NUMERICAL SIMULATION OF SLUG FLOW REGIME FOR AN AIR-WATER TWO-PHASE FLOW IN HORIZONTAL PIPES

Thomas Frank

CFX Development, ANSYS Germany GmbH, Staudenfeldweg 12, D-83624 Otterfing, Germany Phone: +49 (8024) 9054 76, Fax: +49 (8024) 9054 33 Thomas.Frank@ansys.com

ABSTRACT

Fluid mechanics of air-water slug flows in horizontal circular pipes is investigated using detailed, transient, 3-dimensional Computational Fluid Dynamics (CFD) simulations. The inhomogeneous twophase or mixture model combined with the interface sharpening (free surface) algorithm, implemented in the commercial CFD code CFX-5.7, has been used to predict the transition of a segregated air-water flow into slug flow regime. Several cases have been investigated, where the length of the circular pipe segment, the applied boundary conditions (periodic vs. inlet/outlet BC's) and the initial/transient perturbation of the free surface have been varied. With the developed CFD model the slug formation and propagation along the circular pipe can successfully be predicted and can be studied in great detail. Typical slug flow characteristics like e.g. slug frequency, slug length and slug propagation speed have been determined from numerical simulations and are compared to available experimental data. Despite still existing uncertainties in both the experimental data and the numerical model the obtained numerical slug flow characteristics are in reasonably good agreement with the experiments.

KEYWORDS

CFD, two-phase flow, flow regime transition, segregated flow, slug flow, free surface

1. INTRODUCTION

Slug flow is a quite common multiphase flow regime in horizontal pipelines and channels, which can be potentially hazardous to the structure of the pipe system or to apparatus and processes following the slug flow pipe section due to the strongly oscillating pressure levels formed behind liquid slugs. Areas of application are the chemical and process industry as well as safety research and thermo-hydraulic engineering for nuclear power plants. It is the aim of the present paper to study the feasibility and accuracy of CFD simulations for an air-water slug flow in horizontal circular pipes applying different numerical approaches and variations in boundary conditions.

1.1 The phenomena of slug flow formation

For finding a suitable strategy for the numerical simulation of slug flow it is helpful to have a closer look on the physical mechanisms leading to the phenomena of formation of slug flows in circular pipes. A flow in a horizontal circular pipe of diameter D, which is initially filled with 50% volume fraction of the gaseous phase and 50% of the liquid phase, is considered. If the phases are initially at the pipe inlet homogeneously mixed, then demixing due to gravitation and buoyancy occurs within a length of a few pipe diameters (~10D) for an air-water mixture with 1m/s superficial velocities of both phases in the inlet cross-section. Afterwards, phases separate by a free surface, where the upper half of the pipe is occupied by the gaseous phase and the lower half by the liquid phase. The shape of the free surface mainly depends on initial agitation, disturbances arising from the inlet conditions and from the inlet or mixing devices used for the injection of the gaseous phase in the pipe. Further downstream the development of the free surface shape mainly depends on the gaseous and liquid velocities, i.e. from the velocity difference at the interface. Due to wall friction, the liquid phase is decelerated along the

pipe and due to incompressibility and conservation of volume of the liquid phase, the free surface separating both phases is slightly rising with increasing pipe length. This leads to congestion of the cross sectional area above the free surface and a resulting acceleration of the gaseous phase in the upper half of the pipe flow. Consequently, the velocity difference at the phase interface increases. If a critical velocity difference between the two phases is exceeded, the phase interface becomes unstable and wavy structures develop. The acceleration of the gaseous phase in regions of highest liquid wave amplitude leads to a reduction of the gas pressure, which in turn reinforces the build-up of the waves. Finally, this leads to the formation of a liquid slug and complete blockage of the pipe cross-section by the liquid phase. The blockage of the cross-sectional area by the liquid slug gives rise to steep pressure gradients in the gaseous phase. The liquid slug is now driven and accelerated by the upstream gas flow. The observable liquid slug and slug front velocities exceed by far the velocities in the fluid layer in the lower half of the pipe. Depending on the pipe geometry (length and cross-sectional area) and the gas and liquid flow rates, the slug flow can form a stable flow regime, where liquid slugs are moving over long pipe distances.

In other cases depending on the inlet conditions and pipe cross-sectional area, the velocity of the liquid slug front is slower then the wave propagation speed of a surface wave running downstream of the slug front in the pipe direction. In that case the liquid slug disintegrates after a certain distance of propagation due to loss of a critical fluid mass contained in the liquid slug. The pipe cross section is no longer completely blocked by the liquid phase and the gaseous phase can pass over the liquid slug, resulting in a rapid drop of the pressure level behind the liquid slug. This regime usually results in a complete disintegration of the liquid slug and to a repetition of the whole slug formation process further downstream.

From these basic observations on the mechanisms leading to slug flow regime it is obvious, that an attempt of a numerical simulation of this flow process should be based on the 3-dimensional and transient prediction of the two-phase flow. Furthermore, since wall friction and blockage of the crosssectional area by the liquid phase are key factors for the formation of slugs, it can be understood, that a 2-dimensional simulation will often not show slug flow formation for flow conditions, where slug flow can be observed in the experiment, since side wall effects are not taken into account in a 2dimensional simulation. For the same gas and fluid flow rates and for the same cross sectional area flow in cylindrical pipes will earlier tend to slug flow regime than flow in a square channel due to the congestion of the cross-sectional area with increasing liquid level in pipes.

1.2 Numerical simulation of slug flows

Due to the transient and 3-dimensional character of the slug flow regime and the resulting numerical effort only few attempts of numerical simulation have been made. Taking the considerations mentioned above on the slug formation process into account, in principle three different computational approaches can be applied for the simulation of horizontal slug flows:

- "Frozen slug" in a domain with moving wall boundaries, where the absolute value of the prescribed wall velocity is equal to the slug propagation velocity in the pipe. The slug propagation velocity, the slug length and slug period have to be known in advance.
- Transient 3-dimensional simulation in a short computational domain with periodic boundary conditions. A driving pressure force has to be prescribed to compensate the pressure losses due to wall friction. Furthermore it has to be ensured, that the geometrical dimensions of the computational domain do not affect the computed slug flow length and time scales.
- Transient 3-dimensional simulation of slug flow in a long pipe segment with inlet/outlet boundary conditions. In this case the formation of slug flow is largely determined by the exact approximation of the experimental inlet conditions for the numerical simulation. In contrary to the other approaches, no additional experimental knowledge about the slugs is required in order to setup the numerical simulation. Integral parameters like e.g. slug length, period and pressure oscillations are a direct result of the CFD simulation and can be compared with experimental data.

The latter of the three computational approaches provides the highest predictive capability, also it is the most computational intensive approach.

In the past most investigations of the slug flow regime in horizontal pipelines and channels have been carried out on test rigs. Only few publications are known, in which the 2- or 3-dimensional, transient simulation of horizontal slug flows has been attempted. Kunugi et al. (1999) used a multi-

interfaces advection and reconstruction solver (MARS, a volume tracking method) for a direct numerical simulation of a 2-dimensional air-water slug flow in horizontal planar channel. Numerical predictions were carried out for a duct of 0.1m height and 3.0m length. For an initial gas velocity of U_G =9.0m/s, initial gas void fraction of α =0.4 and water at rest growing instabilities developing to slug flow at L~1.7m had been observed in the numerical simulations. Comparison of predicted slug positions with experiments showed good agreement for varying initial gas velocities and void fractions. The reported mesh element size is 10^{-2} m, which results in a quite coarse numerical mesh with respect to a two-phase DNS of only 3.000 mesh elements. Extremely small time steps of 10^{-6} s were used for the transient numerical simulation.

Ujang (2003) provided a 3-dimensional numerical study of the turning of Taylor bubbles in downward directed two-phase flow in inclined circular pipes, i.e. the prediction of an upward moving Taylor bubble in a downward directed water flow. This simulation belongs to the "frozen slug" type, where a moving wall boundary was used in order to keep the Taylor bubble in the computational domain. Additional assumptions of symmetry along the vertical pipe diameter allowed the reduction of the computational domain to a semi-circular pipe segment of 15D with a total mesh resolution of approx. 25.200 grid cells. Simulations were performed with an inhomogeneous two-phase flow model, where Reynolds-averaged Navier-Stokes equations are solved for both phases. A surface sharpening algorithm was used to avoid interface smearing. The velocity of bubble turning has been determined from this 3-dimensional slug flow study and a close agreement with air-water experimental data in a 1 inch diameter pipe for a downward angle of -15° was obtained.

Höhne (2004) presented 3-dimensional simulations of the developing slug flow in a short channel with rectangular cross section using the inhomogeneous two-phase flow mixture model of CFX-5.7, where the general numerical approach is based on the methodology provided in the report of Frank (2004). Experimental data of Vallée & Sühnel (2004) have been used for the transient pressure boundary conditions on the inlet and outlet of the channel. Simulation results show the transient formation of slugs and agree qualitatively well with video observations on the experimental test facility at Research Center Rossendorf, Institute of Safety Research (FZR). Predictions show at the location of pressure transducers in the test section the characteristic strong pressure increase during the transition of liquid slugs. But the predicted amplitude of pressure increase in the gaseous phase showed quantitative differences to the experimental data, which need further investigation.

Other publications are aimed to the numerical simulation of the rise of large Taylor bubbles in a stagnant liquid in vertical pipes. Similar numerical methods like homogeneous or inhomogeneous VOF are applied to this class of gas-liquid flow regimes. The assumption of a stagnant fluid in the pipe leads to certain simplification of the simulation task in comparison with the simulation of horizontal slug flow. Furthermore, the free surface forming the interface between the gaseous and the liquid phases shows a stable shape for Taylor bubbles and transient wave propagation and gas entrainment into the liquid can be neglected in many cases. Tomiyama et al. (1996) used an inhomogeneous VOF method for the prediction of the influence of Eötvos (Eo) and Morton (Mo) numbers on the terminal rise velocities and Taylor bubble shapes. Calculations were performed in a relative coordinate frame ("frozen plug" approach), where a constant downward wall velocity was prescribed. Additionally calculations have been carried out on a 2-dimensional computational domain (r-z) assuming axial symmetry of the Taylor bubble. Results were compared to experimentally observed Taylor bubble shapes obtained by photography and showed good agreement for the investigated range of Eo and Mo numbers.

Similar 2-dimensional simulations of rising Taylor bubbles through stagnant liquids in vertical tubes have been published by Bugg et al. (1998). A homogeneous Volume of Fluid (VOF) method was used for the prediction of Taylor bubble terminal rise velocities and film thickness for $10 \le Eo \le 100$ an $10^{-12} \le Mo \le 10^1$. Predicted axial-symmetric velocity profiles in front and behind the Taylor bubble as well as film velocities have been successfully compared to literature data. Under certain conditions (Fr>0.3) recirculation zones were predicted behind the trailing edge of the Taylor bubble. In reality they will lead to a 3-dimensional fluid flow behind the bubble, which could not be detected by the numerical approach.

3-dimensional transient simulations of Taylor bubble rising and coalescence have been reported by Anglart & Podowski (1999) using the homogeneous VOF method of CFX-4. Taylor bubble shape and terminal rise velocities were successfully predicted and the time-dependent coalescence process of two Taylor bubbles has been studied. Results show the strong 3-dimensional character of the coalescence of Taylor bubbles due to elongation of the trailing bubble from the pipe centerline, which is mainly caused by pressure fluctuations in the bubble wake of the leading Taylor bubble.

2. EXPERIMENTAL INVESTIGATION OF SLUG FLOW

Numerical simulations described in this paper were based on experiments carried out at the Lehrstuhl für Thermodynamik (TD/TUM) at the TU Munich, Germany. A more detailed description of the linear part of the slug flow test section and its instrumentation used at TD/TUM can be found in Lex (2003). The horizontal test section consists of a Pyrex pipe with D=54mm and a pipe length of about L=9.0m, which corresponds to approximately 170 D. Air and water are mixed homogeneously in a pipe T-junction at L=0.0m, while air and water volume flow rates can be adjusted in a certain range. The gas and liquid inlet conditions are monitored by flow meters. The Pyrex glass pipe section allows video observation of the flow development. Pressure can be measured at 9 different locations along the pipe. The gas volume fraction distribution is measured over time at L=8.0m with wire-mesh sensors manufactured by the Research Center Rossendorf (FZR). Their spatial resolution is 16x16 wires over the cross-sectional area of the pipe.



Figure 1: "Virtual top and side view" of slug flow in the horizontal pipe test section reconstructed from wire-mesh sensor data.

The measurements result in time series of gas void fraction distributions over the measurement cross-section. These data allow the prediction of integral and time-averaged values characterizing the slug flow. Furthermore, "virtual side views" of the axial void fraction distribution can be constructed from these data by plotting the sensor data at the vertical symmetry plane over time (see Fig. 1). In contrary to the use of wire-mesh sensors for measurements of fully developed, quasi-stationary void fraction distributions in vertical dispersed bubbly flows, this information from the "virtual side views" for slug flows is not directly comparable with a real side view at a given moment in time obtained from the numerical simulation, since slug flow is permanently evolving over the pipe length. Moreover, similar "virtual side view" plots have to be constructed from the numerical simulation for direct comparison. Time series of pressure in the gaseous phase can be obtained from pressure measurements at different pipe locations. These pressure measurements can be used for comparison with numerical simulations, if they can be synchronized with respect to the time of slug transition at the measurement location.

3. NUMERICAL SOLUTION ALGORITHM

3.1 The Multiphase Flow Model

If we assume, that gaseous and liquid phases in the slug flow regime are fully segregated, then it seems to be the most appropriate and economical approach to use the homogeneous VOF method with shared velocity field assumption for this type of application. In this case only one set of Navier-Stokes equations for the gas-liquid mixture together with two volume fraction equations and turbulence model

equations are solved. However, experimental observations show that on the leading front of a liquid slug the flow tends to form breaking waves, droplets and liquid ligaments. These partial phenomena can lead to gas entrainment into the liquid phase. Since in the homogeneous model both phases share the same velocity field, phases are demixing only through 3-dimensional motion and not by interpenetration of phases. This usually leads to delayed demixing times and a generally different behavior of the multiphase mixture in areas of higher gas entrainment.

Therefore the numerical simulations presented in this work are based on the CFX-5.7 two-fluid (or multifluid) Euler-Euler approach. The Eulerian modeling framework is based on mass-weighted (Favre) averaged mass and momentum transport equations for all phases, gas and liquid. Regarding both phases as continua, which are segregated or mixed at a macroscopic level (mixture model), these equations without mass transfer between phases read:

$$\frac{\partial}{\partial t} (r_{\alpha} \rho_{\alpha}) + \nabla . (r_{\alpha} \rho_{\alpha} \vec{U}_{\alpha}) = 0$$
⁽¹⁾

$$\frac{\partial}{\partial t} \left(r_{\alpha} \rho_{\alpha} \vec{U}_{\alpha} \right) + \nabla \left(r_{\alpha} \rho_{\alpha} \vec{U}_{\alpha} \otimes \vec{U}_{\alpha} \right) = \nabla \left(r_{\alpha} \mu_{\alpha} (\nabla \vec{U}_{\alpha} + (\nabla \vec{U}_{\alpha})^{T}) \right) - r_{\alpha} \nabla p + r_{\alpha} \rho_{\alpha} \vec{g} + \vec{F}_{D}$$
(2)

where U_{α} represents the velocity field of phase α (with α =L for the liquid and α =G for the gaseous phase), r_{α} - volume fraction, ρ_{α} - density, p - pressure, μ_{α} - viscosity, g - gravitation and F_D represents the drag force due to momentum transfer at the interface between phases. The liquid and gaseous phases are coupled through the interfacial drag F_D , which can be expressed through a drag coefficient C_D and the interfacial area density $A_{\alpha\beta}$:

$$F_{D,\alpha} = C_D \rho_{\alpha\beta} A_{\alpha\beta} (U_\beta - U_\alpha) \tag{3}$$

with:

$$C_D = 0.44 \quad , \quad \rho_{\alpha\beta} = r_\alpha \rho_\beta + r_\beta \rho_\alpha \quad , \quad A_{\alpha\beta} = \frac{6r_\alpha r_\beta}{d_{\alpha\beta}} \quad , \quad d_{\alpha\beta} = r_\alpha d_\beta + r_\beta d_\alpha \tag{4}$$

The user supplied phase specific length scales have been set to $d_{\alpha}=d_{\beta}=0.001m$ under the assumption, that entrained droplets or gas bubbles are approximately of this size for the given type of flow. Similar results have been obtained in the numerical predictions using a drag law derived by Ishii (1990, p. 37) for gas-liquid slug flows instead of Newton's drag law:

$$C_D = 9.8 \left(1 - r_\beta\right)^3 \tag{5}$$

where r_{β} is the volume fraction of the gaseous phase. Also it is quite uncertain whether this macroscale drag law can be applied to the microscale interfacial processes between the two phases.

For the present simulations the two-phase mixture model described above have been combined with the CFX-5 free surface model. Hereby the surface tension at the interface has been neglected. The free surface model implemented in CFX-5 consists of a numerically compressive advection scheme, which is often referred to as interface capturing. The model applies a controlled downwinding scheme (based on the high-resolution TVD advection scheme allowing for blending factors $\beta>1$) to the volume fraction equations in order to avoid smearing of the interface between the two phases due to numerical diffusion of the solution algorithm. Here a blending factor of $\beta=0$ recovers the first-order upwind scheme, with $\beta=1$ this scheme is a second-order upwind biased scheme and with $\beta=2$ would result in an advection scheme, which is as anti-diffusive (compressive) as the first-order upwind scheme is diffusive. More in-depth details of the CFX-5 free surface algorithm can be found in Zwart et al. (2003) and Zwart (2005).

In principle it is possible to apply a homogeneous turbulence model to both phases. In the present case no or too small damping of the turbulence was observed at the interface, particularly for the gas phase. Also with fluid-dependent turbulence modeling applied to both phases separately (e.g. a k- ε or k- ω model applied for each phase) the turbulence properties of one phase propagate nearly without any

damping at the free surface into the area occupied by the other phase. A secondary effect of these problems in turbulence modeling for fully segregated flows is the occurrence of non-physical gas velocities at the free surface. In order to improve the turbulence modeling for the segregated flow simulation a fluid-dependent k- ω turbulence models for each phase was used. Additionally damping of turbulent diffusion at the interface between the two phases has been taken into account by an appropriate source term in the ω -equation of the following form:

$$S_{\omega}|_{\alpha} = r_{\alpha}r_{\beta}0.075\rho_{\alpha}\omega_{D\alpha}^{2} \quad , \qquad \omega_{D\alpha} = C_{D\omega}\frac{6\nu_{\alpha}}{d_{\alpha\beta}^{2}} \quad , \qquad \mu_{i\alpha} = \rho_{\alpha}\frac{k_{\alpha}}{\omega_{\alpha}} \quad (6)$$

where $C_{D\omega}$ has been set to a "large" value (in case of the present simulations $C_{D\omega}=1000.0$ was used).

For the gaseous and liquid phases standard physical properties for air and water at 25°C and at a reference pressure of 105500 Pa were used. Finally, the coupled system of transport equations was solved with the commercial CFD package CFX-5.7 using the high resolution advection scheme and a second order backward Euler time integration scheme.

Mesh description	Pipe diameter D	Pipe length L	Spatial mesh resolution		No. of
			No. of grid cells in pipe cross-section	No. of grid cells along the pipe	grid cells
Grid 1 , 4m long pipe (periodic boundary conditions)	0.054m	4.0m	249	250	73.500
Grid 2 , 8m long pipe (inlet/outlet boundary conditions)	0.054m	8.0m	249	500	147.000

Table 1: Parameters of finite volume meshes.

3.2. Finite Volume Meshes and Boundary Conditions

For the slug flow simulation in a horizontal pipe symmetry with respect to the vertical plane at z=0.0m has been assumed. Therefore hexahedral meshes in a half cylinder have been generated using the mesh generation tool ICEM/CFD-Hexa. For the different numerical simulations described in more detail in the next section, two different numerical meshes have been generated. Geometrical dimensions and grid resolutions of the numerical meshes were set as shown in Table 1.

Two different types of numerical simulations have been carried out. For the first slug flow simulation on Grid 1 periodicity of the flow was forced by assigning periodic boundary conditions to the half-pipe cross sections at x=0.0m and x=4.0m. In a second series of simulations (Grid 2) these periodic boundary conditions have been replaced by inlet and outlet boundary conditions, which will be described later. The wall of the horizontal pipe has been set as hydraulically smooth walls with non-slip boundary conditions applied for the gaseous and liquid phases. For the transient simulations velocity fields of air and water phases were initialized with their inlet values in the area, where the phase is present at T=0.0s.

4. NUMERICAL SIMULATION OF HORIZONTAL GAS-LIQUID SLUG FLOW

4.1. Slug Flow Simulation with Periodic Boundary Conditions

Initially shorter pipe segments with L=1.0m and L=2.0m had been used in this study for slug flow simulations with periodic boundary conditions. But it was found, that these simulations showed a significant influence and interference of the periodic boundary conditions with the developing flow regimes. So finally, Grid 1 was selected for the simulations with periodic boundary conditions. The length of L=4.0m was chosen in order to minimize the disturbance of slug formation due to prescribed

periodicity. This selection was based on the experimental observation of a mean slug length of about 1.8m for the given inlet and flow conditions. Air and water were initially fully segregated. The volume fraction distribution of the gaseous and liquid phases in the computational domain was initialized with a mean gas and liquid volume fraction of 0.5 and a sinusoidal disturbed free surface with a liquid level y_I following the function:

$$y_I = y_0 + A_I \sin\left(2\pi \frac{x}{p_I}\right) \tag{7}$$

with $y_0=0.0m$, $A_I=0.1D$ and $p_I=0.5L$. The wavelength again has been determined in similarity to the experimental observations for the mean slug length by Lex (2003). The initial velocities of the gaseous and liquid phase were set to $U_G=U_L=2.0m/s$ ($v_G=v_L=1.0m/s$ gas and liquid superficial velocity). The driving pressure force was again corresponding to a mean pressure drop in the experiments of approx. $\Delta P=800$ Pa/m.



Figure 2: Slug flow simulation in a 4m long pipe with periodic boundary conditions; iso-surface corresponds to liquid volume fraction of 0.5 (T=0.57s)



Figure 3: Superficial velocity distributions for the gaseous and liquid phases in the vicinity of the liquid slug (x=1.0m-3.0m, T=1.24s)



Figure 4: Slug propagation in a 4m long pipe with periodic BCs; iso-surface corresponds to liquid volume fraction of 0.5 (view left to right, top to bottom; x=1.0m-3.0m, T=1.21s-1.34s); symmetry plane at z=0.0m shows the pressure distribution.

In the transient CFD simulation, the preset sinusoidal free surface structure was quickly transformed into a free surface showing two wave structures (Fig. 2) within the computational domain of L=4.0m with an almost flat phase interface between these two surface waves. The formation of the first liquid slug started after T=1.21s at x~1.4m. The time series of this slug formation is shown in Fig. 4 for the time period of T=1.21s-1.34s. The slug shows a complete blockage of the pipe cross-sectional area but has duration of only about 0.05s. A new liquid slug is build up afterwards. The semi-circular pictures in Fig. 4 show the void fraction distribution in cross sections at x=1.0m, x=2.0m and x=3.0m. At the central symmetry plane z=0.0m the pressure distribution is visualized. The location of the isosurface separating the two phases corresponds to a predicted volume fraction of 0.5. Due to the sharpness of the free surface in the slug flow. Additionally, Fig. 3 shows the superficial velocity distributions for the gaseous and liquid phases for T=1.24s in the vicinity of the liquid slug. It can be observed that the superficial air velocity is about 2 times higher then the superficial water velocity. While the water velocity is at a quite low level of about v_L~2.5m/s in the liquid bottom layer, the fluid gets strongly accelerated in the liquid slug to velocities of about 5-6m/s.

The 3-dimensional CFD simulations on the 4m long pipe segment with periodic boundary conditions have clearly shown the feasibility of slug flow simulation with the available multiphase flow model. The combination of the multiphase mixture model with the free surface model ensures the sharpness of the phase interface over long computational times and with complex transient transformation of the free surface separating gas and liquid. Nevertheless, the periodicity assumption for a pipe segment of given length has also two major disadvantages. The numerical simulation needs an a priory known and prescribed pressure drop. In addition, the length of the pipe segment used in the simulation might affect the length scale and period of the liquid slugs in the computational domain. Both disadvantages can be avoided with the following numerical approach.



Figure 5: Pressure distribution for a liquid slug at T=4.04s. Slug visualized as iso-surface of water volume fraction at $r_L=0.5$

4.2. Slug Flow Simulation with Inlet/Outlet BCs and a Periodically Agitated Free Surface

The 3-dimensional slug flow simulations with inlet and outlet boundary conditions were carried out on Grid 2. The length of the pipe segment of L=8.0m almost corresponds to the length of the experimental test section of TD/TUM. Again the volume fraction distribution for both phases in the computational domain was initialized in correspondence to the sinusoidal disturbance of the free surface separating gas and liquid as it was used in the previous case. Four complete periods of the sinus function fit into the computational domain, since the same parameters of the sinusoidal function for the initial liquid level in the pipe were used. In contrary to the periodic BC flow simulation in this case transient inlet boundary conditions for gas and liquid velocities and volume fraction distribution at the inlet cross section were prescribed as a function of time. For the present simulation, constant superficial inlet velocities for gas and liquid phases $v_G=v_L=1.0m/s$ have been used (corresponds to

local gas and liquid velocities of $U_G=U_L=2.0$ m/s where the phases are present), while a transient liquid level at the inlet cross section was set following the function:

$$y_I = y_0 + A_I \sin\left(2\pi \frac{V_I \cdot t}{p_I}\right) \tag{8}$$

While the parameters y_0 , A_I , p_I were set to the same values as used for the domain initialization ($y_0=0.0$ m, $A_I=0.25$ D, $p_I=0.25$ L), the characteristic interface velocity was set to $V_I=U_G=U_L=2.0$ m/s. In using these parameters the resulting perturbation of the free surface at the inlet cross section reproduces the initially set sinusoidal agitation of the free surface in time. The time averaged mean gas and liquid void fraction in the pipe is thereby $r_G=r_L=0.5$ for both phases. Average static pressure outlet conditions with a relative pressure of $P_{rel}=0.0$ Pa have been applied to the downstream outlet cross section of the pipe. The transient simulation has been carried out with a constant time step of dt=0.005s and the solution for a total time of T=7.0s (1400 iterations) has been computed.



Figure 6: Superficial liquid and gas velocity distributions respectively for a liquid slug at T=4.04s.

During the first 2-3s of the transient flow simulation the predicted flow field was characterized by a gravitational settling of the preset initial volume fraction distribution with increasing pipe length. The initial sinusoidal structure of the free surface was almost leveled out (or was at least substantially decreased in amplitude) for x>4.0m. At later times it could be observed that the velocity of the liquid phase decreased with increasing pipe length from $U_L=2.0$ m/s at the inlet cross-section to $U_L\sim0.82$ m/s at the outlet cross-section at x=8.0m. Due to volume conservation this decrease in liquid velocity due to wall friction is accompanied by a rise of the water level with increasing pipe length. This further leads to an acceleration of the gaseous phase in the upper half of the pipe due to the reduction of the cross-sectional area available for the gas flow.

First slug formation occurs at $x \sim 3.8$ m after approx. 670 time steps (T=3.35s). The first stable liquid slug is formed after T=4.04s at approx. $x \sim 4.0$ m. Also the shape of the liquid slug front and tail is continuously changing with its propagation along the pipe. The slug remains stable and covers most of

the time the whole cross-sectional area of the pipe. Towards the end of the pipe the slug length further increases. Figs. 5 and 6 show the pressure and the gas and liquid velocity distributions over this liquid slug at T=4.04s in more detail. The sharp pressure increase over the liquid slug front can be clearly observed from Fig. 5.



Figure 7: Time series of liquid slug propagation from T=3.4s to T=4.7s.

Finally, Fig. 7 shows a time series of pictures from the liquid slug propagation from x=4.0m to the outlet cross-section of the pipe segment at x=8.0m. The picture series covers the time interval between T=3.4-4.7s. The liquid slug length at the end of the pipe segment was about 0.25m. From the observed liquid slugs a mean slug period of approx. ~2.7m and a slug propagation velocity of approx. ~2.7-3.1m/s could be determined, although the small number of slugs does not allow the calculation of reliable mean quantities. From the experiment an averaged slug period of ~1.8m and an averaged propagation speed of ~2.7m/s has been determined, which is in fairly good agreement. Furthermore, the numerical predictions showed strong transient pressure changes corresponding to the propagation of the front of single plugs along the pipe with maximum relative pressure peaks of about 2000-2800

Pa. The resulting averaged pressure drop from the numerical simulations of ~500-700 Pa/m compares quite well to a ~700 Pa/m averaged pressure drop determined at the TD/TUM test section for developed slug flow. Again, these values are subject to larger uncertainties due to the comparable small absolute time frame of the numerical simulation and the limited number of observed slugs propagating through the pipe segment in the computed time.

5. CONCLUSION

In a series of numerical investigations the formation and propagation of slug flows in horizontal circular pipes have been simulated using the commercial CFD software package CFX-5.7. The formation and propagation of liquid slugs in a horizontal pipe could be successfully predicted with the two-phase inhomogeneous mixture model in combination with the free surface model of CFX-5. The flow regime transition from a fully segregated to slug flow could be observed in the numerical predictions with both periodic and inlet/outlet boundary conditions. Once slug flow regime has been developed the observable liquid slugs were stable in time and showed qualitatively similar flow behavior and characteristics (slug frequency, slug length, slug propagation speed) as in the experiments, although a real quantitative comparison could not yet be carried out due to the uncertainties of flow conditions as they were realized in the experiments and due to the uncertainties in the numerical models.

It was found that the process of transition from segregated to slug flow regime is mainly determined by the wall friction of the liquid phase. For simulations using inlet/outlet boundary conditions it was further observed, that the formation of slug flow and the time and location of the first occurrence of a liquid slug strongly depends on the agitation or perturbation of the inlet boundary conditions. In numerical simulations with prescribed constant inlet velocities and volume fractions without any transient perturbation of the free surface between water and air very long pipe segments were necessary in the numerical simulations in order to observe the formation of liquid slugs. But it has to be taken into consideration, that such numerical inlet boundary conditions are quite artificial for real experiments, since it is almost impossible to experimentally realize these kinds of completely undisturbed inlet boundary conditions. Usually, in laboratory experiments the two phases are partially mixed at some cross section further upstream of the test section. In most cases velocities and/or volume fraction ratio of the phases are at least subject to certain fluctuations in time due to the involved technical equipment like pumps, fans, flow meters, bends in the upstream pipe work, etc. resulting in certain disturbance of the inlet boundary conditions. Therefore it will be of greatest importance for the setup of future validation experiments for slug flows to realize fully controlled and monitored inlet boundary conditions, which can then be used in numerical validation testcases.

Furthermore, it was found that the length of the computational domain plays an important role in slug formation. Even with the L=8.0m long pipe segment most numerical simulations carried out in this study were still affected by the pressure fluctuations caused by liquid slugs leaving the computational domain through the outlet cross section. This effect was less pronounced in the CFD simulations with periodic boundary conditions. The effect of the downstream pressure fluctuations is also reduced, if the simulated pipe segment is long enough for the propagation of multiple liquid slugs.

Further experimental research and model development is necessary for the determination of interfacial drag and turbulence damping at the free. Detailed experimental investigations on slug flows are required to obtain reliable data for CFD code and multiphase flow model validation.

ACKNOWLEDGEMENT

This research has been supported by the German Ministry of Economy and Labour (BMWA) in the framework of the German CFD Network on Nuclear Reactor Safety Research.

REFERENCES

- Anglart H., Podowski M.Z. 1999. Fluid mechanics of Taylor bubbles and slug flows in vertical pipes, 9th Int. Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-9), San Francisco, CA, October 3-8. 1999, pp. 1-14.
- Bugg J.D., Mack K., Rezkallah K.S. 1998. A numerical model of Taylor bubbles rising through stagnant liquids in vertical tubes, Int. J. Multiphase Flow, Vol. 24, No. 2, pp. 271-281.
- Frank Th. 2003. "A review on advanced Eulerian multiphase flow modeling for gas-liquid flows", Technical Report, CFD-Network "Nuclear Reactor Safety", ANSYS Germany, pp. 1-23.
- Frank Th. 2003. "Numerical simulations of multiphase flows using CFX-5", German CFX Users Conference, Garmisch-Partenkirchen, Germany, November 3.-5. 2003.
- Frank Th. 2004. Numerical simulation of slug flow in horizontal pipes using CFX-5 (Validation testcase HDH-01/1), Technical Report, CFD-Network "Nuclear Reactor Safety", ANSYS Germany, pp. 1-52.
- Höhne Th. 2004. *CFX-5 Calculation for the horizontal air/water flow channel*, 2nd Joint FZR & CFX Workshop on Multiphase Flows 'Simulation, Experiment and Application', Dresden, June 28.-30. 2004.
- Ishii M. 1990. *Two-fluid model for two-phase flow*, Multiphase Science and Technology, Vol. 5, Edited by G.F. Hewitt, J.M. Delhaye, N. Zuber, Hemisphere Publishing Corporation, pp. 1-64.
- Kunugi T., Banat M., Ose Y. 1999. "Slug-plug flow analysis of stratified multiphase flows in a horizontal duct", 7th Int. Congress on Nuclear Engineering, (ICONE), Tokyo, Japan, April 19-23 1999, Paper-No. 7027, pp. 1-10.
- Lex Th. 2003. *Beschreibung eines Testfalls zur horizontalen Gas-Flüssigkeitsströmung*, Technische Universität München, Lehrstuhl für Thermodynamik, Internal Report, pp. 1-3.
- Sühnel T. 2003. *Aufbau und Inbetriebnahme eines horizontalen Luft-Wasser-Strömungskanals*, Diploma Thesis, Institute of Safety Research, FZ Rossendorf, Dresden, Germany, pp. 1-108.
- Tomiyama A., Sou A., Sakaguchi T., 1996. *Numerical simulation of a Taylor bubble in a stagnant liquid inside a vertical pipe*, JSME Int. Journal, Series B, Vol. **39**, No. 3, pp. 517-524.
- Ujang P.M. 2003. A three-dimensional study of Taylor bubble turning in two-phase downflow, Second MIT Conference on Computational Fluid and Solids Mechanics, Elsevier Science Ltd., Ed.: K.J. Bathe, pp. 1176-1180.
- Vallée Ch., Sühnel T. 2004. *Stratified flow in a horizontal air-water flow duct*, 2nd Joint FZR & CFX Workshop on Multiphase Flows: Simulation, Experiment and Application, Dresden, June 28.-30. 2004.
- Zwart P.J., Scheuerer M., Bogner M. 2003. Free surface flow modelling of an impinging jet, ASTAR Int. Workshop on Advanced Numerical Methods for Multidimensional Simulation of Two-phase Flows, GRS, Garching, Germany, September 15-16, 2003, pp. 1-12.
- Zwart P.J. 2005. Modelling of Free surface flows and cavitation, VKI Lecture Series, Industrial Two-Phase Flow CFD, Bruessels, Belgium, May 23-27 2005, pp. 1-25.
- Zwart P.J. 2005. *Industrial CFD applications of free surface problems and cavitation*, VKI Lecture Series, Industrial Two-Phase Flow CFD, Bruessels, Belgium, May 23-27 2005, pp. 1-29.