

Two-Dimensional and Axisymmetric Viscous Flow in Apertures

By SADEGH DABIRI¹, WILLIAM A. SIRIGNANO¹
AND DANIEL D. JOSEPH²

¹Department of Mechanical and Aerospace Engineering, University of California, Irvine, CA
92697, USA

²Department of Aerospace Engineering and Mechanics, University of Minnesota, Minneapolis,
MN 55455, USA

(Received 27 June 2007)

The flow of a planar liquid jet out of an aperture is simulated by solving the unsteady incompressible Navier-Stokes equations. A convective equation is solved for the level set to capture the interface of the liquid jet with the gaseous environment. The flows for different Reynolds numbers and Weber numbers are calculated. Results show that, for $We = \infty$, a maximum value of discharge coefficient appears for $Re = O(100)$. Using the total-stress criterion for cavitation, the regions that are vulnerable to cavitation are identified and the results are compared to the solution of viscous potential flow. It is proved that the inviscid potential flow satisfies the normal stress boundary condition on free surface of a viscous flow as well. The results are close to viscous potential solution except inside the boundary layers. Navier-Stokes solution for the axisymmetric aperture are also presented for two values of Reynolds number. These axisymmetric results are qualitatively similar to the planar results but have a lower discharge coefficient and less contraction in terms of transverse length dimension.

1. Introduction

High-pressure atomizers and spray generators are of great interest in industry and have many applications such as combustors, drying systems and agricultural sprays.

Although it is known that generally the liquid/air interaction is very important in the break up of liquid jets, recent experimental studies by Tamaki *et al.* (1998, 2001) and Hiroyasu (2000) show that the disturbances inside the nozzle caused by the collapse of traveling cavitation bubbles make a substantial contribution to the break-up of the exiting liquid jet. Even with high pressure drops, the main flow of liquid jet does not atomize greatly when a disturbance caused by cavitation is not present. Nurick (1976) also observed that the presence of cavitation in nozzle will decrease the uniformity of the mixing for unlike impinging doublets. He & Ruiz (1995) studied the effect of cavitation on turbulence in plain orifices flows. In their experiment they measured the velocity field for both cavitating and noncavitating flow in the same geometry. They observed that the impingement of the free surface flow onto the orifice wall increases the turbulence generation behind the cavity. Also, turbulence in the cavitating flow is higher and decays more slowly than that in the noncavitating flow.

Many numerical studies have been performed on the cavitation inside the orifice flow (Xu *et al.* (2004), Chen & Heister (1996), Bunnell & Heister (2000)). Bunnell *et al.* (1999) studied the unsteady cavitating flow in a slot and found that partially cavitating slots show a periodic oscillation with Strouhal number near unity based on orifice length and Bernoulli velocity. Different models for two-phase flow and cavitation are used. For example Kubota *et al.* (1992) derived a constitutive equation for the pseudo density from the Rayleigh-Plesset equation for bubble dynamics. These models are based on pressure and are neglecting the viscous stress effects. In the present work, we consider the effects of viscous stress in finding the potential regions of cavitation.

Many experimental studies have been performed on cavitation in orifices as well. Mishra & Peles (2005*a,b*) looked at the cavitation in flow through a micro-orifice inside a silicon microchannel. Payri *et al.* (2004) studied the influence of cavitation on the internal flow and the spray characteristics in diesel injection nozzles. Ahn *et al.* (2006) experimentally studied the effects of cavitation and hydraulic flip on the breakup of the liquid jet injected perpendicularly in to subsonic crossflow. They showed that cavitation results in shortening the liquid column breakup length. Jung *et al.* (2006) considered the breakup characteristics of liquid sheets formed by a like-doublet injection. They found that liquid jet turbulence delays sheet breakup and shorten wavelength of both ligaments and sheets. Ganippa *et al.* (2004) considered the cavitation growth in the nozzle as they increased the flow rate. First, traveling bubbles are created. These bubbles are detached from the wall and move with the stream. By increasing the flow, the unsteady cloud of cavitation is observed. Further increase of the flow rate caused the non-symmetrical distribution of cavitation within the nozzle and its extension to the nozzle exit. More atomization occurs at the side with stronger cavitation.

The dynamics of liquid sheets also has been extensively studied. These sheets are important in atomization and spray combustion (Lefebvre (1989)) and curtain coating (Brown (1961)). Jets created by slot atomizers are close to 2-D flows.

Flow through an aperture is a simple example of flow with hydraulic flip that occurs in nozzles with sharp corners. In the experiments on cavitating orifices, the occurrence of hydraulic flip coincides with the disappearance of cavitation bubbles and increase in the breakup length of the jet (e.g. Tamaki *et al.* (1998)). However, the present work shows that cavitation is possible in hydraulically flipped flows, especially in the case of low Reynolds numbers where the viscous stress is significant.

In this paper, we are interested in the prediction of possibility of cavitation that would result in the formation of detached vapor bubbles that travel with the liquid.

According to the traditional criterion, cavitation occurs when the pressure drops below the breaking strength of liquid. This threshold value depends on the type of nucleation. In the homogeneous nucleation, the nucleation sites are temporary microscopic voids that are results of the thermal motion within the liquid. In this case, the critical pressure could be much lower than the vapor pressure. However, in heterogeneous nucleation which occurs in the most of the engineering situations, rupture occurs at the boundary between the liquid and solid wall of container or between liquid and small solid particles suspended in the liquid. In this case, rupture could occur at pressures closer to the vapor pressure.

Joseph (1998) proposed that the important parameter in cavitation is the total stress which include both the pressure and viscous stress. Therefore, the cavitation occurs when the maximum principal stress drops below the breaking strength of liquid. Using this criterion, Funada *et al.* (2006) predicted the cavitation of a two-dimensional steady viscous potential flow through an aperture. Also, for axisymmetric viscous flow through an orifice, Dabiri *et al.* (2007) predicted cavitation using Joseph's total-stress criterion to post-process the solutions of the Navier-Stokes equations. These papers take a common approach although the configurations vary widely. Using the new criterion, the likelihood for cavitation at each point in the field is determined. Of course, if cavitation does occur, the flow field can be drastically changed. In that sense, the analytical approach is similar to the approach used in hydrodynamic stability analysis where the field evaluated for likelihood of instability is very different from the field that occurs in the unstable situation. A similar approach is used in this paper.

Except for Dabiri *et al.* (2007), the previous calculations done on the subject of high-

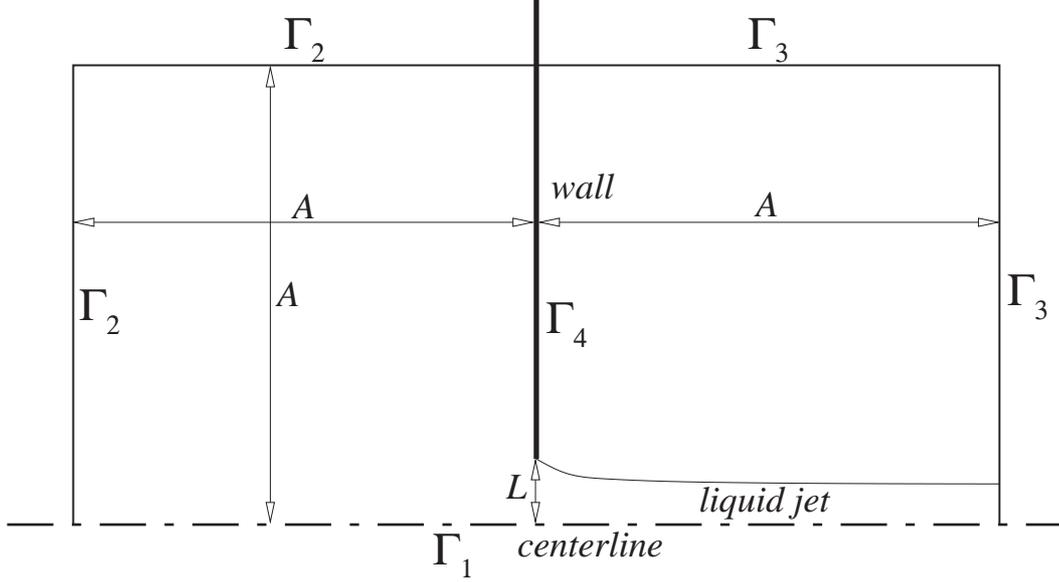


Figure 1: Geometry of the computational domain.

pressure-nozzle cavitation have used the traditional criterion. The purpose of this paper is to use the new criterion to study the cavitation in aperture flow.

2. Theoretical Development

2.1. Navier-Stokes flow

In this study, we consider flow of a liquid departing an aperture in a flat plate and creating a jet in a stagnant gas. The physical problem and the computational domain is shown in figure 1. In this figure, A is the size of computational domain and L is the half width of the aperture. The Navier-Stokes equations for an unsteady, incompressible viscous flow are

$$\rho_i \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot (2\mu_i \mathbf{D}) + \sigma \kappa \delta(d) \mathbf{n} \quad (2.1)$$

$$\mathbf{D} = \frac{1}{2} [(\nabla \mathbf{u}) + (\nabla \mathbf{u})^T] \quad (2.2)$$

$$\nabla \cdot \mathbf{u} = 0 \quad (2.3)$$

where \mathbf{u} , ρ and μ are the velocity, density and viscosity of the fluid, respectively. Subscript i could represent either liquid or gas phase and \mathbf{D} is the strain rate tensor. The last term represents the surface tension as a force concentrated on the interface. Here σ is the surface tension coefficient, κ is the curvature of interface, δ is the Dirac delta function. d represents the distance from interface and \mathbf{n} corresponds to the unit normal vector at the interface. The flow is characterized by the gas-to-liquid density ratio, viscosity ratio and nondimensional parameters: Reynolds number (Re) and Weber number (We) which are defined as follows:

$$Re = \frac{\rho_{liq} U L}{\mu_{liq}}, \quad We = \frac{\rho_{liq} U^2 L}{\sigma}, \quad \rho\text{-ratio} = \frac{\rho_{gas}}{\rho_{liq}}, \quad \mu\text{-ratio} = \frac{\mu_{gas}}{\mu_{liq}} \quad (2.4)$$

$$U = \sqrt{\frac{2(p_u - p_d)}{\rho_{liq}}} \quad (2.5)$$

where L is the half width of the aperture, U is the Bernoulli velocity of jet and p_u and p_d are the upstream and downstream pressures respectively.

After finding the velocities and pressure field, one can calculate the stress tensor using

$$\mathbf{T} = \mu [(\nabla \mathbf{u}) + (\nabla \mathbf{u})^T] - p \mathbf{I} \quad (2.6)$$

where \mathbf{I} is the identity matrix and superscript T refers to transpose of a tensor. In the planar flow the stress tensor has the following form

$$\mathbf{T} = \begin{bmatrix} T'_{11} & T'_{12} & 0 \\ T'_{21} & T'_{22} & 0 \\ 0 & 0 & 0 \end{bmatrix} \quad (2.7)$$

Therefore, the maximum tensile stress, T_{11} can be calculated using the planar stress analysis in $x - y$ plane:

$$T_{11} = \frac{T'_{11} + T'_{22}}{2} \pm \sqrt{\left(\frac{T'_{11} - T'_{22}}{2}\right)^2 + T'^2_{12}} \quad (2.8)$$

The new criterion for cavitation proposed by Joseph (1998) is used to find the cavitating regions in the flow field. According to this criterion, cavitation occurs when the maximum principal stress exceeds the negative of the critical threshold pressure of liquid at local temperature. According to the total-stress criterion, the cavitation occurs where:

$$T_{11} > -p_c \quad (2.9)$$

The critical threshold pressure p_c might be the vapor pressure p_v or some other appropriate value. The cavitation number, K , defines the critical threshold pressure, p_c , in a nondimensional manner:

$$K = \frac{p_u - p_d}{p_d - p_c} \quad (2.10)$$

2.2. Interface tracking and level set formulation

Several methods have been proposed and implemented to capture the interface and model the surface tension in a two-phase flow such as the Front-Tracking method by Tryggvason *et al.* (2001) and Volume-of-Fluid method by Hirt & Nichols (1981). Also, Popinet & Zaleski (1999) did an accurate balance of surface tension forces on a finite volume method by explicit tracking of the interface. A review of different methods of interface tracking and surface tension modeling is done by Scardovelli & Zaleski (1999).

We are considering incompressible flow of two immiscible fluids. The interface between these fluids moves with the local velocity of flow field. To track the motion of interface the level set method is used which has been developed by Osher and coworkers (e.g., Sussman *et al.* (1998) and Osher & Fedkiw (2001)). The level set function, denoted by θ ,

is defined as a signed distance function. It has positive values on one side of the interface (gas phase), and negative values on the other side (liquid phase). The magnitude of the level set at each point in the computational field is equal to the distance from that point to interface.

The level set function is being convected by the flow as a passive scalar variable:

$$\frac{\partial \theta}{\partial t} + \mathbf{u} \cdot \nabla \theta = 0 \quad (2.11)$$

It is obvious that, if the initial distribution of the level-set is a signed distance function, after a finite time of being convected by a nonuniform velocity field, it will not remain a distance function. Therefore, we need to re-initialize the level-set function so it will be a distance function (with property of $|\nabla \theta| = 1$) without changing the zero level set (position of the interface).

Suppose $\theta_0(\mathbf{x})$ is the level-set distribution after some time step and is not exactly a distance function. This can be reinitialized to a distance function by solving the following partial differential equation (Sussman *et al.* (1998)):

$$\frac{\partial \theta'}{\partial \tau} = \text{sign}(\theta_0)(1 - |\nabla \theta'|) \quad (2.12)$$

with initial conditions:

$$\theta'(\mathbf{x}, 0) = \theta_0(\mathbf{x})$$

where

$$\text{sign}(\theta) = \begin{cases} -1 & \text{if } \theta < 0 \\ 0 & \text{if } \theta = 0 \\ 1 & \text{if } \theta > 0 \end{cases} \quad (2.13)$$

and τ is a pseudo time. The steady solution of equation (2.12) is the distance function with property $|\nabla \theta| = 1$ and since $\text{sign}(0)=0$, then θ' has the same zero level set as θ_0 .

Now using the level set definition, the fluid properties can be defined as:

$$\rho = \rho_{liq} + (\rho_{gas} - \rho_{liq})H_\epsilon(\theta) \quad (2.14)$$

$$\mu = \mu_{liq} + (\mu_{gas} - \mu_{liq})H_\epsilon(\theta) \quad (2.15)$$

where H_ϵ is a Heaviside function that is numerically approximated by a smooth jump:

$$H_\epsilon = \begin{cases} 0 & \theta < -\epsilon \\ (\theta + \epsilon)/(2\epsilon) + \sin(\pi\theta/\epsilon)/(2\pi) & |\theta| \leq \epsilon \\ 1 & \theta > \epsilon \end{cases} \quad (2.16)$$

where ϵ represents the numerical thickness of the interface and has the value of 1.5 times is cell size. This Heaviside function corresponds to a delta function that can be used to evaluate the force caused by surface tension:

$$\delta_\epsilon = \begin{cases} [1 + \cos(\pi\theta/\epsilon)]/(2\epsilon) & |\theta| \leq \epsilon \\ 0 & \text{otherwise} \end{cases} \quad (2.17)$$

The last term in the momentum equation (2.1) includes the normal unity vector and the curvature of the interface which can be calculated as follows:

$$\mathbf{n} = \frac{\nabla\theta}{|\nabla\theta|}, \quad \kappa = -\nabla \cdot \mathbf{n} \quad (2.18)$$

expanding the equation (2.18) in Cartesian coordinates leads to the following equation:

$$\kappa(\theta) = \frac{\theta_y^2\theta_{xx} - 2\theta_x\theta_y\theta_{xy} + \theta_x^2\theta_{yy}}{(\theta_x^2 + \theta_y^2)^{3/2}} \quad (2.19)$$

2.3. Viscous potential flow

The solution of inviscid flow has been used widely in the literature to treat the flow problems with finite viscosity. For example, Moore (1965) studied the rise of a deformed bubble in a liquid of small viscosity by calculating the dissipation of an irrotational flow around the bubble. Also, Joseph & Wang (2004) considered the viscous potential flow for decay of surface gravity waves. The viscous potential flow solution does satisfy the

Navier-Stokes equations but does not satisfy the boundary condition for the tangential component of the velocity at a rigid surface and the tangential component of shear at a free surface.

In Appendix A, it is shown that the potential flow solution of flows with free surfaces satisfies the normal stress boundary condition on the free surface in the case of finite viscosity as well. Therefore, the viscous potential flow solution will be used here as a comparison to the Navier-Stokes solution of the aperture problem. The problem of incompressible potential flow through an aperture was solved a long time ago. The complex potential in z -plane for this flow is given implicitly by Currie (1974) (p.129)

$$f(z) = \phi + i\psi = -\frac{2C_cLU}{\pi} \ln \left\{ \cosh \left[\ln \left(U \frac{dz}{df} \right) \right] \right\} - iC_cLU \quad (2.20)$$

where L is half width of the aperture and C_c is the coefficient of contraction. Funada *et al.* (2006) has analyzed the viscous potential flow solution of the aperture flow. The velocity field can be derived from the potential function as follows:

$$u = \frac{1}{2} \left(\frac{df}{dz} + \frac{d\bar{f}}{d\bar{z}} \right), \quad v = \frac{i}{2} \left(\frac{df}{dz} - \frac{d\bar{f}}{d\bar{z}} \right), \quad (2.21)$$

and from there the rate of strain tensor can be calculated:

$$2\mathbf{D} = \begin{bmatrix} \left(\frac{d^2f}{dz^2} + \frac{d^2\bar{f}}{d\bar{z}^2} \right) & i \left(\frac{d^2f}{dz^2} - \frac{d^2\bar{f}}{d\bar{z}^2} \right) \\ i \left(\frac{d^2f}{dz^2} - \frac{d^2\bar{f}}{d\bar{z}^2} \right) & - \left(\frac{d^2f}{dz^2} + \frac{d^2\bar{f}}{d\bar{z}^2} \right) \end{bmatrix} \quad (2.22)$$

To calculate the maximum tension, the principal stresses should be found. The diagonalized rate of strain tensor is

$$2\mathbf{D} = \begin{bmatrix} \lambda & 0 \\ 0 & -\lambda \end{bmatrix}, \quad \lambda = 2 \left| \frac{d^2f}{dz^2} \right| \quad (2.23)$$

Therefore, the maximum tension T_{11} is given by

$$T_{11} = -p + \mu\lambda = -p_u + \frac{\rho}{2}(u^2 + v^2) + \mu\lambda \quad (2.24)$$

3. Numerical Implementation

The numerical solution of the incompressible unsteady Navier-Stokes equations is performed using the finite-volume method on a staggered grid. The convective term is discretized using the Quadratic Upwind Interpolation for Convective Kinematics (QUICK) (Hayase *et al.* (1992)). The Semi-Implicit Method for Pressure-Linked Equation (SIMPLE), developed by Patankar (1980), is used to solve the pressure-velocity coupling. The time integration is accomplished using the second-order Crank-Nicolson scheme. The calculation is done for different Reynolds-numbers based on orifice diameter and average velocity at orifice.

The computational domain is shown in figure 1. The size of the domain is $A = 20L$ and a Cartesian grid with 77924 nodes and 77361 elements is employed. A nonuniform distribution of grids is used with clustering in the region of rapid velocities near the tip of the aperture. The size of smallest elements is $\Delta x/L = 0.001$. Figure 2 shows the cartesian grid used. The following boundary conditions are applied: Γ_1 is the axis of symmetry and the v-velocity is zero also the normal derivative of all other variables vanish. On the upstream boundary Γ_2 the stagnation pressure is specified as the boundary condition. On the downstream boundary Γ_3 the static pressure is specified. On the aperture plate Γ_4 all the velocity components are set to zero.

The dependence of the solution on the size of the domain is investigated. In order to ensure the accuracy of the constant pressure boundary conditions, a larger domain is considered with $A = 30L$. Comparing the results for $Re = 100$ shows that the difference in discharge coefficient is below 0.002%. In addition, calculation is done for a finer grid with the total number of nodes being doubled while keeping the same grid distribution. Comparison between two calculations for $Re = 1000$ has shown that discharge coefficient for the two cases differ by less than a 0.1%.

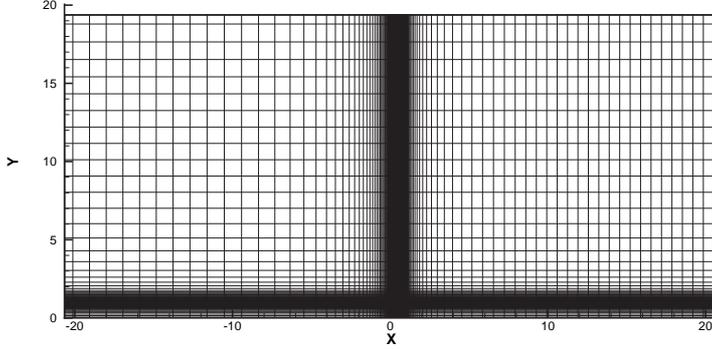


Figure 2: Computational domain and the Cartesian grid that is used.

In order to compare the results with the theoretical viscous potential flow solution, a dynamically inactive environment is required which has been achieved by decreasing the viscosity and density of the gaseous phase. In the case with ρ -ratio and μ -ratio of 10^{-4} each, the flow becomes independent of any further decrease in these parameters. Therefore, the calculations are performed for these ratios.

3.1. *Effects of Reynolds number*

Figure 3 shows the free streamline (liquid-gas interface) for flows with different Reynolds numbers. It can be seen that the free streamline leaves the aperture wall at different angles for different Reynolds numbers. This angle is plotted versus Reynolds number in figure 4.

Figure 5(a) shows the thickness of jet at a distance of $5L$ downstream of the aperture. As Reynolds number decreases the jet thickness increases. This can be explained by the fact that increasing the thickness of boundary layer and decrease in velocity causes the flow to change direction faster. For Reynolds number of one the jet expands. Expansion of Newtonian liquid jets has been observed before, for example by Middleman & Gavis

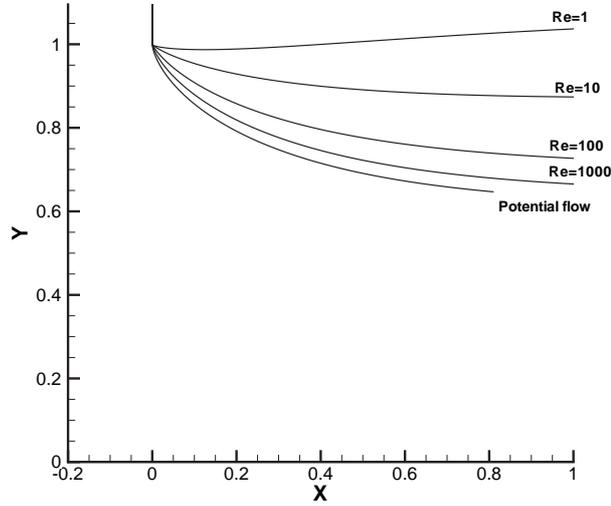


Figure 3: Free streamlines from Navier-Stokes solution for $Re = 1, 10, 100, 1000$ and for potential flow. ρ -ratio= 10^{-4} and μ -ratio= 10^{-4}

(1961). The discharge coefficient of aperture is plotted in figure 5(b). The value of C_d has a peak for $Re = O(100)$. As the Reynolds number decreases from infinity, the thickness of jet increases causing an increase in the discharge coefficient. But for very low Reynolds numbers, the velocity of jet drops, therefore, the discharge coefficient decreases.

Pressure distribution for Navier-Stokes and potential solutions is shown in figure 6 for different Reynolds numbers. (The potential flow solution for lower values of Re is not shown to avoid complexity.) For higher Reynolds numbers, the difference between Navier-Stokes and potential solutions is small but, for lower Reynolds numbers, the pressure field deviates from potential flow. Figure 7 shows the viscous stress in the flow and compares it with the viscous potential flow case. There is a good agreement between them for $Re = 1000$ except in the wall boundary layer. Comparing figures 6 and 7, one can see that the viscous stress is two to three orders of magnitude smaller than the local pressure

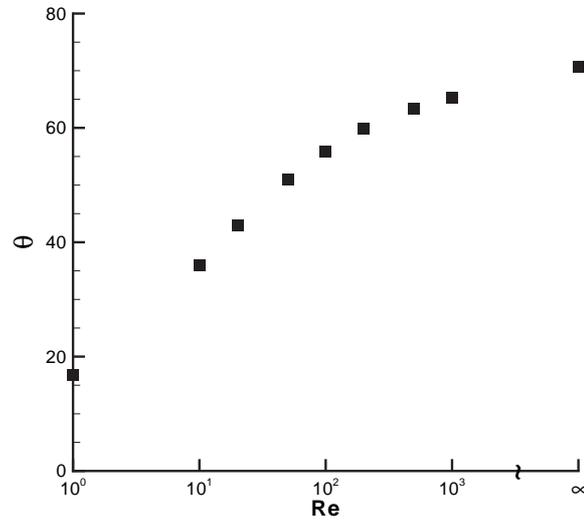


Figure 4: Angle between the free surface and axis of symmetry at $x/L = 0.005$.

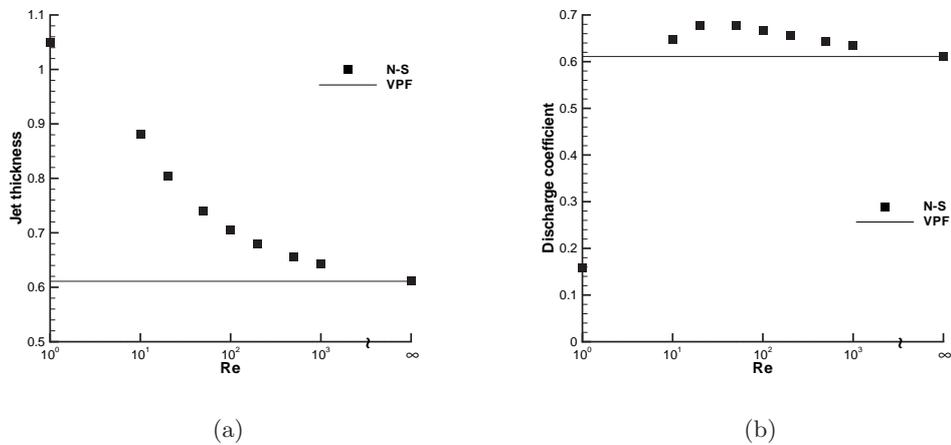


Figure 5: (a) Half thickness of the jet normalized by L and (b) the discharge coefficient for different Reynolds numbers.

drop for the two higher values of Reynolds number. However, for lower Reynolds numbers, such as $Re=10$ or 1 , the viscous stress is the same order of the local pressure drop.

It is important to note that the Reynolds number in these calculations is based on the

Bernoulli jet velocity, which is larger than the actual average velocity of the jet, specially for low Reynolds numbers. For example, for flow with $Re=1$, the jet velocity is about %20 of the Bernoulli velocity and this causes the strain rates and therefore the stresses to be scaled down with the same ratio. This has the significant effect of producing smaller regions of high stresses in the N-S solution as seen in figure 7(d). This is the reason that the regions with large stresses are smaller in the N-S solution.

Calculating the total stress and comparing with the threshold stress, the regions in which the cavitation criterion is satisfied are identified.

The aperture flow is always hydraulically flipped; so, the acceleration of the liquid near the exit corner is not large and the local pressure drop is not significant. Therefore, the chance of having cavitation is related to the contribution of viscous stress in the total stress tensor. Figure 8 shows the regions at risk to cavitation in different flows with the same Reynolds number and different cavitation numbers, K , corresponding to different values of critical pressure, p_c . Although the existence of cavitation bubbles could change the flow field, these bubbles cannot persist far downstream from the inception point where conditions favorable to cavitation have disappeared and the bubbles collapse. We have shown that aperture flows at low Reynolds numbers may cavitate due to viscous stresses under conditions, like hydraulic flip, which are unfavorable to cavitation under the conventional criterion. The experiments on hydraulic flip are done for high Reynolds numbers; the possible cavitation of flipped flows at low Re , due to viscous stresses has not been studied before.

Figure 9 shows how the area of the region vulnerable to cavitation will increase as the cavitation number increases. For a specific value of K , the cavitating domain is larger for lower Reynolds number because the viscous stress is stronger. This agrees with the

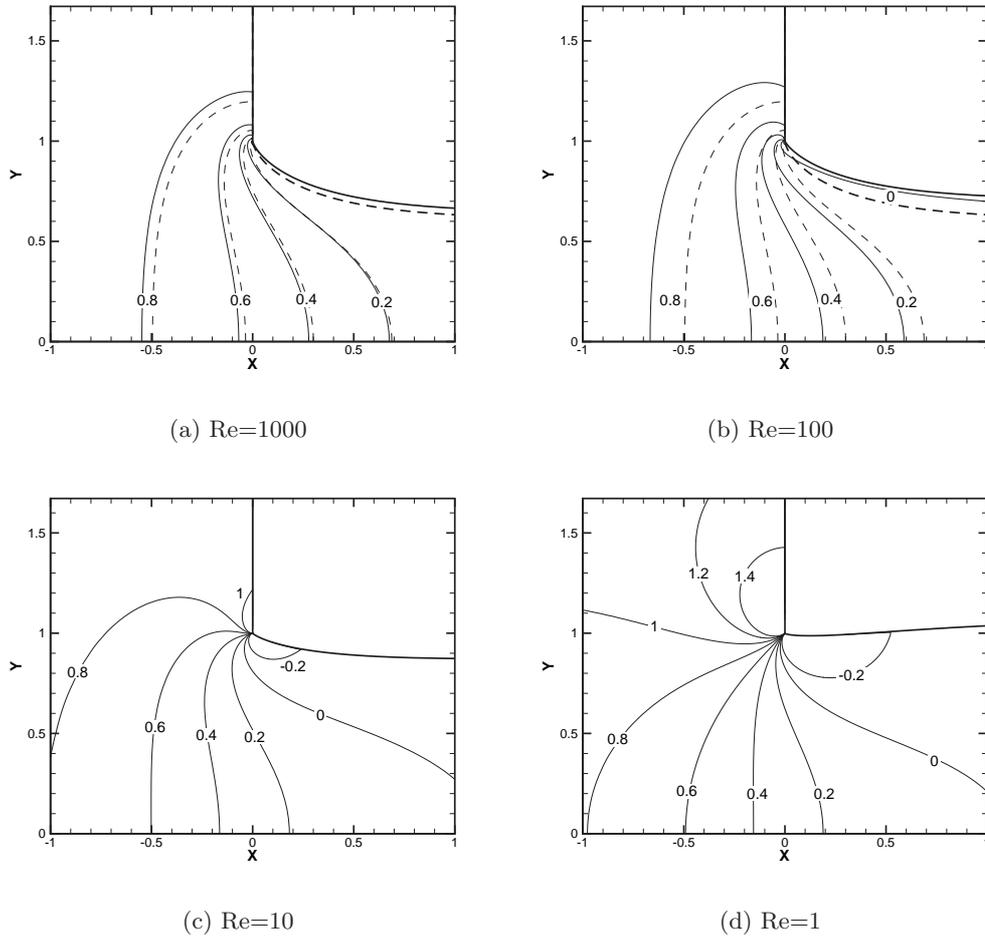


Figure 6: Contours of $(p - p_d)/\frac{1}{2}\rho_{liq}U^2$ for Navier-Stokes solution (solid lines) compared to viscous potential flow solution (dashed lines) for ρ -ratio= 10^{-4} and μ -ratio= 10^{-4} (a) $Re=1000$, (b) $Re=100$, (c) $Re=10$, (d) $Re=1$ (Potential flow solutions are not shown in (c) and (d)).

statement by Padrino *et al.* (2007) about the increase in risk of cavitation for more viscous fluids.

Another important point about the figure 9 is that, for the larger Reynolds numbers, the difference between the cavitating domain predicted by N-S solution and VPF solution

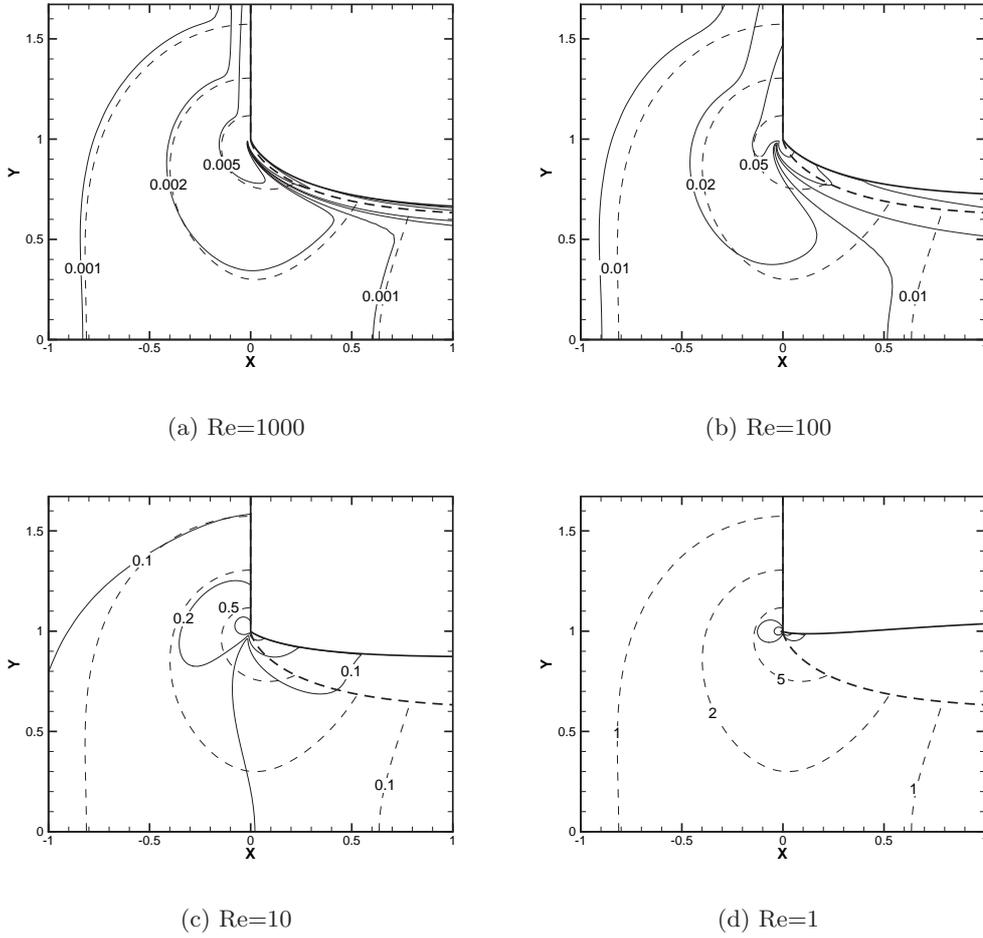


Figure 7: Contours of $T_{11}/\frac{1}{2}\rho_{liq}U^2$ for Navier-Stokes solution (solid lines) compared to viscous potential flow solution (dashed lines) for ρ -ratio= 10^{-4} and μ -ratio= 10^{-4} (a) Re=1000, (b) Re=100, (c) Re=10, (d) Re=1

will be greater. That is, for larger Reynolds numbers, the cavitation area is confined to shear layers and boundary layers which are not present in the potential flow solution.

Calculations were also done for the round aperture assuming axisymmetric flow. A schematic of the flow is shown in figure 10. Figure The results of these calculations are shown in figure 11. The pressure contours in the round aperture (figure 11(a)) are confined to a more compact region than in the planar flow shown in figure 6(b). Flow

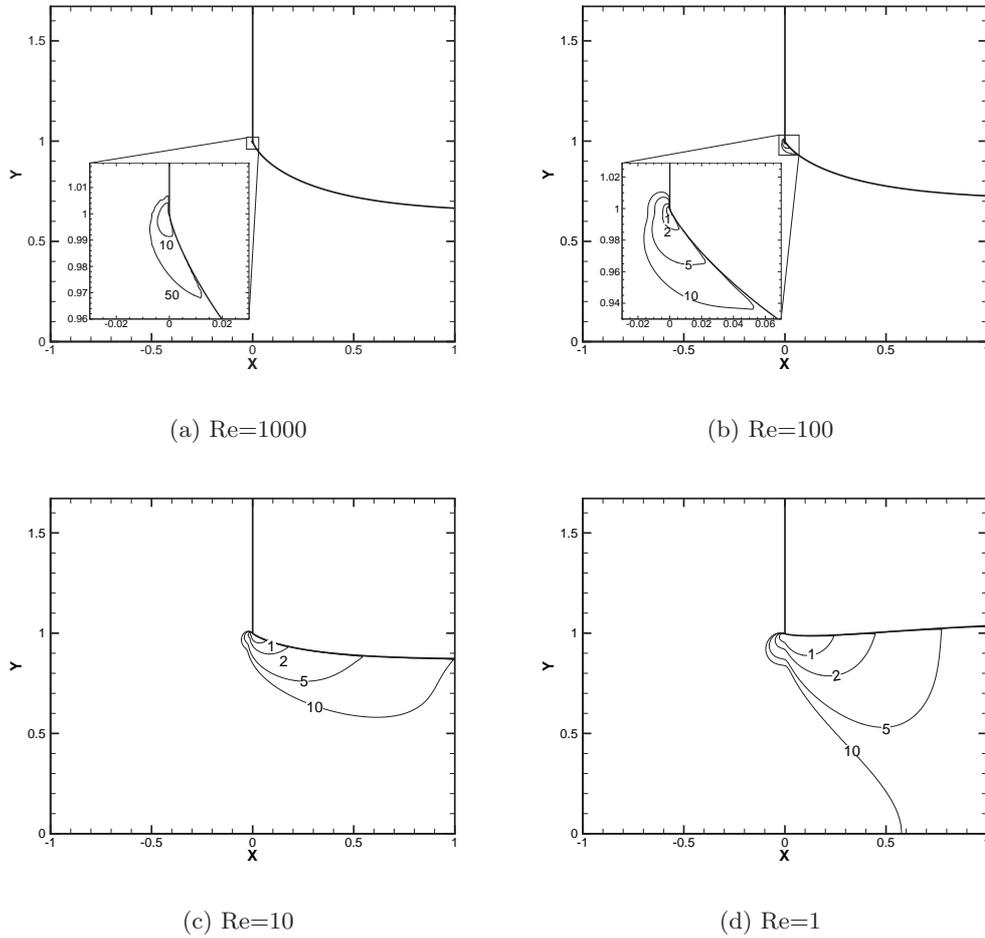


Figure 8: The cavitation threshold curves on which $T_{11} + p_c = 0$ in different flows with $K = 1, 2, 5, 10, 50$. ρ -ratio= 10^{-4} and μ -ratio= 10^{-4} (a) Re=1000, (b) Re=100, (c) Re=10, (d) Re=1

acceleration occurs in a smaller region and gives rise to a higher strain rate and viscous stress in the round aperture as shown in the viscous stress plots in figures 11(c) and 7(b).

3.2. Effects of Weber Number

The flows for Weber numbers of 10, 100, 1000 and infinity are calculated and the free streamlines are shown in figure 12. The flow with Weber number of 1000 is very close

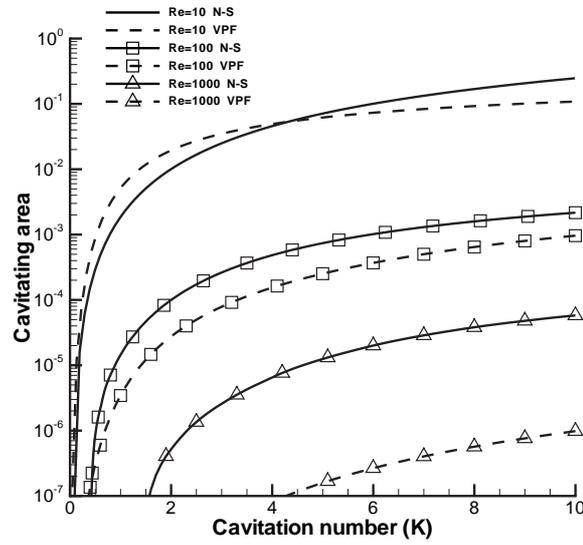


Figure 9: Area of cavitating region normalized by L^2 .

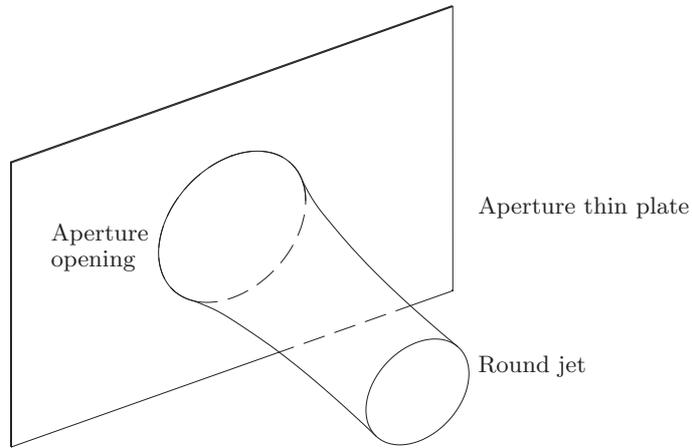


Figure 10: Schematic of axisymmetric flow through a round aperture.

to the flow with no surface tension, or infinite Weber number. The free stream for these two cases cannot be distinguished on these figures. As the Weber number decreases, the jet deviates towards a less contracting jet with smaller curvatures at the interface.

For the flow with Weber number of 10, the potential regions of cavitation is shown

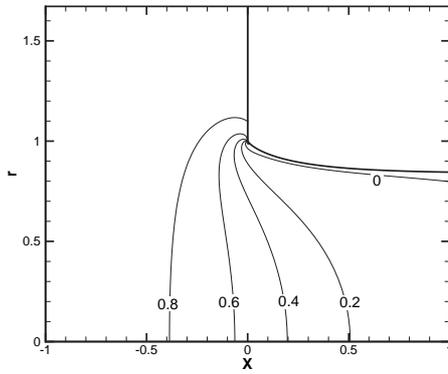
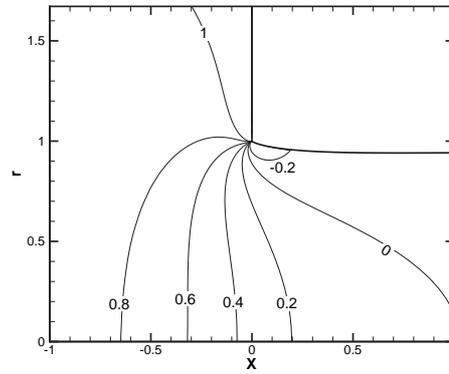
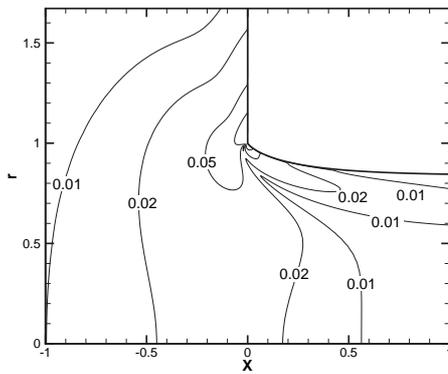
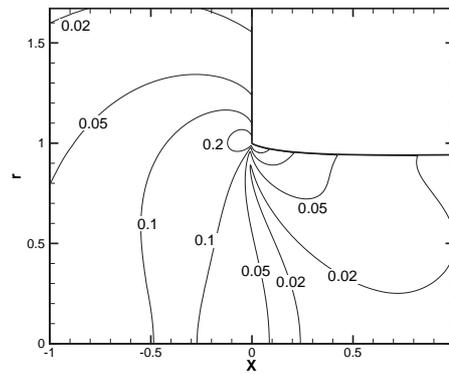
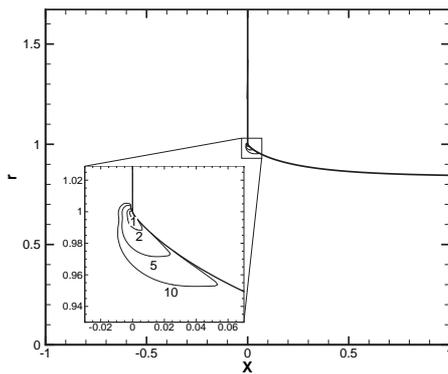
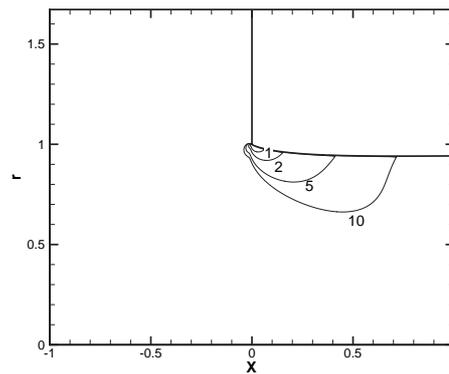
(a) Pressure, $Re=100$ (b) Pressure, $Re=10$ (c) Viscous stress, $Re=100$ (d) Viscous stress, $Re=10$ (e) Cavitation, $Re=100$ (f) Cavitation, $Re=10$

Figure 11: Axisymmetric aperture flow; Contours of $(p - p_d)/\frac{1}{2}\rho U^2$ for (a) $Re=100$, (b) $Re=10$, and maximum principal value of viscous stress for (c) $Re=100$, (d) $Re=10$, and the cavitation threshold curves on which $T_{11} + p_c = 0$ in different flows with $K = 1, 2, 5, 10$ (e) $Re=100$, (f) $Re=10$

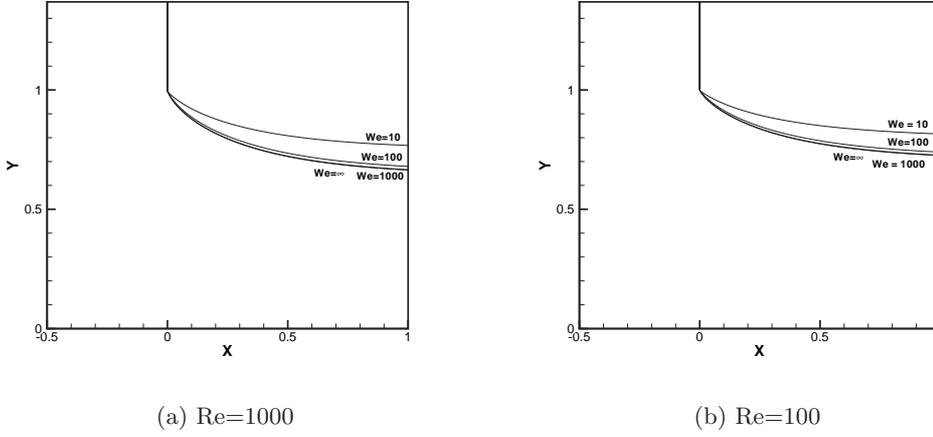


Figure 12: The free streamline for flows with different Weber numbers, ρ -ratio= 10^{-4} and μ -ratio= 10^{-4} (a) Re=1000, (b) Re=100

for Reynolds numbers of 100 and 1000 in figure 13. Comparing these plots with figure 8, reveals a large difference because of surface tension. The pressure on the liquid side of the interface will be smaller due to the curvature of interface and adding the effect of shear stress to that causes a larger domain vulnerable to cavitation at lower Weber numbers. Also, since the boundary layer is larger for the lower Reynolds number, the regions of possible cavitation will be larger.

4. Conclusions

The Navier-Stokes equations for two-dimensional flow of a liquid through an aperture in a flat plate is solved numerically for Reynolds numbers between 1 and 1000. The results are compared to those of the viscous potential flow solution. It is proved that the constant speed condition on the free surface of a potential flow leads to zero normal viscous stress on the free surface, hence, it satisfies the boundary condition of viscous flow

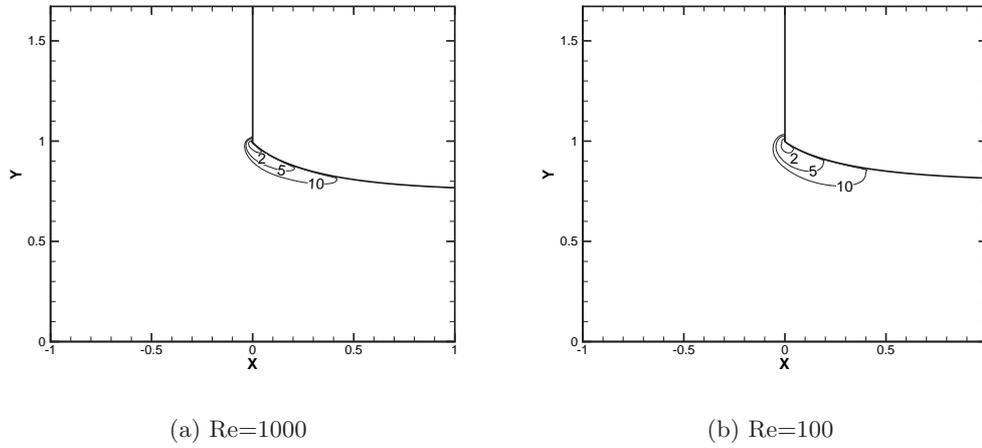


Figure 13: The cavitation threshold curves on which $T_{11} + p_c = 0$ in different flows with $K = 2, 5, 10$. $We = 10$, ρ -ratio= 10^{-4} and μ -ratio= 10^{-4} (a) Re=1000, (b) Re=100

as well. Using the total-stress criterion for cavitation, the regions vulnerable to cavitation are found for flows with different Reynolds number and cavitation number.

Acknowledgment

This research has been supported by the US Army Research Office through grant No. W911NF-06-1-0225, with Dr. Kevin McNesby and Dr. Ralph Anthenien having served sequentially as program managers. D. D. Joseph was also supported by NSF grant No. CBET-0302837.

Appendix A. Boundary condition on normal stress

Here we shall show that the normal strain rate, the derivative of the normal velocity in the direction normal to the free streamline, vanishes on the free streamline of the potential flow solution used by Funada *et al.* (2006). Therefore, the potential flow with free stream satisfies the boundary condition of the normal stress on the free surface of a viscous flow.

We will take the potential function, ϕ , and stream function, ψ , of the potential flow as the orthogonal curvilinear coordinates. The velocity field in this coordinates has a simple form:

$$\begin{cases} x_1 = \phi \\ x_2 = \psi \end{cases} \quad \begin{cases} u_1 = q \\ u_2 = 0 \end{cases} \quad (\text{A } 1)$$

where the velocity in complex domain can be written as:

$$u - iv = qe^{-i\theta} \quad (\text{A } 2)$$

In order to evaluate the stresses, first we define the scale factors:

$$h_1 = h_2 = \frac{1}{\sqrt{u^2 + v^2}} = \frac{1}{q} \quad (\text{A } 3)$$

Calculating the stresses:

$$T_{11} = -p + 2\mu \left[\frac{1}{h_1} \frac{\partial u_1}{\partial x_1} + \frac{u_2}{h_1 h_2} \frac{\partial h_1}{\partial x_2} + \frac{u_3}{h_1 h_3} \frac{\partial h_1}{\partial x_3} \right] = -p + 2\mu q \frac{\partial q}{\partial \phi} \quad (\text{A } 4)$$

$$T_{22} = -p + 2\mu \left[\frac{1}{h_2} \frac{\partial u_2}{\partial x_2} + \frac{u_3}{h_2 h_3} \frac{\partial h_2}{\partial x_3} + \frac{u_1}{h_2 h_1} \frac{\partial h_2}{\partial x_1} \right] = -p - 2\mu q \frac{\partial q}{\partial \phi} \quad (\text{A } 5)$$

$$T_{12} = \mu \left[\frac{h_2}{h_1} \frac{\partial}{\partial x_1} \left(\frac{u_2}{h_2} \right) + \frac{h_1}{h_2} \frac{\partial}{\partial x_2} \left(\frac{u_1}{h_1} \right) \right] = 2\mu q \frac{\partial q}{\partial \psi} \quad (\text{A } 6)$$

Using the Bernoulli equation for viscous potential flow,

$$p_u = p + \frac{1}{2} \rho q^2 \quad (\text{A } 7)$$

Now, we can substitute the pressure back in the equations for normal stresses:

$$T_{22} = -p_u + \frac{1}{2} \rho q^2 - 2\mu q \frac{\partial q}{\partial \phi} \quad (\text{A } 8)$$

Along a streamline we have $d\phi = qds$, where ds is the distance element along the streamline.

$$T_{22} = -p_u + \frac{1}{2} \rho q^2 - 2\mu \frac{dq}{ds} \quad \text{along a streamline} \quad (\text{A } 9)$$

Applying the boundary condition:

$$T_{22} = -p_d \quad \Rightarrow \quad \frac{dq}{ds} = \frac{1}{2\mu} \rho q^2 - \frac{C}{\mu} \quad \text{along the free streamline} \quad (\text{A } 10)$$

where $C = p_u - p_d$ is the pressure difference between stagnation pressure of the flow and ambient pressure.

Now, we show that for the case of a free jet where s is unbounded, the only possible solution is $q = \text{constant}$. If $\frac{dq}{ds} > 0$ initially, then q becomes unbounded, and if $\frac{dq}{ds} < 0$ initially, then q becomes zero and then negative with increasing s . Both of these situations are non-physical, so the only possible solution happens when $\frac{dq}{ds} = 0$ initially, which leads to $q = \text{constant}$. This results in both T_{11} and T_{22} to be constant and equal to $-p$ along the free streamline. Therefore, the irrotational flow with constant pressure at the bounding streamline satisfies the viscous boundary condition of normal stress on the free interface. However, it does not satisfy the condition of zero shear stress on the free surface. To

develop the shear stress more, we consider the irrotationality condition of the flow in Cartesian coordinates:

$$(\nabla \times \mathbf{u})_3 = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 \quad (\text{A } 11)$$

$$= \left(\frac{\partial v}{\partial \psi} \frac{\partial \psi}{\partial x} + \frac{\partial v}{\partial \phi} \frac{\partial \phi}{\partial x} \right) - \left(\frac{\partial u}{\partial \psi} \frac{\partial \psi}{\partial y} + \frac{\partial u}{\partial \phi} \frac{\partial \phi}{\partial y} \right) \quad (\text{A } 12)$$

Using the velocity field,

$$\begin{cases} u = q \cos \theta \\ v = q \sin \theta \end{cases} \quad (\text{A } 13)$$

we get:

$$(\nabla \times \mathbf{u})_3 = -q \frac{\partial q}{\partial \psi} + q^2 \frac{\partial \theta}{\partial \phi} = 0 \quad (\text{A } 14)$$

which results in:

$$\frac{\partial q}{\partial \psi} = q \frac{\partial \theta}{\partial \phi} \quad (\text{A } 15)$$

Substituting back in equation (A 6) and using $d\phi = q ds$ along a streamline again,

$$T_{12} = 2\mu q \frac{d\theta}{ds} = 2\mu q \kappa \quad (\text{A } 16)$$

where κ is the curvature of the streamline. So, in a planar irrotational flow, in an orthogonal coordinates, one of which is parallel to the streamlines, the shear stress is proportional to magnitude of velocity times the curvature of the streamline.

In conclusion, the irrotational flow satisfies the constant normal stress condition on the free surface, but does not satisfy the zero shear stress condition on the free surface and a correction may be necessary.

REFERENCES

- AHN, K., KIM, J. & YOON, Y. 2006 Effects of orifice internal flow on transverse injection into subsonic crossflows: Cavitation and hydraulic flip. *Atomization and Sprays* **16** (1), 15–34.
- BROWN, D. R. 1961 A study of the behaviour of a thin sheet of moving liquid. *Journal of Fluid Mechanics* **10**, 297–305.
- BUNNELL, R. A. & HEISTER, S. D. 2000 Three-dimensional unsteady simulation of cavitating flows in injector passages. *J. Fluid Engineering* **122** (4), 791–797.
- BUNNELL, R. A., HEISTER, S. D., YEN, C. & COLLICOTT, S. H. 1999 Cavitating injector flows: Validation of numerical models and simulations of pressure atomizers. *Atomization and Sprays* **9** (5), 445–465.
- CHEN, Y. & HEISTER, S. D. 1996 Modeling cavitating flows in diesel injectors. *Atomization and Sprays* **6** (6), 709–726.
- CURRIE, I. G. 1974 *Fundamental mechanics of fluids*. McGraw-Hill.
- DABIRI, S., SIRIGNANO, W. A. & JOSEPH, D. D. 2007 Cavitation in an orifice flow. *Physics of Fluids* In press.
- FUNADA, T., WANG, J. & JOSEPH, D. D. 2006 Viscous potential flow analysis of stress induced cavitation in an aperture flow. *Atomization and Sprays* **16** (7), 763–776.
- GANIPPA, L. C., BARK, G., ANDERSSON & CHOMIAK, J. 2004 Cavitation: a contributory factor in the transition from symmetric to asymmetric jets in cross-flow nozzles. *Experiments in Fluids* **36**, 627–634.
- HAYASE, T., HUMPHREY, J. A. C. & GREIF, R. 1992 A consistently formulated quick scheme for fast and stable convergence using finite-volume iterative calculation procedure. *J. Comput. Phys.* **98**, 108–118.
- HE, L. & RUIZ, F. 1995 Effect of cavitation on flow and turbulence in plain orifices for high-speed atomization. *Atomization and Sprays* **5** (6), 569–584.
- HIROYASU, H. 2000 Spray breakup mechanism from the hole-type nozzle and its applications. *Atomization and Sprays* **10** (3-5), 511–527.
- HIRT, C. W. & NICHOLS, B. D. 1981 Volume of fluid (vof) method for the dynamics of free boundaries. *Journal of Computational Physics* **39** (1), 201–225.

- JOSEPH, D. D. 1998 Cavitation and the state of stress in a flowing liquid. *Journal of Fluid Mechanics* **366**, 367–376.
- JOSEPH, D. D. & WANG, J. 2004 The dissipation approximation and viscous potential flow. *J. Fluid Mech.* **504**, 365–377.
- JUNG, K., KHIL, T. & YOON, Y. 2006 Effects of orifice internal flow on breakup characteristic of like-doublet injectors. *J. Propulsion and Power* **22** (3), 653–660.
- KUBOTA, A., KATO, H. & YAMAGUCHI, H. 1992 A new modeling of cavitating flows - a numerical study of unsteady cavitation on a hydrofoil. *Journal of Fluid Mechanics* **240**, 59–96.
- LEFEBVRE, A. H. 1989 *Atomization and Sprays*. Hemisphere.
- MIDDLEMAN, S. & GAVIS, J. 1961 Expansion and contraction of capillary jets of newtonian liquids. *Physics of Fluids* **4** (3), 355–359.
- MISHRA, C. & PELES, Y. 2005a Cavitation in flow through a micro-orifice inside a silicon microchannel. *Physics of Fluids* Art. No. 013601.
- MISHRA, C. & PELES, Y. 2005b Flow visualization of cavitating flows through a rectangular slot micro-orifice ingrained in a microchannel. *Physics of Fluids* **17** (11), art. No. 113602.
- MOORE, D. W. 1965 The velocity of rise of distorted gas bubbles in a liquid of small viscosity. *J. Fluid Mech.* **23**, 749–766.
- NURICK, W. H. 1976 Orifice cavitation and its effect on spray mixing. *Journal of Fluids Engineering* **98** (4), 681–687.
- OSHER, S. & FEDKIW, R. P. 2001 Level set methods: an overview and some recent results. *J. Comput. Phys.* **169**, 436.
- PADRINO, J. C., JOSEPH, D. D., FUNADA, T., WANG, J. & SIRIGNANO, W. A. 2007 Stress-induced cavitation for the streaming motion of a viscous liquid past a sphere. *Journal of Fluid Mechanics* **578**, 381–411.
- PATANKAR, S. V. 1980 *Numerical heat transfer and fluid flow*. Hemisphere, Washington, DC/New York.
- PAYRI, F., BERMUDEZ, V., PAYRI, R. & SALVADOR, F. J. 2004 The influence of cavitation on the internal flow and the spray characteristics in diesel injection nozzles. *Fuel* **83** (4-5), 419–431.

- POPINET, S. & ZALESKI, S. 1999 A front-tracking algorithm for accurate presentation of surface tension. *Int. J. Numer. Meth. Fluids* **30**, 775–793.
- SCARDOVELLI, R. & ZALESKI, S. 1999 Direct numerical simulation of free-surface and interfacial flow. *Annual Review of Fluid Mechanics* **31**, 567–603.
- SUSSMAN, M., FATEMI, E., SMEREKA, P. & OSHER, S. 1998 An improved level set method for incompressible two-phase flows. *Computers and Fluids* **27**, 663–680.
- TAMAKI, N., SHIMIZU, M. & HIROYASU, H. 2001 Enhancement of the atomization of a liquid jet by cavitation in a nozzle hole. *Atomization and Sprays* **11** (2), 125.
- TAMAKI, N., SHIMIZU, M., NISHIDA, K. & HIROYASU, H. 1998 Effects of cavitation and internal flow on atomization of a liquid jet. *Atomization and Sprays* **8** (2), 179–197.
- TRYGGVASON, G., BUNNER, B., A. ESMAEELI, D. JURIC, AL-RAWAHI, N., TAUBER, W., HAN, J., NAS, S. & JAN, Y. J. 2001 A front-tracking method for the computations of multiphase flow. *Journal of Computational Physics* **169** (2), 708–759.
- XU, C., HEISTER, S. D. & BLAISDELL, G. A. 2004 Simulation of cavitated flow in orifices fed by a manifold. *Atomization and Sprays* **14** (1), 37–52.