Applied Mathematical Modelling xxx (2015) xxx–xxx

Contents lists available at ScienceDirect

Applied Mathematical Modelling

journal homepage: www.elsevier.com/locate/apm

Implementation of a height function method to alleviate spurious currents in CFD modelling of annular flow in microchannels

Zhenyi Guo, David F. Fletcher *, Brian S. Haynes

School of Chemical and Biomolecular Engineering, The University of Sydney, NSW 2006, Australia

ARTICLE INFO

Article history:
Received 19 August 2014
Received in revised form 13 February 2015
Accepted 17 April 2015
Available online xxxx

Keywords:
Computational fluid dynamics
Volume of fluid method
Surface tension modelling
Height function method
Microchannel annular flow

ABSTRACT

The use of the continuum surface force (CSF) model in surface tension modelling using the volume of fluid (VOF) method is known to introduce unphysical velocities, called parasitic or spurious currents arising mostly from poor calculation of interface curvature. In this paper, a modified 2-D axisymmetric height function (HF) method is proposed and implemented in parallelised computations. The default CSF method in the commercial software ANSYS Fluent 15.0 is replaced using User-Defined Functions (UDFs) of ANSYS Fluent and the improvements obtained are evaluated in test cases with stationary and moving interfaces. The application of the HF method to annular flow modelling has shown massive improvements in interface modelling and potential for research on annular flow hydrodynamics and heat transfer.

1. Introduction

The application of microchannel two-phase flow is widespread in cutting edge technologies, such as in electrical, chemical and biomedical engineering and in the micro-structured devices used in modern chemical plants [1–4]. Experiments have shown that annular flow is the prevalent flow regime in microchannel flow boiling [5–10]. Eulerian-based interface capturing methods are widely used in the computational fluid dynamics (CFD) analysis of this two-phase flow system [11], such as the volume of fluid (VOF) method [12] and the level-set (LS) method [13,14]. As the interface capturing methods do not track the interface explicitly, a colour function is used to indicate the location of the interface and also to calculate the interfacial transfer terms. Therefore, the choice of colour function has a significant impact on the performance of the modelling of the interfacial surface tension force. The VOF method has been shown to suffer from the so-called spurious currents problem due to the poor surface tension modelling using the conventional continuum surface force (CSF) model proposed by Brackbill et al. [15]. However, the VOF method has been the most widely used interface capturing method due to its low computational cost and ease of implementation [16,17]. Therefore, many efforts have been made to improve the surface tension calculation using the VOF method. In the current work, a height function method proposed by Malik et al. [18] and Hernández et al. [19] is modified and implemented into the commercial code ANSYS Fluent. Test cases are performed to evaluate the improvements in surface tension modelling made by the height function method and its ability to resolve the surface tension forces occurring in the annular flow regime, the study of which is the motivation of this work.

* Corresponding author. Tel.: +61 2 9351 4147; fax: +61 2 9351 2854.
E-mail address: david.fletcher@sydney.edu.au (D.F. Fletcher).

http://dx.doi.org/10.1016/j.apm.2015.04.022

Nomenclature

A	area
D	diameter
$\boldsymbol{F}_{\sigma}$	surface tension force
$\boldsymbol{g}$	gravitational acceleration
H	discretised height function
h	grid spacing
$\boldsymbol{n}$	unit normal vector
n_y	y-component of the volume fraction gradient
P	pressure
R	radius
$\boldsymbol{S}_{\sigma}$	surface tension momentum source
t	time
$\boldsymbol{v}$	velocity
Δx	horizontal grid spacing
Δy	vertical grid spacing
Greek letters	
α	volume fraction of a fluid phase
ε	small positive value
κ	curvature
μ	viscosity ($\tilde{\mu}$ for bulk viscosity)
ρ	density ($\tilde{\rho}$ for bulk density)
σ	surface tension coefficient
$\boldsymbol{\tau}$	unit tangent vector or interfacial shear stress

2. Review of surface tension modelling

At a curved gas–liquid interface, the liquid surface has the tendency to contract to the minimum area and hence creates a pressure jump across the interface which is known as the Laplace pressure [20]. In CFD, the effect of surface tension can be modelled via normal and tangential forces applied at the interface between the two fluids. The normal and tangential forces must balance at the interface giving Eqs. (1) and (2) [21].

$$-P_l + P_g + \boldsymbol{n} \cdot (\boldsymbol{\tau}_l - \boldsymbol{\tau}_g) = \sigma\kappa, \tag{1}$$

$$\boldsymbol{n} \times (\boldsymbol{\tau}_l - \boldsymbol{\tau}_g) = \nabla\sigma. \tag{2}$$

where κ is the curvature of the interface and $\boldsymbol{\tau}_i$ are the interfacial shear stresses with subscripts l and g denoting the liquid and gas phases, respectively. The variation of surface tension coefficient with temperature needs to be modelled if the temperature changes significantly along the interface [22]. It is evident from Eq. (1) that an inaccurate curvature value will result in an incorrect pressure jump across the gas–liquid interface, so the accurate calculation of the interface curvature is vital when modelling the surface tension force.

In interface capturing methods, such as volume of fluid (VOF) and the level-set (LS), these interfacial conditions are not implemented directly, as no explicit interface is present. Rather, they must be built into the numerical method. The most widely used surface tension model for interface capturing methods is the continuum surface force (CSF) model proposed by Brackbill et al. [15]. Lafaurie et al. [23] and Harvie et al. [24] pointed out that this implementation of the surface tension force can induce unphysical velocities near the interface, known as 'spurious or parasitic currents'. In their review of CFD modelling of microchannel flows, Fletcher et al. [25] pointed out that the main challenges in the modelling of multiphase flow in microchannels are the correct identification of the interface location and the minimisation of the parasitic currents near the interface that arise due to surface tension modelling.

There are two problems that are responsible for the spurious currents in the VOF method using the CSF model. The first problem is the mismatch in the discretisation methods for the pressure gradient and the surface tension force [26]. This can be minimised by including the pressure jump from surface tension in the calculation of the velocity field at the pressure correction stage [27]. However, ANSYS Fluent provides a PRESTO! (Pressure STaggering Option) scheme which uses the "staggered" control volumes around the faces to calculate the "staggered" pressure field [28]. Similar to the "staggered" grid method for velocity fields proposed by Patankar [29], the PRESTO! scheme obtains the face-centred values of pressure using the "staggered" pressure field directly without performing interpolations. Therefore, a mismatch between the interpolated pressure and the discretisation of the surface tension force is avoided.