Ninth International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
10-12 December 2012

QUASI-3D MODELLING OF TWO-PHASE SLUG FLOW IN PIPES

Sjur MO*[1], Alireza ASHRAFIAN[1], Jean-Christophe BARBIER[2] & Stein Tore JOHANSEN[1]

[1]Flow Technology Group, SINTEF Materials and Chemistry, 7465 Trondheim, NORWAY
[2]Total E&P, Norway, Stavanger
*Corresponding author's e-mail: sjur.mo@sintef.no

ABSTRACT

In this paper, we present progress obtained by the Quasi 3-Dimensional (Q3D) model for pipe flows. This model is based on a multi-fluid multi-field formulation with construction and tracking of the large-scale interfaces (LSIs). Here the computational time is significantly reduced by performing a slice-averaging technique. However, new terms are created in the model equations which are related to the important mechanisms such as wall shear stress and turbulence production at side walls.
The paper reports some basic performance of the model, including single phase wall friction and the velocities of single Taylor bubbles at inclinations ranging from horizontal to vertical. Finally we report the performance of the model for slug flow in horizontal and 10° inclined pipes.
The model seems to satisfactorily reproduce the two investigated slug flows. This indicates that the model can have a great potential is serving the oil & gas industry.

NOMENCLATURE

D	Pipe diameter (m)
Fr	Froude number ($Fr = v_{drift} \big/ \sqrt{gD}$)
v_{drift}	Drift velocity (m/s)
g	Gravity (9.81 m/s^2)
k_m	Turbulent kinetic energy for phase m (m^2/s^2)
l	Turbulent length scale (m)
Re_D	Pipe Reynolds number ($\mathrm{Re}_\mathrm{D} = \rho UD \big/ \mu$)
U	Stream wise velocity (m/s)
ε	Wall roughness (m)
ε_m	Turbulent dissipation for phase m (m^2/s^3)
μ_m	Molecular viscosity for phase m (Pa·s)
μ_m^T	Turbulent viscosity for phase m (Pa·s)
ρ_m	Density for phase m (kg/m^3)

INTRODUCTION

In industrial pipelines for oil and gas transport unstable flows can cause major operational problems. A main problem is that the liquid is arriving in larger, intermittent chunks (slugs), and not continuously. In this case a separator with huge volume would be needed to handle the liquid in such large slugs. These instabilities are caused by liquid waves that grow and interact to form hydrodynamic slugs. Empirically it has been observed that these hydrodynamic slugs can grow continuously with time and form huge slugs (Shea et al., 2004). However, the mechanisms of initial slug formation are poorly understood, together with the growth mechanisms which lead to the manifestation of large and industrially problematic slugs.
SINTEF, ConocoPhillips and Total have been working with development of the LedaFlow multiphase flow prediction tools (Laux et al., 2007, 2008a, 2008b) to enable more fundamental prediction of multiphase flows, including the phenomena of slugging. The overall idea has been to develop a model which is capable of handling most multiphase flow phenomena that will appear in a pipeline. Typical situations to predict are two and three phase flows where the flow patterns include waves and distribution of dispersed fields. The flow pattern should be fundamentally predicted by the model. In addition, the model should be sufficiently fast to analyze the flow in a relevant pipeline length. The results presented in this paper show the capabilities of this model for some selected applications.

Modeling of slug flow in pipes

The main mechanism leading to the formation of the liquid slugs in pipes and channels is the retardation of the liquid phase by wall friction. Due to incompressibility and conservation of the volume of the liquid phase, the liquid level is slightly rising with increasing distance along the pipe. Simultaneously, this leads to acceleration of the gas phase in the upper part of the pipe leading to increasing local gas velocities and corresponding pressure drop. If a critical velocity difference between the two phases is exceeded, the interface become unstable and wavy structures develop. Further reduction in the local gas pressure reinforces the build-up of the wave which leads to complete blockage of the pipe-cross section by the liquid phase and hence formation of a liquid slug. The blockage of the cross-sectional area gives rise to a steep pressure-gradient in the gas phase which drives the liquid slug. Depending on the pipe geometry (length and diameter) and the gas and liquid flow rates, the slug flow regime can be stable, in which liquid slugs move over long distances in the pipe, or, in other cases, liquid slugs disintegrate after a certain distance of propagation due to loss of the critical liquid mass contained in the slug.

Successful modeling of hydrodynamic slug flow poses several challenges. One of them is modeling the dynamic behavior of the interface which separates the two layers of fluid but, at the same time, where significant entrainment and mixing takes place, leading to simultaneous dispersion of gas bubbles into the liquid and liquid droplets into the gas. These phenomena, as well as the prediction of the bubble and droplet size, have significant importance in determining the slug flow regime and are very difficult to predict due to the complex turbulence phenomena taking place at and in the vicinity of the large scale interface. Due to these effects, accurate physical predictions are beyond the current 1D-modeling capabilities. Hence, we need to address the slug flow process by applying more fundamental principles.

A review on past attempts towards numerical simulation of the slug flow regime in horizontal pipes is presented in the paper by Frank (2005). In general, the current two major modeling approaches for modeling of dispersed and separated flows, the standard multi-fluid Eulerian and the volume-of-fluid (VOF) methods, are not fully capable of handling situations where large scale interfaces and dispersion of phases co-exist. Multi-fluid models are well suited for dispersed flows (with no large scale interfaces) whereas the VOF models are well-suited for separated flows with no mixing at the interface.

The three dimensional nature of the slug flow in pipes has a crucial significance that cannot be ignored. Firstly, and as discussed earlier, formation of the slug flow is strongly influenced and determined by the wall friction on the liquid phase. In a plane 2D approximation of the slug flow in pipe, the effect of sidewalls on the flow is neglected and therefore, the extra retardation of the liquid phase by the pipe walls is less emphasized compared to a full 3D flow. Secondly, the total blockage of the cross sectional area by the liquid phase is more easily established in pipes than in channels. Therefore, plane 2D modeling of slug flow in pipes cannot yield good predictions.

Full 3D simulations of slug flow in pipes are very expensive in terms of computational time, memory requirements, data storage and post-processing. Compared to the diameter of the pipe, the length of the pipe has to be sufficiently long so that hydrodynamic slugs can be generated (see e.g. Lakehal et al. 2012). Hence, a 3D based method, averaged down to two dimensions, may offer a good compromise of speed and accuracy. This approach is explained next.

MODEL DESCRIPTION

Model basis

The model is based on a 3D and 3-phase formulation, where the equations are derived based on volume averaging and ensemble averaging of the Navier-Stokes equations. Conceptually, the model is based on the following elements (Laux et al., 2007):

i) A multi-fluid Eulerian model allowing two types of dispersed fields in each of the three continuous fluids.
ii) The flow domain consists of several zones, each with a well-defined continuous fluid, separated by LSIs
iii) Between the zones local boundary conditions are applied (interface fluxes)
iv) Field based turbulence model with wall functions for interfaces and solid walls.
v) Evolution models for droplet and bubble sizes
vi) By adding together the field-based equations phase based mass, momentum and turbulence equations are obtained

For the turbulence model we have here applied a length scale model, which is solved from a Poisson equation. At solid walls and LSIs the length scale is given as a boundary condition. Turbulent energy equations are solved for each phase, again applying wall laws at solid walls and the LSIs. The turbulent viscosity for phase m is given by:

$$\mu_m^T = \rho_m l \sqrt{0.35 k_m} \quad (1)$$

The turbulent dissipation rate for phase m is:

$$\varepsilon_m = \left(0.35 k_m\right)^{1.5} / l \quad (2)$$

The resulting model gives the volume fractions and momentum for the phases in the flow. In order to apply local boundary conditions inside the flow as described above we need to identify the Large Scale interfaces. This is done based on an evaluation of the predicted phase volume fraction, based on the assumption that there is a critical volume fraction which controls phase inversion. In this work a phase is continuous if the local volume fraction is above 0.5. Based on a relative simple reconstruction algorithm, the interface is reconstructed such that the local boundary conditions can be applied. Presently, the effects of surface tension on the motion of the Large Scale Interface are not included. This simplification is good as long as we use relatively coarse grids and do not want to resolve capillary waves.

This model framework has the capability to handle any 3D 3-phase (or less) multiphase flow as long as the flow can be described by 9 fields – 3 continuous fields with 2 dispersed fields in each. However, fields such as thin liquid wall films are not included. As this model is directed towards predictions of multiphase flows in pipelines long sections of pipes will have to be simulated for a considerable flow-time. This restriction demands simplifications in order to be able to obtain results in a reasonable time. Weeks or months of computer time on parallel machines would not be acceptable for most industrial applications. The simplification we have introduced is the Quasi 3D (Q3D) approximation. By slicing the pipe in one direction (usually the vertical direction), as demonstrated in Figure 1, the flow can be resolved as 2-dimensional, but describing the complete flow in a pipe.

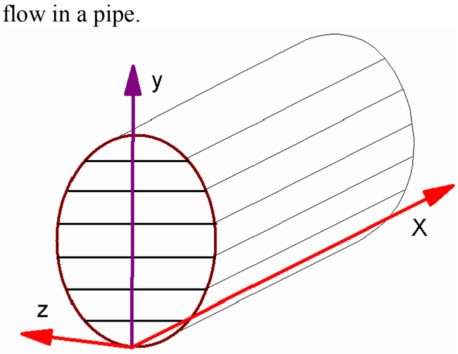

Figure 1: Quasi 3D grid cells, showing one axial (x-direction) and 7 vertical cells.

The full 3D model equations are then averaged over the transversal distance z to create slice averaged model equations. In this process the 3D structures are homogenized and the flow becomes represented by slice averaged fields. One result is that the wall fluxes, such as shear stresses, becomes source terms in what we call

Quasi-3D (Q3D) model equations (for details, see Laux et al., 2007).

The numerical solution is performed on a staggered Cartesian mesh, where the discrete mass, pressure and momentum equations are solved by an extended phase-coupled SIMPLE method (Patankar, 1980). The implicit solver uses first order-time discretization and up to third-order in space (convective terms, Laux et al., 2007).

The Quasi 3D model description is expected to perform well in horizontal stratified and hydrodynamic slug flows where the large scale interface is dominantly horizontal at a given axial position x, as seen in Figure 11 and demonstrated in previous papers (Laux et al., 2007, 2008a, 2008b).

The applicability of the Q3D approximation to high inclination and vertical flows can only be clarified by testing the model versus experiments. This will be discussed below.

BASIC MODEL PERFORMANCE

Performance tests

In order to verify the model single phase calculations were performed to check the prediction quality of wall shear stresses and the resulting pressure drop. In the verification runs good agreement with slice averaged profiles of velocity and turbulent energy was obtained. In Figure 2 we see that the model gives acceptable single phase pressure drops over the entire range of Reynolds numbers and wall roughnesses ε.

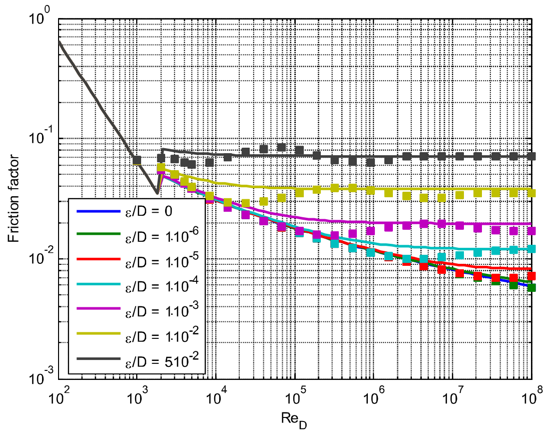

Figure 2: Moody diagram (Moody 1944) showing friction factor calculated using the Colebrook (1939) equation (lines) and Q3D (squares) for different relative wall roughnesses **ε** /D versus pipe Reynolds number.

As a symmetry test the model was tested for the Rayleigh–Taylor instabilities for all spatial directions. It has also been demonstrated that it is possible to obtain the Kelvin Helmholtz instability.

Further validation of the model is discussed next.

Taylor bubble velocities

Another fundamental check of the model is its capability to reproduce the velocity of Taylor bubbles in two phase flows. Accurate representation of the speed of Taylor bubbles, in both horizontal and inclined pipes, is essential for modeling slugs under operational conditions. We therefore investigate the Q3D model's capability to handle Taylor bubbles in pipes with various inclinations, ranging from horizontal to vertical. In a recent paper Jeyachandra et al. (2012) reported measurements of drift velocity for air bubbles in high viscosity oils for different inclinations and pipe diameters. The oil viscosities were (*0.105,0.256,0.378,0.574*) Pa·s, the inclinations (*0°,10°, 30°,50°,70°,90°*) and the pipe diameters (*2,3,6*) inches. In Figure 5 we compare the results for diameter *76.2* mm and oil viscosity *574* mPa·s with CFD results both from Fluent 3D and Q3D. The pipe length was *4* m. For the Q3D simulations we used a $600{\times}15$ mesh[1] while the Fluent mesh had *47704* cells. The cross sections are shown in Figure 3. The pipe configuration and initialization was as shown in Figure 4. The general trend is that both Fluent 3D and Q3D underestimate the drift velocity, but both are able to capture the main trend with a maximum for intermediate inclination angles. The low velocity for horizontal pipe is probably partly related to problems emulating correct boundary conditions, and more work is needed here. The simulation time is typically 2-3 times shorter for Q3D compared with Fluent 3D.

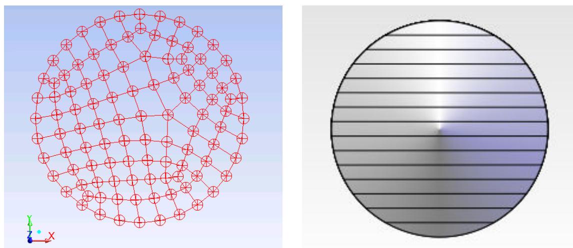

Figure 3: Mesh cross sections. Left: Fluent3D, Right: Q3D.

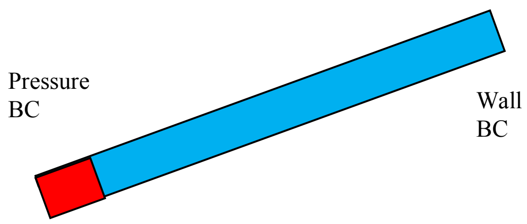

Figure 4: Initial state volume fraction with boundary conditions. Gas (red) is patched in at the bottom end of the liquid (blue) filled pipe. The pressure boundary is a pressure outlet with gas only backflow.

[1] A mesh sensitivity study was performed to ensure sufficiently fine mesh. The conclusion was that 600 cells in the stream wise direction were sufficient. For the transversal direction the results were more inconclusive.

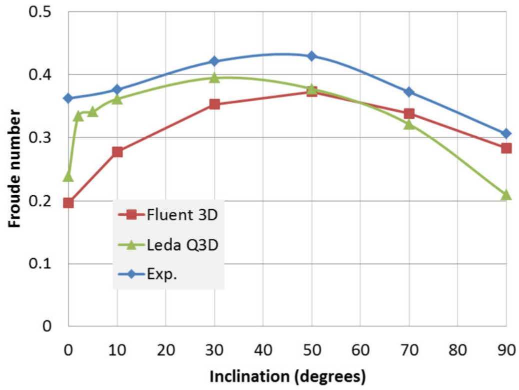

Figure 5: Comparison of experimental and CFD results for the Froude number, versus inclination angle. The pipe diameter is 76.2 mm.

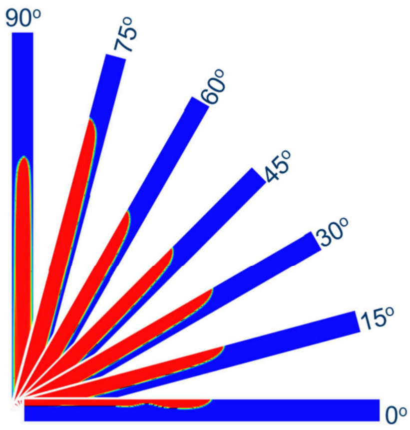

Figure 6: Taylor bubble shapes (red) for different inclinations. Red is gas and blue is liquid.

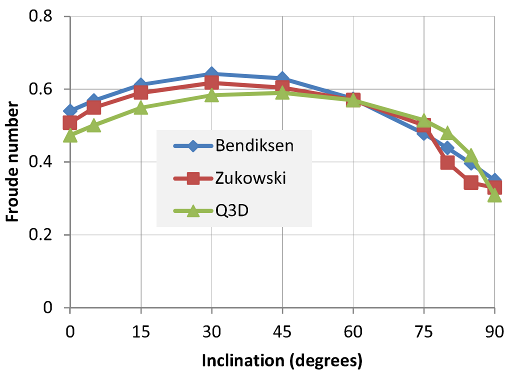

Figure 7: Comparison of Q3D results with experiments (Zukowski 1966) and correlation (Bendiksen 1984).

SLUG FLOW APPLICATIONS

Horizontal flows

As we have demonstrated that the model well reproduces both pressure drops and Taylor bubble slip velocities, we now look into the reproduction of slug flow. The numerical simulation described for this case is based on experiments carried out at Imperial College London (WASP facility) by Ujang *et al.* (2005) in order to study the initiation and the subsequent evolution of hydrodynamic slugs in a horizontal pipe. Air-water experiments were carried out at atmospheric pressure, 4.0 bar(a) and 9.0 bar(a), and the effects of superficial liquid and gas velocities were investigated. The test section used for these experiments was 37 m in length, with an internal diameter of *0.078* m. Further details are described in the paper.

For the numerical simulation presented here, atmospheric pressure with $U_{sg} = 4.64$ m/s and $U_{sl} = 0.611$ m/s were used. The pipe length was *30* m with 10×2440 cells uniformly distributed across the diameter of the pipe and in the axial direction, respectively, leading to a grid aspect ratio of *1.5*. Gas compressibility was taken into account by using a PVT table created for the air-water system. No perturbations were imposed at the inlet so that fluid phases were entering the pipe fully stratified. The details of the inflow arrangement for the fluids were not included in the simulations. The pipe was initially filled with stratified air and water with 50-50 volume percentage and zero velocity. Computations were carried out in parallel on 4 CPUs using MPI. The total flow time for this simulation was 52.7 seconds for which a total clock time of 2.3 days was used[2].

Snapshots from the evolution of slugs in the pipe are shown in Figure 8. The pipe diameter is magnified 5 times for clarity of the flow details. Initially the water phase is smooth and it takes some simulation time until a first wave is created, growing to a slug which blocks the cross section of the pipe (frame a) and grows in size as it progresses in the pipe. However, this initial long slug (frames c-e) is not periodic and is believed to be generated out of the initial condition of the flow inside the pipe. Similar initial slugs are also observed in experiments, e.g. Kristiansen (2004).

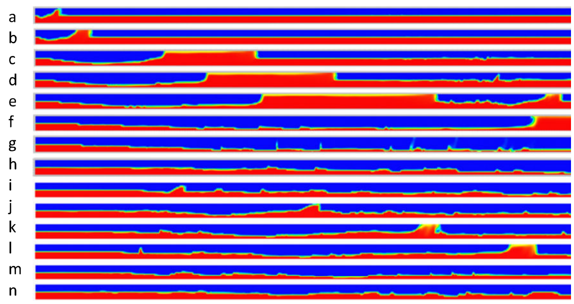

Figure 8: Snapshots of Q3D results showing the time evolution of slugs in a 30 m long horizontal pipe (diameter is magnified 5 times). Here red is liquid and blue is gas. Flow is from left to right.

After that the initial long slug has almost drained the pipe from liquid (frame f), the interface level starts to rise until it reaches a critical level (frame h) at which interfacial disturbances are created and next grow into a new slug (frame j). These disturbances are captured by the model, and as the simulation proceeds further in time, they grow into slugs which completely block the cross section of the

[2] For a comparison with full 3D simulation we note that Lakehal et al. (2012) using the TransAt code used 21 days on 8 cores to run 16 m pipe for 30 seconds real time. The mesh had 1.4 million cells. Their results were better in predicting the slug frequencies the first 5-10 m in the pipe, but had larger errors than Q3D later.

pipe. Figure 9 shows the liquid hold-up time series at different probe locations along the pipe. The development of slug flow in space and time can be studied in great detail. The frequency of the slugs is calculated from these time series based on both 60% and 80% volume fraction of liquid phase as a defined threshold for the slug. The calculated frequency versus distance from the inlet is plotted in Figure 10 and compared with the experimental values. In the experiments, a high frequency of slugs formed in the inlet region of the pipe is observed. This effect is not captured by the model with the current inlet and initial conditions. However, this is believed to be strongly affected by the inlet conditions in the experiments (Ujang, et al., 2005). At distances further from the inlet, the slug frequency compares relatively well with experiments.

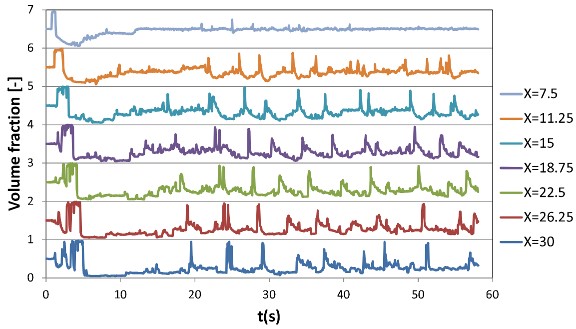

Figure 9: Liquid hold-up time-series for hydrodynamic slug flow at different locations (X [m]) along the horizontal pipe. Vertical axis is shifted by 1.0 for each series for readability.

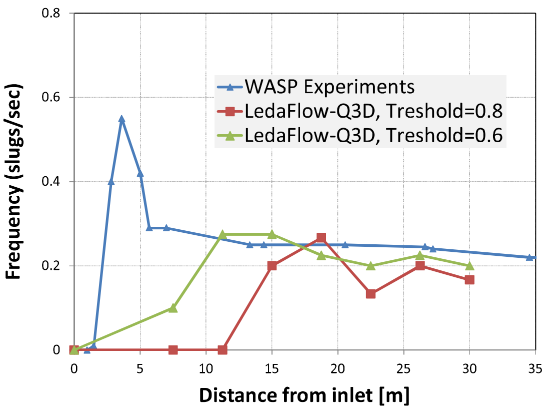

Figure 10: Slug frequency variation along the pipe.

Inclined flows

In this section the Q3D model is applied to simulate flow of oil/gas mixtures in a 12 inch and 10° upward inclined pipe. The simulation results are compared to experimental data obtained in the large scale loop at the SINTEF multiphase flow laboratory. Simulation results are presented and discussed for only one of the many 12 inch experiments. More simulation cases for different experiments in the 12 inch loop are presented and discussed in Laux et al. (2007).

The used fluids reasonably represent a produced oil-gas fluid system. The data on the physical properties, however, is proprietary and can therefore not be given here. The superficial velocities were $U_{sg} = 2.552$ m/s and $U_{sl} = 0.502$ m/s.

The simulations were performed using a compressible gas on a 100 m pipe on a 20×2000 grid[3]. A typical flow situation is shown in Figure 11. Here we see one slug bridging the pipe fully, while some large waves are about to bridge the pipe. The turquoise color shows regions where unresolved gas bubbles have been entertained into the liquid (blue). The entrainment of gas bubbles is seen to be more intense at the slug fronts. In Figure 12 we compare time traces of liquid hold-up from simulations and experiments. The main behavior is very similar, but the amplitude is somewhat larger in the simulated results. The corresponding probability density function (PDF) is shown in **Figure 13**. The main peak is almost exactly at the same volume fraction. The shape of the PDF indicates slug flow since we have two "peaks" even if the high hold-up peak is not very pronounced.

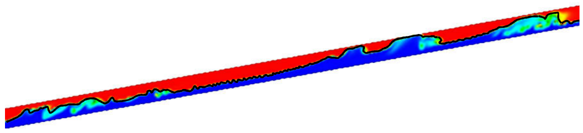

Figure 11: Excerpt of snap-shot from prediction of slug flow in an inclined pipe, 12 inches in diameter and 100 m long (the pipe diameter in the picture is magnified 5 times). The colours denote gas fraction, where red is 100% gas and deep blue is no gas (liquid). Flow is from left to right.

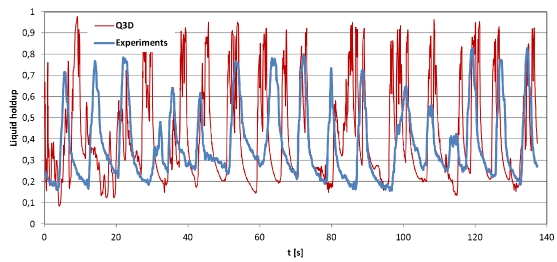

Figure 12: Liquid hold-up signal at a location 90 meters from the pipe inlet as compared to the experimental Gamma-ray signals.

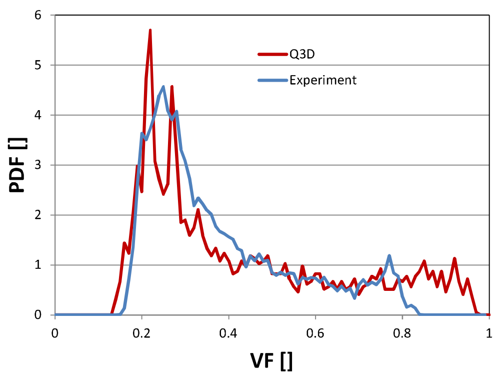

Figure 13: Probability density function (PDF) of the liquid volume fraction (VF) signal at 90 m from inlet compared with that of experimental data.

DISCUSSION

The Q3D model is, as described above, built on several simplifications and sub-scale models. The two most

[3] The simulation time needed to run this case on 8 processors for 3 min real time was about 3.5 days.

important model features are the slice averaging (Q3D approximation) and the modeling of the physics at the Large Scale Interface.

The basic tests with Taylor bubbles show, quite surprisingly, that good estimates for bubble velocities can be obtained for high inclinations, and even for vertical flow (Figure 7). In the vertical case the pipe is sliced in one transversal direction while the experimental flow is expected to be more radial symmetrical in nature. As a result the predicted fluid wall shear stresses, along the bubble body, are expected to deviate from experimental values. Experimental data is needed to quantify such deviations. However, the critical result is the models capability to predict experimental bubble velocities, as these velocities are critical for all processes that control liquid accumulation and pressure drop.

We may note that in vertical flow we are able to work with 2D representations, using either radial symmetry or the Q3D approximation. However, radial symmetry offers one transversal degree of freedom for the flow (in or out from centerline), while Q3D offers two degrees of freedom (independent transversal flow at each side of the center line). As a result the Q3D approximation has a better potential to reproduce complex flow patterns for high inclination and vertical flows. This has already been indicated (Laux et al., 2008a) in studies of riser flows.

In the analyses of the WASP slug experiments (Ujang et al., 2006) we see that the Q3D model is producing slugs from unperturbed inlet conditions. The overall physics is well reproduced, including the *developed* slug frequency. However, the slugs in the simulations appear later than in the experiments. The reason for this discrepancy is partly attributed to the simplification of the inlet section used in the Q3D simulations. The 3D geometry of the inlet section, particularly a horizontal plate, is expected to trigger instabilities and waves. The second issue is the neglect of capillary waves. By running the Q3D simulations on a grid that is too coarse to resolve capillary waves we can run fast simulations. Currently, it is clear that the detailed onset of instabilities may be impacted by capillary waves, but if these are of importance in these actual experiments remains to be investigated.

In the final application, on 12 inch and 10° upward inclined flow, we have seen that frequency and the PDF of liquid volume fraction is in general well reproduced. However, we see from Figure 13 that the experiments indicate that slugs contain significant amounts of dispersed gas (~ 20%), while the simulations indicated slugs with much less gas (~ 3%). This indicates that the gas entrainment in the model may be underestimated, or that 3-dimensionality (secondary flows) in the slug front may impact the entrainment and separation of dispersed gas bubbles. The accuracy of the interpretation of the gamma-densitometer relies on the flow being fully stratified. This may impact the accuracy of the measurements if the gas bubbles are trapped into secondary flows in the slug front. However, experimental uncertainty alone seems not sufficient to explain the high gas fraction in the slugs.

CONCLUSION

Using wall functions for solid walls our Quasi 3D model can reproduce single phase flows as required for engineering simulations. Taylor bubble velocities, being the fundamental building block of slug flows, are reproduced well for all inclinations including perfectly vertical flows.

The model is capable of reproducing onset of slugging and reproduces closely the slugging frequency observed in experiments. In 10° inclined pipe flow the model reproduces well both the shape of the time traces, frequency and the PDF of the cross sectional averaged liquid volume fraction. In the latter case it was found that the model seems to under-predict the gas entrainment into the slugs. The reason for this discrepancy should be identified, as this indicates an area for model improvement.

It has been demonstrated that our Q3D model for multiphase pipe flows is able to reproduce important features of two-phase pipe flow. In particular it has been shown how the model can handle flows containing large resolved bubbles and more complex transitional slug flows with significant amounts of dispersed bubbles and droplets. Due to the 2D numerical representation the Q3D model is significantly faster than full 3D models, allowing longer pipes to be simulated for a longer time. The speed and the accuracy of the model indicate that it may have a great potential in serving the oil & gas industry.

As the model is already extended to 3-phase flows it will in the future be interesting to see and communicate the model performance for such exceedingly complex flows. Finally, it is also realized that the experimental techniques used in pipe flows research are often inadequate to validate multidimensional models. It is therefore a must to provide model developers with more high quality multi-dimensional experimental data.

ACKNOWLEDGEMENTS

The permission of TOTAL, ConocoPhillips, and SINTEF to publish this paper is gratefully acknowledged. Our colleagues Ernst Meese, Wouter Dijkhuizen, Runar Holdahl, Dadan Darmana, Jørn Kjølås, Harald Laux, OSRAM Opto Semiconductors GmbH, Regensburg, and Alain Line, INSA, Toulouse, are acknowledged for their contributions.

REFERENCES

BENDIKSEN, H.K., (1984), "An experimental investigation of long bubbles in inclined tubes", Int. J. Multiphase flow 10, 467-483.

FRANK, T. (2005), "Numerical simulation of slug flow regime for an air-water two-phase flow in horizontal pipes, Paper: 038, *Proc. The 11th Int. Topical Meeting on Nuclear Thermal Hydraulics (NURETH-11),* Avignon, France, October 2-6, 2005.

JEYACHANDRA, B.C., GOKCAL, B., AL-SARKHI, A., SARICA, C., and SHARMA, A.K. (2012), "Drift-Velocity Closure Relationships for Slug Two-Phase High-Viscosity Oil Flow in Pipes". *SPE J.* **17** (2): 593-601. SPE-151616-PA. http://dx.doi.org/10.2118/151616-PA.

KRISTIANSEN, O. (2004), Experiments on the transition from stratified to slug flow in multiphase pipe flow, Ph.D. thesis, Norwegian University of Science and Technology (NTNU), Trondheim, 2004.

D. LAKEHAL, M. LABOIS, S. THOMAS, C. NARAYANAN, (2012), "Advances in the Large-Eddy and Interface Simulation (LEIS) of interfacial multiphase flows in pipes", Progress in Computational Fluid Dynamics, An Int. J., Vol.12, No.2/3: 153 – 163.

LAUX, H., MEESE, E., JOHANSEN S.T.,LADAM Y.,BANSAL, K. M., DANIELSON T.J., GOLDSZAL A. and MONSEN, J.I. (2007), "Simulation of multiphase flows composed of large scale interfaces and dispersed fields", Paper S5_Tue_D_29, *Proc. Int. Conf. Multiphase Flows*, Leipzig, July 9-13th, 2007.

LAUX, H., MEESE, E., MO, S., T. E. UNANDER, JOHANSEN S.T., BANSAL, K. M., DANIELSON T.J., GOLDSZAL A., and MONSEN, J.I., (2008a), "Multidimensional Simulations of Multiphase Flow for Improved Design and Management of Production and Processing Operation", OTC-19434, Proc. *2008 Offshore Technology Conference,* Houston, Texas, U.S.A., 5–8 May 2008.

LAUX, H., MEESE, E., JOHANSEN S.T., BANSAL, K. M., DANIELSON T.J., GOLDSZAL A., and MONSEN, J.I., (2008b), "Multi-dimensional simulations of slug and slug-like flows in inclined pipes and channels", *Proc. 6th North American BHRG Conference on Multiphase Technology*, June 12th 2008, Banff, pp. 21-36.

MOODY, L. F. (1944), "Friction factors for pipe flow", Transactions of the ASME 66 (8): 671–684.

COLEBROOK, C.F. (1939). "Turbulent flow in pipes, with particular reference to the transition region between smooth and rough pipe laws". Journal of the ICE, Vol 11, Issue 4: pp 133 –156.

PATANKAR, S.V., (1980), "Numerical Heat Transfer and Fluid Flow", McGraw-Hill, New York.

SHEA, R., EIDSMOEN, H., NORDSVEEN, M., RASMUSSEN, J. XU, Z., NOSSEN, J., (2004), "Frequency Prediction Method Comparison", BHRG Multi-phase Production Technology Proceedings, Banff, Canada.

UJANG, P.M., LAWRENCE, C.J., Hale C.P., & HEWITT, G.F. (2006), "Slug initiation and evolution in two-phase horizontal flow", *Int. J. Multiphase Flow*, Vol. 32, pp. 527-552.

ZUKOWSKI, E. E. (1966), "Influence of viscosity, surface tension, and inclination angle on motion of long bubbles in closed tubes". J. Fluid Mech. 25, 821–837.

