Two-Dimensional Viscous Aperture Flow: Navier-Stokes and Viscous-Potential-Flow Solutions

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 7 February 2007)

The flow of a planar liquid jet coming 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 liquid jet with gaseous environment. The flow for different Reynolds numbers, Weber numbers, liquid-to-gas density ratios and viscosity ratios 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 has been 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.

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 cavitation 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.

In a different experiment, Otendal et al. (2005) studied the break up of high-speed liquid jet in vacuum, where the pressure is lower than the vapor pressure. By an appropriate design of the nozzle, they avoided the cavitation-induced instabilities inside the nozzle. But decreasing the air pressure below the vapor pressure, they observed a bursting phenomena due to cavitation in the free jet.

Bunnell et al. (1999) studied the unsteady cavitating flow in a slot and found that partially cavitated slots show a periodic oscillation with Strouhal number near unity based on orifice length and Bernoulli velocity.

Tafreshi & Pourdeyhimi (2004) performed a numerical simulation on cavitation and hydraulic flip inside hydroentangling nozzles. They showed under certain conditions cavitation extends to the nozzle outlet and results in hydraulic flip. When hydraulic flip occurs, cavitation vanishes due to the fact that downstream air moves upward into the nozzle.
This leads into the elongation of the jet breakup length. Ahn et al. (2006) experimentally studied the effects of cavitation and hydraulic flip, on the breakup of the liquid jet injected perpendicularly into subsonic crossflow. They showed that cavitation results in shortening the liquid column breakup length. They observed smaller breakup length in the hydraulic flip due to the fact that jet diameter was smaller than the orifice diameter. 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 increasing in the flow 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.

In the traditional criterion of cavitation, cavitation occurs when the pressure drops below the breaking strength of liquid which in the ideal case is the vapor pressure at local temperature. 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.

Most of the calculation done on the subject of high-pressure-nozzle cavitation have used the traditional criterion. The purpose of this paper is to use the new criterion to study the cavitation in liquid atomizers.

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. Governing equations for an unsteady, incompressible viscous flow are the Navier-Stokes equations:

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

\[\mathbf{D} = \frac{1}{2} \left[ (\nabla \mathbf{u}) + (\nabla \mathbf{u})^T \right]\]

\[\nabla \cdot \mathbf{u} = 0\]

where \(\mathbf{u}\) is the velocity, and \(\rho\) and \(\mu\) are the density and viscosity of the fluids, 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 \(\sigma\) is the surface tension coefficient, \(\kappa\) is the curvature of interface, \(\delta\) 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 density ratio of gas to liquid, viscosity ratio and nondimensional parameters: Reynolds number (Re) and
Weber number (We) which are defined as follows:

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

Here \( L \) is the half width of the aperture and \( U \) is the Bernoulli velocity of jet:

\[ U = \sqrt{\frac{2(p_u - p_d)}{\rho_{\text{liq}}}} \] (2.5)

where \( p_u \) and \( p_d \) are the upstream and downstream pressures respectively.

Finding the velocities and pressure field, the stress tensor is calculated using:

\[ T = \mu \left[ (\nabla \mathbf{u}) + (\nabla \mathbf{u})^T \right] - p \mathbf{I} \] (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

\[ T = \begin{bmatrix} T'_{11} & T'_{12} & 0 \\ T'_{21} & T'_{22} & 0 \\ 0 & 0 & 0 \end{bmatrix} \] (2.7)

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

\[ T_{11} = \frac{T'_{11} + T'_{22}}{2} \pm \sqrt{\left( \frac{T'_{11} - T'_{22}}{2} \right)^2 + T'_{12}^2} \] (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 total-stress criterion, the cavitation occurs where:

\[ T_{11} > -p_c \]  

(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:

\[ K = \frac{p_u - p_d}{p_d - p_c} \]  

(2.10)

2.2. Interface tracking and level set formulation

Popinet & Zaleski (1999) did an accurate balance of surface tension forces on a finite volume method by explicit racking of the interface. The method was applied only for two-dimensional calculations because of the geometrical complexity appearing in three-dimensional calculation. 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 \( \theta \), 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:

\[ \frac{\partial \theta}{\partial t} + \mathbf{u} \cdot \nabla \theta = 0 \]  

(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 \( \theta_0(x) \) 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)):

\[ \frac{\partial \theta'}{\partial \tau} = \text{sign}(\theta_0)(1 - |\nabla \theta'|) \]  

(2.12)

with initial conditions:

\[ \theta'(x, 0) = \theta_0(x) \]

where

\[ \text{sign}(\theta) = \begin{cases} 
-1 & \text{if } \theta < 0 \\
0 & \text{if } \theta = 0 \\
1 & \text{if } \theta > 0 
\end{cases} \]  

(2.13)

and \( \tau \) 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 \( \theta' \) has the same zero level set as \( \theta_0 \).
2-D Viscous Aperture Flow

Now using the level set definition, the fluid properties can be defined as:
\[
\rho = \rho_{liq} + (\rho_{gas} - \rho_{liq})H_{\epsilon}(\theta)
\]
\[
\mu = \mu_{liq} + (\mu_{gas} - \mu_{liq})H_{\epsilon}(\theta)
\]
where \(H_{\epsilon}\) is a Heaviside function that has a smooth jump:
\[
H_{\epsilon}(\theta) = \begin{cases} 
0 & \theta < -\epsilon \\
(\theta + \epsilon)/(2\epsilon) + \sin(\pi\theta/\epsilon)/(2\pi) & |\theta| \leq \epsilon \\
1 & \theta > \epsilon
\end{cases}
\]
where \(\epsilon\) represents the thickness of the interface and has the value of \(1.5h\) where \(h\) is the cell size. This Heaviside function corresponds to a delta function that can be used to evaluate the force caused by surface tension:
\[
\delta_{\epsilon} = \begin{cases} 
[1 + \cos(\pi\theta/\epsilon)]/(2\epsilon) & |\theta| \leq \epsilon \\
0 & \text{otherwise}
\end{cases}
\]

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.

In the 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 potential flow solution of the incompressible aperture flow problem has been solved for a long time. The complex potential \(f(z)\) for this flow is given implicitly by Currie (1974) (p.129)
\[
f(z) = \phi + i\psi = -\frac{2C_{c}LU}{\pi} \ln \left\{ \cosh \left( U \frac{dz}{df} \right) \right\} - iC_{c}LU
\]
where \(l\) is half width of the aperture and \(C_{c}\) is the coefficient of contraction. Funada et al. (2006) has analyzed the potential flow solution of the aperture flow. The velocity field can be derived from the potential function as follows:
\[
u = \frac{1}{2} \left( \frac{df}{dz} + \frac{dT}{dz} + \frac{1}{2} \right), \quad v = \frac{i}{2} \left( \frac{df}{dz} - \frac{dT}{dz} \right),
\]
and from there the rate of strain tensor can be calculated:
\[
2D = \begin{bmatrix} 
\frac{d^{2}f}{dz^{2}} + \frac{d^{2}T}{dz^{2}} & i \left( \frac{d^{2}f}{dz^{2}} - \frac{d^{2}T}{dz^{2}} \right) \\
i \left( \frac{d^{2}f}{dz^{2}} - \frac{d^{2}T}{dz^{2}} \right) & - \left( \frac{d^{2}f}{dz^{2}} + \frac{d^{2}T}{dz^{2}} \right)
\end{bmatrix}
\]
To calculate the maximum tension, the principal stresses should be found. The diagonalized rate of strain tensor is
\[
2\mathbf{D} = \begin{bmatrix} \lambda & 0 \\ 0 & -\lambda \end{bmatrix}, \quad \lambda = 2 \left| \frac{d^2 f}{dz^2} \right| \tag{2.23}
\]
Therefore, the maximum tension \(T_{11}\) is given by
\[
T_{11} = -p + \mu \lambda = -p_u + \frac{\rho}{2}(u^2 + v^2) + \mu \lambda \tag{2.24}
\]

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 nonuniform Cartesian grid with 77924 nodes and 77361 elements is employed. The following boundary conditions are applied: \(\Gamma_1\) is the axis of symmetry and the \(v\)-velocity is zero also the normal derivative of all other variables vanish. On the upstream boundary \(\Gamma_2\) the stagnation pressure is specified as the boundary condition. On the downstream boundary \(\Gamma_3\) the static pressure is specified. On the aperture plate \(\Gamma_4\) 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 \(Re = 100\) has shown that discharge coefficient for two cases differ by less than a 0.1%. A momentum integral calculation is conducted to prove the accuracy of the solution. The momentum flux going into and coming out of an arbitrary control volume is calculated and the difference is normalized by the momentum influx. This shows the relative difference is less than 0.1%.

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 \(\rho\)-ratio and \(\mu\)-ratio of \(10^{-4}\) 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 2 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 3.

Figure 4(a) shows the thickness of jet at a distance of 5\(L\) downstream of the aperture. As Reynolds number decreases the jet thickness increases. This could 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 (1961). The discharge coefficient of aperture is plotted in figure 4(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 5 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 becomes different. Figure 6 shows the viscous stress in the flow and compares it with the viscous potential flow case.

It is important to note that the Reynolds number in these calculations is based on the Bernoulli’s velocity of jet, which is larger than the average velocity of jet, specially for low Reynolds numbers. For example, for flow with $Re=1$ the jet velocity is about 20% of the Bernoulli’s velocity and this causes the strain rates and therefore the stresses to be scaled down with the same ratio. This has a significant affect on having smaller regions of high stresses in the N-S solution as seen in figure 6(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 for which the cavitation occurs are identified (figure 7). The size of these areas increase by increasing the cavitation number and by decreasing the Reynolds number.

Figure 8 shows how the area of the region vulnerable to cavitation will increase as the cavitation number increases. For a specific value of $K$, cavitating area is larger for lower Reynolds number because the viscous stress is stronger. This agrees with the statement by Padrino et al. (2007) about the increase in risk of cavitation for more viscous fluids.

Another important point about the figure 8 is that, for the larger Reynolds numbers, the difference between cavitating area predicted by N-S solution and VPF solution 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.

### 3.2. Effects of Weber Number

The flow for Weber numbers of 10, 100, 1000 and infinity are calculated and the free streamlines are shown in figure 9. Flow with Weber number of 1000 is very close to the flow with no surface tension, or infinity Weber number. The free stream for these to 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 for Reynolds numbers of 100 and 1000 in figure 10. Comparing these plots with figure 7, 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 for the Reynolds number of 100 since the boundary layer is larger, the potential regions of 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 potential flow solution. It has been proved that constant
Figure 6: Contours of $T_{11}/\frac{1}{2}\rho_{\text{liq}} U^2$ for Navier-Stokes solution (solid lines) compared to viscous potential flow solution (dashed lines) for $\rho$-ratio=$10^{-4}$ and $\mu$-ratio=$10^{-4}$ (a) Re=1000, (b) Re=100, (c) Re=10, (d) Re=1.

speed condition on the free surface of a potential flow leads to zero normal viscous stress on the free surface, hence satisfies the boundary condition of viscous flow 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/Contract No. W911NF-06-1-0225, with Dr. Kevin Mc Nesby and Dr. Ralph Anthenien having served sequentially as program managers.
Figure 7: The cavitation threshold curves on which $T_{11} + p_c = 0$ in different flows with
$K = 1, 2, 5, 10, 50, \rho$-ratio=10$^{-4}$ and $\mu$-ratio=10$^{-4}$ (a) Re=1000, (b) Re=100, (c) Re=10,
(d) Re=1

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, $\phi$, and stream function, $\psi$, of the potential flow as
the orthogonal curvilinear coordinates. The velocity field in this coordinates has a simple
form:

$$
\begin{align*}
\begin{cases}
  x_1 = \phi \\
  x_2 = \psi
\end{cases}
\quad
\begin{cases}
  u_1 = q \\
  u_2 = 0
\end{cases}
\end{align*}
$$

(A 1)

where the velocity in complex domain can be written as:

$$
u - iv = qe^{-i\theta}$$

(A 2)
Figure 8: Area of cavitating region normalized by $L^2$.

Figure 9: The free streamline for flows with different Weber numbers, $\rho$-ratio=10$^{-4}$ and $\mu$-ratio=10$^{-4}$ (a) Re=1000, (b) Re=100

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

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

Calculating the stresses:

$$T_{11} = -p + 2\mu \left[ \frac{1}{h_1} \frac{\partial u_1}{\partial x_1} + \frac{u_2}{h_1 h_2} \frac{\partial h_1}{\partial x_2} + \frac{u_3}{h_1 h_3} \frac{\partial h_1}{\partial x_3} \right] = -p + 2\mu q \frac{\partial q}{\partial \phi}$$  \hspace{1cm} (A 4)
Using the Bernoulli equation for viscous potential flow:

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

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

\[ T_{22} = -p_u + \frac{1}{2} \rho q^2 - 2\mu \frac{dq}{d\phi} \quad (A 8) \]

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

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

Applying the boundary condition:

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

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:

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

Using the velocity field:

\[
\begin{align*}
u &= q \cos \theta \\
v &= q \sin \theta
\end{align*} \quad (A\ 13)
\]

we get:

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

which results in:

\[
\frac{\partial q}{\partial \psi} = q \frac{\partial \theta}{\partial \phi} \quad (A\ 15)
\]

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

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

where \(\kappa\) 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


