Surface Tension Model for High Viscosity Ratios implemented in VOF model

J. Bohacek*
Heat Transfer and Fluid Flow Laboratory, Faculty of Mechanical Engineering
Brno University of Technology
Technicka 2896/2, 616 69 Brno, Czech Republic

Abstract

In this paper, a new surface tension model suitable for Volume of Fluid (VOF) model is introduced. The VOF model was designed many years ago in order to be solved by FVM. Nowadays, it is used in most of commercial CFD packages, e.g., ANSYS FLUENT, ANSYS CFX, and has the capability to simulate surface tension effects. However, when surface tension is dominant, i.e., Capillary or Weber number is low; it gives rather spurious currents that can cripple a velocity field in a whole domain. In VOF, the interface between the primary and the secondary phases is tracked on a fixed grid. In reality, surface tension acts as a surface force, however, here, it must be imposed as a body force into momentum equations. Usually, CSF approach by Brackbill is used. It does not impose the sharp boundary condition on interface and it rather smears it into perpendicular direction from the interface that causes artificial momentum around the interface. It leads into spurious velocities, which grows especially in low density fluids. Here, we propose the new surface tension model that calculates interface normal from Height Functions. The curvature is calculated as the average value of circle radii. Then, each normal is recomputed from the circle centre and the curvature. Body force of surface tension is not imposed directly on the interface but rather in mesh cells containing only the denser phase and located near the interface. Several tests with different mesh sizes were carried out to test this model. ANSYS FLUENT was used for these purposes. A rectangular domain (4x4 mm) with 2 mm droplet patched in the middle was surrounded by four slip walls. Three different surface tension models were taken into account – the CSF model, the HF model and our surface tension model. Computed data was compared and summarized into results.

Introduction

Nowadays, Computational Fluid Dynamics is extensively developing and is very useful in most of industrial branches. Here, the paper is solely focused on the multiphase modeling of free-surface flows with dominant surface tension effects. Plenty of numerical methods for solving interfacial flows can be found in literature, e.g., front-tracking method, the boundary integral method, the phase-field method, the Second Gradient method, the Level Set method, the Volume of Fluid method (VOF) [2]. Most of commercial CFD packages offers the VOF model since it is very robust, mass conservative and it requires no additional equations to be solved iteratively. Despite other methods, it suffers from the fact that the position of interface is not know a priori and it must be reconstructed from so called volume fractions accessible in each cell.

This paper is solely focused on a modeling of interfacial flows with very high Capillary number. Since the ANSYS FLUENT is presumably the most widely used commercial CFD package, for this reason, all the work is intentionally done in this software with implemented the VOF model with the CSF model discussed later.

In literature [4], several references can be found to so-called spurious currents also known as parasitic currents that cause unphysical vortex-like velocities. Here, we compare spurious currents obtained within standard CSF model, the HF model and our new surface tension model.

Methods

Firstly, the CSF model is briefly discussed [3]. The interface is naturally reconstructed only from volume fractions. The interface normal is computed from local gradients of volume fractions and the interface curvature is defined in terms of the divergence of the unit normal. Since the field of volume fractions is rather a discontinuous function, this approach could be inacurrate. The surface tension force is transformed into the body force, imposed as a source term to the momentum equations and is given by the following equation:

$$F_{s-t} = \sigma \frac{\rho \kappa \frac{\partial F}{\partial x_i}}{0.5\left(\rho_g + \rho_l\right)} \tag{1}$$

* Corresponding author: bohacek@lptap.fme.vutbr.cz

Secondly, the HF methodology [1] was used to improve the estimation of the normal to the interface and the interface curvature. Further, the position of interface is reconstructed in a piecewise linear way from volume fractions and normals. The length of the linear segment is calculated consequently. The source term was applied in cell centres, only in cells that contain the interface and it is given as following formula:

$$F_{s-t} = \frac{\sigma\kappa \cdot l}{V} \tag{2}$$

Finally, in the third model, the HF method gave the first estimation of normals and the linear segments were reconstructed. Around each cell containing the interface, a 3×3 stencil was constructed and the curvature was calculated from radii where each radius corresponds to the circle circumscribed around three points of piecewise linear interface in the vicinity of the cell of interest. Knowing such an average circle centre corresponding to the interface cell, the normal was recomputed in this cell. In next step, it was necessary to define cells where the surface tension force will be applied.

Imagine the cell containing the interface of the mesh size length but containing almost no dense phase, i.e. there is the interface in the cell but almost no water – only air. In such a cell, artificial air velocities would be induced, thus, the source term is always imposed in the cell containing only water. Such a cell is found right under the interface in the normal direction.

Model Settings

The 2D rectangular computational domain (4×4 mm) was mapped with quad mesh and surrounded by four slip walls. Three different mesh densities were tested to reveal mesh sensitivity. The circle of 2 mm diameter was patched in the centre. The source term standing for surface tension effects was the only body force. The first order implicit unsteady solver with PISO pressure-velocity coupling was used. In Fluent, the pressure correction equation does not account for the presence of surface tension, it cannot be even modified and thus, an extra UDF was used to remedy pressures and velocities on interface.

Results and Discussion

As regards the standard CSF model, the L_∞ and L_1 errors for velocities are in order of 0.1 and 0.01 m/s, respectively. Results do not improve with mesh refinement. Unlike the normals computed from gradients of volume fraction, the normals computed from HFs were 6times more accurate in the second model. The best accuracy of predicted normals was in vertical or horizontal direction. On the contrary, in diagonal direction there was an error of 30 %. It led to overestimated surface tension forces, consequently. In the third model, the curvature error was less than 4 %. Spurious currents have been significantly suppressed especially in the third model.

Nomenclature

a	acceleration [$m \cdot s^{-2}$]
F	volume fraction [-]
l	length of linear segment [m]
V	cell volume [m^3]
κ	curvature [$1 \cdot m^{-1}$]
ρ	density [$kg \cdot m^{-3}$]
σ	surface tension [$N \cdot m^{-1}$]

Subscripts

g	gas
l	liquid

References

[1] Afkami,S.,Bussmann, M., *Height functions for applying contact angle to 2D VOF simulations,* International Journal for Numerical Methods in Fluids, 2008, vol. 57, pp. 453-472

[2] Bohacek, J., *Modeling of free-falling water droplet, Handicap of CSF approach and proposals for its improvement,* Water Jet Conference, Ostrava, pp. 1-10, (2009).

[3] Brackbill, J.U., Kothe, D.B., Zemach, C., *A continuum method for modeling surface tension*, J. Comput. Phys., 1992, vol. 100, pp. 335-354

[4] Galusinski, C., Vigneaux, P., *On stability condition for bifluid flows with surface tension: Application to microfluidic*, Journal of Computational Physics, 2008, vol. 227, pp. 6140-6164.