19th Annual Conference on Mechanical Engineering-ISME2011
10-12 May, 2011, The University of Birjand, Birjand, Iran

ISME2011-2329

The Impact of a Solid Object on to a Liquid Surface

Iman Mirzaii[1], Mohammad Passandideh-Fard[2]

[1]Graduate Student, Ferdowsi University of Mashhad; imanmirzaii@gmail.com
[2]Associate Professor, Ferdowsi University of Mashhad; mpfard@um.ac.ir

Abstract

In this study, a numerical algorithm is developed for simulating the entry of a solid object into a liquid medium considering the free surface development at the liquid/gas interface. The presented model is that of the fast-fictitious-domain method integrated into the volume-of-fluid (VOF) technique used for tracking the free surface motion. The developed model considers the solid objects as fluids with a high viscosity resulting in a rigid motion of the objects. Attributing a high viscosity to solid objects improves the accuracy of the results compared to that of the recent models in the literature without additional computational time and cost. The model is validated by comparison of the simulation results with those of the experiments available in the literature for the free fall of a circular disk and a sphere. For all cases considered, the results were in good agreement with those of the experiments and other numerical studies.

Keywords: Solid-liquid interaction, Free surface flows, Volume of Fluid (VOF), Fictitious domain method, Implicit viscous term

Introduction

The impact of a solid object on to a liquid surface has been studied experimentally and theoretically for more than a century. However, this phenomenon is not well understood due to complex effects of turbulency, cavity formation behind the solid object and capillary effects at the contact line between the surface of the solid and liquid. The study of solid-liquid interaction has a wide application in many industries such as designing ships, flying boats, seaplanes, etc.

Worthington [1] was one of the first researchers who studied water entry of spherical objects. Since then many studies were performed to improve the knowledge about solid/liquid impact. May [2, 3] studied the cavity formation behind a solid object impacting a liquid surface and discussed the effects of several experimental parameters, such as object surface condition, the atmospheric density, and the velocity and size of the spheres. These parameters can influence the time and location of various events in the life of the cavity formation behind the spherical objects. Laverty [4] determined a critical impact speed before which no splash cavity is formed. He also studied the cone angle formed behind an impacting object. The cone angle was found to be a function of the depth of the solid object and its impact speed.

The classical view of impacts on free surfaces relies solely on solid and liquid inertia. In strong contrast to this largescale hydrodynamic viewpoint, Duez et al. [5] showed that the wettability of the impacting body is a key factor in determining the degree of splashing. Motivated by Duez et al., other studies [6-9] were conducted for water entry of hydrophobic spheres capable of creating a bigger air cavity behind the solid. This, in turn, affected the subsequent object trajectory and the resulting drag force.

There have been some attempts to simulate the solid-liquid impact numerically. Lin [10] introduced a two dimensional and fixed-grid model for this purpose. Do-Quang [11, 12] presented a numerical study on the influence of solid surface wettability on the splash of a solid sphere impacting a liquid free surface. The main drawback of all these studies is that the speed and trajectory of the solid object had to be known as *a priori*. Yang et al. [13] used a sharp interface immersed-boundary formulation and a level-set/ghost–fluid method for solid–liquid and liquid-gas interface treatments, respectively. However, in this study they had to solve additional equation for modeling the solid objects which leads to a time consuming process.

In this study, a numerical method is presented that can simulate the water entry of solid objects. This method is capable of handling unprescribed motion of the solids without solving any additional equation in the computational domain. This method is based on Sharma et al. [14] study. Although the presented method of Sharma et al. is a fast one, it suffers from low accuracy due to a slip condition on the solid-liquid interface. In addition, Sharma et al. merely used their model to simulate single phase fluid in the computational domain. In order to improve the accuracy of Sharma's method, a simple modification without any additional simulation time is suggested in this study. This method integrated into volume of fluid (VOF) technique is then used for tracking the free surface motion of the entry of spheres through water surface. The model introduced in this study can easily be applied to other fluid flow solvers dealing with un-prescribed motion of solid objects.

Numerical Method

Fluid flow:

The governing equations for the fluid flow are the continuity and momentum equations (Newtonian fluid). For discretization of the governing equations the two-step projection method [15, 16] is used. In this method, momentum and continuity equations are approximated in two separate steps as follows:

step1)

$$\frac{\vec{V}^{n+1/2}-\vec{V}^{n}}{dt}=-\nabla\cdot\left(\vec{V}\vec{V}\right)^{n}+\frac{1}{\rho^{n}}\nabla\cdot\mu\left[\left(\nabla\vec{V}^{n}\right)+\left(\nabla\vec{V}^{n}\right)^{T}\right]+\vec{g}^{n}+\frac{1}{\rho^{n}}\vec{F}_{b}^{n} \qquad (1)$$

step2)

$$\frac{\vec{V}^{n+1}-\vec{V}^{n+1/2}}{dt}=-\frac{1}{\rho^{n}}\nabla\cdot p^{n+1} \qquad (2)$$

$$\nabla\cdot\vec{V}^{n+1}=0 \qquad (3)$$

where $\vec{V}$ is the velocity, ρ the density, p the pressure, μ the dynamic viscosity, $\vec{F}_b$ body forces, and $\vec{g}$ the acceleration due to gravity. The nonlinear advection term is written in conservative form. To handle the effect of surface tension, it is considered as a body force using the Continuum Surface Force (CSF) model [17].

In the first step, a velocity field $\vec{V}^{n+1/2}$ is computed from incremental changes in the field $\vec{V}^{n}$ resulting from viscosity, advection, gravitational accelerations, and body forces. In the second step, the velocity field is projected onto a zero-divergence vector field (Eq. (3)) which leads to a single Poisson equation for the pressure evaluation:

$$\nabla \cdot \left[\frac{1}{\rho^{n}} \nabla p^{n+1} \right] = \frac{\nabla \cdot \vec{V}^{n+1/2}}{\delta t} . \tag{4}$$

The projection method presented above is only adequate for solving fluid flow problem with low viscosity. This fact results from an explicit discretization of the governing equations in the first step. This explicit treatment is therefore subject to a linear stability time step constraint [18]. The time step limitation of viscous term can be presented as:

$$\delta t < \delta t_V = \frac{1}{3\nu} \left[\frac{(\delta x)^2 (\delta y)^2}{(\delta x)^2 + (\delta y)^2} \right]_{\min} . \tag{5}$$

As seen from Eq. (5), increasing the viscosity will decrease the time step which in turn will increase the computational time. In this study, the rigid motion of solid objects is to be modeled using a high viscosity for the solid region. As a result, based on Eq. (5), the allowable time step for numerical simulation will decrease dramatically. A remedy for this problem is proposed in this study as follows. The first step of the projection method, Eq. (1), can be divided into two separate steps [19] as:

$$\frac{\vec{V}^{n+1/2} - \vec{V}^{n}}{\delta t} = -\nabla \cdot \left(\vec{V}\vec{V}\right)^{n} + \vec{g}^{n} + \frac{1}{\rho^{n}} \vec{F}_b^{n} \tag{6}$$

$$\frac{\vec{V}^{n+1/2} - \vec{V}^{n+1/2}}{\delta t} = \frac{1}{\rho^{n}} \nabla \cdot \mu \left[\left(\nabla \vec{V}^{n+1/2}\right) + \left(\nabla \vec{V}^{n+1/2}\right)^{T} \right] \tag{7}$$

Eq. (6) is the same equation as Eq. (1) except that the effect of viscous term is not considered in this step. This effect is evaluated in Eq. (7). In this equation the viscous term is discretized in $t^{n+1/2}$. This leads to implicit treatment of the viscous term effects and will allow using large time steps even when we have to deal with high viscosity fluids. To solve the resulted system of equations TDMA solver was used.

The interface is advected using the VOF method by means of a scalar field (*f*), the so-called liquid volume fraction, defined as:

$$f = \begin{cases} 0 & \text{in gas} \\ 0 < \ , < 1 & \text{in liquig} - \text{gas in terface} \\ 1 & \text{in liquid} \end{cases} \tag{8}$$

The discontinuity in *f* is a Lagrangian invariant, propagating according to:

$$\frac{df}{dt} = \frac{\partial f}{\partial t} + \vec{V} \cdot \nabla f = S \tag{9}$$

where *S* stands for the mass transfer term which in this study is zero. Eq. (9) is used to track the location of the interface and is solved according to Youngs' PLIC algorithm [20].

Solid Object:

In this study, the solid objects are considered as fluids with a high viscosity resulting in a rigid motion of the objects. Attributing a high viscosity to solid objects improved the accuracy of the results compared to that of Sharma et al. [14] which may be considered as a base for the presented method. The main inefficiency of the Sharma et al. [14] method is because it suffers from a slip condition on the solid-fluid interface. In the first step of simulations, the governing equations of fluid motion is solved everywhere in the computational domain without any additional equation for object zones. Next, the rigid body motion can be obtained by imposing an additional condition that the total linear and angular momenta in the individual object domain must be conserved in each time step. This method can be summarized in three steps:

1) The object zones in the computational domain are defined using a scalar parameter (φ_s). This parameter is defined in the computational domain as:

$$\varphi_s = \begin{cases} 0 & \text{out of solid} \\ 0 < \ , < 1 & \text{solid boundary} \\ 1 & \text{within solid} \end{cases} \tag{10}$$

2) The fluid equations are solved as discussed above to obtain V^{n+1}. In this step density and viscosity in each cell is defined as:

$$\rho = f\rho_l + (1 - f - \varphi_s)\rho_g + \varphi_s \rho_s \tag{11}$$

$$\mu = f\mu_l + (1 - f - \varphi_s)\mu_g + \varphi_s \mu_s \tag{12}$$

where subscripts *l*, *g* and *s* refer to liquid, gas and solid, respectively. The viscosity of the solid is set by a large magnitude in comparison with that of the liquid. The effect of this high viscosity on the results will be discussed later in this paper.

3) The average translational velocity and angular velocity are obtained in the object zones based on the conservation of momentum in the object zone using following integrals:

$$M_s \vec{V}_s = \int_{Particle zone} \rho \vec{V} \, dA \tag{13}$$

$$I_s \vec{\omega}_s = \int_{Particle zone} \vec{r} \times \rho \vec{V} \, dA. \tag{14}$$

where M_s and I_s are the mass and moment of inertia of the solid, respectively. The velocity in the object zone will be substituted by:

$$\vec{V}_{Particle\ zone} = \vec{V}_s + \vec{\omega}_s \times \vec{r}. \tag{15}$$

Numerical Results

In this section, numerical results of a few sample cases are presented. To validate the developed numerical approach, first a case is considered for which the experimental measurements and other numerical results are available in the literature. This case is the free fall of a rigid circular disk in a container full of liquid. The model is then used to simulate the water entry of solid spheres where an interface is also involved in the calculations.

Sedimentation of a circular disk:

A solid circular disk is released from the rest in a 2D container full of liquid. The geometry, initial conditions and properties of the liquid and the disk are as follows [21, 22]:

- Disk diameter: 2.5 mm
- Disk is released from the rest at an elevation of 27.5mm
- Liquid and solid densities are 10^6 and 1.25×10^6 kg/m^3, respectively.
- Liquid dynamic viscosity is 10 kg/m.s

The computational domain for this case was considered to be: 20 mm ×30 mm. Released from the rest under the effect of gravity, the disk falls down. The disk will be slowed down and reach a constant falling velocity (called terminal velocity) because of the liquid viscosity and drag force exerted on the disk lateral surface. Numerical results for the evolution of the y-coordinate of the translational velocity of the disk are shown in Figure 1. As the figure shows, increasing the viscosity in the object zone will increase the accuracy of the results compared to that of Sharma et al. [14] method in accordance with the available results in the literature presented by Glowinski [21] and Blasco et al. [22]. Glowinski [21] compared his numerical results with many experimental measurements for terminal velocity of a falling disk in a

Newtonian liquid, therefore, his results as displayed in Figure 1 can be considered those of the experiments.

Figure 2 shows the y-direction velocity contour and the streamlines in the computational domain at t=0.28 sec., respectively. At t=0.28 s the circular disk has reached almost to its terminal velocity.

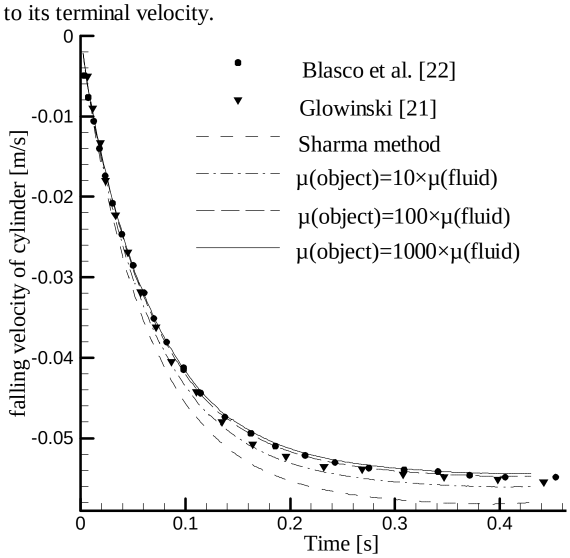

Figure 1.Comparison of vertical velocity of the circular disk with available results in literature, increasing the viscosity in the object zone increases the accuracy of Sharma method

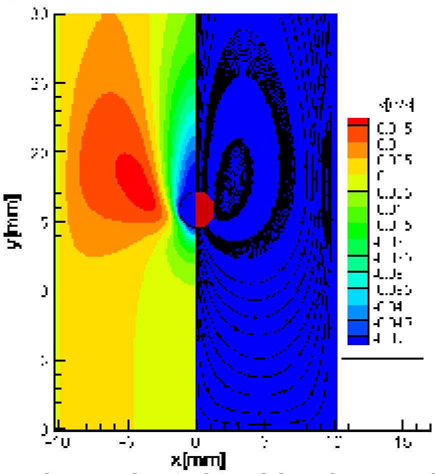

Figure 2. The streamline (right) and the y-direction velocity contour (left) at t=0.28 sec.

Water entry of a sphere:

The presented method in this paper can model the cases in which an interface is involved in the computational domain. To demonstrate this, two different cases of water entry problems are presented. The initial conditions and the properties are as follows:

•Sphere radius: 1.27 cm

• Initial position of sphere center: 1.27 cm above the water surface

•Solid density: 200 and 860 kg/m^3 for Case 1 and Case 2, respectively.

•Impact velocity: 2.40 and 2.17 m/s for Case 1 and Case 2, respectively.

The computational domain for Case 1 was 50 mm×80 mm and for Case 2 was 50 mm×130 mm (radius × height).

The results are compared with the experimental results of Aristoff et al. [9]. Figure 3 and Figure 4 show the 2D numerical results of water distribution after the impact. Particularly, in first steps of the numerical simulations, results are in good agreement with the experimental results.

Conclusion

A fast-fictitious-domain method, integrated into the volume-of-fluid (VOF) technique, was developed for numerical simulation of solid-liquid interaction. The key feature of the developed model is that the solid objects are treated as fluids with a high viscosity and no additional equation is solved for object zone. As a result, in comparison with other available models for numerical modelling of solid-liquid interaction, the presented algorithm is more efficient and needs less CPU time and memory. The developed model was used for the free fall of a circular disk and water entry of a sphere. For all cases considered, the results were in good agreement with those of the experiments and other numerical studies.

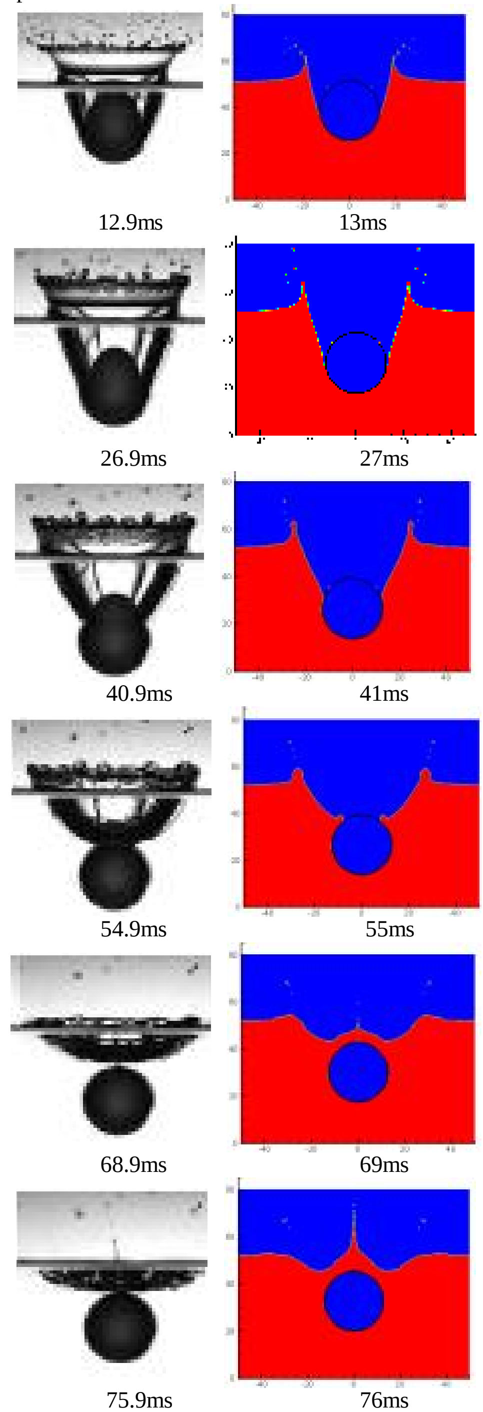

Figure 3. Comparison of the numerical result simulations with experimental results presented by Aristoff et al. [9], Radius: 1.27 (cm), Density: 200 (kg/m3)

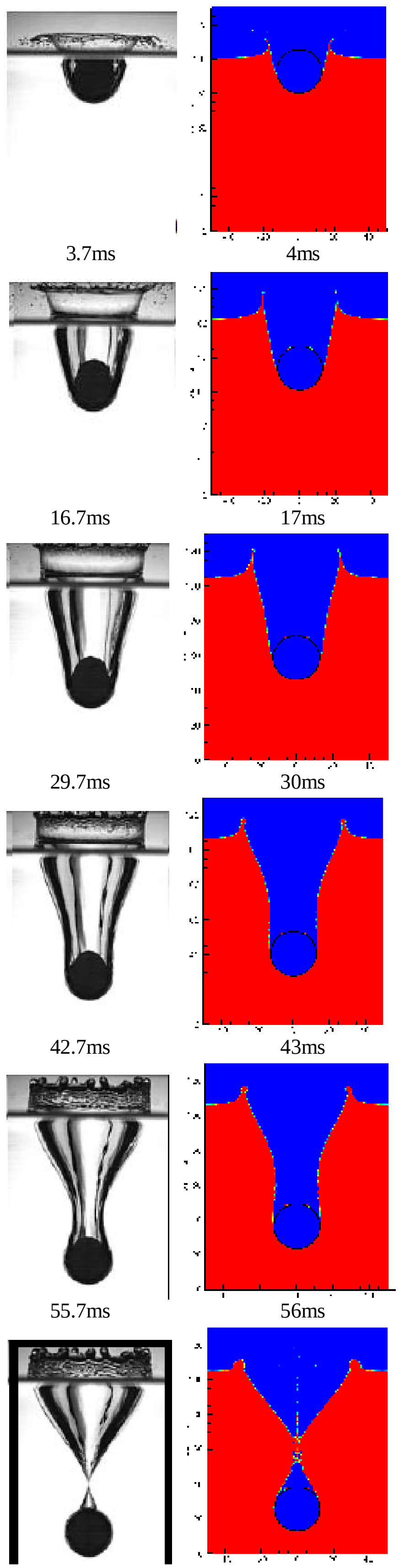

Figure 4. Comparison of the numerical result simulations with experimental results presented by Aristoff et al. [9], Radius: 1.27 (cm), Density: 860 (kg/m3)

References

[1] Worthington, A. M., 1908, A Study of Splashes, Longmans, Green, and Co., London.

[2] May, A., 1951, "Effect of Surface Condition of a Sphere on Its Water Entry Cavity," Journal of Applied Physics, 22(10), p. 4.

[3] May, A., 1952, "Vertical entry of missiles into water," Journal of Applied Physics, 23, pp. 1362-1372.

[4] Laverty, S. M., 2004, "Experimental Hydrodynamics of Spherical Projectiles Impacting On a Free Surface Using High Speed Imaging Techniques," Master of science, Massachusetts institute of technology, Massachusetts.

[5] Duez, C., Ybert, C., Clanet, C., and Bocquet, L., 2007, "Making a splash with water repellency," Nat Phys, 3(3), pp. 180-183.

[6] Lee, D.-G., and Kim, H.-Y., 2007, "Impact of a Superhydrophobic Sphere onto Water," Langmuir, 24(1), pp. 142-145.

[7] Duclaux, V., Caill, F., Duez, C., Ybert, C., Bocquet, L., and Clanet, C., 2007, "Dynamics of transient cavities," Journal of Fluid Mechanics, 591(-1), pp. 1-19.

[8] Aristoff, J. M., and Bush, J. W. M., 2009, "Water entry of small hydrophobic spheres," Journal of Fluid Mechanics, 619(-1), pp. 45-78.

[9] Aristoff, J. M., Truscott, T. T., Techet, A. H., and Bush, J. W. M., 2010, "The water entry of decelerating spheres," Physics of Fluids, 22, p. 8.

[10] Lin, P., 2007, "A fixed-grid model for simulation of a moving body in free surface flows," Computers & Fluids, 36(3), pp. 549-561.

[11] Do-Quang, M., and Amberg, G., 2009, "The splash of a solid sphere impacting on a liquid surface: Numerical simulation of the influence of wetting," Physics of Fluids, 21, p. 13.

[12] Do-Quang, M., and Amberg, G., 2010, "Numerical simulation of the coupling problems of a solid sphere impacting on a liquid free surface," Mathematics and Computers in Simulation, 80(8), pp. 1664-1673.

[13] Yang, J., and Stern, F., 2009, "Sharp interface immersed-boundary/level-set method for wave-body interactions," Journal of Computational Physics, 228(17), pp. 6590-6616.

[14] Sharma, N., and Patankar, N. A., 2005, "A fast computation technique for the direct numerical simulation of rigid particulate flows," Journal of Computational Physics, 205(2), pp. 439-457.

[15] Harlow, F. H., and Welch, J.E., 1965, "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface," Physics of Fluids, 8, p. 2182.

[16] Chorin, A. J., 1968, "Numerical solution of the Navier-Stokes equations " Math. Comp., 22, pp. 745-762.

[17] Brackbill, J. U., Kothe, D. B., and Zemach, C., 1992, "A continuum method for modeling surface tension," Journal of Computational Physics, 100(2), pp. 335-354.

[18] Harlow, F. H., Amsden, A.A., 1971, "Fluid dynamics: A LASL monograph (Mathematical solutions for problems in fluid dynamics) " LA 4700, Los Alamas National Labratory.

[19] Carlson, M., Mucha, P. J., Brooks Van Horn III, R., and Greg, T., "Melting and flowing," Proc. Proceedings of the 2002 ACM SIGGRAPH/Eurographics symposium on Computer animation.

[20] Youngs, D. L., 1984, "An interface tracking method for a 3D Eulerian hydrodynamics code," Technical report 44/92/35, AWRE.

[21] Glowinski, R., 2003, Numerical methods for fluids (Part3), Elsevier Science B.V., Amsterdam, The Netherlands.

[22] Blasco, J., Calzada, M. C., and Marin, M., 2009, "A Fictitious Domain, parallel numerical method for rigid particulate flows," Journal of Computational Physics, 228(20), pp. 7596-7613.