21st International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2018, Karlovy Vary, Czech Republic, 18th – 22nd September, 2018

VOLUME-OF-FLUID METHOD FOR INTERFACE CAPTURING IN FREE-SURFACE FLOW HYDRODYNAMICS IN OPENFOAM®

Edin Berberović
University of Zenica, Polytechnic Faculty
Fakultetska 1, 72000 Zenica
Bosnia and Herzegovina

ABSTRACT
In the present paper the volume-of-fluid (VOF) technique accurate surface capturing in free surface flow hydrodynamics in OpenFOAM® is presented. The presented model is algebraic and utilizes a compressive scheme to effectively shrink the transitional region between the phases. The performance of the model is demonstrated by computing two-dimensional axisymmetric gravity-free droplet shape evolution and drop impact onto a flat wall covered with a film of the same liquid. Results show rather good capabilities of the model for the simulation of free-surface flows.
Keywords: volume-of-fluid, modeling, hydrodynamics, free-surface flow

1. INTRODUCTION
The increasing development of computer power and efficiency have enabled a broader use of numerical methods in investigations of hydrodynamics of flows with free surfaces. Numerical simulations provide a detailed insight into the phenomena and dynamic behavior of such flows. The amount of information which can be obtained is beyond the scope of any experimental or theoretical method. Importance of free surface flows is particularly reflected in spray impact which occurs in many industrial applications, such as ink-jet printing, paint spraying, agriculture spray deposition, internal combustion engines and spray cooling. Such phenomena are very expensive to compute and much effort is made in developing models and algorithms able to approach them [1]. In this paper the algebraic VOF-model for surface capturing in simulations of the hydrodynamics in free surface flows is presented, which is implemented in the open-source CFD toolbox OpenFOAM® [2].

2. DESCRIPTION OF THE MODEL
In the VOF-method the motion of two incompressible immiscible fluids is modeled as the motion of an effective fluid, the physical properties of which are weighted averages depending on the distribution of the phase fraction, being equal to the properties of each of the pure fluids in the regions they occupy and varying only across the interface. The free surface is tracked by using the phase fraction γ of one of the fluids which takes values between 1 and 0 corresponding to each of the two pure phases, respectively. The model for the free-surface flow consists of the governing transport equations for the conservation of mass, phase fraction and momentum in the following form

$$\nabla \cdot \mathbf{U} = 0, \qquad \text{... (1)}$$

$$\frac{\partial \gamma}{\partial t} + \nabla \cdot \left(\mathbf{U} \gamma \right) + \nabla \cdot \left[\mathbf{U}_c \gamma \left(1 - \gamma \right) \right] = 0, \qquad \text{... (2)}$$

$$\begin{aligned} &\frac{\partial \left(\rho \mathbf{U} \right)}{\partial t} + \nabla \cdot \left(\rho \mathbf{U} \mathbf{U} \right) = \\ &\quad -\nabla p_d + \nabla \cdot \left(\mu \nabla \mathbf{U} \right) + \nabla \mathbf{U} \cdot \nabla \mu - \mathbf{g} \cdot \mathbf{x} \nabla \rho + \sigma \kappa \nabla \gamma, \end{aligned} \qquad \text{... (3)}$$

where $\mathbf{U}$ is the velocity of the effective fluid, γ is the phase fraction, ρ and μ are the density and viscosity of the effective fluid, p_d is the modified pressure obtained by absorbing the hydrostatic contribution into static pressure, $\mathbf{x}$ is the position vector, $\mathbf{g}$ is acceleration due to gravity, σ is the surface tension and κ is curvature of the interface. The last term in Eq. (3) is the so-called Continuum Surface Force model for the volumetric surface tension force [3]. Mixture properties are calculated as weighted averages, i.e. for some specific property y in liquid and gas, the corresponding mixture property of the effective fluid is $y = \gamma y_l + (1-\gamma) y_g$, and surface curvature is calculated as $\kappa = -\nabla \cdot \left(\nabla\gamma / |\nabla\gamma| \right)$. The last convective term in Eq. (2) is the so-called compressive term which is active only in the interface region and has the purpose of effectively shrinking the interface thickness and providing a sharp interface representation in the simulation [4]. The model is implemented in OpenFOAM® which utilizes the finite volume method for arbitrary unstructured meshes. The notable feature of the model is the compression velocity $\mathbf{U}_c$ in Eq. (2), which is modeled as

$$\mathbf{U}_c = \min\left[C_\gamma |\mathbf{U}|, \max\left(|\mathbf{U}|\right) \right] \frac{\nabla\gamma}{|\nabla\gamma|} \qquad \text{... (4)}$$

and related to the maximum velocity in the solution domain. The intensity of the interface compression is controlled by the parameter C_γ: no compression is obtained for C_γ =0, conservative compression for C_γ =1, and enhanced compression for C_γ >1 [4]. The time step is adjusted within the time loop according to the maximum prescribed Courant number, commonly Co_{max}=0.2. The phase fraction equation is solved in several sub-cycles within a single time step. The sub-cycle time step is set by dividing the global time step by the preset number of sub-cycles and the total mass flux in the global time step is calculated by summing (accumulating) the sub-cycle mass fluxes. In the calculations of source terms in Eq. (3) the corresponding values in cell-centers are obtained by reconstructing them from the cell-face values as weighted averages of the (staggered) values at cell-faces. The coupling between pressure and velocity is ensured by adopting the PISO algorithm [5]. The same reconstructing procedure is used for recovering velocities from conservative face fluxes in the corrector stage of the PISO algorithm.

3. DEMONSTRATION OF THE MODEL PERFORMANCE

The performance of the model is tested by computing droplet shape evolution in zero gravity and drop impact onto a liquid wall film. The calculations are initialized by prescribing the distribution of phase fraction and also the velocity for the case of drop impact. The set of boundary conditions for the case of drop impact consists of axis of symmetry, the no-slip condition at walls and an open top boundary. For the case of gravity-free droplet shape evolution a plane of symmetry is used instead of the bottom wall. Both cases are two-dimensional with axisymmetric meshes. The source terms are evaluated using the mid-point rule and time derivatives are discretized using the implicit Euler scheme. The spatial derivatives are converted into surface integrals by using Gauss's theorem. Convective terms are discretized by using the Gamma differencing scheme [6]. The physical properties of liquids used are listed in Table 1 and atmospheric air was used as the gaseous environmental fluid.

Table 1. Physical properties of liquids.

	Gravity-free drop	Drop impact
density ρ, kg/m^3	805	1179
viscosity μ, Pas	$2.3\cdot10^{-3}$	$7.1\cdot10^{-3}$
surface tension σ, N/m	$2.36\cdot10^{-2}$	$6.68\cdot10^{-2}$

In the case of the gravity-free droplet, the liquid is initialized as the cylindrical section in two dimensions with a mesh of 100×100 cells. The computed temporal interface evolution is shown in Fig. 1 without and with interface compression. The phase-interface is represented in colors for values from 0.1 to 0.9. It is seen that the interface is resolved more sharply with interface compression. Values of the droplet radius are tracked in three directions: horizontal, vertical and at the angle of 45°. The representative point at the interface is found as the first point satisfying $\gamma \geq 0.5$ and the computed results are shown in Fig. 2. There is a small difference between the values for the droplet radius in different directions, indicating that the computed droplet is not a perfect sphere.

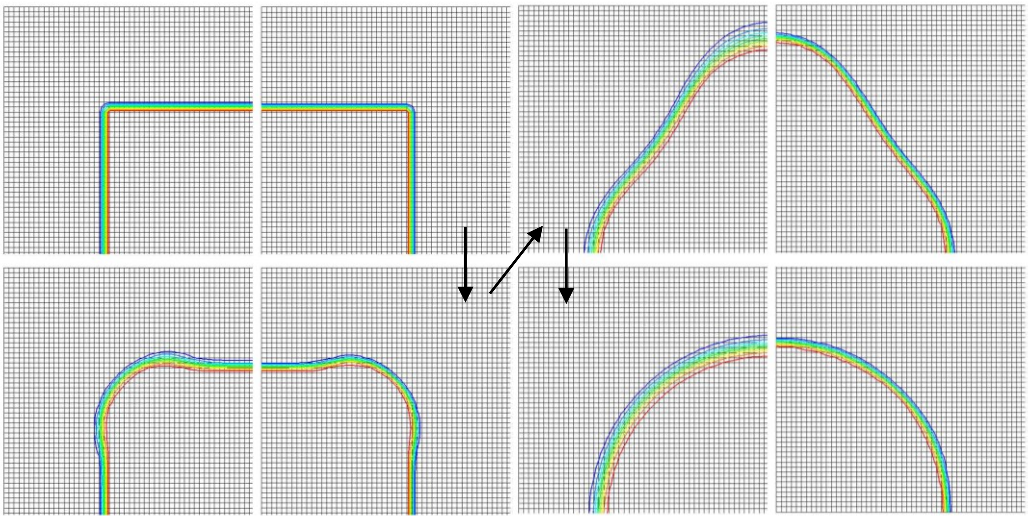

Figure 1. The computed temporal interface evolution: the right-side snapshots are with interface compression.

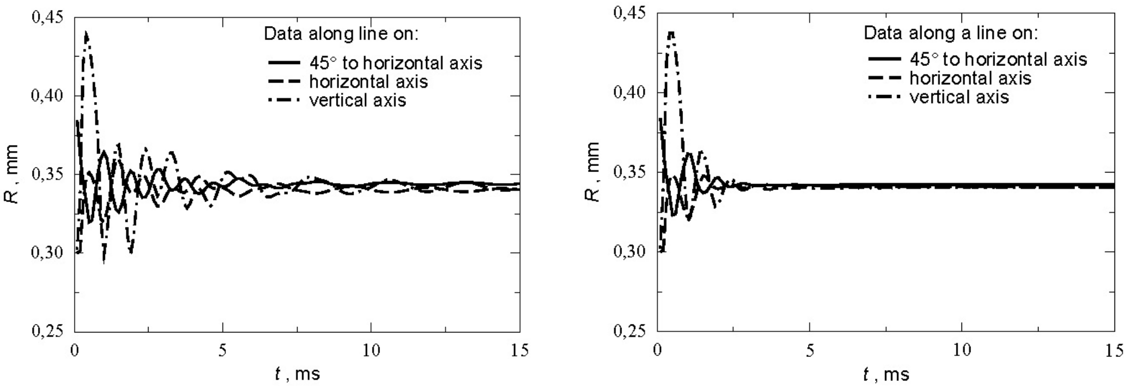

Figure 2. The computed droplet radius evolution without (left graph) and with interface compression (right graph).

The computed equilibrium velocity fields are shown in Fig. 3 at time t=15 ms when the droplet has reached the final bulk shape. It may be seen that when the quasi-equilibrium state is reached, the generated parasitic currents with interface compression are more aligned with the free surface.

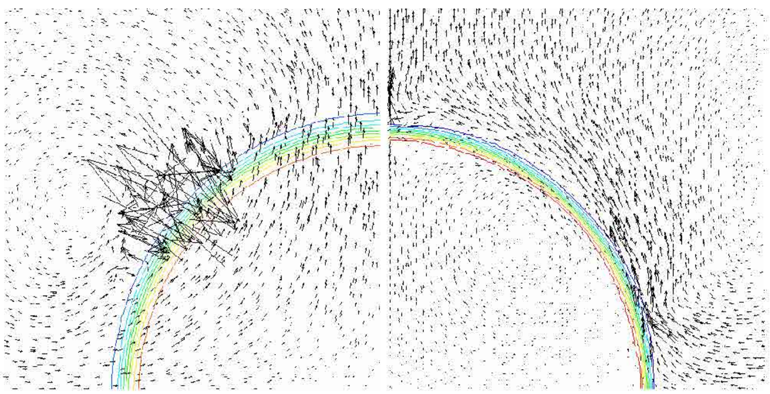

Figure 3. The computed velocity fields without (left) and with interface compression (right).

For the case of drop impact, the liquid in the drop is initialized as a spherical section and the wall film as a cylindrical section in two dimensions. The mesh is refined in the region of the impact and has in total about 70000 cells. The sequence of stages after the impact is observed in experiments and described in [7]. After the first contact of the falling drop with the liquid film a small circumferential liquid jet is ejected upward, then a crater is formed within the wall film which at first expands and then recedes due to surface tension effects, and finally collapses around the axis of symmetry ejecting an upward jet. The computed shape of the interface at various times is shown in Fig. 4. The phase-interface is indicated with black lines representing two γ-values equal 0.1 and 0.9. It is seen that without interface compression the free-surface is non-physically smeared, whereas with interface compression it stays sharp.

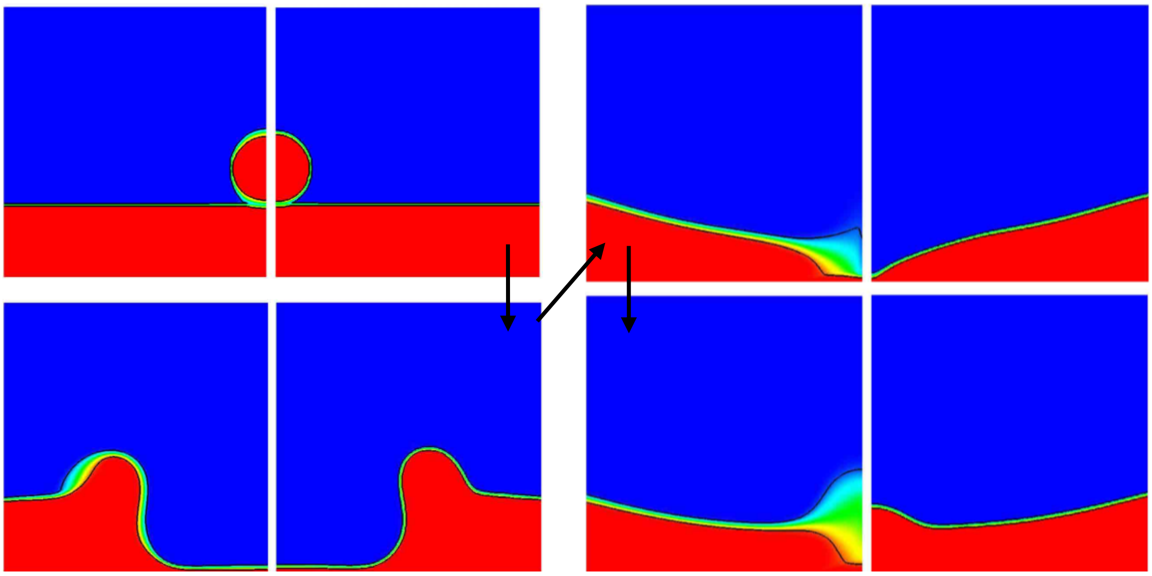

Figure 4. Drop impact onto a wall film: the right-side snapshots are with interface compression.

4. CONCLUSIONS

The algebraic volume-of-fluid model for interface capturing in free-surface flows with OpenFOAM® has been presented. The special scheme is utilized to suppress the numerical diffusion and compress the phase-interface. The potential of the model is demonstrated by simulating gravity-free droplet shape evolution and drop impact onto a liquid film onto the flat underlying wall. In both cases the interface compression yields sharply resolved interface showing good capabilities for the prediction of free-surface flows.

5. REFERENCES

[1] D. Gerlach, G. Tomar, G. Biswas, and F. Durst: Comparison of volume-of-fluid methods for surface tension-dominant two-phase flows, International Journal of Heat and Mass Transfer, 49:740-754, 2006.

[2] OpenFOAM® The open source CFD toolbox, https://www.openfoam.com/

[3] J.U. Brackbill, D.B. Kothe, and C. Zemach: A continuum method for modeling surface tension, Journal of Computational Physics, 100:335-354, 1992.

[4] H.G. Weller: A New Approach to VOF-based Interface Capturing Methods for Incompressible and Compressible Flow, Technical Report TR/HGW/04, OpenCFD, 2008.

[5] H. Rusche: Computational fluid dynamics of dispersed two-phase flows at high phase fractions, PhD thesis, Imperial College of Science, Technology and Medicine, London, 2002.

[6] H. Jasak, H.G. Weller, and A.D. Gosman: High resolution NVD differencing scheme for arbitrarily unstructured meshes, International Journal for Numerical Methods in Fluids, 31:431-449, 1999.

[7] E. Berberovic, N.P. van Hinsberg, S. Jakirlic, I.V. Roisman, and C. Tropea. Drop impact onto a liquid layer: Dynamics of the cavity evolution, Physical Review E, 79:036306, 2009.