Numerical simulation of multiphase flow in a vertically mounted Venturi flow meter

A. Fiebach[1], E. Schmeyer[1], S. Knotek[2], S. Schmelter[1]

[1] Physikalisch-Technische Bundesanstalt (PTB), Abbestr. 2-12, 10587 Berlin, Germany
[2] Czech Metrology Institute (CMI), Okružní 31, 638 00 Brno, Czech Republic
E-mail (corresponding author): andre.fiebach@ptb.de

Abstract

The objective of the European research project "Multiphase flow metrology in Oil and Gas production" (MultiFlowMet) is to explain and reduce the uncertainty in multiphase flow metering. A comparison of measurements using multiphase metering systems with corresponding results from computational fluid dynamics is used to achieve this goal. In this contribution, a two-phase flow through a vertically mounted Venturi flow meter is presented. According to the experimental set-up within the project, the simulations focus on the flow of oil and gas in large pipes of diameter $D = 0.104\,\mathrm{m}$. The simulations represent the whole configuration including a $154D$ long inflow section, where the flow pattern is formed. After a horizontal bend the flow is turned upwards through a blind-T in the vertically mounted measurement unit. This leads to swirl and turbulence and thus another pattern is observed in the Venturi flow meter. The simulation results are compared with experimental data obtained in the project. Furthermore, the main features of the flow have been extracted by means of data analysis. This allows an easier recognition of the flow pattern and is the basis for a quantitative comparison between experimental and numerical results.

1. Introduction

Energy from oil and gas still plays an important role in Europe for the near or mid-term future. Since the number of large reserves is decreasing, one is interested in exploiting new smaller deposits which are more remote and situated in deeper water. New wells are operated and measured under seabed, before a mixture of oil, water and gas is flowing in shared pipelines to the processing facility. Nowadays multiphase flow measurement systems have a high level of uncertainty of up to $20\,\%$. The aim of the European research project "Multiphase flow metrology in Oil and Gas production" (MultiFlowMet) is to improve these levels of uncertainty and to form the basis of a sustainable reference network, reducing financial exposure and risk for industry. To achieve this goal a comprehensive intercomparison on multiphase flow is conducted on one hand. On the other hand the influence of different parameters is studied by computational fluid dynamics (CFD). As an initial step, the flow of several phases through the measurement configuration used for the experimental intercomparison within the project is simulated with the commercial CFD solver ANSYS Fluent [1].

There are several papers concerning the CFD modeling of three-dimensional gas-liquid multiphase flows, see [2, 4, 5, 10, 11, 12] and the references therein. In [4, 5] the numerical simulation of slug flow in horizontal pipes is studied systematically using CFX-5. It is observed that the formation of slug flow regimes strongly depends on the perturbation of the inlet boundary conditions. Moreover, the length of the computational domain as well as the resolution of the velocity gradient in the vicinity of the interface between the different phases plays an important role in slug formation. A detailed description of multiphase CFD modeling is given in [12]. In this reference, the transient formation of different patterns is simulated and the results agree qualitatively well with experimental video observations. However, the predicted amplitude of pressure showed quantitative differences to the experimental data. Two-phase pipe flow simulations with the OpenFoam solver interFoam are carried out in [10]. The liquid holdup and pressure drop calculated with the CFD model are in good agreement with results obtained from a two-phase mechanistic model developed in [7]. In the work of [11] the pattern formation of an air-water two-phase flow in horizontal pipelines is studied using the OpenFoam software. The simulation results agree qualitatively well with experimental video observations.

The paper is organized as follows: In Section 2 the geometry of the measurement configuration is introduced and the numerical scheme used for the CFD simulation with Fluent is sketched. Simulation results as well as a comparison with experimental data are presented in Section 3. Finally, conclusions are drawn in Section 4.

2. Modeling with Fluent

2.1. Mesh and Boundary Conditions

The simulated geometry represents a measurement set-up commonly used in multiphase flows, see Figure 1.

According to the specification of the experiments, the diameter of the pipe network $D = 0.104\,\mathrm{m}$ is chosen. In front of the measurement unit depicted in Figure 1 there is a horizontal, $16\,\mathrm{m}$ long inflow section, where the flow pattern is formed according to the prescribed superficial velocities and fluid parameters of the different phases. In the experimental set-up of the project the pattern can be observed through a glass pipe at the end of this inflow section. Behind a 90° bend a blind-T is attached so that the flow is abruptly turned 90° upstream towards the measuring unit. Hence the flow gets mixed and any prior gravity segregations of the phases break down, c.f., [6]. The volume flow rate is measured by a classical differential pressure Venturi device, whereas the phase fractions are determined using gamma densitometry. The end of the pipe network builds a 45° out of plane double bend.

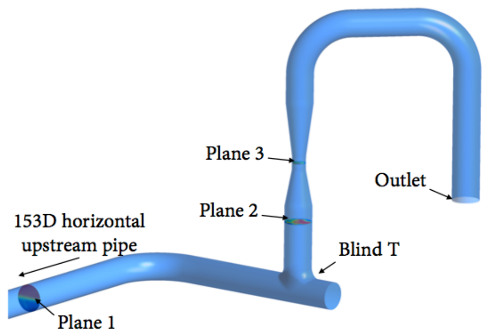


Figure 1: Geometry of the vertical Venturi measurement set-up used within the MultiFlowMet project. In front of Plane 1 there is an additional $154D$ long horizontal pipe. The three sampling planes labeled Plane 1, Plane 2, and Plane 3 are marked in the picture.

For the numerical simulation, the whole geometry has been meshed with hexahedral elements using the mesh generation tool ICEM/CFD-Hexa. The mesh consists of about 1.1 million hexahedral elements, see Figure 2.

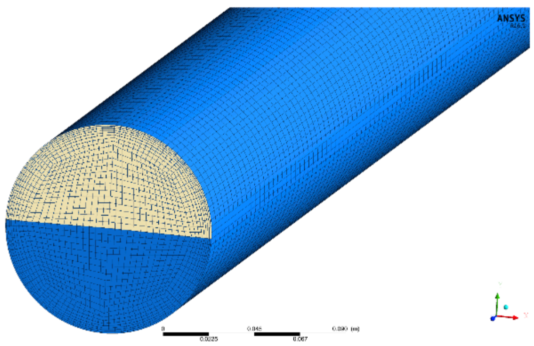


Figure 2: Inlet section of the grid used for the numerical simulations of two-phase flow. The different colors mark the inlet sections of liquid (bottom) and gas (top), respectively.

The pipe walls are treated as hydraulically smooth walls with non-slip boundary conditions applied for the gaseous and liquid phases. At the end of the pipe a pressure outlet boundary condition is set. By using the open-channel option, the solver extrapolates the required information from the interior.

In a flow-loop experiment, the phases flow together at an injection point further upstream. The CFD modeling of such a mixing section is challenging due to the different flow morphology. Therefore, for the numerical simulation, the inlet of the pipe is divided into two parts of equal size representing the inflow of the gaseous and liquid phase, respectively. The inlet section is depicted in Figure 2. The layering of both phases imposes less turbulence into the system than present in reality. Hence, at the inlet, a sinusoidal perturbation is introduced in order to stimulate the Kelvin–Helmholtz instability leading to a more natural behavior. The perturbation is implemented by means of Fluent user defined functions (UDFs).

2.2. Numerical Method

The selection of an appropriate multiphase flow model together with a free surface model is the crucial part in the simulation of multiphase flows in channels and pipes. With today's computer power it is impossible to resolve the spatial structure of the free surface down to the micro scale for the whole pipe geometry, which has a length of several meters. Hence, the evolution of Kelvin-Helmholtz instabilities at the free surface can only be resolved on a coarse scale. An interfacial drag law must be used to model the influence of these free surface instabilities on the macro-scale flow properties. Additional instabilities can be triggered by introducing an oscillation of the free surface at the inlet boundary condition, c.f. [12]. By using the Volume of Fluid (VOF) method it is only possible to resolve structures that are several times larger than the grid spacing.

It is common to use the explicit Geo-Reconstruct scheme for the interface reconstruction in the VOF method in Fluent together with a implicit first order in time discretization, c.f. [1]. This leads to a sharper interface and thus to a more accurate solution. The poor convergence on skewed meshes and the Courant number limitation of the time step are disadvantages of the method. In [9] the use of an implicit compressive scheme along with a bounded second order time discretization scheme is favored. Since this method does not have the Courant number limitation, the simulations can be run with larger time steps than with an explicit solver.

As already said, the focus lies on the modeling of the macro-scale and non-fluctuating features of the flow. Hence, for every phase the fluid-dependent k-ω based shear stress transport (SST) turbulence model of [3] was applied together with a damping of turbulent diffusion at the interface [5, 9]. At the interface between the phases turbulence damping helps to resolve the interfacial instability, since high velocity gradients lead to turbulence generation.

The schemes of the spatial and time discretization are summarized in Table 1. The Pressure Implicit with Splitting of Operator algorithm (PISO) together with an Algebraic Multigrid (AMG) is used for solving the discretized Reynolds-averaged Navier-Stokes equations.

Table 1: Numerical discretization schemes used in Fluent.

Gradient	Green-Gauss node based
Pressure	Body force weighted
Momentum	Second order upwind
Turbulence	First order upwind
Phases	Compressive scheme
Time discr.	Bounded second order

3. Simulation results and analysis

The measurement device is tested for a variety of different operational parameters. In the following, we will present simulation results for one selected case of the test envelope, namely a two-phase oil-gas slug flow.

If the superficial velocities of both phases are high enough, the interface between them becomes unstable and waves occur. For even higher velocities, the waves can growth until they touch the top wall of the pipe so that the whole diameter is covered by liquid phase. Such a flow pattern we call a slug.

This pattern is characterized by its periods, when the pipe is fully filled with liquid, alternating with periods, when the gaseous and liquid phases are separated. Within the project slug flows are of special interest due to periods without liquid or gas followed by very high liquid and gas rates. Such a behavior causes undesired consequences in a pipe network like emergency shutdown of the platform due to the high level of liquid in the separators, floods, corrosion and damages to the equipments, c.f. [8].

Here, "slug flow" means that this pattern is observed at the end of the horizontal upstream section, which is located in front of Plane 1 in Figure 1. After passing the blind-T, which is typically used as a flow conditioner, the phases get mixed so that usually another flow pattern is observed in the vertical section. The superficial velocities as well as the material properties of kerosene and nitrogen, which have been used in this test case, are summarized in Table 2.

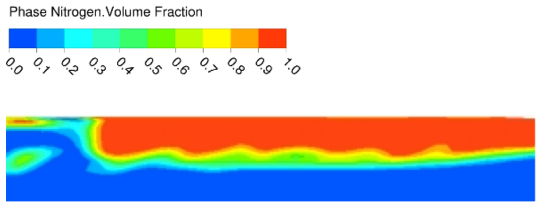
Table 2: Fluid properties and superficial velocities of the two phases used for the CFD simulation.

at 25 °C	kerosene	nitrogen
density in $\mathrm{kg\,m^{-3}}$	998.2	1.138
dyn. viscosity in $\mathrm{Pa\,s}$	$1.003 \cdot 10^{-3}$	$1.663 \cdot 10^{-5}$
superficial vel. in $\mathrm{m\,s^{-1}}$	1.144	1.399

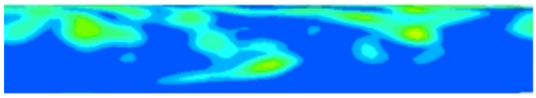
3.1. Simulation results and comparison with experimental data

In the following, the simulation results for the selected test case, see Table 2, are compared with experimental video observations. The data were collected at the National Engineering Laboratory (NEL) Multiphase facility.

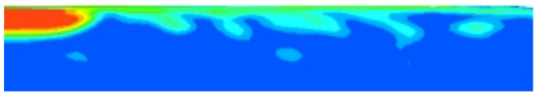
Figure 3 shows a slug traveling in the horizontal upstream pipe close to Plane 1. The pictures depict the volume fraction of nitrogen in a vertical cut through the pipe at different times. In the top picture, one can see the



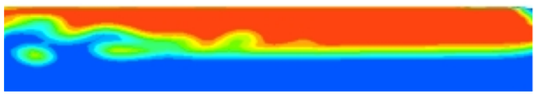
(a) Beginning of slug at $t = 38.174\,\mathrm{s}$, $\Delta t = 0$.



(b) Middle of slug at $t = 38.424\,\mathrm{s}$, $\Delta t = 0.25\,\mathrm{s}$.



(c) End of slug at $t = 38.874\,\mathrm{s}$, $\Delta t = 0.7\,\mathrm{s}$.



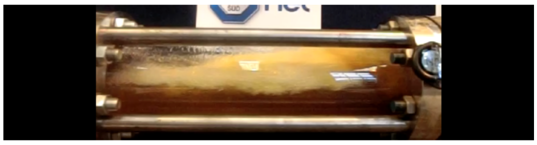
(d) Waves behind slug at $t = 39.374\,\mathrm{s}$, $\Delta t = 1.2\,\mathrm{s}$.

Figure 3: Gas volume fraction of nitrogen in a vertical cross section at the end of the horizontal inflow section at different time points obtained by CFD. Flow direction is from left to right.

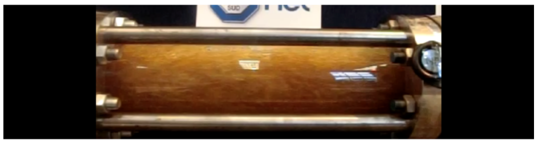
front of the slug entering the displayed area. In front of the slug, waves can be observed. The next picture shows the middle of the slug. Here the pipe is fully filled with liquid. Gas is only present in small bubbles within the slug. In the following picture the end of the slug is depicted. From the time difference between the beginning and the end of the slug (first and third picture), one can roughly estimate that the slug lasts about $0.7\,\mathrm{s}$.

In the experiments, a glass section is introduced shortly before the measurement unit. In Figure 4 a slug traveling through this section is depicted. Note that in this case the flow direction is from right to left.

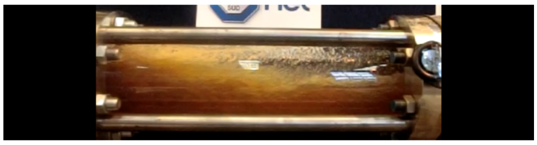
A comparison between the simulation results (Figure 3) and the pictures extracted from the video observations (Figure 4) show that the structure of the slug is reproduced quite well by the numerical simulation. Also the time differences between the beginning, middle, and end of the slug match quite well with the experimental observations. Both in the numerical simulation and in the experiment, one observes smaller waves behind most of the slugs, see bottom pictures in Figures 3 and 4. The time it takes until these waves occur are also very similar in the CFD



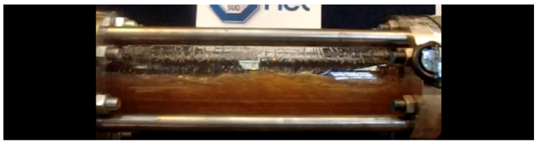
(a) Beginning of slug, $\Delta t_{\text{exp}} = 0$.



(b) Middle of slug, $\Delta t_{\text{exp}} = 0.375\,\text{s}$.

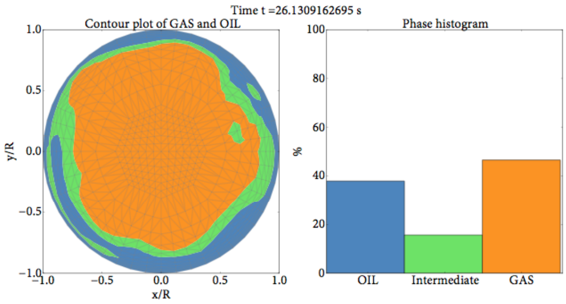


(c) End of slug, $\Delta t_{\text{exp}} = 0.875\,\text{s}$.

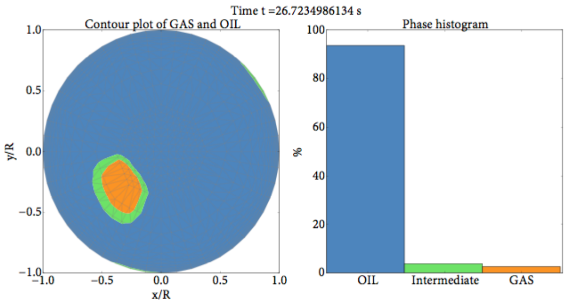


(d) Waves behind slug, $\Delta t_{\text{exp}} = 1.175\,\text{s}$.

Figure 4: View through a glass section located at the end of the inlet section at different times. Flow direction is from right to left. Data were collected at the National Engineering Laboratory (NEL) Multiphase facility.



(a) $t = 26.13\,\text{s}$.



(b) $t = 26.72\,\text{s}$.

Figure 5: Left: contours of the gas volume fraction in the Venturi throat at two different times. Right: corresponding histograms of both phases.

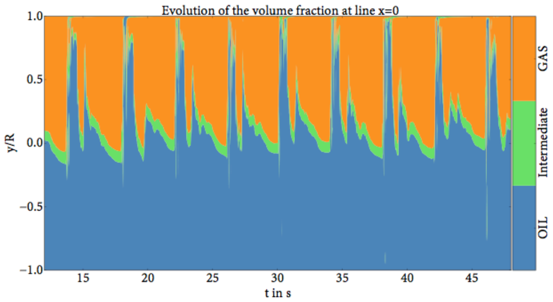
simulation and in the experiments. However, the strong mixing between the phases, which can be seen in the experiments, cannot be reproduced by the used numerical method. This is due to the fact that the mesh is relatively coarse so that the interface between the phases is resolved only roughly.

3.2. Data analysis

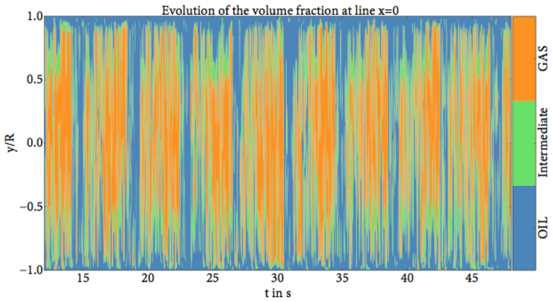
In view of a more quantitative comparison between experimental and numerical results, the huge amount of simulation data need to be condensed so that the main features of the flow can be extracted. This then allows an automatic flow pattern recognition as well as a better comparison between different simulations and with experimental data. In the following, some results will be presented for the selected test case from above.

Figure 5 depicts contours of the gas volume fraction in a horizontal cross section inside the throat of the Venturi at different times. Using the notation from Figure 1, the pictures show Plane 3. The blue color represents volume fractions of nitrogen up to 0.35, the orange one volume fractions of 0.65 and higher. By these classifications, the blue and orange color represents the liquid and gaseous phase, resp.. The green color marks volume fractions of nitrogen between 0.35 and 0.65 and can thus be interpreted as a mixture of the two phases. One observes that due to the blind-T the horizontal slug flow pattern has been transformed into an annular like flow pattern. This flow pattern is characterized by the presence of a liquid film on the pipe wall. Comparing the left top and bottom pictures of Figure 5, one can recognize that the high volume fraction of liquid in the top picture corresponds to a slug, previously present in the horizontal section before the flow conditioning unit.

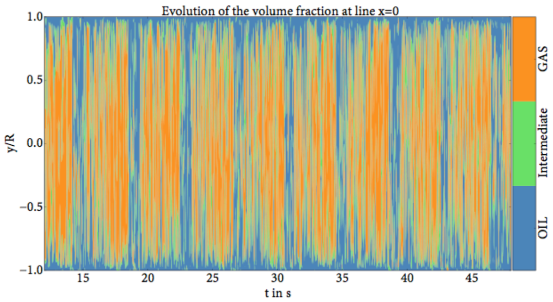
Figure 6 plots the evolution of the gas volume fraction on lines through the middle of the labeled planes from Figure 1. The blue color identifies gas volume fractions greater or equal 0.65, whereas the orange color stands for gas volume fractions less or equal 0.35. By this association the flow pattern can be extracted from the diagram. The top picture shows the gas volume fraction along a vertical line on Plane 1 versus time. One can recognize the periodic occurrence of slugs (the high blue peaks, when the whole pipe is filled with liquid) as well as the waves in between the slugs (the smaller blue peaks). Moreover the green color is an indicator of the height of the fluid interface. The middle and bottom pictures show the gas volume fraction along horizontal lines on Plane 2 and Plane 3, respectively. Here the pattern has changed to annular, since the flow has passed the blind-T flow conditioning unit. The liquid film present on the vertical walls that are typical for annular flow can be recognized in both the middle and bottom pictures of Figure 6. The periodic behavior (previously representing the slugs) is still present in the vertical section. However, it is not so



(a) Plane 1.

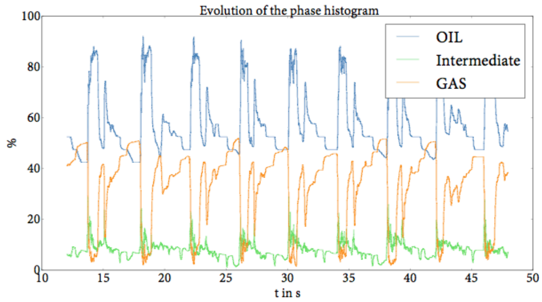


(b) Plane 2.

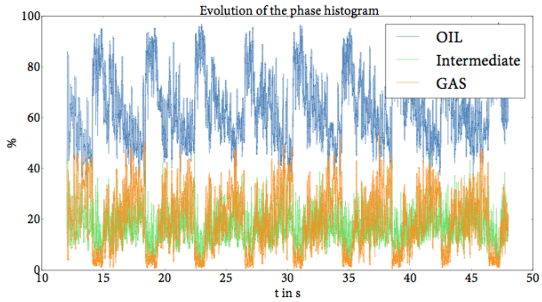


(c) Plane 3.

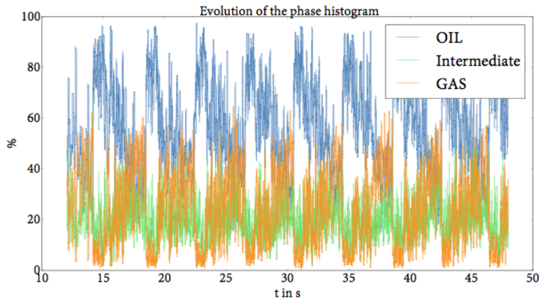
Figure 6: Evolution in time of the gas volume fraction (a) along a vertical line through the middle of Plane 1, (b) along a horizontal line through the middle of Plane 2, and (c) along a horizontal line through the middle of Plane 3.



(a) Plane 1.



(b) Plane 2.



(c) Plane 3.

Figure 7: Evolution in time of the histograms representing the two different phases (a) at the end of the inflow section, (b) shortly in front of the Venturi, and (c) inside the Venturi.

pronounced any more. The fluid is more mixed due to the blind-T.

In Figure 7 the time evolution of the histograms presented in the right pictures of Figure 5 are plotted for all of the three planes in Figure 1. Thus, the graphs give the percentage of very high gas volume fractions (orange) and very low gas volume fractions (blue) in the corresponding plane. For pattern recognition it is sensible to associate the orange curves with gas and the blue ones with oil. Areas where the gas volume fraction is between 0.35 and 0.65 are classified as Intermediate. They may either indicate the interface between the two phases or correspond to areas, where both phases are mixed.

Figure 8 shows the histograms of the gas phase for the three planes in one plot. Note that only a smaller time interval is shown in this picture so that the smaller time scales are resolved better. In the plot, cells with gas volume fractions greater or equal 0.65 are shown. In this picture one can recognize again the periodic occurrence of slugs on Plane 1. Furthermore, the slug frequency can be read off, which allows a more quantitative comparison with experimental data. On Plane 2 and Plane 3 the curves are much more oscillating due to the disturbances introduced by the blind-T.

Also other test points of the intercomparison test matrix of the MultiFlowMet project were simulated. The different superficial velocities lead to other slug frequencies or even other flow patterns. These changes can be observed by the evolution of the histograms.

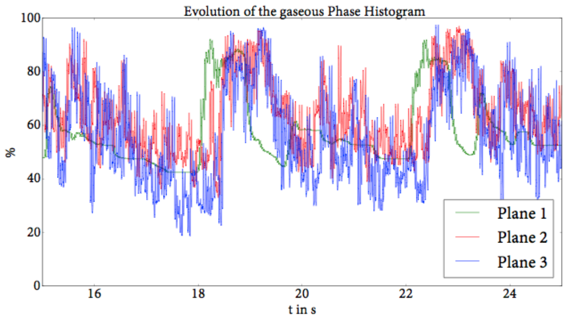


Figure 8: Evolution in time of the histograms of the gas phase in three different planes.

4. Conclusion

For this contribution a two-phase oil-gas flow in a meteorologically relevant geometry was simulated by the commercial CFD solver Ansys Fluent. For tracking the interfaces between the phases the Volume of Fluid (VOF) method was used.

The numerical results are in good agreement with experimental video observations provided by NEL. Since optical comparisons are time consuming and inaccurate, appropriate key data describing relevant flow properties have to be identified. For slug flow quantities of interest are the frequency and length of a slug, the interface height, etc.

The data analysis provides an approach to condense the huge amount of data and to extract the characteristics of the flow. By identifying high and low gas volume fractions with gas and oil, respectively and by neglecting the areas, where both phases are mixed, patterns can be classified. This forms the basis for a quantitative comparison with experimental data coming from tomography or gamma densitometry.

Acknowledgement

The authors acknowledge the support received from the European Metrology Research Programme (EMRP) through the Joint Research Project "Multiphase flow metrology in Oil and Gas production". The EMRP is jointly funded by the European Commission and participating countries within Euramet and the European Union. The authors would like to thank Jiri Polansky for his assistance and for valuable discussions.

References

[1] *ANSYS FLUENT User Guide*. 2015.

[2] D. Bestion et al. "Two-phase CFD: The various approaches and their applicability to each flow regime". In: *Multiph. Scien. Techn.* 23.2-4 (2011), pp. 101–128. ISSN: 0276-1459.

[3] F. R. Menter. "Two-equation eddy-viscosity turbulence models for engineering applications". In: *AIAA J.* 32 (1993), pp. 1598–1605.

[4] T. Frank. "Advances in Computational Fluid Dynamics (CFD) of 3-dimensional Gas-Liquid Multiphase Flows". In: *Proceedings of NAFEMS Seminar on Simulation of Complex Flows (CFD) - Applications and Trends*. 2005.

[5] T. Frank. "Numerical simulation of slug flow regime for an air-water twophase flow in horizontal pipes." In: *Proceedings of The 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics NURETH-11*. 2005.

[6] N. Carter and B. Theuveny. "Phasing Out Separation". In: *Middle East and Asia Reservoir Review* 3 (2003).

[7] N. Petalas and K. Aziz. "A Mechanistic Model for Multiphase Flow in Pipes". In: *J. Can. Pet. Technol.* 39.06 (2000), pp. 101–128.

[8] A. Sausen, P. Sausen, and M. de Campos. "The Slug Flow Problem in Oil Industry and Pi Level Control". In: *New Technologies in the Oil and Gas Industry*. Ed. by J. S. Gomes. InTech, 2012, pp. 103–118.

[9] J. Seo. "Multiphase Flow Modeling with Free Surfaces Flow". In: Presented at the Korea Convergence Conference, Korea, 2014.

[10] A. M. Shuard, H. B. Mahmud, and A. J. King. "Comparison of Two-Phase Pipe Flow in OpenFOAM with a Mechanistic Model". In: *IOP Conf. Ser. Mater. Sci. Eng.* 121.1 (2016), p. 012018.

[11] J. P. Thaker and J. Banerjee. "CFD Simulation of Two-Phase Flow Phenomena in Horizontal Pipelines using OpenFOAM". In: *Proceedings of the Fortieth National Conference on Fluid Mechanics and Fluid Power*. 2013.

[12] C. Vallée et al. "Experimental investigation and CFD simulation of horizontal stratified two-phase flow phenomena". In: *Nucl. Eng. Des.* 238.3 (2008). Benchmarking of CFD Codes for Application to Nuclear Reactor Safety, pp. 637–646. ISSN: 0029-5493.