Journal Pre-proof

Implicitly Coupled Phase Fraction Equations for the Eulerian Multi-Fluid Model

Robert Keser, Vuko Vukčević, Michele Battistoni, Hong Im, Hrvoje Jasak

PII: S0045-7930(19)30240-3
DOI: <https://doi.org/10.1016/j.compfluid.2019.104277>
Reference: CAF 104277

To appear in: *Computers and Fluids*

Received date: 7 June 2019
Revised date: 2 August 2019
Accepted date: 23 August 2019

Please cite this article as: Robert Keser, Vuko Vukčević, Michele Battistoni, Hong Im, Hrvoje Jasak, Implicitly Coupled Phase Fraction Equations for the Eulerian Multi-Fluid Model, *Computers and Fluids* (2019), doi: <https://doi.org/10.1016/j.compfluid.2019.104277>

This is a PDF file of an unedited manuscript that has been accepted for publication. As a service to our customers we are providing this early version of the manuscript. The manuscript will undergo copyediting, typesetting, and review of the resulting proof before it is published in its final form. Please note that during the production process errors may be discovered which could affect the content, and all legal disclaimers that apply to the journal pertain.

© 2019 Published by Elsevier Ltd.

Highlights

- Implicitly coupled solution procedure for the phase fraction equations
- Results were obtained for a bubbly flow which forms a mixing layer inside a duct
- Verification and validation of the implemented implicitly coupled solution procedure
- Performance comparison of the coupled and segregated implementation
- Performance improvement for flows with higher bubble phase fraction

Implicitly Coupled Phase Fraction Equations for the Eulerian Multi-Fluid Model

Robert Keser^{a,*}, Vuko Vukčević^a, Michele Battistoni^c, Hong Im^d, Hrvoje Jasak^{a,b}

^a*University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Ivana Lučića 5, Zagreb, Croatia*

^b*Wikki Ltd, 459 Southbank House, SE1 7SJ, London, United Kingdom*

^c*Department of Engineering, University of Perugia, Perugia 106123, Italy*

^d*King Abdullah University of Science and Technology, Clean Combustion Research Center, Thuwal, Saudi Arabia*

Abstract

In this work, the implementation, verification and validation of an implicitly coupled solution procedure for the phase fraction equations in the Eulerian multi-fluid model are presented. The model is implemented within the `foam-extend` toolbox, a community-driven fork of OpenFOAM. The implicitly coupled system for an arbitrary number of phases is based on the modified formulation of the phase fraction equation. This formulation takes advantage of the mixture divergence-free velocity and the cross-coupling with the remaining phase fraction equations via the non-linear relative velocity term. The linearised and implicitly coupled phase-fraction equations are solved simultaneously within a single block matrix. The model is tested for a bubbly air-water upward flow which forms a mixing layer inside a square duct. In

*Corresponding author.

Email address: `robert.keser@fsb.hr` (Robert Keser)

the first test, the mesh verification analysis is performed on structured grids with different levels of refinement. The second test investigates the influence of the number of bubble phases on the flow solution for the same flow conditions. In the third test, the implemented model is validated against experimental data from the literature. The last test compares the performance of the implemented implicitly coupled solution procedure for the phase fraction equations against the standard segregated implementation. The proposed method shows good agreement with experimental data, and has proven to be consistent both in terms of the number of phases and grid refinement. Furthermore, the method improved the convergence of the solution for flows at higher bubble phase fraction.

Keywords: Finite volume method, Multiphase flow, Eulerian multi-fluid model, Implicit coupling, OpenFOAM

1. Introduction

Multi-phase flows are widely present both in natural and industrial processes. Understanding and predicting behaviour of such flows is of great importance if processes which utilise multi-phase phenomena are to be optimally and safely designed and controlled. The term multi-phase flow covers a huge spectrum of flow patterns and regimes, and this work limits the scope of investigation to dispersed multi-phase flows. The term dispersed defines the topology of the flow, where one of the constituent phases is present in the form of bubbles, particles or droplets which are suspended within a

continuous phase. There are three main CFD approaches in describing the multi-phase flow. The first and computationally the most expensive is Direct Numerical Simulation (DNS) (Hirt and Nichols (1981); Osher and Sethian (1988); Sussman et al. (1994)), where the interface of each dispersed element in the flow domain is individually tracked or reconstructed, while the time scales associated with turbulent fluctuations are resolved and not modelled. Therefore, due to the large computational cost, DNS is most often not applicable to engineering-type industrial simulations. The second option is the Lagrangian approach (Subramaniam (2013)), where individual dispersed elements or groups of dispersed elements (parcels) are tracked through the domain by solving suitable equations of motion, which are expressed in the Lagrangian frame of reference. The continuous phase is still defined and solved in the Eulerian frame of reference. This approach is most often suitable for relatively dilute flows where the dispersed phase fractions are still sufficiently low (Durst et al. (1984); Hoyas et al. (2013)). The third and the most general option is the Eulerian or the multi-fluid (Ishii and Hibiki (2011, 2006)) approach. Here, all phases are defined in the Eulerian frame of reference and are treated as interpenetrating continua, where each phase is represented by appropriately averaged conservation equations (Crowe et al. (2011); Hill (1998)). Due to the averaging, several terms in the conservation equations require closure, e.g. Reynolds stresses, and averaged inter-phase momentum transfer. This approach is suitable for all flow regimes, but due to local homogenisation, local characteristics of the flow are obtained only

in an averaged sense. Despite its broad range of application, the multi-fluid model often has problems with stability and robustness of the solution algorithm, which can lead to problems with the convergence or even to the divergence of the numerical solution. The origin of those problems often lies in the hyperbolicity, grid refinement, inappropriate closure models, unboundedness of the phase-fraction, etc. (Ransom and Hicks (1984); Dinh et al. (2003); Jareteg et al. (2017); Panicker et al. (2018); Ishii and Hibiki (2006); Ferreira et al. (2019); Weller (2002); Oliveira and Issa (2003)). In multi-phase flow with high density ratios, the phase conservation is crucial, because small errors in volume fraction produce significant errors in phase mass conservation. Therefore, the focus of this work is on the appropriate numerical implementation and solution of phase-fraction equations for the Eulerian multi-fluid model.

Boundedness of the phase-fraction is one of the main concerns in the numerics of multi-phase flows, where the solution of the phase-continuity equation should produce conservative and bounded results. Rusche (2002) presented an overview of several approaches, where various authors derived and tested different phase-continuity equations and recombination techniques. The easiest way to transport the phase-fraction field is by solving the original form of the phase-continuity equation (Hill (1998); Politis (1989)), but this approach has problems with boundedness (especially if all phases share the same mixture pressure). Individual phase-fractions can be easily bounded by zero if bounded (temporal and convection) discretisation schemes are em-

ployed, but the sum of all phase-fractions is not necessarily bounded. B. Spalding (1985) suggested a recombination technique, which bounds the solution at both ends but overwrites the solution of the phase-continuity equation. Hence, the solution is not necessarily conservative until convergence is reached, which has to be taken into account for transient simulations. Carver (1982) tried to couple the phases through a composite solution for both phases in the two-fluid model, where the two phase-continuity equations are subtracted, and the resulting equation is solved. This approach is computationally more efficient, but even the converged solution does not guarantee conservativeness nor boundedness. Weller (2002) reformulated the phase continuity equations (for the two-fluid model) in terms of the mean velocity, i.e. he decomposed the velocity into the mean and relative part. This procedure results with all terms in a conservative form and with an additional non-linear term which couples the two phases implicitly through convective terms.

Rusche (2002); Silva and Lage (2011); Jareteg et al. (2017) employed the phase-continuity equation formulation presented by Weller (2002) for the two-fluid model, but due to the additional non-linearity, the authors reported possible difficulties in convergence. As a remedy, they suggested the usage of lower Courant number and full implicit treatment of the non-linear term. Kissling et al. (2010); Ferreira et al. (2019) generalised the modified phase-continuity for the multi-fluid model, but the generalised equations were still solved in a segregated manner. Even Fernandes et al. (2019) who developed

an implicit pressure-velocity coupling for the Eulerian multi-fluid model employed the segregated solution procedure for the modified phase continuity equations.

From the literature review, it is clear that boundedness and conservation over a set of coupled volume fraction equations is an essential property of the solution which needs to be preserved. None of the methods proposed in the literature simultaneously satisfy both criteria and form a robust solution algorithm. Therefore, this work presents a novel implicitly-coupled solution procedure for the modified phase-continuity equation (Weller (2002)) for an arbitrary number of phases. To the authors' knowledge, the implicitly-coupled solution procedure for the phase fraction equations has not been previously adopted for the Eulerian multi-fluid model. It is the objective of this study to test the feasibility of the novel implicitly coupled approach as a foundation for a robust framework for simulating multi-phase flows. The method is implemented in `foam-extend` using the available block-coupled Finite Volume methodology (Uroić and Jasak (2018); Ferreira et al. (2019); Fernandes et al. (2019)).

The paper is organised as follows. Section 2 presents the governing equations for the Eulerian multi-fluid model for an arbitrary number of phases. Section 3 presents the block-system structure, necessary linearisation procedure and the implementation of the implicitly-coupled phase-continuity equations. Section 4 describes the test case and presents the numerical results, which include a mesh refinement study, a multi-fluid consistency study,

validation and performance analysis. Section 5 gives a conclusion regarding the behaviour of the implemented model.

2. Eulerian multi-fluid model

The Eulerian multi-fluid model implemented in this work follows the approach previously presented by Rusche (2002), Weller (2002) and Hill (1998) where the authors derived the conditionally averaged equations, based on the work of Dopazo (1977) for the incompressible two-fluid model without mass transfer between different phases or fluids.

This section presents the Eulerian multi-fluid model for an arbitrary number of incompressible phases which is used in this work.

2.1. Phase-intensive momentum equation

Following the approach presented by Weller (2002) and Rusche (2002), the conditionally averaged momentum equation for phase φ in the phase-intensive form reads:

$$\frac{\partial \bar{\mathbf{U}}_\varphi}{\partial t} + \bar{\mathbf{U}}_\varphi \cdot \nabla \bar{\mathbf{U}}_\varphi + \nabla \cdot \bar{\mathbf{R}}_\varphi^{\text{eff}} + \frac{\nabla \alpha_\varphi}{\alpha_\varphi} \cdot \bar{\mathbf{R}}_\varphi^{\text{eff}} = -\frac{\nabla \bar{p}}{\bar{\rho}_\varphi} + \mathbf{g} + \frac{\bar{\mathbf{M}}_\varphi}{\alpha_\varphi \bar{\rho}_\varphi}, \quad (1)$$

where the subscript φ denotes the phase, $\bar{\mathbf{U}}_\varphi$ denotes the averaged phase velocity, α_φ is the phase fraction, $\bar{\mathbf{R}}_\varphi^{\text{eff}}$ is the combined Reynolds (turbulent) and viscous stress, $\bar{p}$ is the mixture (phase volume fraction mean) pressure shared by all phases, $\bar{\rho}_\varphi$ is the density of phase φ , $\mathbf{g}$ is the acceleration due to gravity, and the $\bar{\mathbf{M}}_\varphi$ is the averaged inter-phase momentum transfer term.

The Reynolds stresses are calculated with the Boussinesq hypothesis and the combined (laminar and turbulent) stress $\overline{\mathbf{R}}_\varphi^{\text{eff}}$ is calculated as:

$$\overline{\mathbf{R}}_\varphi^{\text{eff}} = -\nu_\varphi^{\text{eff}} \left(\nabla \overline{\mathbf{U}}_\varphi + \nabla \overline{\mathbf{U}}_\varphi^\top - \frac{2}{3} \mathbf{I} \nabla \cdot \overline{\mathbf{U}}_\varphi \right) + \frac{2}{3} \mathbf{I} k_\varphi, \quad (2)$$

where the effective viscosity ν_φ^{eff} is calculated as a sum of the molecular and the turbulent (kinematic) viscosity $\nu_\varphi^{\text{eff}} = \nu_\varphi + \nu_\varphi^t$, k_φ represent the turbulence kinetic energy of phase φ , $\mathbf{I}$ is the identity tensor and the superscript $\top$ is the transpose operator.

In general, the averaged inter-facial momentum transfer term $\overline{\mathbf{M}}_\varphi$ is calculated as:

$$\overline{\mathbf{M}}_\varphi = \sum_{i=1, i \neq \varphi}^{n_{\text{phases}}} \overline{\mathbf{M}}_{\varphi,i}, \quad (3)$$

where n_{phases} is the total number of phases or fluids in the system, and $\overline{\mathbf{M}}_{\varphi,i} = -\overline{\mathbf{M}}_{i,\varphi}$ represents the momentum exchanged between phases φ and i , and vice versa.

In this work, the implemented Eulerian multi-fluid model is specialised for bubbly flows, in which water is represented by one continuous phase, and the bubbles are sub-divided into multiple size classes, i.e. multiple dispersed phases. Each bubble class is treated as a different phase in the calculation and will have its momentum equation and phase continuity equation, but the mixture pressure is shared among all phases (including the continuous phase). In this case $n_{\text{phases}} = n_{\text{bubbles}} + 1$, where n_{bubbles} is the number of bubble classes. Furthermore, the implemented model does not account for

breakup or coalescence of bubbles. Therefore, the dispersed phases exchange momentum only with the continuous phase via virtual mass forces, drag and lift, i.e. there is no momentum exchange between the dispersed phases. The averaged interfacial momentum transfer term for the continuous phase is given by:

$$\overline{\mathbf{M}}_c = - \sum_{i=1}^{n_{\text{bubbles}}} \overline{\mathbf{M}}_{d,i}, \quad (4)$$

where the subscript c denotes the continuous phase, subscript d indicates the dispersed phase, and the inter-facial momentum transfer term for the i -th dispersed phase reads:

$$\begin{aligned} \overline{\mathbf{M}}_{d,i} = & \alpha_{d,i} C_{d,i} \frac{3 \bar{\rho}_c}{4 d_i} |\overline{\mathbf{U}}_{r,i}| \overline{\mathbf{U}}_{r,i} && \text{drag} \\ & + \alpha_{d,i} C_l \bar{\rho}_c \overline{\mathbf{U}}_{r,i} \times (\nabla \times \overline{\mathbf{U}}_c) && \text{lift} \\ & + \alpha_{d,i} C_{vm} \bar{\rho}_c \left(\frac{D_c \overline{\mathbf{U}}_c}{Dt} - \frac{D_{d,i} \overline{\mathbf{U}}_{d,i}}{Dt} \right) && \text{virtual mass} \\ & - C_{d,i} \frac{3 \bar{\rho}_c \nu_c^t}{4 d_i \sigma_\alpha} |\overline{\mathbf{U}}_{r,i}| \nabla \alpha_{d,i} && \text{turbulent drag.} \end{aligned} \quad (5)$$

The $D_\varphi \overline{\mathbf{U}}_\varphi / Dt$ term denotes the phase material time derivative. The relative velocity is defined as $\overline{\mathbf{U}}_{r,i} = \overline{\mathbf{U}}_c - \overline{\mathbf{U}}_{d,i}$. $C_{d,i}$, and d_i are the drag coefficient, and the bubble diameter of the i -th dispersed phase. C_l and C_{vm} are the lift and the virtual mass coefficient.

According to Behzadi (2001) and Rusche (2002), the drag coefficient is expressed as a function of phase fraction, i.e. a correction to the drag coeffi-

cient of a single bubble:

$$C_{d,i} = C_{d0,i} \left(\exp(3.64 \alpha_{d,i}) + \alpha_{d,i}^{0.864} \right), \quad (6)$$

where $C_{d0,i}$ is the drag coefficient of a single bubble of the i -th bubble phase which is calculated with the following expression (Rusche (2002); Hill (1998); Haberman and Morton (2011)):

$$C_{d0,i} = \exp \left(-51.8 + 13.2 \ln(\text{Re}_i) - 0.824 (\ln(\text{Re}_i))^2 \right). \quad (7)$$

The Re_i term is the Reynolds number for i -th bubble phase given by:

$$\text{Re}_i = \frac{|\bar{\mathbf{U}}_{r,i}| d_i}{\nu_c}. \quad (8)$$

The lift coefficient C_l and the virtual mass coefficient C_{vm} are treated as constants, i.e. their values do not depend on the phase fraction of bubbles.

2.2. Volumetric mixture continuity equation

The continuity equation for phase φ has the following form:

$$\frac{\partial \alpha_\varphi}{\partial t} + \nabla \cdot (\bar{\mathbf{U}}_\varphi \alpha_\varphi) = 0. \quad (9)$$

Combining eq. (9) for all phases results in the mixture volumetric continuity equation:

$$\nabla \cdot \bar{\mathbf{U}} = 0, \quad (10)$$

where $\bar{\mathbf{U}}$ is the mixture velocity given by:

$$\bar{\mathbf{U}} = \sum_{i=1}^{n_{\text{phases}}} \alpha_i \bar{\mathbf{U}}_i. \quad (11)$$

Since the described model assumes that all phases share the same mixture pressure, eq. (10) is used for the derivation of the mixture pressure equation. Rusche (2002) presented a detailed overview of the reformulation procedure for the two-fluid model.

2.3. Modified phase continuity equation

The original form of the phase continuity equation eq. (9) has problems with the unboundedness, because of the $\bar{\mathbf{U}}_\phi$ in the second term on the l.h.s., which can lead to convergence issues. The only divergence-free velocity is the mixture velocity $\bar{\mathbf{U}}$ because it satisfies the mixture continuity equation $\nabla \cdot \bar{\mathbf{U}} = 0$. Therefore, Weller (2002) rearranged the phase continuity equation in a more conservative form, where the phase fractions could be bounded at both ends, which was later generalised for the multi-fluid formulation by Kissling et al. (2010) and Silva and Lage (2011):

$$\frac{\partial \alpha_i}{\partial t} + \nabla \cdot (\bar{\mathbf{U}} \alpha_i) + \nabla \cdot \left(\alpha_i \sum_{j=1, j \neq i}^{n_{\text{phases}}} \alpha_j (\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j) \right) = 0. \quad (12)$$

Rusche (2002) concluded that this formulation couples the phases implicitly through the relative velocity $\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j$ in the third term on the l.h.s., but the limitation of this approach originates from the non-linear term $\alpha_i \alpha_j$.

Consequently, boundedness at both limits can only be assured if the equations are solved fully implicitly.

All available publications which employ this improved formulation of phase continuity equations solve the continuity equations in a segregated manner for each phase, where the presence of the cross-coupling terms is not fully exploited. Hence, this work presents the implementation and validation of an implicitly-coupled solution procedure for an arbitrary number of phase continuity equations. Reformulation and linearisation of eq. (12) for implicit cross-coupling is shown in section 3.

2.4. Turbulence model

This work adopts the modified k - ϵ turbulence model by Gosman et al. (1992) and Rusche (2002), which consists of the standard single-phase k - ϵ model by Launder and Spalding (1974) for the continuous phase with additional source terms to include the effects of the dispersed phase. The modified equations read:

$$\frac{\partial k_c}{\partial t} + (\bar{\mathbf{U}}_c \cdot \nabla) k_c - \nabla \cdot \left(\frac{\nu_c^{\text{eff}}}{\sigma_k} \nabla k_c \right) = P_c - \epsilon_c + S_k, \quad (13)$$

and

$$\frac{\partial \epsilon_c}{\partial t} + (\bar{\mathbf{U}}_c \cdot \nabla) \epsilon_c - \nabla \cdot \left(\frac{\nu_c^{\text{eff}}}{\sigma_\epsilon} \nabla \epsilon_c \right) = \frac{\epsilon_c}{k_c} (C_1 P_c - C_2 \epsilon_c) + S_\epsilon, \quad (14)$$

where k_c is the turbulence kinetic energy and ϵ_c is the rate of dissipation of turbulence energy of the continuous phase. The effective viscosity is calcu-

lated from $\nu_c^{\text{eff}} = \nu_c + \nu_c^t$, and ν_c^t is given by:

$$\nu_c^t = C_\mu \frac{k_c^2}{\epsilon_c}. \quad (15)$$

Furthermore, P_c stands for the production of turbulent kinetic energy, C_1 , C_2 , C_μ , σ_k and σ_ϵ are adjustable model constants. S_k and S_ϵ are additional multi-phase source terms which account for the effects of the dispersed phase. The dispersed phase, i.e. bubble phase turbulence variables are related to the continuous phase via the turbulence response function C_t as follows:

$$k_{d,i} = C_t^2 k_c, \quad (16)$$

$$\epsilon_{d,i} = C_t^2 \epsilon_c, \quad (17)$$

and

$$\nu_{d,i}^{\text{eff}} = \nu_{d,i} + C_t^2 \nu_c^t. \quad (18)$$

According to Behzadi (2001) and Rusche (2002) C_t is given by:

$$C_t(\alpha_d) = 1 + (C_{t0} - 1) \exp(180\alpha_d - 4.71 \times 10^3 \alpha_d^2 + 4.26 \times 10^4 \alpha_d^3), \quad (19)$$

where C_{t0} is obtained with the procedure described and tested by Hill (1998); Behzadi (2001); Rusche (2002). Additionally, in this work α_d is calculated as:

$$\alpha_d = \sum_{i=1}^{n_{\text{bubbles}}} \alpha_{d,i}. \quad (20)$$

3. Numerical model

The Eulerian multi-fluid model equations are discretised using collocated, cell-centred Finite Volume Method (FVM) method presented by Jasak (1996), and implemented in the `foam-extend` toolbox, the community-driven fork of OpenFOAM (Weller et al. (1998)), an open-source CFD software library for arbitrary polyhedral FVM. The solution procedure uses an adaptation of the PISO (Issa (1986)) algorithm to handle the pressure-velocity coupling.

The following sub-sections present the structure of the block-matrix, which is followed by the description of the linearisation procedure necessary for the implementation of the non-linear terms in the phase continuity equations and a brief overview of the selected discretisation schemes and linear solver settings.

3.1. Block-system structure

The structure of the block-system can be described by two levels (Uroić and Jasak (2018)). The first level represents the coupling between computational cells, which are a product of the spatial discretisation of the computational domain with control volumes, using the collocated cell-centred FVM. The second level represents the coupling between variables.

The spatial discretisation of the computational domain via FVM into N

control volumes results in a linear system of N unknowns:

$$\begin{pmatrix} a_{1,1} & a_{1,2} & \cdots & a_{1,N} \\ a_{2,1} & a_{2,2} & \cdots & a_{2,N} \\ \vdots & \vdots & \ddots & \vdots \\ a_{N,1} & a_{N,2} & \cdots & a_{N,N} \end{pmatrix} \begin{pmatrix} \phi_1 \\ \phi_2 \\ \vdots \\ \phi_N \end{pmatrix} = \begin{pmatrix} b_1 \\ b_2 \\ \vdots \\ b_N \end{pmatrix}, \quad (21)$$

where $a_{i,j}$ is a matrix entry, ϕ_i is the field value to be solved in the cell i , and b_i is the r.h.s. term for the cell i .

In the segregated approach, each matrix entry is a scalar, because every equation is solved sequentially. In the implicitly coupled approach, ϕ_i and b_i are n -dimensional vectors, where n represents the number of the implicitly coupled equations. The matrix entry $a_{i,j}$ is an $n \times n$ tensor, which models the coupling between the implicitly coupled equations.

This work investigates the implicit coupling of phase continuity equations for every phase of the presented multi-fluid system. Therefore, the number of implicitly-coupled equations is equal to $n_{\text{phases}} = n_{\text{bubbles}} + 1$, i.e. the total number of phases in the system. In this case, ϕ_i has the dimension of n_{phases} and reads:

$$\phi_i = \begin{pmatrix} \alpha_{d,1,i} \\ \alpha_{d,2,i} \\ \vdots \\ \alpha_{d,n_{\text{bubbles}},i} \\ \alpha_{c,i} \end{pmatrix}. \quad (22)$$

Each matrix entry $a_{i,j}$ is a $n_{\text{phases}} \times n_{\text{phases}}$ tensor which models the coupling between the equations:

$$a_{i,j} = \begin{pmatrix} a_{\alpha_{d,1,i},\alpha_{d,1,j}} & a_{\alpha_{d,1,i},\alpha_{d,2,j}} & \cdots & a_{\alpha_{d,1,i},\alpha_{d,n_{\text{bubbles}},j}} & a_{\alpha_{d,1,i},\alpha_{c,j}} \\ a_{\alpha_{d,2,i},\alpha_{d,1,j}} & a_{\alpha_{d,2,i},\alpha_{d,2,j}} & \cdots & a_{\alpha_{d,2,i},\alpha_{d,n_{\text{bubbles}},j}} & a_{\alpha_{d,2,i},\alpha_{c,j}} \\ \vdots & \vdots & \ddots & \vdots & \vdots \\ a_{\alpha_{d,n_{\text{bubbles}},i},\alpha_{d,1,j}} & a_{\alpha_{d,n_{\text{bubbles}},i},\alpha_{d,2,j}} & \cdots & a_{\alpha_{d,n_{\text{bubbles}},i},\alpha_{d,n_{\text{bubbles}},j}} & a_{\alpha_{d,n_{\text{bubbles}},i},\alpha_{c,j}} \\ a_{\alpha_{c,i},\alpha_{d,1,j}} & a_{\alpha_{c,i},\alpha_{d,2,j}} & \cdots & a_{\alpha_{c,i},\alpha_{d,n_{\text{bubbles}},j}} & a_{\alpha_{c,i},\alpha_{c,j}} \end{pmatrix}, \quad (23)$$

where $a_{\alpha_{d,1,i},\alpha_{d,1,j}}$ models the coupling between $\alpha_{d,1}$ in cell i with $\alpha_{d,1}$ in cell j , $a_{\alpha_{d,1,i},\alpha_{c,j}}$ models the coupling between $\alpha_{d,1}$ in cell i with α_c in cell j , etc.

3.2. Linearisation and implementation of the coupled phase continuity equations

As previously stated in section 2.3, in eq. (12) the third term on the l.h.s. is non-linear due to the product $\alpha_i \alpha_j$. Since the solution algorithm uses a linear solver, the non-linear term needs to be linearised. To make the linearisation procedure clearer, it is first performed on an arbitrary three-fluid system and is later generalised for the multi-fluid formulation. The modified phase continuity equation for the three-fluid system reads:

$$\frac{\partial \alpha_i}{\partial t} + \nabla \cdot (\bar{\mathbf{U}} \alpha_i) + \nabla \cdot \left(\alpha_i \sum_{j=1, j \neq i}^3 \alpha_j (\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j) \right) = 0, \quad (24)$$

and when $i = 1$ the non-linear term is given by:

$$\alpha_1 \sum_{j=2}^3 \alpha_j (\bar{\mathbf{U}}_1 - \bar{\mathbf{U}}_j) = \alpha_1 \alpha_2 (\bar{\mathbf{U}}_1 - \bar{\mathbf{U}}_2) + \alpha_1 \alpha_3 (\bar{\mathbf{U}}_1 - \bar{\mathbf{U}}_3). \quad (25)$$

In context of the phase continuity equation the relative velocities are treated as constants, i.e. $(\bar{\mathbf{U}}_1 - \bar{\mathbf{U}}_2) = \bar{\mathbf{U}}_{r,1,2} \neq f(\alpha_1, \alpha_2)$. Therefore, the linearisation of the first term on the r.h.s. in eq. (25) around the solution from the previous time-step/iteration reads:

$$\begin{aligned} \bar{\mathbf{U}}_{r,1,2} \alpha_1^n \alpha_2^n &\approx \bar{\mathbf{U}}_{r,1,2} \alpha_1^o \alpha_2^o \\ &+ \left(\frac{\partial (\bar{\mathbf{U}}_{r,1,2} \alpha_1 \alpha_2)}{\partial \alpha_1} \right)^o (\alpha_1^n - \alpha_1^o) \\ &+ \left(\frac{\partial (\bar{\mathbf{U}}_{r,1,2} \alpha_1 \alpha_2)}{\partial \alpha_2} \right)^o (\alpha_2^n - \alpha_2^o) \\ &\approx \bar{\mathbf{U}}_{r,1,2} \alpha_1^o \alpha_2^o + \bar{\mathbf{U}}_{r,1,2} \alpha_2^o (\alpha_1^n - \alpha_1^o) + \bar{\mathbf{U}}_{r,1,2} \alpha_1^o (\alpha_2^n - \alpha_2^o) \\ &\approx \bar{\mathbf{U}}_{r,1,2} \alpha_1^n \alpha_2^o + \bar{\mathbf{U}}_{r,1,2} \alpha_1^o \alpha_2^n - \bar{\mathbf{U}}_{r,1,2} \alpha_1^o \alpha_2^o, \end{aligned} \quad (26)$$

where superscripts n and o denote the new and old time step/iteration. The linearisation products can now be easily generalised for the multi-fluid for-

mulation:

$$\begin{aligned}
 \alpha_i \sum_{j=1, j \neq i}^n \alpha_j (\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j) &\approx \alpha_i^n \sum_{j=1, j \neq i}^n \alpha_j^o (\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j) && \text{implicit} \\
 &+ \alpha_i^o \sum_{j=1, j \neq i}^n \alpha_j^n (\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j) && \text{cross-coupling} \\
 &- \alpha_i^o \sum_{j=1, j \neq i}^n \alpha_j^o (\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j) && \text{explicit.}
 \end{aligned} \tag{27}$$

By substituting eq. (27) into eq. (12), the linearised modified phase continuity equation reads:

$$\begin{aligned}
 \frac{\partial \alpha_i}{\partial t} + \nabla \cdot (\bar{\mathbf{U}} \alpha_i^n) + \nabla \cdot \left(\alpha_i^n \sum_{j=1, j \neq i}^{n_{\text{phases}}} \alpha_j^o (\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j) \right) \\
 + \underbrace{\nabla \cdot \left(\alpha_i^o \sum_{j=1, j \neq i}^{n_{\text{phases}}} \alpha_j^n (\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j) \right)}_{\text{implicit cross-coupling}} \\
 - \nabla \cdot \left(\alpha_i^o \sum_{j=1, j \neq i}^{n_{\text{phases}}} \alpha_j^o (\bar{\mathbf{U}}_i - \bar{\mathbf{U}}_j) \right) = 0,
 \end{aligned} \tag{28}$$

where the new terms are convection terms. During the discretisation of eq. (28), special attention needs to be given to relative velocities, i.e. relative fluxes which are used to interpolate the phase fractions, to ensure that the equations are numerically equivalent (Weller (2002); Rusche (2002)). Furthermore, to ensure the boundedness of the solution, bounded convection schemes should be applied, e.g. upwind differencing if first-order accuracy is sufficient or NVD/TVD schemes (Leonard (1988); Harten (1983); Jasak

et al. (1999)) if less numerical diffusion is required.

As previously mentioned, the Eulerian multi-fluid model implemented in this work is specialised for bubbly flows. Therefore, the linearised phase continuity equation for an arbitrary number of bubble classes is implemented in the following manner. For bubble phases, the implemented phase continuity is given by eq. (28). Whereas the phase fraction of the continuous phase α_c is implemented as:

$$\alpha_c^n + \underbrace{\sum_{i=1}^{n_{\text{bubbles}}} \alpha_{d,i}^n}_{\text{implicit cross-coupling}} = 1. \quad (29)$$

All phase continuity equations, i.e. eq. (28) for all bubble phases and eq. (29) for the water phase are placed into a single block matrix and solved together in an implicitly-coupled manner.

3.3. Numerical procedure

In section 4, the model is tested for a bubbly air-water upward flow of two separate inlet streams which form a mixing layer inside a square duct. The test case has a steady-state solution, and the steady-state calculation is performed with a pseudo-transient time-marching approach (Rusche (2002)). This approach keeps the stabilising influence of the time derivative but uses a large time step to march through time, where the Courant number can be larger than unity, and only one iteration in the PISO loop is performed.

Cell to face interpolations, Laplacians and gradients were calculated using linear interpolation. Turbulence model variables were advected using the

upwind scheme. The momentum and phase fractions were advected using linear approximation, but the phase fractions employed a limiter.

The pressure equation was solved using a selection algebraic multigrid algorithm with the Gauss-Seidel smoother (Uroić and Jasak (2018)). The turbulence model equations were solved using a Bi-Conjugate Gradient Method (BiCGStab) preconditioned by DILU (Saad (2003)), and the block-coupled linear system of phase continuity equations was also solved with the BiCGStab but with the ILUC0 preconditioner (Saad (2003)). All equations were solved with an absolute tolerance of 10^{-10} for the normalised residual of the linear system.

4. Results

This section presents the test case used for the mesh refinement study (section 4.1), consistency testing of the multi-fluid formulation (section 4.2) and the validation of the implemented Eulerian multi-fluid model (section 4.3).

As previously mentioned in section 3, the implemented model is tested for a bubbly two-phase upward flow of two separate co-flowing streams which are brought together to form a mixing layer within a square duct. Roig et al. (1998) performed a series of experimental measurements for the described flow configuration, and numerous authors used this data for the validation of their multi-phase models (Hill (1998); Lathouwers (1999); Rusche (2002)).

Figure 1 schematically shows the geometry of the test case.

Figure 1: Geometry of the test case.

Values on both axes are in centimetres. The dashed lines represent the lines along which the results were sampled for comparison. The inlet conditions for the water and bubble phase are initialised with available experimental data for $x = -1$ cm, following the procedure described by Hill (1998). According to Roig et al. (1998), the bubble diameter is set to a uniform value of 2 mm. In the two-dimensional numerical test case, the splitter plate at the inlet patch is modelled as a zero-thickness wall with a 1 cm height, and the computational domain spreads from $x = -1$ cm to $x = 110$ cm to remove the influence of the outlet boundary condition on the results.

4.1. Mesh refinement study

The mesh refinement study is carried out on three structured computational grids with different levels of refinement. Table 1 shows the number of cells in each direction and the total number of cells for each mesh. Furthermore, fig. 2 shows a visual comparison of the meshes. Due to high flow gradients (i.e. velocity, phase fraction) in the splitter region, the grids have a highly increased mesh density in this region, which is gradually decreasing towards the outer walls and outlet.

Mesh	No. of cells in x -direction	No. of cells in y -direction	Total
coarse	26	20	520
intermediate	52	40	2080
fine	103	80	8240

Table 1: Computational mesh details for the mesh refinement study.

Figures 3 and 4 show the comparison of numerical results obtained with

Figure 2: Computational meshes for the mesh refinement study.

three different computational meshes, i.e. coarse, intermediate and fine mesh, for all five horizontal sampling lines which are located at $x = -1$ cm, 6 cm, 20 cm, 40 cm and 60 cm. Figure 3 shows the comparison of the averaged continuous phase velocity in the x -direction $\bar{U}_{c,x}$, and fig. 4 shows the comparison of the bubble phase fraction α_d .

Both figs. 3 and 4 show good agreement, the only significant difference is present in the high gradient region, especially at $x = -1$ cm. This deviation is mainly due to the difference of the grid density at the inlet patch on which

Figure 3: Mesh refinement study: Comparison of the averaged continuous phase velocity in the x -direction.

Figure 4: Mesh refinement study: Comparison of the bubble phase fraction.

we interpolate the available experimental data (during the initialisation procedure). The deviation is being transferred upstream and fades towards the outlet.

Furthermore, to estimate the achieved accuracy in space a freely available uncertainty estimator is used (ReFRESCO (2018)) which is based on the procedures suggested by Eça and Hoekstra (2014). The numerical uncertainty study is performed for the α_c value at the location $x = 40$ cm and $y = -1.5$ cm. This location is selected because it is sufficiently distant from the inlet and outlet boundaries, and it is also close to the $\min(\alpha_c)$ location at the $x = 40$ cm sampling line. To increase the accuracy of the estimation, three additional grids were introduced, two between the coarse and intermediate refinement level, and one between the intermediate and fine level. Table 2 presents the input data used for the evaluation and table 3 presents

Total mesh size	α_c at (40 cm, -1.5 cm)
520	0.9775
768	0.9732
1170	0.9732
2080	0.9746
4680	0.9751
8240	0.9736

Table 2: Input data for the uncertainty estimator.

the output of the uncertainty calculation. Table 3 follows the notation presented by Eça and Hoekstra (2014), where ϕ_0 denotes the extrapolated exact solution, ϕ_1 denotes the finest level solution, U_ϕ is the uncertainty estimate, and p denotes the achieved accuracy in space.

Item	ϕ_0	ϕ_1	U_ϕ	p
α_c (40 cm, -1.5 cm)	9.738×10^{-1}	9.736×10^{-1}	0.9%	2.00

Table 3: Uncertainty estimation results.

4.2. Multi-fluid consistency analysis

The consistency of the multi-fluid model is tested for the previously described test case using the intermediate mesh, but in this case, the dispersed phase is divided into multiple bubble classes. During the initialisation procedure, the experimental data for the phase fraction field on the inlet patch ($x = -1$ cm) is equally divided into multiple bubble classes with the same bubble diameter. Throughout all simulation, the continuous phase is kept constant, and only the number of bubble classes is varied. The consistency is tested using $n_{\text{phases}} = 2, 4, 8$ and 16 , i.e. $n_{\text{bubbles}} = 1, 3, 7$ and 15 .

Figures 5 and 6 show the comparison of numerical results obtained with 2, 4, 8 and 16 fluids for all five sampling lines. In both figures the difference between the four data sets is negligible, i.e. the implemented model gives consistent results for the same flow condition, independent from the number of dispersed phase classes. For all data sets in fig. 6, the α_d is calculated using eq. (20).

4.3. Validation

The implemented multi-fluid model is validated using the fine mesh from section 4.1 and is compared against experimental data by Roig et al. (1998).

Figure 7 shows the comparison of measured and calculated profiles of

Figure 5: Multi-fluid consistency: Comparison of the averaged continuous phase velocity in the x -direction.

Figure 6: Multi-fluid consistency: Comparison of the bubble phase fraction.

the averaged continuous phase velocity in the x -direction at five measuring locations. Similarly, fig. 8 shows the comparison of the bubble phase fraction. In both figures, the numerical results from the implemented model are in good agreement with the experimental data.

4.4. Performance

The performance of the derived and implemented implicitly coupled solution procedure for the phase fraction equations is compared against the standard segregated implementation (eq. (9)). The performance is assessed by comparing the behaviour of the normalised residuals profiles.

The normalised residual r , for a matrix system $A\mathbf{x} = \mathbf{b}$, is calculated in the following manner (OpenCFD (2019)):

$$r = \frac{1}{n} \sum |\mathbf{b} - A\mathbf{x}|, \quad (30)$$

where n is the normalisation factor defined as:

$$n = \sum (|A\mathbf{x} - A\bar{\mathbf{x}}| + |\mathbf{b} - A\bar{\mathbf{x}}|), \quad (31)$$

and $\bar{\mathbf{x}}$ is the arithmetic average of the solution vector $\mathbf{x}$.

It is important to emphasise that for both performance tests (4.4.1 and 4.4.2) the coupled and segregated cases employed the same computational (intermediate) grid, the solution is initialised with the same initial conditions and the calculations are performed with identical linear solver tolerances.

Figure 7: Validation: Comparison of the averaged continuous phase velocity in the x -direction.

Figure 8: Validation: Comparison of the bubble phase fraction.

4.4.1. Monodispersed test case at lower bubble phase fraction

In this subsection, the performance comparison is performed for the previously described monodispersed bubbly flow using three bubble classes. Figure 9 shows the comparison of the normalised residual profiles, where the upper plot presents the residuals for the three bubble phase fractions, and the lower one presents the residuals profiles for the mixture pressure and turbulence model variables (the phase velocity residuals are absent because the momentum equations are not solved explicitly). Solid lines correspond to the segregated phase fraction equation implementation, and the dashed lines correspond to the implicitly coupled implementation. The residual profiles for the three bubble classes are identical because all three bubble classes have the same amount of identical bubbles (2 mm in diameter). In both plots, the segregated and coupled implementation have practically identical convergence profiles for all variables. Consequently, for bubbly flows with low bubble phase fraction (in this case, approximately 2%), the implicit coupling does not offer any significant improvement in performance, because the small amount of bubbles does not have a significant influence on the flow. The following subsection presents the performance comparison for a polydispersed flow at higher bubble phase fraction.

4.4.2. Polydispersed test case at higher bubble phase fraction

The performance for the polydispersed flow at higher bubble phase fraction is tested for the previously described geometry using the intermediate

Figure 9: Performance: Comparison of residuals convergence history for the monodispersed bubbly flow.

computational grid, but in this case, the amount of bubbles is ten times larger and is equally divided into three bubble classes with the bubble diameter ranging from 1 to 3 mm. Figure 10 shows the comparison of the normalised residual profiles, again the upper plot presents the residuals for the three

bubble phase fractions, and the lower one presents the residuals profiles for the remaining variables. In both plots, the implicit coupling contributed to

Figure 10: Performance: Comparison of residuals convergence history for the polydispersed bubbly flow.

a better convergence profile, i.e. the coupled implementation managed to force the residuals below the segregated implementation practically by one

order of magnitude for all variables (especially for the mixture pressure). For bubbly flows with higher bubble phase fraction (in this case, approximately 20%), the implicit coupling contributes to a better convergence performance.

In future work, the model will account for breakup and coalescence, which considerably contributes to an additional inter-equation coupling between the bubble phase fraction equations. In that case, the implicitly coupled solution procedure should show its full potential.

5. Conclusion

This paper presents an implementation of the implicitly coupled system of phase fraction equations for the Eulerian multi-fluid model with an arbitrary number of phases. The model was implemented within the `foam-extend` toolbox, which is a community-driven fork of OpenFOAM. The implemented model was applied to the simulation of a bubbly flow within a square duct. During the mesh refinement study, it was shown that the solution remained stable on all computational grids, despite a significant difference in the total number of cells. There is an apparent discrepancy in the dispersed phase fraction profile near the centre of the duct, especially near the inlet of the domain. The discrepancy is mainly due to the difference of the grid density and interpolation of the experimental results onto the inlet patch during the solution initialisation. Furthermore, during the testing of the consistency of the multi-fluid model, the implementation proved to be insensitive to the selected number of bubble classes for the same flow conditions. The numeri-

cal results were also compared against available experimental measurements, and an overall good agreement was found. Finally, the performance comparison showed that the implicitly coupled solution procedure contributed to a better convergence of the solution for flows at higher bubble phase fraction. The presented work showed that the novel implicitly coupled solution procedure for the phase fraction equations in the Eulerian multi-fluid model gives a stable and consistent prediction of bubbly flows. The additional performance increase for more demanding flow conditions, suggests that the coupled approach is a promising candidate for the development of a robust framework for predicting the complex behaviour of polydispersed multi-phase flows. Therefore, in future work, the implicitly coupled approach will be tested for multi-phase flows which exhibit breakup and coalescence. Those models introduce additional cross-coupling of the phase fraction equations (through new source/sink terms), which should benefit from the implicitly coupled solution procedure.

Acknowledgements

This work was supported by the King Abdullah University of Science and Technology within the OSR-2017-CRG6-3409.03 research grant, and the Croatian Science Foundation (project number DOK-01-2018).

Nomenclature

In this document normal symbols represent scalar quantities, boldface Roman symbols represent vectors and boldface Greek symbols represent tensor quantities.

Latin

F	force vector	N
g	acceleration due to gravity	m s^{-2}
M	interfacial momentum transfer rate vector	$\text{kg m}^{-2} \text{s}^{-2}$
U	velocity	m s^{-1}
<i>d</i>	diameter	m
<i>k</i>	turbulence kinetic energy	$\text{m}^2 \text{s}^{-2}$
<i>n</i>	total number of phases or equations	—
<i>p</i>	pressure	Pa
<i>t</i>	time	s

Greek

α	volume fraction	—
ϵ	rate of dissipation of turbulence energy	$\text{m}^2 \text{s}^{-3}$

ν	kinematic viscosity	$\text{m}^2 \text{s}^{-2}$
ρ	density	kg m^{-3}

Dimensionless Numbers

Re Reynolds number

C_d drag coefficient

C_l lift coefficient

C_{vm} virtual mass coefficient

Superscripts

q^{eff} effective

q^n q in the new (current) time step or iteration

q^o q in the old (previous) time step or iteration

q^t turbulent

q^T transpose

Subscripts

q_c q in the continuous phase

q_d q in the dispersed phase

q_φ q in phase φ

q_i q in the i -th phase or i -th element

q_j q in the j -th phase or j -th element

Oversymbols

$\bar{q}$ ensemble average

$\bar{q}_\varphi$ conditional ensemble average in phase φ

Abbreviations

l.h.s. left hand side

r.h.s. right hand side

FVM finite volume method

References

References

B. Spalding, D., 1985. The numerical computation of multi-phase flows. NASA STI/Recon Technical Report N .

Behzadi, A., 2001. Turbulence modelling at high phase fractions. Technical Report.

Carver, M.B., 1982. Method of Limiting Intermediate Values of Volume Fraction in Iterative Two-Fluid Computations. Journal of Mechanical Engineering Science 24, 221–224.

Crowe, C., Schwarzkopf, J., Sommerfeld, M., Tsuji, Y., 2011. Multi-

- phase Flows with Droplets and Particles, Second Edition. CRC Press. doi:10.1201/b11103.
- Dinh, T.N., Nourgaliev, R.R., Theofanous, T.G., 2003. Understanding of the Ill-posed two-fluid model, in: The 10th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-10), Seoul, Korea,.
- Dopazo, C., 1977. On conditioned averages for intermittent turbulent flows. *Journal of Fluid Mechanics* 81, 433–438. doi:10.1017/S0022112077002158.
- Durst, F., Milojevic, D., Schönung, B., 1984. Eulerian and Lagrangian predictions of particulate two-phase flows: a numerical study. *Applied Mathematical Modelling* 8, 101–115. doi:10.1016/0307-904X(84)90062-3.
- Eça, L., Hoekstra, M., 2014. A procedure for the estimation of the numerical uncertainty of CFD calculations based on grid refinement studies. *Journal of Computational Physics* 262, 104–130. doi:10.1016/j.jcp.2014.01.006.
- Fernandes, C., Vukčević, V., Uroić, T., Simoes, R., Carneiro, O.S., Jasak, H., Nóbrega, J.M., 2019. A coupled finite volume flow solver for the solution of incompressible viscoelastic flows. *Journal of Non-Newtonian Fluid Mechanics* 265, 99–115. doi:10.1016/j.jnnfm.2019.01.006.
- Ferreira, G.G., Lage, P.L., Silva, L.F., Jasak, H., 2019. Implementation of an implicit pressure-velocity coupling for the Eulerian multi-fluid model. *Computers and Fluids* 181, 188–207. doi:10.1016/j.compfluid.2019.01.018.

- Gosman, A.D., Lekakou, C., Politis, S., Issa, R.I., Looney, M.K., 1992. Multidimensional modeling of turbulent two-phase flows in stirred vessels. *AIChE Journal* 38, 1946–1956. doi:10.1002/aic.690381210.
- Haberman, W.L., Morton, R.K., 2011. An experimental investigation of the drag and shape of air bubbles rising in various liquids / by W. L. Haberman and R. K. Morton. Technical Report. doi:10.5962/bhl.title.47521.
- Harten, A., 1983. High resolution schemes for hyperbolic conservation laws. *Journal of Computational Physics* 49, 357–393. doi:10.1016/0021-9991(83)90136-5.
- Hill, D.P., 1998. The Computer Simulation of Dispersed Two-phase Flows. Ph.D. thesis.
- Hirt, C., Nichols, B., 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics* 39, 201–225. doi:10.1016/0021-9991(81)90145-5.
- Hoyas, S., Gil, A., Margot, X., Khuong-Anh, D., Ravet, F., 2013. Evaluation of the Eulerian-Lagrangian Spray Atomization (ELSA) model in spray simulations: 2D cases. *Mathematical and Computer Modelling* 57, 1686–1693. doi:10.1016/j.mcm.2011.11.006.
- Ishii, M., Hibiki, T., 2006. Thermo-fluid dynamics of two-phase flow. doi:10.1007/978-0-387-29187-1, arXiv:arXiv:1011.1669v3.

- Ishii, M., Hibiki, T., 2011. Thermo-Fluid Dynamics of Two-Phase Flow. doi:10.1007/978-1-4419-7985-8.
- Issa, R.I., 1986. Solution of the implicitly discretised fluid flow equations by operator-splitting. *Journal of Computational Physics* 62, 40–65. doi:10.1016/0021-9991(86)90099-9.
- Jareteg, K., Ström, H., Sasic, S., Demazière, C., 2017. On the dynamics of instabilities in two-fluid models for bubbly flows. *Chemical Engineering Science* 170, 184–194. doi:10.1016/j.ces.2017.03.063.
- Jasak, H., 1996. Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows. Ph.D. thesis.
- Jasak, H., Weller, H.G., Gosman, A.D., 1999. High resolution NVD differencing scheme for arbitrarily unstructured meshes. *International Journal for Numerical Methods in Fluids* 31, 431–449. doi:10.1002/(SICI)1097-0363(19990930)31:2<431::AID-FLD884>3.0.CO;2-T.
- Kissling, K., Springer, J., Jasak, H., Schütz, S., Urban, K., Piesche, M., 2010. A coupled pressure based solution algorithm based on the volume-of-fluid approach for two or more immiscible fluids. *European Conference on Computational Fluid Dynamics*, 14–17.
- Lathouwers, D., 1999. Modelling and simulation of turbulent bubbly flow. Ph.D. thesis. TU Delft.

- Launder, B.E., Spalding, D.B., 1974. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering* 3, 269–289. doi:10.1016/0045-7825(74)90029-2, [arXiv:1204.1280v1](#).
- Leonard, B.P., 1988. Simple high-accuracy resolution program for convective modelling of discontinuities. *International Journal for Numerical Methods in Fluids* 8, 1291–1318. doi:10.1002/fld.1650081013.
- Oliveira, P.J., Issa, R.I., 2003. Numerical aspects of an algorithm for the Eulerian simulation of two-phase flows. *International Journal for Numerical Methods in Fluids* 43, 1177–1198. doi:10.1002/fld.508.
- OpenCFD, 2019. OpenFOAM: User Guide v1906.
- Osher, S., Sethian, J.A., 1988. Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations. *Journal of Computational Physics* 79, 12–49. doi:10.1016/0021-9991(88)90002-2.
- Panicker, N., Passalacqua, A., Fox, R.O., 2018. On the hyperbolicity of the two-fluid model for gasliquid bubbly flows. *Applied Mathematical Modelling* 57, 432–447. doi:10.1016/j.apm.2018.01.011.
- Politis, S., 1989. Prediction of two-phase solid-liquid turbulent flow in stirred vessels. January, University of London.
- Ransom, V., Hicks, D., 1984. Hyperbolic two-pressure models for two-phase flow. *Journal of Computational Physics* 53, 124–151. doi:10.1016/0021-9991(84)90056-1.

- ReFresco, 2018. A community based open-usage and open-source CFD code for the Maritime World.
- Roig, V., Suzanne, C., Masbernat, L., 1998. Experimental investigation of a turbulent bubbly mixing layer. *International Journal of Multiphase Flow* 24, 35–54. doi:10.1016/s0301-9322(97)00046-3.
- Rusche, H., 2002. Computational Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions. Ph.D. thesis.
- Saad, Y., 2003. *Iterative Methods for Sparse Linear Systems*. 2nd ed., Society for Industrial and Applied Mathematics, Philadelphia, PA, USA.
- Silva, L.F., Lage, P.L., 2011. Development and implementation of a polydispersed multiphase flow model in OpenFOAM. *Computers and Chemical Engineering* 35, 2653–2666. doi:10.1016/j.compchemeng.2011.04.011.
- Subramaniam, S., 2013. LagrangianEulerian methods for multiphase flows. *Progress in Energy and Combustion Science* 39, 215–245. doi:10.1016/J.PECS.2012.10.003.
- Sussman, M., Smereka, P., Osher, S., 1994. A Level Set Approach for Computing Solutions to Incompressible Two-Phase Flow. *Journal of Computational Physics* 114, 146–159. doi:10.1006/JCPH.1994.1155.
- Uroić, T., Jasak, H., 2018. Block-selective algebraic multigrid for implicitly coupled pressure-velocity system. *Computers and Fluids* 167, 100–110. doi:10.1016/j.compfluid.2018.02.034.

Weller, H.G., 2002. Derivation modelling and solution of the conditionally averaged two-phase flow equations. Technical Report Tech. Rep.. Nabla Ltd.

Weller, H.G., Tabor, G., Jasak, H., Fureby, C., 1998. A tensorial approach to computational continuum mechanics using object-oriented techniques. Computers in Physics 12, 620. doi:10.1063/1.168744.