Hydrodynamics study of bubbly flow in a top-submerged lance vessel

Yannan Wang ¹, Maarten Vanierschot ², Lingling Cao ^{3*}, Zhongfu Cheng ¹, Bart Blanpain ¹, Muxing Guo ^{1*}

1. Department of Materials Engineering, KU Leuven, 3000 Leuven, Belgium.

2. Mechanical Engineering Technology Cluster TC, Campus Group T Leuven, KU Leuven, 3000 Leuven, Belgium.

3. State Key Laboratory of Advanced Metallurgy, University of Science and Technology Beijing, 100083 Beijing, China.

^{1*} Corresponding author. Tel.: +32 16321183; Email address: muxing.guo@kuleuven.be (M. Guo);

^{3*} Corresponding author. Tel.: +86 15652934497; Email address: meynell0809@gmail.com (L. Cao);

Abstract

Free surfaces of large bubbles are involved in gas injection into liquid through a top-submerged lance. The Volume of Fluid (VOF) multiphase model is chosen to simulate these flows. However, the selection of an appropriate turbulence model remains a point of discussion. Therefore, three representative turbulence models, namely the Renormalization Group (RNG) k - ϵ model, the Reynolds Stress model (RSM) and the Large Eddy Simulation (LES) model, respectively coupled with VOF simulations, are studied. The results are compared with Particle Imaging Velocimetry (PIV) experiments. Afterwards, the selected turbulence model is used to study the two-phase flow characteristics. It has been found that the three turbulence models are able to capture the main averaged features of the flow such as the flow structures and average velocity field. However, the RNG and RSM models are not capable of estimating the fluctuating flow parameters such as the Reynolds stresses. In contrast, the LES model gives very good agreement with experimental data. The subsequent study on flow characteristics also showed that the 3D VOF-LES model can describe the bubble behavior and surface fluctuation. Resorting to Fast Fourier Transform (FFT) method, it has been proven that the bubble frequency is around 20 Hz and the turbulence in the bubbly region is between the single phase $-5/3$ and two-phase $-10/3$ power law in the inertial sub-range. Finally, the effect of the contact angle between the formed bubbles and outer lance wall on the flow has been estimated. The results showed that contact angle has limited influence on the flow.

27 Keywords: top-submerged lance injection; VOF; RNG/RSM/LES; hydrodynamic validation; bubble
28 behavior; flow characteristics.

29 Nomenclature

30	$C_{1\varepsilon}, C_{2\varepsilon}, C_{3\varepsilon}$	constants, 1.42, 1.68, 0 or 1, respectively.
31	C_μ	constant, 0.0845.
32	$D_{T,ij}$	turbulent diffusion, $\text{kg/m}\cdot\text{s}^3$.
33	F_{csf}	surface tension force, N.
34	g	gravitational acceleration, m/s^2 .
35	G_b	turbulence generation due to buoyancy, $\text{kg/m}\cdot\text{s}^3$.
36	G_{ij}	buoyancy production, $\text{kg/m}\cdot\text{s}^3$.
37	G_k	turbulence generation due to the mean velocity gradients, $\text{kg/m}\cdot\text{s}^3$.
38	k	turbulent kinetic energy, m^2/s^2 .
39	K	curvature, 1/m.
40	n	phase number.
41	p	pressure, Pa.
42	P_{ij}	stress production, $\text{kg/m}\cdot\text{s}^3$.
43	R_ε	R - ε term, $\text{kg/m}\cdot\text{s}^4$.
44	S	momentum source term, N/m^3 .
45	u	velocity, m/s.
46	<i>Greek symbols</i>	
47	α	phase volume fraction.
48	$\alpha_k, \alpha_\varepsilon$	inverse effective Prandtl numbers.
49	ε	turbulent energy dissipation rate, m^2/s^3 .
50	μ	molecule viscosity, $\text{Pa}\cdot\text{s}$.
51	μ_{eff}	effective viscosity, $\text{Pa}\cdot\text{s}$.
52	ρ	density, kg/m^3 .

53	σ	surface tension coefficient.
54	$\bar{\tau}$	stress tensor. N/m ² .
55	ϕ_{ij}	pressure strain, kg/m ³ ·s ³ .
56	<i>Subscripts</i>	
57	q	q^{th} phase.
58	i, j, k	coordinate direction indices.

59 1. Introduction

60 Gas-liquid two-phase flows are widely present in industrial processes. The gas phase is usually
61 injected from the bottom or top into the liquid. For instance, gas is injected from the bottom in bubble
62 columns and metallurgical ladles, and it is injected from the top in bath smelting processes, such as
63 Ausmelt [1-4] and Hismelt [5]. Compared to the extensive investigations on gas injection into a
64 bubble column [6-19] and a metallurgical ladle [20-23], gas injection from the top is different in two
65 aspects. The first one is that the gas is injected through a top-submerged lance with an open hole
66 instead of being sparged through a perforated plate or multi-hole plugs from the bottom, therefore, the
67 bubbles formed are several times larger. The second one is that the liquids, *e.g.*, zinciferous slags in
68 bath smelting furnaces and basic oxygen furnace slags in slag pots, normally have a higher viscosity.
69 During the top injection, gas bubbles are generated and then rise up, driving the surrounding liquid to
70 move. Therefore, a certain flow pattern can be formed, which has a significant influence on the
71 transfer efficiency of momentum, mass and energy. As a consequence, the investigation with respect
72 to the fundamental hydrodynamic properties of the two-phase flow in a top-submerged lance vessel
73 has drawn considerable attention, not only due to its theoretical significance, but also due to the
74 importance of its industrial applications.

75 Currently, physical experiments and numerical simulations are performed to study the two-phase flow
76 in a top-submerged lance vessel. Iguchi et al. [24] conducted air-water cold model experiments to
77 investigate the fluid flow phenomena in a top-submerged lance vessel. The experimental results
78 indicated a critical condition for the cessation of the swirl motion. An empirical correlation for the

vertical migration distance of bubbles was derived as a function of the modified Froude number. They also observed a dead region near the bottom of the bath. Morsi et al. [25] experimentally studied the gas injection into a top-submerged lance vessel to gain an understanding of the mixing process by means of water modelling. A Dantec 2D laser Doppler anemometer system was used to obtain the flow information in their study. The results showed that the swirl gas injection can improve the mixing, and two thirds lance submergence can result in a better mixing for both swirl and non-swirl gas injection. In addition to the experimental work, numerical simulations have also contributed to the understanding of the two-phase flow. For the top gas injection, the Euler-Euler model [3,4, 26-29] is mainly employed. Schwarz et al. [30], pioneers in this research topic, built a two-fluid model to investigate the fluid flow in a top-submerged lance vessel. The predicted void fractions were compared with the measured ones from a water model of the system. The important features such as back-penetration into the lance and the penetration depth of the swirling jet were well simulated. Liovic et al. [27, 28] built a 2D VOF model incorporating a volume tracking method based on a piecewise linear interface calculation approach and a monotonic, flux-limited advection scheme playing the role of an implicit sub-grid scale turbulence model. Their work discussed bubble behavior rising in water and liquid pig iron and other relevant pyrometallurgical applications. The results also highlighted the applicability of the VOF model in metallurgical applications such as matte entrainment, splash, bubble behavior and gas injection. However, the turbulence method models the energy dissipation from the resolved-scale eddies to the subgrid-scale eddies in an empirical way and it is not able to model the energy transfer from the subgrid scale to the resolved scale. These limitations can affect the accuracy of the model. Liovic and Lakehal [29] also investigated a large scale bubbling process driven by the downward injection of air into a water pool using VOF-LES approach. Results including the transitional mode of bubble growth, interfacial area evolution, interface/turbulence interactions, coherent structures, energy spectra and dissipation have been documented in detail. The LES model has even been validated by the comparison of the level of upper bubble surface during the start-up bubble formation with the experimental results from Meier et al. [31], demonstrating the predictive capability of the VOF-LES approach for the topology change of interfaces. However, a comprehensive validation, especially for the turbulence quantities (*e.g.*

fluctuating velocity, turbulent kinetic energy), are still needed for the study of turbulence phenomena in the two-phase flow. Huda et al. [3, 4] studied a top-submerged lance smelting process for zinc fuming by using a commercial Computational Fluid Dynamic (CFD) package AVL Fire (AVL, Graz, Austria). This is a study on a high viscosity liquid *i.e.* a slag as opposed to the previous example for a water flow, and very comprehensive information for industrial applications has been obtained. However, the Eulerian model used in their work is not able to give a quantitative analysis of the splash formation due to the absence of the surface tracking methods such as the VOF. Besides, discrepancies appeared between the mean tangential velocities obtained from simulations and experiments. This may be caused by the limitations of the standard $k-\varepsilon$ turbulence model used in their work. The selection of the multiphase model and turbulence model and an appropriate combination of these models is of high importance for representing the flow characteristics. Therefore, a generalized validation of the fundamental aspects of the models is imperative. Regarding top gas injection in this study, a tracking model for the unsteady behavior of large bubbles and the validation of the performance of different turbulence models are of great interest, as well as the in-depth hydrodynamics in a high viscosity liquid. To the best of our knowledge, these aspects have not been covered in the open literature.

In this paper, 2D and 3D VOF models capable of tracking the unsteady behavior of the gas-liquid interface are adopted to describe the free surfaces of large bubbles in the top-submerged gas injection. The VOF model, respectively combined with three representative turbulence models, *i.e.*, the RNG, the RSM and the LES, is qualitatively and quantitatively validated by corresponding PIV experiments. The flow characteristics are subsequently investigated by a coupling of the VOF technique and the selected turbulence model. This study can shed light on the model selection and the hydrodynamics of the two-phase flow, and can provide reference for the fundamental validation of different turbulence models. Additionally, it also lays the foundation for the following work of pneumatic injection of additives into molten slag through a top-submerged lance and the additives transport in a slag pot during slag treatment, since the turbulence generated by the gas injection has influence on the additive movement.

2. Numerical Methodology

2.1. Multiphase model

As mentioned before, large bubbles are formed during the gas injection. Therefore, a robust method capable of gaining the transient interface information is needed. The VOF model with advanced interface-tracking method is employed. For this two immiscible fluids system a parameter, the phase volume fraction, is introduced to track the interface position. The volume fraction of the q^{th} phase, α_q , is zero when the cell is empty of the q^{th} phase; α_q is unity when the cell is entirely occupied by the q^{th} phase; α_q is between zero and unity when the interface exists in the cell. The tracking of the instantaneous interface can be realized by solving a continuity equation for the volume fraction of one of the phases (q^{th} phase) as,

$$\frac{\partial}{\partial t}(\alpha_q \rho_q) + (\alpha_q \rho_q \vec{u}_q) = 0 \quad (1)$$

The volume fraction of the primary phase is not attained from this equation, while it is computed based on the following constraint:

$$\sum_{q=1}^n \alpha_q = 1 \quad (2)$$

It is also noteworthy that the variables and properties are shared by all phases and represent volume-averaged values, which means the variables and properties in each cell is either purely representative of one of the phases, or representative of a mixture of the phases, depending on the volume fraction value.

The explicit Geometric Reconstruction scheme [32] is adopted to track the interface. The advantage of this scheme is that a special interpolation treatment based on a piecewise linear approach is applied to the cell near the interface. It is assumed that the interface between the two fluids has a linear slope within each cell, and the linear shape is used to calculate advection and diffusion fluxes, as well as volume fraction. It has been considered that this scheme is the more accurate one due to its less significant numerical diffusion [33, 34].

Apart from the continuity equation, the governing equations also include momentum conservation equations. Since the variables are shared by all phases, there is only one single set of momentum equations, which is listed as follows:

$$\frac{\partial}{\partial t}(\rho \vec{u}) + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla p + \nabla \cdot \bar{\tau} + \rho \vec{g} + \vec{F}_{csf} \quad (3)$$

The velocity in equation (3) is defined as:

$$\vec{u} = \langle \vec{u} \rangle + \vec{u}' \quad (4)$$

where $\vec{u}$, $\langle \vec{u} \rangle$ and $\vec{u}'$ represents instantaneous velocity, resolved and unresolved velocities, respectively. The variables, $\langle \vec{u} \rangle$ and $\vec{u}'$, can be interpreted as time-averaged velocity and fluctuating velocity as a result of Reynolds-averaging the Navier-Stokes equations, and can also be referred to as large scale component and sub-grid scale component if the equation (3) is obtained by a filtering scheme.

The bubble behavior is closely related to the surface tension. Hence, the surface tension force is considered and added as a source term in the momentum equation. The model used to calculate surface tension is the Continuum Surface Force (CSF) model developed by Brackbill *et al* [35]. The surface tension force is derived by:

$$\vec{F}_{csf} = \sigma K \nabla \alpha \quad (5)$$

2.2. Turbulence models

2.2.1. RNG model

In order to solve the Reynolds stress terms $(\overline{\rho u'_i u'_j})$, the RNG k - ε turbulence model is introduced. The form of this model is similar to that of the standard k - ε model developed by Launder and Spalding [36]. The difference is that a modification based on a dynamic Renormalization Group method was carried out to improve its performance in the RNG model [37]. As a consequence, the RNG model has some advantages over the standard k - ε model for rapidly strained and swirling flows. In this model, transport equations of the two turbulence quantities, turbulent kinetic energy k and its dissipation rate

ε , are calculated. Based on the results of the calculations, the eddy viscosity is computed. The transport equations of k and ε are presented in equations (6) and (7), in which the terms, G_k and G_b , represent the generation of turbulent kinetic energy due to the mean velocity gradient and buoyancy, respectively. The term, R_ε , makes the RNG model more responsive to the effects of rapid strain and streamline curvature than the standard k - ε model.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon \quad (6)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + G_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon \quad (7)$$

2.2.2. Reynolds stress model

There is another way to solve the Reynolds stress terms by directly constructing the transport equations for each stress term, together with an equation for the dissipation rate. This approach is known as the RSM model. In the RSM model, the turbulent kinetic energy can be obtained from the solved values of normal stress. The Reynolds stress transport equation reads:

$$\frac{\partial}{\partial t}(\rho \overline{u'_i u'_j}) + \frac{\partial}{\partial x_k}(\rho u_k \overline{u'_i u'_j}) = \frac{\partial}{\partial x_k} \left[\mu \frac{\partial}{\partial x_k} (\overline{u'_i u'_j}) \right] - P_{ij} + D_{T,ij} + G_{ij} + \phi_{ij} + \varepsilon_{ij} \quad (8)$$

The terms from left to right represent local time derivative, convection, molecular diffusion, stress production, turbulent diffusion, buoyancy production, pressure strain and dissipation, respectively.

2.2.3. LES model

The governing equations for the LES model are derived by filtering the instantaneous Navier-Stokes equations via the built-in filtered approach. After that, the whole turbulence scale is divided into two parts, which are the large scale component and the sub-grid scale component, respectively. Then the large scale ones are directly resolved by the filtered equations, and the sub-grid scale ones are modelled by the developed Sub-Grid Scale (SGS) models. The SGS model adopted in this work is Wall-Adapting Local Eddy-Viscosity (WALE) model developed by Nicoud et al [38]. The WALE model has the advantage of correctly treating the laminar zone in the domain. It makes this model

preferable for our case in which there is complete turbulent behavior in the middle of the domain and a laminar zone near the wall due to the high liquid viscosity.

2.3. Numerical setup

The simulations are implemented in the commercial software package ANSYS FLUENT 16.2. The geometrical information used in the simulations is shown in Table 1. The geometrical model is depicted in Fig. 1. The whole computational domain includes the vessel space and the inner area of the lance. As seen in Fig. 1, the domain is patched by two zones, *i.e.*, the gas zone and the liquid zone. For 3D full simulations, the whole domain is adopted. For the 2D simulations, the median plane (X-Z plane) is adopted, same as the laser sheet generated in the PIV experiments. The velocity inlet and pressure outlet boundary conditions are employed. The liquid surface is set to interface, and the surface variation can be observed in the simulations. The rest of the boundaries are set to walls. All information about boundary conditions is indicated in Table 2. Structured meshes with a different numbers of nodes were built using ICEM. In the 2D simulations, three meshes with number of nodes of 4000 (around 3800 quadrilateral cells), 6000 (around 5750 quadrilateral cells) and 10000 (around 9700 quadrilateral cells) are used. The range of the mesh size is from 0.2 mm to 4 mm. In the full 3D simulations, three meshes are used to perform a mesh sensitivity study. The numbers of nodes are 350000 (around 334500 hexahedral cells), 500000 (around 480300 hexahedral cells) and 820000 (around 792800 hexahedral cells), respectively. The range of the mesh size is from 0.36 mm to 5 mm. Representatives from 3D and 2D meshes are given in Fig. 2. In addition, the operating parameters and materials properties needed in the simulations are also listed in Table 1.

Table 1 Geometrical information, operating parameters and material properties used in the simulations.

Dimensions of the vessel, m	$0.135 \times 0.135 \times 0.195$
Inner diameter of the lance, m	0.005
Outer diameter of the lance, m	0.008
Length of the lance, m	0.071
Submerged depth, m	0.044
Height of oil level, m	0.168

Air flow rate, NL/h	300
Inlet velocity, m/s	4.246
Density of paraffin oil, kg/m ³	880
Viscosity of paraffin oil, kg·m ⁻¹ ·s ⁻¹	0.065
Density of air, kg/m ³	1.225
Viscosity of air, kg·m ⁻¹ ·s ⁻¹	1.789×10 ⁻⁵
Interfacial tension of oil/air, N/m	0.026

Fig. 1 Geometry of the vessel and boundary conditions.

Fig. 2 3D mesh with 500000 nodes (left) and 2D mesh with 6000 nodes (right).

Table 2 Boundary conditions used in the numerical setup.

Inlet condition	Velocity inlet
Outlet condition	Pressure outlet
Lance wall, side and bottom wall conditions	Stationary and no-slip wall
Symmetry axis (2D)	Axis

2.4. Solution strategy

For the numerical solution methods in the RNG and RSM simulations, the PISO (Pressure-Implicit with Splitting of Operators) scheme [39] is adopted for velocity-pressure coupling, and the PRESTO! (PREssure Staggering Option) and second order upwind schemes are employed for the spatial discretization. For the LES simulation, the scheme of SIMPLEC (Semi-Implicit Method for Pressure-Linked Equations - Consistent) [40] is used for velocity-pressure coupling. The pressure and momentum equations are spatially discretized by the PRESTO! and bounded central differencing schemes, respectively. Convergence is monitored by means of the unscaled residual of the continuity equation. In this study, it is considered that the residual of the continuity equation should drop at least 3 orders of magnitude to guarantee a sufficient accuracy. The convergence criterion less than 1×10^{-8} is chosen. The time step is 1×10^{-4} s.

The calculations were submitted to a High Performance Computing cluster. One node with 20 processors was required, and it took around two months to finish all the calculations.

3. Experimental Validation

In order to validate the combined VOF-RNG/RSM/LES model, TR-PIV (Time Resolved Particle Imaging Velocimetry) experiments were performed. Paraffin oil and compressed air were used to form a two-phase system. The paraffin oil was used to simulate high viscosity liquids, for example, metallurgical slags. The cubic vessel is made of transparent glass, and a glass pipe used as the top lance is placed in the center of the vessel. All the information used in the PIV experiment is the same as that listed in Table 1. The experimental setup is shown in Fig. 3.

Fig. 3 Schematic of the experimental setup and the measured area (58 mm × 58 mm): A (-0.065 m, 0.164 m); B (-0.007 m, 0.164 m); C (-0.007 m, 0.106 m); D (-0.065 m, 0.106 m).

Since the selected plane (X-Z median plane shown in Fig. 3) is symmetric, the flow characteristics are assumed to be the same on both sides. Therefore, only the flow information on one side is measured. It can also be easily predicted that the liquid mainly flows in the upper part based on two considerations: 1) the upper part liquid is driven by the rising gas bubbles; and 2) the injected gas cannot penetrate very deeply, so the liquid in the lower part is almost still. Therefore, the selected area to be measured is located in the upper left part of the median plane, as shown in Fig. 3. A PIV laser (wavelength 527 nm, pulse energy 10 mJ @ 1000 Hz) from NewWave is used to generate a laser sheet with a thickness of 0.5 mm overlapping the median plane of the vessel. The flow is seeded with hollow PMMA particles of diameters between 20 and 50 μm . The particles are coated by Rhodamine B to exhibit fluorescence, which can help increase the signal to noise ratio when the back light of the laser sheet is filtered out by an optical filter. In order to correct for light refraction, the PIV system was calibrated by placing a calibration plate in the measured flow field, and the plate exactly overlaps the laser sheet. The images are recorded by a CMOS camera (HighSpeedStar 5) with a resolution of 1024×1024 pixels. The calculation of velocity vector is processed by the Davis 8.1 software (LaVision GmbH). Detailed information can be found in the work of Vanierschot et al [41]. Further data processing is executed by Matlab software (R2016b).

4. Results and Discussion

4.1. Flow field

In order to clearly understand the flow and to prepare for the data extraction position, an experimental flow pattern in the measured area is given in Fig. 4. Considering that the lance is placed in the middle, bubbles generated from the lance tip rise up along the lance, driving the liquid near this region to rise to the surface and pushing the liquid towards the wall. The liquid hits the wall and rebounds back, resulting in a counter-clockwise rotating vortex in the measured zone, as seen in Fig. 4. This recirculation is a characteristic feature in this top-submerged lance injection, and it is very important for mixing. For example, fine particles are also injected into the liquid, the particles may flow along

the recirculation and be transported to somewhere else. Therefore, the size and position of this vortex are important. However, different models may have different simulated vortices, and the flow velocity around the vortex is also different. Given the aforesaid variables, using data from a fixed geometrical height is not comparable. Hence, a line through the vortex center is used as the position of data extraction for the quantitative comparisons in the following sections.

Fig. 4 Experimental flow pattern (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall; h : axial distance; H : height of the measured zone).

4.2. Independency analysis

In order to save computational cost and to obtain stable statistics, it is necessary to perform an independency analysis of the simulations for simulation time and mesh size. As the LES turbulence model requires substantially finer mesh and sufficiently longer time compared to the Reynolds-averaged Navier-Stokes (RANS) methods, the analysis is done for the 2D axisymmetric LES simulations.

Fig. 5 Average axial velocity (a) and average radial velocity (b) along the line through the vortex center at different times (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall).

Fig. 6 Average axial velocity (a) and average radial velocity (b) along the line through the vortex center for different numbers of nodes (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall).

The results of the convergence of statistics are presented in Fig. 5. The average velocities extracted from the line through the vortex center ($h = 0.151$ m) are used for comparison. It is obvious from Fig. 5(a) that a considerable axial velocity variation exists at the very early stage of the calculation, then the axial velocity quickly reaches stable values when the calculated time exceeds 5 s. A small

difference within 5% between 5 s and 140 s is observed, which means the axial velocity for our case is less dependent on time. On the other hand, the radial velocity is quite different, seen in Fig. 5(b). Even when the calculation time is over 90 s, the radial velocity is still changing in the region near the wall. After time 120 s, the radial velocity becomes time-independent. In summary, the calculations need to be run over 120 s, so as to achieve stable velocity information. Therefore, a calculation time of 140 s was chosen.

For the mesh independency analysis, three types of mesh, *i.e.*, numbers of nodes of 4000, 6000 and 10000, were evaluated. As shown in Fig. 6(a), only small difference of the average axial velocity within 5% can be found with the different meshes. While for the average radial velocity in Fig. 6(b), the variation within 15% is slightly larger compared to that of the axial velocity, but still can be neglected compared to the radial velocity magnitude. Therefore, it can be concluded that the mesh with the number of nodes over 4000 is sufficient for a 2D axisymmetric calculation. For the sake of precise calculation, the mesh with the number of nodes of 6000 was selected for the following calculations in this work.

4.3. 2D and 3D simulations

A much larger computational cost is needed for 3D simulations than for the 2D simulations. For the sake of saving computational cost, the comparison between the 2D axisymmetric simulation and 3D simulation was performed to evaluate the effect of the dimension on the flow. As noted in the section 4.2, the axial velocity can quickly attain a steady value. Hence, this parameter is used for the comparison. The 3D simulation was performed for 20 s, which is much longer than the simulation time needed for the convergence of the axial velocity. Before the comparison, the grid study for the 3D simulations was carried out to avoid an effect of the mesh on the calculations. Three types of mesh were used, and the results are shown in Fig. 7. It can be clearly seen that the obtained average axial velocities are very close, which means that the mesh used in our 3D simulations has very limited influence on the simulations. Therefore, the mesh with number of nodes of 350000 was adopted for the comparison between the 2D and 3D simulations. Fig. 8 shows the results, where both of the 2D and 3D simulations agree with the experimental data, and only minor differences can be detected

between 2D and 3D simulations. Therefore, it is reasonable to conclude that the 2D simulation has similar outcomes as the 3D simulation, and the 2D simulation is sufficiently accurate to represent the flow. In addition, a 2D simulation can save lots of computational time. In this context, the 2D simulation is used for the following hydrodynamics validation.

Fig. 7 Average axial velocity along the line through the vortex center for different numbers of nodes (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall).

Fig. 8 Average axial velocity along the line through the vortex center in 2D and 3D simulations (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall).

4.4. Qualitative validation of velocity field

The numerical simulations with the RNG, RSM and LES models are validated with experimental data by comparing the velocity and vortex shape and position. Fig. 9 demonstrates the velocity vector fields and velocity contour profiles obtained from the experiments and simulations. For both the

experimental and calculation results the counter-clockwise rotating vortex in the measured zone is formed. This main simulation flow pattern embodied by the macroscopic recirculation corresponds very well with the experiments. In addition, Fig. 9 also shows the presence of a large velocity gradient in the flow from right to left both in the experiments and the simulations of this work. This can be attributed to the characteristics of the top-submerged lance injection. The region on the right is close to the lance, and the generated bubbles drive the liquid in this region to rise up. This phenomenon leads to a large velocity in this bubbly zone, and this velocity can be dissipated by the viscous liquid to a small velocity in the wall zone. The discrepancies between experiments and simulations are reflected by the velocity magnitude in the bubbly zone and the vortex position and shape. The numerical velocity magnitudes in the bubbly zone are larger than the experimental ones, as seen in the contour profiles. This non-conformity is likely to be caused by gas bubbles in the PIV experiments. The gas bubbles can pose a negative influence on the measurements of the liquid velocity due to the refractive issues, and this makes the measured value in this region less reliable. Another contributing factor for the discrepancy is that the velocity in the VOF model is volume-averaged. As the PIV only measures the liquid velocity and the gas velocity is larger than the liquid velocity in the bubbly zone, resulting in a larger numerical velocity in comparison to the experimental values. With respect to the vortex shape, Fig. 9 clearly shows almost a regular circle in the experiment and the LES model, and oblate in the RNG and RSM models. Besides that, the vortex size is similar between the experiment and the LES model. However, the size is a little larger in the RNG and RSM models. For a direct comparison of the vortex position, the coordinates of the vortex center in experiment and simulations are listed in Table 3. The error is also calculated. According to the results, the LES model has a large advantage over the RNG and RSM models in predicting the x position of the vortex center, while the RSM model is slightly better in predicting the y position of the vortex center.

Fig. 9 Comparison of the experimental and numerical velocity fields (a) Experimental; (b) RNG model; (c) RSM model; (d) LES model (r: radial distance from the axial line; R: radial distance between the axial line and the vessel wall; h: axial distance; H: height of the measured zone).

Table 3 Positions of the vortex center and the calculated error.

Method	r/R	x-error/%	h/H	y-error/%
PIV	0.696	-	0.690	-
RNG	0.459	27.6	0.776	8.6
RSM	0.519	20.6	0.621	6.9
LES	0.711	1.7	0.776	8.6

4.5. Quantitative validation

4.5.1. Comparison of velocity and fluctuating components

Fig. 10(a) and (b) show the average velocity and axial velocity profiles obtained from the PIV experiment, and the RNG, RSM and LES models, respectively. Compared to the experimental data, all the three turbulence models coupled with the VOF model are able to exhibit the velocity trend that is the decrease of the velocity from the lance to the wall. However, there is a large deviation in the bubbly zone (around $r/R < 0.25$). The reason for this deviation has been given in the last section, which is the unreliability of the experimental data and the overestimation of the numerical velocity caused by the gas bubbles. This point can be proven by Fig. 11. It can be clearly seen that the presence of gas phase represented by the gas volume fraction is consistent with the deviations appearing in the average velocity and axial velocity profiles. Except for that part, the velocity and its axial component from the LES model agree very well with the experimental data. However, discrepancies can be found between the RNG/RSM model and experiment.

Fig. 10 Average velocity (a) and average axial velocity (b) along the line through the vortex center (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall).

Fig. 11 Average volume fraction of gas phase along the line through the vortex center (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall).

In the RNG and RSM models, Reynolds averaging is applied to the initial instantaneous Navier-Stokes equations, therefore, it is the RANS velocity solved in the calculations. Hence, the velocity fluctuations are not directly resolved in the RNG and RSM models. The fluctuating components are contained in the turbulent kinetic energy, k , in the RNG model and in the Reynolds stress terms in the

RSM model. In the LES model, the instantaneous velocities of the large eddies are recovered by the calculations. Therefore, the fluctuating components can be directly extracted from the LES model, and they can represent the genuine fluctuating velocities if the mesh is fine enough. Based on the aforementioned information, the fluctuating velocities are extracted and displayed in Fig. 12, where the fluctuating velocities obtained by the LES model shows good agreement with the experimental data. In addition, the subgrid kinetic energy from the filtered-out eddies is 3 orders of magnitude smaller than the directly resolved one, suggesting that the LES model has resolved almost all of the kinetic energy. According to Pope's criterion which is used to evaluate the dependence of the LES calculations on the grid spacing [42], the ratio of the subgrid turbulent kinetic energy to the total turbulent kinetic energy should be less than 20%. As this ratio in our LES calculations is far less than this limit, the mesh used in this work is appropriate with respect to Pope's criterion. Compared to the LES model, the RNG and RSM models greatly overpredict the fluctuating velocities with similar amounts and trend. This suggests that the reason for the discrepancy should be something that the RNG and RSM model have in common in their model constructions. It is obvious that both models adopt the scale-determining equation, *i.e.*, the turbulence dissipation rate equation. This equation is based on physical reasoning and it is rather uncertain. Therefore, this overprediction may be caused by the inaccuracy of the dissipation rate equation in our case. In addition to this most possible factor, the assumptions employed to model the terms such as the pressure production and dissipation in the RSM model may also limit its fidelity. According to the results in Fig. 12, The RNG and RSM models are not able to accurately predict the fluctuating velocity components for our cases.

Fig. 12 Fluctuating velocity (a), axial fluctuating velocity (b) and radial fluctuating velocity (c) along the line through the vortex center (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall).

4.5.2. Comparison of Reynolds stress terms

The comparisons of Reynolds stress terms are shown in Fig. 13(a), (b) and (c), respectively. Significant over-predictions of the Reynolds stress terms are noticed in the RNG and RSM models, while the LES model successfully represents the stress terms. Besides, the magnitudes of Reynolds stress terms in the RSM and the LES models are much larger in the region close to the injection lance due to the intensive turbulence generated by the rising bubbles. This is in line with the phenomenon observed in the PIV experiments. However, the RNG model does not obviously reveal this feature, probably due to the inapplicability of the isotropic theory. In addition, there is a clear hump in the results obtained by the RNG and RSM model. It could be the consequence of the failure of the epsilon equation used in the RNG and RSM model. The epsilon equation underestimates the turbulence dissipation rate in that region, resulting in a hump. It can also be found that the axially normal component of stresses, $\langle u'u' \rangle$, is larger than the radially normal component and the shear component, $\langle u'v' \rangle$. This is caused by the strongly upward flow driven by the rising bubbles. Since the turbulent kinetic energy is included in the normal Reynolds stress terms, the results of the turbulent kinetic energy are not represented in this paper to avoid repetition.

Fig. 13 Reynolds stress term $\langle u'u' \rangle$ (a), $\langle v'v' \rangle$ (b) and $\langle u'v' \rangle$ (c) along the line through the vortex center (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall).

4.6. Flow characteristics

According to the validation above, it is believed that the LES model is superior than the RNG and RSM models in the present work, particularly for the turbulence aspect. Thereby, the LES model is adopted for the study of the flow characteristics.

Bubble behavior is very important for a bubbly flow. To this end, the gas volume fractions from the 2D and 3D models are investigated and displayed in Fig. 14. Firstly, a dome-shaped gas bubble appears at the exit of the top-submerged lance and then it is deformed into a dumbbell shape with narrow necks due to the pressure difference on the bubble surface. The neck becomes even narrower and thinner until it breaks, then the detached bubbles rise up to the liquid surface. As the process continues, bubble breakup occurs. It is interesting to note that the bubble shape is similar in the 2D and 3D models at the very early stage (≤ 0.15 s) when the liquid is less fluctuating. The rather flat liquid surface means that the turbulence is weak. After 0.15 s, a wavy liquid surface can be clearly observed, and bubble breakup also appears. The difference is that the asymmetric phenomena including the bubble shape and number and surface fluctuation can only be captured in the 3D model. These asymmetric characteristics are normally caused by transient turbulence, which is not observed in the 2D simulations due to its axisymmetric setting. In addition, other typical phenomena including symmetric bubble growth and rise, lop-sided bubble detachment and free-rise can also be observed in the 3D simulation. Fig. 15 shows the comparison of bubble profiles between the experiment and 3D simulation. The bubble profile is well captured by the simulation in the aspects of asymmetric shape, lop-sided detachment and free-rise. Therefore, the 3D VOF-LES model is obviously more capable of investigating the bubble behavior generated by top-submerged lance injection.

Fig. 14 Gas volume fraction at different times from 2D model (upper) and 3D model (X-Z median section) (lower).

Fig. 15 Bubbles profile obtained from experiments and 3D simulations.

For bubbly flow, the bubble frequency can be estimated by inspecting the power spectral density obtained by the FFT [43] of the pressure measured at one point ($x = 0.160$ m, $y = 0$ m) in the top lance domain. As show in Fig. 16, there is a clearly dominant bubble frequency detected at around 20 Hz, which indicates the presence of a bubbling mode. In order to understand the turbulence in the bubbly region, the fluctuating velocity in time series was measured at one point ($x = 0.151$ m, $y = 0.015$ m) in the bubbly region. The energy spectrum was obtained through the FFT of the recorded velocity values. Fig. 17 shows the power spectral density, and this one-dimensional spectrum implies a wide range turbulence. Its slope is changing between the single phase $-5/3$ and two-phase $-10/3$ power laws in the inertial sub-range. This result is consistent with the work of Dhotre, et al [16]. The sampling rate for the above FFT treatment is 1 kHz, and the Hanning's window built in FLUENT software is used during the data process.

Fig. 16 Fast Fourier Transform of the measured pressure in the top lance.

Fig. 17 Power spectral density taken at a point ($x = 0.151$ m, $y = 0.015$ m).

4.7. Effect of contact angle

As shown in Fig. 14, the formed bubbles attach to the outer wall of the lance at the tip. Therefore, it is necessary to highlight the influence of the contact angle on the flow. The contact angle by definition is the angle formed between the liquid-gas interface and the surface of outer lance wall. While the prescribed contact angle is only effective with a good refining mesh [44]. In order to check the used mesh here, the interface resolution near the lance tip is demonstrated in Fig. 18. It is evident that the interface spreads over 2-3 cells. Hence, it can be concluded that the present mesh is fine enough to activate the prescribed contact angle [44]. In our study, two simulations with contact angles of 30° and 90° were carried out, respectively. The qualitative and quantitative results represented by the velocity fields, average axial velocity and its fluctuating velocity are shown in Fig. 19 and Fig. 20, respectively. There is only minor difference within 5% between these two simulations, indicating a limited influence of the contact angle on the flow. Nevertheless, further work is needed to confirm this conclusion.

Fig. 18 Interface resolution with the used mesh (node number 6000).

Fig. 19 Velocity fields obtained from LES model with contact angle of 30° (a), with contact angle of 90° (b) (r: radial distance from the axial line; R: radial distance between the axial line and the vessel wall; h: axial distance; H: height of the measured zone).

Fig. 20 Average axial velocity (a) and axial fluctuating velocity (b) obtained from experiment and LES model with different contact angles (r : radial distance from the axial line; R : radial distance between the axial line and the vessel wall).

5. Conclusions

This paper studies the hydrodynamics of large bubbles involved in gas injection into liquid through a top-submerged lance. Simulations have been performed to firstly verify a proper selection of turbulence models coupled with the VOF multiphase model, and then the selected turbulence model has been used to study the flow characteristics resulted from the top-submerged lance injection. Three turbulence models, *i.e.*, the RNG, RSM and LES, have been considered and the simulation results have been compared with the PIV experimental data.

For the qualitative validation of the flow pattern, all of the turbulence models exhibit the main recirculation zone. However, the LES model has a better agreement with the experimental data both in vortex position as in vortex shape compared to the RNG and RSM models. In addition, the RNG and RSM models agree well with the experiments in the mean velocities, but they cannot estimate the fluctuating parameters such as fluctuating velocity and Reynolds stress terms well. In contrast, the results of the LES model are in good agreement with experimental data both for mean velocities and fluctuating parameters. It, therefore, can be concluded that the LES model is more suitable for the study of a top-submerged lance injection. The flow characteristics have been investigated with a combined VOF-LES model. The bubble-forming process is clearly observed. The bubble shape is very similar at the very early stage in both the 2D and 3D simulations. However, the asymmetric phenomena at later stage appear only in the 3D model due to the existing transient turbulence. Thereby, the 3D model is more suitable for studying bubble behavior. It was confirmed that the bubble frequency is around 20 Hz by the FFT of the measured pressure. The one-dimensional energy spectra indicates that the turbulence in the bubbly region is between the single phase $-5/3$ and two-phase $-10/3$ power law in the inertial sub-range. Finally, the effect of contact angle between the formed bubbles and outer lance wall on flow field has been assessed. The results reveal that the contact angle has very limited influence on the flow field by qualitative and quantitative comparisons between the simulations with contact angles of 30° and 90° .

This work is designed for fluids with a relatively high viscosity such as oil and molten slag. The validation seems very successful in representing the mean flow, or in capturing the turbulence characteristics. However, such work regarding to the validation of fluids with a low viscosity, *e.g.*, water and liquid metal, is still needed.

Acknowledgements

The computational support from HPC (KU Leuven) and financial Grant IWT project 140514 (Belgium) are highly acknowledged. The authors, Yannan Wang and Lingling Cao, would like to give thanks to the China Scholarship Council (CSC).

Author contributions

Yannan Wang participated in the PIV experiments, performed numerical simulations, analyzed the data, interpreted the results, wrote the manuscript. Maarten Vanierschot performed the PIV experiments, assisted in the interpretation of the results and reviewed the paper. Lingling Cao participated in the PIV experiments, contributed to the discussion and reviewed the paper. The contribution of other co-authors, Zhongfu Cheng, Bart Blanpain and Muxing Guo, consisted in discussion of the results and reviewing of the paper.

Conflicts of interest

The authors declare no conflicts of interest.

References

- [1] J.M. Floyd, The third decade of top submerged lance technology, in: M. Nilmani, T. Lehner (Eds.), The Howard Worner International Symposium on Injection in Pyrometallurgy, TMS, (1996) 417-429.
- [2] J. Hoang, M.A. Reuter, R. Matusiewicz, S. Hughes, N. Piret, Top submerged lance direct zinc smelting, *Miner. Eng.* 22(2009) 742-751.

- [3] N. Huda, J. Naser, G. Brooks, M.A. Reuter, R.W. Matuszewicz, CFD modeling of swirl and nonswirl gas injections into liquid baths using top submerged lances, *Metall. Mater. Trans. B.* 41B(2010) 35-50.
- [4] N. Huda, J. Naser, G. Brooks, M.A. Reuter, R.W. Matuszewicz, Computational fluid dynamic modeling of zinc slag fluming process in top-submerged lance smelting furnace, *Metall. Mater. Trans. B.* 43B(2012) 39-55.
- [5] D. Stephens, M. Tabib, M.P. Schwarz, CFD simulation of bath dynamics in the HIs melt smelt reduction vessel for iron production, *Progress in Computational Fluid Dynamics*, 12(2012), 196-206.
- [6] D. Pflieger, S. Gomes, N. Gilbert, H.G. Wagner, Hydrodynamic simulations of laboratory scale bubble columns fundamental studies of the Eulerian-Eulerian modelling approach, *Chem. Eng. Sci.* 54(1999) 5091-5099.
- [7] E. Delnoij, J.A.M. Kuipers, W.P.M. van Swaaij, A three dimensional CFD model for gas-liquid bubble columns, *Chem. Eng. Sci.* 54(1999) 2217-2226.
- [8] J. Sanyal, S. Vasquez, S. Roy, M.P. Dudukovic, Numerical simulation of gas-liquid dynamics in cylindrical bubble column reactors, *Chem. Eng. Sci.* 54(1999) 5071-5083.
- [9] Y. Pan, M.P. Dudukovic, M. Chang, Dynamic simulation of bubbly flow in bubble columns, *Chem. Eng. Sci.* 54(1999) 2481-2489.
- [10] N.G. Deen, T. Solberg, B.H. Hjertager, Large eddy simulation of the gas-liquid flow in a square cross-sectioned bubble column, *Chem. Eng. Sci.* 56(2001) 6341-6349.
- [11] J. Chahed, V. Roig, L. Masbernat, Eulerian-Eulerian two-fluid model for turbulent gas-liquid bubbly flows, *Int. J. Multiphase Flow* 29(2003) 23-49.
- [12] G.M. Cartland Glover, S.C. Generalis, The modelling of buoyancy driven flow in bubble columns, *Chem. Eng. Process.* 43(2004) 101-115.
- [13] K. Bech, Dynamic simulation of a 2D bubble column, *Chem. Eng. Sci.* 60(2005) 5294-5304.
- [14] K. Ekambara, M.T. Dhotre, J.B. Joshi, CFD simulations of bubble column reactors: 1D, 2D and 3D approach, *Chem. Eng. Sci.* 60(2005) 6733-6746.

579 [15] M.V. Tabib, S.A. Roy, J.B. Joshi, CFD simulation of bubble column-An analysis of interphase
580 forces and turbulence models, *Chem. Eng. J.* 139(2008) 589-614.

581 [16] M.T. Dhotre, B. Niceno, B.L. Smith, Large eddy simulation of a bubble column using dynamic
582 sub-grid scale model, *Chem. Eng. J.* 136(2008) 337-348.

583 [17] M. Pourtousi, J.N. Sahu, P. Ganesan, Effect of interfacial forces and turbulence models on
584 predicting flow pattern inside the bubble column, *Chem. Eng. Process.* 75(2014) 38-47.

585 [18] D.D. McClure, H. Norris, J.M. Kavanagh, D.F. Fletcher, G.W. Barton, Validation of a
586 computationally efficient Computational Fluid Dynamics (CFD) model for industrial bubble
587 column bioreactors, *Ind. Eng. Chem. Res.* 53(2014) 14526-14543.

588 [19] S. Besbes, M.E. Hajem, H.B. Aissia, J.Y. Champagne, J. Jay, PIV measurements and Eulerian-
589 Lagrangian simulations of the unsteady gas-liquid flow in a needle sparger rectangular bubble
590 column, *Chem. Eng. Sci.* 126(2015) 560-572.

591 [20] S.T. Johansen, F. Boysan, Fluid dynamics in bubble stirred ladles: Part II. Mathematical
592 modeling, *Metall. Trans. B.* 19B(1988) 755-764.

593 [21] D. Guo, G.A. Irons, Modeling of gas-liquid reactions in ladle metallurgy: Part II. Numerical
594 simulation, *Metall. Mater. Trans. B.* 31B(2000) 1457-1464.

595 [22] M.S.C. Terrazas, A.N. Conejo, Effect of nozzle diameter on mixing time during bottom-gas
596 injection in metallurgical ladles, *Metall. Mater. Trans. B.* 46B(2015) 711-718.

597 [23] L. Li, B. Li, Z. Liu, Modeling of gas-steel-slag three-phase flow in ladle metallurgy: Part II.
598 Multi-scale mathematical model, *ISIJ Int.* 57(2017) 1980-1989.

599 [24] M. Iguchi, T. Uemura, H. Yamaguchi, T. Kuranaga, Z. Morita, Fluid flow phenomena in a
600 cylindrical bath agitated by top lance gas injection, *ISIJ Int.* 34(1994) 973-979.

601 [25] Y.S. Morsi, W. Yang, B.R. Clayton, N.B. Gray, Experimental investigation of swirl and non-
602 swirl gas injection into liquid baths using submerged vertical lances, *Can. Metall. Quart.*
603 39(2000) 87-98.

604 [26] M. Adib, M.A. Ehteram, H.B. Tabrizi, Numerical and experimental study of oscillatory behavior
605 of liquid surface agitated by high-speed gas jet, *Appl. Math. Modell.* 62(2018) 510-525.

- [27] P. Liovic, J.L. Liow, M. Rudman, A Volume of Fluid (VOF) method for the simulation of metallurgical flows, *ISIJ Int.* 41(2001) 225-233.
- [28] P. Liovic, M. Rudman, J.L. Liow, Numerical modelling of free surface flows in metallurgical vessels, *Appl. Math. Modell.* 26(2002) 113-140.
- [29] P. Liovic, D. Lakehal, Interface-turbulence interactions in large-scale bubbling processes, *Int. J. Heat and Fluid Flow* 28(2007) 127-144.
- [30] M.P. Schwarz, P.T.L. Koh, Numerical modelling of bath mixing by swirled gas injection, In *SCANINJECT IV: 4th International Conference on Injection Metallurgy*, Lulea, 6:1-6:17, MEFOS, 1986.
- [31] M. Meier, G. Yadigaroglu, M. Andreani, Numerical and experimental study of large steam-air bubbles injected in a water pool, *Nucl. Sci. Eng.* 136(2000) 363-375.
- [32] D.L. Youngs, Time-dependent multi-material flow with large fluid distortion, *Numerical Methods for Fluid Dynamics* 24(2) (1982) 273-285.
- [33] H. Yang, J. Wolters, P. Pischke, H. Soltner, S. Eckert, J. Fröhlich, Improved collision modelling for liquid metal droplets in a copper slag cleaning process, *12th International Conference on CFD in Oil&Gas, Metallurgical and Process Industries*, Trondheim, Norway, (2017) 355-363.
- [34] ANSYS Inc., *Ansys theory guide 16.2*, (2015).
- [35] J.U. Brackbill, D.B. Kothe, C. Zemach, A continuum method for modeling surface tension, *J. Comput. Phys.* 100(1992) 335-354.
- [36] B.E. Launder, D.B. Spalding, The numerical computation of turbulent flows, *Comp. Meth. Appl. Mech. Eng.* 3(1974) 269-289.
- [37] V. Yakhot, S.A. Orszag, Renormalization group analysis of turbulence. I. basic theory, *J. Sci. Comp.* 1(1986) 3-51.
- [38] F. Nicoud, F. Ducros, Subgrid-scale stress modelling based on the square of the velocity gradient tensor, *Flow Turbul. Combust.* 62(1999) 183-200.
- [39] R.I. Issa, Solution of the implicitly discretised fluid flow equations by operator-splitting, *J. Comput. Phys.* 62(1986) 40-65.

633 [40] J.P. Van Doormaal, G.D. Raithby, Enhancements of the SIMPLE method for predicting
634 incompressible fluid flows, Numer. Heat Transfer 7(1984) 147-163.

635 [41] M. Vanierschot, T.A.J. Verrijssen, S. Van Buggenhout, M. Hendrickx, E. Van den Bulck,
636 Experimental and numerical analysis of an apparatus to apply controlled shear/elongation in fluid
637 flows, Chem. Eng. Sci. 113(2014) 88-94.

638 [42] S.B. Pope, Ten questions concerning the large-eddy simulation of turbulent flows, New J. Phys.
639 6(2004) 35-58.

640 [43] E.O. Brigham, The fast fourier transform and its applications, Prentice Hall, New Jersey, (1988).

641 [44] R.V.P. Rezende, R.A. Almeida, A.A.U. de Souza, S.M.A.G.U. Souza, A two-fluid model with a
642 tensor closure model approach for free surface flow simulations, Chem. Eng. Sci. 122(2015) 596-
643 613.