TransAT Report Series
– Presentations –

TransAT for Oil & Gas
Multiphase Flow Simulation in Pipes & Risers

Ascomp Switzerland
Edited by: Dr Djamel Lakehal
Release date: Sep, 2014.
References: TRS-P/ 06-2014
Table of Contents

1. Introduction .................................................................................................................. 2
2. The Numerical Approach .............................................................................................. 3
3. Model Validation: Kelvin-Helmholtz Instabilities in Thorpe’s Experiment .................. 3
   3.1. Problem Description .................................................................................................. 3
   3.2. Simulation Setup & Results ....................................................................................... 4
4. Model Validation: 2d Slug Flow Formation in The Hawac Experiment ....................... 6
   4.1. Problem Description .................................................................................................. 6
   4.2. Simulation Setup & Results ....................................................................................... 7
5. Slug Formation in Condensation-Induced Waterhammer ............................................... 8
   5.1. Problem Description .................................................................................................. 8
   5.2. Problem Setup ........................................................................................................... 9
   5.3. Simulation Results .................................................................................................... 9
6. Slug Formation at the Wasp Facility ............................................................................. 10
   6.1. Problem Description .................................................................................................. 10
   6.2. Problem Setup .......................................................................................................... 11
   6.3. Simulation Results (2D) ............................................................................................ 11
   6.4. Simulation Results (3D) ............................................................................................ 12
7. Droplet Entrainment & Detachment ............................................................................. 15
   7.1. Background ............................................................................................................... 15
   7.2. Problem Definition ................................................................................................... 15
   7.3. Simulation Setup & Results ...................................................................................... 15
8. Multiphase Flow in Vertical Pipes ................................................................................ 18
   8.1. Problem Description and Model Set-up ....................................................................... 18
   8.2. Bubbly Flow Simulation Results .............................................................................. 19
   8.3. Slug Flow Simulation Results .................................................................................... 20
   8.4. 3D Annular Flow Simulation: Coarse Grid .............................................................. 22
   8.5. 3D Annular Flow Simulation: Fine Grid .................................................................... 24
9. Conclusions .................................................................................................................... 27

References .......................................................................................................................... 27
1. Introduction

As stated in the Introduction, gas-liquid flow in pipes is of great practical importance in petroleum engineering, in particular for separation and transportation. Mixtures of gas and liquids (light and heavy components of oil, solid particles, hydrates, wax, condensate and/or water) are produced and transported together under various topologies (e.g. bubbly, slug, annular, mist) (Fig. 1). In addition, the relative volumetric fraction of the phases can change along the pipes either because of heat addition, heat exchange between the phases or flashing due to depressurization. In vertical pipe flows, e.g. risers, the flow regime identification (up to three main phases, plus when possible sand and hydrates) is critical for the success of drilling and production. The main task in modeling multiphase pipeline flows is the identification of the flow-regime map.

Current predictive tools for multiphase flow and heat transfer in pipes are based on the two-fluid, six-equation model, in which the conservation equations are solved for each phase. In the oil and gas industry this model is reduced to 1D, and is commonly referred to as the "Mechanistic Model". Solution of the Mechanistic Model equations requires specification of closure relations for flow characteristics such as local velocities, wall shear stress, liquid holdups, etc. These closure relations carry the largest uncertainties in the model and are typically empirical, making use of over-simplified assumptions; in particular, the geometry of the vapor/liquid interface is always idealized, e.g. spherical or bullet-shaped bubbles, smooth or sinusoidal wavy liquid films, spherical or elliptic droplets. The physical reality of the situation is much more complex, as shown in Fig. 1, displaying the various flow regimes encountered in 'controlled' laboratory experiments. Furthermore, the closure relationships are often developed from low pressure, small diameter pipe (typically 25-75 mm) data using synthetic oil and air, which does not simulate actual field conditions, making upscaling to predictive codes highly uncertain. All these uncertainties in the closure relations reflect on the overall accuracy of the predicted figure of merit, e.g. pressure drop. For example, the codes adopting the Mechanistic Model and used by the oil industry could predict the pressure drop in a vertical riser with an error of more than 60% (Belt et al. 2011). The closure relations are flow regime dependent and it is well known that flow regimes in large pipes (300 mm diameter, like deep-sea riser pipes) differ significantly from those in smaller pipes. For example, slug flow is replaced by cap flow in large pipes because of large-bubbles instabilities. The complexity increases when more than two phases evolve in the pipe, e.g. gas, water and oil. In this case the flow regime map is expected to feature a broader domain for churn and annular flow, the topology of which remains difficult and expensive to investigate experimentally.

Figure 1: Flow regimes in a vertical pipe with gas superficial velocity (Prasser et al., 2003)
In horizontal pipe flows, the main open issue requiring better understanding is the transition from stratified flow to slug flow (Valluri et al., 2008). Slug flow is a commonly observed pattern in horizontal and low upslope gas liquid flows. The regime is associated with large coherent disturbances, due to intermittent appearance of aerated liquid parcels filling the pipe cross-section. As these aerated liquid parcels travel downstream the pipe, large pressure fluctuations and variations in flow rates could occur, which could affect process and separation equipment. Our recent CMFD results (Lakehal et al., 2012) of flow transition in horizontal pipes has clearly shown the importance of relying on advanced 3D simulation techniques, and has shed light on subtle mechanisms in association with surface deformation, sealing and slug displacement.

2. The Numerical Approach
The CMFD code TransAT© (2013) developed at ASCOMP is a multi-physics, finite-volume code based on solving multi-fluid Navier-Stokes equations. The code uses structured meshes, though allowing for multiple blocks to be set together, MPI and OpenMP parallel based algorithms are used in connection with multi-blocking. The grid arrangement is collocated and can thus handle more easily curvilinear skewed grids. The solver is pressure based (Projection Type), corrected using the Karki-Patankar technique for subsonic to supersonic compressible flows. High-order time marching and convection schemes can be employed; up to third order Monotone schemes in space. Multiphase flows can be tackled using (i) interface tracking techniques for both laminar and turbulent flows (Level Set, VOF with interface reconstruction, and Phase Field), (ii) N-phase, phase-averaged mixture model with Algebraic Slip, and (iii) Lagrangian particle tracking (one-to-four way coupling). As to the level set, use is made of the 3rd order Quick scheme for convection, and 3rd order WENO for re-distancing. Mass conservation is enforced using global and local mass-conserving schemes (Lakehal et al. 2002). To mesh complex geometries, use is made of the Immersed Surfaces Technology (IST) developed by implemented in the code (TransAT, 2013).

3. Model Validation: Kelvin-Helmholtz Instabilities in Thorpe’s Experiment

3.1. Problem Description

Thorpe’s (1969) experiment is a setup enabling to observe and characterize Kelvin-Helmholtz instabilities in an interfacial, stratified two-fluid flow. The complete details about the experiment and comparisons with CFD results can be found in Bartosiewicz et al. (2008). The experiment consists of a rectangular channel (Fig. 2) half-filled by water and half-filled by paraffin, with fluid material properties $\rho_1, \rho_2 = 783.1000 \, kg/m^3$, $\mu_1, \mu_2 = 0.001 \, Pa.s$, and surface tension $\sigma = 0.032 \, N/m$. The channel is then gradually tilted at an angle of $\beta = 4.12890$ until surface instabilities develop and grow, before decaying. The authors of the experiment have changed the channel height (3 and 5cm) and found no appreciable differences as to the
main flow-instability parameters, including time for onset, critical wave length, wave number wave amplitude and wave speed \((t_{\text{onset}}, \lambda_c, k_c, 2a_{\text{max}}, c)\), respectively. Thorpe (1969) took ten pictures of the interface, separated by a time interval of 0.059 s. The first picture is taken at a time of onset of the instability has been observed. This time is \(t_{\text{onset}} = 1.88 \pm 0.07s\) and includes half the time taken to tilt the channel (about 0.25 s). In his paper, Thorpe claims that this uncertainty might be even larger. The most unstable wave-number is estimated with the distance between two wave crests (Fig. 2) to be equal to \(\lambda_c = 2.5 - 4.5\) cm (uncertainty raises from the different critical wavelengths observed under the same operating conditions). After the onset of the instability, Thorpe observed the growth of the waves for approximately 0.52 seconds, before rolling up. At this time the amplitude of the waves was about \(2a_{\text{max}} = 6-8\) mm (Fig. 2). The downward wave speed was also measured to 2.6 cm/s.

UCL Belgium (Schrooyen & Thiry, 2010) has reproduced the experiment of Thorpe, using more advanced flow acquisition material and control. This is now called the “The refurbished” Thorpe experiment”. Novel features of this experiment compared to the previous one include: (i) calculated acceleration ramp to minimize initial perturbations, (ii) high speed camera, (iii) PIV (2D and stereoscopic), and (iv) fluids fully characterized in-house (surface tensions, densities, viscosities). The measured flow-instability characteristics are shown in Table 1. The main differences with the original Thorpe experiment relate to the values of the most unstable wave-number and time for onset of the instability.

### 3.2. Simulation Setup & Results

First qualitative results are presented in Fig (3). Use was made of the TransAT CMFD code, using for this flow the Level Set technique to track the interface. High-order schemes were employed for space discretization (Quick) and interface re-distancing (WENO). The time-stepping was adaptive and second-order implicit. The pressure solver was GMRES with SIP as pre-conditioner. The grid employed consisted of 1831x31 cells to cover a calculation domain of \(1.8\) m \(x\) \(0.03\) m. The liquid-oil interface was initialized as a sine-wave with different modes with amplitude of \(0.2\) mm. The fluids properties were set as specified in the experiment, and the channel was then tilted at an angle of \(\beta = 4.12890\).

It is perhaps useful to note that this problem is dependent on the way initial conditions are set. Several trials were performed before the right ingredients were found for a workable test-case. Although this piece of work lacks a detailed flow analysis as has been done previously, the results obtained compare very well with the experiment (Fig. 3). The first panels up to \(t=2.29\) s reflect the period of time during which the interface builds its inherent instability. The panels for \(t=2.09\) and 2.19 s are important ones, since they reflect the onset of the KH instability. TransAT simulations predict perfectly the trend, but with additional sub-modes that cannot be visualized in the experiment. Beyond the point of onset of instability, the prediction of the instability characteristics (e.g. inclined steep, travelling waves) is overall excellent, in particular for the wave amplitude and time for onset, as shown in Table 1. Finally, while Thorpe observed that the wave ceases to grow at \(t_{\text{decay}} \approx 2.4\) s, the experiments show that this occurs slightly later, at \(t_{\text{decay}} \approx 2.59\) s, which is the outcome of the simulations by TransAT, a difference that can only attributed to the difference in fluid properties.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>(\lambda_c, [cm])</td>
<td>2.5 - 4.5</td>
<td>3.32 - 4.45</td>
<td>4.1 - 4.7</td>
</tr>
<tr>
<td>(k_c, [1/m])</td>
<td>139.6 - 251.3</td>
<td>141.2 - 189.2</td>
<td>133.7 - 153.2</td>
</tr>
<tr>
<td>(a_{\text{max}}, [mm])</td>
<td>6 - 8</td>
<td>6.1 - 7.3</td>
<td>6.05 - 7.23</td>
</tr>
<tr>
<td>(t_{\text{onset}}, [s])</td>
<td>1.88 +/- 0.007</td>
<td>2.09 - 2.19</td>
<td>2.1 - 2.2</td>
</tr>
</tbody>
</table>

Table 1: Flow instability characteristics, experiments vs. CMFD
Figure 3: Comparison of interfacial model evolution and propagation (TransAT (b) vs. Experiment performed at UCL (a))
4. Model Validation: 2d Slug Flow Formation in The Hawac Experiment

4.1. Problem Description
The Horizontal Air/Water Channel (HAWAC) shown in Fig. 4 was devoted to conduct co-current flow experiments. A special inlet device provides defined inlet boundary conditions by separate injection of water and air into the test-section. A blade separating the streams can be moved up and down to control the free inlet cross-section for each phase (Fig. 5), which influences the evolution of the flow regime. The cross-section of this channel is smaller than the one used earlier by Vallée et al. (2008), with dimensions 100 x 30 mm² (height x width). The test-section is about 8 m long, and therefore the length-to-height ratio L/h is 80, or L/Dh = 173 (Dh is the hydraulic diameter).

![Figure 4: Schematic view of the horizontal channel with inlet device for a separate injection of water and air into the test-section](image)

![Figure 5: Schematic of the inlet device](image)

The inlet device (Fig. 5) was designed for separate injection of air (through the upper part) and water (through the lower part) into the channel. As the inlet geometry introduces perturbations into the flow, four wire-mesh filters were mounted in each part of the device. The filters were made of stainless steel wires with a diameter of 0.63 mm and have a mesh size of 1.06 mm. The wire-mesh filters were used to provide homogenous velocities at the inlet, and to produce a pressure drop that attenuates the effect of the pressure surge created by slug flow on the fluid supply systems.

Air and water streams merge at the trailing edge of a 500 mm long blade separating the phases downstream of the filter segment. The free inlet cross-section for each phase can be controlled by inclining the blade up and down. In this way, the perturbation caused by the first contact between gas and liquid can be either minimized or, if required, a perturbation can be introduced (e.g. hydraulic jump).
4.2. Simulation Setup & Results

Figure 6: Interface contour predicted by TransAT at JL = 1.0 m/s and JG = 5.0 m/s, with ∆t = 50 ms (depicted large part of the channel)

Figure 7: Interface contour predicted by TransAT (same conditions): flow zone at the sealing position

Figure 8: Measured sequences at JL = 1.0 m/s and JG = 5.0 m/s with ∆t = 50 ms (depicted part of the channel: 0 to 3.2 m from inlet)

HAWAC was simulated by TransAT using the level set technique to track the interface. The same space-time discretization and interface re-distancing schemes as in the previous case were employed. Time-stepping was adaptive and second-order implicit. The grid employed consisted of 1831 x 31 cells covering only a 2D portion (in length) of the calculation domain. The interface was initialized at rest. The fluids properties were those of water and air. As to turbulence, use was made of the V-LES, with a filter scale of 10% the channel height, with the k-ε model being employed to model the effect of sub-scale turbulence.

The results shown in Fig. 6 depict clearly the process evolving from an interface populated by ripples and wrinkles to sealing and slugging. The process is very precise, in that sealing occurs only after 3 waves have merged together in the direction of the flow. In all cases, the interface ripple-scale instabilities are predicted everywhere in the channel, including upstream the main perturbations, similar to the picture provided by the experiment (Fig. 8).

Fig. 7 shows the sealing mechanism over a larger portion of the channel, equivalent to Fig. 8. Initial ripples (developed in the course of the simulation and not imposed) develop into
larger waves, which when travelling at different speeds with the main flow come to merge together, creating a blocking effect leading to sealing or slugging. The slug travels at a speed dependent on the mixture velocity and density difference. The same process repeats itself with new flow events: early ripple-scale instabilities, to long waves, to slugs. The figure also shows other subtle details not visible in the phase-averaged simulations, including the detachment of some ligaments from the upcoming waves and far downstream from the slugs. Because of two-dimensional calculation, the sealing/slugging mechanism could not be reproduced in detail: slugging in a 2D channel tends to produce a higher water cut, since the waves cannot spread laterally. Be it as it may, the comparison with the former calculations and experiment is rather good.

5. Slug Formation in Condensation-Induced Waterhammer

5.1. Problem Description

The experiment of Martin et al. (2007) was designed for the purpose of investigating the phenomenon of condensation-induced waterhammer in an ammonia refrigeration system. Waterhammer was initiated by introducing warm ammonia gas over static subcooled ammonia liquid placed in a horizontal carbon steel pipe 6.0 m in length. The only data used for comparison with the simulation include the isothermal subsonic case, with no condensation. The apparatus was designed to simulate an industrial environment whereby ammonia liquid is standing in a partially-filled horizontal pipe in thermal equilibrium with ammonia gas above it. The essential elements of the test setup consist of a horizontal pipe and a high pressure tank containing warm ammonia gas, as shown in Figure 9. The test pipe has a nominal diameter of 150 mm and length of 6 m, made of 80 carbon steel, having internal diameter of 146.3 mm, and wall thickness 11.0 mm. The pressure tank contains warm ammonia gas on top of liquid in thermal equilibrium at ambient conditions inasmuch as the entire test facility was outdoors. Between the pressure tank and the test pipe were three valves and a metering orifice. The angle valve remained fully open, while flow was initiated by a solenoid valve for a given position of the throttle valve. The ammonia in the insulated test pipe was introduced from an ancillary system containing a compressor, an auxiliary tank, and another tank for purging non-condensable gases. For each test, care was exercised to transfer ammonia liquid to or from the test pipe to establish the desired depth and equilibrium temperature.

Instrumentation consisted of an orifice meter to determine the transient mass flow rate utilizing both upstream P0 and differential pressure transducers AP. In order to calculate the mass flow rate through the orifice, which was calibrated with water, the upstream temperature was monitored by RTD1. Another pressure transducer (PD) was located in the

Figure 9: Schematic of Test Pipe, Orifice, and Pressure Tank [11]
downcomer. The initial liquid temperature within the test pipe was recorded by RTD2, mounted on the bottom of the pipe. For the measurement of waterhammer or shock pressures four piezoelectric pressure transducers – PCB1, PCB2, PCB3, and PCB4 – were mounted as shown. The first three piezoelectric transducers were on the bottom of the pipe, while PCB4 was mounted on the pipe centerline at the end cap. In order to determine the gas pressure within the test section during the transient event four diaphragm differential pressure transducers – labeled PACE1, PACE2, PACE3, and PACE4 were mounted on the top of the pipe, as shown in Figure 9. By maintaining one side of the diaphragm open to the atmosphere and the other side connected to the top of the pipe gage pressures were measured during the test. The test procedure consisted of reducing the liquid and gas temperature within the test pipe to the desired value by means of a compressor.

5.2. Problem Setup
A full 3D computational domain is considered in these simulations. The pipe length is 6.3m, and the diameter is 0.14 m. The multi-block grid strategy is used to cover the domain with adjacent sub-domains (coloured differently). Here boundary fitted grids were used rather than the IST. The blocks are distributed between 12 processors for MPI parallel execution. The results presented here were obtained for a grid composed by 360,000 curvilinear cells distributed over 12 blocks. The LEIS approach was employed here, relying on the Level-Set technique for interface tracking. Sub-grid scale (SGS) modelling was achieved using the MILES approach [6], where the diffusive effects of unresolved turbulence motion is left to the scheme. The inflow boundary conditions involve fixing the superficial gas and liquid velocities and the void fraction as specified in the experiments. The right-end of the cap was left open now since we deal with the non-phase case; Pressure boundary conditions were used, in combination with a special scheme for the void fraction to control global and phase-specific volume conservation. Specifically at the inflow, we have set the following values for the turbulent flow conditions: gas superficial velocity $U_{sg} = 14 m/s$; liquid superficial velocity $U_{sl} = 0.5 m/s$; void fraction = 50%. An initial flow disturbance was applied in the flow domain, based on the wall shear Reynolds number.

5.3. Simulation Results

The sealing mechanism is clearly depicted in Figure 10, which seems to repeat itself and successive slugs form along the pipe. The results show something unusual, that is the formation of vortex shedding past the slug immediately after sealing. This observation is corroborated by all three instantaneous velocity- component contours, in particular $w'$. The mechanism is similar to what may be expected in flows past fixed blunt bodies; the slug plays somehow the same role, as it travels with a lower velocity than the gas after sealing. The breaking of the free surface after slug sealing is also perfectly illustrated in Figure 10. Again, the panels combine free-surface and velocity iso-contours. The figure shows the vigorous plunging of the breaker after sealing, pretty much similar to what we observe in plunging breakers. The 3rd panel shows the impact of the breaker over the stratified liquid surface, a
region characterized by a high level of turbulence production. Vortex shedding is again visible as in the previous panels, though the breaker seems to have affected the coherence of their motion.

![Figure 11: Slug speed (tail and centre): LEIS vs. Analytical solution](image)

The slug tail and centre speeds are discussed in Figure 11 below, displaying the position of the slug versus time. A linear dependency is revealed, which is in agreement with the measurements of Martin et al. (2007), who obtained an average slug speed of \( U_s = 9.4 \, \text{m/s} \), under these conditions. The slug speed is rather constant, as it has been found in the earlier 2D simulations (results not shown here). In case of the hydrodynamic slugging, the slug velocity can be calculated from gas and liquid flow rates if the void fraction is known, in horizontal lines the mean velocity of the liquid in the body of the slug is approximately equal to the mixture velocity, or can be estimated analytically using:

\[
U_s = 1.201U_m + 0.532 \sqrt{Dg \left( \rho_l - \rho_g \right) / \rho_l}
\]

where \( U_m \) stands for the mixture velocity. The result shown in Figure 11 reveal that our LEIS simulation predicts the slug speed (tail and centre) in accord with the theory (1), which gives \( U_s = 9.33 \, \text{m/s} \), and experiment of Martin et al. \( U_s = 9.4 \, \text{m/s} \). This is an interesting result for practical applications, which shows that although a coarse grid has been used, the LEIS concept is capable to predict one of the most important flow features of pipe flows.

6. Slug Formation at the Wasp Facility

6.1. Problem Description

The experiments were performed at the Imperial College WASP facility with the test section mounted horizontally. Gas and water were fed from two different entries perpendicular to the main pipe (Figure 12). Slugs were monitored from close to the point where they were first initiated until they decayed or exited the pipe. Twin-wire holdup probes were used to monitor the liquid level at a series of locations along the pipe. Slugs were discriminated from large waves by measuring the velocity using cross correlation of the outputs of successive probes. The length of the stainless steel test section is 37 m and its diameter is 77.92 mm, the pressure at the outlet is 1 atm, and the temperature is 25°C. The liquid water is introduced below a stratification plate at the bottom of the test-line and the gas is introduced above it. The superficial velocities of the two phases (air and water) are: \( U_sL = 0.611 \, \text{m/s} \) and \( U_sG = 4.64 \, \text{m/s} \), respectively.
6.2. Problem Setup

Use was made here of the IST technique to mesh the pipe. The pipe CAD file was created using Rhinoceros software, and immersed into a Cartesian grid, as shown in Fig. 12. The 2D simulations were performed in a pipe of length 17m. The 3D simulations were performed in a shorter domain of 8m, consisting of 715.000 cells, then in a longer one of 16m, consisting of 1.200.000 cells. The simulation time for the 8m pipe simulation was 10 days on a low bandwidth Dell PC (2 nodes x 4 cores; Open MP parallel protocol) for 20s real time, and 53H on a high bandwidth 18 nodes IBM multicore computer (OpenMP protocol). The LEIS approach was employed here, with a filter width of 0.1D, combined with the Level-Set technique for interface tracking. Subscale modelling of turbulence was achieved with the k-ε model with filter width set equal to 0.1D [10]. The inflow boundary conditions involve fixing the superficial velocities and the void fraction, as specified in the experiment. Specifically at the inflow, we have set the following values for the turbulent flow conditions: gas superficial velocity $U_{sG} = 4.64 \text{ m/s}$; liquid superficial velocity $U_{sL} = 0.611 \text{ m/s}$; void fraction = 50%.

6.3. Simulation Results (2D)

Figure 13 shows the measured liquid hold-up at different probe locations along the axis: 5.01m, 5.69m, 6.99m and 13.32m. The signals display distinct large-wave structures developing along the pipe that could in fact be considered as slug-structures [13]. While a traditional slug is a structure blocking the cross section of the pipe completely, large-wave structures with a length scale larger than the pipe diameter can also be termed as ‘slugs’. The 3D simulations discussed next will help make the distinction between the different structures. Slugs or large-wave structures are captured around location x = 3m and beyond (results not shown here). The periodicity of slug occurrence is clearly visible from these locations (x > 5 m). Figure 14 depicts the calculated water holdup in 2D at different probe locations along the axis, from 5.65 to 15m. While the signal is qualitatively similar to the measured one in terms of slug or large-wave structures intermittency, it is unclear whether slugs were indeed captured; various locations exhibit water holdup of about $hL/D = 0.8-0.9$. Be it as it may, large surface perturbations were already captured upstream close to the inflow, at x = 0.76m, while the experiment there shows liquid hold-up not exceeding $hL/D =$
0.2. For the locations considered, the data and CFD provide a similar picture as to wave frequency.

Figure 13: Measured liquid holdup for \( U_{SL} = 0.611 \text{m/s} \) and \( U_{SG} = 4.64 \text{m/s} \).

Figure 14: Simulated liquid holdup for \( U_{SL} = 0.611 \text{m/s} \) and \( U_{SG} = 4.64 \text{m/s} \).

6.4. Simulation Results (3D)

To address the effect of pipe length the flow was reproduced in pipes with different lengths: 8 m and 16 m, using the same flow conditions. Figure 15 shows the development structures at different probe locations along the axis, at 5.01 m, 5.695 m and 6.995 m. These were obtained from 3D simulations in the short pipe \( (L=8\text{m}) \). The distinct patterns at different locations show the variations in the slug frequencies. Slug frequency decays as the location of the probe is moved further downstream. Slugs or large-wave structures are predicted at downstream locations close to the pipe end: \( x = 7 \text{m} \), in contrast to the experiment and longer-pipe simulations, both indicating a shorter position for the early slugs. Further, in contrast to the 2D results discussed earlier, slugs or large disturbances of the surfaces are not predicted upstream close to the inflow, but downstream. These results are interpreted later on in terms of slug frequency, and compared to the longer pipe results.
Turning now to the 16m long pipe, the formation of the different types of slugs is well illustrated in Figure 16. The first panel exhibits a ‘large-scale slug’, which, in the experiments is often referred to as ‘operation slug’. This slug is formed upstream (x < 3 m) and fills entirely the pipe (hL/D = 1) with an average size of the order of 2–4 D. Although the lower panels do not show a 100% water holdup filling the pipe as in the first one, the liquid structures are travelling at a higher speed than the mean flow, which makes them ‘slugs’. Here one observes that gas bubbles are caught inside the slug, which explains that the measured liquid hold up hL/D is less than unity (usually hL/D is between 0.80 and 0.95).

The slug- or large-wave structures frequency results shown in Figure 17 are qualitatively similar to the structures observed in the experiment. The lines in green correspond to the 16m pipe simulations; the red ones to the 8m simulations; both in 3D. The shift in the frequency peak observed for the two simulations is clearly due to the difference in pipe length, as the outflow boundary condition has an important impact on the flow. The frequency of the slugs is measured as a function of the abscissa. In the 16m case, better results are obtained as a peak frequency around 3.5m can be seen, which is almost equal to the measured value. There is however a difference in terms of interpretation, when the frequency of slugs is evaluated based on hL/D of 0.8 or 0.85. It is clear the simulation and measurement agree best for hL/D = 0.85. Moreover, the evolution of the slug frequency along the axis of the pipe is in good agreement with the data, although the result suggests that the simulation time was not enough to acquire all the slugs with lower frequency (0.3 Slug/s). We thus conclude that the data could probably be better predicted for longer simulation time.
Figure 16: The formation of different kinds of slugs (long pipe: 16m).

Figure 17: Comparison of experiment and simulations of slug frequency for 2 pipe lengths; 8m and 16m. Left panel: threshold hL/D = 0.8; Right panel: threshold hL/D = 0.85.
7. Droplet Entrainment & Detachment

7.1. Background

In annular interfacial pipe flow the wall-adjacent liquid layer maybe subject to droplet detachment and dispersion in the gas core. Droplets are entrained from the film at the bottom of the pipe and are transported towards the upper part of the pipe where they deposit and form a liquid film which drains downwards towards the bottom. Thus, there is a process of continuous renewal and drainage of the film in the upper part of the pipe. Replenishment of the film by droplet deposition is then equal, in fully developed flow, to the deposition of the droplets on the surface (Anderson and Russell, 1970). Understanding the relationship between the liquid film around the periphery and the liquid pool at the bottom of the pipe (Brown et al., 2008) is of importance from the flow-assurance standpoint.

7.2. Problem Definition

In a first step, prior to tackling the annular flow with peripheral liquid film drainage, we first attempted to reproduce droplet entrainment and detachment in a stratified gas-liquid flow in a shorter pipe than the previous one, of diameter 0.5m and of length 5m. The pipe contains though water at a low water holdup, i.e. $h_L/D = 0.14$, injected at a velocity of 0.2m/s. Air is injected at a bulk velocity of 20m/s, making the flow rather turbulent ($Re_G = 1.6 \times 10^5$). A splitter plate has been placed by purpose between the two phases for the first 16cm to help flow development and avoid raising issues with two-phase inlet conditions.

To perform the simulations, V-LES has been used in order to capture the unsteady behaviour of the flow, in line with the recommendations and learnt-lessons from the section presenting the 2D simulations, that three-dimensional effects are important for free surface deformations. All the more, 3D simulations increase the likelihood of correctly representing the appearance of the first droplets including their size and shape. To be pragmatic though, it is clear that predicting droplets of the mm size would require a very dense grid, which is beyond reach of basic computational resources. To alleviate this limitation, one could rather resort to a time evolving, interfacial flow, and using periodic boundary conditions (e.g. Gulati et al., 2011).

7.3. Simulation Setup & Results

This flow is simulated on a grid consisting of 1.6 million cells, with regular cells of 1cm in all directions (580 along the pipe and 58 x 54 for the cross section). Use was made of the IST meshing technique introduced in an early section to represent the pipe inside a simple Cartesian grid (Fig. 18). Only high order schemes were employed, for both space discretization and time marching (3rd order RK). Five days were necessary on a 16-core PC to achieve 3.6 s of flow time (5600 time steps at $\Delta t = 6.5e-4$ s).
As was to be expected, the significant difference in the resulting velocity between the two phases in an important shear at the interface, as shown in Fig. 19, displaying streamwise velocity contours and the corresponding turbulent eddy viscosity and x-vorticity contours. The figure clearly shows the boundary layer evolution on top of the interface, which may evolve further to reach spatially fully developing conditions. Be it as it may, we can safely conclude that the computational parameters employed for this simulations are sufficient for the flow to develop (though probably not 'fully developed') and return the main mechanisms of interface deformation and water droplet entrainment, as discussed next. The calculations show at least that the code is able to predict shear-induced fragmentation at the gas-liquid interface.

Figure 20 depicts the contours of negative and positive vorticity contours to mimic turbulent coherent structures in the flow. The structures are shown to be localized at the interface where the strong imposed gas-side shear creates turbulence. The phenomenon has been previously shown to occur in other similar interfacial turbulent flows. A better description of the phenomenon is shown in Fig. 22, displaying surface level with vorticity contours.

Figure 19: Space evolution of the interfacial boundary layer: contours of (a) x-vorticity, (b) U-velocity and (c) eddy viscosity

Figure 20: 3D snapshot of negative and positive-vorticity contours
Figure 21 suggests the presence of small amplitude perturbations at the sheared surface, which finally results in droplet detachment, as can be clearly seen from the snapshots displayed in Fig. 21. These instabilities (of Kelvin-Helmholtz type) start growing downstream around \( x/D = 1.4 \) (55 cm downstream of the separator plate), and the first droplet detachment occurs only around a streamwise location of 1.5 m, corresponding to \( x/D = 3 \). These are then entrained in the turbulent air flow, as can be judged from the snapshots presented in Fig. 21. There are subjective reasons to think that the droplets are not well resolved by the grid such that they disappear from the computational domain. But this is actually not sure and further analysis is thus required. This is said, one could think of a better alternative to solve this sort of problems is then to couple ITM with a Lagrangian droplet tracking. In short, droplets larger than the grid are directly captured by ITM, while smaller ones of size equivalent to the grid scale will be tracked in a Lagrangian way. This is work under development at ASCOMP.
8. Multiphase Flow in Vertical Pipes

8.1. Problem Description and Model Set-up

Measurements of a mixture of gas-water flowing in a vertical pipe of 6.7 cm diameter and 6 m length with various superficial air and water velocities (Figure 22) were conducted at the Chemical Engineering Laboratory of the University of Nottingham, UK (Szalinski et al., 2010). Liquid and air were mixed at the bottom of the pipe by a special mixing device. The liquid enters the mixing chamber from one side and flows around a perforated cylinder; air is injected through a large number of 3 mm diameter orifices, thus, gas and liquid could be well mixed at the test section entry. Inlet volumetric flow rates of liquid and air were determined by a set of rotameters. Gas void fractions were measured at a height of 5 m using a wire-mesh sensor. Three cases were reproduced numerically using code TransAT: in Case 1, the liquid and gas superficial velocities are 0.25 m/s and 0.05 m/s, respectively; in Case 2, the liquid superficial velocity is increased to 0.7 m/s. These two flow regimes are bubbly flows, as shown in the flow pattern map in Figure 22. In Case 3, the gas superficial velocity is $\mathrm{JG}=0.57 \, \text{m/s}$, for a comparable liquid superficial velocity: $\mathrm{JL}=0.25 \, \text{m/s}$. According to Figure 7, this is a flow fearing a clear Taylor bubble, corresponding to the 7th panel of Fig. 1. The equivalent Reynolds numbers for Case 1 and 2 are 17’000 and 47’000, respectively, based on the water properties and the pipe diameter. Case 3 is an intermittent slug flow, featuring coherent Taylor bubbles, followed by a cloud of smaller bubbles trapped in the wake of the large cell.

The mixture model with algebraic slip closure for the interphase slip has been used, in combination with the k-ε model for turbulence, employed with the standard near-wall functions (Manninen & Taivassalo, 1996). The lift force model is based on Tomiyama’s (1998) proposal. Two-phase flow in a circular pipe can be simulated under two-dimensional axisymmetric assumption, or in three-dimensions. The solution can be either forced to convergence via steady-state iteration process or through time marching until steady-state conditions are reached; this is true for flows laden with small bubbly structures only. Inflow conditions were set by fixing superficial velocities of gas and water streams. Since no data are available as to the exact inflow flow topology, the velocity and void fraction profiles are assumed in calculations to be flat. Note that in the experiments, 3 mm radius bubbles were released, but merging occurred immediately near the sparger, which explains the reasons why bubble size distributions show a radius spectrum extending from 0.5 to 6.0 mm.
8.2. Bubbly Flow Simulation Results

Sensitivity simulations were first performed in order to define adequate domain size and grid refinement. The analysis was performed under 2D axisymmetric steady-state conditions for Case 1 (JL=0.2 m/s, JG=0.05 m/s). Results of a fine mesh consisting of 507x24 cells (Δx=1cm, Δy=0.4mm) were compared to those of a coarser mesh of 307x15 cells (Δx=1.6cm, Δy=1mm). Although calculated void fraction and velocity profiles were comparable at the centre of the pipe, there seems to be an important loss of precision with the coarser mesh close to the wall (Figure 23). The 507x24 cells mesh was then used for all subsequent studies. As to the length of the pipe, simulations of Case 1 performed on a 5.1 m long pipe did provide the same results as with a 3m (L/D=44.7) pipe. However, for Case 2 (JL=0.7m/s, JG=0.05 m/s), it was found that the flow could not yet be developed after 5m (L/D=74.6). Therefore, final simulations were performed on a 5.1m long pipe, and results were taken at 5m as in the experiment. Final sensitivity studies related to steady versus unsteady simulations. For Case 1 at least, steady and unsteady (once averaged over time) results were exactly the same (see also Figure 24a).

This first detailed simulation was aimed to reproduce Case 1 & Case 2 flow conditions, with superficial velocities of JL=0.2 m/s and JG=0.05m/s, and JL=0.7 m/s and JG=0.05m/s, respectively. Steady and unsteady simulations were performed, providing both a reasonable qualitative agreement with the data as shown in Figure 24. The maximum void fraction at the centre of the pipe is well predicted, albeit is slightly over predicted close to the wall. This may be due to an underestimation of the lift force, which remains one of the critical closure laws in phase-averaged models. Phasic velocity profiles were also plotted; however, since no experimental data is available, these are not shown here.
8.3. Slug Flow Simulation Results

We now turn to Case 3 with a gas superficial velocity of $J_G=0.57$ m/s (and $J_L=0.25$ m/s). Initial conditions are the same as inflow conditions, fixed as per the measurements. This case includes the formation of air slugs or Taylor bubbles travelling upward along the pipe, which makes it more complicated to model because of the simultaneous presence of large and small bubble structures trapped in the wake, and associated unsteady – meandering– behavior of the flow and more active turbulence generated due to the interaction of the phases, and between the aerated structures of different size.

Case 1 includes the formation of air slugs or Taylor bubbles traveling upward along the pipe, which makes it more complicated to model because of the simultaneous presence of large and small bubble structures trapped in the wake, and associated unsteady “meandering” of the flow and more active turbulence generated due to the interaction of the phases, and between the aerated structures of different sizes. The chosen pipe length is only 3.1 m to ensure sufficient grid resolution and reasonable CPU time. The algebraic slip model has been used, although it is well known that this approach applies only for flows laden with small bubbles of sizes comparable to the grid size. The modeling of the drift velocity is another issue, since it requires the average bubble radius. Although we are aware that this class of models cannot be applied for slug flows, we have used it nonetheless, assuming a bubble radius of 3.0 mm (otherwise the model crashes).

The 3D simulation was performed on a HPC supercomputer with 128 Processors. Level Set has been combined with LES for turbulence, using the WALE subgrid-scale model (Nicoud & Ducros, 1999). The grid consists of three million cells. The Immersed Surface Technique (IST) was employed, in that a CAD representing the pipe has been simply immersed in a Cartesian grid. Near-interface treatment of turbulence follows the model proposed by Liovic and Lakehal (2007). High-order schemes were employed, up to 3rd-order RK for time marching, and 2nd order central scheme for convection fluxes; the Quick scheme was used for solving the level set equation. The time step varies in time (bounded by convection, diffusion and surface tension CFL-like limiters $\sim 0.4-0.7$), decreasing when small bubbles appear, down to 10-5s sometimes. Understandably this first attempt has been made possible thanks to the available HPC resources only. The simulation of 6 seconds reproducing 4 slugs required 22 hours on the HPC supercomputer.

Figure 25 clearly shows that the LEIS approach provides a rich picture of the flow, as it may look in reality and is qualitatively much closer to the experiment: c.f. 6th and 7th panels in Fig. 1). Slugs of different sizes and elongations form naturally without triggering their onset, occupying the entire pipe and travel upwards. Swarms of bubbles are also generated in the wake, populating the area between the Taylor bubbles. Our videos show actually that the bubble cloud is primarily a result of fragmentation of the Taylor bubble wake due to strong interfacial shear dominating surface tension. Despite the qualitative picture, this grid is not sufficient to resolve the cloud of bubbles as depicted in the experimental images. Furthermore, because of the grid resolution, grid-size bubbles formed tend to disappear. Also, time averaged profiles could not be generated, because steady-state ergodic conditions require averaging over at least 10 Taylor bubbles traveling along the pipe.
Figure 25: 3D LEIS simulation of slug flow regime.

Figure 26: Evolution of slug flow at a selected instance.
Figure 26 shows close-up images of the flow immediately in the wake of the Taylor bubble, with cross-sections colored by the streamwise velocity. The images show better the fragmented portion of the Taylor bubble, featuring gas bubbles of different sizes. It also shows that the subsequent two slugs or Taylor bubbles are actually smaller in size. It may be that the slug flow forms as two small cells merge later on into a larger one. The second image gives also the impression that these two small Taylor bubbles are the result of the fragmentation of a preceding larger Taylor cell. Figure 27 shows snapshots of the SGS viscosity in the Taylor bubble wake normalized by the molecular viscosity. While the image does not reflect important flow physics, apart from the secondary flow motion typical of confined turbulent flows (though here, in contrast to channel flow, this motion is promoted by the interaction between the bubbles and the carrier water phase), it has the merit to indicate the degree or LES resolution of the flow. A ratio of this order is a good indication of the resolution of the flow, in fact, the unresolved SGS part of it.

Global flow parameters are discussed in the context of subsequent figures, 28 and 29, including pressure drop and wall frictional velocity along the pipe. While the instantaneous pressure drop values shown in the upper panel indicate changes with time that can vary up to 10% from one case to the other, the lower panel of the figure shows, as was expected, that the average (of the upper instantaneous values) pressure drop in the slug case is 20% lower than in the equivalent single water phase flow. Instantaneous wall frictional velocities along the pipe walls plotted in Fig. 29 reveal that the signal is very sensitive to the passage of the voids, with a rate of variation around the mean that exceeds 100%. This is an important result to consider when employing 1D system codes that could only use empiricism to fit the model.

8.4. 3D Annular Flow Simulation: Coarse Grid

For the annular flow regime (Case 2), the dimensions of the pipe were reduced to 3cm x 100cm. With a grid of 800,000 points, the simulation required 32 hours to perform 20,000 iterations on a HPC cluster, using 128 cores. The numerical method and parameters were the same as for the slug flow simulation discussed above. The resulting time step was approximately 6.10^-6 s. Here, too, the level set approach was used in connection with LES for turbulence; basically the same model setup as in the previous slug flow. In reality the film is set purposely thicker compared to the experiment; otherwise a much larger grid would have been required to resolve the film, which was not possible at the time the case was simulated. The flow conditions were set as follows: \( VG=2000 \text{ L/m} \) corresponding to \( Re_G=8.55\times10^4 \), and \( V_L=70 \text{ L/m} \) corresponding to \( Re_L=4.22\times10^4 \). The initial film is \( h/D = 0.04 \). Both phases are therefore turbulent.
Density contours with interface and cross-flow velocity vectors of the air-water pipe flow are shown in Fig. 30. The coherent wavy structures of the water film can be seen in both the streamwise and spanwise directions, in particular in the 3rd panel of the figure. Only in a few locations we could observe water parcels migrating to the core flow. The secondary flow motion is also noticeable, as in the slug-flow case, very much similar to confined flow in enclosures.

This is a remarkable result that has been so far within the realm of speculation only. The evolution of the water film thickness on the walls is reported in Fig. 30 at different locations, indicating that the flow is not yet fully developed and the comparison with the data of velocity and density profiles would require a longer pipe. Although presented as a proof-of-concept only, the present LEIS results are very encouraging and demonstrate TransAT’s capabilities for simulating multiphase vertical pipe flows. Such an approach provides a novel versatile method for exploring/explaining riser flows. Figure 31 presents three cross-flow
locations of the pipe, featuring the liquid film deformation. It is interesting to note that there is a certain radial coherence of the events. Occasionally we could observe detached droplets migrating to the core flow. Note the 3D view gives the wrong impression that the film is considerably thick.

Figure 31: 3D LEIS in the annular flow regime. Snapshots of the pipe flow at three cross-flow locations.

Figure 32 depicts the evolution of the liquid film thickness in the annular flow regime taken at z = 70, 80 and 90 cm. The plotted values are normalized by the initial liquid film height. One observes how intense the deformation of the interface could be, with fluctuations around the mean reaching ~400%. The frequency of the coherent waves can be extracted from the results below, indicating that it may change with time by up to 1 order of magnitude, i.e. $125 < f < 275$ Hz. The results suggest that the disturbance waves develop with the distance, and this is true of their circumferential coherence. This result confirms the finding of Hall-Taylor & Nedderman (1969) that the wave frequency decreases with the distance.

Figure 32: Evolution of film thickness in annular flow regime at $z = 70, 80$ and 90 cm.

8.5. 3D Annular Flow Simulation: Fine Grid
This flow regime is more complex than the previous one since it involves a very thin film of water. The experiments were performed at Imperial College London [19]. The objective was to study the initiation, growth and development of circumferential coherence of disturbance waves in annular flow and shed new light on its physics, four decades since the pioneering Harwell Laboratory experiments of Hewitt & Lovegrove (1969) and Hall-Taylor & Nedderman (1969).
In contrast to the previous thicker-film case, here the flow has a more complex interfacial structure, because of the presence of both small ripple waves and large disturbance waves. In small tubes, disturbance waves are travelling like short milky ring around the periphery along the pipe, with different velocities and frequencies over a continuous liquid layer near the tube wall. To obtain a better understanding of the disturbance wave structure, a series of experiments were carried out in a pipe of 3.45 cm diameter and 4m length, where salty water (0.6 g/L KNO) and air flow upward under room temperature and atmospheric pressure conditions (c.f. Table 2). Two sets of four-pin probes were placed at different flange section.

<table>
<thead>
<tr>
<th>$V_G$ (L/min)</th>
<th>2250</th>
<th>1950</th>
<th>1650</th>
<th>1350</th>
<th>1050</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re_G (10^4)$</td>
<td>9.33</td>
<td>8.09</td>
<td>6.84</td>
<td>5.60</td>
<td>4.35</td>
</tr>
<tr>
<td>$Re_L$</td>
<td>211</td>
<td>302</td>
<td>452</td>
<td>603</td>
<td>-</td>
</tr>
<tr>
<td>$V_L$ (L/min)</td>
<td>0.35</td>
<td>0.50</td>
<td>0.75</td>
<td>1.00</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 2: Experimental matrix of flow parameters.

Ripple waves were revealed for low water flow rates. For higher water flow rates, the flow was seen to feature a mix of disturbance and ripple waves, with a clear circumferential coherence of the disturbance ones (Fig. 33).

The LEIS setup of the problem is very similar to the previous one, except that the grid is way denser than in the thicker film case (Fig. 34), and periodic boundary conditions were applied to the flow including the water film in order to reproduce turbulence. Since the wave length if the larger waves is small (could be inferred analytically), a shorter pipe domain was considered, in line with previous DNS and LES studies of turbulent pipe flows. The domain size is: 0.2m length and 0.0328m diameter. A BFC grid consisting of 937,000 cells was employed, with the cross section covered by 12500 cells. This grid clearly allows resolve the viscous sublayer, which avoids having to resort to wall functions. No turbulence model was selected. The simulation took 763 CPU hours on a supercomputer, using 15 parallel...
processors. As in the previous case, the level set technique was employed; though using a
time-marching high-order scheme on a narrow band was needed to conserve mass via re-
distancing. The film thickness was inferred from the imposed mass flow rate of water and as
such it was not fixed as in the previous case.

Figure 34: 3D evolution of film thickness in annular flow.

Figure 35: Cross-sectional views of film thickness in annular flow and secondary flow structure and
turbulence.
Figure 34 shows various snapshots of the thin film after 65,000 time steps of simulation, which took almost 15,000 time steps before revealing the first interfacial deformations. The flow seems to exhibit both circumferential coherent disturbance waves (Panel 1 and 4; counting from upper left) and non-coherent smaller ripple waves (other Panels). It is unfortunate that no interface signal is reported here to better infer information on the circumferential coherence; in fact a rich signal needs more than 100,000 times steps of simulation data. Note that periodic conditions as in this flow do not allow predict the development of the waves with distance; but only a fully developed portion of the pipe.

The secondary flow motion depicted in Fig 35 reveals how fine the resolution of this problem was, in terms of grid and flow structures. By purpose, we show here various snapshots revealing turbulence gradually. Fully developed turbulence is achieved in the lower 4 panels only, featuring a broadband spectrum of scales; the larger ones are half-diameter size. The water film is tiny and cannot be well perceived, and so is the case for the interface perturbation. However, one could notice that at some locations, the film is almost dried-out.

9. Conclusions
Progress made in predicting multi-phase flows with high fidelity, in the context of oil and gas problems, have been reported. Complex multi-phase, oil and gas problems are shown to be within reach of modern 3D CMFD techniques implemented in the code TransAT. The examples were presented to demonstrate the capabilities of CMFD simulations in general and TransAT in particular. Deeper insight into each individual problem would of course require a dedicated study to exploit the rich database generated in the simulations. Most of the models can be further refined and adapted for various scenarios. It is true that various roadblocks need to be alleviated before CMFD could be efficiently used to provide an added value to understanding the various processes, as is the case today with 1D codes like OLGA, LEDAFLOW, etc. The present effort is dedicated to the coupling of TransAT with 1D codes to cope with specific problems requiring more than one single strategy.

References

Gulati S., Narayanan C., Buongiorno J. and Lakehal D., 2013, Simulation of liquid entrainment in stratified and annular flow regimes using Large-Eddy and Interface Simulation, LEIS, ICMF2013 (Paper 859), May 26-31


