Scale-adaptive simulation of a square cross-sectional bubble column

Citation for published version (APA):

Ma, T., Lucas, D., Ziegenhein, T., Fröhlich, J., & Deen, N. G. (2015). Scale-adaptive simulation of a square cross-sectional bubble column. *Chemical Engineering Science*, 131, 101-108.
<https://doi.org/10.1016/j.ces.2015.03.047>

Document license:
TAVERNE

DOI:
[10.1016/j.ces.2015.03.047](https://doi.org/10.1016/j.ces.2015.03.047)

Document status and date:
Published: 01/01/2015

Document Version:
Publisher's PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:

- A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
- The final author version and the galley proof are versions of the publication after peer review.
- The final published version features the final layout of the paper including the volume, issue and page numbers.

[Link to publication](#)

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:

www.tue.nl/taverne

Take down policy

If you believe that this document breaches copyright please contact us at:

openaccess@tue.nl

providing details and we will investigate your claim.

Scale-Adaptive Simulation of a square cross-sectional bubble column

T. Ma^{a,*}, D. Lucas^a, T. Ziegenhein^a, J. Fröhlich^b, N.G. Deen^c

^a Helmholtz-Zentrum Dresden-Rossendorf, Dresden, Germany

^b Technische Universität Dresden, Dresden, Germany

^c Eindhoven University of Technology, Eindhoven, The Netherlands

HIGHLIGHTS

- Improved mean flow and turbulence prediction with Scale-Adaptive Simulations (SAS) for bubbly flows.
- SAS are compared with LES in the same geometry.
- Discussion of the effect of using bubble-induced turbulence modelling.
- Spectra analysis for a time signal in the centre of the bubble column.

ARTICLE INFO

Article history:

Received 1 December 2014

Received in revised form

27 February 2015

Accepted 27 March 2015

Available online 3 April 2015

Keywords:

Bubble column

Two-fluid model

Scale-adaptive simulation

Large eddy simulation

Energy spectra

ABSTRACT

This paper presents detailed Euler–Euler Scale-Adaptive Simulations of the dispersed bubbly flow in a square cross-sectioned bubble column. The main objective is to investigate the potential of this approach for the prediction of bubbly flows with anisotropic liquid velocity fluctuations. The set of physical models describing the momentum exchange between the phases was chosen according to previous experiences of the authors. Experimental data and Euler–Euler Large Eddy Simulation are used for comparison. It was found that the presented combination of sub-models provides good agreement with experimental data for the mean flow and liquid velocity fluctuations. The energy spectra of the resolved velocity obtained from the simulations are presented and compared to the experimental spectra.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

Many technical flows feature a continuous liquid phase and a dispersed gaseous phase. The turbulence of the liquid phase is an important phenomenon in such multiphase flows, as it has a strong influence on the local distribution of the dispersed phase, including bubble coalescence and breakup. A bubble column provides a good experimental system for the study of turbulent phenomena in bubbly flows and the development of computational models. In bubble columns a wide range of length and time scales exists on which turbulent mixing takes place. The largest turbulent scales are comparable to the characteristic length of the mean flow and depend on the column geometry and boundary conditions. The smaller scales depend on the bubble dynamics and are proportional to the bubble diameter. The smallest scales relate to the Kolmogorov scale and are generally smaller than the bubble diameter, in bubble column. In bubbly flows, the smallest scales are responsible for the dissipation of the turbulent

kinetic energy as in single-phase flow, but the bubbles can also generate back-scatter, i.e. energy transfer from smaller to larger scales (Dhotre et al., 2013). The combination of both effects can yield an overall enhancement or an attenuation of the turbulence intensity.

In the present paper several turbulence models are investigated. In CFD simulations of bubble columns, Reynolds-Averaged Navier–Stokes (RANS) models are traditional. The type of RANS model typically employed, however, is isotropic and based on an eddy-viscosity, while the turbulence created by the rising bubbles exhibits fine-scale anisotropy. Furthermore, the flow in bubble columns usually exhibits large-scale unsteadiness. In principle, all RANS models can be solved in unsteady mode, which then is termed URANS. The basic RANS model, however, is not conceived for partial resolution of the turbulent motion, even if the grid and time step would be small enough, which makes the URANS approach delicate in cases where the unsteadiness is due to an internal instability and not to unsteady boundary conditions, as discussed in Fröhlich and von Terzi (2008).

In that reference, an overview over Scale-Resolving Simulations (SRS) of various types is provided, together with a classification

* Corresponding author.

E-mail address: tian.ma@hzdr.de (T. Ma).

scheme. These methods are particularly designed to resolve large portions of the turbulent motion and hence seem to be more appropriate for bubble columns than URANS. Large Eddy Simulation (LES) falls into the category and offers the opportunity to resolve the large-scale anisotropic turbulence directly and to model the small scales with a Subgrid-Scale (SGS) model. But this approach is too expensive for most industrial flows due to high resolution requirements, especially in boundary layers. The Scale-Adaptive Simulation (SAS) approach (Menter and Egorov, 2010) is based on a transport equation for the turbulent length scale using the local von Karman length-scale as a reference. The information provided by the latter allows the SAS model to adjust to resolved structures in an unsteady simulation, so that generally less computational effort is required compared to LES.

Simulations related to the present study have been performed by different authors combining the Euler–Euler approach with LES or RANS (URANS). Deen et al. (2001) used LES with the Smagorinsky model to simulate a bubble plume in a bubble column with a square cross-section. They compared the results of LES with the prediction of a transient k – ϵ model and concluded that the better results can be obtained with LES. Dhotre et al. (2008) reported LES with a two-fluid model for the same experiment of Deen et al. (2001). They investigated the influence of SGS models in LES: the Smagorinsky model and the dynamic Smagorinsky model of Germano et al. (1991). It was found that the performance of both models was similar and gave good results compared to the experiment. This resulted from the fact that in that simulation the averaged value of C_s in the dynamic Smagorinsky model was close to the value of C_s with the Smagorinsky model. Niceno et al. (2008) supplied the one-equation model solving for the SGS kinetic energy (Davidson, 1997) with an extra source term (Pfleger and Becker, 2001) in the transport equation to represent bubble-induced turbulence (BIT) and applied it to a square cross-sectional bubble column. It was found that this approach gives good predictions, however, the unresolved part of the turbulent kinetic energy reaches about 20% with this approach, which might be too high for a traditionally LES (Fröhlich and von Terzi, 2008). For the URANS simulations, most previous work used two-equation eddy viscosity models. To consider BIT in those simulations, the model of Sato et al. (1981) or BIT-source terms in transport equation of the turbulence quantities were used, e.g. by Troshko and Hassan (2001), Politano et al. (2003), Ziegenhein et al. (2015), and Morel (1997). The model of Sato et al. (1981) defines an additional term for the turbulent viscosity to represent the influence of the bubbles on the liquid turbulence. The BIT-source term which is usually used in the k – ϵ equation assumes that the energy transferred by drag contributes to the turbulent kinetic energy. Corresponding source terms in ϵ - or ω -equation are obtained in an analogous way, like for a single phase flow, but there are different proposals in literature regarding the required time scale. Krepper et al. (2007) demonstrated the possible importance of the lateral forces including lift and wall force also for bubble columns by experimental and numerical investigations with URANS. They found a need for consideration a complete set of forces. Pfleger et al. (1999) studied the motion of a bubble plume in a rectangular column and used the k – ϵ model without source terms. The averaged liquid vertical velocity fitted well with the experimental data at different heights in this column. But the temporal period of the bubble plume could not be reproduced with this approach. Dhotre et al. (2008) compared the k – ϵ model, into which source terms proposed by Simonin and Viollet (1988) were included, with their LES simulation mentioned above. It was found, that this approach gives reasonably good agreement with almost all mean values of the experiment. However, the averaged anisotropic velocity fluctuation of the liquid was not well predicted with URANS, even if using the source terms to account BIT

Fig. 1. Geometry of the bubble column experiment of Deen et al. (2000). The broken line shows the measurement position. The marker on that line is the measurement point for the results presented in Figs. 10–12.

influence. Furthermore, a review of literature and a discussion of two-dimensional simulations can be found in the work of Bech (2005) and an extensive discussion of the merits of LES can be found in Dhotre et al. (2013), providing a systematic evaluation of prior work on the modelling of turbulent bubbly flows.

In the present work, Euler–Euler SAS are performed for a square cross-sectioned bubble column (Fig. 1). To the best of the author's knowledge, this is the first use of SAS for this type of flow. The results are compared to the experimental data from Deen et al. (2000) and to own LES. Furthermore, the effect of modelling bubble-induced turbulence with SAS and LES is discussed.

2. Physical modelling

2.1. Euler–Euler approach

In this work the Euler–Euler two-fluid model is used. The conservation equations are discussed in detail in a number of books, such as (Ishii and Hibiki, 2011), and a broad consensus on this model has been reached. For the special case of adiabatic flows without phase transition the governing equations in this approach are the continuity and momentum equations including source terms for the interfacial momentum transfer

$$\frac{\partial}{\partial t}(\alpha_i \rho_i) + \nabla \cdot (\alpha_i \rho_i \bar{\mathbf{u}}_i) = 0, \quad (1)$$

$$\frac{\partial(\alpha_i \rho_i \bar{\mathbf{u}}_i)}{\partial t} + \nabla \cdot (\alpha_i \rho_i \bar{\mathbf{u}}_i \bar{\mathbf{u}}_i) = -\nabla(\alpha_i \mu_i \bar{\mathbf{s}}_i) - \alpha_i \nabla p + \alpha_i \rho_i \mathbf{g} + \mathbf{M}_i - \nabla(\alpha_i \boldsymbol{\tau}_i), \quad (2)$$

here, the lower index i denotes the different phases, with α , ρ , μ and $\bar{\mathbf{u}}$ the volume fraction, density, molecular viscosity and resolved velocity, respectively. $\bar{\mathbf{s}}$ is the strain rate tensor. The vector $\mathbf{M}$ represents the sum of all interfacial forces acting between the phases such as drag force, lift force, wall lubrication and turbulent dispersion force. The unresolved stress tensor $\boldsymbol{\tau}$, and all interfacial forces have to be modelled. The applied modelling is discussed below.

2.2. Turbulence

In this study, turbulence is treated differently for the two phases. The turbulence in the dispersed gas phase is of little relevance and is modelled with a simple zero equation model $\mu_G^t = \frac{\rho_g}{\rho_l} \mu_L^t$, where μ_G^t and μ_L^t are the eddy viscosity of the gas and the liquid phase, respectively. It was found that this model has nearly no influence on the result, because of the low density of the gas and the low volume fraction in this case. For the liquid phase, SAS and LES were used in separate runs.

2.2.1. Scale-Adaptive Simulation for continuous liquid phase

The liquid velocity $\bar{\mathbf{u}}_L$ in (1) and (2) represents the resolved part only. The corresponding unresolved components are

$$\mathbf{u}'_L = \mathbf{u}_L - \bar{\mathbf{u}}_L, \quad (3)$$

where $\mathbf{u}_L$ is the true velocity of the liquid. The unresolved stress tensor $\tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j$, is modelled by the SAS model of [Menter and Egorov \(2010\)](#). It employs an equation for the turbulent kinetic energy K and an equation for $\Phi = \sqrt{K} L_t = \nu_t c_\mu^{-1/4}$, with L_t the turbulent length scale, and the components of $\mathbf{u}_L$ denoted U_j , reading:

$$\frac{\partial(\rho_L K)}{\partial t} + \frac{\partial(\rho_L \bar{U}_j K)}{\partial x_j} = P_K - c_\mu^{3/4} \rho_L \frac{K}{\Phi^2} + \frac{\partial}{\partial x_j} \left(\frac{\mu_L^t}{\sigma_K} \frac{\partial K}{\partial x_j} \right), \quad (4)$$

$$\frac{\partial(\rho_L \Phi)}{\partial t} + \frac{\partial(\rho_L \bar{U}_j \Phi)}{\partial x_j} = \frac{\Phi}{K} P_K \left(\zeta_1 - \zeta_2 \left(\frac{L_t}{L_{vk}} \right)^2 \right) - \zeta_3 \rho_L K + \frac{\partial}{\partial x_j} \left(\frac{\mu_L^t}{\sigma_\Phi} \frac{\partial \Phi}{\partial x_j} \right), \quad (5)$$

here, L_{vk} is the von Karman length-scale defined by

$$L_{vk} = \kappa \frac{S}{|\bar{\mathbf{U}}'|}, \quad S = \sqrt{2S_{ij}S_{ij}}, \quad S_{ij} = \frac{1}{2} \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right), \quad |\bar{\mathbf{U}}'| = \sqrt{\frac{\partial^2 \bar{U}_i}{\partial x_j \partial x_j} \frac{\partial^2 \bar{U}_i}{\partial x_k \partial x_k}}, \quad (6)$$

and

$$P_K = \mu_L S^2. \quad (7)$$

The values of the constants are $\zeta_1 = 0.8$, $\zeta_2 = 1.47$, $\zeta_3 = 0.0288$, $\sigma_\Phi = 2/3$, $\kappa = 0.41$, $c_\mu = 0.09$, according to [Menter and Egorov \(2010\)](#). The information provided by the von Karman length-scale allows the SAS model to treat large turbulent structures in a URANS fashion so that less computational effort is required compared to LES. The L_{vk} term can be transformed and implemented into any other scale-defining equation. Here, the Shear Stress Transport model (SST) is used, resulting in an SAS–SST model. [Menter and Egorov \(2005\)](#) introduced an additional SAS source term F_{SAS} in the ω -equation resulting from the transformation. The finally used version of this source term reads ([Egorov and Menter, 2008](#)):

$$F_{SAS} = \max \left\{ \rho_L \zeta_2 \kappa S^2 \left(\frac{L}{L_{vk}} \right)^2 - C \frac{2\rho_L K}{\sigma_\Phi} \max \left\{ \frac{1}{\omega^2} \frac{\partial \omega}{\partial x_j} \frac{\partial \omega}{\partial x_j}; \frac{1}{K^2} \frac{\partial K}{\partial x_j} \frac{\partial K}{\partial x_j} \right\}; 0 \right\}, \quad (8)$$

with $\zeta_2 = 3.51$ and $C = 2$. The equilibrium eddy viscosity of the SAS–SST model is:

$$\mu_L^t = \rho_L \left(\sqrt{((\beta/C_\mu) - \alpha)/(\kappa \zeta_2)} \cdot L_{vk} \right)^2 S, \quad (9)$$

with $\beta = 0.075$. This formula has a similar structure as the Smagorinsky SGS eddy viscosity used for the LES in this study as described below. Then, the SAS model is employed in its two-phase version, with the void fraction α_L incorporated in a

straightforward way into each term of (4) and (5). Furthermore, BIT is accounted for as described below.

Because the liquid is stagnant at the beginning, the initial condition for the turbulence parameters are considered only for the numerical stability, defined as: liquid turbulent kinetic energy $K = 3.75 \cdot 10^{-7} \text{ m}^2/\text{s}^2$ and eddy frequency $\omega = 0.042 \text{ s}^{-1}$ ([ANSYS, 2011](#)).

SAS aims to resolve the unsteady large-scale turbulent motion without explicit grid dependence and was for this reason, termed second generation URANS in [Fröhlich and von Terzi \(2008\)](#).

2.2.2. Large Eddy Simulation for continuous liquid phase

Results obtained with LES for the same configuration and with the same computational grid are used for comparison here. They were obtained with the Smagorinsky SGS model based on the eddy viscosity

$$\mu_{SGS} = \rho_L (C_s \Delta)^2 S, \quad (10)$$

here, the model constant C_s was chosen to be $C_s = 0.12$, while the filter width Δ was determined by the grid size from $\Delta = \sqrt[3]{\text{Vol}}$, with Vol. standing for the volume of a grid cell. LES employs a reduced eddy viscosity compared to URANS. Full details of this approach for two-phase flow are provided in [Ma et al. \(2014a, 2014b\)](#).

2.2.3. Bubble-induced turbulence

In the Euler–Euler approach bubbles are treated statistically, i.e. single bubbles are not resolved. The resolved part of the velocity field in SAS and LES only represents the shear-induced turbulence. The influence of bubbles traveling through the liquid on the liquid turbulence has to be modelled. Usually it is assumed that all energy lost by the bubble due to drag is converted to turbulent kinetic energy in the wake of the bubble. For classical two equation RANS models, additional source terms have been developed to describe BIT. Detailed information about BIT models in RANS can be found in the recent review by [Rzehak and Krepper \(2013\)](#). Such an approach is not suitable for LES with zero equation SGS model, because there is no transport equation for K available. For SAS, up to now, non-BIT models are available in the literature. For these reasons, the common BIT model of [Sato et al. \(1981\)](#) is employed here. In this model BIT is represented by an additional extra term in the SGS turbulent viscosity so that the effective viscosity

$$\mu_L^{\text{eff}} = \mu_L^{\text{mol}} + \mu_L^{\text{SGS}} + \mu_L^{\text{bub}}, \quad (11)$$

with

$$\mu_L^{\text{bub}} = C_B \rho_L \alpha_G d_B |\mathbf{u}_G - \mathbf{u}_L|, \quad (12)$$

is employed. Here, C_B is a model constant equal to 0.6, and d_B represents the bubble diameter. In SAS, the source for unresolved turbulent kinetic energy is proportional to the modified turbulent viscosity $\mu_L^{\text{t,mod}}$:

$$P_K = \mu_L^{\text{t,mod}} S^2 = (\mu_L^t + \mu_L^{\text{bub}}) S^2. \quad (13)$$

2.3. Interfacial forces

In the Eulerian two-fluid model the interaction between the bubbles and the liquid phase is modelled through exchange terms in the momentum equation of the liquid and the gas phase. There is still no agreement in the community on the closures to be used. The corresponding interfacial transfer models employed here are listed in [Table 1](#). A complete description of all interfacial transfer models can be found in the so-called baseline model of Helmholtz–Zentrum Dresden–Rossendorf ([Rzehak and Krepper, 2013](#)).

The turbulent dispersion force represents the bubble dispersion caused by the turbulent fluctuations of the liquid velocity. In URANS simulations, it has to be modelled, because these turbulent fluctuations are not simulated. In LES, however, the resolved part of the turbulent dispersion is explicitly calculated, and the unresolved part has little influence on bubble dispersion if the bubble size is on the scale of the filter size (Niceno et al., 2008). In contrast, using SAS the unresolved part is often not negligible, and larger than in LES. Here, the model from Burns et al. (2004) derived by Favre averaging of the drag force is used:

$$F_{Disp} = -\frac{3}{4}C_D \frac{\alpha_G}{d_b} |\mathbf{u}_G - \mathbf{u}_L| \frac{\mu_L^t}{\sigma_{TD}} \left(\frac{1}{\alpha_L} + \frac{1}{\alpha_G} \right) \nabla(\alpha_G), \quad (14)$$

with C_D the drag coefficient according to the correlation by Ishii and Zuber (1979) and σ_{TD} the turbulent Schmidt number taken to be 0.9, here. This model is not employed with LES.

3. Experimental data and simulation setup

3.1. Experimental data

The simulations were carried out for an air/water bubble column at ambient pressure (Deen et al., 2001), schematically shown in Fig. 1. The bubble column has a square cross-section of $0.15 \text{ m} \times 0.15 \text{ m}$ and is filled with distilled water up to a height of

0.45 m. A distributor plate containing 49 needles with an inner diameter of 1 mm was placed in the centre of the bottom with a square section of $A_{in} = 0.03 \text{ m} \times 0.03 \text{ m}$. Measurements were performed for the gas superficial velocity of 4.9 mm/s and took place 0.25 m above the distributor plate ($y=0.25 \text{ m}$) in the centre plane ($z=0.075 \text{ m}$). A particle image velocimetry system was used to simultaneously measure the gas and liquid velocities. In order to obtain a non-coalescing system, 4 g of kitchen salt per litre water was used. The bubble size was 4 mm (Deen et al., 2001). More details on the experiment are provided in that reference.

3.2. Simulation details

The gas inlet is defined as a distributed source in the central area $0.03 \times 0.03 \text{ m}^2$ at the bottom of the domain, representing the experimental needle setup. A superficial gas velocity of 4.9 mm/s is imposed leading to a gas velocity U_G at the inlet of 0.12 m/s with $\alpha_G = 1$ and $\alpha_L = 0$. At the walls, a no-slip boundary condition is applied for the continuous phase and a free slip condition for the disperse phase. At the top of the column a degassing boundary is imposed, which means an outlet for the dispersed gas phase. Here, the outlet pressure is not specified. Instead, a pressure distribution is computed on this fixed-position boundary, and can be interpreted as representing the weight of the surface height variations in the real flow. The continuous liquid phase sees the boundary as a free-slip wall and does not leave the domain. For the spatial discretization, a central difference scheme is employed, and a second-order backward Euler scheme is used in time. The simulations were carried out using a time step of $\Delta t = 0.0025 \dots 0.005 \text{ s}$ to satisfy $CFL < 1$. The simulations were run for 200 s.

The rectangular bubble column was discretized with uniform cubic cells of $\Delta x = \Delta y = \Delta z = 5 \text{ mm}$, resulting in 81,000 cells, overall. The bubbles are treated as mono-disperse with bubble diameter $d_b = 4 \text{ mm}$, according to the experiment. The spatial

Table 1
Models for interfacial forces employed for both cases in the present work.

Drag force	Ishii and Zuber (1979)
Lift force	Tomiyama et al. (2002)
Wall force	Hosokawa et al. (2002)
Virtual mass	$C_{VM} = 0.5$

Fig. 2. Instantaneous data from SAS and LES at $t = 70 \text{ s}$. The left two graphs show instantaneous isosurfaces of constant void fraction ($\alpha = 0.05$). The right two graphs show the instantaneous void fraction in the centre plane at the same instant.

Fig. 3. Vertical component of liquid velocity obtained from SAS, LES without BIT model and experiments.

Fig. 4. Vertical component of gas velocity obtained from SAS, LES without BIT model and experiments.

resolution hence is in the order of the bubble size ($\Delta x \approx d_B$). Resolution requirements for Euler–Euler methods are still subject of debate in the literature. The approach was conceived to represent bubble swarms on very coarse grids, far coarser than individual bubbles. The equations are derived by means of volume averaging, and the averaging volume then has to be substantial larger than individual bubbles. Once the equations derived, the problem to be solved is composed of a partial differential equation (PDE) with appropriate boundary conditions. This continuous problem can be solved numerically on an arbitrarily fine grid. One then obtains a very smooth solution and negligible numerical error. The numerical solution converges to the exact solution of the Euler–Euler PDE. Caution is needed only when interpreting the result. The fact that grid points are very close does not necessarily mean that the fluid motion is resolved with the same detail. The details have just been removed from the equations and the quantity determined is an averaged quantity. This is the same as solving the equations of the low-Reynolds $k-\varepsilon$ model on a grid with step size finer than the Kolmogorov scale which provides the almost exact solution to the RANS problem, but not a DNS. Starting from the limit of a very fine grid for the Eu–Eu equations and coarsening the cells the finest fluctuations may at some stage not be resolved anymore, so that an LES-SGS model has to be employed. This threshold, however, only depends on the Eu–Eu equations and has no relation to the bubble diameter. In the present work, the container is relatively small compared to the bubble size: $w=150$ mm and $d_B \approx 4$ mm yields a factor of $w/d_B \approx 38$. Discretizing the container cross section with 30×30 cells allows reliable discretization of derivatives, while substantially coarser grids do not. According to the above discussion, this grid is well suitable for the Euler–Euler method and hence

Fig. 5. Fluctuations of vertical liquid velocity component obtained from SAS, LES without BIT model and experiments. Continuous lines indicate the total fluctuations obtained from adding the resolved and the unresolved part. The unresolved part computed as explained in the text.

selected here according to the mesh studies of Niceno et al. (2008) and Sungkorn et al. (2011) to supply the best prediction. The simulations were conducted with the software Ansys CFX 14.5 using a finite volume solver.

4. Results

4.1. Instantaneous results

Snapshots of isosurfaces of constant void fraction with SAS and LES are shown in Fig. 2 together with the corresponding contour plots of the void fraction in the centre plane. On average, the bubbles reach the top of the column after about 1.2 s. The isosurfaces provide a good illustration of the dynamics of the bubble plume and look very similar to the numerical results presented in Niceno et al. (2009). For LES, the plume is more distorted than in case of SAS. This is caused by the smaller effective viscosity compared to SAS, leading to a better resolution of small fluctuations. The corresponding void fraction in the centre plane shows a difference between the results for the two turbulence models. In the results of SAS, due to the application of turbulent dispersion model, the gas volume fraction is distributed somewhat more homogeneously over the cross section. It was confirmed by an own simulation (not reported here), that using SAS without turbulent dispersion model yields a bubble plume rising up to the top of the column without much transversal spreading. This is in contrast to the experiment, where the bubble plume moves around in the bubble column in a random fashion (Deen et al., 2001).

4.2. Time-averaged results

In this section, the simulation results averaged from $t = 20$ s to $t = 200$ s are quantitatively compared with the experiment. The experimental data were taken at the centreline of the horizontal plane at height 25 cm, as indicated in Fig. 1. Time averaged vertical liquid and gas velocity, vertical and lateral liquid velocity fluctuation, gas void fraction and effective viscosity are presented for both cases.

Figs. 3 and 4 show profiles of the averaged vertical liquid and gas velocity. Satisfying quantitative agreement with the experiment is obtained for both simulations. The results of SAS have a bit lower values compared to the LES in both profiles. The liquid velocity profile is somewhat underpredicted with SAS, but well reproduced using LES. On the other hand, the gas velocity agrees better using SAS compared to the LES. This effect can be attributed to the drag force modelling and should be subject of further investigations. In both simulations the relative velocity between liquid and gas is about 0.2 m/s, i.e. it is nearly the same, resulting from the use of the same drag force model. The vertical liquid velocity changes its sign about

Fig. 6. Fluctuations of lateral liquid velocity component obtained from SAS, LES without BIT model and experiments. Continuous lines indicate the total fluctuations obtained from adding the resolved and the unresolved part. The unresolved part computed as explained in the text.

Fig. 7. Vertical liquid velocity obtained from SAS, LES and experiments, with and without Sato model.

0.02 m away from wall, which is in agreement with the experimental data. The downflow in the near-wall region is caused by the liquid mass balance in the bubble column.

Figs. 5 and 6 present a comparison of the vertical and lateral fluctuating liquid velocity. In the previous studies by Deen et al. (2001) and Dhotre et al. (2008), only the resolved velocity fluctuations were provided, while the unresolved part was neglected. In the present SAS and LES, the total velocity fluctuation is considered, summing the resolved part $\bar{u}'$ and the unresolved part $\bar{u}''$ as mentioned in Section 2.2 above. For the resolved part, $\bar{u}'\bar{u}' = [\bar{u} - \langle \bar{u} \rangle]^2$ is computed using the difference between the instantaneous resolved velocity $\bar{u}$ and the time average of the resolved velocity, $\langle \bar{u} \rangle$. Modelling the unresolved part is different for SAS and LES. SAS has a transport equation for the turbulent kinetic energy. Assuming isotropy of turbulence leads to $\overline{w'w'} = \overline{v'v'} = \overline{u'u'} = \frac{2}{3}K$. For LES, only the anisotropic part of the SGS stress tensor is considered in the Smagorinsky model, so that the information about the isotropic part, $\tau_{kk} = 2K$, is lost. Here, the following estimate for K_{SGS} is used (Menter, 2013):

$$K_{SGS} = \sqrt{\frac{\nu_{SGS} \varepsilon_{SGS}}{C_\mu}}, \quad \varepsilon_{SGS} = \nu_{SGS} S^2, \quad (15)$$

with ε_{SGS} being the SGS dissipation. The constant C_μ is taken from the $k-\varepsilon$ model.

The qualitative comparison of the fluctuations of the vertical liquid velocity shown in Fig. 5 is encouraging for both, SAS and LES. However, the twin-peaked shape seen in the experiments is only reproduced by the SAS. The unresolved vertical velocity fluctuation with the SAS is higher than that with the LES, due to

Fig. 8. Liquid phase effective viscosity obtained from SAS and LES, with and without Sato model.

Fig. 9. Gas void fraction obtained from SAS, LES, with and without Sato model.

the higher effective viscosity of the SAS (see Fig. 8). This effect is expected and matches the basic concept of corresponding hybrid LES/RANS methods (here SAS) as discussed by Fröhlich and von Terzi (2008). The fluctuating lateral liquid velocity shown in Fig. 6 is not predicted as well as the one of the vertical component. Quantitatively, both SAS and LES overpredict the magnitude to some extent. This overprediction is caused by the high ratio of the modelled lateral fluctuations to the total lateral fluctuations (about 35% in SAS and 16% in LES). Compared to the vertical direction, the resolved fluctuations in lateral direction are much smaller. But determining the unresolved parts isotropic turbulence is assumed with both turbulence models and provides the same values as for the vertical direction.

4.3. Effect of using the Sato model for BIT

Previous studies of Deen et al. (2001), Dhotre et al. (2008) and Niceno et al. (2008) mentioned that there was no significant change in the predictions using LES with and without the BIT model of Sato. This is not confirmed in the present work.

Fig. 7 shows the mean vertical liquid velocity profiles with and without the Sato model for both turbulence models. There is a more pronounced peak in the centre with both approaches, when the model is included, clearly overpredicting the experimental data. This can be attributed to the fact that, as shown in Fig. 8, the additional viscosity caused by the Sato BIT model in (12) yields an increased effective viscosity in both SAS and LES, which in turn damps the bubble plume dynamics. A behaviour similar to Fig. 7, with an even higher centre peak in the vertical liquid velocity profile was obtained by Zhang et al. (2006). He performed LES with an increased value of $C_s = 0.2$ in the Smagorinsky SGS model instead of $C_s = 0.12$. The

Fig. 10. Time history of the liquid vertical velocity obtained with SAS at the centre of measurement line, compared to experimental data from Deen et al. (2001).

Fig. 11. Time history of the liquid vertical velocity obtained with LES at the centre of measurement line, compared to experimental data from Deen et al. (2001).

application of the Sato model in the LES has a similar effect as increasing C_s , since it enhances the SGS viscosity.

Fig. 9 presents the comparison of the void fraction with and without the Sato model. Using the model in both SAS and LES, a more pronounced peak in the centre is obtained, compared to a flatter profile without the model. The larger gas volume fraction in this region also increases the liquid velocity in the centre and thus is a second reason for the steeper velocity profiles. Unfortunately, a comparison with the experimental data is not possible, because no void fraction profiles were measured.

In the earlier results of Deen et al. (2001), Dhotre et al. (2008) and Niceno et al. (2008) who also used the Sato model with LES, an increase of the vertical velocity in the centre was not observed as in the present work. One reason could be the fact that these authors used a different value of the lift coefficient, $C_l = 0.5$. In the present study, the lift force correlation of Tomiyama is used with a lower effective lift coefficient $C_l \approx 0.28$, along with a turbulent dispersion force that also results in dispersing the bubble plume. As reported by Lucas and Tomiyama (2011), the lift force may have a significant effect on the gas plume transverse spreading.

4.4. Analysis of spectra

Figs. 10 and 11 show time series of the resolved vertical liquid velocity at the centre of the measurement line shown in Fig. 1 for SAS and LES without BIT model. The period of 50 s presented in these figures corresponds to 189 sample points for the experimental data and 500 sample points for the simulations. Both, SAS and LES reflect the transient behaviour observed in the experiment of Deen et al. (2001) in terms of the amplitude of the fluctuations. The differences observed in the signals are caused by the different sampling rates, so that an investigation of the spectra is helpful.

The power spectrum densities (PSD) of SAS and LES without BIT model obtained with 18,000 sample points are shown in Fig. 12. The data were also extracted from the centre of the measurement line and cover the time interval $t = 20$ s to $t = 200$ s. A fast Fourier transform of the vertical liquid velocity signal using the Welch method (Welch, 1967) with non-overlapping sections and a Hanning window was performed. Because of the low sample frequency, the experimental spectrum can only be determined up to about 4 Hz.

Fig. 12. Power spectrum density of vertical liquid velocity taken at the centre of measurement line for SAS and LES in comparison to the result obtained from the experimental time signals.

Both spectra with SAS and LES exhibit a broad range of frequencies and slopes of about $-5/3$ in the inertial subrange, like single phase turbulent flow. Both predictions are in good agreement with the experiment in the low frequency region. Hence the large-scale vortical magnitudes are well reflected in the simulations. Compared to LES, the spectrum of SAS is more damped caused by the larger eddy viscosity. This is a similar effect as increasing the Smagorinsky constant C_s in a LES, which is discussed by Fröhlich (2006).

5. Conclusions

The simulation results obtained with SAS for a bubble plume show good agreement with LES results and experiment, both for the instantaneous bubble plume shape and for the mean-flow properties. The turbulence quantities determined with SAS are similar in quality as those obtained from LES in this case. Especially the trend of the fluctuations in the mean flow direction is well captured. SAS hence might be a promising option to be used with the Euler–Euler approach for the interfacial momentum exchange to predict bubbly flows.

The BIT model of Sato was investigated in SAS and LES. It is found, that the increased effective viscosity resulting from this model damps the bubble plume dynamics. This causes a more pronounced peak in the centre of the liquid velocity profile compared to experimental one. Such an effect of the Sato BIT model was not found by Ma et al. (2014a) and Ziegenhein et al. (2015) for a different bubble column with a homogeneous distributed gas inlet at the bottom. One reason could be the flatter gas volume fraction profile in their simulated bubble column. The impact of the increased effective viscosity caused by the Sato model for this reason is much higher in the present case.

Further investigations of the relationship between resolved and non-resolved turbulence and interfacial force modelling will be conducted in the future for different bubbly flow configurations.

Acknowledgments

This work was carried out in the frame of a current research project funded by the German Federal Ministry of Economic Affairs and Energy, project number 150 1411.

References

- ANSYS, 2011. CFX R14 Modelling Guide.
- Bech, K., 2005. Dynamic simulation of a 2D bubble column. Chem. Eng. Sci. 60, 5294–5304.

- A. Burns, T. Frank, I. Hamill, J.M. Shi, 2004. The Favre averaged drag model for turbulent dispersion in Eulerian multi-phase flows. In: Fifth International Conference on Multiphase Flow, ICMF2004, Yokohama, Japan, Paper no. 392.
- L. Davidson, 1997. Large Eddy Simulations: A Note on Derivation of the Equations for the Subgrid Turbulent Kinetic Energies, Technical Report no. 97/12, 980904, Chalmers University of Technology, Gothenburg, Sweden.
- N.G. Deen, B. H. Hjertager, T. Solberg, 2000. Comparison of PIV and LDA measurement methods applied to the gas liquid flow in a bubble column. In: Proceedings of the 10th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon, Portugal, July 2000.
- Deen, N.G., Solberg, T., Hjertager, B.H., 2001. Large Eddy Simulation of the gas-liquid flow in a square cross-sectioned bubble column. *Chem. Eng. Sci.* 56 (21–22), 6341–6349.
- Dhotre, M.T., Niceno, B., Smith, B.L., 2008. Large Eddy Simulation of a bubble column using dynamic sub-grid scale model. *Biochem. Eng. J.* 136 (2–3), 337–348.
- Dhotre, M.T., Deen, N.G., Niceno, B., Khan, Z., Joshi, J.B., 2013. Large Eddy Simulation for dispersed bubbly flows: a review. *Int. J. Chem. Eng.* 2013.
- Egorov, Y., Menter, F.R., 2008. Development and application of SST-SAS turbulence model in the DESIDER project. In: Peng, S-H, Haase, W (Eds.), *Advances in Hybrid RANS-LES Modelling. Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, vol. 97. Springer, Berlin, pp. 261–270.
- Fröhlich, J., von Terzi, D., 2008. Hybrid LES/RANS methods for the simulation of turbulent flows. *Prog. Aerosp. Sci.* 44, 349–377.
- Fröhlich, J., 2006. Large Eddy Simulation turbulenter Strömungen. Teubner-Verlag.
- Germano, M., Piomelli, U., Moin, P., Cabot, W.H., 1991. A dynamic subgrid-scale eddy viscosity model. *Phys. Fluids A* 3, 1760–1765.
- S. Hosokawa, A. Tomiyama, S. Misaki, T. Hamada, 2002. Lateral migration of single bubbles due to the presence of wall. In: *Proc. ASME Joint U.S.–European Fluids Engineering Division Conference, FEDSM2002*, Montreal, Canada, 855.
- Ishii, M., Zuber, N., 1979. Drag coefficient and relative velocity in bubbly, droplet or particulate flows. *AIChE J.* 25, 843.
- Ishii, M., Hibiki, T., 2011. *Thermo-fluid Dynamics of Two-phase Flow*, second ed. Springer.
- Krepper, E., Vanga, B.N.R., Zaruba, A., Prasser, H.M., M. A. Lopez, de Bertodano, 2007. Experimental and numerical studies of void fraction distribution in rectangular bubble columns. *Nucl. Eng. Des.* 237, 399–408.
- Lucas, D., Tomiyama, A., 2011. On the role of the lateral lift force in poly-dispersed bubbly flow. *Int. J. Multiphase Flow* 37, 1178.
- T. Ma, T. Ziegenhein, D. Lucas, J. Fröhlich, E. Krepper, 2014a. Large Eddy Simulation for Dispersed Bubbly Flows, *CFD4NRS-5*, Zürich.
- Ma, T., Lucas, D., Ziegenhein, T., Fröhlich, J., 2014b. Large Eddy Simulation of a bubble plume. In: Fröhlich, J., Odenbach, S., Vogeler, K. (Eds.), *Strömungstechnische Tagung*, TUDpress, pp. 337–344 2014.
- Menter, F.R., Egorov, Y., 2010. Scale-Adaptive Simulation method for unsteady flow predictions. Part 1: Theory and model description. *Flow Turbul. Combust.* 85 (1), 113–138.
- F.R. Menter, Y. Egorov, A Scale-Adaptive Simulation Model Using Two-equation Models. *AIAA Paper* 2005-1095, 2005.
- Menter, F.R., 2013. Private Communication.
- Morel, C., 1997. *Turbulence Modeling and First Numerical Simulations in Turbulent Two-phase Flows*. CEA Grenoble, France.
- Niceno, B., Dhotre, M.T., Deen, N.G., 2008. One-equation subgrid scale (SGS) modelling for Euler–Euler Large Eddy Simulation (EELES) of dispersed bubbly flow. *Chem. Eng. Sci.* 63 (15), 3923–3931.
- Niceno, B., Boucker, M., Smith, B.L., 2009. Euler–Euler Large Eddy Simulation of a square cross-sectional bubble column using the Neptune CFD code. *Sci. Technol. Nucl. Install.* 2009, Article ID 410272.
- Pfleger, D., Gomes, S., Gilbert, N., Wagner, H.G., 1999. Hydrodynamic simulation of laboratory scale bubble columns fundamental studies of the Eulerian–Eulerian modelling approach. *Chem. Eng. Sci.* 54, 5091–5099.
- Pfleger, D., Becker, S., 2001. Modelling and simulation of the dynamic flow behaviour in a bubble column. *Chem. Eng. Sci.* 56 (4), 1737–1747.
- Politano, M., Carrica, P., Converti, J., 2003. A model for turbulent polydisperse two phase flow in vertical channels. *Int. J. Multiphase Flow* 29, 1153–1182.
- Rzehak, R., Krepper, E., 2013. Bubble-induced turbulence: comparison of CFD models. *Nucl. Eng. Des.* 258, 57–65.
- Sato, Y., Sadatomi, M., Sekoguchi, K., 1981. Momentum and heat transfer in two-phase bubble flow-I. Theory. *Int. J. Multiphase Flow* 7 (2), 167–177.
- O. Simonin, P.L. Viollet, On the computation of turbulent two-phase in Eulerian formulation. In: *Proceedings of EUROMECH 234*, Toulouse, France, 1988.
- Sungkorn, R., Derksen, J.J., Khinast, J.G., 2011. Modeling of turbulent gas-liquid bubbly flows using stochastic Lagrangian model and lattice-Boltzmann scheme. *Chem. Eng. Sci.* 66 (12), 2745–2757.
- Tomiyama, A., Tamai, H., Zun, I., Hosokawa, S., 2002. Transverse migration of single bubbles in simple shear flows. *Chem. Eng. Sci.* 57 (1849).
- Troshko, A.A., Hassan, Y.A., 2001. A two-equation turbulence model of turbulent bubbly flows. *Int. J. Multiphase Flow* 27, 1965–2000.
- Welch, P.D., 1967. The use of fast Fourier transform for the estimation of power spectra: a method based on time averaging over short, modified periodograms. *IEEE Trans. Audio Electroacoust.* AU-15, 70–73.
- Zhang, D., Deen, N.G., Kuipers, J.A.M., 2006. Numerical simulation of the dynamic flow behavior in a bubble column: a study of closures for turbulence and interface forces. *Chem. Eng. Sci.* 61, 7593–7608.
- Ziegenhein, T., Rzehak, R., Lucas, D., 2015. Transient simulation for large scale flow in bubble columns. *Chem. Eng. Sci.* 122, 1–13.