 9.

It is assumed that the upstream hydraulic jump at \(y = 0\) occurs near the stagnation point where the momentum of the uniform flow is balanced by that of the radial flow. Further, photographic observation reveals that the upstream thin film zone is stably formed if the distance between the hydraulic jump and the jet impact point is roughly four times larger than the estimated jet radius \(r_j\), as defined in Eq. (6). Therefore, we have the following correlation for presenting the upstream thin film zone:

\[
\frac{q}{2\pi V_f} \geq 4 r_j \quad \text{.......................... (11)}
\]

The strength of the source, \(q\), can be related to the radial jet velocity \(V_f\) and the estimated jet radius \(r_j\) as

\[
q = 2\pi r_j V_f \quad \text{.......................... (12)}
\]

Substituting Eq. (12) into Eq. (11), we have

\[
\left( \frac{V_s}{V_f} \right) \left[ 1 + \frac{2}{Fr_{ji}^2} \right] \geq 4. \quad \text{............... (13)}
\]

The critical boundary, at which the upstream thin film zone is present, is now given by

\[
\left( \frac{V_f}{V_s} \right) \left[ 1 + \frac{2}{Fr_{ji}^2} \right] = 4 \quad \text{............... (14)}
\]

The results obtained from Eq. (14) are plotted in Fig. 8 for validating the present model. The predicted critical boundaries agree moderately well with the experiments in all cases, although the model assumptions are too ideal. The present model takes no account of the thickness \(d_i\) of the preexisting liquid film, which must be one of factors that affect the critical boundary. Its effect is included implicitly in the coefficient of “4” in Eq. (11) as a fitting parameter. Thus, for establishing a more useful correlation capable of predicting the critical boundary, Eq. (11) should be modified. This is a challenge for future works.

5.2. Numerical Results

Figure 10 shows the comparison of the numerical results with the experimental results for \(V_0 = 0.43 \text{ m/s, } V_f = V_s = 0.2 \text{ m/s, } H = 4.0 \text{ mm, } d = 7.0 \text{ mm, and } d_i = 2.0 \text{ mm}\). The flow is categorized as Mode 1. It should be noted that the boldface symbols \((x, y, z)\) in the figure represent dimensionless coordinates normalized by the circular nozzle diameter \(d\). In both the numerical and the experimental results, the presence of an upstream thin film zone is detected. The numerical result agrees reasonably well with the experimental result. Figure 11 shows the velocity profile and the pressure contour in the liquid in the symmetric plane \((y = 0)\). Pressure is high at the impact point and decreases on moving away from it. On the downstream side (a), the film thickness decreases very slightly on moving away from the impact point. Since the flow direction of the thin film is the same as that of the moving surface, the velocity gradient across the liquid film is small. On the upstream side (b), the hydraulic jump occurs at \(x/d \sim -1.1\). In the region between the vertical jet and the hydraulic jump, the liquid near the free surface moves in the upstream direction, whereas the liquid close to the solid surface flows in the downstream direction owing to the motion of the solid. Thus, a large velocity gradient occurs in the thin liquid film. In the hydraulic jump region, a vortex is stably formed.

Figure 12 shows the comparison of the numerical result...
with the experimental result for $V_0 = 0.8$ m/s, $V_f = V_s = 0.8$ m/s, $H = 40.0$ mm, $d = 7.0$ mm, and $d_f = 2.0$ mm. The flow is Mode 3. In both the numerical and the experimental results, the upstream thin film zone is absent. The numerical result agrees qualitatively with the experimental result. In addition, the hydraulic jump structure is unstable.

**Figure 13** shows the time evolution of the predicted free liquid surface for $V_0 = 0.43$ m/s, $V_f = V_s = 0.9$ m/s, $H = 20.0$ mm, $d = 7.0$ mm, and $d_f = 2.0$ mm. Conditions as those in Fig. 13. The resolution of the camera was set as $336 \times 96$ pixels, and its frame rate was 1200 fps. For convenience, the time is set to $t = 0$ s in (a). The free surface is almost flat in the downstream thin film zone near the impact point in (a). The film flow interacts with the cir-
impinging on a moving surface covered with a preexisting water film were investigated through experiments and three-dimensional computer simulations. The main results of this study are summarized below.

(1) Three types of flow structures were found to exist depending on the experimental conditions.

(2) A simple theoretical model for predicting the critical condition, at which the upstream thin film zone is present, was built on the basis of the steady, irrotational flow theory for a two-dimensional inviscid fluid.

(3) Numerical simulations were conducted to study the flow structures of Modes 1 and 3. In Mode 1, the flow in the vicinity of the jet impact point is relatively stable. In Mode 3, the flow in the vicinity of the jet impact point is complex and unsteady. Non-periodical solitary waves appeared at the downstream side. The numerical results agree qualitatively well with the experimental results.

REFERENCES

1) S.-J. Chen and A. A. Tseng: Int. J. Heat Fluid Flow, 4 (1991), 343.
2) N. Hatta and M. A. Saiman: ISA Trans., 36 (1997), 11.
3) F. Xu and M. S. Gadala: Steel Res. Int., 78 (2007), 71.
4) M. Gradeck, A. Kouachi, A. Dani, D. Arnoult and J. L. Borean: Exp. Therm. Fluid Sci., 30 (2006), 193.
5) R. Ishida, A. Mizuta, K. Korida, S. Yasunaga and K. Takisawa: ISIJ Int., 52 (2009), 1094.
6) M. J. Cho, B. G. Thomas and P. J. Lee: Metall. Mater. Trans. B Proc. Metal. Mater. Proc., 39 (2008), 593.
7) T. Yabe, F. Xiao and T. Utsumi: J. Computational Phys., 169 (2001), 556.
8) T. Bohr, P. Dimon and V. Putkaradze: J. Fluid Mech., 254 (1993), 635.
9) A. Rao and J. H. Arakeri: Exp. Fluids, 31 (2001), 542.
10) S. Togashi, T. Ohta and H. Asuma: Trans. Jpn. Soc. Mech. Eng. Part B, 69 (2003), 61 (in Japanese).
11) B. Elison and B. W. Webb: Int. J. Heat Mass Trans., 36 (1993), 3751.
12) N. T. Obot, A. S. Majumdar and W. J. M. Douglas: ASME paper, 79-WA/HT-53 (1979).
13) S.-J. Chen, J. Kothari and A. A. Tseng: Exp. Therm. Fluid Sci., 4 (1991), 343.
14) J. Stevens and B. W. Webb: Int. J. Heat Mass Trans., 37 (1994), 1207.
15) J. Stevens and B. W. Webb: Int. J. Heat Mass Trans., 36 (1993), 1227.
16) K. Kokado: Steel Res. Int., 59 (1988), 339.
17) M. Gradeck, A. Kouachi, M. Lebouche, F. Volle, D. Maillet and J. L. Borean: Int. J. Heat Mass Trans., 52 (2009), 1094.
18) N. S. Samaras and M. A. Saiman: ISIJ Trans., 36 (1997), 11.
19) I. Elison and B. W. Webb: Exp. Therm. Fluid Sci., 254 (1993), 635.
20) N. T. Obot, A. S. Majumdar and W. J. M. Douglas: ASME paper, 79-WA/HT-53 (1979).
21) T. Yabe, F. Xiao and T. Utsumi: J. Computational Phys., 169 (2001), 556.
22) T. Bohr, P. Dimon and V. Putkaradze: J. Fluid Mech., 254 (1993), 635.
23) A. Rao and J. H. Arakeri: Exp. Fluids, 31 (2001), 542.
24) S. Togashi, T. Ohta and H. Asuma: Trans. Jpn. Soc. Mech. Eng. Part B, 69 (2003), 61 (in Japanese).
25) S. Yamagami and S. Nishio: Trans. Jpn. Soc. Mech. Eng. Part B, 72 (2006), 1682 (in Japanese).
26) H. Lamb: Hydrodynamics, 6th ed., Cambridge University Press, London, (1932).