Numerical Study of a Planar Liquid Jet Impinging on a Solid Substrate

Hitoshi FUJIMOTO,1) Daisuke OHNO,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 Kawasaki Heavy Industries.

(Received on January 9, 2008; accepted on March 5, 2008)

The flow structure of a planar water jet impinging on a solid substrate was studied by means of three-dimensional numerical simulations. In the system studied, the planar water jet issues from a slot nozzle into quiescent air, falls vertically, impinges on a horizontal smooth surface, and then a thin film forms on the solid surface. The liquid flow was assumed to obey the Navier-Stokes equations in three-dimensional Cartesian coordinates. The simulations took into account the effects of gravity, viscosity, and surface tension at the free surface. Experiments were also conducted for model validation. The predictions of the model were in reasonable agreement with the experimental results. The effects of the velocity profile at the nozzle exit, the liquid flow rate, and the nozzle-plate distance on the flow structures were investigated. The physics of these phenomena are discussed in detail from the viewpoint of fluid mechanics.

KEY WORDS: planar jet; impinging jet; three-dimensional numerical analysis; free surface flow.

1. Introduction

Planar liquid jets issued from a slot nozzle into quiescent air arise in various industrial applications, including the cooling of hot solid materials by impinging liquid jets1–10) and curtain coating.11–14) Figure 1 shows a schematic diagram of a planar liquid jet impinging vertically on a horizontal substrate. The rim of the vertical jet is round and thickened due to surface tension. The width of the planar jet in the horizontal direction decreases with the vertical distance from the nozzle exit. After the jet impinges on the horizontal smooth substrate, a thin liquid film forms on the surface. The flow structure is three-dimensional.

Various prior studies of the cooling or quenching of hot substrates as a result of the impingement of planar liquid jets have been carried out. There have been numerous experimental studies. Vader et al.4) measured the rate of convective heat transfer between a flat hot surface and a planar water jet. They examined the effects of varying the jet velocity on the local surface temperature. Zumbrunnen5) proposed a boundary layer model for predicting the heat transfer from a moving plate to a planar impinging jet. Wolf et al.6) investigated the correlation between the mean jet velocity and the heat transfer coefficient. Robidou et al.7) investigated the effect of varying the nozzle-plate distance on the heat transfer rate. In these experimental studies, the heat transfer coefficient was determined from the experimental data by using two-dimensional equations of heat conduction.

The cooling process has been studied with numerical simulations and analytical models. Tong8) examined the heat transfer of an impinging oblique planar jet with numerical simulations. Timm et al.9) constructed an analytical model of heat transfer during subcooled jet impingement. Chen10) proposed a model for predicting the local heat transfer coefficients of heated surfaces that is based on the laminar boundary layer theory, and compared its predictions with experimental data. In these theoretical studies, the planar impinging jets were treated as two-dimensional flows. However, a detailed understanding of the three-dimensional flow structure is required to evaluate the local heat transfer rate in such cooling processes. Two-dimensional analytical studies of planar jets have also been conducted for other industrial applications,13,15,16) but three-dimensional analyses are rare.

The objective of this study was to examine the flow structure of a planar liquid jet impinging on a solid substrate at room temperature by means of three-dimensional computer simulations and experiments. In the experiments,
each planar impinging jet was observed with photography and the local thickness of the liquid film was measured with the stylus method. In the numerical simulations, we assumed that the liquid flow obeys the Navier–Stokes equations for an unsteady, incompressible, and viscous fluid in three-dimensional Cartesian coordinates. The effects of gravity, viscosity, and surface tension at the free liquid surface were taken into account. The conservation equations were approximated and solved with a finite difference method. The free liquid surface was tracked with the VOF (Volume-of-Fluid) method. The numerical results were then compared to the experimental results. The effects on the flow structures of the velocity profile at the nozzle exit, the nozzle-plate distance, and the flow rate were investigated and are discussed in this paper in terms of fluid mechanics.

2. Experiments

A schematic diagram of the experimental apparatus used in this study is shown in Fig. 2. Tap water was used as the test liquid. The water flows through a regulating valve, a flow meter, and then enters a reservoir. The reservoir is composed of transparent acrylic plastics and its internal dimensions are $40 \times 40 \times 250$ mm$^3$. Water flows vertically downwards through a 40 mm thick particle bed flow distributor, which is filled with 5 mm diameter glass beads, and then through a 40 mm long honeycomb unit, which is present to minimize flow disturbances. A slot nozzle consisting of two parallel plates is located underneath the reservoir. The cross-section of the flow area between the two plates is $1.62$ mm ($d_0$) $\times 30$ mm ($l_0$). The length of the slot nozzle along the flow direction (vertical direction) is 75 mm. The water issues vertically downwards from the nozzle exit in the form of a thin sheet. The liquid sheet impinges on a horizontal smooth surface made of optical glass. The dimensions of the solid surface on which the water jet impinges are 20 mm $\times 28$ mm. The nozzle-plate distance, $h$, was set at 12 mm.

The water flow rate, $Q$, is maintained at a preset value with a regulating valve located upstream of the reservoir, and is determined from the volume of water discharged during the sampling time ($=60$ s). The mean velocity at the nozzle exit, $W_0$, is evaluated from the volume flow rate, $Q$, and the cross-sectional area, $A_0 (=d_0 \times l_0)$ of the slot nozzle:

$$W_0 = \frac{Q}{A_0} \quad (1)$$

The flow rate used in this study was 4.0 L/min. The corresponding mean velocity at the nozzle exit is 1.4 m/s. The Reynolds number was determined from the mean velocity and the gap of the slot nozzle, $d_0$, and found to be 2220.

The Cartesian coordinate system ($x, y, z$) is aligned as shown in Fig. 1. The origin is positioned at the center of the substrate. The impingement line of the planar jet corresponds to the line ($y, z$)=$0, 0$). The velocity components in the $x, y,$ and $z$ directions are denoted by the symbols, $u, v,$ and $w,$ respectively.

The flow structure of each planar impinging jet was investigated with photography. A digital camera with an effective resolution of 3026 $\times$ 2018 pixels was used. The shapes of the planar jets were obtained directly from these images.

The stylus method was adopted to measure the local thicknesses of the planar jet and the film flow on the solid substrate, as shown in Fig. 3. In the measurements of the local thickness of the vertical liquid sheet, a pair of conical needle edges (point micrometer) is mounted on a transverse table, which can only move in the $y$ direction with a spatial resolution of 0.01 mm (see Fig. 3(a)). The position of one of the conical edges is adjusted to touch one side of the free surface. The other conical edge is then moved to contact the other side of the free surface. The local film thickness can then be determined from the distance between the two edges.

In the measurements of the thickness of the liquid film on the solid, one conical edge is positioned with a spatial resolution of 0.01 mm on a transverse table that can only move in the $z$ direction (see Fig. 3(b)). First, the conical edge is adjusted to touch the free liquid surface, and then moved downward to contact the solid surface. The film
thickness is determined from the distance from the surface to the solid.

3. Numerical Simulations

3.1. Conservation Equations

Two kinds of fluids are present in the simulated system, namely water and air. Both fluids were treated as incompressible and viscous. Their flows were assumed to obey the Navier–Stokes equations and the equation of continuity in the three-dimensional Cartesian coordinate system. The effects of gravity, viscosity, and surface tension at the free surface were taken into account. The effects of turbulence were neglected because the Reynolds numbers are relatively low. The conservation equations for constant thermo-fluid properties are then as follows:

\[ \rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = \rho F_x - \frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \]  \( (2) \)

\[ \rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = \rho F_y - \frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \]  \( (3) \)

\[ \rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = \rho F_z - \frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \]  \( (4) \)

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \]  \( (5) \)

where \( t \) is the time, and \( p, \rho, \) and \( \mu \) are the pressure, the apparent fluid density, and the apparent viscosity, respectively. \( F_x, F_y, \) and \( F_z \) are the components of the volume force in the \( x, y, \) and \( z \) directions, respectively. The volume force includes not only the gravitational force, but also the surface tension force, which was converted from a surface force to a volume force by using the CSF (continuum surface force) model.\(^{19}\)

In the present study, the VOF (Volume-of-Fluid) method\(^{19}\) was applied to track the time evolution of the free liquid surface. A color function, \( \phi \), was introduced to represent the volume fraction of liquid in each computational cell. The equation for the color function is given by

\[ \frac{\partial \phi}{\partial t} + u \frac{\partial \phi}{\partial x} + v \frac{\partial \phi}{\partial y} + w \frac{\partial \phi}{\partial z} = 0 \]  \( (6) \)

3.2. Numerical Procedure

The conservation equations were dispersed and solved with a finite difference method. As the numerical procedure was the same as that used in the authors’ previous study,\(^{19}\) only a brief description of this procedure is given here. The computational domain is divided into numerous small cells.

A staggered mesh is used, \( i.e., \) pressure, \( p \), and density, \( \rho \), are defined at the center of each cell and the components of the velocity, \( u, v, \) and \( w \), are defined at the edge of each cell. A fractional step method is then used to solve the conservation equations in accordance with the CCUP (CIP-combined and unified procedure) developed by Yabes et al.\(^{20}\) In this study, the R-CIP (rational-cubic interpolation program) scheme\(^{20}\) was applied to the calculation of the advection terms in the Navier–Stokes equations and of the advection equation for the color function. The diffusion terms were approximated with a 2nd-order central difference method.

Figure 4 shows the computational domain and the grid system. Only one quarter of the computational domain was used, as the flow structure is assumed to be symmetric with respect to both the \( x \) and \( y \) axes. A non-uniform staggered mesh system focusing on the vertical jet region and the region near the solid substrate was used. The mesh sizes in the \( x \) and \( y \) directions are minimal in the vertical planar jet region. The mesh size in the \( y \) direction is relatively coarse in the region downstream of the film flow that occurs on the solid substrate. In addition, the mesh size in the \( z \) direction is minimal adjacent to the solid substrate and increases in the upward direction because a thin velocity boundary layer is present on the solid surface. In the present study, the number of cells is \( 92 \times 137 \times 152 \) in the \( x, y, \) and \( z \) directions.

Initially, the computational domain is filled with quiescent air. The calculation commences when a planar jet issues from the slot nozzle. The steady-state solution is obtained with a time-dependent method.

The boundary conditions are as follows. At \( y=0 \) and \( x=0 \), the symmetric condition is imposed. The zero-velocity condition \( (u=v=w=0) \) is employed at the solid surface \( (z=0, 0 \leq x \leq 6.17) \). The zero-gradient condition was adopted on the bottom plane of the computational domain, except at the solid plate. The upper boundary coincides with the nozzle exit and nozzle wall. The velocity profile is specified at the outlet of the slot nozzle. The zero-velocity condition \( (u=v=w=0) \) is employed at the wall boundary associated with the wall of the slot nozzle. At the other boundaries, the zero gradient condition is used.

In the present study, the velocity profile at the nozzle exit was determined by evaluating the steady-flow distribution inside the slot nozzle. In the calculations of the flow profile in the slot nozzle, the no-slip condition was imposed at the wall boundaries and the velocity at the inlet boundary was
assumed to be uniform. As the velocity profile in the slot nozzle is dependent on the water flow rate, the velocity profile had to be calculated for each flow condition. For convenience, the velocity profile evaluated from the calculations for the slot nozzle is called the calculated flow profile. A uniform velocity distribution \( (u, v, w) = (0, 0, W_0) \) was used to specify the inlet conditions in order to investigate the effect of varying the velocity profile at the nozzle exit on the flow structures.

4. Results and Discussion

4.1. Model Validation

The usefulness of the numerical model was confirmed by comparing its predictions with the experimental results for \( W_0 = 1.4 \text{ m/s} \) and \( h = 12 \text{ mm} \). Figure 5 shows the observed planar impinging jet and the predicted free liquid surface. The rim of the planar jet is round and thick due to surface tension. The liquid film on the solid surface is slightly wavy. The numerical and experimental results are in qualitative agreement.

Figure 6(a) shows the calculated and measured jet widths. The definition of the jet width is given in Fig. 5. The measured jet widths were obtained from photographs obtained with the digital camera set normal to the \( x-z \) plane. The width of the planar jet decreases with increases in its distance from the nozzle exit. The numerical predictions are in reasonable agreement with the experimental results.

Figure 6(b) shows the calculated and measured thicknesses of the planar jet at \( z = 7 \text{ mm} \). The thickness of the planar jet was measured with the stylus method and its definition is shown in Fig. 5. The planar jet is almost uniform in the central region \((0 \leq x/d_0 \leq 5)\) and thick at the rim. The dimensionless thickness of the jet in the central region is smaller than unity, indicating that the jet sheet is thinner than the nozzle gap, which arises because gravity leads to an increase in the vertical velocity of the liquid, resulting in a reduction in the thickness of the planar jet that is governed by mass conservation. The prediction is in agreement with the experimental result, although a slight difference is evident.

After the impingement of the planar jet on the solid, the liquid film flows on the solid surface. Figure 7 shows the distributions of the calculated and measured film thicknesses on the solid surface in the \( x \) direction at \( y = 3 \text{ mm} \) (a), \( y = 6 \text{ mm} \) (b), \( y = 9 \text{ mm} \) (c), and \( y = 12 \text{ mm} \) (d). It is obvious that the liquid film is non-uniform and three-dimensional. The dimensionless film thickness on the substrate is approximately 0.5 at the center \((x=0)\), and the film is thin near the edges of the substrate. The predicted film thickness is in reasonable agreement with the experimental data for \( y = 3 \text{ mm} \). There is a discrepancy between the predictions and the experimental results in the downstream region. One reason for this discrepancy is the problem of numerical errors arising from the finite difference approximation and the neglect of turbulence. Experimental difficulties are also a factor: the calculations were conducted assuming a steady-state flow, but the experimental flows are unsteady. The flow fluctuation is appreciable in the downstream region, which contributes to the discrepancy between the predicted and measured data.

4.2. Flow Structure

Numerical experiments were conducted under the four flow conditions listed in Table 1. First, the effects of varying the velocity profile were examined for a mean velocity at the nozzle exit of \( W_0 = 1.4 \text{ m/s} \) and a nozzle-plate distance of \( h = 12 \text{ mm} \). Figure 8 shows the predicted free liquid surface, the pressure contours in the \( y = 0 \) and \( z = 0 \) planes, and the streamlines. In both cases, the horizontal width of the planar jet decreases with increases in the distance from the nozzle exit. The streamlines in the falling planar jets are almost vertical except at the rims. The flow direction shifts from vertical to horizontal in the stagnation region where the pressure is high. The streamlines are almost parallel to the \( y \)-axis on the solid substrate.

Figure 9 shows the predicted free surface, velocity distribution, and pressure profile before the impact for the horizontal cross-section of the planar jet at \( z = 7 \text{ mm} \). Note that
the velocity components \((u, v)\) in the \(x\) and \(y\) directions are plotted, but the component, \(w\), in the \(z\) direction is not shown. The cross-section of the planar jet consists of two round rims and an almost uniform liquid sheet between them. In the uniform sheet region, the velocity components in the \(x\) and \(y\) directions are very small and the pressure is almost equal to the atmospheric pressure. At the rims, the pressure is high because of surface tension. The pressure just inside the rims in the \(x\) direction is lower than atmospheric pressure, because the free liquid surface is concave with respect to the atmosphere. The pressure gradient induces inward flows at the rims, resulting in the retraction of the jet width. The jet width in (a) is slightly smaller than that in (b). As the vertical velocity in the calculated flow profile at the nozzle exit is small near the edges (nozzle wall), the liquid velocity at the rims is smaller than that for the case of uniform flow. As a consequence, the surface tension acts on the rims for a longer time, which promotes the retraction of the liquid sheet.

Figure 10 shows the predicted free surface and the velocity distribution for a horizontal cross-section of the planar jet at \(z=2\) mm. The two dotted lines in the figure are the edges of the solid substrate \((x/d_0=\pm6.17)\). The rims are located outside the solid surface and only a uniform sheet impinges on the substrate, which gives rise to streamlines parallel to the \(y\)-axis on the solid (see Figs. 8(a) and 8(b)). In addition, the rims are larger than those at \(z=7\) mm in Fig. 9.

Figure 11(a) shows the distribution of the pressure relative to atmospheric pressure at the stagnation line \((y=0, z=0, 0\leq x/d_0\leq6.17)\) for the four impact conditions listed in Table 1. The stagnation pressure is associated with the velocity at the nozzle exit, and is larger for a larger velocity at the nozzle exit at \(y=0\) and for a larger nozzle-plate distance.
In all these cases, the stagnation pressure is almost uniform except near the edge \(x/d_0 = 6.17\), where it sharply decreases down to atmospheric pressure. This fact suggests that three-dimensional nature of the flow appears in the vicinity of the plate edge. Figure 11(b) shows the pressure distribution at the plate surface in the stagnation region \((x/H_1 = 0, z/H_1 = 0, y/d_0 = 2)\). The pressure is maximum at the stagnation line and decreases sharply with \(y\).

Figure 12 shows the free surface and velocity profiles of the liquid film on the solid substrate for vertical cross-sections at \(y = 3, 6, 9, \) and 12 mm. Note that the velocity components \((u, w)\) in the \(x\) and \(z\) directions are plotted, but the component, \(v\), in the \(y\) direction is not shown. The two calculated flow fields are not identical, but are similar. The liquid moves from the edge to the atmosphere at \(y = 3\) mm, resulting in a reduction in the film thickness near the edge. This outward flow is induced by the sharp pressure gradient between the stagnation pressure and atmospheric pressure at the impact line, as shown in Fig. 11(a). The outward velocity at the edge becomes small in the downstream region. The shape of the free surface is round and convex with respect to the atmosphere in the vicinity of the edge. As shown in Fig. 13, the pressure near the edge is slightly higher than atmospheric pressure due to surface tension. Thus the pressure profile leads to inward flow at \(x/d_0 = 5.5\), although it is very slow. The presence of the plate edge appreciably affects the flow structure of the liquid film, which cannot be predicted by 2-dimensional simulations.

The numerical results for a longer nozzle-plate distance are now discussed. As expected, the horizontal jet width in the \(x\) direction at the impact line decreases with increases in the nozzle-to-plate distance. We focus on the cases where the planar jet width is smaller than the plate size at the impact line. Figure 14 shows the streamlines, the shape of the free surface, and the pressure profiles for the \(y = 0\) and \(z = 0\)
planes for (a) $W_0=1.4 \text{ m/s}$ and (b) $W_0=2.1 \text{ m/s}$. The nozzle-plate distance is $h=24 \text{ mm}$ and the velocity profile at the nozzle exit was found by evaluating the flow profile in the slot nozzle. In both cases, the rims of the planar jet impinge partially on the solid, resulting in a wavy free surface on the solid. In Fig. 14(a), large portions of the rims impinge on the solid. The inward flow in the rims gives rise to oblique streamlines from the edge at the impact line to the center. The streamlines are almost parallel to the $y$-axis except near the edge in Fig. 14(b).

Figure 15 shows the velocity fields in the vertical planes at $y=3$, 6, 9, and 12 mm for (a) $W_0=1.4 \text{ m/s}$ and (b) $W_0=2.1 \text{ m/s}$. In (a), a pair of liquid swellings formed by the impingement of the rims on the solid are seen at $y=3 \text{ mm}$. The flow direction in the liquid swelling is toward the center, and outward flows appear at the edges. The outward flows at the edges vanish in the region $y=6 \text{ mm}$. The liquid film becomes thin near the edges in the downstream region in accordance with the motion of the liquid swellings. In Fig. 15(b), small liquid swellings are observed near the edges at $y=3 \text{ mm}$. The liquid in the swelling moves inward and outward flows are seen at the edges. As a consequence, a pair of weak vortex structures is formed. The velocity in the vertical plane is relatively small compared to the case shown in Fig. 15(a).

Figure 16 shows the free surface and the velocity distribution in the vertical plane $x/d_0=0$ for $W_0=1.4 \text{ m/s}$, with nozzle-plate distances of (a) $h=12 \text{ mm}$ and (b) $24 \text{ mm}$. In both cases, a stagnation region appears near the impact point. The velocity boundary layer forms along the solid surface from the impact point. The dimensionless film thickness is almost constant ($f=0.5$) in the region $y/d_0<6$. In principle, wall friction causes a reduction in the mean velocity of the liquid film in the $y$ direction, resulting in an increase in the film thickness with increases in the distance from the impact point. The change in the film thickness due to wall friction is, however, negligibly small in the present case. The liquid film thickens in the downstream region in...
Fig. 16(b). This is because the two liquid swellings (oblique flows) in Fig. 16(a) coalesce at the center axis ($x=0$). Such flow structure of the liquid film cannot be predicted by 2-dimensional simulations.

5. Conclusions

The flow structure of a planar liquid jet impinging on a solid substrate was studied with three-dimensional computer simulations and experiments. The calculated results were compared to the experimental results for model validation, and reasonable agreement was found. The flow structures were then investigated in detail. The stagnation pressure was found to be dependent on the velocity profile at the nozzle exit, the nozzle-plate distance, and the mean jet velocity. The liquid film formed by the impingement of the planar jet on the solid substrate becomes wavy when the rims of the planar jet impinge on the plate. Oblique flows are seen at the edge on the impact line that meet at $y/H=0$ in a downstream point, resulting in a slight increase in the liquid film thickness.

In all the cases examined in the present study, the liquid flows out from the edge to the atmosphere near the line of impact of the planar jet onto the solid and the liquid film becomes thin. In the downstream region, the outward flows vanish at the edges. Because the shape of the free surface at the edges is round and convex with respect to the atmosphere, the surface tension increases the pressure slightly. As a consequence, a flow toward the center ($x=0$) is induced and a three-dimensional flow arises.

REFERENCES