The 8th Iranian Aerospace Society Conference
Feb. 17-19/2009, Malek-e-Ashtar University of Technology

IAS2009-AE954

Parametric study on the stability of a liquid jet using VOF method

Ali.Nematbakhsh[1], Ebrahim.Shirani[2], Ahmad Reza Pishevar[3], Ahmad Sedaghat [4]
1,2,3,4- Isfahan University of Technology

Abstract
The object of this study is to investigate the behavior of liquid jet which is discharged in to air. This problem has great importance in aerospace engineering especially in lubrication of the airplane engines such as bearing box lubrication. The main advantage of this study is consideration of the second phase of the flow which is air in this study. The problem is numerically simulated by using volume of fluid (VOF) method which is based on conservation of each fluid in the domain. Surface tension effect is also considered by including surface tension as a volumetric force in Navier-Stokes equations. Because our numerical simulation is unsteady, Navier-Stokes equation is solved by explicit projection method. Both Navier-Stokes equation and interface tracking equation are solved in each time step. The simulation is performed for different inlet velocities, also the effect of jet diameter is studied. Critical velocities are found for different inlet diameters which beyond these values the effect of gravity is negligible. Also it is found that by reducing the diameter of the liquid jet this critical velocity also reduced respectively. In order to verify the code used for the numerical simulation, dam break problem is simulated and numerical results are compared with available experimental data. Comparison of the numerical results with available experimental data reveals the accuracy of the method.
Keywords: *instability-liquid jet-VOF-surface tension-multiphase*

Introduction

Deformation of a liquid jet in air is a phenomenon which has great importance in aerospace engineering especially in lubrication of the airplane engines such as bearing box lubrication. Therefore accurate prediction of stability and deformation of the liquid jet and using the suitable flow rate for lubrication is of great importance. Studying the deformation of liquid jets is not only useful in the field of aerospace engineering but also has a vast application in other industries such as metal coating and painting industries.

The preliminary study on the stability of the liquid jets was done by Rayleigh who studied the stability of unlimited jet, neglecting the effects of the viscosity. He obtained an exact solution, assuming the instability in the form of exponential function. Recently many numerical methods are used for modeling the instability of liquid jets. Richard et al. [1] modeled the deformation of liquid jet by using VOF method. Ashgriz and Mashayek [2] used finite element method for solving a two-dimensional jet problem. In both of the above mentioned references the influence of the second phase which is air is neglected. Yu et al.[3] used the level set method for modeling flow of a droplet jet. Their model accurately simulates the jet flow but their model was only applicable to flows which are separated in to small droplets . Store and Behina [4] modeled falling of the liquid jet under the influence of gravity. They also performed an experimental study and compared their numerical results with experimental data. Their model was fairly in good agreement with experimental results but they didn't present relation between the parameters governing the deformation and instability of the liquid jet. Yeh [5] studied the deformation of liquid jet with the assistant of VOF method. they used a nozzle for discharging the liquid flow in to air but he study the problem as a free surface flows and do not consider the second phase of the flow.

The purpose of this work is to study effect of inlet velocity and jet diameter on the deformation and stability of liquid jets and find critical velocities for different diameters which beyond these value the effect of gravity is negligible. Our numerical model is validated by experimental results of dam-break problem.

Governing equations and numerical algorithm

In order to track the position of the liquid jet we need to find the position of the interface between air and the liquid jet. For tracking this interface we are to solve additional equation other than momentum and mass conservation equations. In this simulation it is assumed that the fluids are immiscible and the temperature is constant. By this assumption the procedure of interface tracking becomes much easier. This additional equation is obtained based on

[1] Graduate student, Nematbakhsh@me.iut.ac.ir
[2] Full professor, Tel.(+983113915205), e.shirani@cc.iut.ac.ir
[3] Associate professor, Apishe@cc.iut.ac.ir
[4] Assistnat professor, Sedaghat@cc.iut.ac.ir

volume fraction conservation of each fluid in the domain at it is known as volume of fluid (VOF) method. For implementing this fact in to numerical simulation we assign "C" number to each cell in the numerical simulation. This value can be one for the cells which contain first liquid, zero for the cells of second liquid and a number between one and zero for the cells which are not completely fill with one fluid. The values of these cells are calculated by the volume fraction of the first liquid in cell divided by the total volume of the cell. Because of the conversation for "C" value in numerical domain, we can write:

$$\frac{DC}{Dt} = 0 \tag{1}$$

where $\frac{D}{Dt}$ represent and complete derivative of the "C" value. Equation (1) imply a lagrangian view for interface tracking. If we stated this equation in eulerian view, it can be stated as:

$$\frac{\partial C}{\partial t} + U_i \frac{\partial C}{\partial X_i} = 0 \tag{2}$$

where "C" is the volume fraction of liquid in a computational domain, U_i is the velocity in i[th] direction, X_i is space coordinate in i[th] direction and t is time. Beside this equation in order to find the values of the velocities for equation (2), Navier-Stokes equation in combined with mass conservation equation for incompressible flows should be solved. Because surface tension has great influence on the numerical results we add surface tension as volumetric force to Navier-Stokes equation. Mass conservation and Navier-Stokes equations can be written in the form of:

$$\frac{\partial U_i}{\partial X_i} = 0 \tag{3}$$

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_i U_j}{\partial X_j} = -\frac{\partial p}{\partial X_i} + F^{st}.\hat{i} + \mu \frac{\partial^2 U_i}{\partial X_j^2} + \rho g_i \tag{4}$$

In these equations ρ and μ are the density and dynamic viscosity in each cell. p represents the pressure, g is the gravity and finally F^{st} introduced volumetric surface tension force.
Navier-Stokes equation accompanied with volume fraction of fluid conservation equation should be solved in all of the cells in the numerical domain. In the cells which only fluid one or fluid two is present, the solution of Navier-Stokes is straight forward, because the properties such as ρ , μ are known exactly, but for the elements which a fraction of the cell contain fluid one and the rest contain fluid two, we should place a value between the values of properties fluid one and two. A good estimate for viscosity and density in these cells are:

$$\rho = \rho_2 + C(\rho_1 - \rho_2) \tag{5}$$

$$\mu = \mu_2 + C(\mu_1 - \mu_2) \tag{6}$$

Where μ_1, μ_2 are dynamic viscosity of fluid one and two and ρ_1, ρ_2 are density of fluid one and two respectively.
The only thing that is remained for solving Navier-Stokes equation is volumetric form of the surface tension (F^{st}). Surface tension is a discontinuous force which is applied only to the place where interface is present and it is equal to:

$$\vec{F}^{st} = \sigma \kappa \vec{N} \tag{7}$$

In this equation σ is a coefficient of surface tension, κ is the curvature of the interface and $\vec{N}$ is normal to the interface. But as we want to solve the problem numerically and position of the interface is not always exactly at center of the cells, therefore surface tension can't be implemented in the exact place.
A numerical model which is used in this paper is based on Brackbill et al[6] model which approximate surface tension force with the assistant of numerical value "C" and it is in the form of:

$$F^{st} = \sigma \left(-\nabla . \frac{\nabla C}{|\nabla C|} \right) . \frac{\nabla C}{[C]} \tag{8}$$

In this equation $\nabla.$ is divergence operator, "∇" is gradient operator and $[C]$ is the difference between "C" values assigned two fluid one and two.

By considering surface tension force as a volumetric force, the surface tension force is not implemented in its exact location, therefore spurious currents flows will produced in the vicinity of the interface. In order to reduce these spurious currents we can smooth the value of "C" in the domain by a smoothing function. The smoothing function which is used in this study is weighted smooth function and it is in the form of:

$$\begin{aligned} \tilde{C}(i,j) = {} & .25C(i,j) + \\ & 0.125[C(i+1,j) + C(i,j+1)] + \\ & 0.125[C(i-1,j) + C(i,j-1)] + \\ & 0.0625[C(i+1,j+1) + C(i-1,j+1)] + \\ & 0.0625[C(i+1,j-1) + C(i-1,j-1)] \end{aligned} \tag{9}$$

Note that smoothing function should be in the form that conserves the parameter in the numerical domain.

By using this smoothing function the values of spurious currents reduced considerably. An important point that should be considered is that by smoothing the value of "C" we also reduce the accuracy of the

numerical simulation. This happens because the nature of surface tension is discontinuous, so when the surface tension is replaced by volumetric force it changes to a continuous force which acts in a thin band near the interface and by smoothing, this thin band become wider. Considering this fact the smoothing should used when the spurious currents are comparable to physical velocities.

By modeling the value of surface tension, all the parameters are known in Navier-Stokes equations and also volume fraction conservation equation can be solved in each time step.

After each time step by knowing the values of "C" in each cell, the interface of two fluids should be constructed. To construct the interface at each cell a second order function model known as "PLIC" method is used. In this method each cell is divided in to two separate parts by a second order straight line. The inclination of the line is calculated by knowing the normal of the line. The normal of the line is in the direction of the "C" gradient. "C" gradient shows maximum change of the C values and therefore it is normal to the interface (straight line). Second parameter which is needed to construct the straight line in the cell is the value of the C. By knowing a constant value C and the inclination of the line, the interface is constructed in each cell.

In order to solve Navier –Stokes equations and also volume fraction conservation equation , a FORTRAN code called "SURFER" written by by Lafaurie et.al[6] and improved by Gueyffier et.al[7] and also by Shirani et al.[8] is used. Details of numerical methods are not mentioned here, but can be found in the reference [8]. Uniform rectangular cells are used in the entire numerical domain. Staggered algorithm is used to discrete Navier-stokes and volume fraction conservation equation. A multigrid technique was employed for the solution of the pressure Poisson equation to enforce the divergence-free velocity.

Verification of the numerical method

In order to validate the code and the procedure of solution used here, dam break problem is solved and numerical results are compared with experimental data given by Martin and Moyce [9]. This experimental results are so useful because it can be considered as a two-dimensional problem with a fairly good approximation.

Dam break problem can be stated as follows. A rectangular motionless water is restricted between two vertical walls. In Fig. 1 the right wall of the shaded area is removed at t=0. The height of the restricted water is 0.1143 meter and the width is 0.0254 meter. The water at t=0 is in the hydrostatic equilibrium. The right wall is removed at t=0 therefore a flow is begun to move by the influence of the gravity the properties of the fluid and the air are stated in the Table (1).

Table 1 – properties used for modeling dam break problem

Properties	Values
Water density	$998\,kg/m^3$
Water viscosity	$0{,}99\times10^{-3}\,kg/ms$
Air density	$1.19\,kg/m^3$
Air viscosity	$1.84\times10^{-5}\,kg/ms$
Surface tension coefficient	$0.0728\,N/m$
Gravity	$9.81\,m/s^2$

A rectangular numerical domain is used for simulation. A solid boundary condition is used for four side of the domain. In order to neglect the influence of the boundary the top and also right boundary of the numerical domain are far enough from the flow of water. The size of the numerical domain is 0.2032 meter in the "x" direction and 0.2286 meter in the "y" direction in Cartesian coordinate. A 64×128 rectangular elements is used for numerical modeling and VOF method is used for tracking the interface. Fig. 1 shows the initial state and boundary condition of the dam break problem

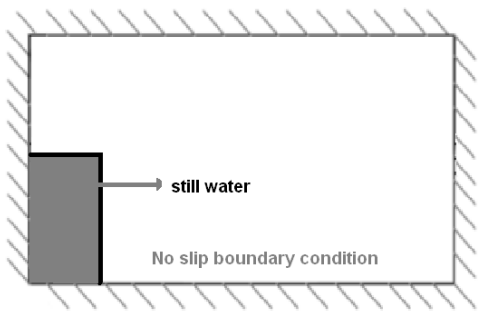


Fig. 1 Initial state and boundary conditions of a dam break problem

Fig. 2 shows the comparison of numerical results with experimental data stated by Martin and Moyce[7].

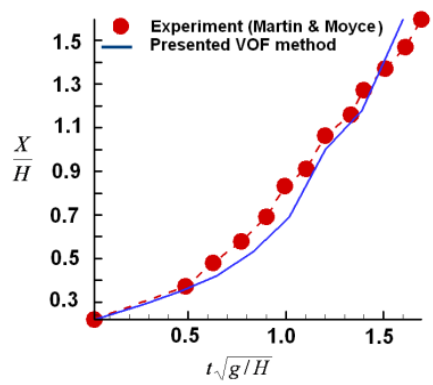


Fig. 2 Comparison of experimental and numerical results for dam-break problem

In this figure X is the distance of leading front from the left boundary of the box, H is the height of the hydrostatic water and **t** is time and **g** is the gravity. Comparison of numerical results based on VOF

method with experimental results reveals the accuracy of computational model and numerical simulation used in this paper.

Simulation of liquid jet

Simulation of liquid jet discharged in to air is studied with the assistant of VOF method. Brackbill model for surface tension which is stated in equation (8) is used for numerical modeling of surface tension. A rectangular domain is chosen. Solid boundary condition is applied to the top and bottom of the numerical domain. The right and left boundaries are divided into two parts. For the left side, the middle part is the place where liquid jet is discharged, therefore inlet boundary condition is used and the rest soild boundary condition is used. Same boundary is implemented to the right side except that instead of inlet boundary condition, outlet boundary condition is used. The outlet boundary condition is used in order to satisfy the mass conservation in the numerical domain.

Fig. 3 shows the boundary conditions used for simulation of liquid jet discharged in to air.

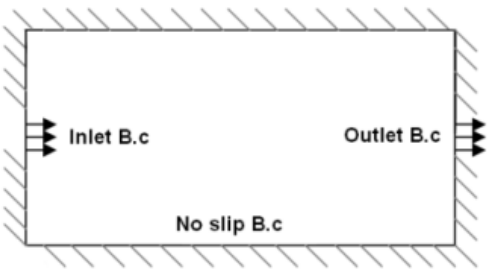
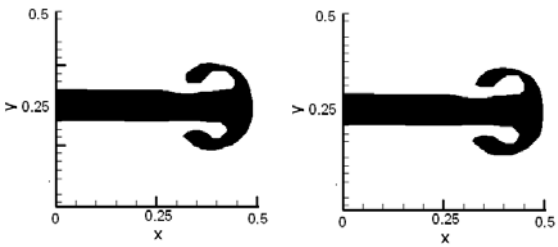
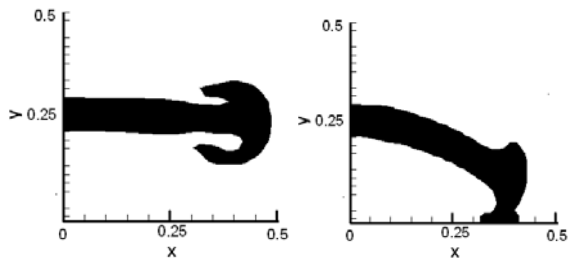


Fig. 3 Boundary conditions applied for studying the instability of jets

The height of numerical domain is 3.2d and the width is equal to 6.4d, where "d" represents the diameter of the liquid jets. The simulation is performed for different inlet diameters and also for different inlet velocities. Fig. 4 shows the numerical simulation of liquid jet discharging in to air for different inlet velocities. The properties of liquid and air are as same as dam break problem. In these four figures the diameter of the liquid jet is equal to 14 mm. In Fig. 4 X and Y axis (x,y divided by length of the numerical domain L=6.4d) represent none dimensional length.



(a)…… .(b)



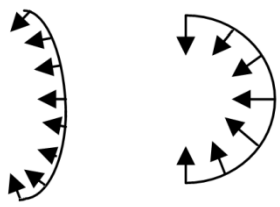
(c)…… .(d)

Fig. 4 numerical simulation of liquid jet discharging in to air for different inlet velocities, (a) v=6 m/s (b) v=4 m/s (c)v=3 m/s, (d) v=1 m/s

As it can be seen from Fig. 4 the shape of the jet for v=6 m/s and also for v=4 m/s are approximately the same. Numerical simulation shows that for these inlet velocities the effect of gravity is negligible. If we simulate this problem without gravity force for such inlet velocities the results make no major difference. The numerical results for inlet velocity equal to 3 m/s (Fig. 4(c)) shows that at this velocity the leading front of the liquid jet make a decline slightly and finally for the velocity equal to 1m/s (Fig.4(a)) the effect of gravity is so considerable and the liquid jet decline completely.

From Fig.4 we notice that a convex shape is created in the front of the liquid jet. The reason which results in creation of this shape for the leading edge of the liquid jet is surface tension force. As it is mentioned in the section "Governing equation " surface tension is a discontinuous force which is applied only at the interface. The direction of this force according to equation (1) is in the normal direction of the surface. If we assumed a straight line with a small curvature as the interface of two fluid and assumed that the surface tension force is the only force that is applied to the interface and also interface do not shows any resistance to the deformation, therefore by applying this force the shape of the line change to circle. The same thing is happened to the leading front of the liquid jet. Surface tension force act on the interface of the liquid jet and because the direction is normal to the interface it tends to change the interface shape in to a circle.

Because of the behavior of surface tension, free surface of the fluids with high surface tension in restricted environment such as Mercury are in convex shape. Fig.5 shows the surface tension force which is applied to interface. If the surface tension force is high enough it can change the shape of the interface form Fig.5 (a) to Fig.5 (b).



(a) (b)

Fig. 5 surface tension is applied to the normal direction and tends to curve the interface

In Fig. 5(a) the shape of the interface is approximately flat, but under the effect of gravity force it changes to Fig. 5(b) which is nearly part of a circle.

Effect of liquid jet diameter and inlet velocity

In order to study the effect of liquid jet diameter on the shape and stability of the liquid jet, liquid jets with different diameters and the same inlet velocities equal to 4.5m/s are simulated.

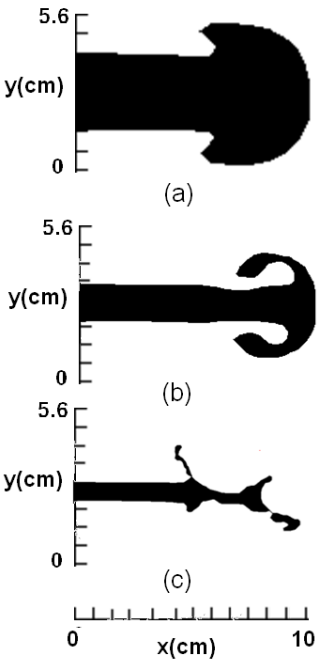


Fig. 6 Effect of liquid jet diameter on the stability of liquif jet, (a) d=28 mm, (b) d=14 mm and (c) d=7 mm

The shape of the liquid jet in distance of 0.09 meter is shown in Fig. 6. As it can be seen from the Fig. 6 by increasing the jet diameter the jet become more stable at the same distance. The liquid jet with 7mm diameter before reaching distance equal to 0.09cm becomes unstable and the jet with 28 mm diameter at the same distance is stable.

In order to study the effect of inlet velocities for different liquid jet diameters, we simulate the liquid jet motion for different velocities and also for different diameter. For each diameter there is a certain velocity at which the effect of gravity is negligible and the jet moves on a horizontal straight line and a also the shape of the leading edge of the liquid jet is symmetric respect to horizontal axis.

Fig. 6 represent the motion of liquid jet with diameter equal to 14 mm (d=14 mm) and inlet velocity of 1m/s. as it can be seen the liquid jet deviate from the horizontal line. This deviation is occurred because of gravity force. In order to calculate this deviation numerically, an angle θ is introduced. This angle is shown in fig. 6. In this figure X and Y are dimesionless length and are equal to x divided by the length of the domain which is 6.4d (d is the diameter of the liquid jet).

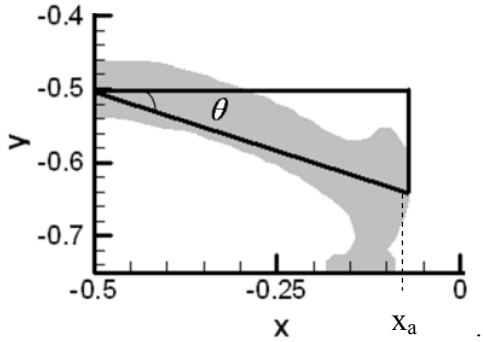


Fig. 7 liquid jet profile for inlet velocity 1(m/s) and diameter 14 mm. θ is introduced for calculating the deviation of liquid jet

in order to calculate θ parameter x_a is introduced. x_a as shown in fig. 7 is equal to maximum distance that the leading edge of the liquid jet go forward and if Xa>0.5L then it is replaced by 0.5L. the other line of the triangular for calculating θ is obtain by drawing a straight line at middle of liquid jet diameter from the placed discharge to the middle of the leading front of the liquid jet. By knowing these two lines the angle θ is calculated easily.

Fig. 7 shows the variation of deviation parameter of the liquid jet (θ) for different inlet velocities. The velocities are change from 0.1 m/s to 7 m/s and in each simulation parameter (θ) is calculated. The simulation is performed for three different diameters (d=7 mm, d=14 mmm, d=28 mm) in order to study the effect of liquid jet diameter on the stability of liquid jet.

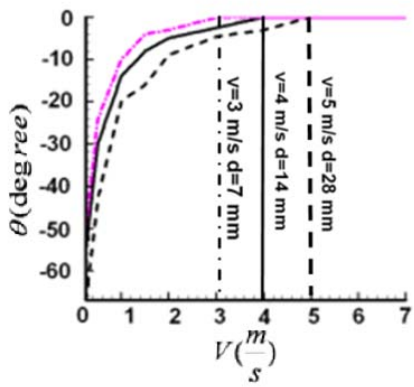


Fig. 8 variation of θ respect to different inlet velocities and critical velocities (dashed line jet diameter: 28mm, solid line jet diameter: 14mm, dash dot line, jet diameter: 7mm)

From Fig. 7 it can be understood that for different jet diameters, the deviation of the liquid jet (θ) at low inlet velocities changes rapidly but as the velocity increase the rate of change decrease and eventually become zero and the effect of the gravity become negligible.

The velocity at which deviation of the liquid jet is equal to zero, changes for different liquid jet diameters. Our numerical simulation shows that, increase of jet diameter results in the increase of critical velocity and vice versa.

Fig. 7 also shows that the deformation of the jet regarding to inlet velocity change nonlinearly for different jet diameters.

Conclusion

In this paper by considering the effect of surface tension and the second phase in the numerical domain, parametric study is done for stability and deformation of liquid jet. The effect of inlet velocity and the jet diameter on the instability of liquid jets is investigated. Critical velocities are found for different jet diameters which beyond these values the effect of gravity is negligible and the liquid jet moves in horizontal straight line up to certain distance. By modeling the dam-break problem and comparing the numerical results with experimental data the accuracy of the numerical model is studied.

References

1- Richards, J.R., Lenho, A.M., Beris, A.N , Dynamic breakup ofliqui d-liquid jets. *Physics of Fluid*, Vol. 6, No. 8, 1994, PP. 2640-2654

2- Ashgriz, N., Mashayek, F., Temporal analysis of capillary jet breakup, *Journal of Fluid Mechanic,* Vol. 291, 1995, PP. 163-190

3- Jiun-Der Yu a, Shinri Sakai b, James Sethian, A coupled quadrilateral grid level set projection method applied to ink jet simulation, *Journal of Computational Physics,* Vol. 206, 2005, PP. 227–251

4- Storr G.J. a, Behnia M., Comparisons between experiment and numerical simulation using a free surface technique of free-falling liquid jets, *Experimental Thermal and Fluid Science,* Vol.22, 2000, PP. 79-91

5-.Yeh Ch.l, Numerical investigation of liquid jet emanating from plain orifice atomizers with chamfered or rounded orifice inlets, *JSME International Journal,* Vol. 47, 2004, PP 37-47

6- . Brackbill, J.U. Kothe D.B,. Zemach C, A continuum method for modeling surface tension, *Journal of Computational Physics,* Vol. 100, 1992, PP. 335–354

6- Lafaurie B.,. Nardone C, cardovelli.S. Zaleski., RZanetti, , G. Modeling merging and fragmentation in multiphase flows with SURFER, *Journal of Computational Physics,* Vol. 113, 1994, PP. 154–175

7- . Gueyffier, D. Li J, Nadim A, Scardovelli, S. Zaleski. . R, Volume-of-fluid interface tracking with smoothed surface stress methods for three-dimensional flows, *Journal of Computational Physics,* Vol. 152, 1999, PP. 423–456

8- Shirani E, Ashgriz N, Mostaghimi J., Interface pressure calculation based on conservation of momentum for volume tracking methods, *Journal of Computational Physics,* Vol. 203, 2005, PP. 134–147

9- . Martin J.C,. Moyce, W.J, An experimental study of the collapse of liquid columns on a rigid horizontal plane, *Mathematical physic Science,* Vol. 882, 1952, PP. 312-32