Proceedings of the 8th World Congress on Mechanical, Chemical, and Material Engineering (MCM'22)
Prague, Czech Republic – July 31, 2022 - August 02, 2022
Paper No. HTFF 188
DOI: 10.11159/htff22.188

Eulerian Approach to CFD Analysis of a Bubble Column Reactor – A Review

Mohammed W. Abdulrahman, Nibras Nassar
Rochester Institute of Technology
Dubai, UAE
mwacad@rit.edu; nin8507@rit.edu

***Abstract* -** Bubble Column Reactors (BCR) / Slurry Bubble Column Reactors (SBCR) have many advantageous characteristics as such they are used in numerous industrial applications. This work reviews the Eulerian Computational Fluid Dynamics (CFD) approach when analysing BCR/ SBCRs. Several studies have been reviewed which vary parameters such as the reactor design, superficial gas velocity, pressure, CFD models (drag, turbulence), particle concentration, and phase material to investigate their effects on the reactor's performance in terms of hydrodynamics or heat transfer. This review indicates that using a Eulerian CFD model can accurately predict the BCR/SBCR's performance. Key findings include that increasing the superficial gas velocity, column pressure, and gas phase density increases the gas holdup. Gas holdup is unevenly distributed in the BCR where most of the gas holdup is in the centre of the column. Increasing solid particles decreases the bubble breakup rate and gas holdup. Furthermore, it was concluded that increasing the superficial gas velocity increases the average slurry temperature and volumetric heat transfer. However, decreasing the column height increases the slurry temperature and volumetric heat transfer.

***Keywords*:** CFD; bubble columns; hydrodynamics; heat transfer; Eulerian

1. Introduction

Bubble column reactors are multiphase vertical reactors which contain a gas, a liquid or slurry, and fine particles where the gas is fed into the reactor by a sparger where bubbles are formed inside the liquid/ slurry. BCR/SBCR are advantageous as they can be used in a variety of industrial applications. Furthermore, the liquid in the three-phase reaction is beneficial as it allows for a more precise temperature control due to the liquid's high heat capacity [1]. Scaling up, modelling, and designing slurry bubble column reactors is a complex process as it requires detailed knowledge in relation to kinetics, hydrodynamics, heat and mass transfer, chemical reaction rates, phase holdup, flow regimes, pressure change, and solid distribution. CFD simulations can be used to help model the reactions and design the reactors. When studying BCR/SBCR it is important to focus on the gas holdup as it is one of the most important characteristics of the BCR [2]. Over the years research has been conducted in relation to bubble column reactors both experimentally and by means of CFD software. Research conducted has investigated theeffects of gas holdup, coalescence, reactor dimension, operational pressure, and heat transfer onthe reactions. This review will aid in future development and design of BCR/SBCRs by highlighting the CFD findings of previous studies.

2. Hydrodynamic Investigations of BCR/ SBCR

Meier et al. [3] conducted a 3D CFD investigation of a gas-liquid flow in a churn turbulent regime in order to compare the effectiveness of several models in relation to predicting the drag closures, breakup, and coalescence. Twelve combinations of the breakup and coalescence models were created and simulated. The combinations included a mixture of the breakage closures and coalescence closures. The Breakage closures used by Luo and Svendsen [4], Lehr et al. [5], and Laakkonen et al. [6] with Generalized PDF distribution. The coalescence closures used were Prince and Blanch [7], Luo [8] and Das [9]. Once simulated the CFD models were compared to the experimental data of Manjrekar and Dudukovic [10]. The Manjrekar and Dudukovic [10] bubble column reactor has a cylindrical diameter of 20.32 cm and a height of 2m. Experiments were carried out on the combination of 12 models with air-water flows at superficial gas velocities of 20 cm/s and 40 cm/s. The results highlighted the importance of selecting the appropriate breakage and coalescence closure model. In particular the breakup model, as the breakup model had a larger impact on the flow prediction than the coalescence model. The key findings of this report indicated that the Luo and the Prince and Blanch models for coalescence were similar in results and were both able to best predict the experimental data points at 6.2% and 7.2% relative error ata superficial gas velocity of 20 cm/s. When considering the breakup closure a combination of the Laakkonen et al. [6] model with the generalized PDIF distribution coupled with the breakagegenerating three daughter bubbles resulted in an acceptable prediction of the experimental results. Additionally, this combination was the least computationally expensive for the break up closure simulations.

The Schiller and Naumann [11] model was found to be the best model to predict the drag closure in comparison to experimental data. Overall, the simulated results were able to predict the experimental data with low relative errors for the gas axial velocity (7.7% at 20 cm/s and 14.0% at 40 cm/s) and for the gas holdup (14% at 20 cm/s and 21.9% at 40 cm/s). The computational predicted data for the gas axial velocity in the centre of the bubble column showed better agreement with the results than the predictions near the wall. The simulated results near the wall were over predicted in comparison to the experimental data. Ertekin et al. [12] validated the hydrodynamic CFD models presented by Fletcher et al. [13] while varying conditions such as column diameter from 0.19 m to 3m and the superficial gas velocities which varied from 0.03 m/s to 0.25 m/s based on the experimental data of Raimundo et al. [14] and McClure et al. [15]. Furthermore, Ertekin et al. [12] validated the experimental model using the effects of different phase materials on the gas holdup versus superficial gas velocity as well as the oxygen transfer rates versus the superficial gas velocity.

The Fletcher et al. [13] model used in this validation was a two-phase Euler- Euler model using a fixed single bubble size. A standard k-Ɛ turbulence model was used for the liquid phase, while the gas phase was modelled using a dispersed phase zero equation. Bubble induced turbulence was accounted for using the Yao and Morel model. Ertekin et al. [12] verified the models used by Fletcher et al. [13], finding that there was a minimal percent difference when studying the holdup. It was observed that there was a percentdifference of 3%- 10% between experimental and simulated data for the water, air phases. Simulated results were under predicted near the centre of the column when the height to diameterratio was 3.75. This was the largest discrepancy. When using organic liquid phases and phases which included surfactants, the proposed model showed good agreement in the gas holdup to superficial velocity data points. Some phase results were more agreeable than others. Furthermore, good agreement of the simulation with the experimental data was observed for results pertaining to the oxygen transfer rate vs the superficial velocity. The results obtained from this test however may vary depending on the surfactant concentration and type of surfactantpresented in the experiment.

Yan et al. [16] used three different optimized drag models to simulate the hydrodynamics of a high pressure, air-water bubble column. The effects of changing the superficial gas velocities (0.121, 0.174, 0.233 and 0.296 m/s) and the effects of changing the reactors pressure (0.5, 1.0, 1.5 and 2.0 MPa) on the radial gas holdup were investigated. The bubble column reactor had a diameter of 0.3m and a height of 6.6m. The gas sparger has 128 opening at a size of 5mm each located 0.2m from the base of the reactor. The data was investigated using 2D and 3D CFD simulations and compared to experimental calculated data using the electrical resistance tomography method. The first optimized drag model was based on Roghair's drag model in which correction were made by introducing the gas holdup of large and small bubbles to aid in density correction. The second model neglected the gas holdup effects of small bubbles with diameters smaller than 0.08m and combined their effects with the liquid phase. The third model was modified using an energy minimization multiscale (EMMS) approach based on the double bubble size model and a link between the drag coefficient and bubble diameter was implemented in the model. After completing the analysis several conclusionswere drawn. It was noticed that as the superficial gas velocity increased so did the radial gas holdup in the bubble column in the cold-water air model. Additionally, increasing pressure within the reactor increases the radial gas holdup. Although all three models tested in this report were different they were all capable of depicting the trends of the drag coefficient in relation to the changes with superficial gas velocity and pressure differences. It was noted that bubble sizehas a key impact on the mass and heat transfer and that using a PBM (Population Balance Model) model should be investigated. Lastly, it was noted that the middle of the bubble columnis most likely to have the greatest gas holdup. This was observed in both the 2D and 3D simulations as well as in the experimental bubble column.

Sarhan et al. [17] investigated the effects of the physio-chemical properties of the liquid and gas phases on bubble formation and hydrodynamics of a bubble column reactor using the population balance equation combined with a 3D CFD model. The bubble column reactor was designed after the experimental reactor constructed by Abdulrahman [18-21] with a diameter of 0.216m, a column height of 0.915m and a liquid height of 0.65m (Fig. 1). The experimental data conducted by Abdulrahman [18-19] was used to validate the CFD model predictions. The two-phase flows were conducted with different materials for gas and liquid. The gases used were helium, air and argon. The liquids used were water and paraffin oil. The CFD model was then used to predict the gas holdup, Sauter mean bubble size distribution and local time average bubble velocity within the column reactor using the different phase flows at different velocities (0.01, 0.05, 0.09 and 0.13 m/s). Sarhan et al. [17] were able to create a Euler-Euler CFD model to predict experimental results of the gas holdup in a bubble column reactor using different phase flows within the rage of ±7%. Additional conclusions which were drawn from the experiment include, the observation that the gas holdup will increase slightly as the gas phase density increases. If the liquid phase has a lower density such as paraffin oil the average gas holdup will increase significantly. If water is used as the liquid phase the bubble rise velocity will increase significantly. However, the bubble rise velocity will decrease if there is an increase in the gas phase density. Lastly, Sarhan et al. concluded that if the liquid phase density increases so will the Sauter mean bubble diameter. The Sauter mean bubble diameter will decrease as the gas phase density increases.

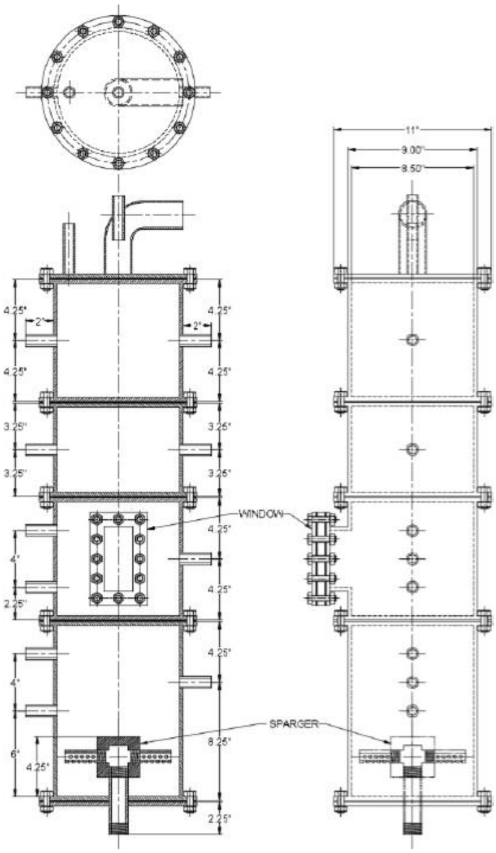

Fig. 1: Experimental bubble column reactor used by Abdulrahman [18].

Adam and Tuwaechi [22] generated a 2 phase, gas - liquid, Eulerian- Eulerian, k-Ɛ mixture turbulence CFD model, to study the effects of gas holdup and superficial gas velocity onthe hydrodynamics using a course and fine mesh. The bubble column reactor simulated had a height of 0.96 m and a diameter of 0.19m. From the CFD model it was observed that as the time step increased so did the volume fraction. The finer mesh with a grid resolution of 0.005 lead to aclearer observation. The last observation made by Adam and Tuwaechi was that the highest pressure was observed near the gas inlet and gradually decreases when moving away from the inlet.

Li et at. [23] conducted both experimental and CFD hydrodynamic analysis of an air-water- glass beads slurry bubble column (Fig. 2). The experiments conducted investigating the effects of changing the reactor diameters (0.2, 0.5 and 0.8m). The reactions occurred at high superficial gas velocities ranging from 0.12 to 0.62 m/s with varying solid concentrations from 0% to 30%. The effects of the changing parameters on the gas holdup, time average liquid flow and kinetic energy were recorded. The slurry bubble column reactor was modelled using a 2D axisymmetric two fluid Euler k-Ɛ model. The research conducted lead to several conclusions. It was observed that as the superficial gas velocity increases so the does the average gas holdup. Additionally, it was noted that as the average gas holdup increases so does the averaged liquid velocity and turbulent kinetic energy. Bubble breakage was hindered by the presence of solid particle which lead to an increase in bubble rise velocity and a reduction in gas holdup. The solidconcentration lead to a small change in time average axial liquid velocities. Furthermore, it was noted that the change of hydrodynamic characteristics with column diameters is the major cause of bubble column scale-up rules. Bubble columns with wider reactor diameters result in the axialliquid velocity rising dramatically within the column core, while the gas holdup is very minimally influenced. Lastly, it was noted that with increasing column scales, turbulent kinetic energy rises.

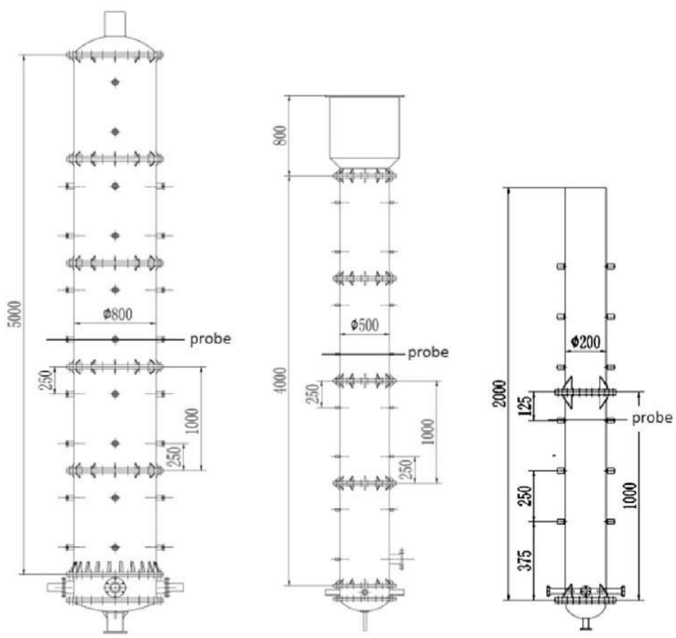

Fig. 2: Slurry bubble columns used by Li et al. [19].

Pourtousi et al. [24] investigated the bubble column regime and the effects of changingsuperficial gas velocity (0.0025 – 0.015 m/s) and varying bubble diameters (3, 4, 5 and 5.5mm) on the Euler-Euler simulation flow pattern predictions. A 3D air water CFD simulation was created with a slurry bubble column with a height of 2.6m and diameter of 0.288m. The simulated ring sparger had 20 holes and a diameter of 0.14m. The simulated data was then compared to experimental data to ascertain the effectiveness. It was noted that it is crucial to observe the size and shape of the bubbles formed near the sparger in the experiment to accurately predict the bubble's simulated hydrodynamics. A 3mm bubble diameter for superficial gas velocities ranging from 0.0025-0.015 m/s lead to acceptable simulated predictions near the bulk region. However, when considering data near the sparger, a bubble diameter of 5 or 5.5 resulted in a better predicted simulated result. Considering a single bubble diameter can result is an acceptable prediction in the homogeneous regime as it is less computationally expensive. The same cannot be assumed for a heterogeneous regime as it would lead to inaccurate results. In order to achieve more accurate simulated results for a heterogeneous regime a range of bubble diameters should be included as well as the implementation of a drag model in the simulation. Heterogeneous flows have bubbles with different shapes and sizes which range from 0.05mm to50mm. Conversely, homogenous flows have consistent bubble shapes and sizes that have minimal interaction with each other within the bubble column reactor.

Li and Zhong [25] conducted a 3D, Eulerian-Eulerian-Eulerian, three phases (air-water –glass powder), time dependent, CFD analysis of three different bubble column reactors to studythe hydrodynamics in relation to time step, momentum discretization schemes and wall boundaryconditions as well as a sensitivity analysis using different drag models. The three different models used where the Gandhi et al. (height:2500mm, Static height: 1500mm diameter:150mm),the Rapure et al. model (height: 2000mm static height: 1000mm diameter:200mm) and the Li and Zhong model (height:800mm width: 100mm depth:10mm) (Fig. 3). The turbulence model used for their simulations was the RNG k-Ɛ model. Li and Zhong concluded that the conditions that best reflected experimental results were the use of a no slip condition, momentum discretization using the second order upwind, and a time step of 0.001s. While considering the drag forces it was found that modelling the gas-liquid drag model was best using the Zhang- Vanderheyden model. While considering liquid-solid drag models the Schiller –Naumann modelwas best. Furthermore, it was noted that the bed gas holdup was heavily influenced by the superficial gas velocity. It was observed that as the superficial gas velocity increased so did the time average gas holdup and the liquid axial velocity however, the local solid holdup dropped. On the distributions of solid holdup and liquid axial velocity, the impacts of solid volume fraction (Vs=0.03– 0.30) and particle size (dp =75 μm–270 μm) were stronger than those of particle density (ρ = 2500 kg/m^3–4800 kg/m^3). When the particle size dp was less than 150 m and the solid volume fraction Vs was less than 0.09, the effects of solid volume fraction and particle size were more noticeable. The bed solid volume fraction distributions can be effectively described by the axial distributions of solid holdup. The axial solid concentration gradient was higher the larger the solid volume fraction, particle size, and particle density values were. Solid volume fraction solid volume fraction had a bigger influence on radial gas holdup than the other two particle attributes, while both solid volume fraction and particle size had a greater effect on liquid axial velocity. When the solid volume fraction and particle size rose, the liquid axial velocity in the bed centre portion dropped.

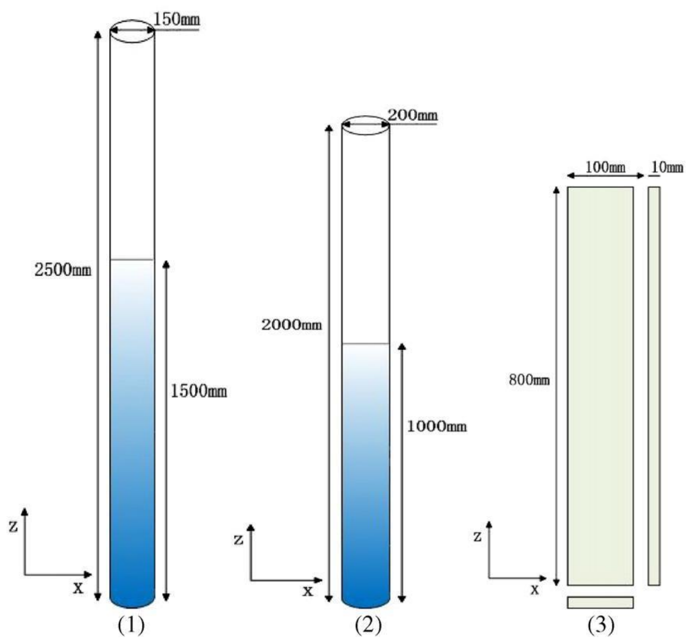

Fig. 3: Bubble column reactors investigated by Li and Zhong [21].

Dhotre et al. [26] conducted a Euler- Euler, gas-liquid, CFD simulations based on Deen et al. [27] experimental work to assess the LES (two sub grid scale models) and K-Ɛ model’s performance. The two sub grid scale models investigated were the Smagorinsky model and the dynamics model of Germano. Additionally, the Sato model was used to include the effects of turbulence caused by the bubbles on the effective viscosity. The reactor being simulated had a square base with a width and depth of 0.15m and a height of 0.45m. The square gas inlet has a width and depth of 0.03m and the superficial gas velocity at the inlet was set to 0.12 m/s. Dhotre et al. [26] study resulted in several conclusions. It was observed that the addition of the Sato model to include the effects of bubble induced turbulence had little impact on the results. It was also concluded that the Smagorinsky and the dynamic approach of Germano led to similar results when the sub grid scale model constant (Cs) was 0.12. Using the Germano method can help predict the Cs values when it is not previously known. Lastly, Dhotre et al. [27] stated that the K-Ɛ model can give comparable results to the mean experimental data when adjusting the turbulent kinetic energy and dissipation rate. However, the K-Ɛ model's results for the radial and axial distribution of the liquid velocity and turbulent kinetic energy near the wall does not reasonably follow the mean experimental data.

3. Effects of Solid Particles in BCR/ SBCR

Abdulrahman [28] investigated the effects of varying the solid concentrations from 0%, 5% and 10% on the gas holdup with reactor diameter of 21.6 cm and heights of 45 cm and 65cm. The study was conducted using a 2D CFD software assuming a two-phase slurry of water and helium. The system was modelled using a multiphase Eulerian model with a viscous-standardk-Ɛ turbulence model. The results indicated that as the concentration of solid particle increases the gas holdup decreases. This relationship remains true at any static liquid height and any specific superficial gas velocity.

Zhou et al. [29] analysed the effects of particles on a gas liquid flows in a slurry bubblecolumn using a conceptual model. The particle dependent dual bubble size (PDBS) model was created to investigate the effects of viscosity and density changes due to the addition of particles,as well as the change to the bubble drag co-efficient due to presence of particles. The model was a three-phase model composing of air, water, and glass beads. The PDBS was tested using CFD simulations using three different reactor designs. The three different bubble column reactor setups that were used to test the PDBS was the 3D square Ojima et al. reactor, the 3D cylindrical Gandhi et al. and the 2D Tyagi and Buwa reactor as depicted in Fig. 4.

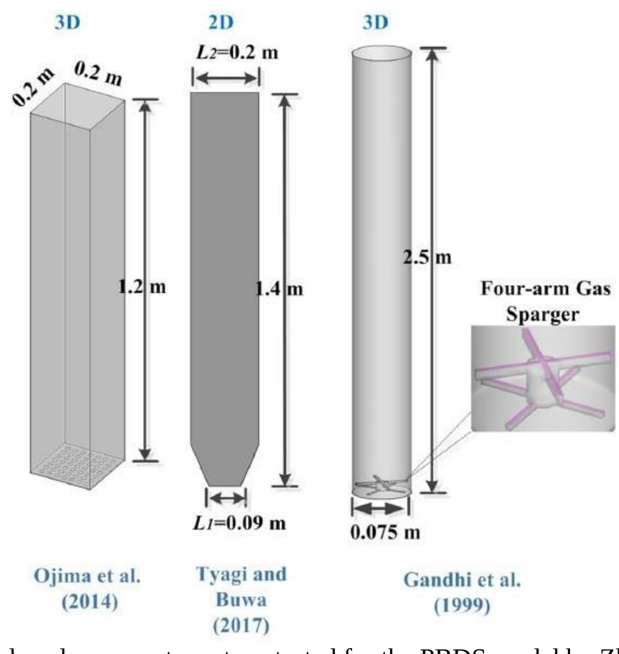

Fig. 4: Bubble column reactor setups tested for the PBDS model by Zhou et al. [29]

Zhou et al. [29] came to several conclusions while testing the PDBS model. When consideringthe effects of viscosity and density it was observed that there was a higher level of stability with increased slurry viscosity and density. This was apparent as there was a delay in the flow regime transition to a higher flow rate. In regards to the changes with the bubble's drag coefficient usingthe PDBS model it was concluded that bubble coalescence is encouraged in the presence of wettable particles. This conclusion in agreeable with experimental data as the overall gas holdup and energy consumption of bubble breakage decreases using the PDBS model. The PDBS modelcan predict results for the gas holdup and bubble breakage with changes in solid concentration. Overall, it was concluded that the increase in solid concentration will result in a decrease of thegas holdup.

Abdulrahman [30] developed a CFD simulation to predict the effects of changing static-liquid heights (45cm, 55cm, and 65mm) and superficial gas velocities (0.05 m/s, 0.1 m/s and 0.15 m/s) on gas holdup and compared the results to experimental results. A 2D CFD simulation of a two-phase slurry consisting of water and helium to analyse the hydrodynamics in steady state was developed. The reactor diameter was fixed at 21.6 cm. The study concluded that at any superficial gas velocity the overall gas holdup would increase if the superficial gas velocity increased. However, at any superficial gas velocity the gas holdup would decrease if the static liquid height was increased. It was noticed that the gas holdup was unevenly distributed along the cross section of the column, where higher gas holdups where found in the centre and lower gas holdups where found near the wall. When comparing the CFD results to experimental data itwas found that the gas holdup in the simulations where under-predicted. While, CFD results of the reactor's height effects on the gas holdup was correctly predicted.

Chen and Brooks [31] conducted both experimental and 3D analysis of a cylindrical bubble column to study the mass transfer and hydrodynamics. The cylindrical reactor had a diameter of 5 inches, a high of 6.55 inches, and a gas inlet at the centre of the reactor with a diameter of 0.12 inches (Fig. 5). The experiment was conducted with 2 phase flows of deionizedwater and air. Scenarios in this experiment were limited to small void fraction cases in which thebubbles may be easily observed. Flow visualization was then used to create a new approach for obtaining a 2D void fraction profile when the void fraction was low. The approach of image processing is used in the Particle Image Velocimetry tests to reduce reflection disturbance. Concentration distribution was not measured in relation to the dissolved oxygen experiment as the mixing occurred rapidly in the small reactor. Under the same conditions, comparisons are done between experiments and the simulations for gas holdup, void fraction profile, liquid velocity field, and volumetric mass transfer coefficient. Conclusions reached in this paper suggest that validation should be conducted with time averaged data, if bubble induced turbulence and turbulent dispersion forces are taken into consideration. Volume average void fractions using the CFD model were predicted within an acceptable limit. It was observed that monitoring the void fraction and velocity profile could be predicted a few centimetres away fromthe inlet, however predictions directly above the inlet was limited. CFD is a valuable tool when simulating low void fractions as it shows good agreement with experimental data. However, for higher gas flow rates the results obtained from visualization and Particle Image Velocimetry methods stray from the CFD data leading to higher levels of uncertainty which may result in an unreliable CFD model. When considering complex conditions and higher void fractions to study heat and mass transfer it is important to include turbulence for two phase flows as it will lead to more agreement between the experimental and simulated results.

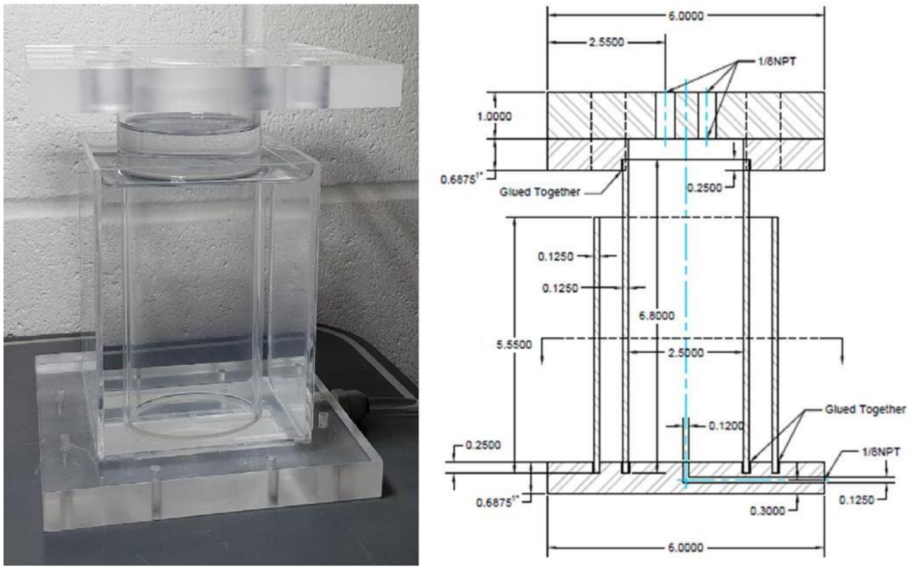

Fig. 5: Experimental bubble column reactor investigated by Chen and Brooks. (Chen and Brooks [31]).

Sarhan et al. [17] used the experimental data of Bhunia, Kundu and Mukherjee to create a 3-dimensional flow simulation of a flotation column reactor. The simulation was used predict the bubble surface area flux while varying the superficial gas velocity (0.64 cm/ to 2.76 cm/s), and the solid particle concentration (5% and 10%) and type (sphalerite and coal). The model to conduct the simulation was a Eulerian- Eulerian with a k-Ɛ dispersed turbulence model. The reactor simulated had a height of 1.66m with a diameter of 0.1m. Conclusions drawnby Sarhan et al. include increasing superficial gas velocity increase the bubble surface area flux. Increasing the solid concentration and or the hydrophobicity will decrease the bubble surface area flux at any given superficial gas velocity. Additionally, it was noted that as the coal concentration was increased from 0% to 10% the surface area flux reduced by 28%. However, ifsphalerite was present the bubble surface area flux increased by 7%. Lastly, it was observed thatthe addition of hydrophobic particles leads to a decrease in bubble concentration by approximately 23% assuming the same operating parameters.

Wodolazski [32]. Generated a 3D CFD simulation of a slurry bubble column reactor to analyse the flow of syngas in a 3-phase flow (syngas, paraffin oil, solid particles). A Eulerian- Eulerian approach was used with a k-Ɛ turbulence model. The reactor modelled had a height of 1.4 and a diameter of 0.4. The distributor was located 0.14m away from the centre of the columnbase with a diameter of 0.027m. Parameters analysed in this study included the superficial gas velocity (0.05, 0.1, 0.2 and 0.3m/s), initial solid particle concentration (10%, 30% and 50%), gasholdup, and bubble size distribution. The report concluded that increasing the inlet gas velocity led to the improvement of the axial gas holdup. Increasing the slurry concentration lead to a decrease in axial gas holdup. Additionally, the increase of the slurry concentration lead to a decrease in the bubble breakup rate. An approximate parabolic relationship was observed between the effects of the gas velocity and the axial solids holdup profile.

4. Heat Transfer

Abdulrahman [19, 33-34] studied the direct contact heat transfer of a helium – water – alumina slurry bubble column reactor using a 2D CFD simulation to ascertain the average temperature of the surly when the superficial gas velocity, static liquid height, and solid particleconcentration are varied. The inlet gas (helium) in this simulation would enter the reactor at a high temperature (90°C) and interact with the water alumina slurry (22°C). The model used was a 2D, 2 phase Eulerian-Eulerian model with a Eulerian sub model with a standard k-Ɛ turbulence model. The simulated reactor was designed with a diameter of 21.6cm. The column heights for this simulated experiment were varied between 45, 55, and 65 cm. The superficial gas velocities ranged from 0.03 to 0.15 m/s. While the solid concentration examined were 0%, 5% and 10%. Abdulrahman [18, 35-36] was able to conclude that increasing the superficial gas velocity will increase the average slurry temperature. The average slurry temperature can also be increased by decreasing the column height. Furthermore, it was observed that changing the concentration of the solid particles in the slurry has a minimal effect on the average slurry temperature. While comparing the CFD data to experimental data it was observed that the average slurry temperature was under predicted with a maximum relative error of 0.4% which shows good agreement. Overall, the CFD data was able to predict the trends that were affected by the changing the reactor height and slurry particle concentration on the average slurry temperature.

In another study, Abdulrahman [19, 34] investigated the volumetric heat transfer. A simulation was created with similar parameters as mentioned above. The simulated results showed good agreement with experimental data. Conclusions derived from this simulation revealthat when the superficial gas velocity increases so does the volumetric heat transfer. However, when the column height or the solid concentration decreases the volumetric heat transfer coefficient increases. Additionally, it was observed that approximately the same decrease in solidconcentration was also observed in the volumetric heat transfer coefficient at different superficialgas velocities.

Pu et al. [37] conducted a 2D CFD simulation of a molten salt bubble column, to investigate the hydrodynamics and direct heat transfer characteristics of a two-phase flow model (air –molten salt). The simulated bubble column had a diameter of 0.15m and height of 1.2m. AEuler- Euler multiphase model with a k-Ɛ turbulence model was used. Factors investigated during the simulation include changing the superficial gas velocity (0.05, 0.1, 0.15, 0.2 and 0.3 m/s), varying the static liquid heights (0.55, 0.61, 0.65 and 0.7m), and using different operationalpressure (1, 2, 3 and 5 bar) and inlet gas temperature (650K, 700K and 800K). Pu et al. made several observations. It was observed that as the superficial gas velocity or the operational pressure increases so does the molten salt temperature and rising rate of the molten salt temperature over time. However, when the static liquid height rises, the rate of increase in average molten salt temperature falls. Increases in superficial gas velocity or operating pressureenhance the volumetric heat transfer coefficient, whereas increases in static liquid height lowersthe volumetric heat transfer coefficient.

Zhang and Luo [38] generated a 2 phase (air-water) CFD model to investigate a bubblecolumn’s local gas-liquid slip velocity distribution in relation to heat transfer in a heterogeneous regime. The study also investigated the simulated time average of the local 2 phase slip velocitieswhen varying superficial gas velocities, axial locations and scale of the bubble column. The model used was a CFD –PBM (population balance model) simulation with an RNG k-Ɛ turbulence. The reactor size investigated follows the experimental data of Al-Dahhan and co-workers with a diameter of 0.44m and varying heights of 2.4, 3.2, 3.66 and 3.65m. The superficial gas velocities simulated include 0.05, 0.2 and 0.35 m/s. It was observed that raising the superficial gas velocities raised the local gas-liquid slip velocities in the region of developed flow. While raising the r/R lead to a decrease in the local gas-liquid slip velocities in the region of developed flow. At larger superficial gas velocities, it was notices that the slip velocities were more affected by the radial position. The slip velocities near the centre of the column where lower than that of the fully developed region. The slip velocities were minimally affected by the axial heights for the fully developed flow regions. The radial profiles of gas–liquid slip velocitiesin the sparger and fully developed flow areas, on the other hand, show clear discrepancies.

Furthermore, the radial profiles of gas–liquid slip velocities, gas velocities, and liquid velocitiesdiffer significantly. The variations between gas holdup profiles and slip velocity profiles are minor in comparison. The local heat transfer coefficient in the pilot scale bubble column raised as the local gas liquid slip velocities rose in the fully developed flow region. At superficial gas velocities of 0.05m/s and 0.35m/s the local heat transfer coefficient and local gas-liquid slip velocities demonstrated a linear relationship. The findings imply that in the fully developed region of bubble columns, there is a strong relationship between local gas–liquid slip velocities and local heat transfer coefficients.

Lie et al [39] investigated the effects of a circular heat exchanger using a 2D CFDPBM model on the hydrodynamics of a pilot scale slurry bubble column reactor (Fig. 6). A Euler –Euler multiphase model with an RNG k-Ɛ turbulence model was used. The reactor investigated had a diameter of 30cm, height of 200cm and the circular heat exchanger had a height of 108cm. Paraffin oil and catalyst particles where the assumed materials in the simulation. Variables changed in the simulation to investigate the hydrodynamics included the variation of the gas distributor’s axial position from 0.025-0.06m, the superficial gas velocity was changed within the range of 0.017-0.085 m/s, and the amount of slurry that was initially loading from (0.845 -0.900m. Several conclusions were drawn from this experiment. It was observed that the gas phase was notably distributed, local circular vertices were generated, and the slurry was strongly circulated due to the implementation of the circular heat exchanger tube.This aids in mass and interphase momentum transfer. The bimodal profile of the gas holdup profile in the radial direction is caused by the circular gas distributor's particular layout. Furthermore, the circular heat exchanger tube increases this distribution, resulting in a larger gas holdup, which facilitates momentum transfer. Moreover, it was noted that variations inthe operational parameters did not affect the gas holdup profile's bimodal structure. The local and whole-time average gas holdup is enhanced by raising the superficial gas velocity. Too muchslurry loading causes the reactor to be evacuated at increased gas velocity. While an excessive increase in the initial slurry loading volume has little influence on momentum transmission in thearea around the heat exchanger tube. Lastly it was concluded that the ideal gas distributor axial height is 0.03m for this experiment. By placing the gas distributor at this height, the momentum and mass transfer was improved.

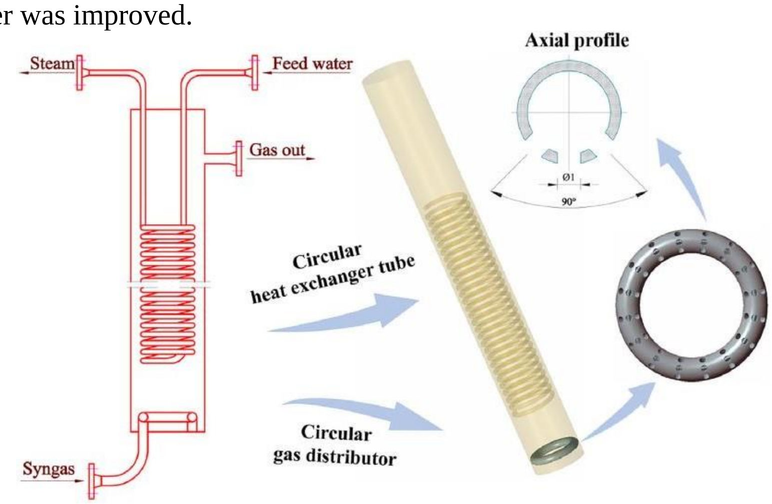

Fig. 6: Bubble slurry column reactor and gas distributor used by Li et al. (Li et al., 2020).

5. Conclusions

BCR/SBCR are advantageous tools when conducting numerous industrial processes. As such it is critical to understand factors which effect BCR/SBCR's performance. Using CFD analysis is one method to aid in the design and performance of BCR/SBCRs. For this reason, this study aimed at investigating numerus Eulerian CFD models used to study the performance of BCR/ SBCR in terms of hydrodynamics and heat transfer. This study's main findings are summarized as follows:

- Increasing the superficial gas velocity increases the average gas holdup.
- Increasing pressure within the reactor increases the average gas holdup.
- Increasing the gas phase density increase the average gas holdup.
- Higher Gas holdups was observed in the centre of the bubble column in 2D and 3D simulations.
- Increasing the solid particle concentration decreases the gas holdup and bubble breakup rate.
- The solid particle concentration in the slurry has minimal effect on the average slurry temperature.
- Increasing the superficial gas velocity increases the average slurry temperature and volumetric heat transfer.
- Decreasing the column height increases the slurry temperature.
- Decreasing the column height or solid concentration increases the volumetric heat transfer coefficient.

In future studies a focus on direct contact heat transfer in BCR/SBCR using 3D CFD methods would further aid in the improving the design and performance of the BCR/SBCR s.

References

[1] K. D. P. Nigam, and A. Schumpe, *Three-phase sparged reactors*. Amsterdam, the Netherlands: Gordon and Breach Publishers, 1996.

[2] L. S. Fan, G. Q. Yang, D. J. Lee, K. Tsuchiya, and X. Luo, "Some aspects of high-pressure phenomena of bubbles in liquids and liquid–solid suspensions," *Chemical engineering science*, vol. 54, no. 21, pp. 4681-4709, 1999.

[3] H. Meier, M. K. Silva, J.C. S. C. Bastos, R. K. Decker, and T. Matiazzo, "Investigation of Breakup and Coalescence Models for Churn-Turbulent Gas-Liquid Bubble Columns," *Journal of Applied Fluid Mechanics*, vol. 13, no. 2, pp. 737-751, 2019.

[4] H. Luo, and H. F. Svendsen, "Theoretical model for drop and bubble breakup in turbulent dispersions," *AIChE Journal*, vol. 42, no. 5, pp. 1225-1233, 1996.

[5] F. Lehr, M. Millies, M., and D. Mewes, "Bubble-size distributions and flow fields in bubble columns," *AIChE Journal*, vol. 48, no. 11, pp. 2426-2443, 2002.

[6] M. Laakkonen, V. Alopaeus, and J. Aittamaa, "Validation of bubble breakage, coalescence and mass transfer models for gas–liquid dispersion in agitated vessel," *Chemical engineering science*, vol. 61, no. 1, pp. 218-228, 2006.

[7] M. J. Prince, and H. W. Blanch, "Bubble coalescence and break-up in air-sparged bubble columns," *AIChE journal*, vol. 36, no. 10, pp. 1485-1499, 1990.

[8] H. Luo. *Coalescence, breakup and liquid circulation in bubble column reactors*. 1995.

[9] S. K. Das, "Development of a coalescence model due to turbulence for the population balance equation," *Chemical Engineering Science*, vol. 137, pp. 22-30, 2015.

[10] O. N. Manjrekar, and M. P. Dudukovic, "Application of a 4-point optical probe to a Slurry Bubble Column Reactor. *Chemical Engineering Science*, vol. 131, pp. 313-322, 2015.

[11] L. Schiller, "Über die grundlegenden Berechnungen bei der Schwerkraftaufbereitung," *Z. Vereines Deutscher Inge.*, vol. 77, pp. 318-321, 1933.

[12] E. Ertekin, J. M. Kavanagh, D. F. Fletcher, and D. D. McClure, "Validation studies to assist in the development of scale and system independent CFD models for industrial bubble columns," *Chemical Engineering Research and Design*, vol. 171, pp. 1-12, 2021.

[13] D. F. Fletcher, D. D. McClure, J. M. Kavanagh, and G. W. Barton, "CFD simulation of industrial bubble columns: Numerical challenges and model validation successes," *Applied Mathematical Modelling*, vol. 44, pp. 25-42, 2017.

[14] P. M. Raimundo, A. Cloupet, A. Cartellier, D. Beneventi, and F. Augier, "Hydrodynamics and scale-up of bubble columns in the heterogeneous regime: Comparison of bubble size, gas holdup and liquid velocity measured in 4 bubble columns from 0.15 m to 3 m in diameter," *Chemical Engineering Science*, vol. 198, pp. 52-61, 2019.

[15] D. D. McClure, J. Deligny, J. M. Kavanagh, D. F. Fletcher, and G. W. Barton, "Impact of surfactant chemistry on bubble column systems," *Chemical Engineering & Technology*, vol. 37, no. 4, pp. 652-658, 2014.

[16] P. Yan, H. Jin, G. He, X. Guo, L. Ma, S. Yang, and R. Zhang, "CFD simulation of hydrodynamics in a high-pressure bubble column using three optimized drag models of bubble swarm," *Chemical Engineering Science*, vol. 199, pp. 137-155, 2014.

[17] A. R. Sarhan, J. Naser, and G. Brooks, "CFD modeling of bubble column: Influence of physico-chemical properties of the gas/liquid phases properties on bubble formation," *Separation and Purification Technology*, vol. 201, pp. 130-138, 2018.

[18] M. W. Abdulrahman, "Analysis of the thermal hydraulics of a multiphase oxygen production reactor in the Cu-Cl cycle*"* Ph.D. dissertation, University of Ontario Institute of Technology, Canada.

[19] M. W. Abdulrahman, "Experimental studies of direct contact heat transfer in a slurry bubble column at high gas temperature of a helium–water–alumina system," *Applied Thermal Engineering*, vol. 91, pp. 515-524, 2015.

[20] M. W. Abdulrahman, "Experimental studies of the transition velocity in a slurry bubble column at high gas temperature of a helium–water–alumina system," *Experimental Thermal and Fluid Science*, vol. 74, pp. 404-410, 2016.

[21] M. W. Abdulrahman, "Experimental studies of gas holdup in a slurry bubble column at high gas temperature of a helium− water− alumina system," *Chemical Engineering Research and Design*, vol. 109, pp. 486-494, 2016.

[22] S. Adam, and K. Tuwaechi, "Hydraulic Simulation of Bubble Flow in Bubble Column Reactor," *Journal of Complex Flow*, vol. 1, no. 2, 2019.
[23] Z. Li, X. Guan, L. Wang, Y. Cheng, and X. Li, "Experimental and numerical investigations of scale-up effects on the hydrodynamics of slurry bubble columns," *Chinese journal of chemical engineering*, vol. 24, no. 8, pp. 963-971, 2016.
[24] M. Pourtousi, P. Ganesan, and J. N. Sahu, "Effect of bubble diameter size on prediction of flow pattern in Euler–Euler simulation of homogeneous bubble column regime," *Measurement*, vol. 76, pp. 255-270, 2015.
[25] W. Li, and W. Zhong, "CFD simulation of hydrodynamics of gas–liquid–solid three-phase bubble column," *Powder Technology*, vol. 286, pp. 766-788, 2015.
[26] M. T. Dhotre, B. Niceno, and B. L. Smith, "Large eddy simulation of a bubble column using dynamic sub-grid scale model," *Chemical Engineering Journal*, vol. 136, no. 2-3, pp. 337-348, 2008.
[27] N. G. Deen, B. H. Hjertager, and T. Solberg, "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*, 2000.
[28] M. W. Abdulrahman, "Effect of Solid Particles on Gas Holdup in a Slurry Bubble Column," in *Proceedings of the 7th World Congress on Mechanical, Chemical, and Material Engineering (MCM'20)*, 2020.
[29] R. Zhou, N. Yang, and J. Li, "A conceptual model for analyzing particle effects on gas-liquid flows in slurry bubble columns," *Powder Technology*, vol. 365, pp. 28-38, 2020.
[30] M. W. Abdulrahman, "CFD Simulations of Gas Holdup in a Bubble Column at High Gas Temperature of a Helium-Water System," in *Proceedings of the 7th World Congress on Mechanical, Chemical, and Material Engineering (MCM'20)*, 2020.
[31] J. Chen, and C. S. Brooks, "Experiments and CFD simulation of mass transfer and hydrodynamics in a cylindrical bubble column," *Chemical Engineering Science*, vol. 234, p. 116435, 2021.
[32] A. Wodołażski, "CFD Hydrodynamics Analysis of Syngas Flow in Slurry Bubble Column," *Journal of Oil, Gas and Petrochemical Technology*, vol. 3, no. 1, pp. 83-96, 2016.
[33] M. W. Abdulrahman, "Direct contact heat transfer in the thermolysis reactor of hydrogen production cu-cl cycle" U.S. Patent 10 059 586, August 18, 2018.
[34] M. W. Abdulrahman, "CFD simulations of direct contact volumetric heat transfer coefficient in a slurry bubble column at a high gas temperature of a helium–water–alumina system," *Applied thermal engineering*, vol. 99, pp. 224-234, 2016.
[35] M. W. Abdulrahman, "CFD Analysis of Temperature Distributions in a Slurry Bubble Column with Direct Contact Heat Transfer," in *Proceedings of the 3rd International Conference on Fluid Flow, Heat and Mass Transfer (FFHMT'16)*, 2016.
[36] M. W. Abdulrahman, "Temperature profiles of a direct contact heat transfer in a slurry bubble column," *Chemical Engineering Research and Design*, Vol. 182, pp. 183-193, 2022.
[37] W. Pu, N. Yang, C. Yue, S. Bai, and Y. Chen, "Simulation on direct contact heat transfer in gas-molten salt bubble column for high temperature solar thermal storage," *International Communications in Heat and Mass Transfer*, vol. 104, pp. 51-59, 2019.
[38] X. B. Zhang, and Z. H. Luo, "Local gas–liquid slip velocity distribution in bubble columns and its relationship with heat transfer," *AIChE Journal*, vol. 67, no. 1, e17032, 2021.
[39] L. Li, W. Lian, B. Bai, Y. Zhao, P. Li, Q. Zhang, and W. Huang, "CFD-PBM investigation of the hydrodynamics in a slurry bubble column reactor with a circular gas distributor and heat exchanger tube," *Chemical Engineering Science: X*, vol. *9*, p. 100087, 2021.