Basic flow modelling for long distance transport of wellstream fluids

D Biberg, H Holmås, G Staff
SPT Group, Norway

T Sira, J Nossen, P Andersson, C Lawrence, B Hu, K Holmås
Institute for Energy Technology, Norway

ABSTRACT

This paper reports improvements in transient multiphase flow simulation to meet future industrial challenges. Multiphase flowlines can be hundreds of km long and feature dynamics with time spans of months, dictating the use of a 1D model on a coarse grid for engineering calculations. The 1D conservation equations for multiphase flow are robust but require closures. Robust mechanistic closures are obtained by integrating a 2D velocity field over the cross-section. In combination with the 1D equations, this gives a 3D flow description, greatly improving scale-up from experiments to field conditions. Extensive comparisons with experimental data reveal substantial model improvement.

1 INTRODUCTION

Multiphase flow technology has had a remarkable development over the past two decades. The distance and the range of conditions under which well stream fluids can be transported economically have increased drastically. This development has only been possible because of the ability to predict the flow behaviour with sufficient reliability. The inherent nature of multiphase pipeline flow implies that the technology cannot be exploited in a safe, controlled way unless the dynamic behaviour of the flow can be predicted. This is crucial with respect to early phase feasibility studies, optimal design of pipeline systems, and the safe operation of the transport system. In this respect commercial dynamic multiphase flow simulators, such as OLGA and TACITE, have served the industry with access to state of the art predictive tools.

Longer distances, deeper water, more difficult fluids and cheaper transport solutions are important key words for the future challenges in well-stream transport. In addition, the industry faces new problems with mature fields that are going into the tail end production phase, with high water cut and reduced wellhead pressure increasing the tendency towards slugging. The HORIZON JIP was initiated in 2004 to address these problems, acquire the new basic knowledge, develop the methods that are necessary and then implement them in the tools used by the industry.

The Institute for Energy Technology (IFE) developed the first version of OLGA for Statoil in 1980. In 1983 IFE and the SINTEF Two-Phase Flow Laboratory together launched a large joint industrial project where SINTEF ran experiments in their large scale, high pressure two-phase flow loop, while IFE developed the flow models and the
OLGA was commercialised through Scandpower in 1990. IFE later developed several new OLGA versions for different groups of oil companies, mostly in cooperation with Scandpower (now SPT Group). During these projects, ideas for new mechanistic multiphase flow models were initiated and gradually matured leading to the HORIZON programme.

Traditionally 1D modelling has been based directly on bulk balances for mass momentum and energy, supplemented with ad-hoc empirical closures for frictions and dispersions. Biberg (2007) demonstrated how a parameterized cross sectional flow description can be integrated, yielding a so called pre-integrated 1D model, featuring a consistent set of mechanistic friction models which replace traditional ad-hoc closures. The basic idea the present work was to expand this idea to all flow regimes, in order to obtain a complete 1D flow multiphase flow model with intrinsic scale up properties.

In the 1980’s multidimensional computer simulations of pipeline networks were practically impossible, and this remains the case today for all but the simplest systems with short pipe lengths. Recognising this, the pioneers of the field derived one-dimensional equations to describe the conservation of mass, momentum and energy in multiphase pipe flow (Ishii 1975; Wallis, 1969). These equations are rigorous balance equations, but averaging over the pipe cross-section leads to a loss of information, and the need to introduce closure models. These closures can be supplied either by correlations of experimental data, or by integration of a more detailed underlying model.

For separated (stratified) flows, the main closures that are needed are the frictional interactions between the phases and with the pipe wall. In the majority of applications, the phases are not completely separated, but either partially or fully dispersed, and then closures are also required for the dispersion characteristics (phase fractions, inter-phase slip, etc.).

Dimensionless correlations have a long history of success in single phase flow and heat transfer applications. A relatively small number of experimental data can be used to produce generic correlations that cover a wide range of operating parameters. For example, the celebrated experiments of Nikuradse (1937) showed that the friction coefficient for single phase flow in sand-roughened pipes could be completely characterised by a few hundred experimental measurements for different values of the Reynolds number Re and dimensionless roughness $k/D$. The Colebrook-White equation, which represents an integration of the underlying physics of the turbulent boundary layer, was subsequently developed and shown to match experimental data for commercial pipes over a very wide range of roughness ratios and Reynolds numbers:

$$\frac{1}{\sqrt{\lambda}} = -2\log_{10}\left(\frac{2.51}{Re\sqrt{\lambda}} + \frac{k}{3.7D}\right)$$

For the pioneers of multi-phase flow, it was natural to attempt to extend the approach of dimensionless correlations to multiphase flow, and this approach has been followed until quite recently, albeit with mixed success. Typical correlations for multiphase flow are adaptations of the successful correlations for single-phase flow, with additional parameters or tuning to account for the presence of more than one phase (Bendiksen et al., 1986; Barnea & Taitel, 1993). However, a dimensional analysis of even a two-phase flow shows that the number of relevant parameters increases from 2 to at least 9, with additional parameters for the pipe inclination, the density and viscosity ratios, the ratio of the flow rates, a Froude number characterizing gravitational effects, a Weber number characterizing surface tension effects, and the contact angle for the fluids at the pipe wall.
The Nikuradse data characterize single phase flow with $O(10^5)$ experiments, but a two-phase equivalent would require a completely impractical $O(10^9)$ experiments!

The fact that dimensionless correlations based on data have very poor properties of extrapolation beyond the data set to which they were tuned inevitably leads to poor performance of one-dimensional flow codes under some circumstances. This is particularly a problem because the majority of experiments are carried out for relatively small diameter pipes with model fluids at modest pressures, while industrial applications involve large diameter pipes with crude oil, formation water and gas at high pressures. The limitation of closure approaches based on empirical data has led some scientists to argue for the abandonment of the one-dimensional approach altogether. However, the problem lies not in the one-dimensional approach itself, which is quite rigorous, but in the poor quality of the closure correlations. We have adopted a strategy of analytical integration of the underlying physical models to obtain “pre-integrated” mechanistic closure relationships which require little or no tuning to data, and thus have much better extrapolation properties.

2  HIGH SPEED VIDEO RECORDINGS (VISUALIZATION EXPERIMENTS)

In order to better understand the basic flow phenomena to be considered, high speed video recordings were made of wavy stratified flows and slug flows. The recordings were made in the Well Flow Loop at IFE, which has an inner diameter of 10 cm and is 15 m long (150D). The gas used was SF₆, with a pressure of 7 bar for wavy flows and 3.5 bar for slug flows. The liquids were tap water or Exxsol D80 (1.8 mPa.s, 829 kg/m³, 0.021 N/m). Pipe inclinations were 0°, 1° and 2° for wavy flows and 1° and 3° for slug flows. Superficial gas velocities were between 0.5 m/s and 3.5 m/s and superficial liquid velocities were between 0.05 m/s and 1.5 m/s. Wavy flows were recorded at a frame rate of 800 fps and slug flows at 2000 fps. All recordings were converted to Windows Media Video format with a playback frame rate of 30 fps. This gave a slowdown factor of 27 times for wavy flow and 67 times for slug flow.

![Figure 1 Large wave flow. (a) Horizontal flow, $U_{SG} = 1$ m/s, $U_{SW} = 0.3$ m/s (b) Inclination 1°, $U_{SG} = 1$ m/s, $U_{SW} = 0.05$ m/s.](image)

In the recordings of wavy flow, the light source was placed above the pipe and reflected from a white plate behind it. A 1.3 kW flicker-free theatre lamp was used. In the recordings of slug flow, reflected light was not enough due to entrained bubbles that
scattered the light. Therefore two lamps were placed behind the pipe, slightly upstream and downstream of the camera, directed towards the area in front of the camera.

![Figure 2 Slug flow. Inclination 3°, \( U_{SG} = 1 \) m/s, \( U_{SW} = 0.3 \) m/s (a) slug front (b) slug tail.](image)

Figure 1 shows two snap shots of wavy flow. The resolution in the images allows for a detailed study of the flows. Features such as the shape of the waves, entrainment of gas bubbles and tear-off of liquid drops can be seen clearly. Figure 2 shows snap shots of the front and the tail of a slug. The distribution of bubbles in the mixing zone and through the slug and the shape of the tail are both seen in great detail. Studying the videos gives detailed information and a better understanding of the mechanisms involved in the dynamics of waves and slugs.

3 STRATIFIED FLOW

Stratified flow or layered flow is the dominant flow regime in gas-condensate pipe-lines. It is also frequently encountered in oil-dominated flows. A stratified flow is dominated by turbulent boundary layers near the pipe walls and interface, and is thus governed by the boundary-layer equations. Considering a three phase flow, and integrating over the pipe cross section, yields the 1D three-fluid model, which accounts for the bulk mass and momentum conservation for a slowly evolving stratified flow, and constitutes the basic framework for 1D models.

Traditional 1D models apply the bulk balances given by the three-fluid model directly, without accounting for the actual flow distribution over the pipe cross section. This approach leaves the crucial wall and interface frictions undetermined, which thus must be represented by ad-hoc closures. This inevitably leads to an inconsistent model, which reduces the potential for scale-up.

In a more refined model, the frictions are given by the velocity distribution, and will thus be part of the solution. Introducing a parameterised velocity distribution based on a turbulence model in the boundary layer equations is a standard method for calculating boundary layer flows. This approach is also adopted in the pre-integrated stratified flow model by Biberg (2007), which constitutes the basis for the stratified flow modelling in the present work.

The Biberg model features a two-parameter velocity distribution within each fluid based on the (generic) algebraic eddy viscosity distribution:
where $v_T$ is the eddy viscosity, $u^* = \sqrt{R_f} / \rho$ is the friction velocity, $h$ is the fluid depth and $Y = y/h$ is the scaled vertical coordinate. The first parameter $R = \tau_f / \tau_w$ is the ratio of interfacial to wall shear stress, and is determined by the model. The second parameter $K$ represents the turbulence level at the interface, and is subject to modelling. The flow is assumed to be in a fully-developed quasi-steady state on either side of a sharp (flat) interface, represented by a jump in fluid properties and turbulence levels.

Mechanistic friction formulas reflecting the velocity distribution are obtained by analytic pre-integration and coupling of the phases by imposing the hydrodynamic boundary conditions at the interface. The corresponding pipe flow formulas, obtained by use of hydraulic similarity, replace the ad-hoc expressions traditionally applied in 1D multiphase flow simulators. The model shows how the coupling of the turbulent fields at the interface represents the main closure challenge in the modelling of stratified flow. It compares well with standard CFD simulations (Holmås and Biberg, 2007), and has nearly the same level of detail; it also compares favourably with both detailed measurements and bulk flow data.

We also derived a “two-equation” algebraic $k - \omega$ model in order improve the understanding of the turbulence coupling at the interface. This model was obtained by supplementing the eddy viscosity distribution Eq. [2] with a similar expression for the turbulent kinetic energy (TKE) $k$. The corresponding expression for $\omega$ was then given by $\omega = k / \nu_T$. The algebraic distributions allowed us to compute the turbulent production $P$ and dissipation $\varepsilon$. This in turn made it possible to use the $k$ -equation to compute the flux of TKE over the interface, as the integrated difference between $P$ and $\varepsilon$ over the fluid depth i.e.

$$J = \frac{\tau_f \rho}{h} \int (P - \varepsilon) dy$$

[3]

in which the upper and lower signs apply for the gas and liquid respectively.

The algebraic $k - \omega$ model has four model parameters which (indirectly) represent $k$ and $\omega$ on either side of the interface, in analogy with the corresponding CFD model, which has four boundary conditions for the $k$ and $\omega$ fields at the interface. One closure parameter could be (implicitly) eliminated by the fact that the turbulent flux $J$ over the interface should balance. We will not consider the remaining closures here, but rather focus on the nature of the TKE flux between the phases, which is crucial for understanding the nature of the coupling of the turbulent fields between the phases. The equations reveal that the TKE flux will always be directed from the high-production side of the interface towards the low-production side. The special case of turbulent equilibrium on one side of the interface implies zero interfacial TKE flux and thus also turbulent equilibrium on the other side of the interface.

Computing the flux of TKE over the interface using Eq. [3] and the Lorencez et al. (1997) air-kerosene data for smooth and wavy interfaces indicated a clear link between the interfacial waves and the balance of turbulent production and dissipation in the vicinity of the interface.

Smooth interfaces are associated with symmetric velocity distributions in each phase, as shown in the left hand plot in Figure 3. Smooth interfaces were also found to be associated with low or zero interfacial TKE flux. Smooth interfaces thus correspond to a
situation in which the bulk production and dissipation are in balance within each phase, and the turbulent fields are effectively isolated from each other. Wavy interfaces, on the other hand, were found to represent a situation with a distinct TKE flux from the gas to the liquid.

Wavy interfaces are associated with asymmetric velocity distributions, as shown in the right hand plot in Figure 3. Wavy interfaces were also found to correspond to a situation in which the gas produces more TKE than it dissipates. The surplus TKE is transferred to the liquid, which dissipates more TKE than it produces. The influx of TKE to the liquid enhances the turbulence level, which acts to flatten the velocity distribution and drive up the wall friction. This observation agrees with the flat liquid velocity distributions in the Akai et al. (1980) and (1981) air-mercury channel flow data. It also agrees with the findings of Espedal (1998), who observed a distinct increase in the liquid-wall friction associated with the onset of interfacial waves of a certain size.

![Figure 3: The TKE flux for smooth and wavy interfaces. Here P and ε are the bulk production and dissipation. Full drawn line: velocity distribution as given by the model, dots (...) Akai et al. (1980) and (1981) air-mercury channel flow data.](image)

Water will in general be present as a third phase in oil and gas production lines. A stratified flow model should thus be a three phase gas, oil and water model. However, basing a three phase pipe flow model on the channel flow geometry does not work very well, since the middle layer (oil) is not in contact with the wall in the channel. There will thus be no natural occurrence of an oil-wall friction term to be applied in the pipe flow model. The three phase stratified flow model in the present work was consequently based directly on the pipe flow geometry.

A single generic layer model accounting for the flow geometry of the middle layer (oil) was applied to represent the flow within each phase. The generic layer model applies directly to the oil layer. Special cases in which the upper and lower interfaces disappear are used for the gas and water phases respectively. Mechanistic wall and interface friction formulas, reflecting the velocity distribution over the pipe cross section, are given by pre-integration and coupling of the phases via the interfacial boundary conditions. Linearizing the weakly non-linear expressions gives explicit expressions securing expedient evaluation. The model has four closure parameters reflecting the turbulence levels on either side of the gas-oil and oil-water interfaces.

The pre-integrated three phase flow model follows the classical boundary layer approach of introducing parameterised velocity distributions into the boundary layer equations. The model thus adds the two cross-sectional dimensions to the axial dimension in the 1D three-fluid model, as illustrated in Figure 4. This results in a consistent 3D description of a slowly evolving three phase stratified flow with 1D computation speed.
The model was tuned to the Espedal (1998) data-set for air-water flow in a small-diameter pipe. Without further tuning, the model gives improved predictions over a wide range of high pressure large-diameter data, when compared to the commercial OLGA code, thus confirming the extraordinarily good scale up properties of the model, see Figure (5).

![Figure 4](image4.png)

**Figure 4**: The velocity distribution for a three phase stratified flow as given by the pre-integrated stratified flow model. A single generic layer model is applied for each phase. Interfacial coupling is given by the boundary conditions.

![Figure 5](image5.png)

**Figure 5**: Measured vs. predicted pressure drop for the Tiller 8” ID, 20, 45 and 90 bar nitrogen/diesel data. Results using the commercial OLGA code to the left, the new pre-integrated stratified flow model to the right.

4 LARGE WAVE FLOW

A typical flow with large waves is shown in Figure 2a. There are several characteristics that distinguish this flow regime from regular stratified flow: First of all, experiments indicate that there is a sudden increase in the pressure drop when interfacial waves above a certain size (large waves) are formed. Standard smooth-interface friction models are not able to predict this behaviour. Also, large waves often have breaking wave-fronts, as shown in Figure 2b, and are then commonly referred to as roll waves. The flow in these breaking wave-fronts is very complex and requires modelling beyond the boundary-layer equations. For these reasons, large wave flow was regarded as a distinct flow regime in the present work.
We have invested much effort into examining different modelling concepts for large waves and obtaining a better understanding of the flow regime. One of the main activities has been a PhD project focusing on numerical simulation of waves in two-phase flow (Holmås, 2008). The approach taken in this PhD project was to use a very accurate numerical method combined with fine numerical grids to capture waves that evolve naturally as instabilities in 1D two-fluid models. In order to model the increased dissipation in breaking wave fronts, the interfacial turbulence levels in the stratified flow model were modified (see Section 3). While this modification led to a redistribution of the wall and interfacial shear stresses along a wavelength, it did not significantly change the predictions of average pressure drop and holdup.

Figures 6 and 7 show comparisons of simulation results with the experimental data of Johnson (2005). In Figure 6, the measured and the simulated holdup time traces are plotted for cases with different pipe inclinations and superficial liquid velocities. If we disregard the high frequency oscillations found in the measured time trace, it can be seen that the characteristic steep oscillations and long tails of the waves are reproduced in the simulations. Furthermore, it was interesting to see that while some simulations (including the one in Figure 6a) reproduced a relatively stable large wave regime, in other cases (including the one in Figure 6b) the waves continued to merge, forming larger and larger waves until a slug was initiated. The latter behaviour is in agreement with the experimental observations by Woods et al. (2000) who reported observations of slug initiation from coalescing roll-waves.

![Figure 6: Comparison of simulated (Holmås, 2008) and observed (Johnson, 2005) time traces of the holdup in two-phase flow of water and SF6 at 8 bara.](image)

(a) 1° upward pipe inclination, $U_{SG} = 4.5$ m/s, $U_{SL} = 0.55$ m/s.
(b) 0.25° upward pipe inclination, $U_{SG} = 4.5$ m/s, $U_{SL} = 0.3$ m/s.

Figure 7 presents comparisons of simulated and measured wave heights, wave speeds, average holdup and average pressure gradient for 471 of the experimental cases reported by Johnson (2005). For each of these flow properties more than 90 % of the predictions are within a 20 % error margin, and in general the agreement between the simulations and the measurements is good.

In addition to the PhD project, a Watson-type travelling wave model (Watson, 1989) was developed for large waves in two-phase flow, and the governing equations were rigorously studied. One of the main findings was that it does not seem to be necessary to
account for the actual wave shape in order to obtain good predictions of the average holdup and pressure drop in the large wave flow regime. In fact, the pre-integrated stratified flow model described in Section 3 yields very similar results to those presented in Figure 7c,d, even though the individual waves are not resolved, and properties such as wave speeds and lengths are not predicted. The pre-integrated stratified flow model is thus also applied for the large wave flow regime.

Figure 7: Comparison of simulated (Holmås, 2005) and measured (Johnson, 2005) data with pipe inclinations: ○ horizontal; ● 0.1°; □ 0.25°; ■ 1° (upwards). (a) wave heights; (b) wave speeds; (c) average holdup; (d) average pressure gradient. The dashed lines indicate errors of ±20%.

5 SLUG FLOW

The slug flow model in the present work is based on the unit cell model concept, where a representative slug unit is modelled. This unit consists of a slug body zone, where the flow is a dispersed bubbly flow and a separated flow zone, where the flow is stratified or annular. Two important improvements over the previous approaches used in the commercial code were made. Firstly, the pre-integrated stratified flow friction model was implemented to replace the previous combination of friction correlations used in the separated flow zone. Secondly, the development of the flow behind the slug, the so-called tail profile, was incorporated (Taitel & Barnea, 1990). The result of these models is an effectively three-dimensional representation of slug flow, where the velocity profile is modelled in each cross-section, while the holdup and fluid velocities are allowed to vary through the slug unit.
In most commercial codes, slug flow is modelled by assuming that the separated flow zone behind the slug body is in a local equilibrium flow and has a spatially uniform holdup profile. However, as can be seen in experimental data, for example the high speed video images described in Section 2, the flow in the separated region is actually a developing flow. This is clearly exemplified in upward inclined pipes where the liquid layer has been seen to move forwards immediately behind the slug, but to reverse into backwards flow before the arrival of the next slug. Thus, it is inappropriate and in many cases inaccurate to use uniform equilibrium flow values to represent the developing flow along the separated region. Furthermore, the shape of the large gas bubble behind a slug is not really square, but a curved nose is generally observed at the bubble front (e.g. Cook & Behnia, 1997; Hale, 2000). Previous studies have suggested that the curvature and the cross-sectional location of the bubble nose are dependent on the Froude number (Fagundes Netto et al., 1999). For several reasons, therefore, it is desirable to use a model that accounts for the development of the holdup and phase velocities in the separated flow region.

In this work, a new point model has been developed for two-phase slug flows with the separated flow zone characterized by a tail-profile. The model for the separated zone is based on the two-fluid model for stratified flow with the friction laws given by the pre-integrated stratified flow model. The basic approximation in the model is that the flow in the separated region is a quasi-steady spatially-developing flow relative to the tail of the slug body. The hydrodynamic behaviour of the fluids behind a slug can therefore be calculated using a local incompressible steady-state two-fluid model. The tail-profile model is integrated numerically using a fast quadrature algorithm; examples of typical slug unit cells are shown schematically in 2D and 3D in Figure 8.

Figure 8: Schematic diagrams of a slug unit cell, (a) plane vertically through the pipe centreline; (b) 3D visualisation.

A preliminary extension of the model to three-phase systems was made using the gross simplification that the two liquids are homogeneously mixed without slip, with physical properties of the mixture based on linear interpolation between those of the two liquid phases. This provides basic three-phase functionality, but will be replaced with a more detailed model as part of our future work.

Figure 9 shows comparisons with experimental data of the mean pressure gradient over the slug unit cell calculated by the “tail-profile” point model and the “square-bubble” point model. The fraction of points within the error bounds and the root-mean-square values (RMS) are also shown. As can be seen, there is fairly good agreement for both models. The RMS error and the fraction of points within the error bounds for the two models are very similar, but the scatter of the points is much reduced for the tail-profile model. As shown, most of the calculations from the tail-profile model show underestimations of experimental data. A possible reason for the under prediction of the two-phase data is the exclusion of the extra friction due to turbulence production at the slug front in the current model; this will be considered as part of our continuing work. The degree of agreement seen for the three phase model is quite surprising given the
gross simplification used in this preliminary model. The prospect of obtaining better agreement with a more detailed three-phase flow model is therefore very encouraging.

![Figure 9: Comparison of mean pressure gradient with experimental data in two-phase (left) and three-phase (right) slug flows. ▲ tail-profile point model; □ square-bubble point model. The dashed lines indicate errors of ±20%.](image)

In this work, the mean liquid holdup in the bubble region, mean liquid holdup over the slug unit cell, slug fraction, and mean shear stresses are also calculated. These values have been compared with a large range of experimental data. In general, the tail profile model gives better agreement for mean liquid holdup in the bubble region and slug fraction. Nevertheless, the results are still not entirely satisfactory. The main reason is believed to be inaccuracies of the empirical correlations used for gas entrainment in the slug and slip between the phases in the slug body.

The slug flow model necessarily builds upon advances in the stratified flow model, so its progress lags somewhat behind the model for stratified flow. Our continuing work therefore focuses more on improvements in the slug flow model: incorporation of effects of developing flow on the friction in the slug body; detailed models for three-phase flow; replacement of the correlations used for gas entrainment at the slug front with mechanistic models; improved models for dispersed flow in the slug; mechanistic models for slug growth and decay; and improved dynamics in the separated flow zone.

### 6 FLOW REGIME TRANSITIONS

Commercial codes such as OLGA and TACITE, which are designed to run on coarse computational grids, require flow regime criteria in order to determine transitions between flow regimes. In the past, four different flow regimes have been used: stratified, slug, dispersed bubble and annular flow. As described in Section 4, a new flow regime named large wave flow was introduced in the present work. While the concepts behind the flow regime transition criteria for dispersed bubble flow and annular flow remain the same, new transition criteria have been developed to describe the transitions between stratified flow, large wave flow and slug flow. In addition to the minimum slip (MS) criterion, the linear stability of stratified flow is also taken into account; together, these are referred to as the Dual Transition Criteria.
The Dual Transition Criteria are based on the compound transition criterion formulated by Hurlburt and Hanratty (2002). In essence, this criterion states that for a given gas flow rate, stratified flow will exist as the liquid flow rate is increased until both (i) slug flow is stable and (ii) either inviscid (IKH) or viscous (VKH) Kelvin-Helmholtz analysis predicts an unstable liquid film. The IKH instability is believed to be of most importance for laminar flow, and is not included in the present work. The Dual Transition Criteria also make use of the ideas put forward by Soleimani and Hanratty (2003) and Holmås et al. (2008), wherein VKH instability can govern the onset of large waves or roll waves. Finally, the continuity of the flow regime in time and space, and the possibility of hysteresis are also taken into account when determining the local flow regime at a particular point in a simulation.

The Dual Transition Criteria are summarised as follows:

1. If the flow is currently stratified or large wave flow, it will continue as:
   a. **Stratified flow** if VKH predicts a stable liquid film
   b. **Large wave flow** if VKH predicts an unstable liquid film and MS predicts unstable slug flow
   c. **Slug flow** if VKH predicts an unstable liquid film and MS predicts stable slug flow

2. If the flow is currently slug flow, it will continue as:
   a. **Stratified flow** if VKH predicts a stable liquid film and MS predicts unstable slug flow
   b. **Large waves** if VKH predicts an unstable liquid film and MS predicts unstable slug flow
   c. **Slug flow** if MS predicts stable slug flow

![Figure 10: Predicted flow diagram together with experimental data points (Bendiksen et al., 1986) for horizontal flow of diesel and nitrogen at 30 bar in a pipe of diameter 18.9 cm.](image-url)

Figure 10 presents a typical flow pattern map for high pressure data based on the Dual Transition Criteria. The flow conditions correspond to the experiments reported by...
Bendiksen et al. (1986), and the experimentally observed flow regimes are indicated in the diagram. It can be seen that there is a small region at low gas flow rates where VKH predicts stable stratified flow and MS predicts stable slug flow. In this region, the continuity of the flow regime will determine whether the Dual Transition Criteria yields stratified or slug flow. For low pressure conditions, this region of ambiguity or hysteresis will typically be larger. There is a substantial region at high gas flow rates where large wave flow is predicted. Unfortunately, the experimental data do not distinguish between stratified flow and large wave flow, so it is not possible to assess the validity of the predictions based on this data set. At lower pressure the large wave region is typically smaller or it may vanish altogether. More comparisons can be found in Holmås et al. (2008).

7 PIPELINE SURGES

Surge waves in gas condensate pipelines are a well-known phenomenon. When starting up a pipeline or when the production level is increased, so called surge waves or oscillations in liquid flow at the pipeline outlet are observed. These oscillations are very slow, with a typical period of the order of 1 hour and they may persist over a day or two for a 100-200 km pipeline before the liquid flow stabilizes. The observed oscillations are due to liquid mass waves propagating down the pipeline with a velocity fairly close to the liquid transport velocity.

The physics behind these very long waves has not been well understood. Moreover, commercial transient multiphase simulators are not able to satisfactorily simulate and predict these phenomena in gas condensate systems. In production systems where the liquid storage capacity is limited, it is important to be able to predict the surge wave oscillations under different operating conditions, as this can have important consequences for the operation of the pipeline.

We have carried out basic research in order to understand and predict the surge wave oscillations in a better way. Since the wavelengths we are looking at are very long, the wave propagation is described by the kinematic wave equation which is the long wavelength asymptotic limit of the incompressible two phase equations. The surge waves are essentially friction-dominated waves in contrast to the large waves described in Section 4, which are gravity-dominated.

The two-phase surge waves are always stable, and they will eventually disappear due to nonlinear distortion. However, this is a slow process, and long surge waves can easily propagate over a distance of 50-100 km.

[Figure 11: Illustration of surge wave formation and propagation.]
Since the waves are stable, a disturbance is needed in order to initiate the surge waves. This initial disturbance is provided by the change in production level. The simple simulation example shown in Figure 11 illustrates what happens. The figure shows the holdup profile in a 7 km section of a pipeline, where there is an uphill section between the 1 and 2 km mark. The equilibrium holdup at the low production level in the uphill section is fairly high (Curve A). The equilibrium holdup for the new higher production level (Curve B) is lower. When the production is increased, the surplus liquid (the difference between curves A and B) is ejected from the incline and starts propagating downstream as a wave (shown at two later points in time C and D in the Figure).

The dynamics in a real pipeline are of course much more complex. Nevertheless, it is clear that a change in production leads to a change in the equilibrium holdup in upwards-inclined pipe sections. This releases a surge of liquid, leading to the initial wave disturbance which then propagates downstream.

8 CONCLUSION

The work reported here represents a significant step forward in the modelling of multiphase pipeline systems. The basic one-dimensional dynamic equations have been known for many years; they are robust but require closures for friction and dispersion characteristics. Historically these closures have been based on correlations of experimental data yielding limited ranges of validity and poor scalability to field conditions. The resulting one-dimensional models have also had a limited range of validity, and often fail under field conditions. As a result, people have come to distrust one-dimensional models, or even to give up on them altogether. Here we have followed a different approach where the closure models are obtained through integration of more detailed physical models. The resulting mechanistic models give very good agreement with data and have much better, built-in scale-up properties.

The first step was to produce a detailed analytical model for the profiles of velocity and turbulence in stratified flow. This model couples together the flow of the different phases through continuity of stress and velocity at the interfaces and integration produces a set of mutually consistent interdependent frictions at the interfaces between the fluids and at the wall. This replaces the ad-hoc, inconsistent, mutually independent correlations that have been used in the past. The pre-integrated stratified flow model is tuned to a single set of data for low pressure air-water flow in a small diameter pipe and, without further tuning, produces significantly better agreement with data for other fluids in large diameter pipes than other models that have been specifically tuned to such data.

In subsequent steps, we have shown how the pre-integrated approach can be adapted to produce consistent models for large wave flow and for slug flow. These models also show much improved agreement with experimental data, but are not yet developed to the same extent as the stratified flow model. Work continues with the aim of producing a consistently high quality flow model in which an underlying three-dimensional analytical representation of the flow is used to provide closures for a high-accuracy high-speed one-dimensional flow solver.

ACKNOWLEDGEMENT

This paper describes work carried out in the HORIZON programme at the Institute for Energy Technology (IFE) and SPT Group. The HORIZON JIP is sponsored by ENI, ExxonMobil, StatoilHydro, Shell, Chevron and SPT Group.
REFERENCES


