CFD Development for Multiphase Flows in HZDR

Ryan Anugrah Putra[1,2*], Dirk Lucas[1]
[1]Hemholtz-Zentrum Dresden-Rossendorf (HZDR), Institute of Fluid Dynamics,
Bautzner Landstr.400, 01328 Dresden, Germany
[2]Department of Mechanical & Industrial Engineering, Faculty of Engineering, Gadjah Mada University, Jalan Grafika No. 2, 55281 Yogyakarta, Indonesia
r.putra@hzdr.de (Ryan Anugrah Putra)

Abstract

Over the last 20 years, Helmholtz-Zentrum Dresden-Rossendorf (HZDR) has contributed in the development of Computational Fluid Dynamics (CFD) for multiphase flows. In the CFD department, a lot of efforts have been performed to provide CFD as a reliable simulation tool for the design, optimization and safety analyses of medium and large scale applications. The tool is expected to contribute in the improvement of the efficient use of energy and resources as well as safety operation. Most of the research activities are related to gas-liquid flows covering dispersed flows, stratified flows and mixture or transition flows. The development is carried out using Euler-Euler two or multi fluid model as the basis. The paper discussed some characteristics of the CFD model development in HZDR started with "simple", reliable experimental data for validation and stepwise and continuous improvement. The examples of each characteristic were given. The discussions covered three CFD modeling frames which had been developed in HZDR; they were iMUSIG for dispersed flows, AIAD for stratified flows and GENTOP concept for mixture or transitional flows.

Keywords: CFD, HZDR, Multiphase flows, Dispersed flows, Stratified flows

1. Introduction

Multiphase flows occur in nature as well as in many industrial processes (e.g., power plants, chemical plants, oil and gas industries, etc). Rapid development in the computer resources leads to the use of CFD in various fields. CFD seems to be an appropriate alternative to physical experiments in terms of financial cost, time and the transferability to modified geometries, flow condition, or scales. Reliable and predictive CFD is beneficial for the design of industrial facilities, the optimization of processes and the safety analyses. In order to achieve that level, the development of CFD model has been carried out over the last 20 years in the CFD-department of HZDR. Considering large scale applications, Euler-Euler two or multi fluid models is used as the basis of the development. Since this model heavily relies on closure models, one of the focuses in the development activities is the improvement of the closure models. Most of the research activities are related with gas-liquid flows. Based on the size of interfacial scales in the relation of cell size used in CFD, the flows can be divided into two different morphologies: dispersed and segregated flows. In the case of dispersed flows, in which the interfacial scales are smaller or similar to computational cell, the inhomogeneous Multiple Size Group (iMUSIG) model has been developed in HZDR in co-operation with ANSYS and validated over various cases. In the stratified flows, which are characterized with the interfacial scales larger than the cell size, the Algebraic Interfacial Area Density (AIAD) model has also been developed and validated. In many flows, both morphologies occur simultaneously and the transition between them exists. To handle such situation, the Generalize Two Phase (GENTOP) concept has been developed.

The discussion in this paper would be presented in section two to four–, covering some important characteristics of CFD model development in HZDR. Section 2 will discuss the principal of start with "simple" for the early stages of model development. Some examples would be given in modeling of dispersed flows and stratified flows. The next section described the efforts to provide high quality experimental data for CFD model validation. Section 4 gave examples on the stepwise and continuous improvement for the three modeling frames: iMUSIG, AIAD and GENTOP. Finally, this paper was closed with some conclusions which would be given in section 5.

2. Start with "Simple"

The development of CFD models either for dispersed flows or stratified flows were always started with simple models or set-up. This fact could be observed from the early publications of the CFD works for those particular topics. From these simulations, then the limitation of the model and the possible improvement could be identified. As an addition, the experimental data required for the next validation of the CFD model could also be known. For the case of dispersed flows, the first attempt of CFD simulation was carried out using Euler-Euler two fluid approaches considering a uniform size of dispersed phase under CFX-4.2 code (Krepper & Prasser, 1999). The simulation was based on the experiment of co-current upward gas-liquid flow in a vertical pipe conducted at HZDR. The test section had inner diameter of 51.2 mm and 4 m in height. The detail information of the void distribution was given from the measurement using a wire mesh sensor. The detail description of this measurement technique could be found in a research conducted by Prasser, Böttger, and Zschau (1998). The non-drag forces including lift force, turbulent dispersion force and wall lubrication force were also considered in the calculation. The classical lift force formulation (Ẑun, 1980) which was induced by shear and acts toward the wall was used. The lift coefficient was set at a constant value of 0.05.

From this work, it could be known that there were some "simple" models or set-ups that were used; the assumption of a uniform bubble size and the use of classical lift force without considering the change of sign of the lift coefficient according to bubble size was as proposed by Tomiyama (Tomiyama, 1998). However, some lessons could be learned from the work. The results of simulation which were able to predict the wall peak void distribution but was not able to well describe the center peak profile showed the influence of this "simple" selection of model or set-up. The use of uniform bubble size and the use of classical lift force formulation were no longer appropriate to describe the evolution of flow pattern in the pipe.

The lesson learned from above opened the way for other works such as the research conducted by Lucas et al. (Lucas, Krepper, & Prasser, 2001). In order to elucidate the influence of the bubble size distribution on radial gas fraction profiles, the calculation using 1D model based on the assumption of the non-drag forces equilibrium was performed (Lucas et al., 2001). In this work, the lift force model as proposed by Tomiyama et al. (Tomiyama, Zun, Tamai, Shimomura, & Hosokawa, 1999) was used. The model not only did consider the shear-induced lift force but also include the force caused by the interaction between the wake and the shear field. Both forces were summarized in the net transverse lift force in which the lift coefficient changed its sign at bubble diameter of 5.8 mm for the water-air system at normal condition. According to this model, bubble with a diameter less than 5.8 mm would experience the net transverse lift force which acted toward the wall while for larger bubbles the force acted toward the pipe center (Lucas et al., 2001).

From the two works described above, one could learn that the limitation identified in the early works was useful to obtain a better model or set-up selection as represented in the second work. In addition to the use of 1D model at that time, it could be viewed as a simple solution for investigating a wide range of bubble size distribution with less computational efforts in comparison with 3D CFD. However, further research focused more in 3D CFD since the nature of the flow is 3D.

The start with "simple" principal was also reflected from the early stage of modeling separated flows in HZDR. This could be observed from the CFD simulation of stratified air/water co-current flow (slug flow) using two-fluid model with the free surface option under CFX-5 code (Vallée, Höhne, Prasser, & Sühnel, 2005). The CFD work was based on the experiment in a horizontal channel with rectangular cross section conducted at HZDR. A constant drag coefficient of 0.44 was used which meant that the dependence of drag coefficient on the fluid morphology was not considered. The k-ω based SST model was selected for each phase without the consideration of damping of turbulent diffusion at the interface. The results well described the position as well as the velocity of the slug flow propagation in the channel. The significant discrepancy to the experiment was observed for the pressure peak behind the slug in which the CFD results over predicted the experimental data which were attributed to the limitation in defining boundary conditions. This discrepancy led to an effort to provide experimental data suitable for CFD validation. To improve this situation, HAWAC (The Horizontal Air/Water Channel) experimental facility was built in HZDR (see Figure 1). This new experimental facility provided well-defined inlet boundary conditions (e.g., homogeneous velocity profiles at the inlet of the test section) for the CFD model; thus, a good CFD validation could be expected (Vallée, Höhne, Prasser, & Sühnel, 2007). From this case, one could learn that even though the simulation was performed with some "simple" selection, constant drag coefficient and the non-consideration of turbulence damping however it was beneficial for identifying the required experimental data which was appropriate for CFD validation. This experimental data further were used to validate AIAD model and turbulence damping.

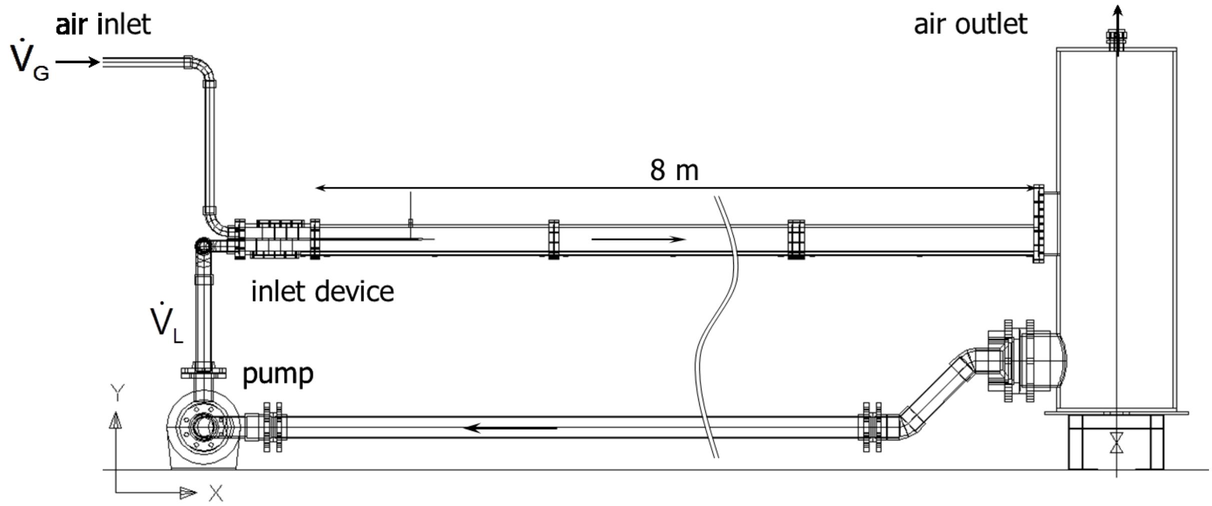

Figure 1. Schematic view of HAWAC experimental facility (Vallée et al., 2007)

3. Reliable experimental data for validation

One of the important factors in the successful development of CFD model in HZDR was the reliable experimental data for the validation. Continuous improvement to assure high quality data was conducted. In the case of gas-liquid dispersed flows, the efforts to obtain the high quality data was reflected in the example of TOPFLOW experimental facility (see Figure 2). The use of so-called variable gas injection was a new idea at that time. Usually, the common measuring techniques were based on needle probes which might influence on the flow downstream the measuring position (Lucas, Beyer, Szalinski, & Schütz, 2010). This made single measurements for a given set of boundary conditions with varying L/D could not be performed since the facility should be reassembled for shifting the measurement techniques. The drawbacks from this limitation were the information of the flow evolution that was obtained from the separate experimental run would lead to the additional discrepancies in the data. The example was bubble size distributions which were very sensitive to the variation in boundary conditions and fluid properties. To overcome these problems, in the case of TOPFLOW facility, the measurement plane was always located at the upper end of the pipe while the gas was injected through orifices in the pipe wall at various distances from this measurement plane. The reassembling of the facility to shift the measurement plane was not necessary since it could be done by just switching between the gas injection devices (Lucas, Beyer, Szalinski, et al., 2010).

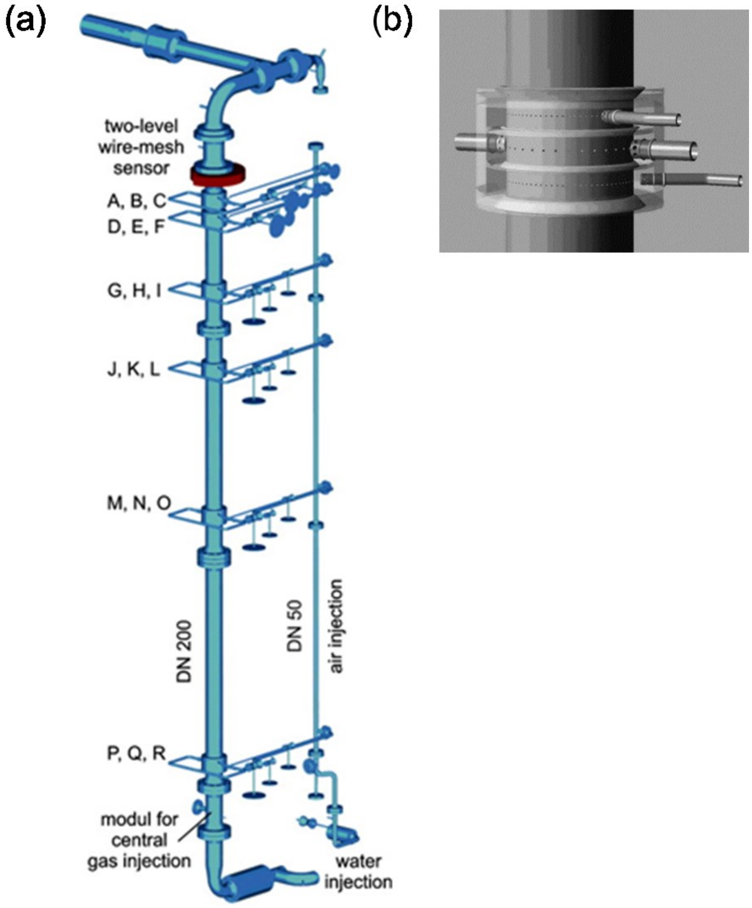

Figure 2. Schematic view of: (a) TOPFLOW test section, (b) gas injection device (Lucas, Beyer, Kussin, & Schütz, 2010)

In contrast to previous test series where the pressure at the location of gas injection was varied with varying L/D due to the hydrostatic pressure, the new TOPFLOW facility maintained the constant pressure. Besides providing suitable boundary conditions for CFD model validation, maintaining a constant pressure was important to avoid the contribution from the variation of the pressure on the evolution of bubble size distribution. The influence of the pressure was larger than the coalescence and break-up process for the flow with relatively low void fractions. Another improvement was the nearly constant water temperature of 30 °C that was maintained in the new facility, contrasting to the previous experiments where the water temperature was varied between 20 °C and 37 °C during measurement series. The importance of maintaining the temperature was the fact that coalescence rate and break-up frequency was sensitive to temperature variation due to changes in surface tension(Lucas, Beyer, Kussin, et al., 2010).

In the case of stratified flows, HAWAC experimental facility, which had been discussed in the previous section, was an example how the improvement was made to provide suitable data for CFD model validation. If HAWAC provided the data for co-current air water stratified flows, the next experiment should provide the data for counter-current flows. To reach the goal, the experimental facility for Counter-current flow limitation (CCFL) in a model of the hot-leg of a pressurized water reactor (PWR) was built and operated in HZDR (see Figure 3). The detail description of the experiments as well as data processing were well documented in (Deendarlianto, Höhne, Lucas, & Vierow, 2012; Deendarlianto et al., 2008; Deendarlianto, Vallée, et al., 2011; Vallée et al., 2010; Vallée et al., 2012; Vallée, Seidel, Lucas, Tomiyama, & Murase, 2011). The data obtained from these CCFL experiments were useful in providing more quantitative validation of AIAD modeling frames.

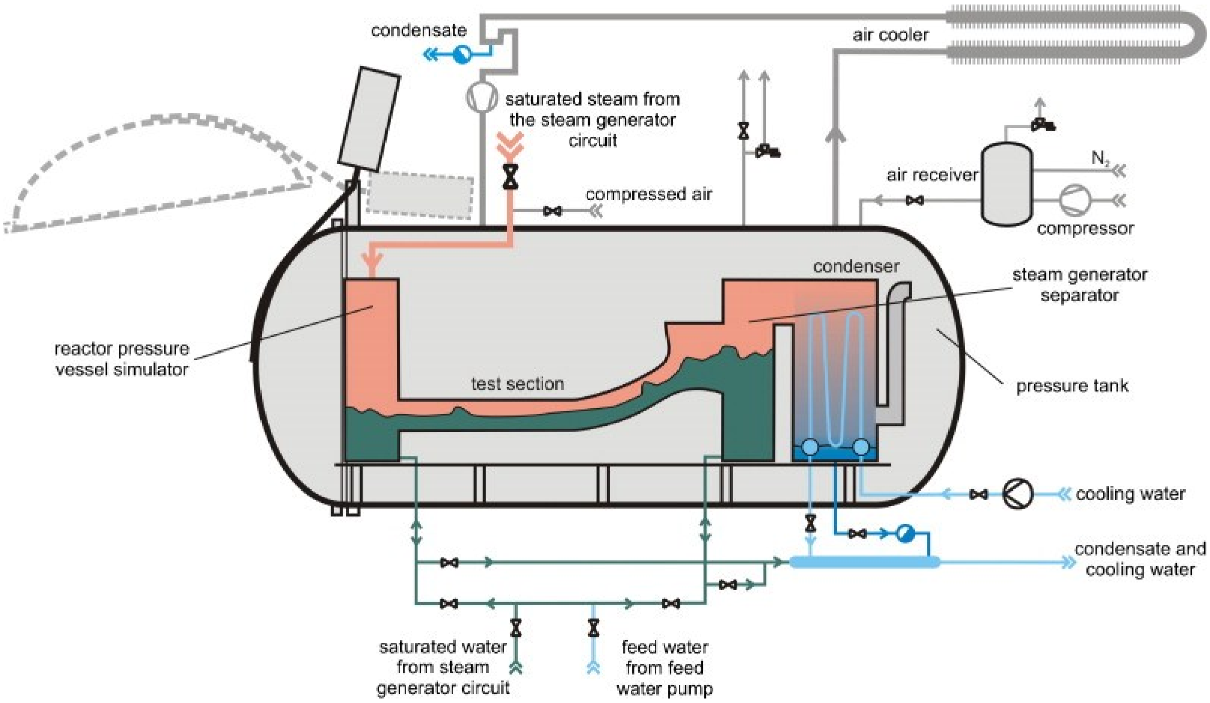

Figure 3. Schematic diagram of CCFL experimental facility in HZDR (Deendarlianto et al., 2008)

4. Stepwise and continuous improvement

For gas-liquid dispersed flows, after the early stages that used a ”simple” model for the CFD work (see Section 2) stepwise and continuous efforts were carried out to improve the identified limitations. The improvement were made by considering bubble size distribution, bubble break-up and coalescence models and also the use of lift force formulation as proposed by Tomiyama et al., (1999), which could also be found in (Krepper, Lucas, & Prasser, 2005). This work identified the limitation of existing approach for modeling poly-dispersed flow. Multiple bubble size groups (MUSIG) approach of Lo (1996) was the only available option in ANSYS CFX code for modeling poly-dispersed flow at that time. The limitation came from the fact that all bubble size groups within MUSIG frames shared a single velocity field which was not suitable to describe the evolution of the flow. Therefore, extending the models to ensure that the dispersed phase had more than one velocity field was proposed (Shi, Krepper, Lucas, & Rohde, 2003). Through good collaboration with ANSYS CFX then inhomogeneous Multiple Size Group (iMUSIG) which allowed the dispersed phases to have multiple velocity groups was implemented in the ANSYS CFX-Code (Frank, Zwart, Krepper, Prasser, & Lucas, 2008; Krepper, Lucas, Frank, Prasser, & Zwart, 2008). The results showed the radial void fraction profiles in the experiments could be well reproduced. Those works also identified the weakness of the available break-up and coalescence models.

In order to improve the break-up and coalescence models, the extensive literature studies were performed (see (Liao & Lucas, 2009, 2010). Based on those studies, a generalized break-up and coalescence models were proposed (Liao, Lucas, Krepper, & Schmidtke, 2011). The validation of these new models showed a better prediction in bubble size distribution in comparison to the standard models (i.e., the models of (Luo & Svendsen, 1996; Prince & Blanch, 1990).

In the case of stratified flows, the first improvement made after the first CFD simulation described in section 2 was the use of turbulence damping. CFD work was performed based on HAWAC experiment (Valle, Hohne, Prasser & Suehnel, 2008). A simple grid dependent symmetric damping procedure proposed by Yegorov (Yegorov, 2004) which provided the solid wall-like damping of turbulence in both gas and liquid phases was applied. Qualitative comparison to the experimental results showed that the generation and propagation of the slug were well reproduced by the CFD simulation. However the stratified flow formed after the slug was too smooth and the time required for the process to form the next slug from the small waves after the previous slug was significantly longer in comparison to the experiment (Vallée, Höhne, Prasser, & Sühnel, 2008).

In order to have a better physical model, Algebraic Interfacial Area Density (AIAD) model was introduced into the two-fluid Euler-Euler simulation (Höhne & Vallée, 2010) allowing the detection of fluid morphology on either bubble, droplet or free surface. The drag coefficient and the area density were then calculated according to the detected morphology. The switch from one to another correlation was realized by the blending function. The new formulation to calculate drag coefficient for free surface was proposed. Next, more validations of AIAD frames were performed for CCFL cases. A good quantitative agreement was obtained for the CCFL characteristics between the CFD simulation and the experimental data. As an addition, the water level inside the hot leg channel was also in a quantitatively good agreement with the experiment (Deendarlianto, Höhne, Lucas, Vallée, & Zabala, 2011). Hoehne et al. in (Höhne, Deendarlianto, & Lucas, 2011) added more quantitative comparison in the term of flooding curve and showed the ability of the model to describe typical flow processes of the CCFL.

The improvement in modeling turbulence for stratified flows was then further processed with the introduction of sub-grid wave turbulence (SWT) model into AIAD frames (Höhne, 2013; Höhne & Mehlhoop, 2014). The aim was to consider waves created by Kelvin Helmholtz instability which were smaller than the grid size (Höhne, 2013). Another improvement which was further performed was the implementation of droplet entrainment model into AIAD frame. The validation was conducted for annular flow and the results showed that some important phenomena could be calculated (Höhne, Geissler, Bieberle, & Hampel, 2015). The recent progress in AIAD frames was the modification of free surface drag model and morphology detection mechanism (Höhne et al., 2015). The revised detection mechanisms led to a better result for detecting sharp interface while the modified free surface drag model gave a good quantitative agreement to the experiment. As an addition, the work showed the importance of turbulence damping.

The improvements of the modeling frames did not only stop with iMUSIG and AIAD frames. The question was then raise on how to deal the flow where both morphologies, bubbly flows and stratified flows, occurred and there was a transition between them as in the case of impinging jet where the liquid jet flows towards the stagnant liquid and further led to the entrainment of continuous gas, broke into bubble with different sizes and then some bubbles went up due to buoyancy and after that some bubble coalesced and went back to the continuous gas in the above the liquid surface (Hänsch, Lucas, Krepper, & Höhne, 2012).

The experience in developing iMUSIG and AIAD led to a genuine idea to combine iMUSIG and AIAD in the unified concept so-called as GENTOP (generalized two-phase flow) concept. The concept was promising for handling the flow where the multi-scale interfacial structures existed as described in the two demonstration cases: impinging jet and bubble column (Hänsch et al., 2012). Along the time, some improvements were made for this new concept. The SWT model which was previously implemented in AIAD frames was implemented into GENTOP concept and validated for the dram-break case with an obstacle The results showed that the CFD simulation was able to describe the main flow characteristics (Hänsch, Lucas, Höhne, & Krepper, 2014). The Continuum surface tension model proposed by Brackbill et al. (Brackbill, Kothe, & Zemach, 1992) was also implemented into GENTOP framework (G. Montoya, Baglietto, & Lucas, 2015). The quantitative comparison was carried out for churn-turbulent flows (G. A. Montoya, 2015). In order to describe the detachment of bubbles from continuous gas then the entrainment model proposed by Ma et al. (Ma, Oberai, Drew, Lahey Jr, & Hyman, 2011) was also introduced (Krepper, Zidouni, & Lucas, 2015). Since GENTOP concept was the newest modeling frames, more validation and improvements were required, in which some parts of them were on-going in HZDR.

5. Conclusion

There were some points that could be concluded from the experience of HZDR in developing CFD models especially for the three 3 modeling frames: iMUSIG, AIAD and GENTOP. The early stages of development were always started with some simple available set-up or models. The limitations of the available models or the further required experimental data were recognized from these early works. The further efforts then could be directed to overcome the limitations. The reliable experimental data with high quality were the backbone for the successful validation of the CFD models. A lot of efforts were made to achieve CFD-grade experimental data. Finally, a stepwise and continuous improvement was required to achieve a better CFD models.

References

Brackbill, J. U., Kothe, D. B., & Zemach, C. (1992). A continuum method for modeling surface tension. *Journal of Computational Physics, 100*(2), 335-354. doi:http://dx.doi.org/10.1016/0021-9991(92)90240-Y

Deendarlianto, Höhne, T., Lucas, D., Vallée, C., & Zabala, G. A. M. (2011). CFD studies on the phenomena around counter-current flow limitations of gas/liquid two-phase flow in a model of a PWR hot leg. *Nuclear Engineering and Design, 241*(12), 5138-5148. doi:http://dx.doi.org/10.1016/j.nucengdes.2011.08.071

Deendarlianto, Höhne, T., Lucas, D., & Vierow, K. (2012). Gas-liquid countercurrent two-phase flow in a PWR hot leg: A comprehensive research review. *Nuclear Engineering and Design, 243*, 214-233. doi:10.1016/j.nucengdes.2011.11.015

Deendarlianto, Vallée, C., Lucas, D., Beyer, M., Pietruske, H., & Carl, H. (2008). Experimental study on the air/water counter-current flow limitation in a model of the hot leg of a pressurized water reactor. *Nuclear Engineering and Design, 238*(12), 3389-3402. doi:http://dx.doi.org/10.1016/j.nucengdes.2008.08.003

Deendarlianto, Vallée, C., Lucas, D., Beyer, M., Pietruske, H., & Carl, H. (2011). Erratum: Experimental study on the air/water counter-current flow limitation in a model of the hot leg of a pressurized water reactor (Nuclear Engineering and Design). *Nuclear Engineering and Design, 241*(8), 3359-3372. doi:10.1016/j.nucengdes.2011.02.028

Frank, T., Zwart, P. J., Krepper, E., Prasser, H. M., & Lucas, D. (2008). Validation of CFD models for mono- and polydisperse air–water two-phase flows in pipes. *Nuclear Engineering and Design, 238*(3), 647-659. doi:http://dx.doi.org/10.1016/j.nucengdes.2007.02.056

Hänsch, S., Lucas, D., Höhne, T., & Krepper, E. (2014). Application of a new concept for multi-scale interfacial structures to the dam-break case with an obstacle. *Nuclear Engineering and Design, 279*, 171-181. doi:http://dx.doi.org/10.1016/j.nucengdes.2014.02.006

Hänsch, S., Lucas, D., Krepper, E., & Höhne, T. (2012). A multi-field two-fluid concept for transitions between different scales of interfacial structures. *International Journal of Multiphase Flow, 47*, 171-182. doi:http://dx.doi.org/10.1016/j.ijmultiphaseflow.2012.07.007

Höhne, T. (2013). *Modelling and validation of turbulence parameters at the interface of horizontal multiphase flows*. Paper presented at the The 8th international conference multiphase flow, ICMF, Jeju, Korea.

Höhne, T., Deendarlianto, & Lucas, D. (2011). Numerical simulations of counter-current two-phase flow experiments in a PWR hot leg model using an interfacial area density model. *International Journal of Heat and Fluid Flow, 32*(5), 1047-1056. doi:http://dx.doi.org/10.1016/j.ijheatfluidflow.2011.05.007

Höhne, T., Geissler, T., Bieberle, A., & Hampel, U. (2015). Numerical modeling of a horizontal annular flow experiment using a droplet entrainment model. *Annals of Nuclear Energy, 77*, 351-360. doi:http://dx.doi.org/10.1016/j.anucene.2014.11.041

Höhne, T., & Mehlhoop, J.-P. (2014). Validation of closure models for interfacial drag and turbulence in numerical simulations of horizontal stratified gas–liquid flows. *International Journal of Multiphase Flow, 62*, 1-16. doi:http://dx.doi.org/10.1016/j.ijmultiphaseflow.2014.01.012

Höhne, T., & Vallée, C. (2010). Experiments and numerical simulations of horizontal two-phase flow regimes using an interfacial area density model. *The Journal of Computational Multiphase Flows, 2*(3), 13

Krepper, E., Lucas, D., Frank, T., Prasser, H.-M., & Zwart, P. J. (2008). The inhomogeneous MUSIG model for the simulation of polydispersed flows. *Nuclear Engineering and Design, 238*(7), 1690-1702. doi:http://dx.doi.org/10.1016/j.nucengdes.2008.01.004

Krepper, E., Lucas, D., & Prasser, H.-M. (2005). On the modelling of bubbly flow in vertical pipes. *Nuclear Engineering and Design, 235*(5), 597-611. doi:http://dx.doi.org/10.1016/j.nucengdes.2004.09.006

Krepper, E., & Prasser, H. M. (1999). *Measurements and CFX-simulations of a bubbly flow in a vertical pipe*. Paper presented at the CFX International Users Conference, Friedrichshafen.

Krepper, E., Zidouni, F., & Lucas, D. (2015). *Analysis and applications of a generalized multi-field two fluid approach for plunging jet configuration*. Paper presented at the NURETH-16, Chicago, IL.

Liao, Y., & Lucas, D. (2009). A literature review of theoretical models for drop and bubble breakup in turbulent dispersions. *Chemical Engineering Science, 64*(15), 3389-3406. doi:http://dx.doi.org/10.1016/j.ces.2009.04.026

Liao, Y., & Lucas, D. (2010). A literature review on mechanisms and models for the coalescence process of fluid particles. *Chemical Engineering Science, 65*(10), 2851-2864. doi:http://dx.doi.org/10.1016/j.ces.2010.02.020

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

Lo, S. M. (1996). *Application of population Balance to CFD modeling of bubbly flow via the MUSIG model* (AEAT–1096). Retrieved from

Lucas, D., Beyer, M., Kussin, J., & Schütz, P. (2010). Benchmark database on the evolution of two-phase flows in a vertical pipe. *Nuclear Engineering and Design, 240*(9), 2338-2346. doi:http://dx.doi.org/10.1016/j.nucengdes.2009.11.010

Lucas, D., Beyer, M., Szalinski, L., & Schütz, P. (2010). A new database on the evolution of air–water flows along a large vertical pipe. *International Journal of Thermal Sciences, 49*(4), 664-674. doi:http://dx.doi.org/10.1016/j.ijthermalsci.2009.11.008

Lucas, D., Krepper, E., & Prasser, H.-M. (2001). Prediction of radial gas profiles in vertical pipe flow on the basis of bubble size distribution. *International Journal of Thermal Sciences, 40*(3), 217-225. doi:http://dx.doi.org/10.1016/S1290-0729(00)01211-4

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

Ma, J., Oberai, A. A., Drew, D. A., Lahey Jr, R. T., & Hyman, M. C. (2011). A comprehensive sub-grid air entrainment model for RANS modeling of free surface bubbly flows. *Journal of Computational Multiphase Flows, 3*, 16.

Montoya, G., Baglietto, E., & Lucas, D. (2015). *Implementation and validation of a surface tension model for the multi-scale approach GENTOP*. Paper presented at the NURETH-16, Chicago, IL.

Montoya, G. A. (2015). Development and validation of advanced theoretical modeling for churn-turbulent flows and subsequent transitions. *Wissenschaftlich-Technische Berichte, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), HZDR-063*.

Prasser, H. M., Böttger, A., & Zschau, J. (1998). A new electrode-mesh tomograph for gas–liquid flows. *Flow Measurement and Instrumentation, 9*(2), 111-119. doi:http://dx.doi.org/10.1016/S0955-5986(98)00015-6

Prince, M. J., & Blanch, H. W. (1990). Bubble coalescence and break-up in air-sparged bubble columns. *AIChE Journal, 36*(10), 1485-1499. doi:10.1002/aic.690361004

Shi, J. M., Krepper, E., Lucas, D., & Rohde, U. (2003). Some concepts for improving the MUSIG model, internal note.

Tomiyama, A. (1998). *Struggle with computational bubble dynamics*. Paper presented at the Third International Conference on Multiphase Flow, ICMF, Lyon, France.

Tomiyama, A., Zun, I., Tamai, H., Shimomura, H., & Hosokawa, S. (1999). *Spatial evolution of developing air–water bubble flow in a vertical pipe.* Paper presented at the 2nd Int. Symp. on Two-phase Flow Modelling and Experimentation, Pisa.

Vallée, C., Höhne, T., Prasser, H.-M., & Sühnel, T. (2008). Experimental investigation and CFD simulation of horizontal stratified two-phase flow phenomena. *Nuclear Engineering and Design, 238*(3), 637-646. doi:http://dx.doi.org/10.1016/j.nucengdes.2007.02.051

Vallée, C., Höhne, T., Prasser, H. M., & Sühnel, T. (2005). *Experimental modelling and cfd simulation of air/water flow in a horizontal channel*. Paper presented at The 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France.

Vallée, C., Höhne, T., Prasser, H. M., & Sühnel, T. (2007). Experimental investigation and CFD simulation of slug flow in horizontal channels. *Wissenschaftlich-Technische Berichte, Forschungszentrum Dresden-Rossendorf e.V., Institut für Sicherheitsforschung*(FZD-485).

Vallée, C., Lucas, D., Beyer, M., Pietruske, H., Schütz, P., & Carl, H. (2010). Experimental CFD grade data for stratified two-phase flows. *Nuclear Engineering and Design, 240*(9), 2347-2356. doi:http://dx.doi.org/10.1016/j.nucengdes.2009.11.011

Vallée, C., Seidel, T., Lucas, D., Beyer, M., Prasser, H. M., Pietruske, H., . . . Carl, H. (2012). Counter-current flow limitation in a model of the hot leg of a PWR - Comparison between air/water and steam/water experiments. *Nuclear Engineering and Design, 245*, 113-124. doi:10.1016/j.nucengdes.2012.01.001

Vallée, C., Seidel, T., Lucas, D., Tomiyama, A., & Murase, M. (2011). Comparison of countercurrent flow limitation experiments performed in two different models of the hot leg of a pressurized water reactor with rectangular cross section. *Journal of Engineering for Gas Turbines and Power, 133*(5). doi:10.1115/1.4002884

Yegorov, Y. (2004). *Contact condensation in stratified steam-water flow.* (EVOLECORA-D 07)

Ẑun, I. (1980). The transverse migration of bubbles influenced by walls in vertical bubbly flow. *International Journal of Multiphase Flow, 6*(6), 583-588. doi:http://dx.doi.org/10.1016/0301-9322(80)90053-1