

**CFD simulation of bubbly flow around an obstacle in a vertical pipe with
a focus on breakup and coalescence modelling**

Tas-Köhler, S.; Neumann-Kipping, M.; Liao, Y.; Krepper, E.; Hampel, U.;

Originally published:

November 2020

International Journal of Multiphase Flow 135(2021), 103528

DOI: <https://doi.org/10.1016/j.ijmultiphaseflow.2020.103528>

Perma-Link to Publication Repository of HZDR:

<https://www.hzdr.de/publications/Publ-31210>

Release of the secondary publication
on the basis of the German Copyright Law § 38 Section 4.

CC BY-NC-ND

CFD simulation of bubbly flow around an obstacle in a vertical pipe with a focus on breakup and coalescence modelling

Sibel Tas-Koehler ^{a*}, Martin Neumann-Kipping ^b, Yixiang Liao ^a, Eckhard Krepper ^a, Uwe Hampel ^{a,b}

^a Helmholtz-Zentrum Dresden-Rossendorf, Institute of Fluid Dynamics, Bautzner Landstr. 400, 01328 Dresden, Germany

^b Technische Universität Dresden, Chair of Imaging Techniques in Energy and Process Engineering, 01062 Dresden, Germany

* CORRESPONDING AUTHOR (E-MAIL: s.tas@hzdr.de)

ABSTRACT

In the present study, we assessed the capabilities of Eulerian-Eulerian CFD two-phase flow simulation with the homogeneous Multiple Size Group Model (MUSIG) and consideration of breakup and coalescence under three-dimensional flow conditions. We compared void fraction, bubble size and bubble velocity distributions against experimental data from vertical gas-disperse two-phase flow in a pipe with a flow obstruction. The simulation results generally agree well upstream the obstacle, where we have a typically developed pipe flow. Downstream of the obstacle void fraction is overpredicted while bubble velocity is underpredicted. The bubble size distribution has no clear trend. With higher liquid velocities, the deviations increase. As a conclusion, the simulation has difficulties to balance the gas fraction in the strong vortex in the shadow of the obstacle. Here further model improvement is needed.

Keywords: CFD, bubbly flow, model validation, bubble breakup, bubble coalescence

Nomenclature

Latin symbols

d_B	bubble diameter [m]	S_K	source term due to turbulent kinetic energy [N·m ⁻² ·s ⁻¹]
d_{cr}	critical bubble diameter [m]	S_ε	source term due to turbulent dissipation rate [N·m ⁻² ·s ⁻²]
C_D	drag coefficient [-]	S_ω	source term due to turbulent frequency [N·m ⁻⁴]
C_L	lift coefficient [-]	u	velocity [m·s ⁻¹]
C_W	wall lubrication coefficient [-]	Greek symbols	
C_{VM}	virtual mass coefficient [-]	α	gas volume fraction [-]
d_\perp	maximum horizontal bubble dimension [mm]	ρ	density [kg·m ⁻³]
E_{O_\perp}	Eötvös number depending on d_\perp [-]	τ	bubble-induced time scale [s]
E_o	Eötvös number [-]	τ_{ij}^{Lam}	laminar stress tensor [kg·m ⁻¹ ·s ⁻²]
F_D	drag force per unit volume [N·m ⁻³]	τ_{ij}^{Turb}	turbulent stress tensor [kg·m ⁻¹ ·s ⁻²]
F_L	lift force per unit volume [N·m ⁻³]	ω	turbulence frequency [s ⁻¹]
F_W	wall lubrication force per unit volume [N·m ⁻³]	ν_L^{turb}	kinematic viscosity [m ² ·s ⁻¹]
F_{Disp}	turbulent dispersion force per unit volume [N·m ⁻³]	σ_{TD}	Schmidt number [-]
F_{VM}	virtual mass force per unit volume [N·m ⁻³]	ε	turbulence dissipation rate [m ² ·s ⁻³]
k	phase indicator, turbulent kinetic energy [m ² ·s ⁻²]	Subscripts	
M_i	source term in i-th direction [kg·m ⁻² ·s ⁻²]	B	bubble
p	pressure [Pa]	L	liquid phase
Re	Reynolds number [-]	G	gas phase

26

27 Acronyms

28 BIT: Bubble-Induced Turbulence

29 CFD: Computational Fluid Dynamics

30 FAD: Favre-Averaged Drag

31 MUSIG: Multiple Size Group Model

32 SST: Shear Stress Transport

33 UFXCT: Ultrafast X-ray Computed Tomography

34 1. Introduction

35 Bubbly flows are encountered in different industrial applications such as chemical, petroleum and
36 nuclear engineering. In these applications, it is important to know the flow behavior to enhance
37 efficiency such as in chemical reactors or to increase safety margins such as in nuclear reactors. For this
38 purpose, computational fluid dynamics (CFD) that is based on the Eulerian-Eulerian framework has
39 become a popular tool. However, the accuracy of CFD simulations highly relies on correct modeling of
40 phase interactions including interfacial forces (i.e. drag, lift, wall lubrication, turbulent dispersion, and
41 virtual mass), bubble-induced turbulence (BIT) and bubble breakup/coalescence. The modeling of the
42 latter one is possible by a poly-dispersed approach in which the gas phase is divided into a number of
43 size groups. In most of the industrial applications, there is a wide distribution of bubble sizes and a non-
44 uniform radial gas fraction profiles. Thus, the correct prediction of bubble size by breakup and
45 coalescence plays an important role in the accuracy of CFD simulations.

46 For adiabatic bubbly flow, numerous numerical studies were done, especially focusing on the interfacial
47 forces and BIT. Frank *et al.* (2008) indicated for two-phase pipe flow that a mono-dispersed simulation
48 model, including the Tomiyama lift force, the Frank wall lubrication force, the Favre-averaged drag
49 (FAD) turbulent dispersion force and the Shear Stress Turbulence (SST) model, gives good agreement
50 with the experimental data in terms of gas void fraction. Besagni *et al.* (2018) investigated the effects
51 of the interfacial forces for small-scale and large-scale bubble columns with the mono-dispersed
52 approach. The authors suggested a baseline model with the following interfacial forces: the Tomiyama
53 drag force, the Antal wall lubrication force and the Lopez or Burns turbulence dispersion force. Jin *et al.*
54 (2019) investigated the influence of different models of interfacial forces on the phase distribution
55 for vertical and inclined bubbly flow. They reported that the combination of the Ishii-Zuber drag force,
56 the Saffman-Mei lift force, the Hosokawa wall lubrication force and the FAD turbulent dispersion force
57 provides good radial void fraction results for vertical bubbly flow. Rzehak *et al.* (2012) examined
58 different wall lubrication force models in case of bubbly flow. They compared the Antal, Tomiyama
59 and Hosokawa wall force models and found that the Hosokawa model provides the best performance.
60 Jareteg *et al.* (2017) investigated the effect of a virtual mass force on the stability of the bubbly flow
61 simulations. The authors showed that the implementation of virtual mass force importantly changes the
62 growth rate of void instabilities. Colombo *et al.* (2019) showed the capability of Eulerian-Eulerian CFD
63 for a bubbly flow in a pipe and square duct by focusing on the lift and turbulence forces. According to
64 the results, the effect of turbulence on the phase distribution is as important as the lift force. They further
65 concluded that the wall lubrication force is not necessary if the near-wall region is appropriately
66 resolved.

67 BIT was also taken into account in many studies. Rzehak *et al.* (2017) developed a closure model for
68 bubbly flow simulations including bubble forces and BIT. However, they did not consider bubble
69 breakup and coalescence yet. The simulation results were compared to experimental data for gas volume
70 fraction, axial liquid velocity and turbulent kinetic energy. Although an overall satisfying agreement
71 between experiments and simulations was found, the authors pointed out that further improvements in
72 turbulence modeling and implementation of bubble breakup/coalescence are highly necessary. Colombo
73 *et al.* (2015) presented the validation of the two-phase Eulerian-Eulerian mono-dispersed model for pipe
74 flow by using experimental data from 6 different literature sources. They reported that their BIT model
75 gives better results in terms of r.m.s velocity fluctuations as compared to Troshko *et al.* (2001) and
76 Rzehak *et al.* (2013). Besagni *et al.* (2018) reported that while BIT inclusion causes convergence
77 problems for large-scale bubble columns, the model of Sato improves the results slightly compared to
78 Simonin *et al.* (1990) for small-scale bubble columns. Parekh *et al.* (2018) compared Launder, Reece,
79 Rodi (LRR) RSM and Speziale-Sarkar-Gatski (SSG) RSM turbulence models as well as the SST model

80 for air-water pipe flow to capture the anisotropy of turbulent fluctuations concerning BIT. The
81 simulation results showed that predictions of LRR and SSG RSM including BIT are comparable to the
82 SST model over radial profiles of the liquid velocity and gas fraction. However, all three models
83 underestimated the wall peaks of the turbulent kinetic energy and Reynolds stresses by comparing them
84 with the experimental data. Liao *et al.* (2018) applied the MUSIG approach for air-water bubbly flow
85 to test the performance of the BIT model developed by Ma *et al.* (2017). The results showed that the
86 model of Ma *et al.* (2017) predicts the radial gas void fraction and gas velocity well compared to the
87 experiments.

88 Bubble breakup and coalescence was less considered in the literature compared to interfacial forces and
89 BIT. Frank *et al.* (2008) showed that the inhomogeneous MUSIG (i-MUSIG) model predicts radial void
90 fraction profiles well but further investigation is needed in terms of bubble breakup and coalescence.
91 Liao *et al.* (2015) performed simulations to assess the capability of the bubble breakup and coalescence
92 modeling which was proposed by Liao *et al.* (2011). The results showed that the mean bubble size is
93 overestimated at low superficial liquid velocities and is slightly underestimated at high superficial liquid
94 velocities. The authors concluded that further studies are necessary considering BIT, which has a high
95 impact on bubble breakup and coalescence mechanisms.

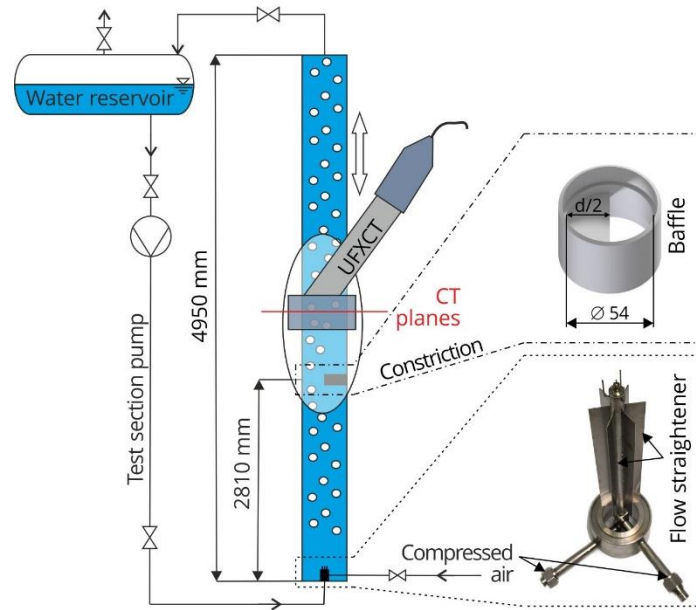
96 While CFD codes have meanwhile been well qualified for simple pipe and column geometries there is
97 yet little analysis and qualification for more complex three-dimensional flow domains. One reason is
98 the lack of appropriate experimental validation data. Prasser *et al.* (2008) provided such data for bubbly
99 flow in a pipe with a semi-circular obstacle using a wire-mesh sensor. Follow-up CFD simulations
100 assuming mono-disperse bubbles provided good void fraction estimation upstream the obstacle but
101 overestimated void fraction downstream. Krepper *et al.* (2009) used the i-MUSIG model including
102 bubble breakup and coalescence and found that the standard breakup and coalescence models (Luo *et*
103 *al.* (1996), Prince *et al.* (1990)) do not predict the bubble size distribution (BSD) well. Continuing in
104 this line the objectives of this study were to show the capability of bubble breakup and coalescence
105 modelling (Liao *et al.* (2015)) under complex flow conditions with new high-resolution two-phase flow
106 data around an obstacle in a pipe. The 3D flow field was simulated for two different liquid velocity
107 conditions using the MUSIG model. For the simulations, a baseline model for air-water bubbly flow
108 was applied that includes the model of Rzehak *et al.* (2017) for interfacial forces, the model of Liao *et*
109 *al.* (2015) for the bubble breakup/coalescence and the model of Ma *et al.* (2017) for the BIT. Moreover,
110 the performance of a new discretization scheme for bubble coalescence and breakup modelling
111 developed by Liao (2020) was also tested within this work.

112 By validation against experimental benchmark data, the capability of bubble breakup/coalescence model
113 is demonstrated in terms of gas volume fraction, gas velocity, mean bubble diameter and bubble size
114 distribution with the aid of Ultrafast X-ray Computed Tomography (UFXCT) measurements
115 (Neumann-Kipping *et al.* (2020)). Furthermore, hydrodynamics of the 3D bubbly flow is analyzed and
116 bubble interaction within the vortex region formed by the obstacle is examined under different liquid
117 velocities. Thus, this paper contributes to two main fields: (i) the assessment of breakup and coalescence
118 model accuracy under 3D flow conditions: (ii) understanding of two-phase flow hydrodynamics in
119 complex geometries.

120 **2. Experimental setup**

121 The experimental study used for validation is described in detail in Neumann-Kipping *et al.* (2020). The
122 experiments were performed in an acrylic pipe with an inner diameter of 54 mm and a total length of
123 4950 mm. A semi-circular obstacle that blocks half of the inner pipe cross-section was utilized to
124 generate 3D flow fields. A sketch of the test facility, as well as the gas injection module, is illustrated

125 in Figure 1. The experiments were performed at 4 bar pressure and a liquid temperature of 30°C.
 126 Deionized water and compressed air are injected at the bottom of the test section as the liquid and gas
 127 phase, respectively.



128
 129 Figure 1: Schematic representations of the vertical test section (left) with details of the gas injection module
 130 (bottom right) and the flow obstacle for generation of three-dimensional flow fields (top right).

131 Various operating conditions in bubbly flow regime were tested by setting appropriate liquid and gas
 132 flow rates. Two operating conditions that are used for the present study are described in Table 1.

133 Table 1: Experimental operating conditions based on combinations of liquid and gas superficial velocities.

Test run	j_l [$\text{m}\cdot\text{s}^{-1}$]	j_g [$\text{m}\cdot\text{s}^{-1}$]
#072	0.4050	0.0368
#074	1.0170	0.0368

134 Ultrafast X-ray computed tomography (UFXCT), which is a well-established non-invasive imaging
 135 technique for multiphase flow, was applied to quantitatively analyze the distribution of gas and liquid
 136 within the test section. The UFXCT scanner can be freely moved to allow for imaging of the flow field
 137 in several imaging planes up- and downstream of the flow obstacle, as depicted in Table 2 (Neumann-
 138 Kipping *et al.* (2020)). By the means of UFXCT, cross-sectional information of the gas velocity and gas
 139 volume fraction, as well as bubble size distribution were determined. A detailed discussion of
 140 measurement uncertainty UFXCT and quality evaluation of the experimental results can be found in
 141 Neumann-Kipping *et al.* (2020). Here, the time-averaged cross-sectional gas holdup and axial gas
 142 velocity were used to calculate the inlet superficial gas velocity. This estimated velocity was compared
 143 to the set value, showing maximum deviation of $\pm 15\%$ for all cases.

144

145 Table 2: Image plane identifier along the vertical test section with relative distances of the upper image plane to
 146 the center of the flow constriction.

Identifier	A	B	C	D	E	F	G	H	I
Z (mm)	-200	-60	0	5	20	50	100	200	400

147

148 3. Numerical method

149 3.1 General remarks

150 For simulation, the geometry was defined as a vertical half tube using a symmetry xz-plane. All
 151 simulations were performed using the solver ANSYS CFX 19.2. The fluid domain was modelled from
 152 1.5 m upstream to 1 m downstream the obstacle (Figure 2). The results were obtained by applying the
 153 Multiple Size Group Model (Lo (1996)). The dispersed phase was divided into 15 size fractions where
 154 the bubble diameters ($d_b = 0 \dots 15 \text{ mm}$) were defined with equidistant bubble diameter. As inlet
 155 conditions, a developed flow condition for liquid velocity, liquid turbulent kinetic energy and turbulent
 156 dissipation obtained from previous single-phase simulations were assigned. Further, experimentally
 157 determined radial void fraction and bubble size distribution at the largest upstream ($Z = -211 \text{ mm}$)
 158 position were applied. A constant pressure was defined as outlet condition. All the simulations were
 159 performed at steady state condition. Turbulence was modeled only for the liquid phase using the SST
 160 model (Menter (1994)). The dimensionless wall distance value for the liquid phase y^+ was kept greater
 161 than 30. The single-phase law of wall was used for the wall treatment. The density change of gas was
 162 considered by treating the gas phase as ideal gas and its change depending on the height was considered
 163 according to

$$\rho_G(P) = \rho_{G,atm} \frac{P_H}{P_{atm}}, \quad (1)$$

164 where P_{atm} is the atmospheric pressure and $\rho_{G,atm}$ is the gas density depending on atmospheric
 165 pressure. P_H is the pressure depending on the height and it is calculated with the following equation:

$$P_H = 4P_{atm} + \rho_L g h. \quad (2)$$

166 Here, ρ_L is the liquid density, g is the gravitational acceleration and h is the height. On the pipe wall, a
 167 no-slip condition was applied for the liquid phase and a free-slip condition for the gas phase. The
 168 convergence criteria were set to $RMS < 10^{-6}$.

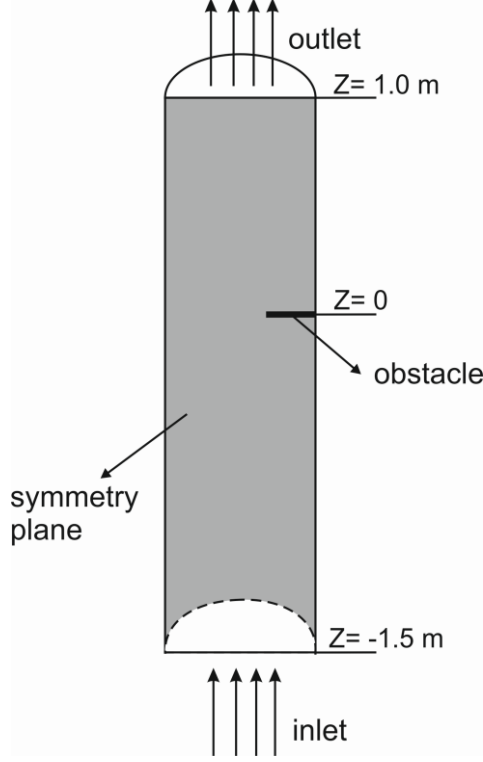


Figure 2: Schematic view of computational domain.

169

170

171

172 3.2 Governing equations

173 An Eulerian-Eulerian two-fluid model was applied in the present simulations. This framework defines
 174 every phase by a set of averaged conservation equations. Detailed information and derivations of the
 175 conservation equations were given by Yeoh *et al.* (2009). Since there is no heat transfer considered for
 176 the current study, the conservation equations include only the continuity equation

$$\frac{\partial(\alpha_k \rho_k)}{\partial t} + \frac{\partial}{\partial x_i} (\alpha_k \rho_k \mathbf{u}_{i,k}) = 0 \quad (3)$$

177 and the momentum equation

$$\begin{aligned} \frac{\partial}{\partial t} (\alpha_k \rho_k \mathbf{u}_{i,k}) + \frac{\partial}{\partial x_i} (\alpha_k \rho_k \mathbf{u}_{i,k} \mathbf{u}_{j,k}) \\ = -\alpha_k \frac{\partial p_k}{\partial x_i} + \frac{\partial}{\partial x_j} [\alpha_k (\tau_{ij,k}^{Lam} + \tau_{ij,k}^{Turb})] + \alpha_k \rho_k \mathbf{g}_i + \mathbf{M}_{i,k}. \end{aligned} \quad (4)$$

178 Here, k is the phase indicator, α is the volume fraction, ρ is the density, \mathbf{u}_i is the velocity component in
 179 the i -th direction, p is the pressure, τ_{ij}^{Lam} is the laminar stress tensor, τ_{ij}^{Turb} is the turbulence stress
 180 tensor and \mathbf{M}_i is the source term in the i -th direction, which will be explained in the following section.

181 3.3 Interfacial momentum transfer

182 The Eulerian-Eulerian framework considers interpenetrating continua and therefore closure models for
 183 interfacial momentum transfer are applied by means of a source term of the form

$$\mathbf{M}_k = \mathbf{F}_D + \mathbf{F}_L + \mathbf{F}_W + \mathbf{F}_{Disp} + \mathbf{F}_{VM} \quad (5)$$

184 that is added in the momentum equation including drag force \mathbf{F}_D , lift force \mathbf{F}_L , wall lubrication force
 185 \mathbf{F}_W , turbulent dispersion force \mathbf{F}_{Disp} and virtual mass force \mathbf{F}_{VM} . In the present study, all these forces
 186 were considered for the simulations.

187 The drag force

$$\mathbf{F}_D = -\frac{3 C_D}{4 d_B} \alpha \rho_L |\mathbf{u}_G - \mathbf{u}_L| (\mathbf{u}_G - \mathbf{u}_L) \quad (6)$$

188 acts opposite to the relative motion of bubbles relative to the surrounding liquid. Here, d_B is the bubble
 189 diameter, α is the gas void fraction, ρ_L is the liquid density, \mathbf{u}_G is the gas velocity, \mathbf{u}_L is the liquid
 190 velocity and C_D is the drag coefficient calculated by the correlation of Ishii *et al.* (1979).

191 The lift force

$$\mathbf{F}_L = -C_L \alpha \rho_L (\mathbf{u}_G - \mathbf{u}_L) \times (\nabla \times \mathbf{u}_L) \quad (7)$$

192 occurs due to the interaction of the bubble with the shear flow of the liquid. Here, C_L is the lift force
 193 coefficient and determined by Tomiyama *et al.* (2002). The lift force coefficient changes its sign from
 194 positive to negative if the bubble diameter exceeds a critical bubble diameter. For water-air at ambient
 195 conditions, as is the case here, this critical diameter has a value of $d_{cr} = 5.8 \text{ mm}$. (Tomiyama *et al.*
 196 (1998)).

197 The wall lubrication force

$$\mathbf{F}_W = \frac{2}{d_B} C_W \rho_L \alpha |\mathbf{u}_G - \mathbf{u}_L|^2 \hat{\mathbf{y}} \quad (8)$$

198 drives the bubbles away from the wall to avoid the maximum gas fraction at the wall. Here, $\hat{\mathbf{y}}$ is the unit
 199 normal perpendicular to the wall and C_W is the wall force coefficient. The model of Hosokawa *et al.*
 200 (2002) was applied to predict the wall force coefficient.

201 The turbulent dispersion force

$$\mathbf{F}_{Disp} = -\frac{3 C_D}{4 d_B} \alpha \rho_L |\mathbf{u}_G - \mathbf{u}_L| \frac{\nu_L^{turb}}{\sigma_{TD}} \left(\frac{1}{(1-\alpha)} + \frac{1}{\alpha} \right) \nabla \alpha \quad (9)$$

202 describes the impact of liquid phase turbulent fluctuations on the gas phase. Here, ν_L^{turb} is the kinematic
 203 viscosity of the liquid phase and σ_{TD} is the Schmidt number, generally taken as 0.9. The turbulent
 204 dispersion force was modeled by Burns *et al.* (2004).

205 The virtual mass force

$$\mathbf{F}_{VM} = -C_{VM} \alpha \rho_L \left(\frac{D_G \mathbf{u}_G}{Dt} - \frac{D_L \mathbf{u}_L}{Dt} \right) \quad (10)$$

206 acts on the bubbles in case of bubble sudden acceleration. Here, C_{VM} is the virtual mass coefficient,
 207 which was set to 0.5 for the simulations. Table 3 shows the equations for calculating the force
 208 coefficients.

Table 3: Mathematical description for interfacial force coefficients.

Force	Reference	Mathematical description
Drag	Ishii <i>et al.</i> (1979)	$C_D = \max[C_{D_{sphere}}, \min(C_{D_{ellipse}}, C_{D_{cap}})]$
		$C_{D_{sphere}} = \frac{24}{Re_d} (1 + 0.1Re_d^{3/4}), C_{D_{ellipse}} = \frac{2}{3} \sqrt{Eo}, C_{D_{cap}} = \frac{8}{3}$
Lift	Tomiyama <i>et al.</i> (2002)	$C_L = \begin{cases} \min[0.288 \tanh(0.121Re), f(Eo_\perp)] & Eo_\perp < 4 \\ f(Eo_\perp) & 4 < Eo_\perp < 10 \\ -0.27 & 10 < Eo_\perp \end{cases}$
		$f(Eo_\perp) = 0.00105Eo_\perp^3 - 0.0159Eo_\perp^2 - 0.0204Eo_\perp + 0.474$
Wall lubrication	Hosokawa <i>et al.</i> (2002)	$Eo_\perp = \frac{g(\rho_L - \rho_L)d_\perp^2}{\sigma} \quad d_\perp = d_B \sqrt[3]{1 + 0.163 Eo^{0.757}}$ $C_W(y) = f(Eo) \left(\frac{d_B}{2y}\right)^2, \quad f(Eo) = 0.021Eo$
Turbulent dispersion	Burns <i>et al.</i> (2004)	Favre averaging the drag force
Virtual mass	Auton <i>et al.</i> (1988)	Constant coefficient $C_{VM} = 0.5$

210

211 3.4 Bubble induced turbulence

212 Two turbulence sources affect the gas-liquid two-phase flow. The first one is the shear-induced
 213 turbulence that calculates the turbulence parameters by the applied single-phase flow turbulence model.
 214 Therefore, the SST turbulence model was applied to the continuous phase in this study. Dispersed phase
 215 turbulence was obtained from continuous phase calculations. This approximation is valid for flows with
 216 a low-density ratio, such as air-water flows (Colombo *et al.* (2015)).

217 The second one is the BIT, which accounts for the turbulence generation due to bubble-liquid
 218 interaction. Ma *et al.* (2017) proposed a BIT model with the source terms

$$S_K = C_I F_D (\mathbf{u}_G - \mathbf{u}_L), \quad (11)$$

$$S_\varepsilon = \frac{C_\varepsilon}{\tau} S_K, \quad (12)$$

219 and

$$S_\omega = \frac{1}{C_\mu k} S_\varepsilon - \frac{\omega}{k} S_K. \quad (13)$$

220 Here, k is the turbulent kinetic energy, ω is the turbulent frequency, C_μ is the shear-induced turbulence
 221 coefficient that is taken as 0.09, τ is the BIT time scale

$$\tau = \frac{d_B}{|\mathbf{u}_G - \mathbf{u}_L|} \quad (14)$$

222 and C_I and C_ε are the model coefficients defined as

$$C_I = \min(0.18Re_B^{0.23}, 1) \quad (15)$$

223 and

$$C_\varepsilon = 0.3C_D. \quad (16)$$

224 These BIT source terms are added in the SST turbulent equations and the turbulent viscosity is calculated
225 with the aid of standard equation

$$\mu_L^{Turb} = C_\mu \rho_L \frac{k_L^2}{\varepsilon_L}. \quad (17)$$

226

227 **3.5 Bubble breakup and coalescence**

228 As can be seen from the literature review, less attention has been paid to bubble breakup and coalescence
229 mechanisms in the Eulerian modelling of bubbly flow. However, for the bubbly flow systems, there are
230 strong interactions between bubbles from different classes, which results in bubble breakup and
231 coalescence. The MUSIG model (Lo (1996)), which is based on population balance approach was
232 applied to the simulations. In the MUSIG model, the dispersed phase is divided into M size fractions
233 and the population balance equation is used to determine the mass conservation of the size fractions
234 considering the interaction mass transfer due to bubble breakup and coalescence. The size fraction
235 equations are given as

$$\frac{\partial \rho_G \alpha f_i}{\partial t} + \frac{\partial}{\partial x_i} (\rho_G \alpha f_i \mathbf{u}_{i,G}) = B_{Ci} - D_{Ci} + B_{Bi} - D_{Bi}. \quad (18)$$

236 The source and sink terms contain the birth rates due to coalescence and breakup, B_{Ci} , B_{Bi} , and death
237 rates due to coalescence and breakup of the bubbles, D_{Ci} , D_{Bi} . They are calculated as

$$B_{Ci} = (\rho_G \alpha)^2 \left(\frac{1}{2} \sum_{j \leq i} \sum_{k \leq i} Q(m_j; m_k) X_{jki} \frac{m_j + m_k}{m_j m_k} f_j f_k \right) \quad (19)$$

$$B_{Bi} = \rho_G \alpha \sum_{j > i} g(m_j; m_i) f_j \quad (20)$$

$$D_{Ci} = (\rho_G \alpha)^2 \left(\sum_j Q(m_i; m_j) \frac{1}{m_j} f_i f_j \right) \quad (21)$$

$$D_{Bi} = \rho_G \alpha f_i \sum_{j < i} g(m_i; m_j) \quad (22)$$

238 where the functions Q denotes the coalescence rate and g denotes the breakup rate. In this context, the
 239 coalescence and breakup model of Liao *et al.* (2015) was applied to determine these rates, which
 240 considers various bubble interaction mechanisms such as turbulent fluctuation, shear, buoyancy and
 241 wake. In the simulations, the change of bubble size due to pressure changes was considered in the
 242 calculation of the Sauter mean diameter (d_{32}) by

$$d_{32}^* = d_{32}(\rho_{ref}/\rho_g)^{1/3} \quad (23)$$

243 where ρ_{ref} is the reference gas density at the inlet and ρ_g is the gas density at a certain height.

244 Recently, Liao (2020) found that the formulations from Eq. (19) to Eq. (22) preserve only the mass of
 245 bubbles but not their number when assigning the breakup/coalescence source term to size groups. A
 246 consequence is underprediction of the bubble size, especially in breakup-dominant cases. She developed
 247 an internally consistent discretization scheme for the terms of birth rates, which is:

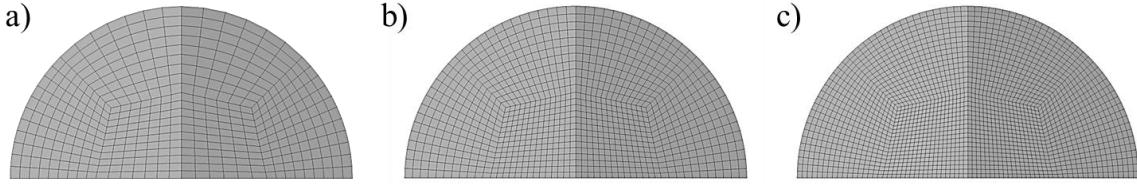
$$B_{Ci}' = (\rho_G \alpha)^2 \left(\frac{1}{2} \sum_{j \leq i} \sum_{k \leq i} Q(m_j; m_k) X_{jki} \frac{m_i}{m_j m_k} f_j f_k \right) \quad (24)$$

$$B_{Bi}' = \rho_G \alpha \sum_{j > i} \frac{m_i}{m_j} f_j \left(g(m_j; m_i) + \sum_{k < j} g(m_j; m_k) Y_{jki} \right) \quad (25)$$

248 Note that m_j+m_k in Eq. (19) has now been replaced by m_i yielding Eq. (24). The size fraction equation
 249 of Eq. (18) is derived from the population balance equation for the bubble number concentration N_i by
 250 multiplying it with m_i , so m_i instead of m_j+m_k should be contained in the coalescence source term. The
 251 computation of the breakup source term according to Eq. (25) considers two situations of the daughter
 252 bubble positioning. One is that the daughter bubble size coincides with the representative value of a size
 253 class, and the other is that the daughter bubble size lies between two representative values. In the former
 254 case, the source term can be calculated in a similar way given in Eq. (20) or the first term in the bracket
 255 of Eq. (25). In the latter one, a mass matrix Y_{jki} like X_{jki} in the coalescence representing the fraction of
 256 mass going to group i is needed for the calculation of the source term. In binary breakage, if the parent
 257 bubble and one daughter bubble are fixed at a representative value, the size of the second daughter
 258 usually does not coincide with any representative values. Therefore, the formulation in Eq. (25) is
 259 general. This scheme preserves both the mass and the number of bubbles. The effect of this discretization
 260 scheme was investigated in the present study.

261 4. Mesh independence studies

262 The flow domain was discretized using structured meshes. Mesh study was done for test 072 with three
 263 different meshes (Figure 3): 103,050 elements (mesh 1, subfigure a), 252,000 elements (mesh 2,
 264 subfigure b), and 553,850 elements (mesh 3, subfigure c). The mesh refinement was applied both in
 265 axial and lateral directions.



266

267

Figure 3: Mesh views: a) mesh 1 b) mesh 2 and c) mesh 3.

268

269

270

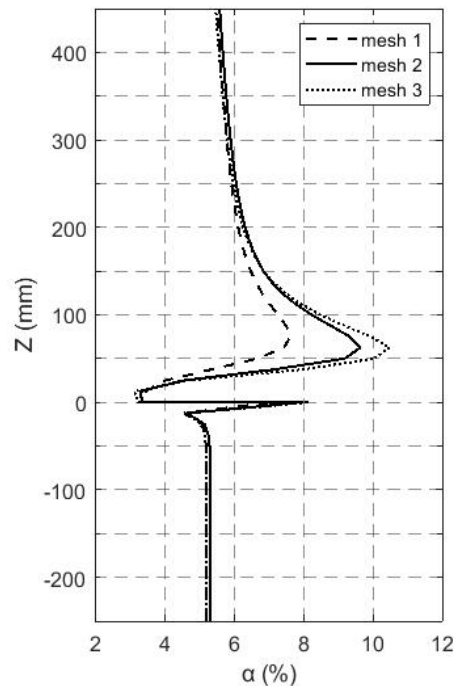
271

272

273

274

Figure 4 shows the simulated average gas fraction along the axial direction for different mesh densities. Upstream of the obstacle $Z < 0$ mm, the mean gas fraction is very much mesh independent. The effect of the mesh refinement is largely noticeable downstream the obstacle $0 \text{ mm} < Z < 200$ mm where high flow complexity occurs. This mesh sensibility to the flow complexity is in line with that previously reported by Tas-Koehler *et al.* (2020). Whereas a maximum relative difference between mesh 1 and mesh 2 is 29%, it is 8% between mesh 2 and mesh 3. Hence, to reduce the computational effort, mesh 2 is applied for this study.



275

276

Figure 4: Effect of grid refinement on the average gas fraction for test 072.

277 5. Results

278

5.1 Phase distribution

279

280

281

282

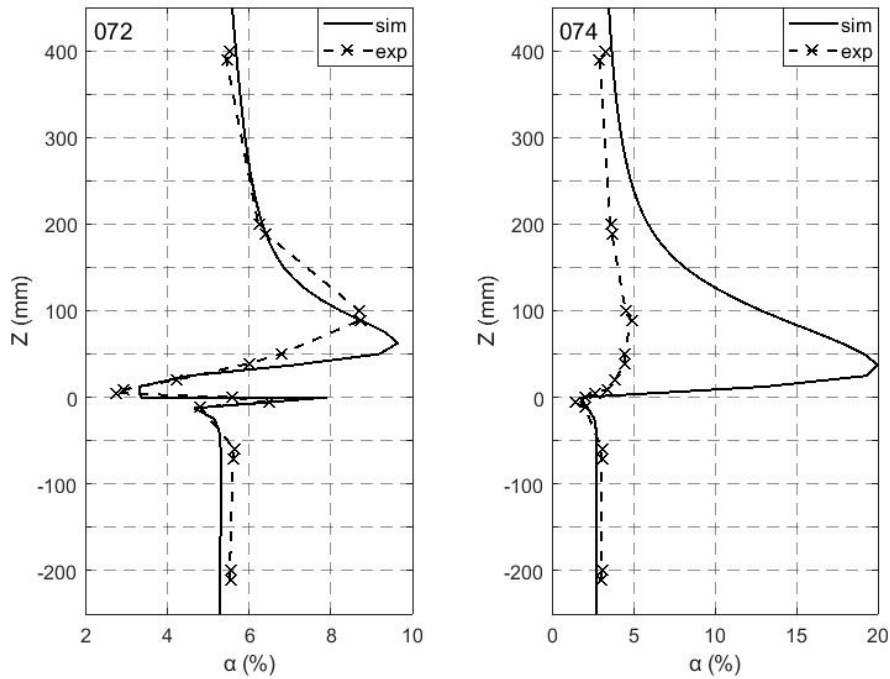
283

284

285

286

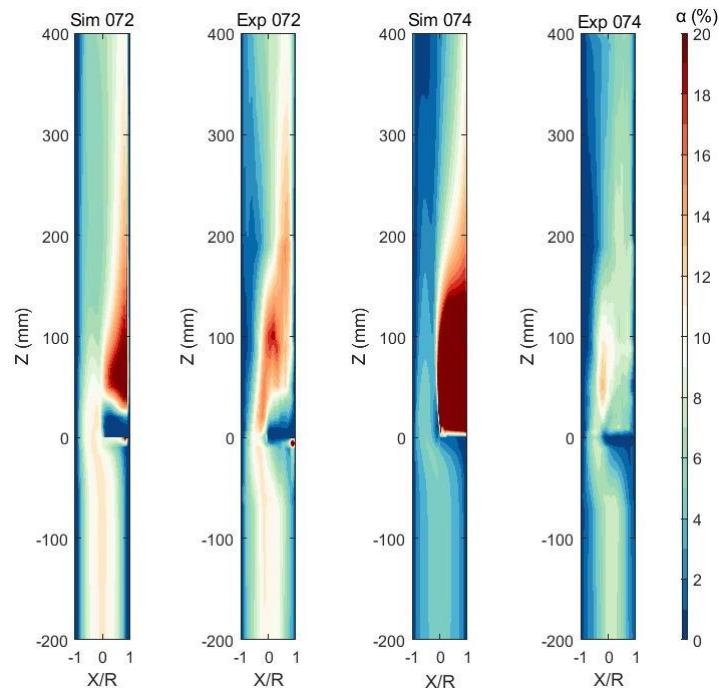
Figure 5 shows the evolution of the averaged gas fraction in the axial direction of the pipe for test cases 072 and 074. Upstream of the obstacle, both cases agree well with the experiments. For test 072, void fraction decreases dramatically downstream the obstacle. After that point, it starts to increase up to around $Z = 60$ mm and it decreases again. Although the peaks that occur after the obstacle in the experiment and simulation do not coincide, generally there is a good agreement between experiment and simulation. For test section 074, void fraction increases downstream the obstacle up to $Z = 40$ mm and it starts to decrease after that point. However, the averaged void fraction is highly overestimated downstream of the obstacle.



287

288 Figure 5: Cross-sectional averaged void fraction along the axial direction for test cases 072 and 074.

289 Figure 6 shows the void fraction for test 072 and 074. The strong gas accumulation after the obstacle
 290 for both tests is calculated. For test 072, a very small near-wall maximum void fraction region
 291 underneath the obstacle and the region with void accumulation after the obstacle are well captured by
 292 the simulation. However, in line with Figure 5 for test 074, there is a high void region after the obstacle
 293 that shows a large discrepancy in terms of its magnitude compared to the experiment. Another finding
 294 is that whereas void starts to accumulate just behind the obstacle for test 074, it begins to accumulate
 295 after a certain Z position that is around 25 mm for test 072.

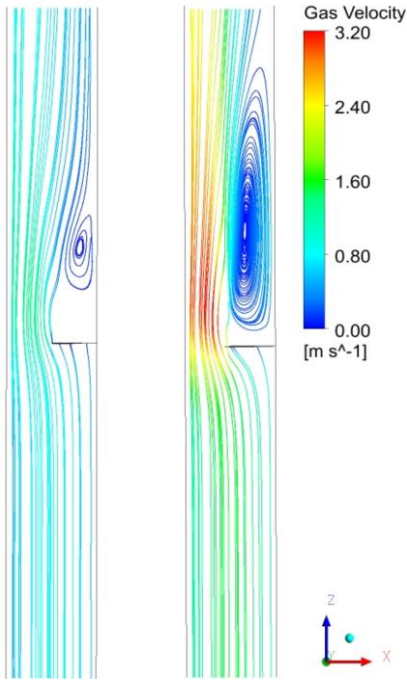


296

297 Figure 6: Visualization plots of simulated and measured void fraction for test cases 072 and 074.

298 In order to explain the different void fraction peaks of test cases 072 and 074, streamlines of the gas
 299 velocity of are shown in Figure 7 for $-200 \text{ mm} < Z < 200 \text{ mm}$. While there is a region free of bubbles

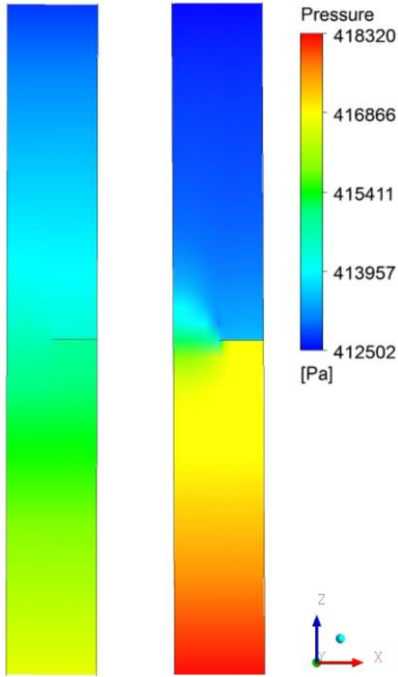
300 directly behind the obstacle for test case 072, this region nearly vanishes for test case 074. Here, the
 301 obstacle causes a downstream wake region (vortex region) due to lateral pressure differences (Figure 8).
 302 As the liquid superficial velocity increases, the pressure differences increase and so does the wake
 303 region. The vortex flow in the wake region leads to void fraction accumulation due to the density
 304 difference between the liquid and gas phases. For test case 072, the wake region appears between around
 305 $30 \text{ mm} < Z < 80 \text{ mm}$, while for test case 074 it develops between around $10 \text{ mm} < Z < 150 \text{ mm}$. This is
 306 in good accordance with the averaged void fraction peaks and changes that are shown in Figure 5 and
 307 Figure 6.



308

309

Figure 7: Streamline for test 072 (left) and 074 (right).

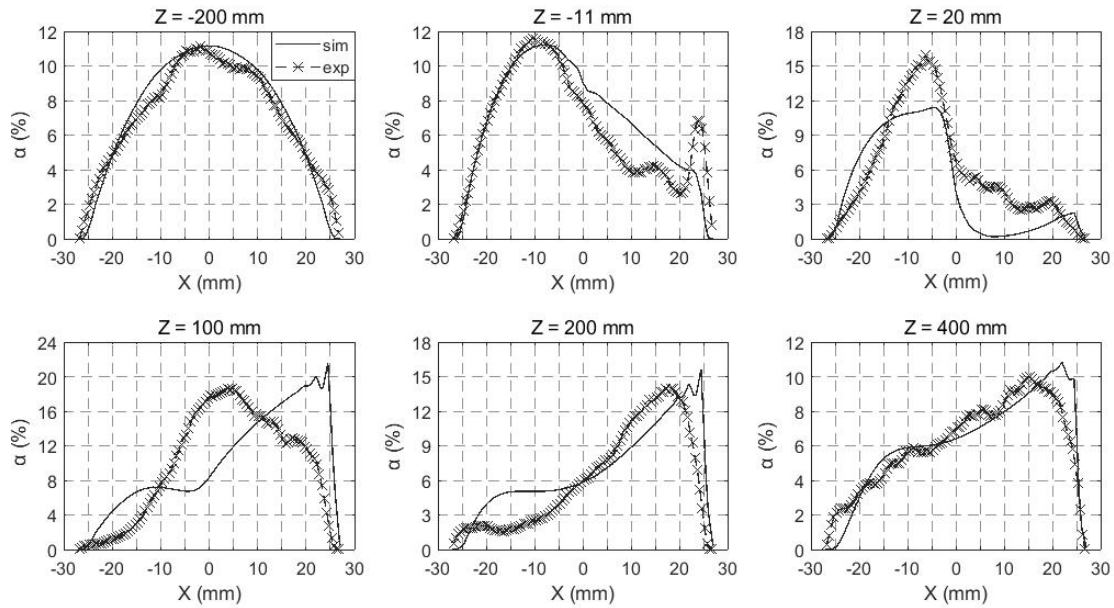


310

311

Figure 8: Pressure distribution for test 072 (left) and 074 (right).

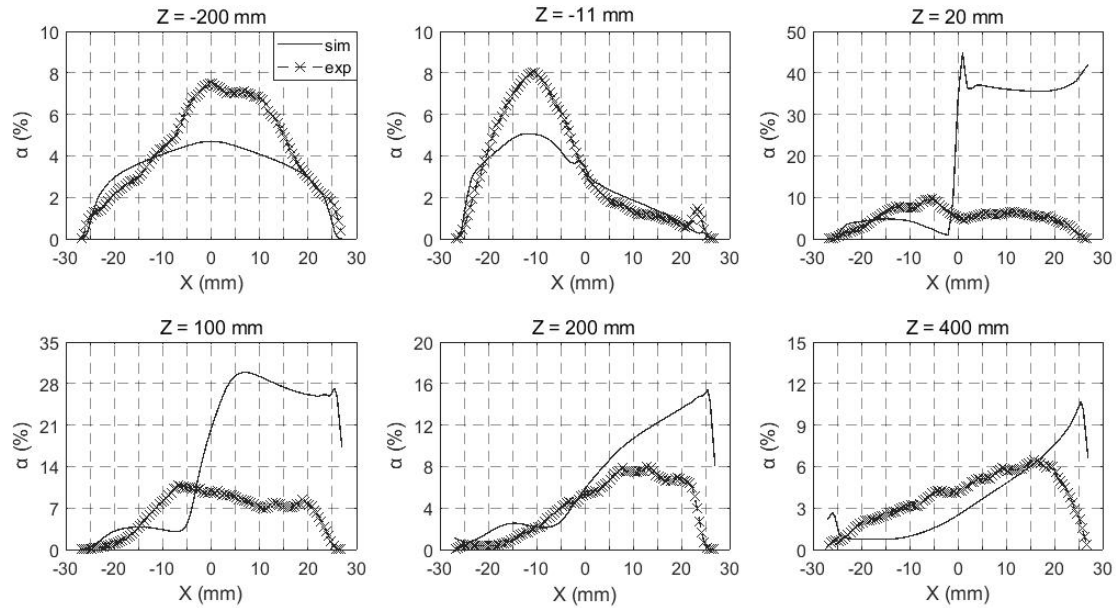
312 Radial void fraction distributions at different cross sections are given in Figure 9 and Figure 10 for both
 313 test cases 072 and 074, respectively. At $Z = -200$ mm, the results of test case 072 are in good agreement
 314 with the experiments, but in the case of 074, the void fraction is underestimated in the pipe center. At
 315 $Z = -11$ mm, better agreement for the obstructed (right) side of the pipe is obtained for test case 074 than
 316 for 072. This is in contrast to the results of the unobstructed (left) side of the pipe. At $Z = 20$ mm both
 317 test cases provide better results for the unobstructed side of the pipe. Further downstream, the
 318 inconsistencies between the experiments and simulations for both sides of the pipe significantly increase
 319 for $Z = 100$ mm and both test cases, but decrease again at higher Z positions, respectively. However,
 320 discrepancies for test case 074 are significantly higher than for test case 072. Downstream of the
 321 obstacle, test case 074 is generally overestimated, especially at the obstructed side of the pipe.



322

323 Figure 9: Radial gas fraction distribution for test 072 for different Z positions.

324 Another point is that without any disturbance, bubbles accumulate in the center of the pipe.
 325 Consequently, they change their position near the obstacle to the unobstructed side of the pipe until they
 326 reach the wake region. Here, they are drawn into the recirculating flow area. Thus, the bubbles start to
 327 accumulate on the obstructed side of the pipe.



328

329

Figure 10: Radial gas fraction distribution for test 074 for different Z positions.

330

5.2 Bubble dynamics

331

332

333

334

335

336

337

338

339

340

In Figure 11 the average bubble diameter changes along the axial direction for both test cases are presented, showing a slight overestimation of the bubble size upstream of the flow obstacle for test case 072. However, here, simulation results for test case 074 are in better accordance with experimental results. Although simulation results are in satisfying agreement with experimental data for both tests at the obstacle, peaks that are obtained from the experiments could not be captured by the simulations. Also, according to the experiments, the average bubble diameter peaks downstream the obstacle at around $Z = 200$ mm for both test cases. None of the simulations can capture these peaks. In addition, the average bubble size is underestimated for $Z > 30$ mm for test case 074. For test case 072, though average bubble size is underestimated for $70 \text{ mm} < Z < 350$, it gives good agreement between around $350 \text{ mm} < Z < 400$ mm.

341

342

343

344

345

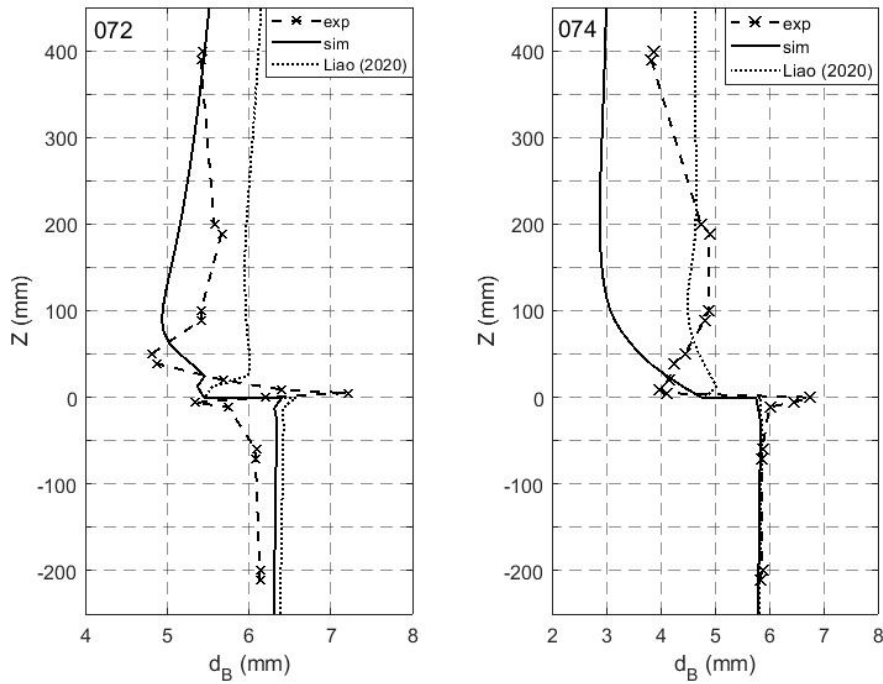
346

347

348

349

Liao (2020) investigated the inconsistencies of the population balance equation in MUSIG, and updated the model by discretizing the source and sink terms that result from bubble coalescence and breakup with an internally consistent scheme, which preserves both the mass and the number of bubbles. The comparison with the updated model is also shown in Figure 11. Whereas the updated MUSIG model provides similar results to the standard MUSIG model upstream the obstacle, it predicts differently at the downstream. For the test case 072, simulation results with the updated model are overestimated compared to experimental results, but the breakup and coalescence tendency behind the obstacle is well captured. For the test case 074, it provides better average bubble diameter prediction than the standard one.



350

351

Figure 11: Average bubble diameter for test cases 072 and 074.

352

353

354

355

356

357

358

359

360

361

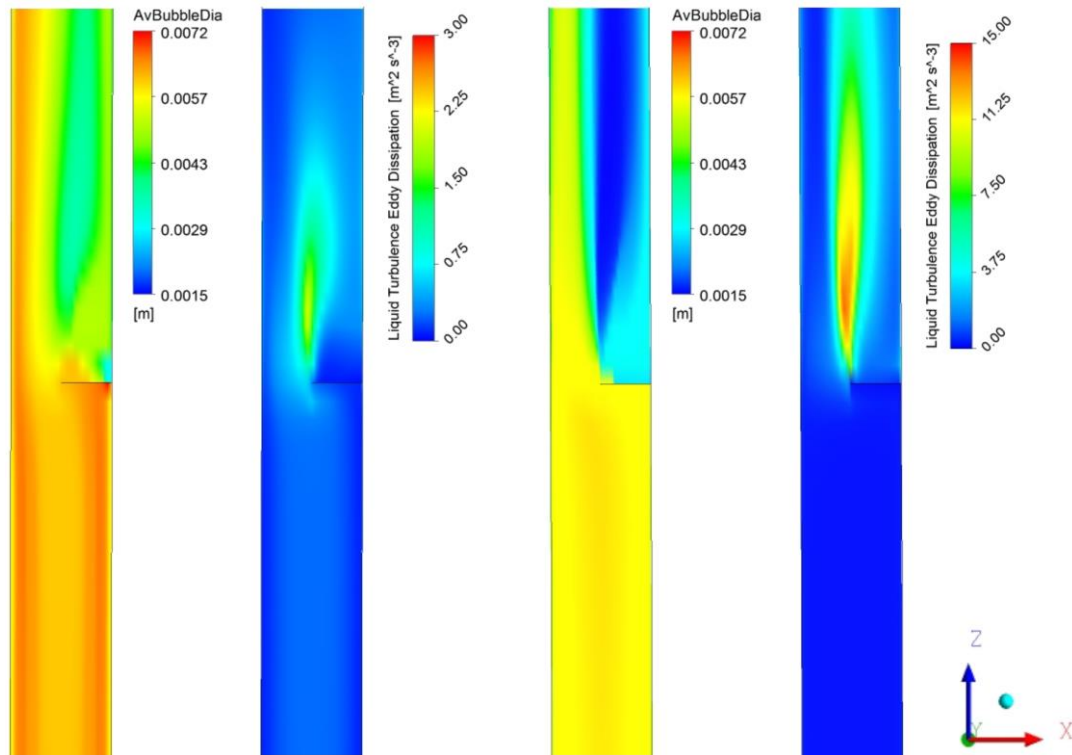
362

363

364

365

Figure 12 presents the average bubble diameter and turbulent dissipation visualization for $-200 \text{ mm} < Z < 200 \text{ mm}$ and both test cases. Upstream of the obstacle, the average bubble diameter near the pipe wall is larger than in the pipe center for test case 072, while it is vice versa for test case 074. This can be explained by the bubble breakup that is dominant for test case 074 due to higher turbulence as compared to test case 072, where more coalescence events take place in the near wall region. Due to the obstacle, a strong liquid jet with high velocity is established, creating a strong shear flow. This, in turn, leads to higher turbulent dissipation as can be seen from the comparison of both test cases in Figure 12. Thus, higher bubble breakup rates are induced. Downstream of the obstacle larger bubble sizes are found on the unobstructed side of the pipe, whereas smaller bubbles occur on the obstructed side because of bubble breakup and liquid circulation for both test cases. This effect increases for test case 074 due to the higher turbulent dissipation. Thus, bubble breakup is more dominant than bubble coalescence in the downstream wake region of the flow obstacle. The accumulation of small bubbles in the circulation region is more obvious (see Figure 7).



366

367
368

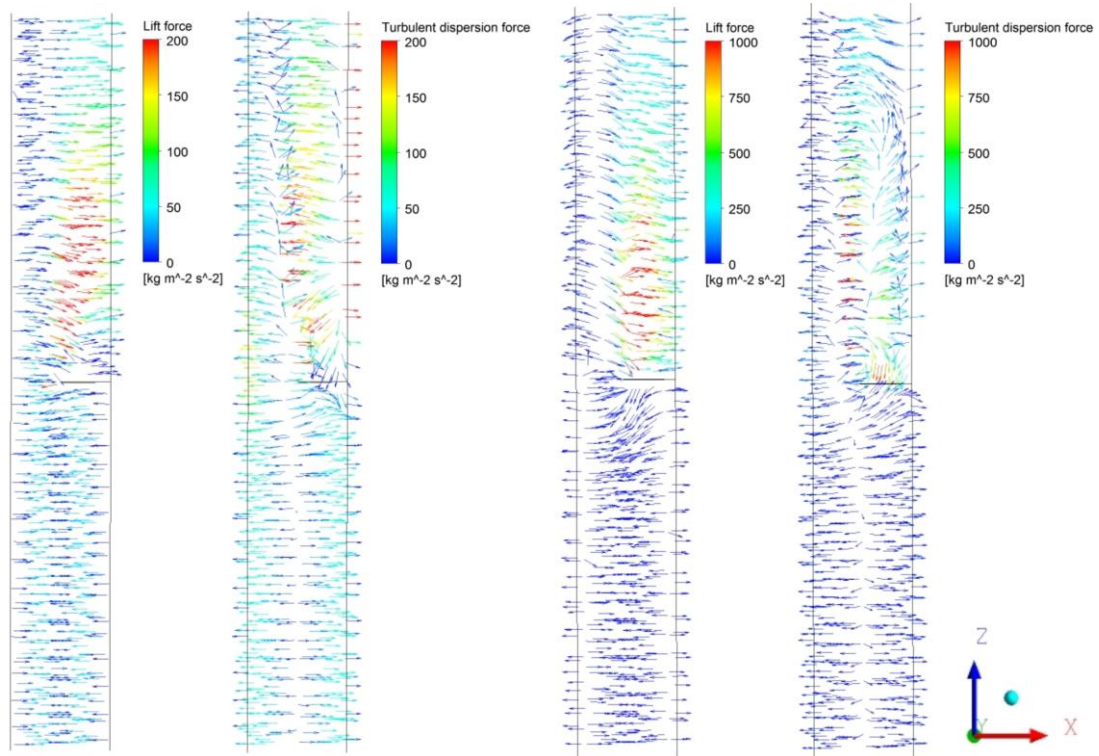
Figure 12: Comparison of average bubble diameter and turbulent dissipation for test cases 072 (left) and 074 (right).

369
370
371
372
373
374
375
376
377
378
379
380
381

Figure 13 presents the lift and turbulent dispersion force vectors for $-200 \text{ mm} < Z < 200 \text{ mm}$ and both test cases. For test 072, by the lift force, all the bubbles are directed to the pipe center upstream the obstacle due to a negative lift force coefficient. Unlike the lift force, the turbulent dispersion force directs the bubbles to the pipe walls, where the volume fraction is lower, and the magnitude of the turbulent dispersion force is bigger than the magnitude of the lift force near the pipe walls. As a result, as the lift force directs the bubbles to the pipe center and increases further the void fraction there. At the obstacle downstream, bubbles that are in the jet region directed to the pipe center by the lift force. On the other hand, bubbles that are in the wake region behind the obstacle are directed by the lift force to the pipe wall, because of a smaller average diameter. Besides this, the turbulent dispersion force directs the bubbles, which are in the jet region, to the pipe wall as expected to counterwork the accumulation of bubbles. Yet, on the obstacle side, the turbulent dispersion force directs the bubbles to the pipe center. The reason is that turbulent dispersion force influences from high void fraction to low void fraction since it is related to the void fraction gradient.

382
383
384
385
386
387
388
389
390
391

For test 074, whereas some bubbles, which are close to the wall, are directed to the pipe wall by the lift force upstream the obstacle, the others move to the pipe center. The turbulent dispersion forces direct all the bubbles to the pipe walls due to a core-peak volume fraction profile as shown in Figure 10. Bubbles which have higher lift force magnitude than dispersion force, move to the pipe center. At the obstacle downstream, for the bubbles that are located undisturbed part of the pipe, the lift force directs them further to pipe wall except for the area that is between the jet region and the region behind the obstacle, where negative velocity gradients prevail. Additionally, whereas turbulent dispersion force directs the bubbles, which are on the left-hand side of the pipe and right-hand side just after the obstacle, to the wall, it changes its direction from pipe wall to pipe center after a certain Z distance on the right-hand side of the pipe, because of high accumulation of bubbles there.



392

393

Figure 13: Bubble lift and turbulent dispersion vectors for test 072 (left) and 074 (right).

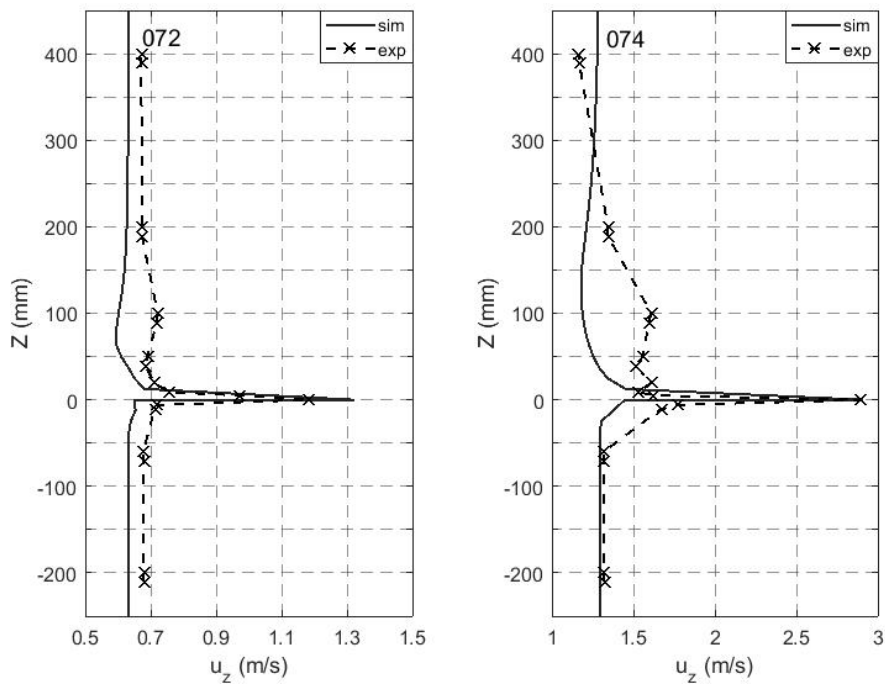
394

The average axial gas velocity is shown in Figure 14. It is clear that the flow experiences strong accelerations due to the obstacle. Both simulation results are generally in good agreement with the experiments. However, there are some inconsistencies according to the experiments like that the simulations could not predict the velocity peaks where are $Z = 100 \text{ mm}$ for both tests.

395

396

397



398

399

Figure 14: Cross-sectional averaged gas velocity for test 072 and 074.

400

5.3 Bubble breakup and coalescence

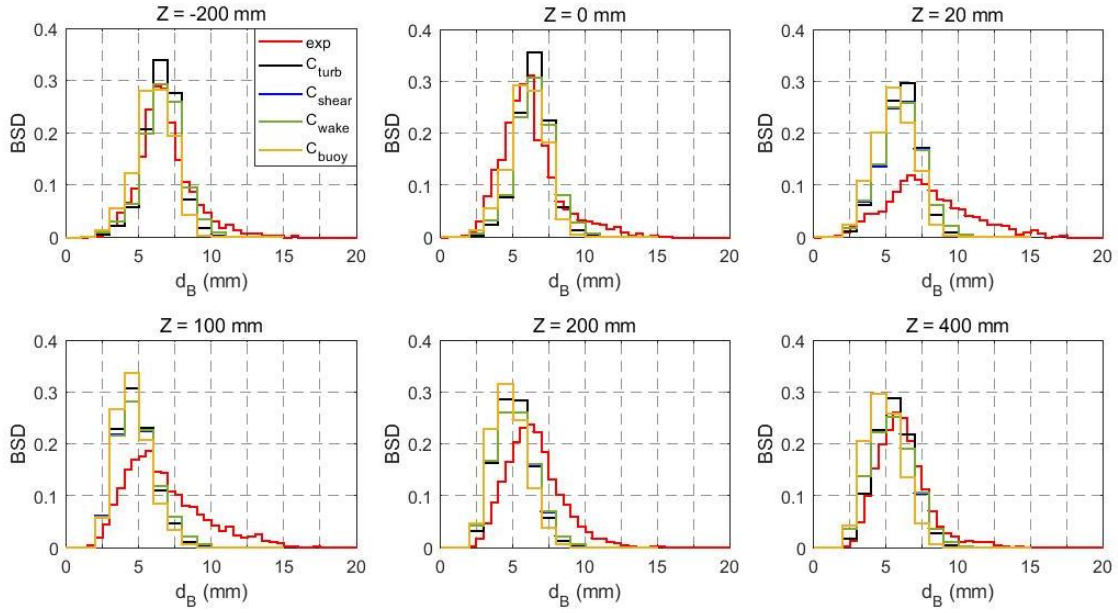
According to Liao *et al.* (2015), there are four mechanisms leading to bubble breakup: laminar shear, turbulent shear, interfacial slip and turbulence fluctuation and likewise five mechanisms leading to coalescence: buoyancy, eddy capture, velocity gradient, wake entrainment and turbulence fluctuation.

The laminar and turbulent shear mechanisms account for viscous shear force in the bulk flow and eddies, respectively. The interfacial slip mechanism considers the impact of interfacial friction. The turbulence fluctuation mechanism describes the effect of turbulent velocity fluctuation on bubble breakup. For coalescence modeling, the buoyancy mechanism accounts for the collision if a faster bubble approaches a slower one. The eddy capture mechanism occurs between bubbles, which are smaller than the Kolmogorov length scale. The velocity gradient mechanism is due to the velocity gradient in the bulk flow. The wake entrainment mechanism acts in the wake region of a bubble where relatively small bubbles can accelerate and catch up with the big one that forms the wake.

The effects of these breakup and coalescence mechanisms were investigated in this study. Thereby, we neglected the turbulent shear mechanism for breakup modelling and the eddy capture mechanism for coalescence modelling as these mechanisms are only important when bubbles are much smaller than the Kolmogorov length scale.

5.3.1 Bubble coalescence mechanisms

Bubble coalescence occurs due to bubble-bubble collision and can be described by different mechanisms as mentioned in Section 5.3. Detailed information can be found in Liao *et al.* (2015). Figure 15 shows the impact of turbulence, velocity gradient, wake entrainment and buoyancy mechanisms on the bubble size distribution at different cross-sections for test case 072. The mechanisms are switched on or off by setting C_{turb} , C_{shear} , C_{wake} and C_{buoy} respectively equal to one or zero. As can be seen from the Figure 15, considering each mechanism separately provide similar results for all Z positions. Here, calculated coalescence rate of all mechanisms might be negligible in comparison to breakup rates, leading to approximately the same predictions. Consequently, the combination of different coalescence mechanisms does not improve the simulation results, as can be seen in Figure 16.

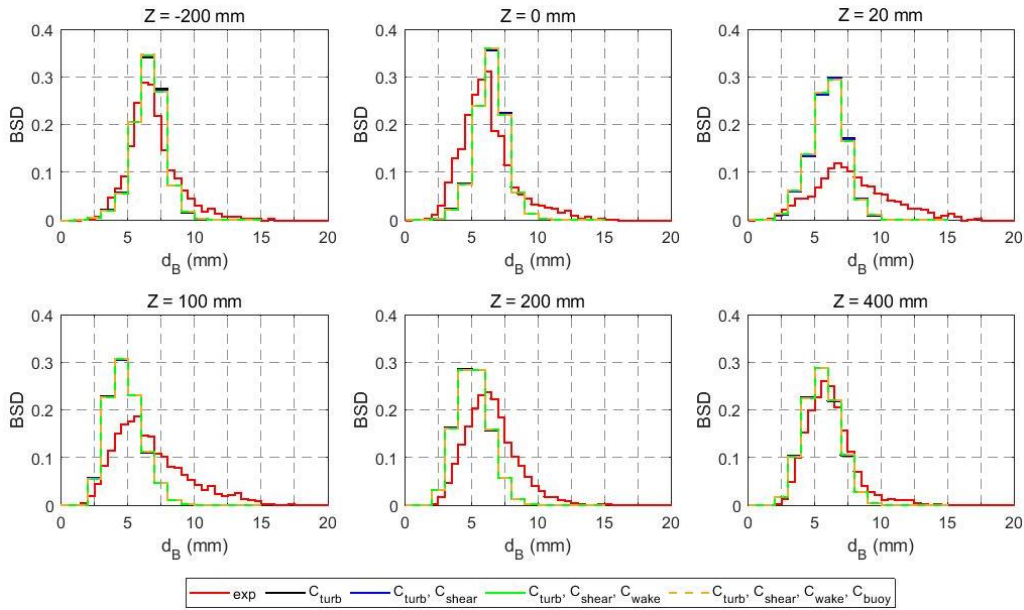


427

428

429

Figure 15: Comparison of bubble coalescence mechanisms for test case 072: C_{turb} , C_{shear} , C_{wake} and C_{buoy} (a legend is given in the first graph).



430

431

432

Figure 16: Comparison of bubble coalescence mechanisms for test case 072: C_{turb} , $C_{turb} + C_{shear}$, $C_{turb} + C_{shear} + C_{wake}$ and $C_{turb} + C_{shear} + C_{wake} + C_{buoy}$ (a legend is given below the graphs).

433

5.3.2 Bubble breakup mechanisms

434

435

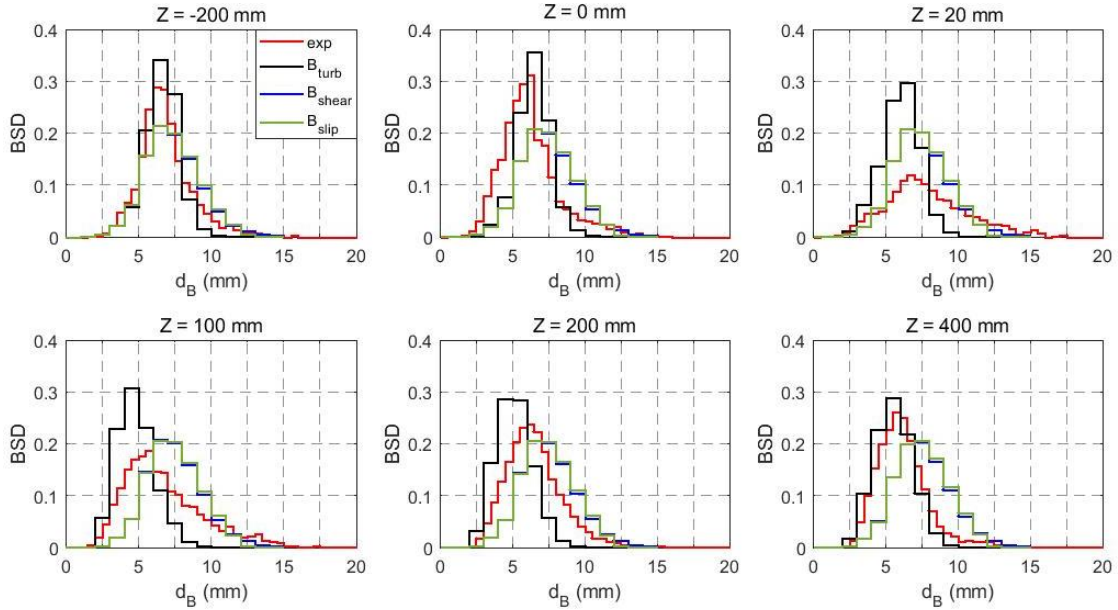
436

437

438

439

Bubble breakup takes place due to flow stresses that act on the bubble surface. As mentioned in Section 5.3, there are different mechanisms causing bubble breakup. Figure 17 presents the impacts of turbulent fluctuation (B_{turb}), laminar shear (B_{shear}) and interfacial slip (B_{slip}) on the bubble size distributions for the varied Z positions. B_{shear} and B_{slip} give almost similar results for every Z positions. Also, as can be seen from Figure 18, laminar shear and interfacial slip play a negligible effect in the breakup, since it is mainly caused by turbulence.

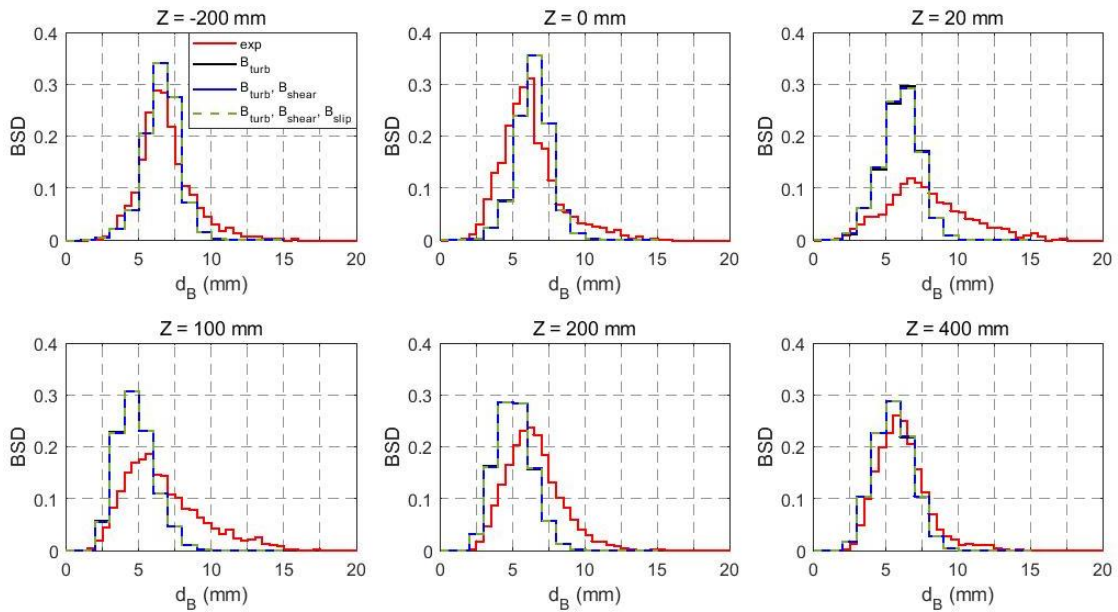


440

441

442

Figure 17: Comparison of bubble breakup mechanisms for test 072: B_{turb} , B_{shear} and B_{slip} (a legend is given in the first graph).



443

444

445

Figure 18: Comparison of bubble breakup mechanisms for test 072: B_{turb} , $B_{turb} + B_{shear}$ and $B_{turb} + B_{shear} + B_{slip}$ (a legend is given in the first graph).

446

447

448

As can be seen from Figure 15 to 18, other mechanisms than turbulence fluctuation have negligible effects. Thus, we considered only the turbulence fluctuation mechanism to model the breakup and coalescence in this study.

449

450

451

452

453

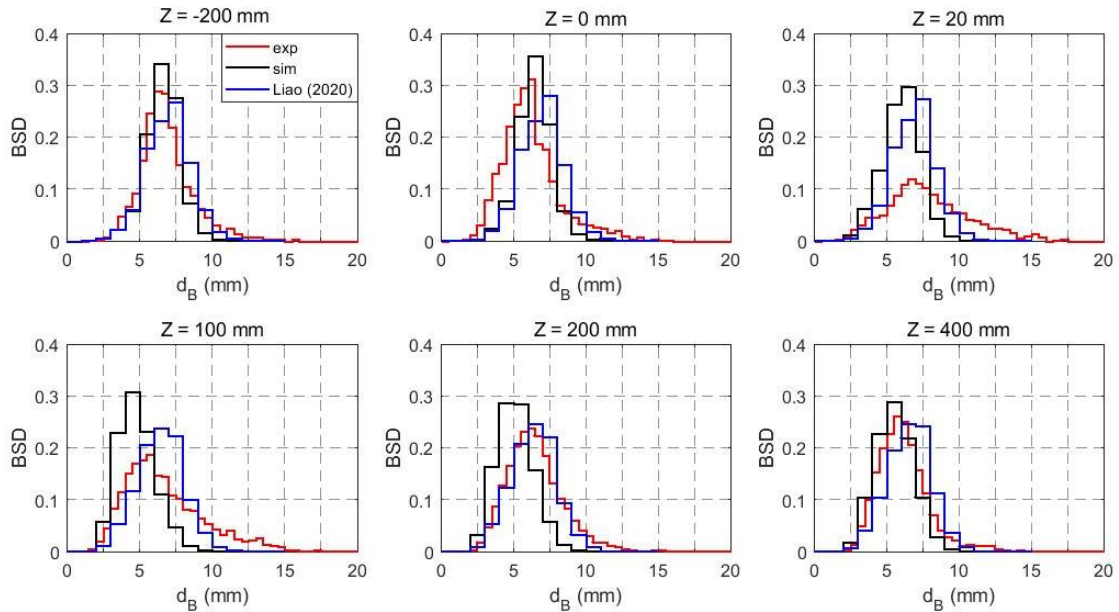
454

455

456

Figure 19 and Figure 20 present the bubble size distribution at different Z positions for both test cases. The bubble size distribution (BSD) is determined as gas volume fraction of each bubble size class divided by the class width and total gas void fraction. For the standard MUSIG model, there is a good agreement of simulation and experiment at position $Z = -200$ mm for test case 072. Although the simulation gives satisfying results for the larger bubbles ($d_B > 7.5$ mm) at position $Z = 0$ mm, it is underestimated for smaller ones ($d_B < 6.5$ mm). Here, slight breakup and coalescence tendency as compared to position $Z = -200$ mm is observed in the experiment, but the model captures none of them. At positions $Z = 20$ mm, $Z = 100$ mm and $Z = 200$ mm, the gas fraction of the bubbles in the size class

457 below 2.5 mm are well predicted. However, the simulation overestimates the gas fraction of bubbles
 458 around $2.5 \text{ mm} < d_B < 7.5 \text{ mm}$ for position $Z = 20 \text{ mm}$ and around $2.5 \text{ mm} < d_B < 6.5 \text{ mm}$ for positions
 459 $Z = 100 \text{ mm}$ and $Z = 200 \text{ mm}$, but underestimates in the remaining size classes for all three positions.
 460 Behind the obstacle, more coalescence events take place leading to a coalescence-dominant trend in the
 461 experiments, which is obviously under-predicted by the model. From position $Z = 200 \text{ mm}$ to
 462 $Z = 400 \text{ mm}$ bubble breakup is slightly overweighted, while the model predicts almost no change.
 463 However, similar to position $Z = -200 \text{ mm}$, good agreement is captured at position $Z = 400 \text{ mm}$ again.
 464 The updated MUSIG model predicts a slightly larger coalescence to breakup ratio, which leads to a
 465 larger mean bubble size as shown in Figure 11.

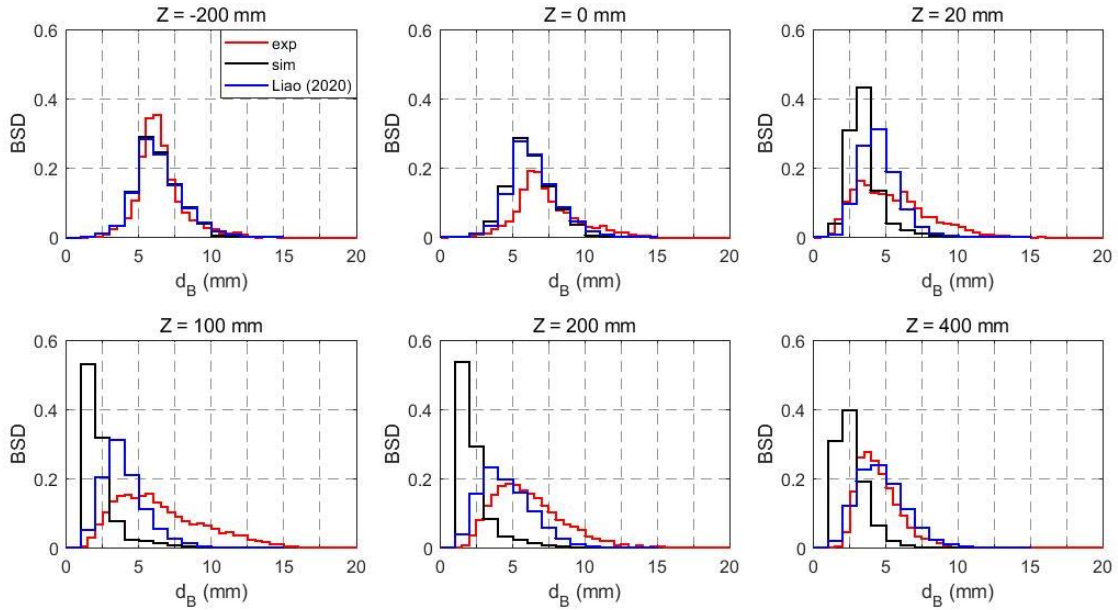


466

467 Figure 19: Bubble size distribution at different sections for test case 072 (a legend is given in the first graph).

468

468 For the standard MUSIG model, test case 074 shows similar tendencies as compared to test case 072.
 469 Good agreement is obtained at position $Z = -200 \text{ mm}$. However, a slight coalescence trend from position
 470 $Z = -200 \text{ mm}$ to $Z = 0 \text{ mm}$ is observed in the experiment, while in the simulation coalescence and
 471 breakup are almost in balance. In addition, from position $Z = 0 \text{ mm}$ to $Z = 20 \text{ mm}$ breakup seems to be
 472 dominant, which may be captured in the simulation, but the breakup rate is highly overestimated. One
 473 of the reasons may be the overprediction of the turbulence dissipation rate. Further, slight coalescence
 474 is observed again in the experiment from position $Z = 0 \text{ mm}$ to $Z = 20 \text{ mm}$ that cannot be captured by
 475 the simulation. Unlike for test case 072, the obvious inconsistency between experiment and simulation
 476 continues to exist also at position $Z = 400 \text{ mm}$.



477

478 Figure 20: Bubble size distribution at different sections for test case 074 (a legend is given in the first graph).

479 On the one hand, the updated MUSIG model gives similar results to standard one at the positions
 480 $Z = -200$ mm and $Z = 0$ mm, where the coalescence and breakup is nearly in equilibrium. On the other
 481 hand, it improves the downstream results significantly, and captures the coalescence trend well. The
 482 improvement increases from position $Z = 20$ mm to $Z = 400$ mm. However, there are still notable
 483 inconsistencies between experiments and updated MUSIG predictions at the positions $Z = 20$ mm and
 484 $Z = 100$ mm where the flow is highly complex.

485

486 6. Conclusion

487 We investigated the performance of state-of-the-art bubble breakup and coalescence modelling for 3D
 488 bubbly flow around an obstacle in a pipe. For this purpose, we performed simulations with the
 489 homogeneous MUSIG model for two different liquid velocities and compared gas velocity, void
 490 fraction, mean bubble diameter and bubble size distribution with 3D UFXCT data. A main feature of
 491 the flow is that there is a vortex region behind the obstacle. This vortex region causes a strong gas
 492 accumulation downstream the obstacle. While in the high-velocity case void accumulates directly
 493 behind the obstacle it does so further downstream in the low-velocity case. The reason is that the obstacle
 494 leads to a downstream wake region whose length increases with velocity. Thus, this wake region causes
 495 void accumulation depending on the area where it acts.

496 While the prediction of axial and radial void fractions for the low-velocity case agrees generally well
 497 with the experimental results for both the upstream and downstream region of the obstacle, the prediction
 498 for the high-velocity case overestimates the average void fraction compared to experimental data
 499 downstream of the obstacle. Moreover, downstream of the obstacle the average bubble diameter and
 500 average gas velocity are underestimated while they are in good agreement for both the low-velocity and
 501 high-velocity cases upstream of the obstacle.

502 Considering the BSD, we found that coalescence is the dominant mechanism behind the obstacle for the
 503 low-velocity case in the experiments. However, the simulations could not capture it. In addition, for the
 504 high-velocity case the breakup rate is highly overestimated compared to the experimental data. The
 505 difference between experimental data and numerical results may be because of the overestimation of
 506 turbulence, which is related to the shear-induced and bubble-induced turbulence models. Simulation

507 results show that the predicted coalescence rate of all mechanisms might be negligible in comparison to
508 breakup rates since all the coalescence mechanisms provide approximately the same predictions. In
509 addition, it is clear that turbulence is the dominant mechanism for both breakup and coalescence.

510 The performance of the updated MUSIG model recently developed by Liao (2020) was also tested in
511 the present study. The results show that the updated MUSIG model predicts an obviously lower breakup
512 rate. Consequently, it is able to reproduce the coalescence-dominant trend for the low-velocity test, and
513 predicts a reasonable mean bubble size for the high-velocity test, where turbulence-induced breakup is
514 significant. Investigation on the effect of two-phase turbulence models and improving the coalescence
515 and breakup model in the updated MUSIG model will be topics of future work.

516 As already mentioned, the low-velocity case shows good agreement with the experimental data
517 downstream of the obstacle, whereas this is not the case for the high-velocity case. The explanation of
518 void fraction overestimation for the high-velocity case may be due to more than one reason. Firstly,
519 applied bubble force models, which are widely used in the literature, do not consider the turbulence
520 effects. Turbulence effects are thought to have an impact, especially on drag force. Secondly, the bubble
521 breakup and coalescence model may need to be improved. Thirdly, the applicability of bubble-induced
522 turbulence that was used for the simulations is still not clear for high shear-induced turbulence cases,
523 since it has been obtained under low turbulence conditions.

524 **References**

- 525 Auton, T. R., Hunt, J. C. R., & Prudhomme, M. (1988). The Force Exerted on a Body in Inviscid
526 Unsteady Non-Uniform Rotational Flow. *Journal of Fluid Mechanics*, 197, 241-257.
527 doi:10.1017/S0022112088003246
- 528 Besagni, G., Guedon, G. R., & Inzoli, F. (2018). Computational fluid-dynamic modeling of the mono-
529 dispersed homogeneous flow regime in bubble columns. *Nuclear Engineering and Design*,
530 331, 222-237. doi:10.1016/j.nucengdes.2018.03.003
- 531 Burns, A. D., Frank, T., Hamill, I., & Shi, J.-M. (2004). *The Favre averaged drag model for turbulent*
532 *dispersion in Eulerian multi-phase flows*. Paper presented at the 5th International Conference
533 on Multiphase Flow, ICMF, Yokohama, Japan.
- 534 Colombo, M., & Fairweather, M. (2015). Multiphase turbulence in bubbly flows: RANS simulations.
535 *International Journal of Multiphase Flow*, 77, 222-243.
536 doi:10.1016/j.ijmultiphaseflow.2015.09.003
- 537 Colombo, M., & Fairweather, M. (2019). Influence of multiphase turbulence modelling on interfacial
538 momentum transfer in two-fluid Eulerian-Eulerian CFD models of bubbly flows. *Chemical*
539 *Engineering Science*, 195, 968-984. doi:10.1016/j.ces.2018.10.043
- 540 Frank, T., Zwart, P. J., Krepper, E., Prasser, H. M., & Lucas, D. (2008). Validation of CFD models for
541 mono- and polydisperse air-water two-phase flows in pipes. *Nuclear Engineering and Design*,
542 238(3), 647-659. doi:10.1016/j.nucengdes.2007.02.056
- 543 Hosokawa, S., Tomiyama, A., Misaki, S., & Hamada, T. (2002). *Lateral migration of single bubbles*
544 *due to the presence of wall*. Paper presented at the Proceedings of the ASME Joint U.S.-
545 European Fluids Engineering Division Conference, FEDSM2002, Montreal, Canada.
- 546 Ishii, M., & Zuber, N. (1979). Drag Coefficient and Relative Velocity in Bubbly, Droplet or
547 Particulate Flows. *Aiche Journal*, 25(5), 843-855. doi:10.1002/aic.690250513
- 548 Jareteg, K., Strom, H., Sasic, S., & Demaziere, C. (2017). On the dynamics of instabilities in two-fluid
549 models for bubbly flows. *Chemical Engineering Science*, 170, 184-194.
550 doi:10.1016/j.ces.2017.03.063
- 551 Jin, D., Xiong, J. B., & Cheng, X. (2019). Investigation on interphase force modeling for vertical and
552 inclined upward adiabatic bubbly flow. *Nuclear Engineering and Design*, 350, 43-57.
553 doi:10.1016/j.nucengdes.2019.05.005

554 Krepper, E., Beyer, M., Frank, T., Lucas, D., & Prasser, H. M. (2009). CFD modelling of
555 polydispersed bubbly two-phase flow around an obstacle. *Nuclear Engineering and Design*,
556 239(11), 2372-2381. doi:10.1016/j.nucengdes.2009.06.015

557 Liao, Y. X. (2020). Update to the MUSIG model in ANSYS CFX for reliable modelling of bubble
558 coalescence and breakup. *Applied Mathematical Modelling*, 81, 506-521.
559 doi:10.1016/j.apm.2020.01.033

560 Liao, Y. X., Lucas, D., Krepper, E., & Schmidtke, M. (2011). Development of a generalized
561 coalescence and breakup closure for the inhomogeneous MUSIG model. *Nuclear Engineering
562 and Design*, 241(4), 1024-1033. doi:10.1016/j.nucengdes.2010.04.025

563 Liao, Y. X., Ma, T., Liu, L., Ziegenhein, T., Krepper, E., & Lucas, D. (2018). Eulerian modelling of
564 turbulent bubbly flow based on a baseline closure concept. *Nuclear Engineering and Design*,
565 337, 450-459. doi:10.1016/j.nucengdes.2018.07.021

566 Liao, Y. X., Rzehak, R., Lucas, D., & Krepper, E. (2015). Baseline closure model for dispersed bubbly
567 flow: Bubble coalescence and breakup. *Chemical Engineering Science*, 122, 336-349.
568 doi:10.1016/j.ces.2014.09.042

569 Lo, S. (1996). *Application of the MUSIG model to bubbly flows*. Paper presented at the AEAT-1096,
570 AEA Technology.

571 Luo, H., & Svendsen, H. F. (1996). Theoretical model for drop and bubble breakup in turbulent
572 dispersions. *Aiche Journal*, 42(5), 1225-1233. doi:10.1002/aic.690420505

573 Ma, T., Santarelli, C., Ziegenhein, T., Lucas, D., & Frohlich, J. (2017). Direct numerical simulation-
574 based Reynolds-averaged closure for bubble-induced turbulence. *Physical Review Fluids*,
575 2(3). doi:10.1103/PhysRevFluids.2.034301

576 Menter, F. R. (1994). 2-Equation Eddy-Viscosity Turbulence Models for Engineering Applications.
577 *Aiaa Journal*, 32(8), 1598-1605. doi:10.2514/3.12149

578 Neumann-Kipping, M., Bieberle, A., & Hampel, U. (2020). Investigations on bubbly two-phase flow
579 in a constricted vertical pipe. *International Journal of Multiphase Flow*, 130.
580 doi:10.1016/j.ijmultiphaseflow.2020.103340

581 Parekh, J., & Rzehak, R. (2018). Euler-Euler multiphase CFD-simulation with full Reynolds stress
582 model and anisotropic bubble-induced turbulence. *International Journal of Multiphase Flow*,
583 99, 231-245. doi:10.1016/j.ijmultiphaseflow.2017.10.012

584 Prasser, H. M., Beyer, M., Frank, T., Al Issaa, S., Carl, H., Pietruske, H., & Schutz, P. (2008). Gas-
585 liquid flow around an obstacle in a vertical pipe. *Nuclear Engineering and Design*, 238(7),
586 1802-1819. doi:10.1016/j.nucengdes.2007.11.007

587 Prince, M. J., & Blanch, H. W. (1990). Bubble Coalescence and Break-up in Air-Sparged Bubble-
588 Columns. *Aiche Journal*, 36(10), 1485-1499. doi:10.1002/aic.690361004

589 Rzehak, R., & Krepper, E. (2013). CFD modeling of bubble-induced turbulence. *International Journal
590 of Multiphase Flow*, 55, 138-155. doi:10.1016/j.ijmultiphaseflow.2013.04.007

591 Rzehak, R., Krepper, E., & Lifante, C. (2012). Comparative study of wall-force models for the
592 simulation of bubbly flows. *Nuclear Engineering and Design*, 253, 41-49.
593 doi:10.1016/j.nucengdes.2012.07.009

594 Rzehak, R., Ziegenhein, T., Kriebitzsch, S., Krepper, E., & Lucas, D. (2017). Unified modeling of
595 bubbly flows in pipes, bubble columns, and airlift columns. *Chemical Engineering Science*,
596 157, 147-158. doi:10.1016/j.ces.2016.04.056

597 Simonin, O., & Viollet, P. L. (1990). *Predictions of an oxygen droplet pulverization in a compressible
598 subsonic coflowing hydrogen flow*. Paper presented at the Numerical Methods for Multiphase
599 Flows, FED91.

600 Tas-Koehler, S., Lecrivain, G., Krepper, E., Unger, S., & Hampel, U. (2020). Numerical investigation
601 on the effect of transversal fluid field deformation on heat transfer in a rod bundle with mixing
602 vanes. *Nuclear Engineering and Design*, 361. doi:10.1016/j.nucengdes.2020.110575

603 Tomiyama, A., Kataoka, I., Zun, I., & Sakaguchi, T. (1998). Drag coefficients of single bubbles under
604 normal and micro gravity conditions. *Jsm International Journal Series B-Fluids and Thermal
605 Engineering*, 41(2), 472-479. doi:10.1299/jsmeb.41.472

606 Tomiyama, A., Tamai, H., Zun, I., & Hosokawa, S. (2002). Transverse migration of single bubbles in
607 simple shear flows. *Chemical Engineering Science*, 57(11), 1849-1858. doi:10.1016/S0009-
608 2509(02)00085-4

- 609 Troshko, A. A., & Hassan, Y. A. (2001). A two-equation turbulence model of turbulent bubbly flows.
610 *International Journal of Multiphase Flow*, 27(11), 1965-2000. doi:10.1016/S0301-
611 9322(01)00043-X
612 Yeoh, G. H., & Tu, J. Y. (2009). *Computational Techniques for Multiphase Flows*: Butterworth-
613 Heinemann.
- 614

CFD simulation of bubbly flow around an obstacle in a vertical pipe with a focus on breakup and coalescence modelling

Sibel Tas-Koehler ^{a*}, Martin Neumann-Kipping ^b, Yixiang Liao ^a, Eckhard Krepper ^a, Uwe Hampel ^{a,b}

^a Helmholtz-Zentrum Dresden-Rossendorf, Institute of Fluid Dynamics, Bautzner Landstr. 400, 01328 Dresden, Germany

^b Technische Universität Dresden, Chair of Imaging Techniques in Energy and Process Engineering, 01062 Dresden, Germany

* CORRESPONDING AUTHOR (E-MAIL: s.tas@hzdr.de)

ABSTRACT

In the present study, we assessed the capabilities of Eulerian-Eulerian CFD two-phase flow simulation with the homogeneous Multiple Size Group Model (MUSIG) and consideration of breakup and coalescence under three-dimensional flow conditions. We compared void fraction, bubble size and bubble velocity distributions against experimental data from vertical gas-disperse two-phase flow in a pipe with a flow obstruction. The simulation results generally agree well upstream the obstacle, where we have a typically developed pipe flow. Downstream of the obstacle void fraction is overpredicted while bubble velocity is underpredicted. The bubble size distribution has no clear trend. With higher liquid velocities, the deviations increase. As a conclusion, the simulation has difficulties to balance the gas fraction in the strong vortex in the shadow of the obstacle. Here further model improvement is needed.

Keywords: CFD, bubbly flow, model validation, bubble breakup, bubble coalescence

Nomenclature

Latin symbols

d_B	bubble diameter [m]	S_K	source term due to turbulent kinetic energy [N·m ⁻² ·s ⁻¹]
d_{cr}	critical bubble diameter [m]	S_ε	source term due to turbulent dissipation rate [N·m ⁻² ·s ⁻²]
C_D	drag coefficient [-]	S_ω	source term due to turbulent frequency [N·m ⁻⁴]
C_L	lift coefficient [-]	u	velocity [m·s ⁻¹]
C_W	wall lubrication coefficient [-]	Greek symbols	
C_{VM}	virtual mass coefficient [-]	α	gas volume fraction [-]
d_\perp	maximum horizontal bubble dimension [mm]	ρ	density [kg·m ⁻³]
EO_\perp	Eötvös number depending on d_\perp [-]	τ	bubble-induced time scale [s]
EO	Eötvös number [-]	τ_{ij}^{Lam}	laminar stress tensor [kg·m ⁻¹ ·s ⁻²]
F_D	drag force per unit volume [N·m ⁻³]	τ_{ij}^{Turb}	turbulent stress tensor [kg·m ⁻¹ ·s ⁻²]
F_L	lift force per unit volume [N·m ⁻³]	ω	turbulence frequency [s ⁻¹]
F_W	wall lubrication force per unit volume [N·m ⁻³]	ν_L^{turb}	kinematic viscosity [m ² ·s ⁻¹]
F_{Disp}	turbulent dispersion force per unit volume [N·m ⁻³]	σ_{TD}	Schmidt number [-]
F_{VM}	virtual mass force per unit volume [N·m ⁻³]	ε	turbulence dissipation rate [m ² ·s ⁻³]
k	phase indicator, turbulent kinetic energy [m ² ·s ⁻²]	Subscripts	
M_i	source term in i-th direction [kg·m ⁻² ·s ⁻²]	B	bubble
p	pressure [Pa]	L	liquid phase
Re	Reynolds number [-]	G	gas phase

26

27 Acronyms

28 BIT: Bubble-Induced Turbulence

29 CFD: Computational Fluid Dynamics

30 FAD: Favre-Averaged Drag

31 MUSIG: Multiple Size Group Model

32 SST: Shear Stress Transport

33 UFXCT: Ultrafast X-ray Computed Tomography

34 1. Introduction

35 Bubbly flows are encountered in different industrial applications such as chemical, petroleum and
36 nuclear engineering. In these applications, it is important to know the flow behavior to enhance
37 efficiency such as in chemical reactors or to increase safety margins such as in nuclear reactors. For this
38 purpose, computational fluid dynamics (CFD) that is based on the Eulerian-Eulerian framework has
39 become a popular tool. However, the accuracy of CFD simulations highly relies on correct modeling of
40 phase interactions including interfacial forces (i.e. drag, lift, wall lubrication, turbulent dispersion, and
41 virtual mass), bubble-induced turbulence (BIT) and bubble breakup/coalescence. The modeling of the
42 latter one is possible by a poly-dispersed approach in which the gas phase is divided into a number of
43 size groups. In most of the industrial applications, there is a wide distribution of bubble sizes and a non-
44 uniform radial gas fraction profiles. Thus, the correct prediction of bubble size by breakup and
45 coalescence plays an important role in the accuracy of CFD simulations.

46 For adiabatic bubbly flow, numerous numerical studies were done, especially focusing on the interfacial
47 forces and BIT. Frank *et al.* (2008) indicated for two-phase pipe flow that a mono-dispersed simulation
48 model, including the Tomiyama lift force, the Frank wall lubrication force, the Favre-averaged drag
49 (FAD) turbulent dispersion force and the Shear Stress Turbulence (SST) model, gives good agreement
50 with the experimental data in terms of gas void fraction. Besagni *et al.* (2018) investigated the effects
51 of the interfacial forces for small-scale and large-scale bubble columns with the mono-dispersed
52 approach. The authors suggested a baseline model with the following interfacial forces: the Tomiyama
53 drag force, the Antal wall lubrication force and the Lopez or Burns turbulence dispersion force. Jin *et al.*
54 (2019) investigated the influence of different models of interfacial forces on the phase distribution
55 for vertical and inclined bubbly flow. They reported that the combination of the Ishii-Zuber drag force,
56 the Saffman-Mei lift force, the Hosokawa wall lubrication force and the FAD turbulent dispersion force
57 provides good radial void fraction results for vertical bubbly flow. Rzehak *et al.* (2012) examined
58 different wall lubrication force models in case of bubbly flow. They compared the Antal, Tomiyama
59 and Hosokawa wall force models and found that the Hosokawa model provides the best performance.
60 Jareteg *et al.* (2017) investigated the effect of a virtual mass force on the stability of the bubbly flow
61 simulations. The authors showed that the implementation of virtual mass force importantly changes the
62 growth rate of void instabilities. Colombo *et al.* (2019) showed the capability of Eulerian-Eulerian CFD
63 for a bubbly flow in a pipe and square duct by focusing on the lift and turbulence forces. According to
64 the results, the effect of turbulence on the phase distribution is as important as the lift force. They further
65 concluded that the wall lubrication force is not necessary if the near-wall region is appropriately
66 resolved.

67 BIT was also taken into account in many studies. Rzehak *et al.* (2017) developed a closure model for
68 bubbly flow simulations including bubble forces and BIT. However, they did not consider bubble
69 breakup and coalescence yet. The simulation results were compared to experimental data for gas volume
70 fraction, axial liquid velocity and turbulent kinetic energy. Although an overall satisfying agreement
71 between experiments and simulations was found, the authors pointed out that further improvements in
72 turbulence modeling and implementation of bubble breakup/coalescence are highly necessary. Colombo
73 *et al.* (2015) presented the validation of the two-phase Eulerian-Eulerian mono-dispersed model for pipe
74 flow by using experimental data from 6 different literature sources. They reported that their BIT model
75 gives better results in terms of r.m.s velocity fluctuations as compared to Troshko *et al.* (2001) and
76 Rzehak *et al.* (2013). Besagni *et al.* (2018) reported that while BIT inclusion causes convergence
77 problems for large-scale bubble columns, the model of Sato improves the results slightly compared to
78 Simonin *et al.* (1990) for small-scale bubble columns. Parekh *et al.* (2018) compared Launder, Reece,
79 Rodi (LRR) RSM and Speziale-Sarkar-Gatski (SSG) RSM turbulence models as well as the SST model

80 for air-water pipe flow to capture the anisotropy of turbulent fluctuations concerning BIT. The
81 simulation results showed that predictions of LRR and SSG RSM including BIT are comparable to the
82 SST model over radial profiles of the liquid velocity and gas fraction. However, all three models
83 underestimated the wall peaks of the turbulent kinetic energy and Reynolds stresses by comparing them
84 with the experimental data. Liao *et al.* (2018) applied the MUSIG approach for air-water bubbly flow
85 to test the performance of the BIT model developed by Ma *et al.* (2017). The results showed that the
86 model of Ma *et al.* (2017) predicts the radial gas void fraction and gas velocity well compared to the
87 experiments.

88 Bubble breakup and coalescence was less considered in the literature compared to interfacial forces and
89 BIT. Frank *et al.* (2008) showed that the inhomogeneous MUSIG (i-MUSIG) model predicts radial void
90 fraction profiles well but further investigation is needed in terms of bubble breakup and coalescence.
91 Liao *et al.* (2015) performed simulations to assess the capability of the bubble breakup and coalescence
92 modeling which was proposed by Liao *et al.* (2011). The results showed that the mean bubble size is
93 overestimated at low superficial liquid velocities and is slightly underestimated at high superficial liquid
94 velocities. The authors concluded that further studies are necessary considering BIT, which has a high
95 impact on bubble breakup and coalescence mechanisms.

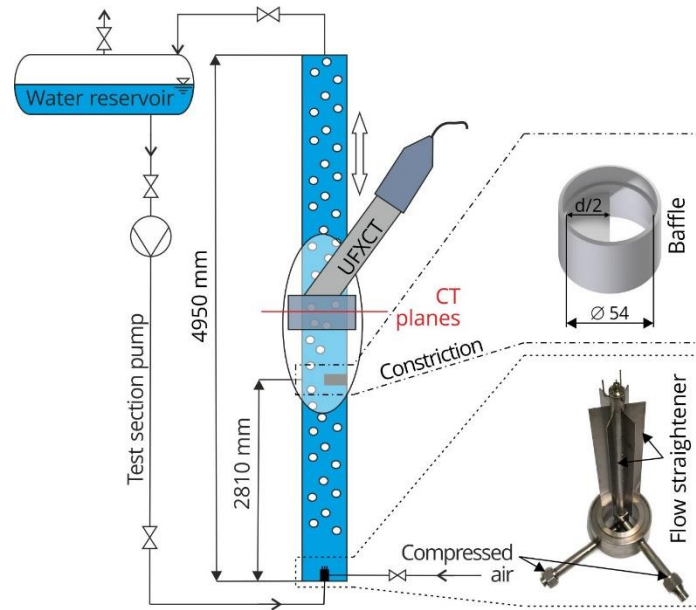
96 While CFD codes have meanwhile been well qualified for simple pipe and column geometries there is
97 yet little analysis and qualification for more complex three-dimensional flow domains. One reason is
98 the lack of appropriate experimental validation data. Prasser *et al.* (2008) provided such data for bubbly
99 flow in a pipe with a semi-circular obstacle using a wire-mesh sensor. Follow-up CFD simulations
100 assuming mono-disperse bubbles provided good void fraction estimation upstream the obstacle but
101 overestimated void fraction downstream. Krepper *et al.* (2009) used the i-MUSIG model including
102 bubble breakup and coalescence and found that the standard breakup and coalescence models (Luo *et*
103 *al.* (1996), Prince *et al.* (1990)) do not predict the bubble size distribution (BSD) well. Continuing in
104 this line the objectives of this study were to show the capability of bubble breakup and coalescence
105 modelling (Liao *et al.* (2015)) under complex flow conditions with new high-resolution two-phase flow
106 data around an obstacle in a pipe. The 3D flow field was simulated for two different liquid velocity
107 conditions using the MUSIG model. For the simulations, a baseline model for air-water bubbly flow
108 was applied that includes the model of Rzehak *et al.* (2017) for interfacial forces, the model of Liao *et*
109 *al.* (2015) for the bubble breakup/coalescence and the model of Ma *et al.* (2017) for the BIT. Moreover,
110 the performance of a new discretization scheme for bubble coalescence and breakup modelling
111 developed by Liao (2020) was also tested within this work.

112 By validation against experimental benchmark data, the capability of bubble breakup/coalescence model
113 is demonstrated in terms of gas volume fraction, gas velocity, mean bubble diameter and bubble size
114 distribution with the aid of Ultrafast X-ray Computed Tomography (UFXCT) measurements
115 (Neumann-Kipping *et al.* (2020)). Furthermore, hydrodynamics of the 3D bubbly flow is analyzed and
116 bubble interaction within the vortex region formed by the obstacle is examined under different liquid
117 velocities. Thus, this paper contributes to two main fields: (i) the assessment of breakup and coalescence
118 model accuracy under 3D flow conditions: (ii) understanding of two-phase flow hydrodynamics in
119 complex geometries.

120 **2. Experimental setup**

121 The experimental study used for validation is described in detail in Neumann-Kipping *et al.* (2020). The
122 experiments were performed in an acrylic pipe with an inner diameter of 54 mm and a total length of
123 4950 mm. A semi-circular obstacle that blocks half of the inner pipe cross-section was utilized to
124 generate 3D flow fields. A sketch of the test facility, as well as the gas injection module, is illustrated

125 in Figure 1. The experiments were performed at 4 bar pressure and a liquid temperature of 30°C.
 126 Deionized water and compressed air are injected at the bottom of the test section as the liquid and gas
 127 phase, respectively.



128
 129 Figure 1: Schematic representations of the vertical test section (left) with details of the gas injection module
 130 (bottom right) and the flow obstacle for generation of three-dimensional flow fields (top right).

131 Various operating conditions in bubbly flow regime were tested by setting appropriate liquid and gas
 132 flow rates. Two operating conditions that are used for the present study are described in Table 1.

133 Table 1: Experimental operating conditions based on combinations of liquid and gas superficial velocities.

Test run	j_l [$\text{m}\cdot\text{s}^{-1}$]	j_g [$\text{m}\cdot\text{s}^{-1}$]
#072	0.4050	0.0368
#074	1.0170	0.0368

134 Ultrafast X-ray computed tomography (UFXCT), which is a well-established non-invasive imaging
 135 technique for multiphase flow, was applied to quantitatively analyze the distribution of gas and liquid
 136 within the test section. The UFXCT scanner can be freely moved to allow for imaging of the flow field
 137 in several imaging planes up- and downstream of the flow obstacle, as depicted in Table 2 (Neumann-
 138 Kipping *et al.* (2020)). By the means of UFXCT, cross-sectional information of the gas velocity and gas
 139 volume fraction, as well as bubble size distribution were determined. A detailed discussion of
 140 measurement uncertainty UFXCT and quality evaluation of the experimental results can be found in
 141 Neumann-Kipping *et al.* (2020). Here, the time-averaged cross-sectional gas holdup and axial gas
 142 velocity were used to calculate the inlet superficial gas velocity. This estimated velocity was compared
 143 to the set value, showing maximum deviation of $\pm 15\%$ for all cases.

144

145 Table 2: Image plane identifier along the vertical test section with relative distances of the upper image plane to
 146 the center of the flow constriction.

Identifier	A	B	C	D	E	F	G	H	I
Z (mm)	-200	-60	0	5	20	50	100	200	400

147

148 3. Numerical method

149 3.1 General remarks

150 For simulation, the geometry was defined as a vertical half tube using a symmetry xz-plane. All
 151 simulations were performed using the solver ANSYS CFX 19.2. The fluid domain was modelled from
 152 1.5 m upstream to 1 m downstream the obstacle (Figure 2). The results were obtained by applying the
 153 Multiple Size Group Model (Lo (1996)). The dispersed phase was divided into 15 size fractions where
 154 the bubble diameters ($d_b = 0 \dots 15 \text{ mm}$) were defined with equidistant bubble diameter. As inlet
 155 conditions, a developed flow condition for liquid velocity, liquid turbulent kinetic energy and turbulent
 156 dissipation obtained from previous single-phase simulations were assigned. Further, experimentally
 157 determined radial void fraction and bubble size distribution at the largest upstream ($Z = -211 \text{ mm}$)
 158 position were applied. A constant pressure was defined as outlet condition. All the simulations were
 159 performed at steady state condition. Turbulence was modeled only for the liquid phase using the SST
 160 model (Menter (1994)). The dimensionless wall distance value for the liquid phase y^+ was kept greater
 161 than 30. The single-phase law of wall was used for the wall treatment. The density change of gas was
 162 considered by treating the gas phase as ideal gas and its change depending on the height was considered
 163 according to

$$\rho_G(P) = \rho_{G,atm} \frac{P_H}{P_{atm}}, \quad (1)$$

164 where P_{atm} is the atmospheric pressure and $\rho_{G,atm}$ is the gas density depending on atmospheric
 165 pressure. P_H is the pressure depending on the height and it is calculated with the following equation:

$$P_H = 4P_{atm} + \rho_L g h. \quad (2)$$

166 Here, ρ_L is the liquid density, g is the gravitational acceleration and h is the height. On the pipe wall, a
 167 no-slip condition was applied for the liquid phase and a free-slip condition for the gas phase. The
 168 convergence criteria were set to $RMS < 10^{-6}$.

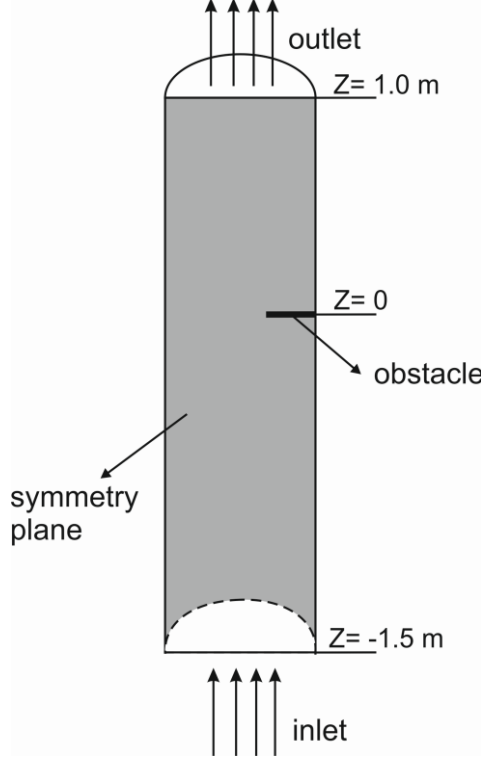


Figure 2: Schematic view of computational domain.

169

170

171

172 3.2 Governing equations

173 An Eulerian-Eulerian two-fluid model was applied in the present simulations. This framework defines
 174 every phase by a set of averaged conservation equations. Detailed information and derivations of the
 175 conservation equations were given by Yeoh *et al.* (2009). Since there is no heat transfer considered for
 176 the current study, the conservation equations include only the continuity equation

$$\frac{\partial(\alpha_k \rho_k)}{\partial t} + \frac{\partial}{\partial x_i} (\alpha_k \rho_k \mathbf{u}_{i,k}) = 0 \quad (3)$$

177 and the momentum equation

$$\begin{aligned} \frac{\partial}{\partial t} (\alpha_k \rho_k \mathbf{u}_{i,k}) + \frac{\partial}{\partial x_i} (\alpha_k \rho_k \mathbf{u}_{i,k} \mathbf{u}_{j,k}) \\ = -\alpha_k \frac{\partial p_k}{\partial x_i} + \frac{\partial}{\partial x_j} [\alpha_k (\tau_{ij,k}^{Lam} + \tau_{ij,k}^{Turb})] + \alpha_k \rho_k \mathbf{g}_i + \mathbf{M}_{i,k}. \end{aligned} \quad (4)$$

178 Here, k is the phase indicator, α is the volume fraction, ρ is the density, \mathbf{u}_i is the velocity component in
 179 the i -th direction, p is the pressure, τ_{ij}^{Lam} is the laminar stress tensor, τ_{ij}^{Turb} is the turbulence stress
 180 tensor and \mathbf{M}_i is the source term in the i -th direction, which will be explained in the following section.

181 3.3 Interfacial momentum transfer

182 The Eulerian-Eulerian framework considers interpenetrating continua and therefore closure models for
 183 interfacial momentum transfer are applied by means of a source term of the form

$$\mathbf{M}_k = \mathbf{F}_D + \mathbf{F}_L + \mathbf{F}_W + \mathbf{F}_{Disp} + \mathbf{F}_{VM} \quad (5)$$

184 that is added in the momentum equation including drag force \mathbf{F}_D , lift force \mathbf{F}_L , wall lubrication force
 185 \mathbf{F}_W , turbulent dispersion force \mathbf{F}_{Disp} and virtual mass force \mathbf{F}_{VM} . In the present study, all these forces
 186 were considered for the simulations.

187 The drag force

$$\mathbf{F}_D = -\frac{3 C_D}{4 d_B} \alpha \rho_L |\mathbf{u}_G - \mathbf{u}_L| (\mathbf{u}_G - \mathbf{u}_L) \quad (6)$$

188 acts opposite to the relative motion of bubbles relative to the surrounding liquid. Here, d_B is the bubble
 189 diameter, α is the gas void fraction, ρ_L is the liquid density, \mathbf{u}_G is the gas velocity, \mathbf{u}_L is the liquid
 190 velocity and C_D is the drag coefficient calculated by the correlation of Ishii *et al.* (1979).

191 The lift force

$$\mathbf{F}_L = -C_L \alpha \rho_L (\mathbf{u}_G - \mathbf{u}_L) \times (\nabla \times \mathbf{u}_L) \quad (7)$$

192 occurs due to the interaction of the bubble with the shear flow of the liquid. Here, C_L is the lift force
 193 coefficient and determined by Tomiyama *et al.* (2002). The lift force coefficient changes its sign from
 194 positive to negative if the bubble diameter exceeds a critical bubble diameter. For water-air at ambient
 195 conditions, as is the case here, this critical diameter has a value of $d_{cr} = 5.8 \text{ mm}$. (Tomiyama *et al.*
 196 (1998)).

197 The wall lubrication force

$$\mathbf{F}_W = \frac{2}{d_B} C_W \rho_L \alpha |\mathbf{u}_G - \mathbf{u}_L|^2 \hat{\mathbf{y}} \quad (8)$$

198 drives the bubbles away from the wall to avoid the maximum gas fraction at the wall. Here, $\hat{\mathbf{y}}$ is the unit
 199 normal perpendicular to the wall and C_W is the wall force coefficient. The model of Hosokawa *et al.*
 200 (2002) was applied to predict the wall force coefficient.

201 The turbulent dispersion force

$$\mathbf{F}_{Disp} = -\frac{3 C_D}{4 d_B} \alpha \rho_L |\mathbf{u}_G - \mathbf{u}_L| \frac{\nu_L^{turb}}{\sigma_{TD}} \left(\frac{1}{(1-\alpha)} + \frac{1}{\alpha} \right) \nabla \alpha \quad (9)$$

202 describes the impact of liquid phase turbulent fluctuations on the gas phase. Here, ν_L^{turb} is the kinematic
 203 viscosity of the liquid phase and σ_{TD} is the Schmidt number, generally taken as 0.9. The turbulent
 204 dispersion force was modeled by Burns *et al.* (2004).

205 The virtual mass force

$$\mathbf{F}_{VM} = -C_{VM} \alpha \rho_L \left(\frac{D_G \mathbf{u}_G}{Dt} - \frac{D_L \mathbf{u}_L}{Dt} \right) \quad (10)$$

206 acts on the bubbles in case of bubble sudden acceleration. Here, C_{VM} is the virtual mass coefficient,
 207 which was set to 0.5 for the simulations. Table 3 shows the equations for calculating the force
 208 coefficients.

Table 3: Mathematical description for interfacial force coefficients.

Force	Reference	Mathematical description
Drag	Ishii <i>et al.</i> (1979)	$C_D = \max[C_{D_{sphere}}, \min(C_{D_{ellipse}}, C_{D_{cap}})]$
		$C_{D_{sphere}} = \frac{24}{Re_d} (1 + 0.1Re_d^{3/4}), C_{D_{ellipse}} = \frac{2}{3} \sqrt{Eo}, C_{D_{cap}} = \frac{8}{3}$
Lift	Tomiyama <i>et al.</i> (2002)	$C_L = \begin{cases} \min[0.288 \tanh(0.121Re), f(Eo_\perp)] & Eo_\perp < 4 \\ f(Eo_\perp) & 4 < Eo_\perp < 10 \\ -0.27 & 10 < Eo_\perp \end{cases}$
		$f(Eo_\perp) = 0.00105Eo_\perp^3 - 0.0159Eo_\perp^2 - 0.0204Eo_\perp + 0.474$
Wall lubrication	Hosokawa <i>et al.</i> (2002)	$Eo_\perp = \frac{g(\rho_L - \rho_L)d_\perp^2}{\sigma} \quad d_\perp = d_B \sqrt[3]{1 + 0.163 Eo^{0.757}}$ $C_W(y) = f(Eo) \left(\frac{d_B}{2y}\right)^2, \quad f(Eo) = 0.021Eo$
Turbulent dispersion	Burns <i>et al.</i> (2004)	Favre averaging the drag force
Virtual mass	Auton <i>et al.</i> (1988)	Constant coefficient $C_{VM} = 0.5$

210

211 3.4 Bubble induced turbulence

212 Two turbulence sources affect the gas-liquid two-phase flow. The first one is the shear-induced
 213 turbulence that calculates the turbulence parameters by the applied single-phase flow turbulence model.
 214 Therefore, the SST turbulence model was applied to the continuous phase in this study. Dispersed phase
 215 turbulence was obtained from continuous phase calculations. This approximation is valid for flows with
 216 a low-density ratio, such as air-water flows (Colombo *et al.* (2015)).

217 The second one is the BIT, which accounts for the turbulence generation due to bubble-liquid
 218 interaction. Ma *et al.* (2017) proposed a BIT model with the source terms

$$S_K = C_I F_D (\mathbf{u}_G - \mathbf{u}_L), \quad (11)$$

$$S_\varepsilon = \frac{C_\varepsilon}{\tau} S_K, \quad (12)$$

219 and

$$S_\omega = \frac{1}{C_\mu k} S_\varepsilon - \frac{\omega}{k} S_K. \quad (13)$$

220 Here, k is the turbulent kinetic energy, ω is the turbulent frequency, C_μ is the shear-induced turbulence
 221 coefficient that is taken as 0.09, τ is the BIT time scale

$$\tau = \frac{d_B}{|\mathbf{u}_G - \mathbf{u}_L|} \quad (14)$$

222 and C_I and C_ε are the model coefficients defined as

$$C_I = \min(0.18Re_B^{0.23}, 1) \quad (15)$$

223 and

$$C_\varepsilon = 0.3C_D. \quad (16)$$

224 These BIT source terms are added in the SST turbulent equations and the turbulent viscosity is calculated
225 with the aid of standard equation

$$\mu_L^{Turb} = C_\mu \rho_L \frac{k_L^2}{\varepsilon_L}. \quad (17)$$

226

227 **3.5 Bubble breakup and coalescence**

228 As can be seen from the literature review, less attention has been paid to bubble breakup and coalescence
229 mechanisms in the Eulerian modelling of bubbly flow. However, for the bubbly flow systems, there are
230 strong interactions between bubbles from different classes, which results in bubble breakup and
231 coalescence. The MUSIG model (Lo (1996)), which is based on population balance approach was
232 applied to the simulations. In the MUSIG model, the dispersed phase is divided into M size fractions
233 and the population balance equation is used to determine the mass conservation of the size fractions
234 considering the interaction mass transfer due to bubble breakup and coalescence. The size fraction
235 equations are given as

$$\frac{\partial \rho_G \alpha f_i}{\partial t} + \frac{\partial}{\partial x_i} (\rho_G \alpha f_i \mathbf{u}_{i,G}) = B_{Ci} - D_{Ci} + B_{Bi} - D_{Bi}. \quad (18)$$

236 The source and sink terms contain the birth rates due to coalescence and breakup, B_{Ci} , B_{Bi} , and death
237 rates due to coalescence and breakup of the bubbles, D_{Ci} , D_{Bi} . They are calculated as

$$B_{Ci} = (\rho_G \alpha)^2 \left(\frac{1}{2} \sum_{j \leq i} \sum_{k \leq i} Q(m_j; m_k) X_{jki} \frac{m_j + m_k}{m_j m_k} f_j f_k \right) \quad (19)$$

$$B_{Bi} = \rho_G \alpha \sum_{j > i} g(m_j; m_i) f_j \quad (20)$$

$$D_{Ci} = (\rho_G \alpha)^2 \left(\sum_j Q(m_i; m_j) \frac{1}{m_j} f_i f_j \right) \quad (21)$$

$$D_{Bi} = \rho_G \alpha f_i \sum_{j < i} g(m_i; m_j) \quad (22)$$

238 where the functions Q denotes the coalescence rate and g denotes the breakup rate. In this context, the
 239 coalescence and breakup model of Liao *et al.* (2015) was applied to determine these rates, which
 240 considers various bubble interaction mechanisms such as turbulent fluctuation, shear, buoyancy and
 241 wake. In the simulations, the change of bubble size due to pressure changes was considered in the
 242 calculation of the Sauter mean diameter (d_{32}) by

$$d_{32}^* = d_{32}(\rho_{ref}/\rho_g)^{1/3} \quad (23)$$

243 where ρ_{ref} is the reference gas density at the inlet and ρ_g is the gas density at a certain height.

244 Recently, Liao (2020) found that the formulations from Eq. (19) to Eq. (22) preserve only the mass of
 245 bubbles but not their number when assigning the breakup/coalescence source term to size groups. A
 246 consequence is underprediction of the bubble size, especially in breakup-dominant cases. She developed
 247 an internally consistent discretization scheme for the terms of birth rates, which is:

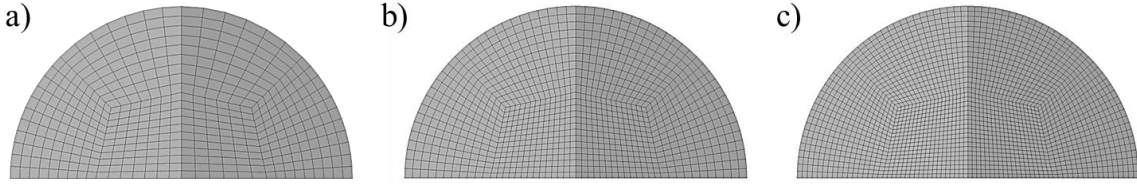
$$B_{Ci}' = (\rho_G \alpha)^2 \left(\frac{1}{2} \sum_{j \leq i} \sum_{k \leq i} Q(m_j; m_k) X_{jki} \frac{m_i}{m_j m_k} f_j f_k \right) \quad (24)$$

$$B_{Bi}' = \rho_G \alpha \sum_{j > i} \frac{m_i}{m_j} f_j \left(g(m_j; m_i) + \sum_{k < j} g(m_j; m_k) Y_{jki} \right) \quad (25)$$

248 Note that m_j+m_k in Eq. (19) has now been replaced by m_i yielding Eq. (24). The size fraction equation
 249 of Eq. (18) is derived from the population balance equation for the bubble number concentration N_i by
 250 multiplying it with m_i , so m_i instead of m_j+m_k should be contained in the coalescence source term. The
 251 computation of the breakup source term according to Eq. (25) considers two situations of the daughter
 252 bubble positioning. One is that the daughter bubble size coincides with the representative value of a size
 253 class, and the other is that the daughter bubble size lies between two representative values. In the former
 254 case, the source term can be calculated in a similar way given in Eq. (20) or the first term in the bracket
 255 of Eq. (25). In the latter one, a mass matrix Y_{jki} like X_{jki} in the coalescence representing the fraction of
 256 mass going to group i is needed for the calculation of the source term. In binary breakage, if the parent
 257 bubble and one daughter bubble are fixed at a representative value, the size of the second daughter
 258 usually does not coincide with any representative values. Therefore, the formulation in Eq. (25) is
 259 general. This scheme preserves both the mass and the number of bubbles. The effect of this discretization
 260 scheme was investigated in the present study.

261 4. Mesh independence studies

262 The flow domain was discretized using structured meshes. Mesh study was done for test 072 with three
 263 different meshes (Figure 3): 103,050 elements (mesh 1, subfigure a), 252,000 elements (mesh 2,
 264 subfigure b), and 553,850 elements (mesh 3, subfigure c). The mesh refinement was applied both in
 265 axial and lateral directions.



266

267

Figure 3: Mesh views: a) mesh 1 b) mesh 2 and c) mesh 3.

268

269

270

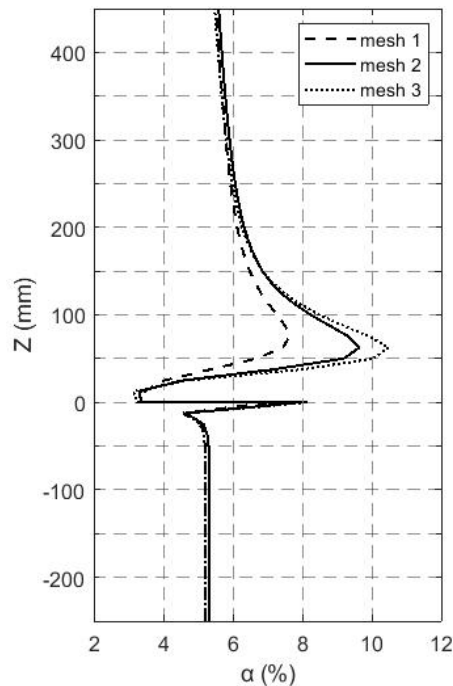
271

272

273

274

Figure 4 shows the simulated average gas fraction along the axial direction for different mesh densities. Upstream of the obstacle $Z < 0$ mm, the mean gas fraction is very much mesh independent. The effect of the mesh refinement is largely noticeable downstream the obstacle $0 \text{ mm} < Z < 200 \text{ mm}$ where high flow complexity occurs. This mesh sensibility to the flow complexity is in line with that previously reported by Tas-Koehler *et al.* (2020). Whereas a maximum relative difference between mesh 1 and mesh 2 is 29%, it is 8% between mesh 2 and mesh 3. Hence, to reduce the computational effort, mesh 2 is applied for this study.



275

276

Figure 4: Effect of grid refinement on the average gas fraction for test 072.

277 5. Results

278

5.1 Phase distribution

279

280

281

282

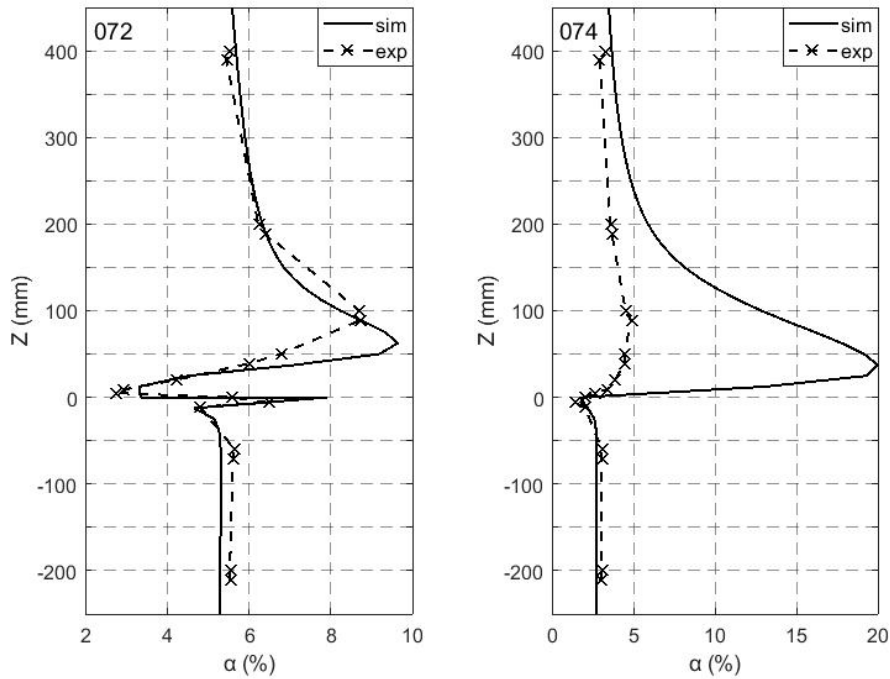
283

284

285

286

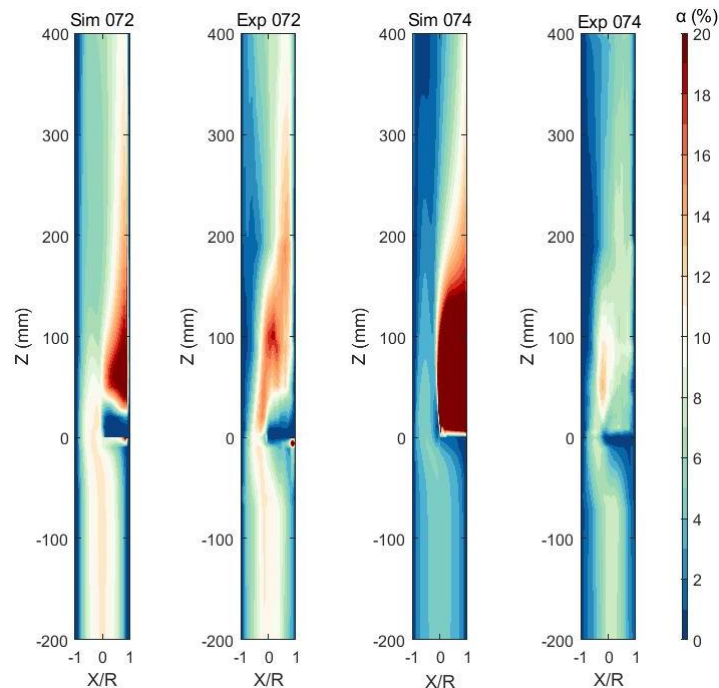
Figure 5 shows the evolution of the averaged gas fraction in the axial direction of the pipe for test cases 072 and 074. Upstream of the obstacle, both cases agree well with the experiments. For test 072, void fraction decreases dramatically downstream the obstacle. After that point, it starts to increase up to around $Z = 60$ mm and it decreases again. Although the peaks that occur after the obstacle in the experiment and simulation do not coincide, generally there is a good agreement between experiment and simulation. For test section 074, void fraction increases downstream the obstacle up to $Z = 40$ mm and it starts to decrease after that point. However, the averaged void fraction is highly overestimated downstream of the obstacle.



287

288 Figure 5: Cross-sectional averaged void fraction along the axial direction for test cases 072 and 074.

289 Figure 6 shows the void fraction for test 072 and 074. The strong gas accumulation after the obstacle
 290 for both tests is calculated. For test 072, a very small near-wall maximum void fraction region
 291 underneath the obstacle and the region with void accumulation after the obstacle are well captured by
 292 the simulation. However, in line with Figure 5 for test 074, there is a high void region after the obstacle
 293 that shows a large discrepancy in terms of its magnitude compared to the experiment. Another finding
 294 is that whereas void starts to accumulate just behind the obstacle for test 074, it begins to accumulate
 295 after a certain Z position that is around 25 mm for test 072.

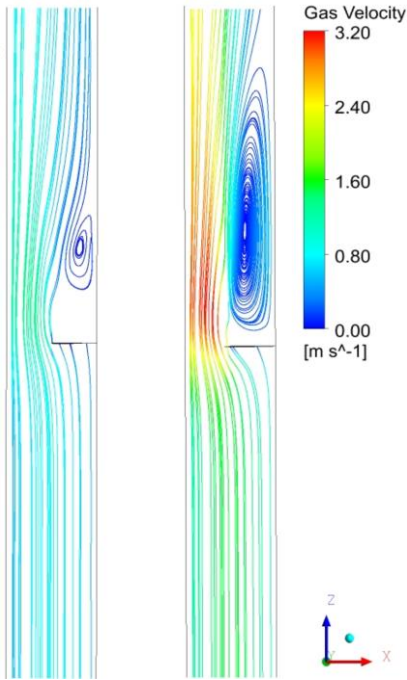


296

297 Figure 6: Visualization plots of simulated and measured void fraction for test cases 072 and 074.

298 In order to explain the different void fraction peaks of test cases 072 and 074, streamlines of the gas
 299 velocity of are shown in Figure 7 for $-200 \text{ mm} < Z < 200 \text{ mm}$. While there is a region free of bubbles

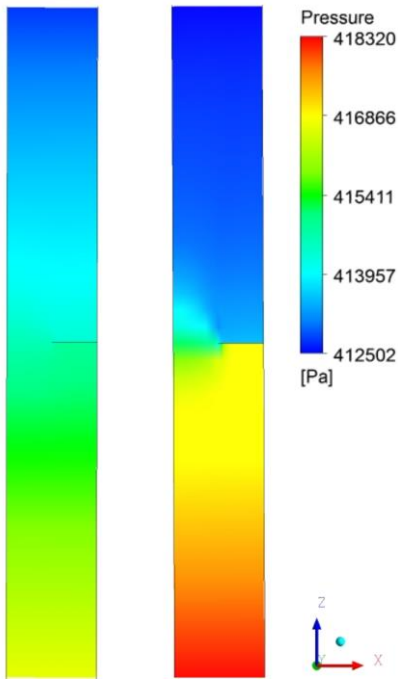
300 directly behind the obstacle for test case 072, this region nearly vanishes for test case 074. Here, the
 301 obstacle causes a downstream wake region (vortex region) due to lateral pressure differences (Figure 8).
 302 As the liquid superficial velocity increases, the pressure differences increase and so does the wake
 303 region. The vortex flow in the wake region leads to void fraction accumulation due to the density
 304 difference between the liquid and gas phases. For test case 072, the wake region appears between around
 305 $30 \text{ mm} < Z < 80 \text{ mm}$, while for test case 074 it develops between around $10 \text{ mm} < Z < 150 \text{ mm}$. This is
 306 in good accordance with the averaged void fraction peaks and changes that are shown in Figure 5 and
 307 Figure 6.



308

309

Figure 7: Streamline for test 072 (left) and 074 (right).

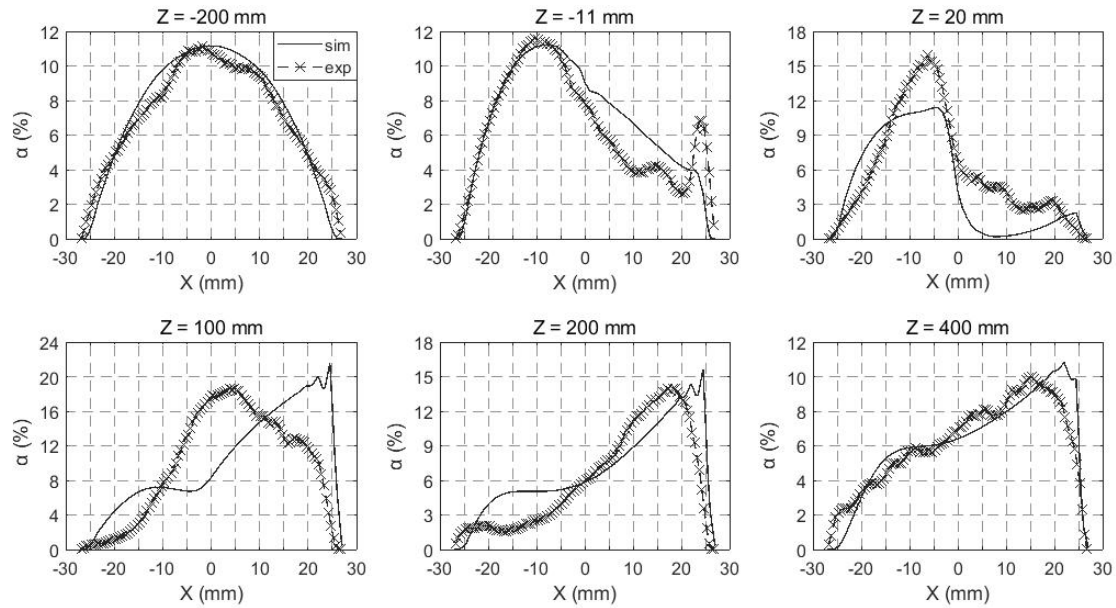


310

311

Figure 8: Pressure distribution for test 072 (left) and 074 (right).

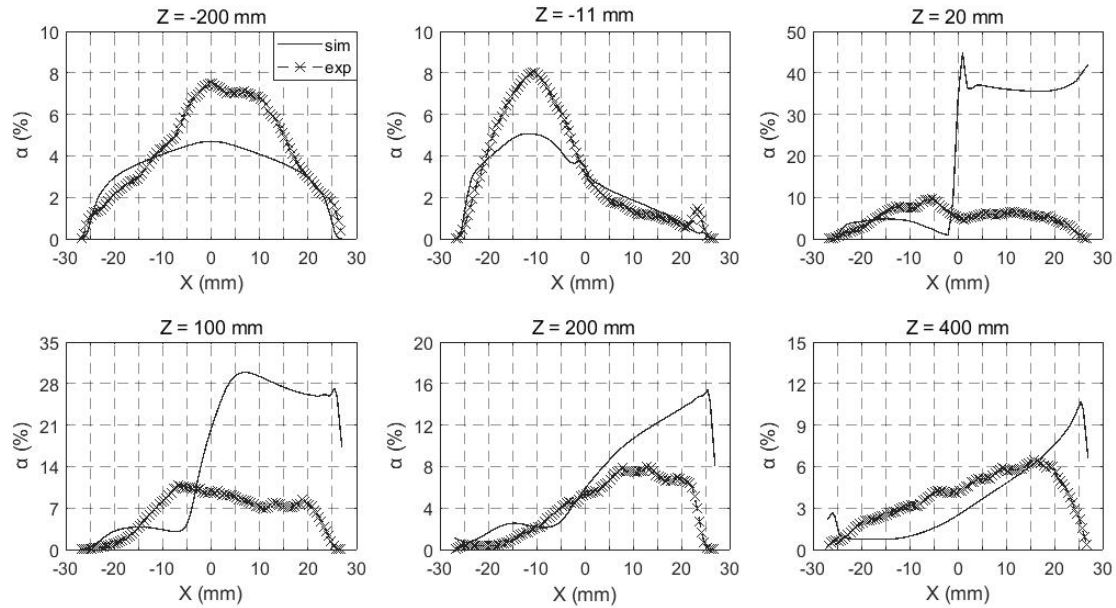
312 Radial void fraction distributions at different cross sections are given in Figure 9 and Figure 10 for both
 313 test cases 072 and 074, respectively. At $Z = -200$ mm, the results of test case 072 are in good agreement
 314 with the experiments, but in the case of 074, the void fraction is underestimated in the pipe center. At
 315 $Z = -11$ mm, better agreement for the obstructed (right) side of the pipe is obtained for test case 074 than
 316 for 072. This is in contrast to the results of the unobstructed (left) side of the pipe. At $Z = 20$ mm both
 317 test cases provide better results for the unobstructed side of the pipe. Further downstream, the
 318 inconsistencies between the experiments and simulations for both sides of the pipe significantly increase
 319 for $Z = 100$ mm and both test cases, but decrease again at higher Z positions, respectively. However,
 320 discrepancies for test case 074 are significantly higher than for test case 072. Downstream of the
 321 obstacle, test case 074 is generally overestimated, especially at the obstructed side of the pipe.



322

323 Figure 9: Radial gas fraction distribution for test 072 for different Z positions.

324 Another point is that without any disturbance, bubbles accumulate in the center of the pipe.
 325 Consequently, they change their position near the obstacle to the unobstructed side of the pipe until they
 326 reach the wake region. Here, they are drawn into the recirculating flow area. Thus, the bubbles start to
 327 accumulate on the obstructed side of the pipe.



328

329

Figure 10: Radial gas fraction distribution for test 074 for different Z positions.

330

5.2 Bubble dynamics

331

332

333

334

335

336

337

338

339

340

In Figure 11 the average bubble diameter changes along the axial direction for both test cases are presented, showing a slight overestimation of the bubble size upstream of the flow obstacle for test case 072. However, here, simulation results for test case 074 are in better accordance with experimental results. Although simulation results are in satisfying agreement with experimental data for both tests at the obstacle, peaks that are obtained from the experiments could not be captured by the simulations. Also, according to the experiments, the average bubble diameter peaks downstream the obstacle at around $Z = 200$ mm for both test cases. None of the simulations can capture these peaks. In addition, the average bubble size is underestimated for $Z > 30$ mm for test case 074. For test case 072, though average bubble size is underestimated for $70 \text{ mm} < Z < 350$, it gives good agreement between around $350 \text{ mm} < Z < 400$ mm.

341

342

343

344

345

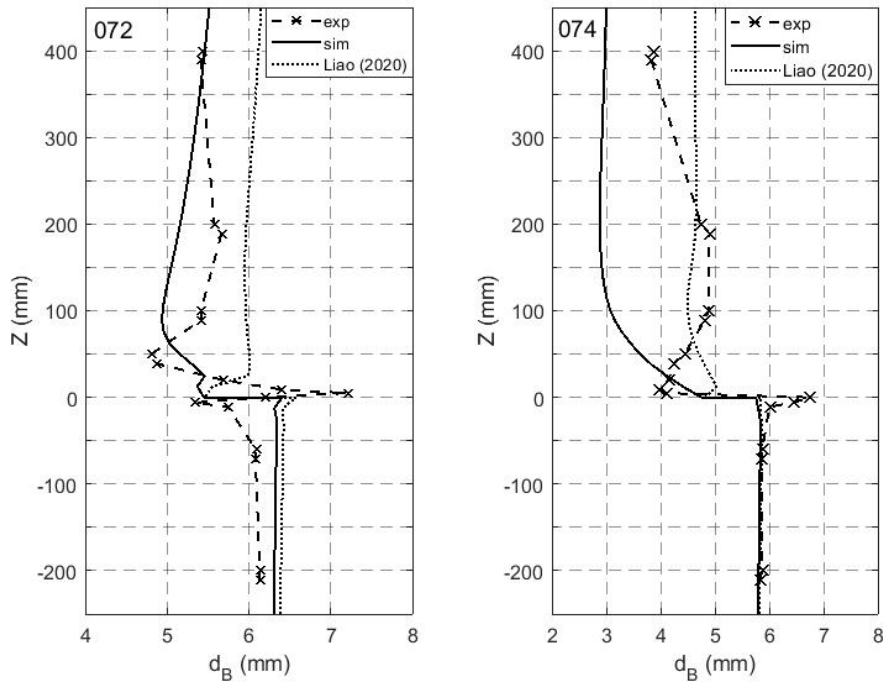
346

347

348

349

Liao (2020) investigated the inconsistencies of the population balance equation in MUSIG, and updated the model by discretizing the source and sink terms that result from bubble coalescence and breakup with an internally consistent scheme, which preserves both the mass and the number of bubbles. The comparison with the updated model is also shown in Figure 11. Whereas the updated MUSIG model provides similar results to the standard MUSIG model upstream the obstacle, it predicts differently at the downstream. For the test case 072, simulation results with the updated model are overestimated compared to experimental results, but the breakup and coalescence tendency behind the obstacle is well captured. For the test case 074, it provides better average bubble diameter prediction than the standard one.



350

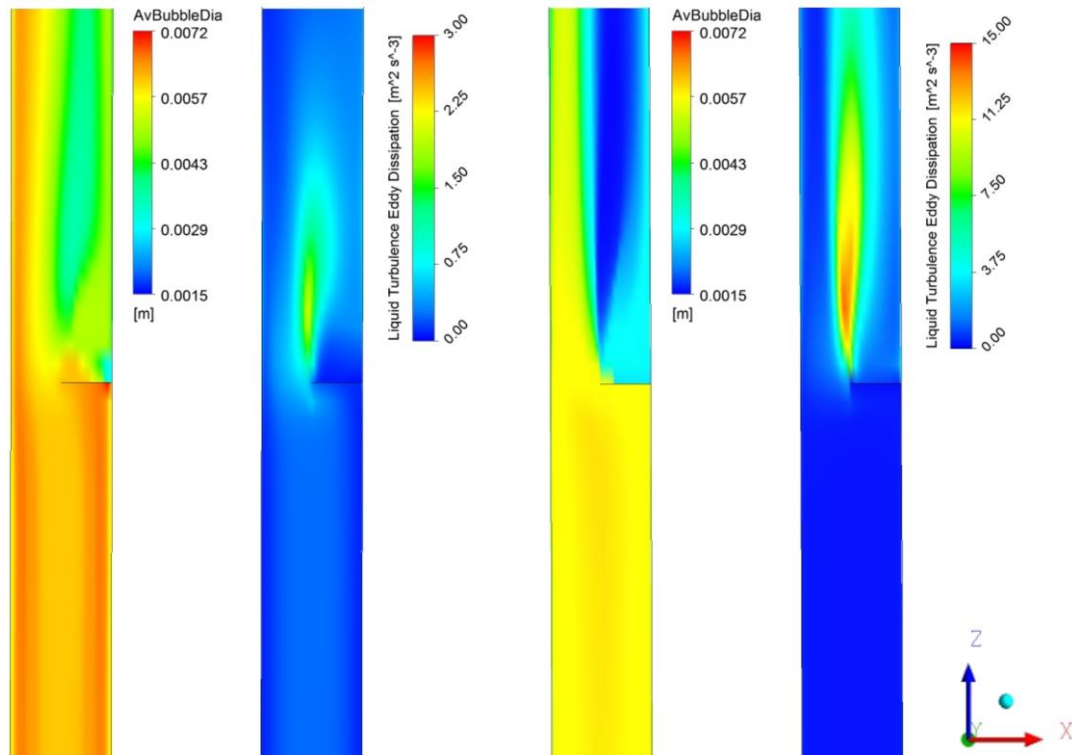
351

Figure 11: Average bubble diameter for test cases 072 and 074.

352

Figure 12 presents the average bubble diameter and turbulent dissipation visualization for $-200 \text{ mm} < Z < 200 \text{ mm}$ and both test cases. Upstream of the obstacle, the average bubble diameter near the pipe wall is larger than in the pipe center for test case 072, while it is vice versa for test case 074. This can be explained by the bubble breakup that is dominant for test case 074 due to higher turbulence as compared to test case 072, where more coalescence events take place in the near wall region. Due to the obstacle, a strong liquid jet with high velocity is established, creating a strong shear flow. This, in turn, leads to higher turbulent dissipation as can be seen from the comparison of both test cases in Figure 12. Thus, higher bubble breakup rates are induced. Downstream of the obstacle larger bubble sizes are found on the unobstructed side of the pipe, whereas smaller bubbles occur on the obstructed side because of bubble breakup and liquid circulation for both test cases. This effect increases for test case 074 due to the higher turbulent dissipation. Thus, bubble breakup is more dominant than bubble coalescence in the downstream wake region of the flow obstacle. The accumulation of small bubbles in the circulation region is more obvious (see Figure 7).

365



366

367
368

Figure 12: Comparison of average bubble diameter and turbulent dissipation for test cases 072 (left) and 074 (right).

369

370

371

372

373

374

375

376

377

378

379

380

381

Figure 13 presents the lift and turbulent dispersion force vectors for $-200 \text{ mm} < Z < 200 \text{ mm}$ and both test cases. For test 072, by the lift force, all the bubbles are directed to the pipe center upstream the obstacle due to a negative lift force coefficient. Unlike the lift force, the turbulent dispersion force directs the bubbles to the pipe walls, where the volume fraction is lower, and the magnitude of the turbulent dispersion force is bigger than the magnitude of the lift force near the pipe walls. As a result, as the lift force directs the bubbles to the pipe center and increases further the void fraction there. At the obstacle downstream, bubbles that are in the jet region directed to the pipe center by the lift force. On the other hand, bubbles that are in the wake region behind the obstacle are directed by the lift force to the pipe wall, because of a smaller average diameter. Besides this, the turbulent dispersion force directs the bubbles, which are in the jet region, to the pipe wall as expected to counterwork the accumulation of bubbles. Yet, on the obstacle side, the turbulent dispersion force directs the bubbles to the pipe center. The reason is that turbulent dispersion force influences from high void fraction to low void fraction since it is related to the void fraction gradient.

382

383

384

385

386

387

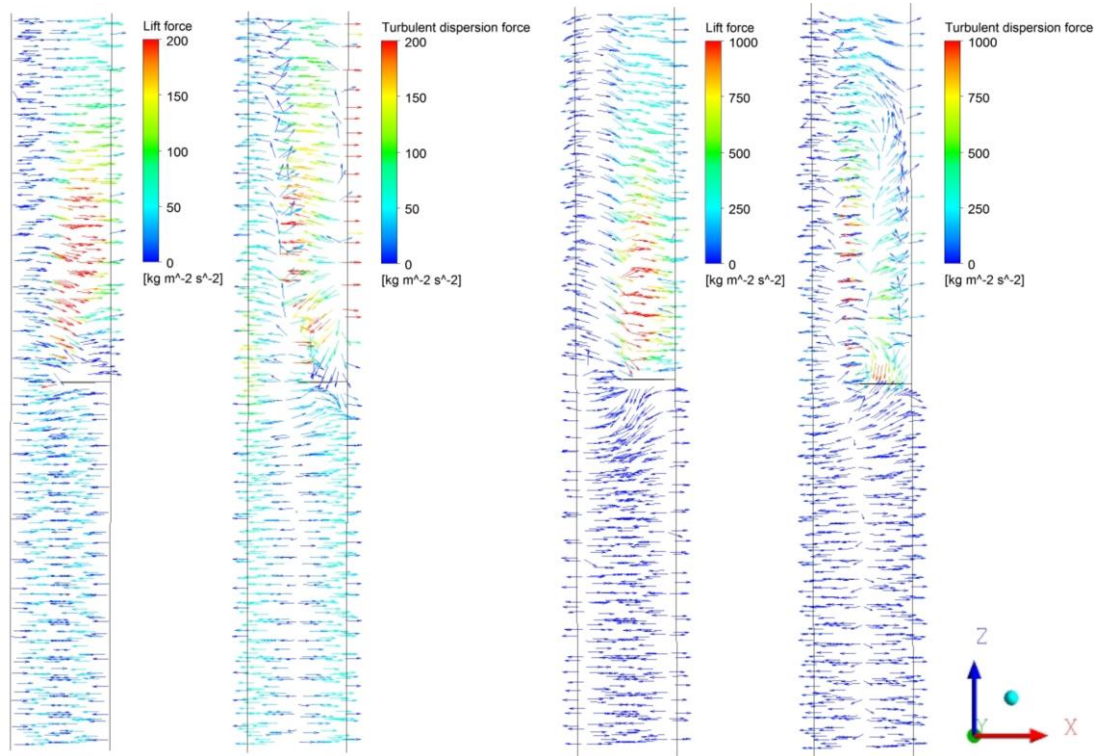
388

389

390

391

For test 074, whereas some bubbles, which are close to the wall, are directed to the pipe wall by the lift force upstream the obstacle, the others move to the pipe center. The turbulent dispersion forces direct all the bubbles to the pipe walls due to a core-peak volume fraction profile as shown in Figure 10. Bubbles which have higher lift force magnitude than dispersion force, move to the pipe center. At the obstacle downstream, for the bubbles that are located undisturbed part of the pipe, the lift force directs them further to pipe wall except for the area that is between the jet region and the region behind the obstacle, where negative velocity gradients prevail. Additionally, whereas turbulent dispersion force directs the bubbles, which are on the left-hand side of the pipe and right-hand side just after the obstacle, to the wall, it changes its direction from pipe wall to pipe center after a certain Z distance on the right-hand side of the pipe, because of high accumulation of bubbles there.



392

393

Figure 13: Bubble lift and turbulent dispersion vectors for test 072 (left) and 074 (right).

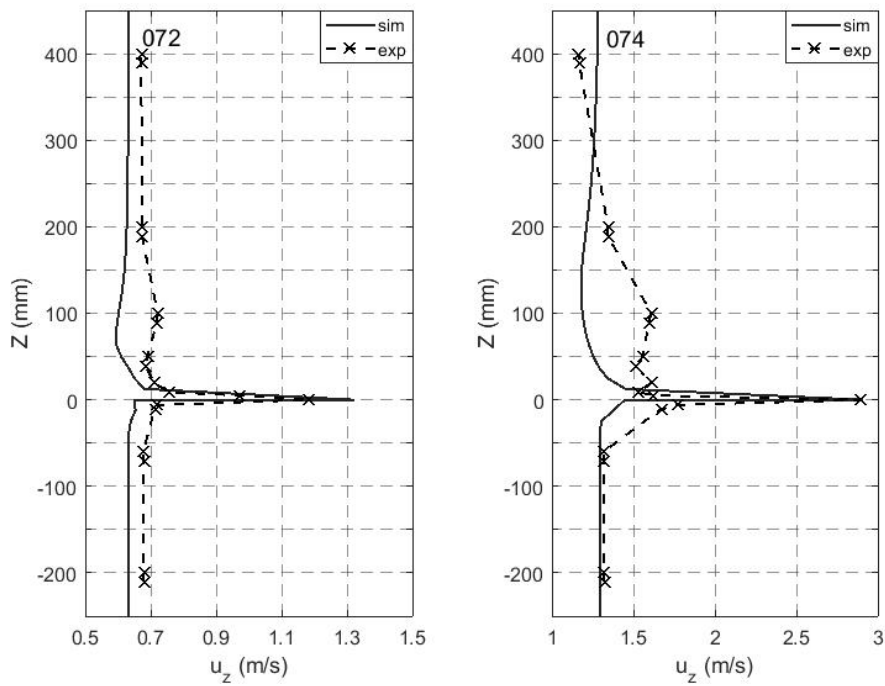
394

The average axial gas velocity is shown in Figure 14. It is clear that the flow experiences strong accelerations due to the obstacle. Both simulation results are generally in good agreement with the experiments. However, there are some inconsistencies according to the experiments like that the simulations could not predict the velocity peaks where are $Z = 100 \text{ mm}$ for both tests.

395

396

397



398

399

Figure 14: Cross-sectional averaged gas velocity for test 072 and 074.

400

5.3 Bubble breakup and coalescence

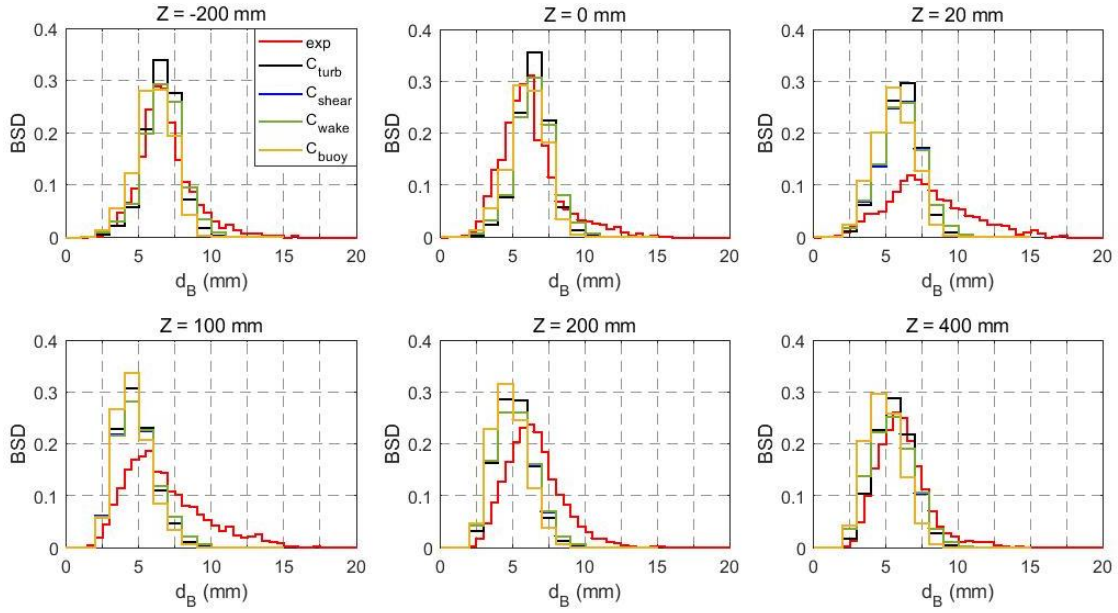
According to Liao *et al.* (2015), there are four mechanisms leading to bubble breakup: laminar shear, turbulent shear, interfacial slip and turbulence fluctuation and likewise five mechanisms leading to coalescence: buoyancy, eddy capture, velocity gradient, wake entrainment and turbulence fluctuation.

The laminar and turbulent shear mechanisms account for viscous shear force in the bulk flow and eddies, respectively. The interfacial slip mechanism considers the impact of interfacial friction. The turbulence fluctuation mechanism describes the effect of turbulent velocity fluctuation on bubble breakup. For coalescence modeling, the buoyancy mechanism accounts for the collision if a faster bubble approaches a slower one. The eddy capture mechanism occurs between bubbles, which are smaller than the Kolmogorov length scale. The velocity gradient mechanism is due to the velocity gradient in the bulk flow. The wake entrainment mechanism acts in the wake region of a bubble where relatively small bubbles can accelerate and catch up with the big one that forms the wake.

The effects of these breakup and coalescence mechanisms were investigated in this study. Thereby, we neglected the turbulent shear mechanism for breakup modelling and the eddy capture mechanism for coalescence modelling as these mechanisms are only important when bubbles are much smaller than the Kolmogorov length scale.

5.3.1 Bubble coalescence mechanisms

Bubble coalescence occurs due to bubble-bubble collision and can be described by different mechanisms as mentioned in Section 5.3. Detailed information can be found in Liao *et al.* (2015). Figure 15 shows the impact of turbulence, velocity gradient, wake entrainment and buoyancy mechanisms on the bubble size distribution at different cross-sections for test case 072. The mechanisms are switched on or off by setting C_{turb} , C_{shear} , C_{wake} and C_{buoy} respectively equal to one or zero. As can be seen from the Figure 15, considering each mechanism separately provide similar results for all Z positions. Here, calculated coalescence rate of all mechanisms might be negligible in comparison to breakup rates, leading to approximately the same predictions. Consequently, the combination of different coalescence mechanisms does not improve the simulation results, as can be seen in Figure 16.

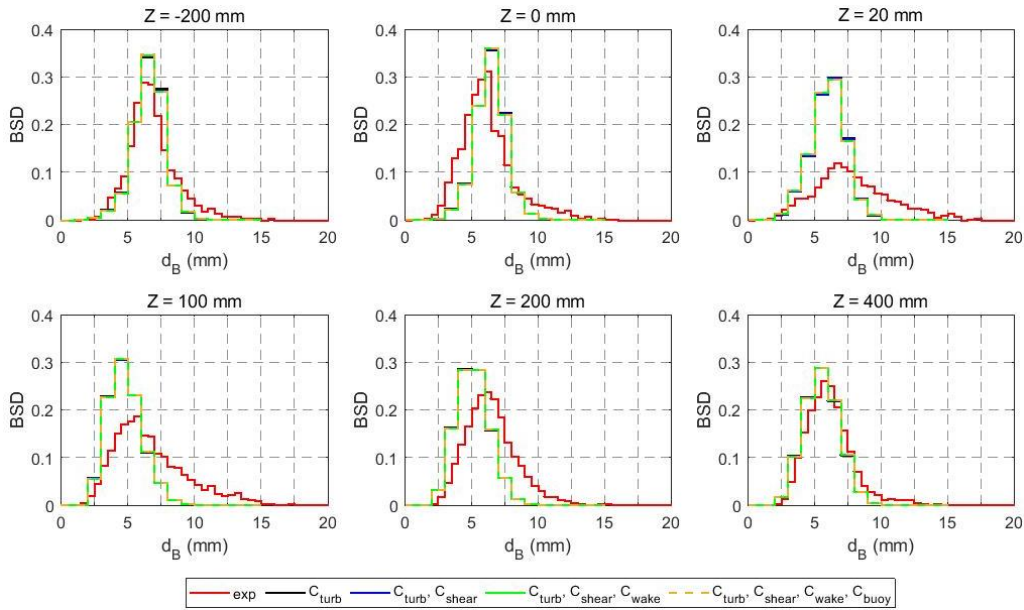


427

428

429

Figure 15: Comparison of bubble coalescence mechanisms for test case 072: C_{turb} , C_{shear} , C_{wake} and C_{buoy} (a legend is given in the first graph).



430

431

432

Figure 16: Comparison of bubble coalescence mechanisms for test case 072: C_{turb} , $C_{turb} + C_{shear}$, $C_{turb} + C_{shear} + C_{wake}$ and $C_{turb} + C_{shear} + C_{wake} + C_{buoy}$ (a legend is given below the graphs).

433

5.3.2 Bubble breakup mechanisms

434

435

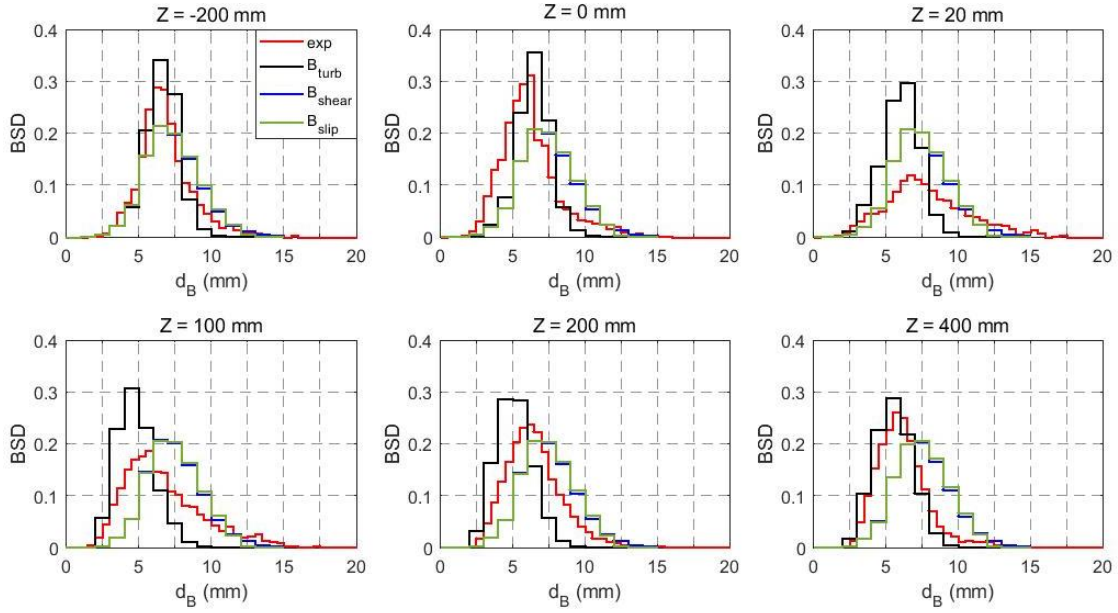
436

437

438

439

Bubble breakup takes place due to flow stresses that act on the bubble surface. As mentioned in Section 5.3, there are different mechanisms causing bubble breakup. Figure 17 presents the impacts of turbulent fluctuation (B_{turb}), laminar shear (B_{shear}) and interfacial slip (B_{slip}) on the bubble size distributions for the varied Z positions. B_{shear} and B_{slip} give almost similar results for every Z positions. Also, as can be seen from Figure 18, laminar shear and interfacial slip play a negligible effect in the breakup, since it is mainly caused by turbulence.

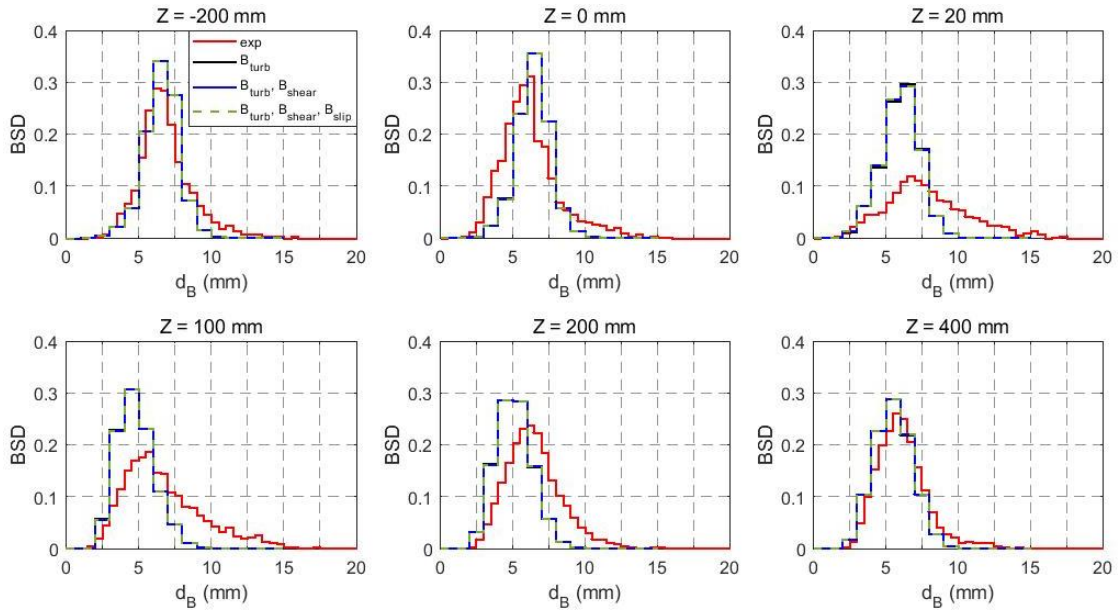


440

441

442

Figure 17: Comparison of bubble breakup mechanisms for test 072: B_{turb} , B_{shear} and B_{slip} (a legend is given in the first graph).



443

444

445

Figure 18: Comparison of bubble breakup mechanisms for test 072: B_{turb} , $B_{\text{turb}} + B_{\text{shear}}$ and $B_{\text{turb}} + B_{\text{shear}} + B_{\text{slip}}$ (a legend is given in the first graph).

446

447

448

As can be seen from Figure 15 to 18, other mechanisms than turbulence fluctuation have negligible effects. Thus, we considered only the turbulence fluctuation mechanism to model the breakup and coalescence in this study.

449

450

451

452

453

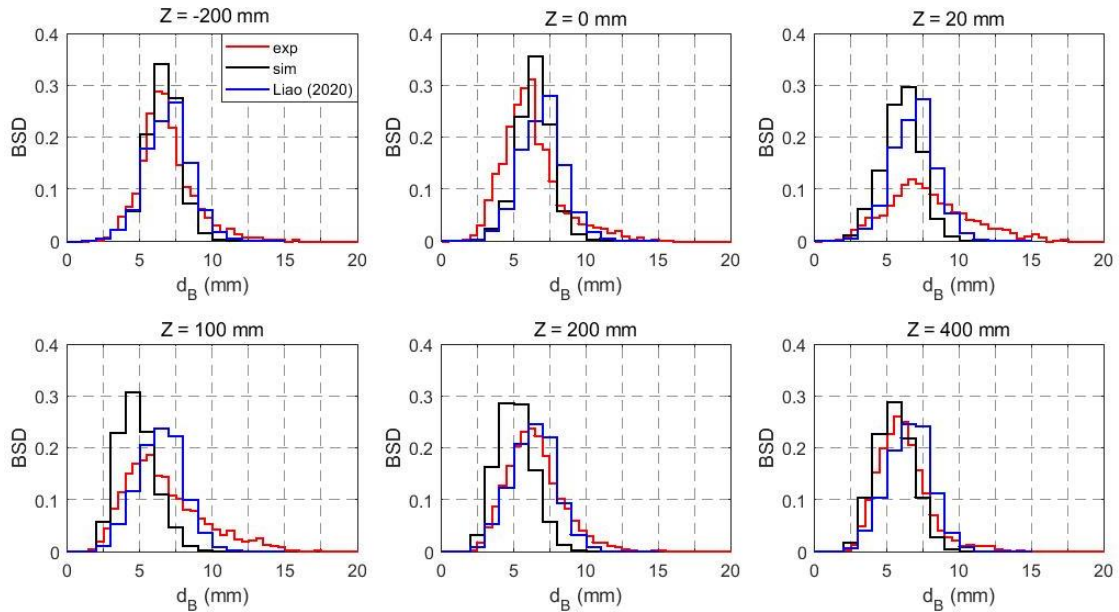
454

455

456

Figure 19 and Figure 20 present the bubble size distribution at different Z positions for both test cases. The bubble size distribution (BSD) is determined as gas volume fraction of each bubble size class divided by the class width and total gas void fraction. For the standard MUSIG model, there is a good agreement of simulation and experiment at position $Z = -200$ mm for test case 072. Although the simulation gives satisfying results for the larger bubbles ($d_B > 7.5$ mm) at position $Z = 0$ mm, it is underestimated for smaller ones ($d_B < 6.5$ mm). Here, slight breakup and coalescence tendency as compared to position $Z = -200$ mm is observed in the experiment, but the model captures none of them. At positions $Z = 20$ mm, $Z = 100$ mm and $Z = 200$ mm, the gas fraction of the bubbles in the size class

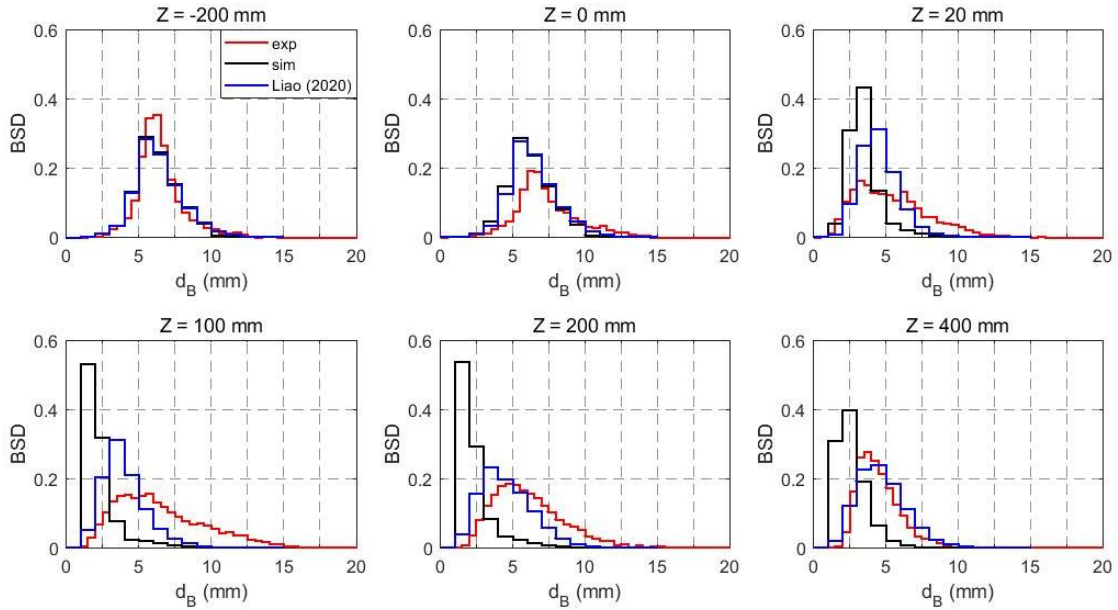
457 below 2.5 mm are well predicted. However, the simulation overestimates the gas fraction of bubbles
 458 around $2.5 \text{ mm} < d_B < 7.5 \text{ mm}$ for position $Z = 20 \text{ mm}$ and around $2.5 \text{ mm} < d_B < 6.5 \text{ mm}$ for positions
 459 $Z = 100 \text{ mm}$ and $Z = 200 \text{ mm}$, but underestimates in the remaining size classes for all three positions.
 460 Behind the obstacle, more coalescence events take place leading to a coalescence-dominant trend in the
 461 experiments, which is obviously under-predicted by the model. From position $Z = 200 \text{ mm}$ to
 462 $Z = 400 \text{ mm}$ bubble breakup is slightly overweighted, while the model predicts almost no change.
 463 However, similar to position $Z = -200 \text{ mm}$, good agreement is captured at position $Z = 400 \text{ mm}$ again.
 464 The updated MUSIG model predicts a slightly larger coalescence to breakup ratio, which leads to a
 465 larger mean bubble size as shown in Figure 11.



466

467 Figure 19: Bubble size distribution at different sections for test case 072 (a legend is given in the first graph).

468 For the standard MUSIG model, test case 074 shows similar tendencies as compared to test case 072.
 469 Good agreement is obtained at position $Z = -200 \text{ mm}$. However, a slight coalescence trend from position
 470 $Z = -200 \text{ mm}$ to $Z = 0 \text{ mm}$ is observed in the experiment, while in the simulation coalescence and
 471 breakup are almost in balance. In addition, from position $Z = 0 \text{ mm}$ to $Z = 20 \text{ mm}$ breakup seems to be
 472 dominant, which may be captured in the simulation, but the breakup rate is highly overestimated. One
 473 of the reasons may be the overprediction of the turbulence dissipation rate. Further, slight coalescence
 474 is observed again in the experiment from position $Z = 0 \text{ mm}$ to $Z = 20 \text{ mm}$ that cannot be captured by
 475 the simulation. Unlike for test case 072, the obvious inconsistency between experiment and simulation
 476 continues to exist also at position $Z = 400 \text{ mm}$.



477

478 Figure 20: Bubble size distribution at different sections for test case 074 (a legend is given in the first graph).

479 On the one hand, the updated MUSIG model gives similar results to standard one at the positions
 480 $Z = -200$ mm and $Z = 0$ mm, where the coalescence and breakup is nearly in equilibrium. On the other
 481 hand, it improves the downstream results significantly, and captures the coalescence trend well. The
 482 improvement increases from position $Z = 20$ mm to $Z = 400$ mm. However, there are still notable
 483 inconsistencies between experiments and updated MUSIG predictions at the positions $Z = 20$ mm and
 484 $Z = 100$ mm where the flow is highly complex.

485

486 6. Conclusion

487 We investigated the performance of state-of-the-art bubble breakup and coalescence modelling for 3D
 488 bubbly flow around an obstacle in a pipe. For this purpose, we performed simulations with the
 489 homogeneous MUSIG model for two different liquid velocities and compared gas velocity, void
 490 fraction, mean bubble diameter and bubble size distribution with 3D UFXCT data. A main feature of
 491 the flow is that there is a vortex region behind the obstacle. This vortex region causes a strong gas
 492 accumulation downstream the obstacle. While in the high-velocity case void accumulates directly
 493 behind the obstacle it does so further downstream in the low-velocity case. The reason is that the obstacle
 494 leads to a downstream wake region whose length increases with velocity. Thus, this wake region causes
 495 void accumulation depending on the area where it acts.

496 While the prediction of axial and radial void fractions for the low-velocity case agrees generally well
 497 with the experimental results for both the upstream and downstream region of the obstacle, the prediction
 498 for the high-velocity case overestimates the average void fraction compared to experimental data
 499 downstream of the obstacle. Moreover, downstream of the obstacle the average bubble diameter and
 500 average gas velocity are underestimated while they are in good agreement for both the low-velocity and
 501 high-velocity cases upstream of the obstacle.

502 Considering the BSD, we found that coalescence is the dominant mechanism behind the obstacle for the
 503 low-velocity case in the experiments. However, the simulations could not capture it. In addition, for the
 504 high-velocity case the breakup rate is highly overestimated compared to the experimental data. The
 505 difference between experimental data and numerical results may be because of the overestimation of
 506 turbulence, which is related to the shear-induced and bubble-induced turbulence models. Simulation

507 results show that the predicted coalescence rate of all mechanisms might be negligible in comparison to
508 breakup rates since all the coalescence mechanisms provide approximately the same predictions. In
509 addition, it is clear that turbulence is the dominant mechanism for both breakup and coalescence.

510 The performance of the updated MUSIG model recently developed by Liao (2020) was also tested in
511 the present study. The results show that the updated MUSIG model predicts an obviously lower breakup
512 rate. Consequently, it is able to reproduce the coalescence-dominant trend for the low-velocity test, and
513 predicts a reasonable mean bubble size for the high-velocity test, where turbulence-induced breakup is
514 significant. Investigation on the effect of two-phase turbulence models and improving the coalescence
515 and breakup model in the updated MUSIG model will be topics of future work.

516 As already mentioned, the low-velocity case shows good agreement with the experimental data
517 downstream of the obstacle, whereas this is not the case for the high-velocity case. The explanation of
518 void fraction overestimation for the high-velocity case may be due to more than one reason. Firstly,
519 applied bubble force models, which are widely used in the literature, do not consider the turbulence
520 effects. Turbulence effects are thought to have an impact, especially on drag force. Secondly, the bubble
521 breakup and coalescence model may need to be improved. Thirdly, the applicability of bubble-induced
522 turbulence that was used for the simulations is still not clear for high shear-induced turbulence cases,
523 since it has been obtained under low turbulence conditions.

524 **References**

- 525 Auton, T. R., Hunt, J. C. R., & Prudhomme, M. (1988). The Force Exerted on a Body in Inviscid
526 Unsteady Non-Uniform Rotational Flow. *Journal of Fluid Mechanics*, 197, 241-257.
527 doi:10.1017/S0022112088003246
- 528 Besagni, G., Guedon, G. R., & Inzoli, F. (2018). Computational fluid-dynamic modeling of the mono-
529 dispersed homogeneous flow regime in bubble columns. *Nuclear Engineering and Design*,
530 331, 222-237. doi:10.1016/j.nucengdes.2018.03.003
- 531 Burns, A. D., Frank, T., Hamill, I., & Shi, J.-M. (2004). *The Favre averaged drag model for turbulent*
532 *dispersion in Eulerian multi-phase flows*. Paper presented at the 5th International Conference
533 on Multiphase Flow, ICMF, Yokohama, Japan.
- 534 Colombo, M., & Fairweather, M. (2015). Multiphase turbulence in bubbly flows: RANS simulations.
535 *International Journal of Multiphase Flow*, 77, 222-243.
536 doi:10.1016/j.ijmultiphaseflow.2015.09.003
- 537 Colombo, M., & Fairweather, M. (2019). Influence of multiphase turbulence modelling on interfacial
538 momentum transfer in two-fluid Eulerian-Eulerian CFD models of bubbly flows. *Chemical*
539 *Engineering Science*, 195, 968-984. doi:10.1016/j.ces.2018.10.043
- 540 Frank, T., Zwart, P. J., Krepper, E., Prasser, H. M., & Lucas, D. (2008). Validation of CFD models for
541 mono- and polydisperse air-water two-phase flows in pipes. *Nuclear Engineering and Design*,
542 238(3), 647-659. doi:10.1016/j.nucengdes.2007.02.056
- 543 Hosokawa, S., Tomiyama, A., Misaki, S., & Hamada, T. (2002). *Lateral migration of single bubbles*
544 *due to the presence of wall*. Paper presented at the Proceedings of the ASME Joint U.S.-
545 European Fluids Engineering Division Conference, FEDSM2002, Montreal, Canada.
- 546 Ishii, M., & Zuber, N. (1979). Drag Coefficient and Relative Velocity in Bubbly, Droplet or
547 Particulate Flows. *Aiche Journal*, 25(5), 843-855. doi:10.1002/aic.690250513
- 548 Jareteg, K., Strom, H., Sasic, S., & Demaziere, C. (2017). On the dynamics of instabilities in two-fluid
549 models for bubbly flows. *Chemical Engineering Science*, 170, 184-194.
550 doi:10.1016/j.ces.2017.03.063
- 551 Jin, D., Xiong, J. B., & Cheng, X. (2019). Investigation on interphase force modeling for vertical and
552 inclined upward adiabatic bubbly flow. *Nuclear Engineering and Design*, 350, 43-57.
553 doi:10.1016/j.nucengdes.2019.05.005

554 Krepper, E., Beyer, M., Frank, T., Lucas, D., & Prasser, H. M. (2009). CFD modelling of
555 polydispersed bubbly two-phase flow around an obstacle. *Nuclear Engineering and Design*,
556 239(11), 2372-2381. doi:10.1016/j.nucengdes.2009.06.015

557 Liao, Y. X. (2020). Update to the MUSIG model in ANSYS CFX for reliable modelling of bubble
558 coalescence and breakup. *Applied Mathematical Modelling*, 81, 506-521.
559 doi:10.1016/j.apm.2020.01.033

560 Liao, Y. X., Lucas, D., Krepper, E., & Schmidtke, M. (2011). Development of a generalized
561 coalescence and breakup closure for the inhomogeneous MUSIG model. *Nuclear Engineering
562 and Design*, 241(4), 1024-1033. doi:10.1016/j.nucengdes.2010.04.025

563 Liao, Y. X., Ma, T., Liu, L., Ziegenhein, T., Krepper, E., & Lucas, D. (2018). Eulerian modelling of
564 turbulent bubbly flow based on a baseline closure concept. *Nuclear Engineering and Design*,
565 337, 450-459. doi:10.1016/j.nucengdes.2018.07.021

566 Liao, Y. X., Rzehak, R., Lucas, D., & Krepper, E. (2015). Baseline closure model for dispersed bubbly
567 flow: Bubble coalescence and breakup. *Chemical Engineering Science*, 122, 336-349.
568 doi:10.1016/j.ces.2014.09.042

569 Lo, S. (1996). *Application of the MUSIG model to bubbly flows*. Paper presented at the AEAT-1096,
570 AEA Technology.

571 Luo, H., & Svendsen, H. F. (1996). Theoretical model for drop and bubble breakup in turbulent
572 dispersions. *Aiche Journal*, 42(5), 1225-1233. doi:10.1002/aic.690420505

573 Ma, T., Santarelli, C., Ziegenhein, T., Lucas, D., & Frohlich, J. (2017). Direct numerical simulation-
574 based Reynolds-averaged closure for bubble-induced turbulence. *Physical Review Fluids*,
575 2(3). doi:10.1103/PhysRevFluids.2.034301

576 Menter, F. R. (1994). 2-Equation Eddy-Viscosity Turbulence Models for Engineering Applications.
577 *Aiaa Journal*, 32(8), 1598-1605. doi:10.2514/3.12149

578 Neumann-Kipping, M., Bieberle, A., & Hampel, U. (2020). Investigations on bubbly two-phase flow
579 in a constricted vertical pipe. *International Journal of Multiphase Flow*, 130.
580 doi:10.1016/j.ijmultiphaseflow.2020.103340

581 Parekh, J., & Rzehak, R. (2018). Euler-Euler multiphase CFD-simulation with full Reynolds stress
582 model and anisotropic bubble-induced turbulence. *International Journal of Multiphase Flow*,
583 99, 231-245. doi:10.1016/j.ijmultiphaseflow.2017.10.012

584 Prasser, H. M., Beyer, M., Frank, T., Al Issaa, S., Carl, H., Pietruske, H., & Schutz, P. (2008). Gas-
585 liquid flow around an obstacle in a vertical pipe. *Nuclear Engineering and Design*, 238(7),
586 1802-1819. doi:10.1016/j.nucengdes.2007.11.007

587 Prince, M. J., & Blanch, H. W. (1990). Bubble Coalescence and Break-up in Air-Sparged Bubble-
588 Columns. *Aiche Journal*, 36(10), 1485-1499. doi:10.1002/aic.690361004

589 Rzehak, R., & Krepper, E. (2013). CFD modeling of bubble-induced turbulence. *International Journal
590 of Multiphase Flow*, 55, 138-155. doi:10.1016/j.ijmultiphaseflow.2013.04.007

591 Rzehak, R., Krepper, E., & Lifante, C. (2012). Comparative study of wall-force models for the
592 simulation of bubbly flows. *Nuclear Engineering and Design*, 253, 41-49.
593 doi:10.1016/j.nucengdes.2012.07.009

594 Rzehak, R., Ziegenhein, T., Kriebitzsch, S., Krepper, E., & Lucas, D. (2017). Unified modeling of
595 bubbly flows in pipes, bubble columns, and airlift columns. *Chemical Engineering Science*,
596 157, 147-158. doi:10.1016/j.ces.2016.04.056

597 Simonin, O., & Viollet, P. L. (1990). *Predictions of an oxygen droplet pulverization in a compressible
598 subsonic coflowing hydrogen flow*. Paper presented at the Numerical Methods for Multiphase
599 Flows, FED91.

600 Tas-Koehler, S., Lecrivain, G., Krepper, E., Unger, S., & Hampel, U. (2020). Numerical investigation
601 on the effect of transversal fluid field deformation on heat transfer in a rod bundle with mixing
602 vanes. *Nuclear Engineering and Design*, 361. doi:10.1016/j.nucengdes.2020.110575

603 Tomiyama, A., Kataoka, I., Zun, I., & Sakaguchi, T. (1998). Drag coefficients of single bubbles under
604 normal and micro gravity conditions. *Jsm International Journal Series B-Fluids and Thermal
605 Engineering*, 41(2), 472-479. doi:10.1299/jsmeb.41.472

606 Tomiyama, A., Tamai, H., Zun, I., & Hosokawa, S. (2002). Transverse migration of single bubbles in
607 simple shear flows. *Chemical Engineering Science*, 57(11), 1849-1858. doi:10.1016/S0009-
608 2509(02)00085-4

- 609 Troshko, A. A., & Hassan, Y. A. (2001). A two-equation turbulence model of turbulent bubbly flows.
610 *International Journal of Multiphase Flow*, 27(11), 1965-2000. doi:10.1016/S0301-
611 9322(01)00043-X
612 Yeoh, G. H., & Tu, J. Y. (2009). *Computational Techniques for Multiphase Flows*: Butterworth-
613 Heinemann.
- 614

FIGURES

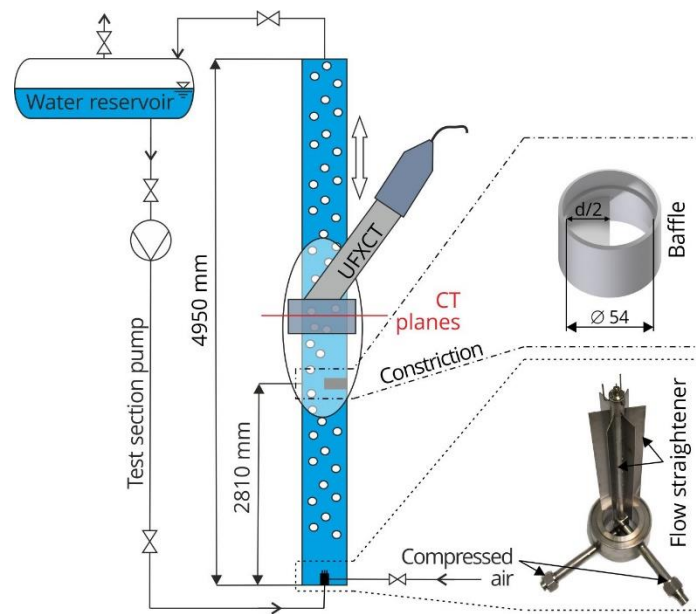


Figure 1: Schematic representations of the vertical test section (left) with details of the gas injection module (bottom right) and the flow obstacle for generation of three-dimensional flow fields (top right).

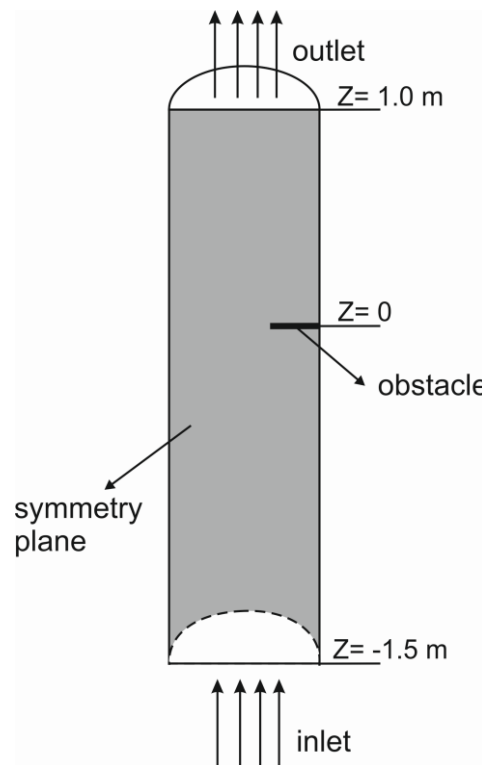


Figure 2: Schematic view of computational domain.

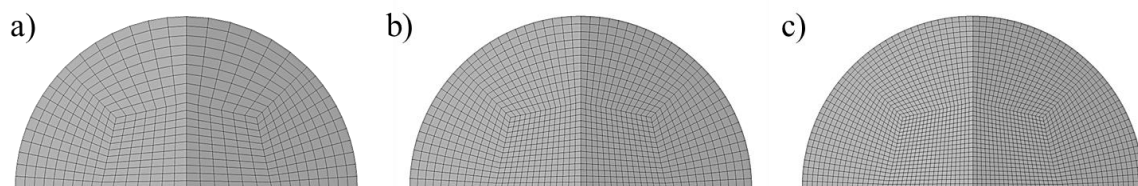


Figure 3: Mesh views: a) mesh 1 b) mesh 2 and c) mesh 3.

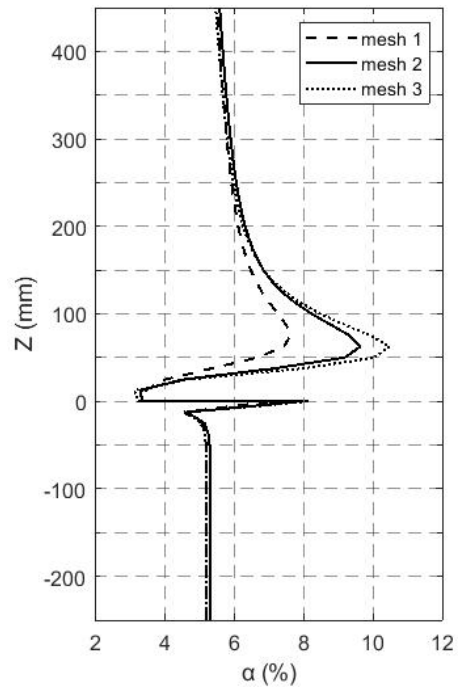


Figure 4: Effect of grid refinement on the average gas fraction for test 072.

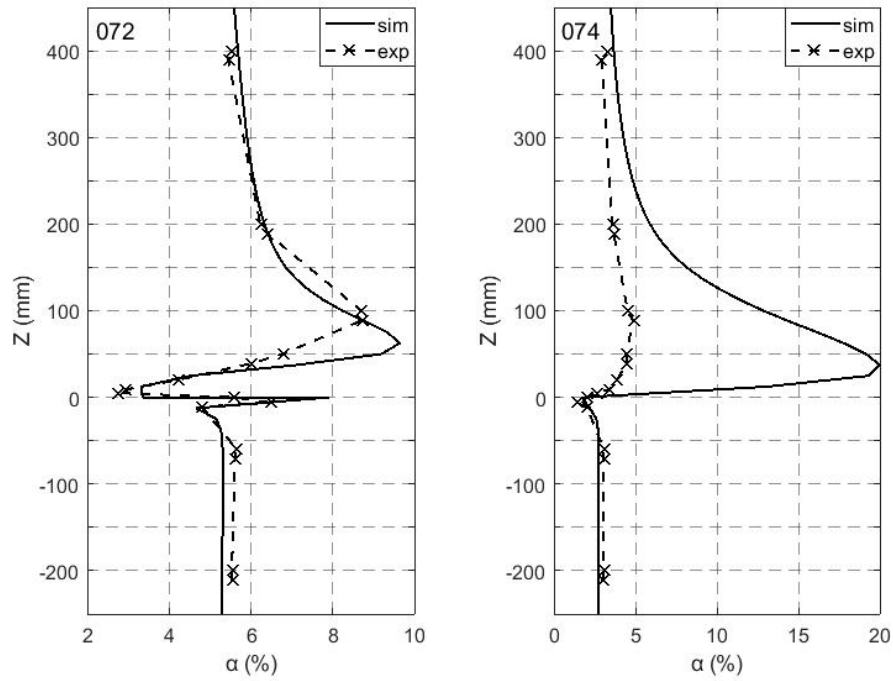


Figure 5: Cross-sectional averaged void fraction along the axial direction for test cases 072 and 074.

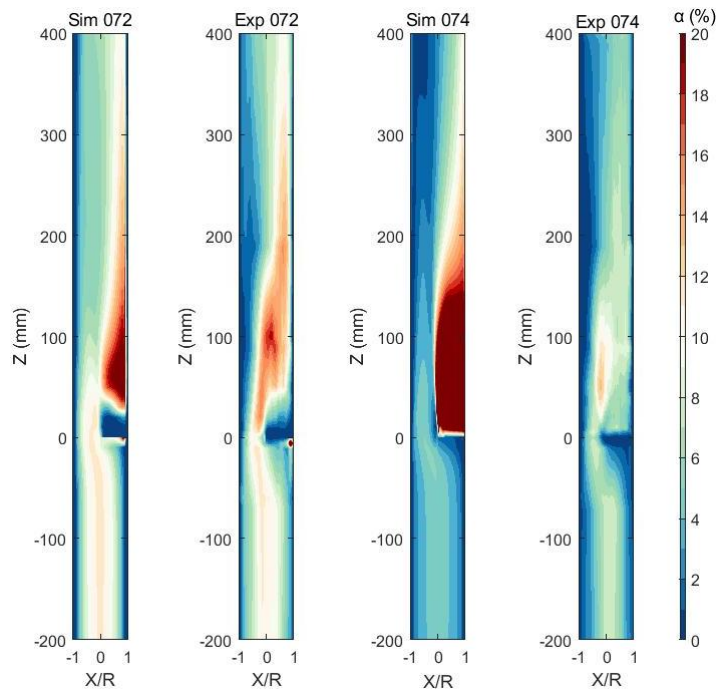


Figure 6: Visualization plots of simulated and measured void fraction for test cases 072 and 074.

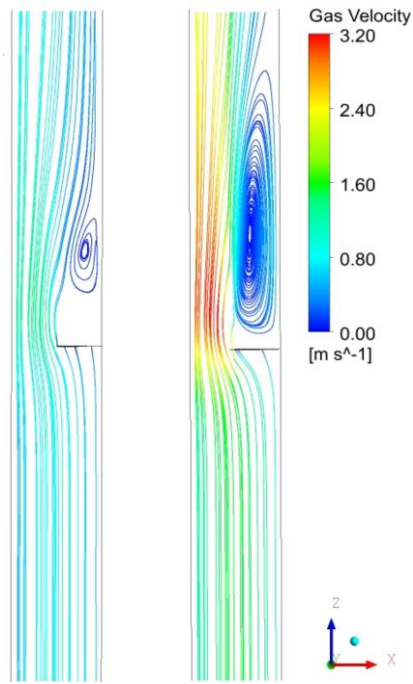


Figure 7: Streamline for test 072 (left) and 074 (right).

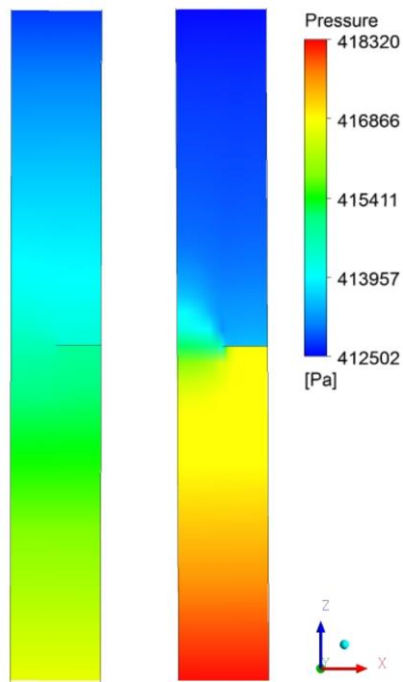


Figure 8: Pressure distribution for test 072 (left) and 074 (right).

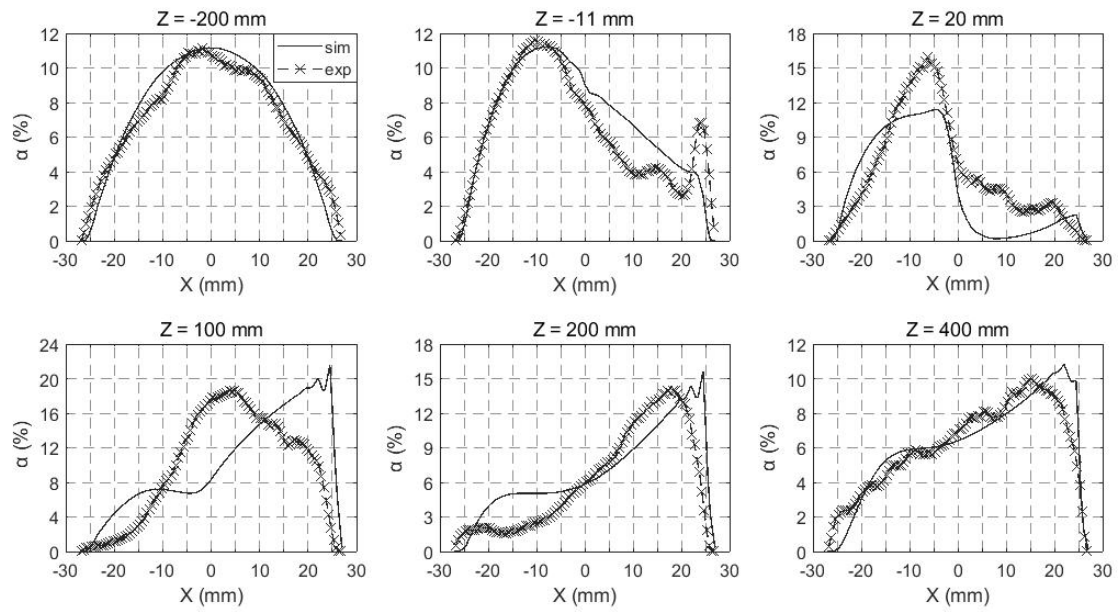


Figure 9: Radial gas fraction distribution for test 072 for different Z positions.

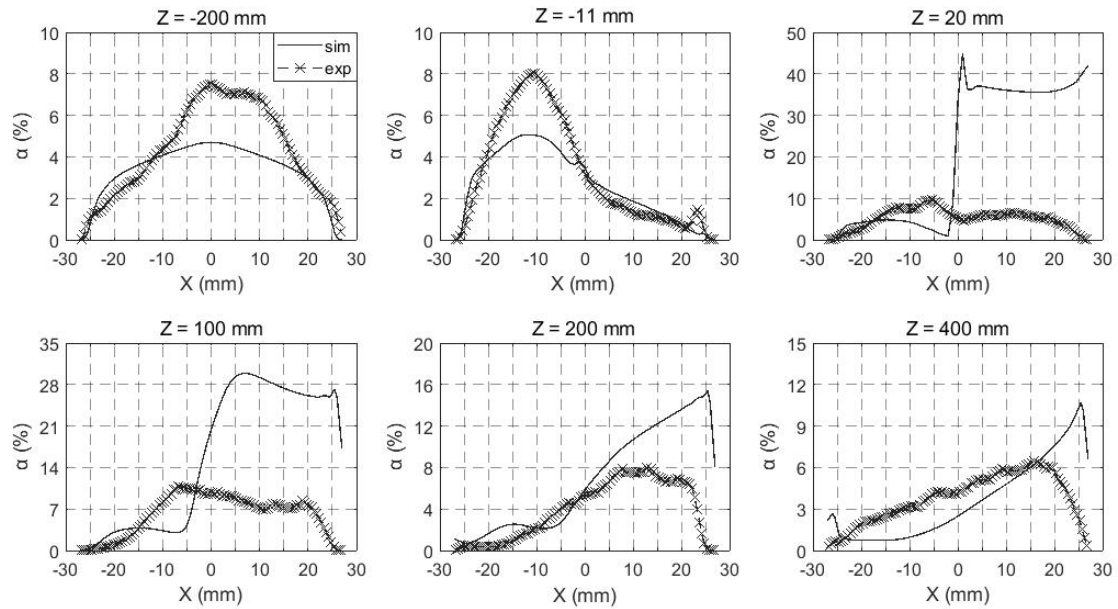


Figure 10: Radial gas fraction distribution for test 074 for different Z positions.

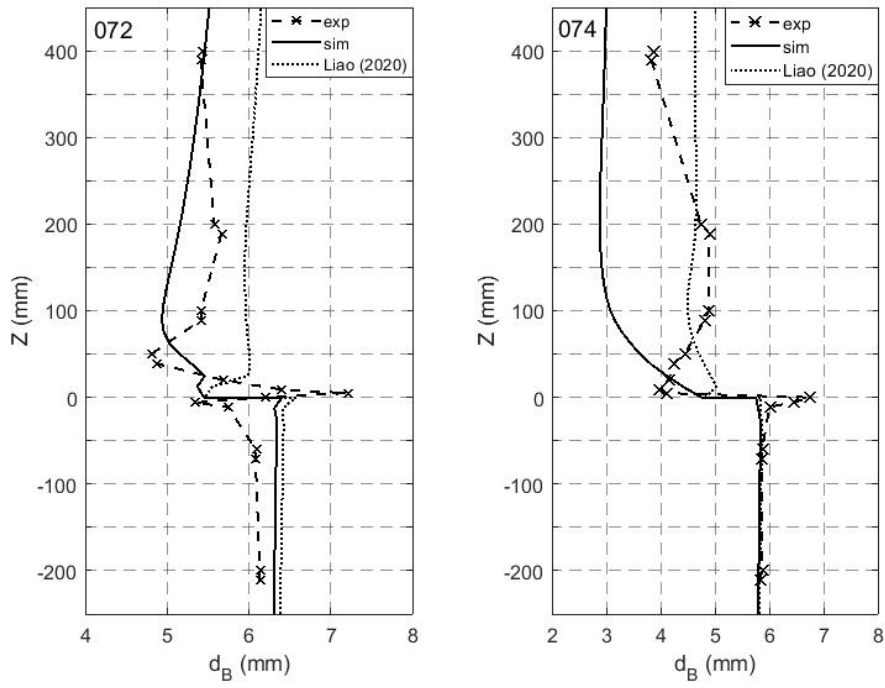


Figure 11: Average bubble diameter for test cases 072 and 074.

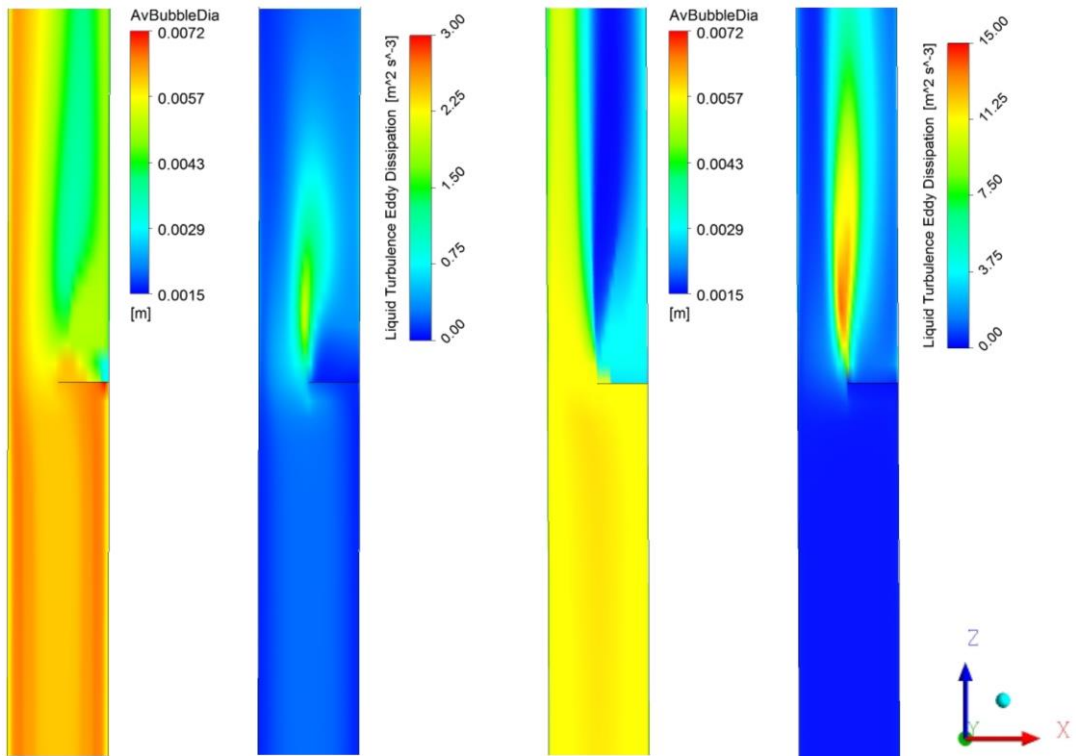


Figure 12: Comparison of average bubble diameter and turbulent dissipation for test cases 072 (left) and 074 (right).

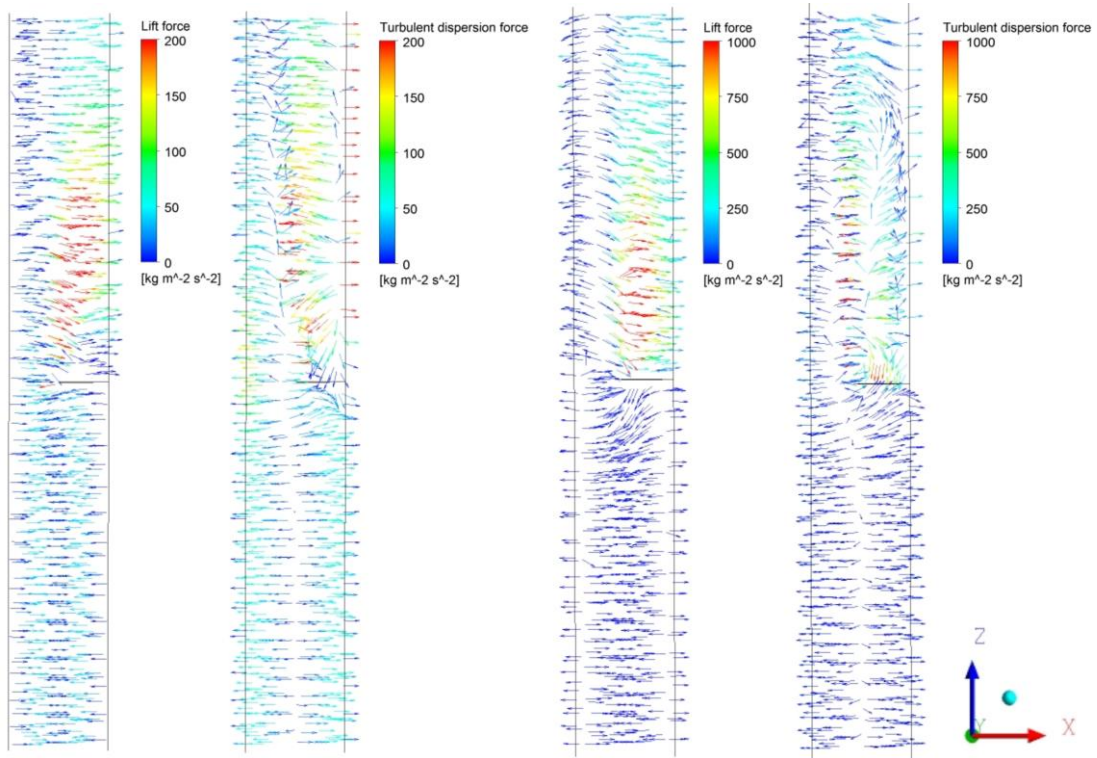


Figure 13: Bubble lift and turbulent dispersion vectors for test 072 (left) and 074 (right).

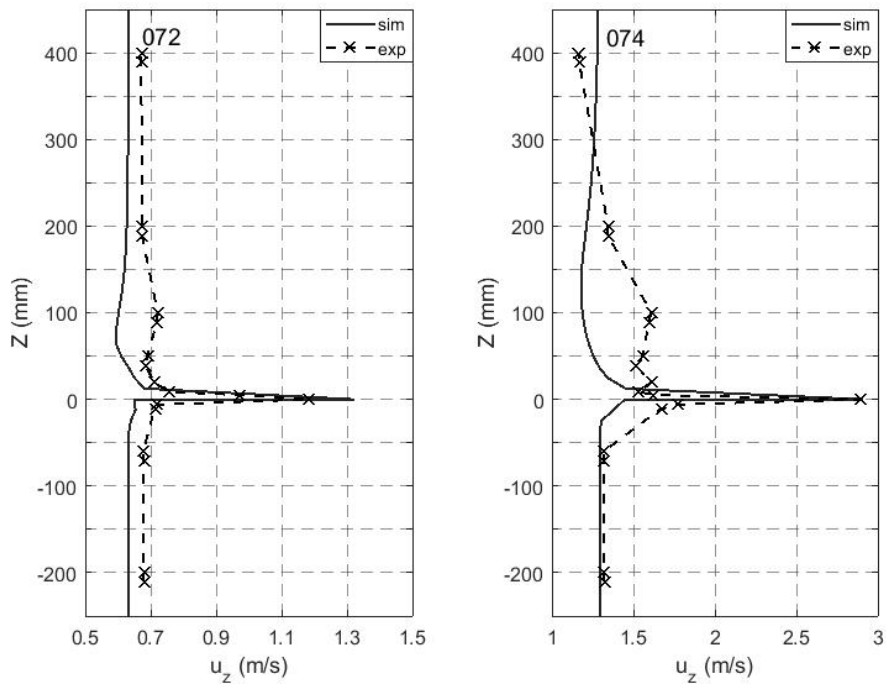


Figure 14: Cross-sectional averaged gas velocity for test 072 and 074.

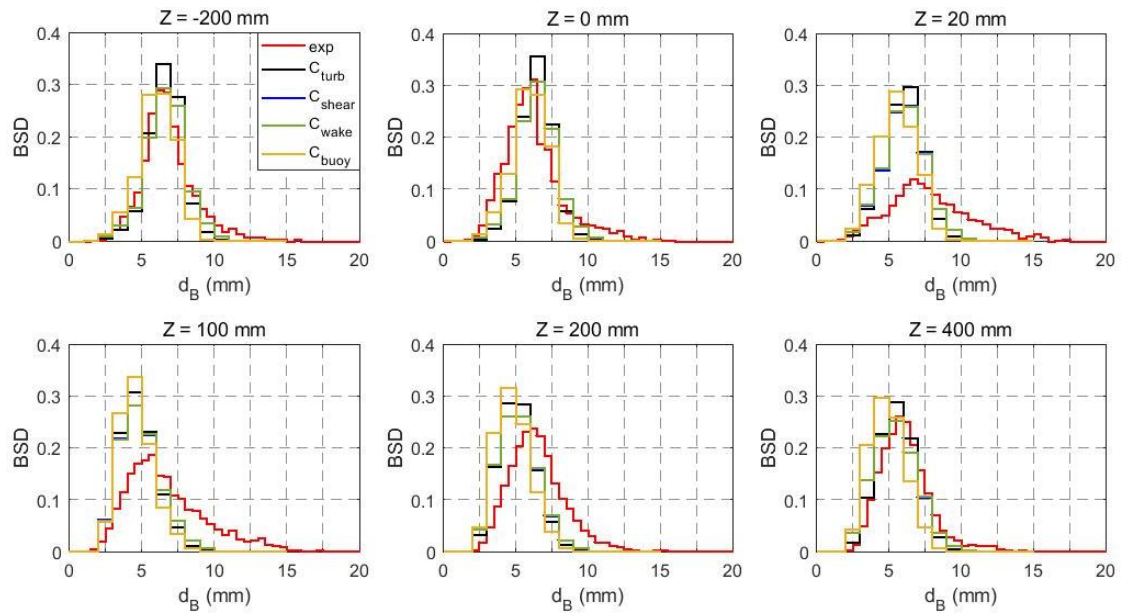


Figure 15: Comparison of bubble coalescence mechanisms for test case 072: C_{turb} , C_{shear} , C_{wake} and C_{buoy} (a legend is given in the first graph).

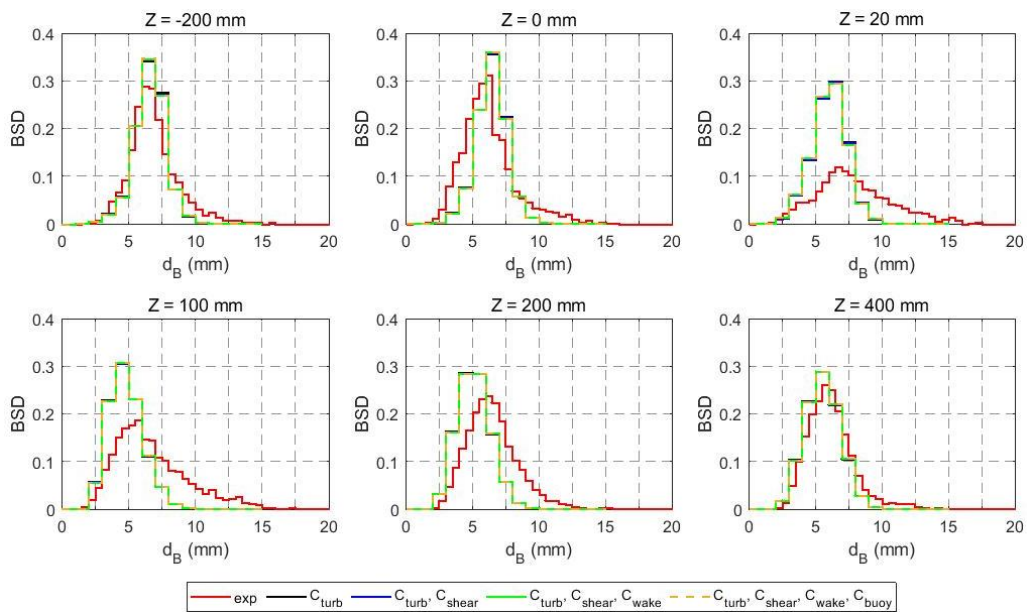


Figure 16: Comparison of bubble coalescence mechanisms for test case 072: C_{turb} , $C_{\text{turb}} + C_{\text{shear}}$, $C_{\text{turb}} + C_{\text{shear}} + C_{\text{wake}}$ and $C_{\text{turb}} + C_{\text{shear}} + C_{\text{wake}} + C_{\text{buoy}}$ (a legend is given below the graphs).

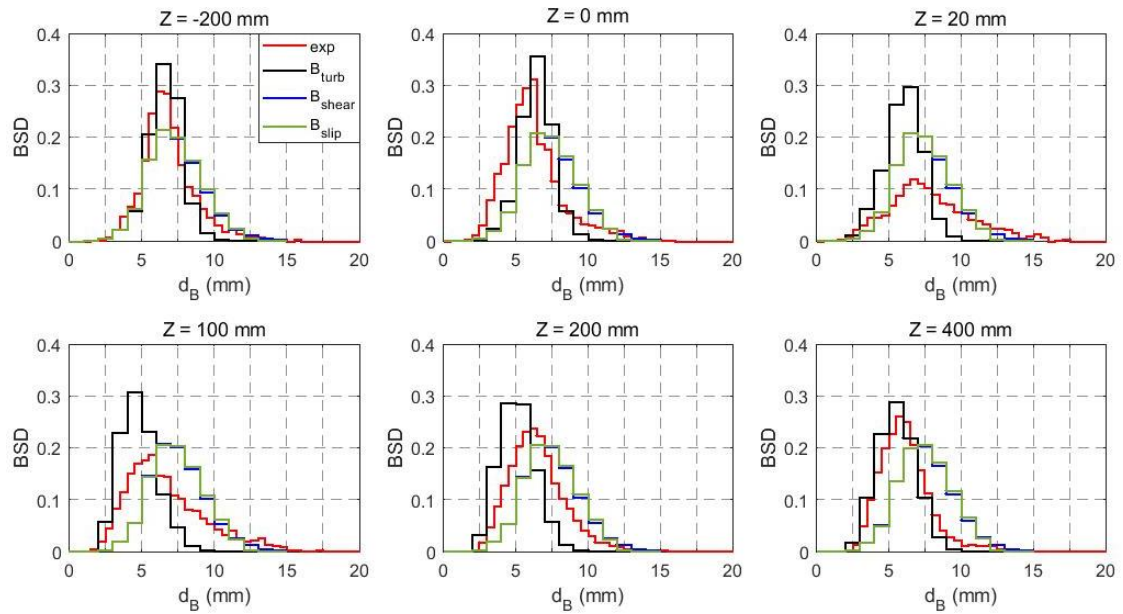


Figure 17: Comparison of bubble breakup mechanisms for test 072: B_{turb} , B_{shear} and B_{slip} (a legend is given in the first graph).

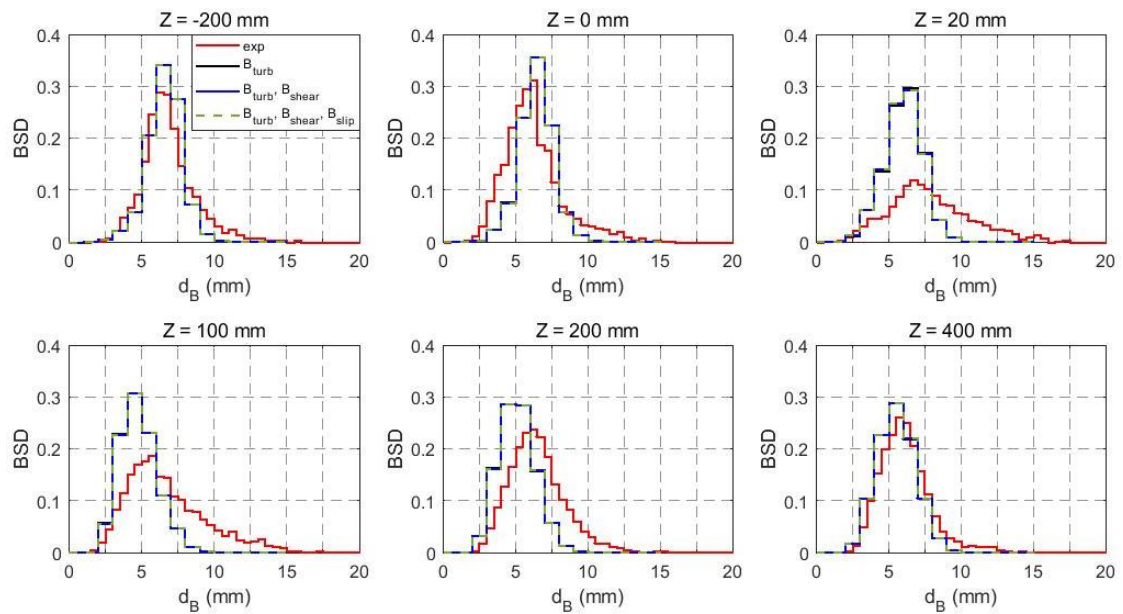


Figure 18: Comparison of bubble breakup mechanisms for test 072: B_{turb} , $B_{\text{turb}} + B_{\text{shear}}$ and $B_{\text{turb}} + B_{\text{shear}} + B_{\text{slip}}$ (a legend is given in the first graph).

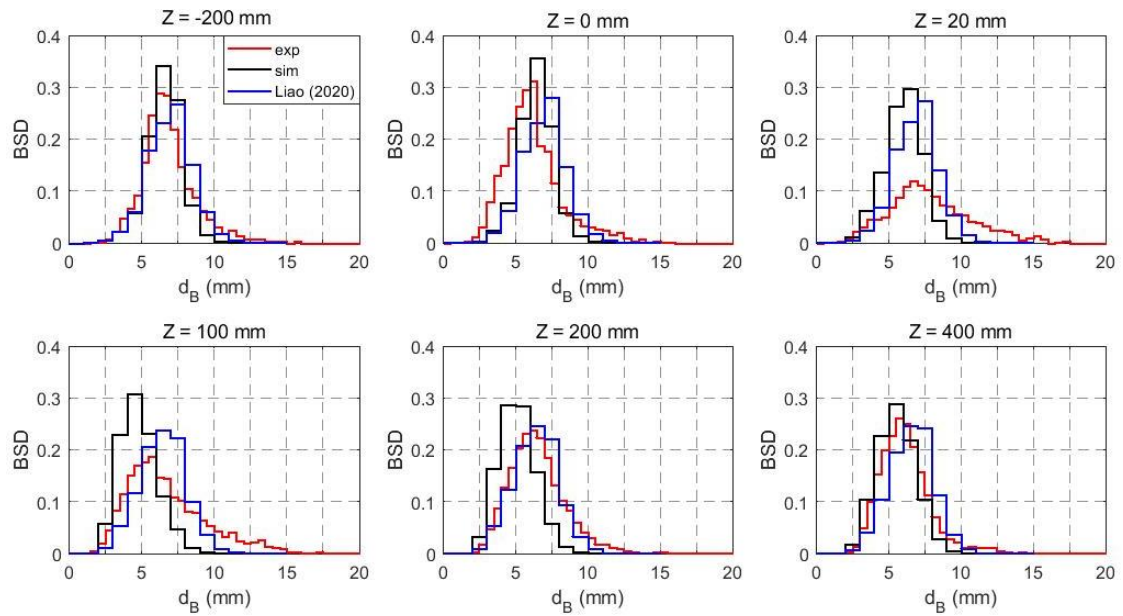


Figure 19: Bubble size distribution at different sections for test case 072 (a legend is given in the first graph).

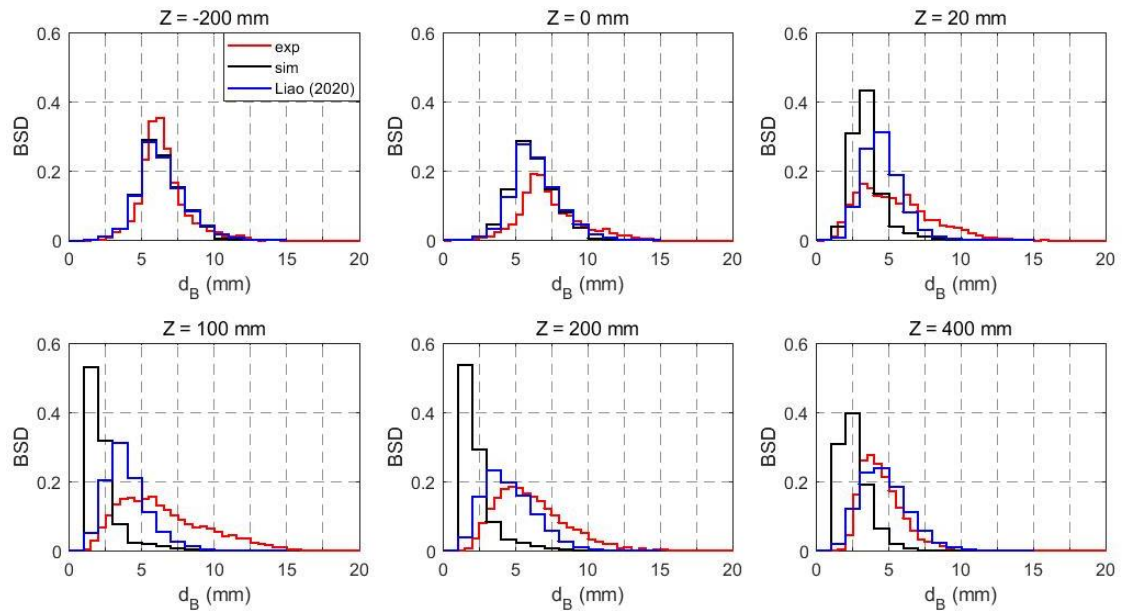


Figure 20: Bubble size distribution at different sections for test case 074 (a legend is given in the first graph).

TABLES

Table 1: Experimental operating conditions based on combinations of liquid and gas superficial velocities.

Test run	j_l [m·s ⁻¹]	j_g [m·s ⁻¹]
#072	0.4050	0.0368
#074	1.0170	0.0368

Table 2: Image plane identifier along the vertical test section with relative distances of the upper image plane to the center of the flow constriction.

Identifier	A	B	C	D	E	F	G	H	I
Z (mm)	-200	-60	0	5	20	50	100	200	400

Table 3: Mathematical description for interfacial force coefficients.

Force	Reference	Mathematical description
Drag	Ishii <i>et al.</i> (1979)	$C_D = \max[C_{D_{sphere}}, \min(C_{D_{ellipse}}, C_{D_{cap}})]$
		$C_{D_{sphere}} = \frac{24}{Re_d} (1 + 0.1Re_d^{3/4}), C_{D_{ellipse}} = \frac{2}{3} \sqrt{Eo}, C_{D_{cap}} = \frac{8}{3}$
Lift	Tomiyama <i>et al.</i> (2002)	$C_L = \begin{cases} \min[0.288 \tanh(0.121Re), f(Eo_\perp)] & Eo_\perp < 4 \\ f(Eo_\perp) & 4 < Eo_\perp < 10 \\ -0.27 & 10 < Eo_\perp \end{cases}$
		$f(Eo_\perp) = 0.00105Eo_\perp^3 - 0.0159Eo_\perp^2 - 0.0204Eo_\perp + 0.474$
Wall lubrication	Hosokawa <i>et al.</i> (2002)	$Eo_\perp = \frac{g(\rho_L - \rho_L)d_\perp^2}{\sigma} \quad d_\perp = d_B \sqrt[3]{1 + 0.163 Eo^{0.757}}$ $C_W(y) = f(Eo) \left(\frac{d_B}{2y}\right)^2, \quad f(Eo) = 0.021Eo$
Turbulent dispersion	Burns <i>et al.</i> (2004)	Favre averaging the drag force
Virtual mass	Auton <i>et al.</i> (1988)	Constant coefficient $C_{VM} = 0.5$

Conflict of Interest

15.06.2020

Dear Editor,

The authors certify that they have no affiliations or involvement in any organization or entity with any financial interest or non-financial interest in the subject matter or materials discussed in this manuscript.

We confirm that this work is original and has not been published elsewhere, nor is it currently under consideration for publication elsewhere.

Sincerely,

Sibel Tas-Koehler

Credit Author Statement

19.10.2020

Dear Editor,

The authors are responsible and ensure an accurate description. All authors contributed to scientific content and the editorial content of the present paper. The order of the authors represents the share of the contribution to the article.

Sincerely,

Tas-Koehler Sibel, Neumann-Kipping Martin, Liao Yixiang, Krepper Eckhard, Hampel Uwe