30th International North Sea Flow Measurement Workshop
23 – 26 October 2012

Using CFD to Understand Multiphase and Wet Gas Measurement

Neil Barton, NEL
Andrew Parry, Schlumberger

ABSTRACT

Computational fluid dynamics (CFD) flow modelling has proved to be a useful tool in single-phase flow metering. More recently, the increase in computer power has made multiphase models viable. However, there is a range of modelling approaches, which makes practical simulation of the behaviour of multiphase and wet gas flows in metering and/or sampling devices as much an art as a science.

This paper introduces alternative modelling approaches to show what can be achieved using CFD modelling and to highlight the limitations of the technique. A series of case studies are presented in which CFD analysis has been used to study the following issues in multiphase and wet gas measurement or sampling:

- Effects of heavy oil multiphase flow in a Venturi
- Behaviour of multiphase swirl flow in a Venturi
- Modelling of wet gas flows in Venturis, cone meters, and orifice plates
- Effective sampling methods

BACKGROUND

CFD modelling is a computer simulation method that predicts fluid flow in complex two-dimensional and three-dimensional domains. Modern CFD modelling started in the 1970s, with early work mostly taking place in the aerospace sector. In the 1990s, CFD modelling moved from being an academic curiosity to a widely used engineering design and analysis tool. Since then, the technique has been applied in a wide range of industries, including the process, automotive, medical, electronics, architectural, and oil and gas sectors.

CFD modelling has now largely been accepted as having a place in single-phase flow metering. It has been used to assess meter installation errors [1], incorrect installation [2], fouling effects [3, 4], erosion [5], thermal effects [6], and flow conditioner designs [7]. The performance of flowmeters with moving parts can be simulated using fluid-structure interaction methods [8]. Two-phase and multiphase CFD modelling is a more challenging proposition. However, the ever increasing computer power has started to make the modelling of flow meters in wet gas and multiphase flow a practical proposition.

The first part of this paper presents a basic overview of alternative modelling approaches. A series of case studies then illustrates how CFD modelling can be used to address particular problems, and the limitations of CFD modelling are also identified.

MULTIPHASE CFD MODELLING TECHNIQUES

A number of commercial and open-source CFD software packages are currently available. The more popular software that can model multiphase flow includes ANSYS Fluent, ANSYS CFX, Adapco STAR-CCM+, Flow3D, and OpenFOAM. These software packages typically comprise a pre-processor for geometry creation and problem setup; a solver, which calculates the solution; and a post-processor, which displays the calculation results.

Single-Phase Flow Model

Figure 1 illustrates the CFD modelling technique for a single-phase liquid flow through a simple nozzle in a pipe. The first step in the calculation is to define the shape of the nozzle boundaries. The space occupied by the liquid is divided into a mesh of small cells. Boundary conditions, such as inlet flow velocity and outlet pressure, are defined and the fluid density and viscosity are set.

This information is used by the solver, which calculates flow variables such as pressure and velocity in each cell. The results can then be viewed in the post-processor software in terms of flow vectors, streamlines, and coloured contours of the computed variables. Point values of variables and graphs can also be extracted from the solution data.

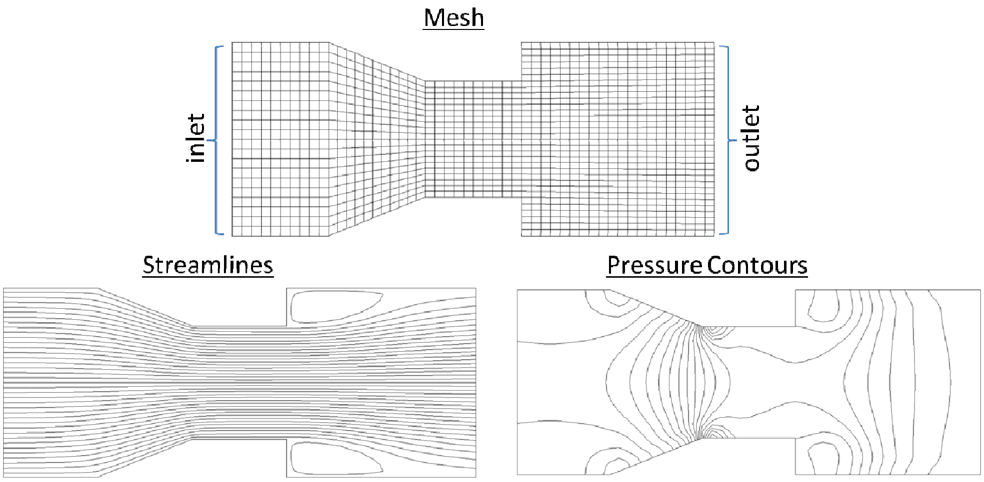


Figure 1. Illustration of setup and predictions for a single-phase CFD model of a nozzle.

Various alternative approaches can be used to simulate the flow of multiphase mixtures. There are numerous variations, but the main modelling methods are homogeneous flow approximation, Lagrangian particle modelling, volume-of-fluid modelling, and Eulerian modelling.

Homogenous Flow Approximation

The simplest approach to multiphase CFD modelling is to treat the multiphase mixture as a homogeneous (i.e. well-mixed) single-phase fluid. For two-phase gas-liquid flow, the mixture density and viscosity are calculated using the gas volume fraction (GVF) thus:

$$\rho_{\text{mixture}} = \text{GVF}.\rho_{\text{gas}} + (1-\text{GVF}).\rho_{\text{liquid}} \tag{1}$$

$$\mu_{\text{mixture}} = \text{GVF}.\mu_{\text{gas}} + (1-\text{GVF}).\mu_{\text{liquid}} \,. \tag{2}$$

Once fluid parameters have been defined, the model is run as a single-phase simulation.

Lagrangian Particle Model

Figure 2 illustrates Lagrangian particle modelling (also known as discrete particle modelling (DPM)). In this case, a carrier fluid is modelled as a single-phase CFD model. A second phase is modelled as individual spherical particles that are tracked, step-by-step, through the nozzle as they are carried by the surrounding fluid. By altering the density of the particles, they can be representative of gas bubbles, liquid droplets, or solid particles such as sand.

Particles are affected by drag and buoyancy. They can also impart drag on the carrier fluid. They bounce off wall surfaces, but they have a fixed, user-defined size and do not bounce off each other or coalesce. Hence, this model is only appropriate for low fractions of the second phase and in applications where particle, drop, or bubble size is well defined and changes little.

Figure 2 shows gas bubble tracks and liquid streamlines through the nozzle. The bubble trajectories are partly determined by drag from the liquid and partly by buoyancy effects that cause the bubbles to rise to the top of the pipe.

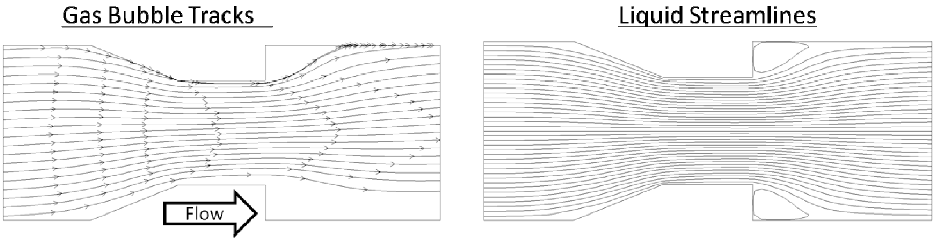


Figure 2. Illustration of particle model representation of bubbly flow.

Volume-of-Fluid Model

Volume-of-fluid (VOF) modelling is typically used to model free-surface flows such as calculation of hull form in ship design and calculation of stratified flows, as illustrated in Figure 3. In VOF modelling, in addition to the normal single-phase flow parameters, the liquid fraction in each cell is calculated. Each cell can contain either liquid or gas, and slip occurs between the two phases at the interface.

In Figure 3, the wave in the liquid level restricts the gas flow, causing tightly packed streamlines in the top half of the nozzle throat.

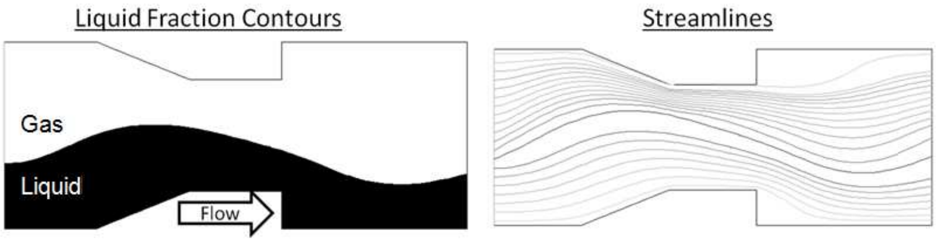


Figure 3. Illustration of predictions for a stratified-flow VOF model.

Eulerian Model

In the VOF model, each cell contains one of the two possible phases. In contrast, the Eulerian model can be viewed as modelling a cloud of droplets or bubbles in a carrying fluid, with the volume fraction varying continuously from 0 to 1 in each cell. Each phase has its own velocity, and slip between phases is controlled by setting a fixed droplet or bubble size.

Figure 4 shows a wet gas flow calculation in which liquid droplets impinge on the nozzle to form a film and a liquid-rich jet downstream. Slip between the phases causes a small difference in the liquid and gas streamlines.

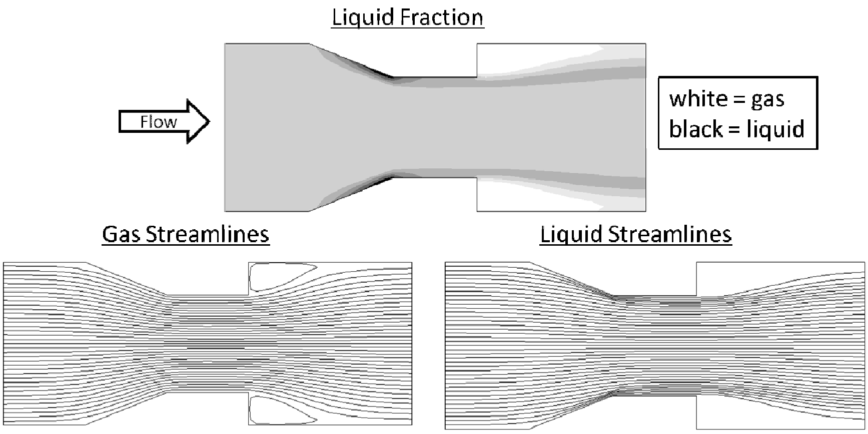


Figure 4. Illustration of predictions for a wet gas Eulerian model.

In principle, VOF modelling can also model droplets and wall films, provided that the computational mesh is fine enough to resolve their edges. The Eulerian model addresses this issue up to a point, provided that an appropriate droplet or bubble size can be defined. However, the Eulerian model tends to blur or smear out sharp liquid/gas interfaces.

EXAMPLE 1—WATER-IN-OIL SAMPLING

CFD methods are currently being employed at the National Engineering Laboratory (NEL) to study how sub-isokinetic and super-isokinetic sampling rates and upstream mixing from bends affect oil sampling systems. In the latter case, the conventional approach is to use empirical methods specified in ISO 3171 [9] to determine whether bends upstream of the sampler provide sufficient mixing to ensure that the sampled oil has a similar water cut to the oil in the main pipeline. However, the ISO-3171 method can be highly conservative; it is based on very limited laboratory test data and only applies to horizontal pipes.

In the mid-1980s, British Petroleum performed tests on a sampling system at one of their refineries. CFD predictions have been compared against published data from these tests [10] to assess the practicality and viability of replacing or supplementing the ISO-3171 methodology with CFD methodology. In the tests, water concentration profiles were measured across the diameter of a 48-inch pipe downstream of a combination of a bend, tees, and valve, as shown in Figure 5.

Based on past experience, it was believed that the water would be in the form of dispersed droplets, with the possibilities of a stratified stream of water running along the bottom of the pipe in some locations. Alternative CFD simulations using the Lagrangian and Eulerian models were compared against the field data.

Both modelling methods required definition of a representative water droplet size. Various approaches exist, but there is no known robust and universal method for doing this. In this work, a range of different droplet sizes were tried. Figure 6 shows typical results. Very large droplets (> 5000 microns) cannot be lifted up into the 48-inch header. The 2000-micron droplets are mixed by the upstream bends but start to stratify before the sampling point. Figure 7 shows a reasonable agreement between field measurements and both CFD methods if the droplet size is set to 300-micron droplets.

This example flags a number of issues common to many multiphase CFD applications. It is clear that the predictions are sensitive to the droplet size. However, fairly good results can be achieved if this value can be well estimated. Also, droplets may break up or coalesce in reality, and accounting for this adds extra complication. Finally, it should be noted that the concentration measurement method used and its uncertainty are not known. Multiphase measurements are often difficult; hence, tuning a simulation to match test data should only be done with care.

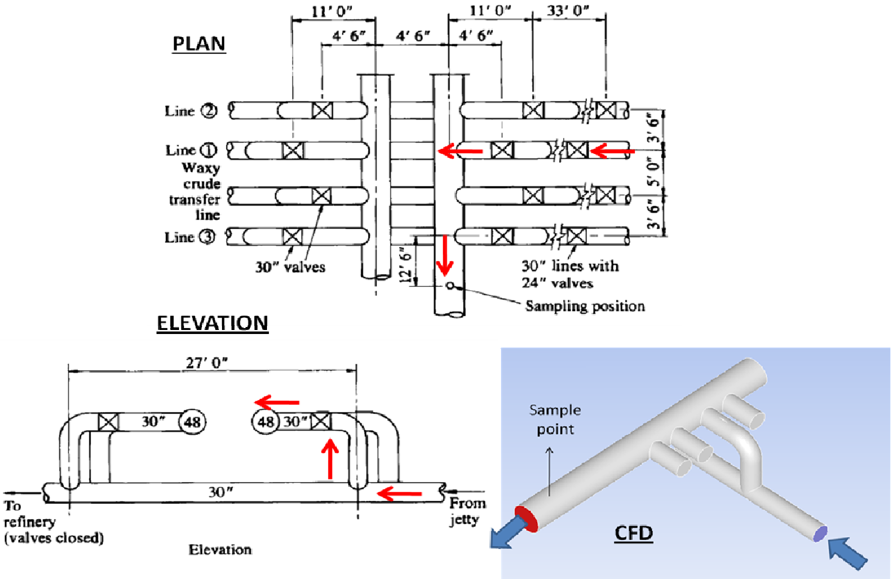


Figure 5. Geometry of pipework in sampler tests.

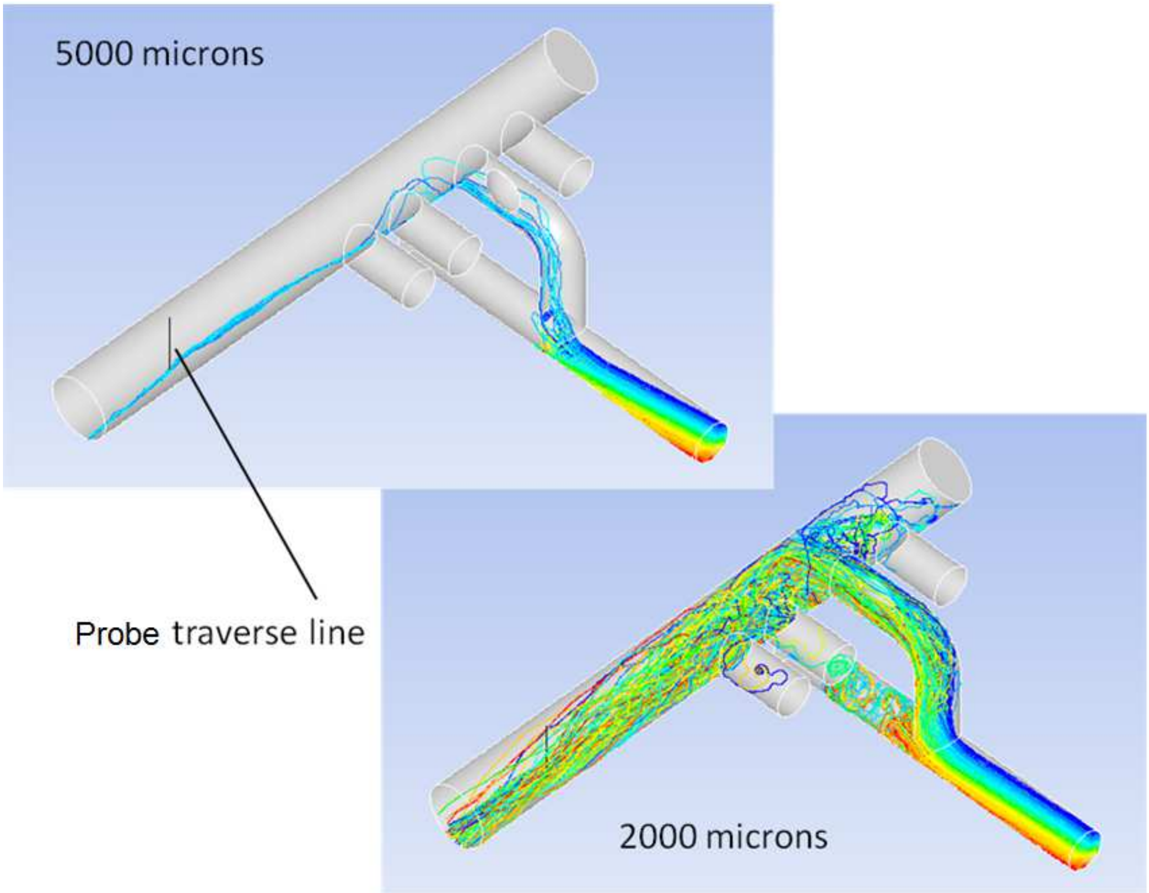
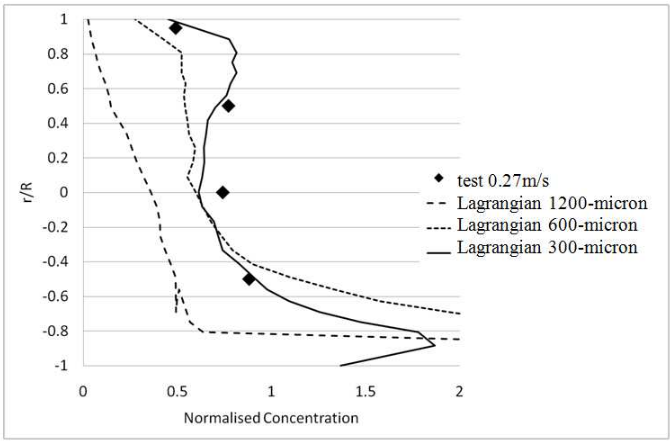
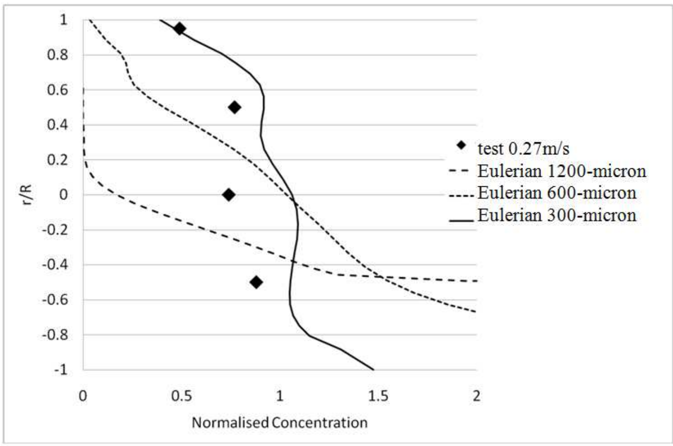


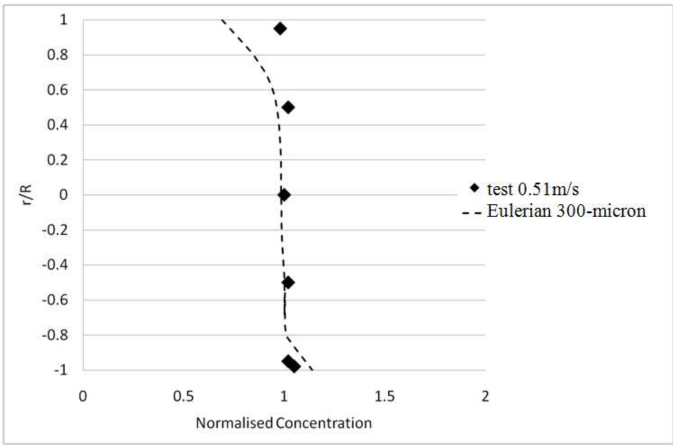
Figure 6. Lagrangian particle trajectories of different-sized water droplets in low-flow-rate sampler assessment simulations.



a) Low flow rate, 2.5% water cut, Lagrangian model



b) Low flow rate, 2.5% water cut, Eulerian model



c) High flow rate, 3.5% water cut, Eulerian model

Figure 7. Comparison of CFD model predictions and field measurements of water concentration for different water droplet sizes, r/R is vertical coordinate normalised by pipe radius.

EXAMPLE 2—THE EFFECT OF HEAVY OIL ON MULTIPHASE FLOW IN A VENTURI

A cooperative project between Schlumberger and NEL (part-funded by the Technology Strategy Board (TSB) [11]) looked at the effects of heavy oil on the measurements of Venturis in liquid-gas flow [12]. In that project, a 4-inch Perspex™ Venturi was calibrated in a water-nitrogen and oil-nitrogen flow in vertical and horizontal orientations. The oil viscosity ranged up to 300 cP. The CFD predictions were compared against test measurements and observations. CFD modelling was also used to gain a better understanding of the test results and to extrapolate test results to liquids with much higher viscosities.

The liquid fraction in these tests varied from 100% to about 5%; hence, the Eulerian and VOF models were chosen for this work. A bubble size of 1 mm was set in the Eulerian model in the hope that this would allow sufficient separation of the liquid and gas whilst accounting for the expected entrainment of bubbles in the liquid.

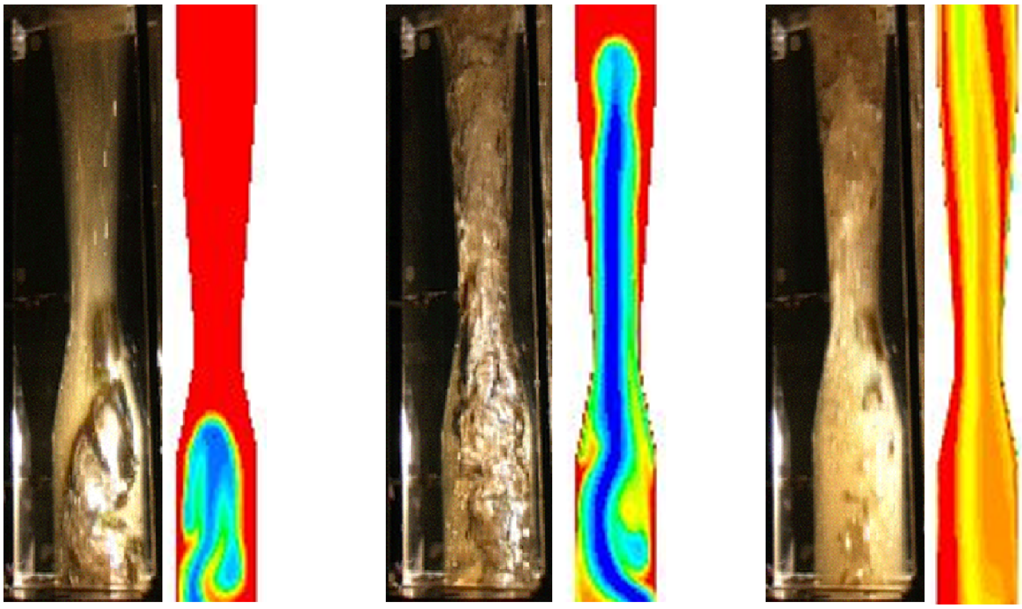
The test was set up such that, over most of the test matrix, the flow entering the meter was slugging. Hence, transient (time varying) simulations were required. Transient models require large computational resources. To mitigate this, a relatively low-resolution (coarse) mesh was used. It was found that the VOF and Eulerian models produced similar results. This was probably partly because the mesh resolution was insufficient to resolve sharp liquid/gas interfaces in the VOF model. Only Eulerian results are shown in this paper.

Figure 8 shows a typical comparison of the Eulerian CFD model and test images for a vertical Venturi. The CFD model generally produced good predictions of the liquid distribution and correctly described the transition from churn to slug flow. Slug flow was mimicked by using prerecorded test data to vary the height of a liquid layer at the bottom of the pipe at the meter inlet. This was partially successful— time-averaged values were fairly well predicted (e.g. the discharge coefficient as shown in Figure 9). Instantaneous values were not as well predicted, probably because the definition of slug flow was too simplistic. In particular, the CFD model struggled to match the instantaneous behaviour of a gamma densitometer downstream of the Venturi, which was found to be sensitive to small changes in liquid distribution.

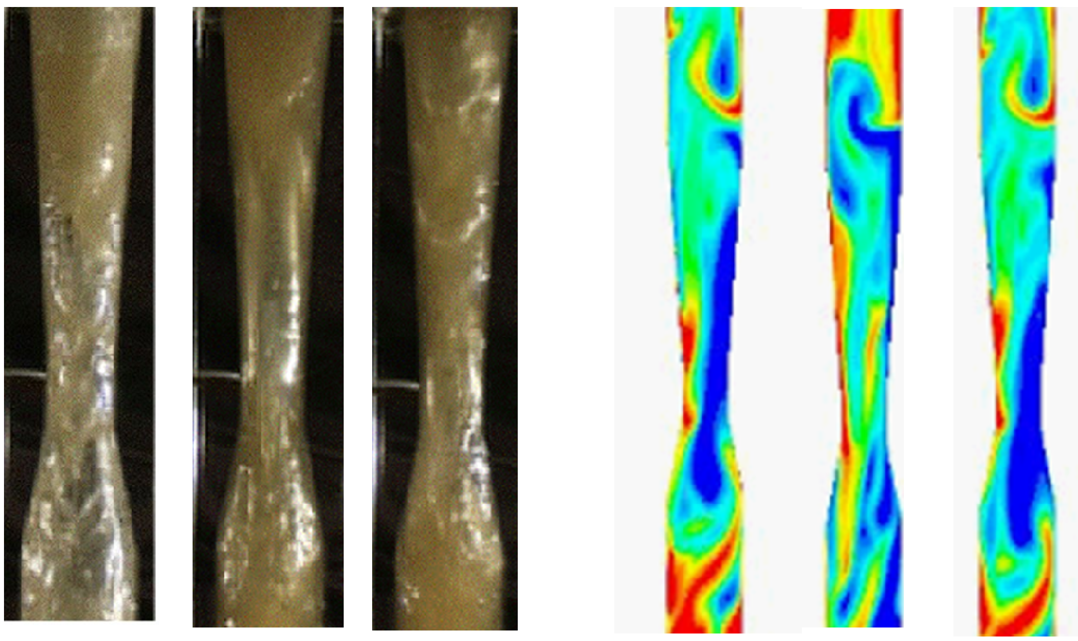
In general, it has been found that CFD modelling is best at predicting time-averaged differential pressure measurements. Instantaneous point or line values, such as those taken by the diametric beam of a gamma densitometer, can be more difficult to predict.

One of the advantages of CFD modelling is that many parameters can be simply changed or controlled in a manner that would be very difficult or expensive to achieve in the laboratory. Figure 10 shows the results of further simulations in which the liquid viscosity was increased significantly beyond that in the tests (10,000 cP). It can be seen that low-viscosity liquid tends to churn, whereas high-viscosity fluid sticks to the meter walls, causing an annular flow.

In a second exercise, a simple square-wave slugging behaviour was defined at the meter inlet for water-gas and oil-gas flow. This simplified behaviour allowed a perfect like-for-like comparison of oil and water flow in a manner that could not be achieved experimentally (Figure 11). It was found that the ratio of the two-phase discharge coefficient to the liquid discharge coefficient is independent of the liquid viscosity (Figure 12). Hence, although some details of the CFD predictions varied from test results, they were found to be sufficiently accurate to demonstrate the basic physics of the phenomenon of interest and, hence, to derive a correction methodology.



a) Vertical Slug Flow



b) Vertical Churn Flow

Figure 8. Liquid distribution as seen in tests and CFD simulations (red = liquid, blue = gas).

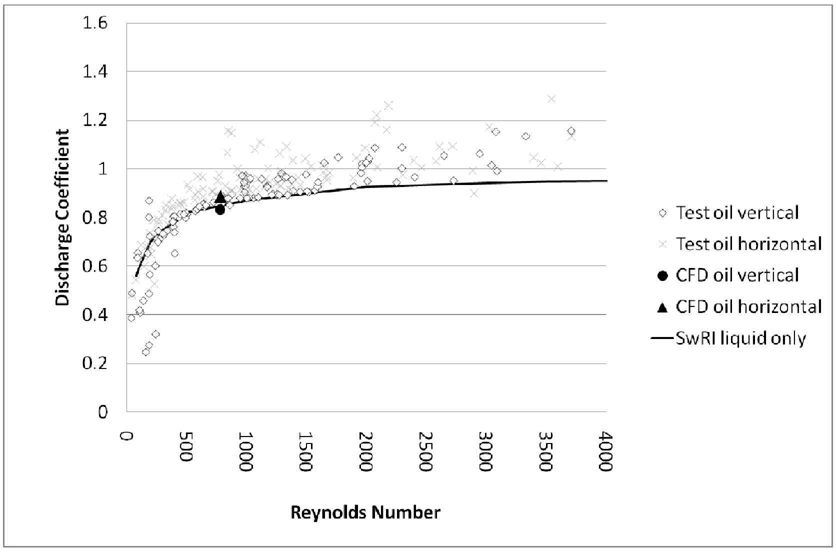
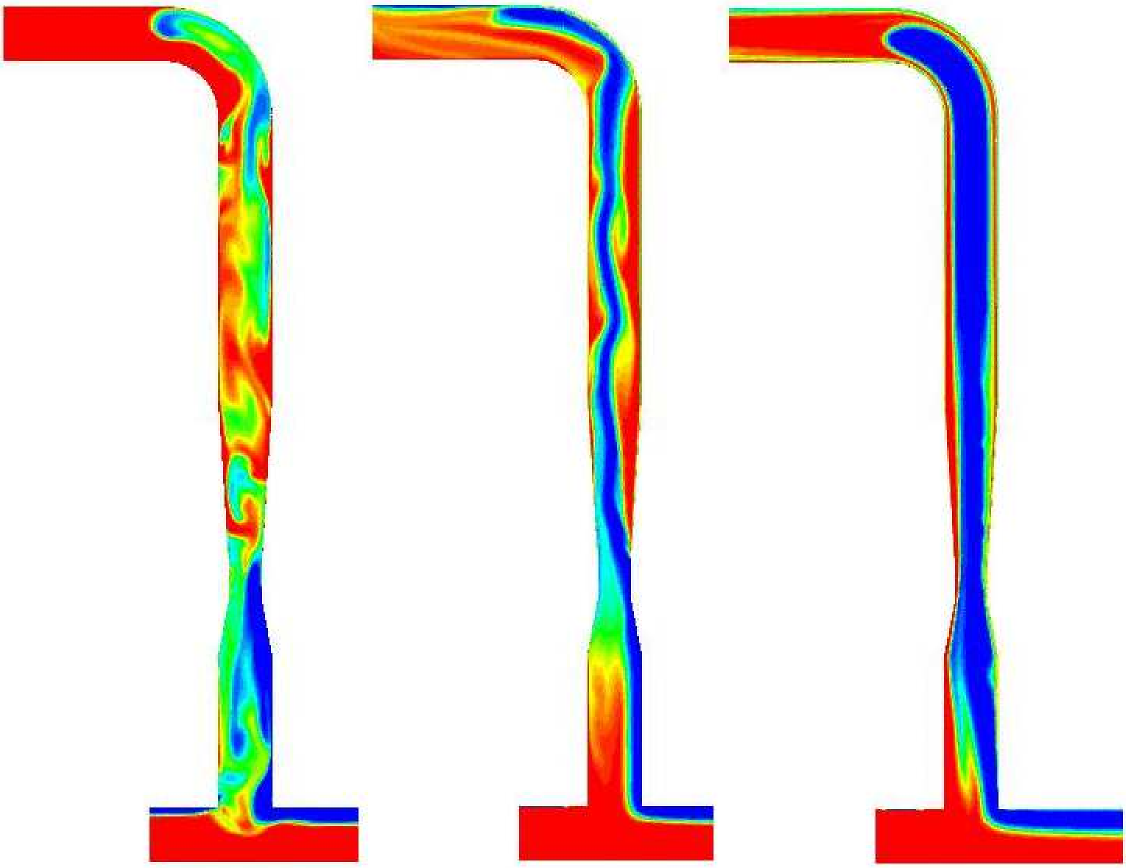


Figure 9. Discharge coefficient plotted against mixture Reynolds number for Venturis in oil and oil-gas flow. SwRI – Southwest Research Institute



1 cP **300 cP** **10,000 cP**

Figure 10. CFD model predictions showing how increasing viscosity suppresses churn flow (red = liquid, blue = gas).

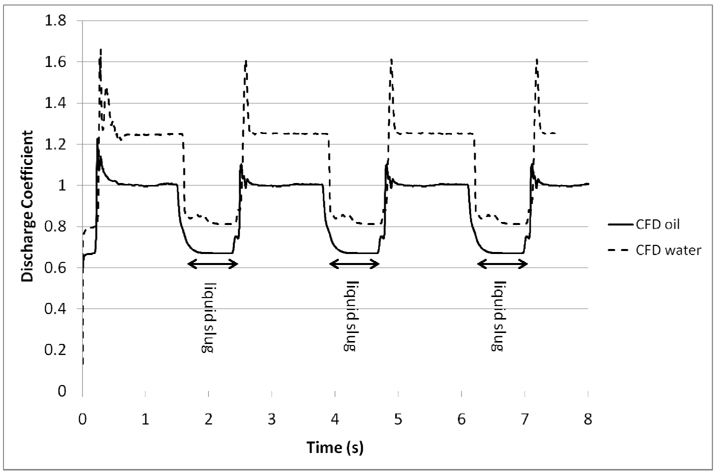


Figure 11. CFD predictions of discharge coefficient for oil and water (vertical flow).

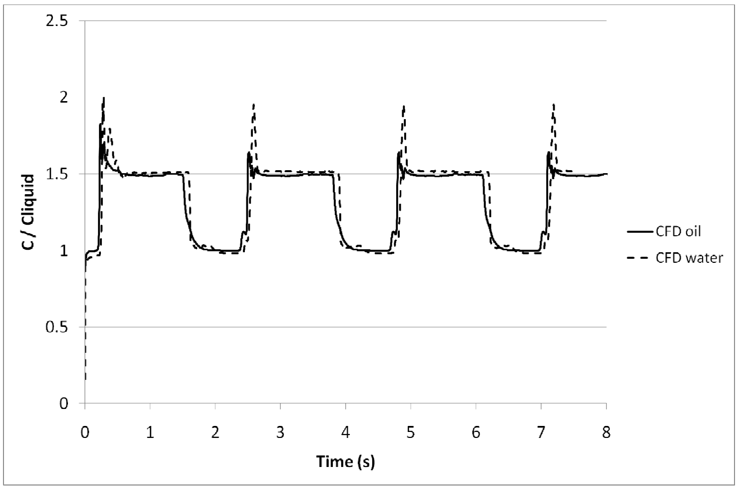


Figure 12. CFD predictions of normalised discharge coefficient.

EXAMPLE 3—THE EFFECT OF SWIRL ON MULTIPHASE FLOW IN A VENTURI

A second research project funded by Schumberger, NEL, and TSB investigated an experimental apparatus which used a swirl generator to condition multiphase flow entering a Venturi.

During testing with oil and water mixtures with a fixed total liquid volumetric flow rate, it was found that the discharge coefficient suddenly stepped down at the phase inversion point, as shown in the test results in Figure 13. Because the pure oil is about ten times more viscous than the water, a higher differential pressure and a lower discharge coefficient would be expected in oil for a standard Venturi. The reason the experimental apparatus displayed the opposite behaviour was not fully understood.

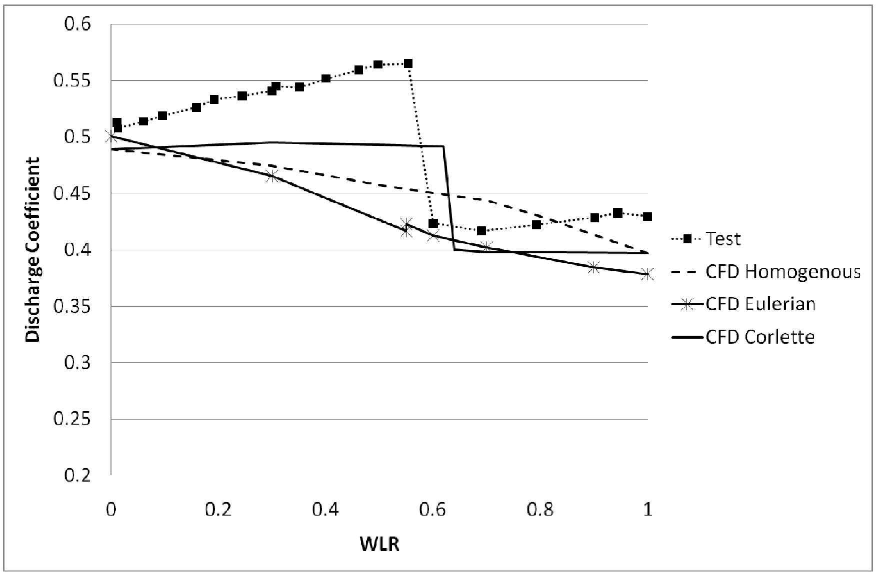


Figure 13. Measured and predicted discharge coefficient.

Initial CFD models represented the oil-water mixture as a simple homogeneous fluid, using the water/liquid ratio (WLR):

$$\rho_{\text{mixture}} = \text{WLR}.\rho_{\text{water}} + (1-\text{WLR}).\rho_{\text{oil}} \tag{3}$$

$$\mu_{\text{mixture}} = \text{WLR}.\mu_{\text{water}} + (1-\text{WLR}).\mu_{\text{oil}}\ . \tag{4}$$

However, as shown in Figure 13, this model did not pick up the step at the inversion point.

A second approach used the Eulerian model. Again, this failed to pick up the step (over a range of droplet sizes).

A third approach assumed a homogeneous fluid density, but the fluid viscosity was varied on the basis of the mean WLR according to a model developed by Anne Corlette at NEL [13] (Figure 14):

$$\rho_{\text{mixture}} = \text{WLR}.\rho_{\text{water}} + (1-\text{WLR}).\rho_{\text{oil}} \tag{5}$$

$$\mu_{\text{mixture}} = f(\text{WLR})\,. \tag{6}$$

Figure 14 shows that this approach predicted a step of roughly the correct magnitude at the inversion point. Further simulations demonstrated that swirl entering the Venturi depends on the fluid viscosity at the walls (Figure 15). High viscosity dissipates swirl entering the Venturi, reducing the differential pressure and, thus, increasing the discharge coefficient. In oil-continuous flow, the viscosity and discharge coefficient are high. In water-continuous flow, water is centrifuged to the walls and forms a lubricating film. Swirl entering the Venturi is high, resulting in a low discharge coefficient.

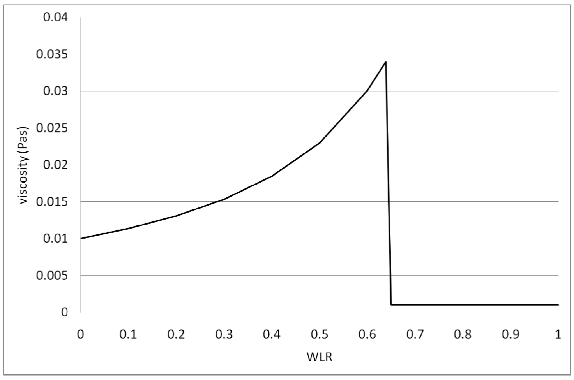


Figure 14. Mixture viscosity predicted by the Corlette model.

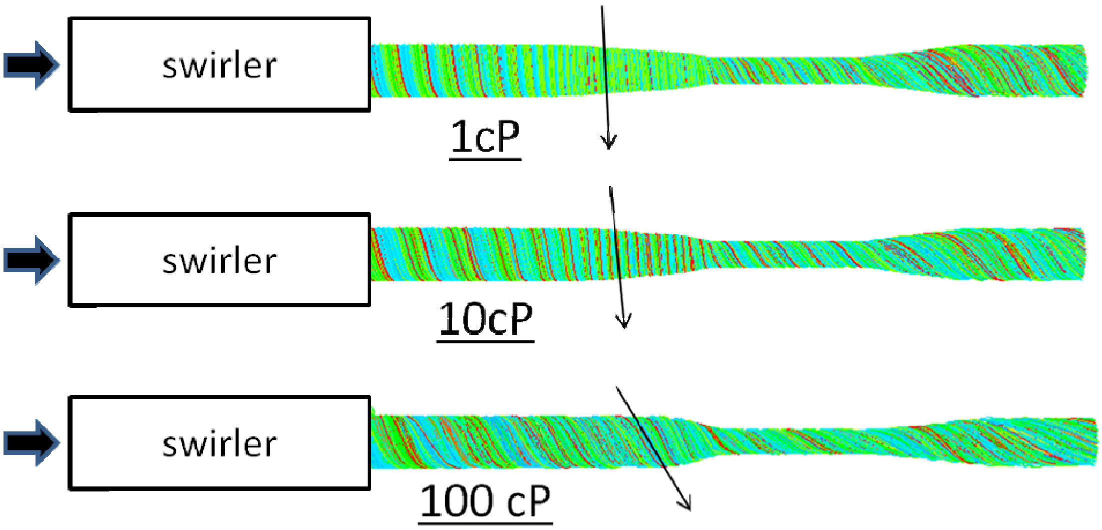


Figure 15. Streamlines showing swirl entering the Venturi reduces as the fluid viscosity increases.

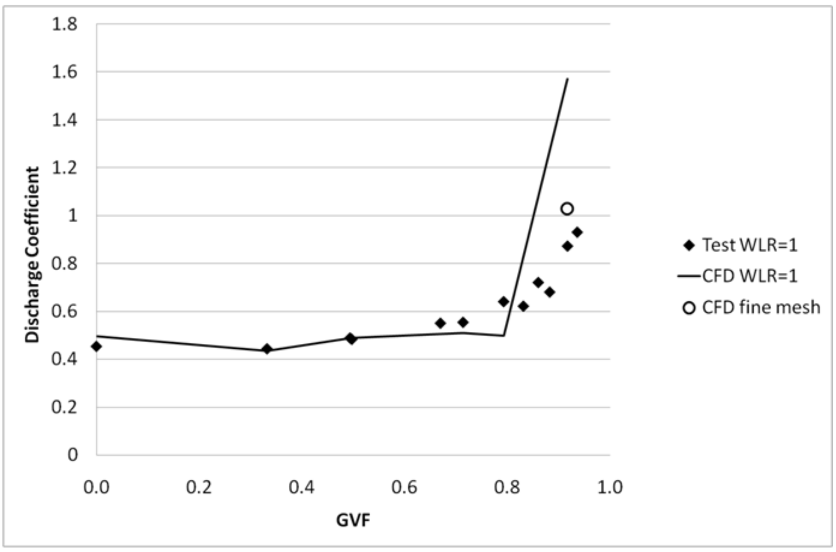


Figure 16. Comparison of Eulerian water-gas simulations with test data.

Further work modelled liquid-gas swirl flow through the Venturi. The CFD predictions agreed well with the tests in oil-gas flow. In water-gas flow, agreement was good except at high gas fractions (Figure 16). This was because thin wall films were not well resolved by the computational mesh. This was confirmed when a finer mesh produced much better predictions. Interestingly, this was not an issue in oil-gas flow because the more viscous oil produced a thicker film that could be adequately represented by a coarse CFD mesh.

This example illustrates how simple homogeneous models can give useful information on multiphase flows. It also shows that discrepancies between CFD and test data can give clues about the physical processes in the meter—a poor prediction is often as informative as a good prediction.

EXAMPLE 4— WET GAS MODELLING

Accurate CFD modelling of wet gas flow differential pressure meters is a challenging proposition. Meter over-reading depends on the fraction of liquid in droplet form and in the wall-film, and this depends on very small-scale droplet entrainment and film coalescence processes. Progress has been made in modelling wet gas flows using VOF-type approaches [15]. However, this requires a very fine mesh and significant computing power. An alternative approach is to use the Eulerian model with an approximate “effective” droplet size to mimic wet gas behaviour. The methodology is as follows:

1. An Eulerian model of a differential pressure metering device is set up that assumes homogeneous mist flow at the device inlet (regardless of the true flow regime).

2. For a set gas Froude number (Fr_g) and one Lockhart-Martinelli number (X), the droplet size is adjusted until the predicted over-reading matches test results at identical conditions.

3. This effective droplet size is assumed to depend on the operating pressure and the gas Froude number only. Different differential-pressure flow-metering device designs operating at different Lockhart-Martinelli numbers use the same effective droplet size.

Figure 17 illustrates the results of this type of analysis. For a beta-0.6 Venturi, the droplet size is adjusted at a Lockhart-Martinelli number, $X = 0.3$, and $Fr_g = 1.5$ until the over-reading matches test values. When X is reduced, but droplet size is kept constant, test and CFD results agree well. Similarly, droplet sizes set at $X = 0.3$ and $Fr_g = 2.5$ and 3.5 agree well with tests. Figure 18 shows that droplet sizes tuned for a beta-0.6 Venturi at $X = 0.3$ work well for a beta-0.75 cone meter. It has been shown that these values also work well for Venturis, orifice plates, and cones over a range of common betas.

Note that the effective droplet size is a measure of fluid drag, not the droplet size that would occur in a physical test. It is probably reasonably realistic in mist flow. As would be expected, errors increase when the true flow regime is stratified. However, despite the fairly gross approximations in this model, it produces surprisingly accurate results in all regimes. This indicates that the flow regime is not a major factor in determining the over-reading of wet gas meters based on differential-pressure sensing devices.

As an extension of this work, a Venturi has been modelled immediately downstream of a combination of a long straight pipe, a single bend, and a double out-of-plane bend. At Lockhart-Martinelli numbers of 0.3, flow maps indicate that the flow regime will be predominantly mist flow and, hence, the mist-flow CFD model should be fairly representative. Figure 19 shows that an upstream bend centrifuges liquid droplets to the wall and most of the liquid enters the Venturi as a wall-film. (Note, the contour scale in Figure 19 has been exaggerated to clearly show this effect.) This changes liquid drag in the Venturi throat and reduces the over-reading by about 5% at $X = 0.3$ (Figure 20). The effect of a double bend is even greater; however, the effect is not predicted at lower liquid fractions ($X > 0.1$).

This effect has not been confirmed experimentally, and upstream installation effects have not been widely studied in wet gas flow. However, it would be interesting to know whether this finding is in line with test results.

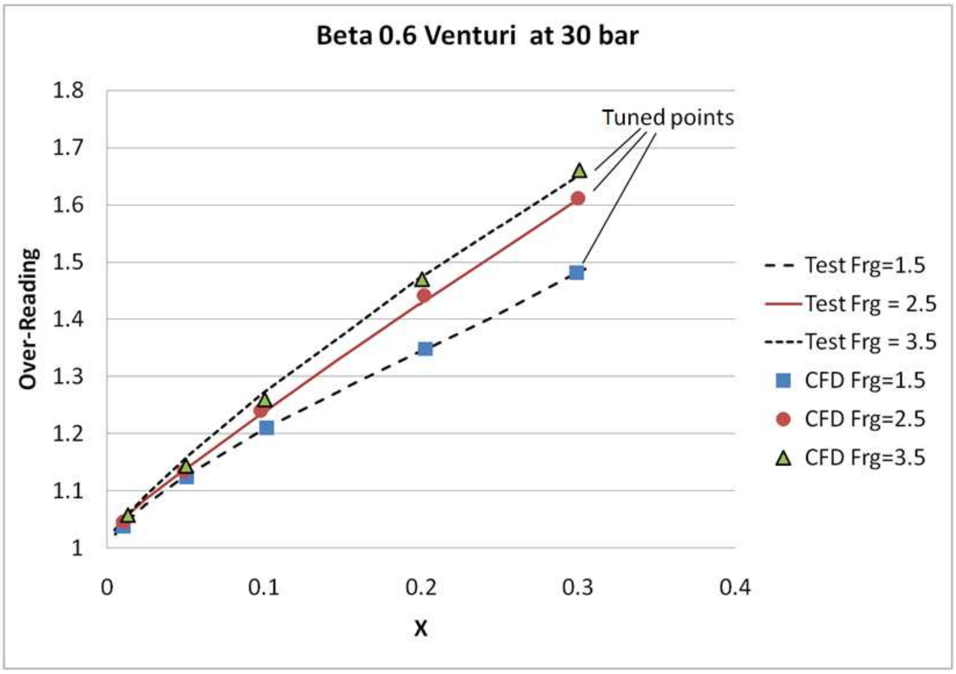


Figure 17. Predicted and measured over-reading for a beta-0.6 Venturi at 30 bar.

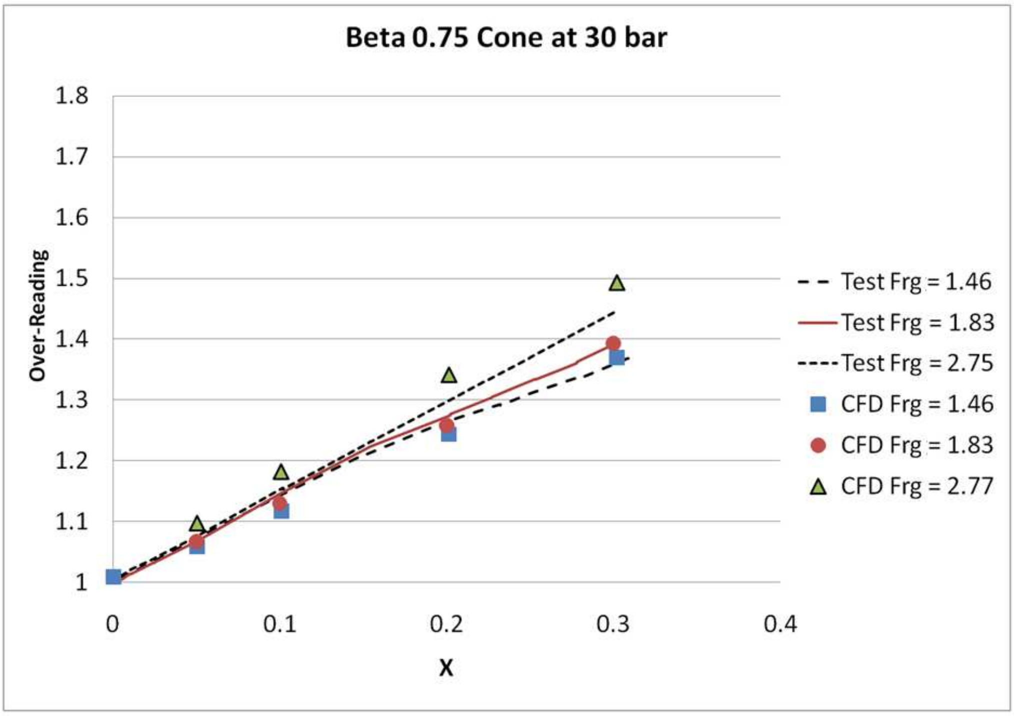


Figure 18. Predicted and measured over-reading for a beta-0.75 cone meter at 30 bar.

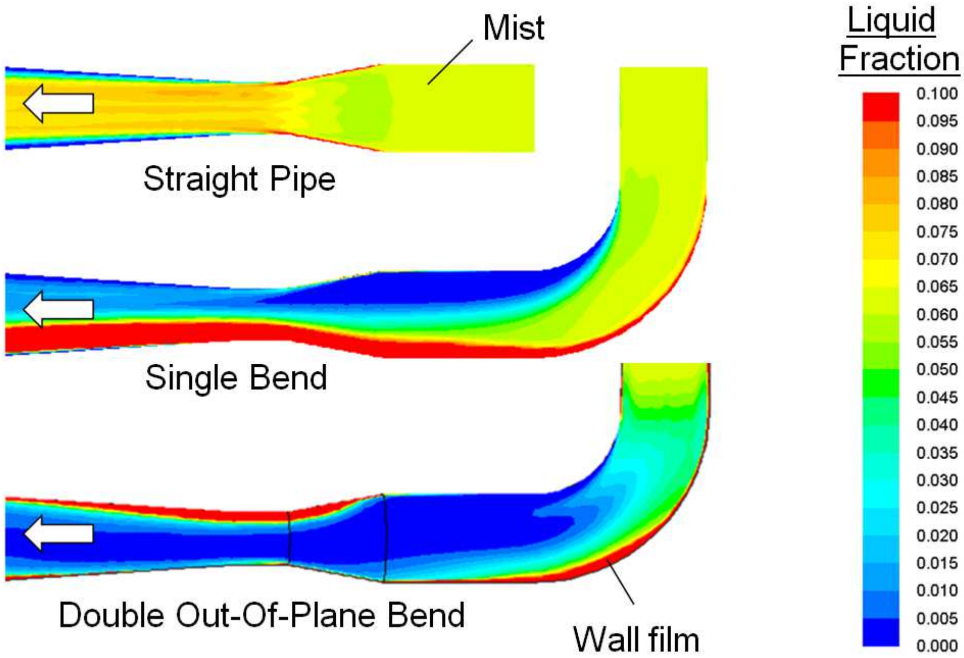


Figure 19. CFD predictions of liquid distribution in a wet gas Venturi downstream of a straight pipe, a single bend, or a double bend.

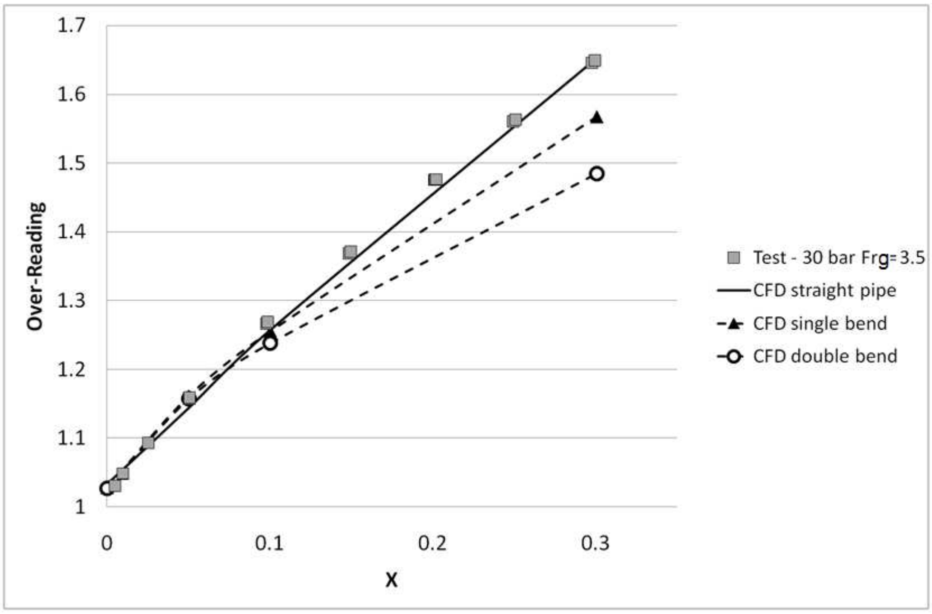


Figure 20. Predicted and measured over-reading for a Venturi downstream of a straight pipe, a single bend, and a double out-of-plane bend.

CONCLUSIONS

CFD modelling has been shown to be a useful tool in understanding the behaviour of flowmeters in single-phase flow. An increase in computational power has now made the use of simplified multiphase CFD models a viable proposition. This paper has shown that, if used carefully, these simple models provide revealing insight into multiphase flow behaviour that can improve metering systems.

CFD analysis is particularly useful for extrapolating experimental results to larger-scale, higher-flow-rate, or higher-pressure conditions that cannot be practically achieved in the laboratory. It is good for diagnosing confusing test results. Sometimes a discrepancy between simulations and tests can be more revealing than good agreement.

Depending on the complexity and sensitivity of the problem, CFD predictions can be used in a quantitative manner (e.g. predicting measurement errors within a few percent) or in a qualitative manner (e.g. showing that one design of meter is less sensitive to problems than another). However, to do this with confidence, a good understanding of the specific application is required.

In theory, CFD predictions of multiphase flow should be as accurate as predictions for single-phase flow. In practice, some multiphase flows are heavily influenced by small-scale effects, such as droplet breakup in wet gas. Current computing power is still not sufficient to model these small-scale effects. The simplified multiphase models discussed in this paper rely on spatial- and time-averaging approaches using approximate values for unknown parameters to reduce computational requirements. These limitations and the fact that solution parameters (e.g. oil-water emulsion viscosity) are often not well defined will always reduce the accuracy of CFD predictions.

However, provided that the limitations of the technique and the flow phenomena being modelled are well understood, CFD modelling can be very effectively used in a wide range of applications.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the National Measurement System Engineering and Flow Programme and the Technology Strategy Board, both of whom supported parts of the work described in this paper.

REFERENCES

1. COULL, C., Investigation of Installation Effects on Ultrasonic Flowmeters and Evaluation of CFD Prediction Methods, 20th North Sea Flow Measurement Workshop, 22-25 October 2002, St Andrews, Scotland.

2. BARTON, N., HODGKINSON, E. & READER-HARRIS, M., Estimation of the Measurement Error of Eccentrically Installed Orifice Plates, 23rd North Sea Flow Measurement Workshop, 24-27 October 2005, Tønsberg, Norway.

3. BARTON, N., ZANKER, K. & STOBIE, G., The Effects of Scale in Subsea Multiphase Flow Meters. 29th North Sea Flow Measurement Workshop, 25-28 October 2011, Tønsberg, Norway.

4. READER-HARRIS, M.J. & HODGES, D., The Effect of Contaminated Orifice Plates on the Discharge Coefficient, TUV NEL report 2008/266, September 2008.

5. BARTON, N., MACLEOD, M., ZANKER, K. & STOBIE, G., Erosion in Subsea Multiphase Flow Meters, The Americas Workshop, 25-28 April 2011, Houston, USA.

6. BROWN, G., AUGENSTEIN, D. & COUSINS, T., Thermal Gradient Effects on Ultrasonic Flowmeters in the Laminar Flow Regime, 28th North Sea Flow Measurement Workshop, 22-25 October 2002, St Andrews, Scotland.

7. BARTON, N., The Effect of Varying Reynolds Number on a Zanker Flow Conditioner, 20th North Sea Flow Measurement Workshop, 26-29 October 2002, St Andrews, Scotland.

8. ABOURI, D., PARRY, A., HAMDOUNI, A. & LONGATTE, É, A Stable Fluid Rigid Body Interaction Algorithm: Application to Industrial Problems, Journal of Pressure Vessel Technology, November 2006, 128, pp 516-524.

9. International Organization for Standardization. ISO 3171:1988, Petroleum liquids - Automatic pipeline sampling, 2nd Edition.

10. BAKER, R. C., Computational Method to Assess Concentration of Water in Crude Oil Downstream of a Mixing Section, Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 1988, 202, pp 117-127.

11. Technology Strategy Board website. Available at: http://www.innovateuk.org. Accessibility verified 13th September 2012.

12. BARTON, N., MILLER, G., PINGUET, B., XIE, C. & PARRY, A. An Investigation into Multiphase Flow Streams Containing a Viscous Oil Component, The Americas Workshop, 3-5 February 2009, Houston, USA.

13. CORLETT, A.E. & HALL, A.R.W., Viscosity of oil and water Mixtures, Multiphase 99, 1999, pp 595-603.

14. National Measurement System website. Available at http://www.tuvnel.com/tuvnel/engineering_flow_programme/. Accessibility verified 13th September 2012.

15. CADALEN, S. & FOURNIER, B., Modelling of Wet Gas Flows, Energy & Low Carbon Measurement Conference, London, 18-19 September 2012.

16. BARTON, N., Simulating Horizontal Wet Gas Flowmeters with CFD, Poster at 28th North Sea Flow Measurement Workshop, 26-28 October 2010, St Andrews, Scotland.