Seventh International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
9-11 December 2009

THE FEASIBILITY OF SMOOTHED PARTICLE HYDRODYNAMICS FOR MULTIPHASE OILFIELD SYSTEMS

Paul DICKENSON[1*]

[1] CFD Laboratory, Cambridge University Engineering Department, Trumpington Street, Cambridge, UK, CB2 1PZ
and
Schlumberger Cambridge Research, High Cross, Madingley Road, Cambridge, UK, CB3 0EL
*Corresponding author, E-mail address: pd268@cam.ac.uk

ABSTRACT

Multiphase flows are common in the oil and gas industry. Computational Fluid Dynamics (CFD) has an important role in understanding such flows with many techniques available. Smoothed Particle Hydrodynamics (SPH) has been chosen as a target for research and has been matched with rotating gas/liquid flows as a suitable test case.

A feasibility study has been conducted to assess the likely computational cost of simulating the selected test case using SPH. It shows usable results are achievable with a serial code but higher resolution results will require parallel computing. The study has also shown that the rotating flow will generate high centrifugal accelerations. These would cause unacceptable compressibility and stratification effects if a Weakly Compressible SPH code were to be used. The Incompressible SPH formulation therefore appears preferable.

NOMENCLATURE

a	centrifugal acceleration
A_f	liquid film cross sectional area
D	diameter
N	number of particles
Q_L	liquid flow rate
Q_T	total flow rate
t	liquid film thickness
v_{av}	average liquid axial velocity
v_i	inlet flow velocity
V_L	volume of liquid
x	interparticle spacing
α_g	gas holdup
β	inlet area ratio
μ	perturbation factor
λ	length ratio

INTRODUCTION

Multiphase flows are ubiquitous within the oil and gas industry. Devices such as pumps, valves, separators and flow meters often involve the flow of complex mixtures of oil, water, gas and non-Newtonian fluids. Many are high value applications and yet the multiphase flows often remain poorly understood. Their behaviour is difficult to measure in the field and experiments are vital but expensive. In this environment, Computational Fluid Dynamics (CFD) has an important role to play.

Unfortunately, Multiphase CFD (MPCFD) in the oilfield is far less well developed than Single Phase CFD (SPCFD). This situation is caused by several technical challenges in MPCFD:

- Large range of length scales
- Coupling between phases
- Complex small scale physics
- Less effective averaging techniques

Together these factors make the solution of almost any MPCFD problem significantly more complex than its single phase equivalent (Prosperetti and Tryggvason, 2007).

Much oilfield CFD is understandably driven by immediate industrial concerns. However, there is also a need for a longer term view on where the next major advance in MPCFD techniques may be found. The strategy used here is to select the most promising novel MPCFD method, choose a well matched oilfield problem and look at the feasibility of its solution.

OPTIONS FOR MULTIPHASE CFD

The field of multiphase computational fluid dynamics (MPCFD) remains immature. Consequently, there exist a variety of techniques, all having their own benefits and drawbacks. Previous work by the author (Dickenson, 2009) has provided an overview of the methods suited to fluid-fluid oilfield multiphase flows while Prosperetti and Tryggvason (2007) provide mathematical detail on many of the methods. Included here are brief details for a selection of the most relevant and commonly used fluid-fluid MPCFD techniques

Homogeneous Model

The homogeneous model uses single phase CFD techniques but with modified properties for the fluid. A gas/liquid flow, for example, might be modelled using a code intended to model single phase liquid but with the density suitably reduced. As might be expected, such an approach works well for cases where the second phase is well dispersed (such as foams and emulsions) and has the benefit of low computational cost. However, there is a total loss of detail and exchange of mass, momentum or energy between the phases cannot be modelled.

Two-Fluid Models

The family of models known as "two-fluid" use some form of averaging to simplify the governing equations such that the volume fraction of the second phase becomes a variable in the problem. This allows the preservation of some local detail for feasible computational cost. First proposed by Ishii (1975), they are now commonly used in the oil and gas industry, and are available in many commercial CFD packages.

The most significant drawback to the two-fluid family of models is the loss of topological detail caused by averaging. For example, it can be difficult to distinguish a single large gas bubble from many smaller dispersed bubbles. Many two-fluid models also rely on empirically defined closure relations causing problems when applying them across a diverse range of problems.

One Fluid Models – VOF and Level Set

The Volume of Fluid (VOF) and Level Set methods are the most common examples of one-fluid models. In these models a marker function is used to indicate the phase distribution and is advected with the flow. Therefore topological detail is preserved.

VOF uses a binary marker function: it is zero in one phase and one in the other. This ensures mass conservation but causes smearing at the interface unless complex advection schemes are used. Indeed, some smearing is essential if curvature, and hence surface tension, is to be calculated.

The Level Set method was developed to reduce interface smearing. A continuous marker function is used with the interface indicated by the surface at which the function has zero value. This gives a well defined interface even after advection. However, it also introduces mass loss in under resolved regions, typically those having fine detail and hence of most interest. For a full account of Level Set methods see Osher and Fedkiw (2003).

Lattice Boltzmann

The Lattice Boltzmann method solves the Boltzmann equations instead of the more usual Navier-Stokes equation. In theory, this gives improved accuracy by applying discretisation at a smaller scale. However, it is not clear that these benefits are realised in reality. For multiphase applications a colour function is still required to indicate the movement of the phases. This is much like the marker functions used in one fluid models.

Smoothed Particle Hydrodynamics

The Smoothed Particle Hydrodynamics (SPH) method does not use a mesh. Instead, the fluid properties are distributed over a cloud of points using kernel functions and the points move with the flow. Since the points have mass, they are commonly regarded as particles. Originally developed in the astrophysics community (Monaghan, 1988), SPH is being increasingly applied to engineering applications.

SPH has the advantage of being inherently multiphase (particles can be defined to have different interaction properties) and is capable of automatically handling free surfaces. Topology changes are also automatic making the method especially suited to so-called "energetic flows". Mass conservation is guaranteed since mass is attached to the particles and there is usually efficient mapping of computational resources onto physical needs. The latter refers to the distribution of particles only in the liquid phase of a free surface problem.

However, SPH is undeniably computationally expensive. It also remains an immature method with little consensus on which SPH formulation is optimal. Codes must therefore be tailored to each application.

WHY SPH?

Smoothed Particle Hydrodynamics has been chosen for this work primarily because it shows the potential to be an excellent match to many oilfield problems. Often such problems involve energetic multiphase flows with frequent topology changes. SPH has the fundamental properties to accurately model these.

There are also non-technical reasons for the selection of SPH. It is a novel method yet to reach full maturity meaning there are many research opportunities. The indications are that the current implementations of the method are yet to reach the full potential of the underlying theory. This also means that SPH is a good candidate for the next "major advance" referred to in the Introduction to this paper. Figure 1 shows the author's view of how MPCFD might develop in the coming years. While SPH currently lags behind the Level Set method, the scope for its development looks set to make it a front runner in the future.

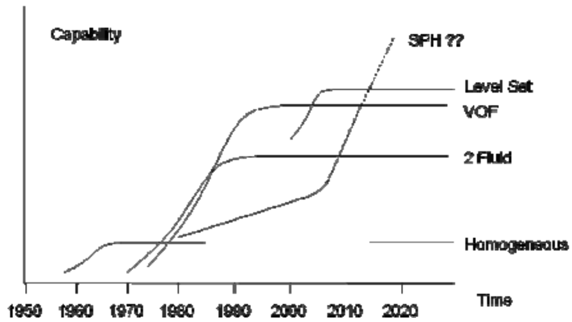

Figure 1: An illustrative schematic of MPCFD development with speculation for the future. Note that "capability" has been nominally normalised by available computational power.

ROTATING FLOWS

Rotating gas-liquid flows have been chosen as the problem to which SPH will be applied in this work. They have been chosen for industrial, technical and practical reasons as described in this section.

Industrial Considerations

Rotating multiphase flows are present within many oilfield systems. They occur in separators, pumps and other flow conditioning devices. Such devices are widely used, financially valuable and of current industrial interest.

Technical Considerations

The known properties of gas-liquid rotating flows make them a good fit for the capabilities of SPH. In particular, a simple scaling analysis of momentum transfer at the interface indicates that under most conditions the behaviour of the gas in such systems has only a limited effect on the liquid (Hewitt and Hall-Taylor, 1970). Hence it is possible to neglect the gas and exploit the free surface capabilities of SPH.

While being a good fit for SPH, this application will also provide technical challenges and encourage fundamental method development. As described later in the paper, developments will be required to enforce incompressibility despite centrifugal acceleration experienced by the liquid.

Practical Considerations

In order for the work to be freely published, the application must be capable of abstraction to a non-confidential test case. For rotating gas-liquid flows, this is possible while still maintaining technical relevance to real applications. For further work there are examples of separators whose geometries and properties are publicly available.

Abstracted Test Case

Modelling a real rotating system such as a gas/liquid separator would be a complex starting point and likely to make detailed experimental comparison both challenging and expensive. Instead, an abstracted test case has been developed as shown in Figure 2. This has a rectangular tangential inlet to generate swirling flow which then travels along a pipe section before exiting the system.

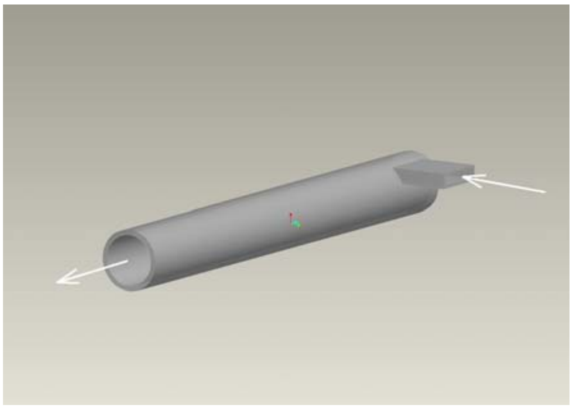

Figure 2: Simplified CAD model of rotating flow test case. The gas/liquid mixture enters through the tangential inlet (top right) and leaves through the pipe section (bottom left).

METHOD FOR FEASIBILITY STUDY

It is important to determine whether it is feasible to model this test case using SPH. This section describes a short study to determine the feasibility with respect to two important factors: computational expense and compressibility.

Like all Lagrangian methods, SPH is known for its high computational expense. Some large problems are not feasible, at least not at a satisfactory spatial resolution. It is therefore important to check whether this rotating flow test case can be modelled using reasonable computational resources.

Most existing SPH codes use a weakly compressible formulation (WCSPH) to model liquids. This treats all fluids as gases with modified equations of state. Testing of the open-source WCSPH code SPHYSICS (http://wiki.manchester.ac.uk/sphysics/index.php/Main_Page), showed that unphysical compressibility effects can be seen in the liquid. In particular, there is stratification of the fluid under the effects of gravity. Hence there were concerns that significant compression and stratification might be caused by the high accelerations found in rotating systems.

The approach used has been to determine the typical values of parameters for this problem, estimate a likely configuration for fluid within the device, and draw conclusions regarding the number of particles required and the likely accelerations.

Typical Parameters

Some likely parameters for this test case are shown in Table 1. They are based on typical commercial devices and available experimental facilities.

Parameter	Value	Units
Liquid Flow Rate	0.003	m^3/s
Gas Hold Up	0.1 - 0.95	
Internal Pipe Diameter	0.1	m
Pipe length	10	Diameters
Area ratio (inlet:pipe)	0.1 – 0.25	

Table 1: Typical parameters for gas/liquid rotating flow test case. Gas hold up is the cross sectional area of the gas in the pipe divided by the total cross sectional area of the pipe. The area ratio refers to the ratio of inlet area to pipe area.

No-Slip Estimate

To estimate the number of particles and the acceleration it is first necessary to find the gas/liquid interface position as shown in Figure 3. Unfortunately there is no unique solution for the interface position. This is caused by the same property which allows us to model the system using free surface techniques; the weak influence of gas on the behaviour of the liquid. The gas and liquid do not necessarily move together but can "slip" with a "slip velocity". However, it is possible to generate a range of solutions by perturbing the no-slip solution. These show plausible agreement with existing experimental results.

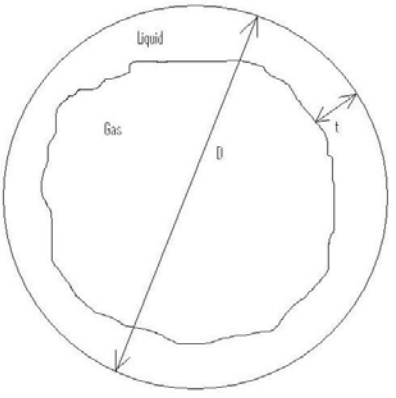

Figure 3: Cross section of rotating flow test case showing gas/liquid interface. This section is taken a few diameters downstream of the tangential inlet.

The ratio of cross sectional areas is given by

$$\frac{LiquidArea}{TotalArea} = 1 - \alpha_g \qquad (1)$$

where α_g is gas holdup, the ratio of gas cross sectional area to total cross sectional area. The area of the liquid film, A_f, is also given by

$$A_f = \frac{\pi}{4}\left(4Dt - 4t^2\right) \qquad (2)$$

where D is the pipe diameter and t is the liquid film thickness. Equating these gives a quadratic in t

$$4t^2 - 4Dt + D^2(1-\alpha_g) = 0 \qquad (3)$$

whose solution is

$$\frac{t}{D} = \frac{1-\sqrt{\alpha_g}}{2}\mu \qquad (4)$$

including a perturbation factor, μ. This is set to μ=1, 0.9, 1.1 to move the interface by 10% in each direction.

Now the mean axial velocity for the liquid, V_{av}, can be calculated from the liquid film area and liquid flow rate, Q_L

$$V_{av} = \frac{Q_L}{\pi(Dt - t^2)} \qquad (5)$$

The residence time, T, for liquid in the system is then given by

$$T = \frac{\lambda D\pi\left(Dt - t^2\right)}{Q_L} \qquad (6)$$

where λ is the ratio of length to diameter.

The inlet velocity, v_i, can be estimated from the total volumetric flow rate, Q_T, and inlet area, A_i

$$v_i = \frac{Q_T}{A_i} \qquad (7)$$

where

$$Q_T = \frac{Q_L}{1-G} \qquad (8)$$

$$A_i = \beta\frac{\pi}{4}D^2 \qquad (9)$$

In Equation 8, G is the Gas Volume Fraction. This is the ratio of input volumetric gas flow rate to total input volumetric flow rate. In this case we are assuming the solution has little or no gas/liquid slip and so G can be approximated by gas holdup, α_g. In equation 9, β is the ratio of inlet area to pipe section area.

Combining equations 9 and 10 gives

$$v_i = \frac{4Q_L}{(1-\alpha_g)\beta D^2\pi} \qquad (10)$$

Assuming the fluid mixture leaves the tangential inlet at a distance D/2 from the axis of the pipe section, the centrifugal acceleration will be

$$a = \frac{32Q_L^{\ 2}}{\left(1-\alpha_g\right)^2\beta^2D^5\pi^2} \qquad (11)$$

The volume of liquid in the pipe section, V_L, is estimated from the film area and pipe length.

$$V_L = \pi D\lambda\left(Dt - t^2\right) \qquad (12)$$

For an inter-particle separation, x, the number of particles required can be found from equation 12.

$$N = \frac{V}{x^3} = \frac{\pi D\lambda\left(Dt - t^2\right)}{x^3} \qquad (13)$$

Elevated Gravity Tests

Without actually modelling a rotating system with SPH, it is possible to simulate the effects of centrifugal acceleration by imposing a body force on an ordinary static problem. This can be achieved in SPHYSICS by changing the value of gravitational acceleration, g, in the source code to reflect the results from this study.

The static test case chosen is a standard 2D bucket test case. This has a 1m x 1m square of water contained within boundaries described using fixed SPH particles. All other parameters have been left at those recommended in the SPHYSICS documentation for the simulation of similar problems.

RESULTS AND DISCUSSION

Fixing β, the inlet area ratio, at a typical value of 0.2, we can plot the variation of liquid film thickness with gas holdup for the no slip and perturbed solutions as shown in Figure 4. It can be seen that film thickness decreases with gas holdup and ranges from about 5mm to 35mm. This is plausible for a 100mm diameter system and consistent with expectations from experiments.

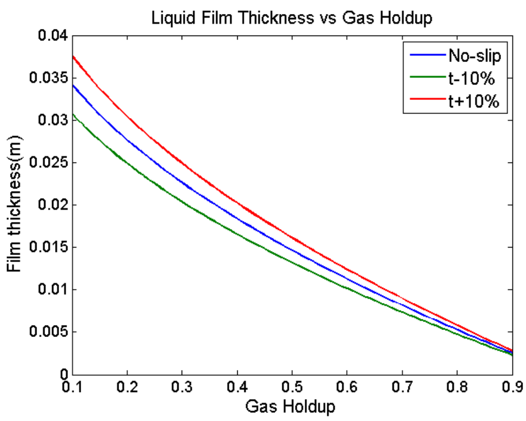

Figure 4: Plot of liquid film thickness versus gas holdup. The three solutions are for the no gas/liquid slip and perturbed solutions.

The residence time for liquid in the system is plotted versus gas holdup in Figure 5. As expected, it decreases with increasing gas holdup due to the higher axial velocity of the liquid. The values are consistent with the short times (a few seconds) observed for similar systems to reach equilibrium in experiments.

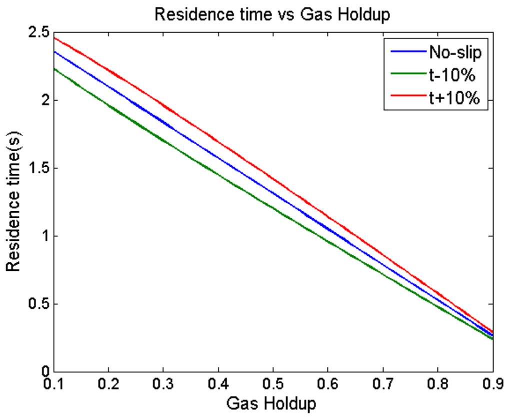

Figure 5: Residence time for liquid plotted versus gas holdup for no slip and perturbed cases.

From film thickness and liquid residence time it is possible to calculate the volume of liquid in the system and hence the number of SPH particles required for a given spatial resolution. This is plotted in Figure 6 for particle a separation of 5mm and in Figure 7 for 1mm. The values are large but plausible; the system when completely filled with liquid at these particle spacings would require 6.3×10^4 and 7.9×10^6 particles respectively.

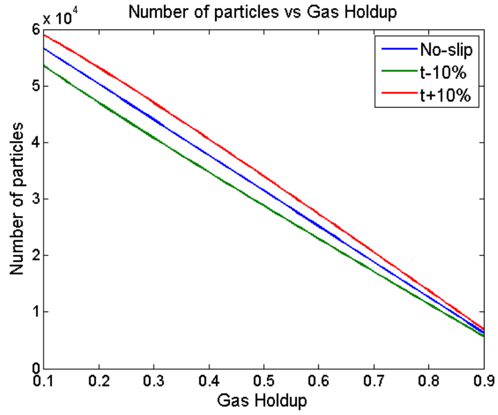

Figure 6: Number of SPH particles for a spatial resolution of 5mm. Note the (much smaller) volume of liquid in the tangential inlet is not included here.

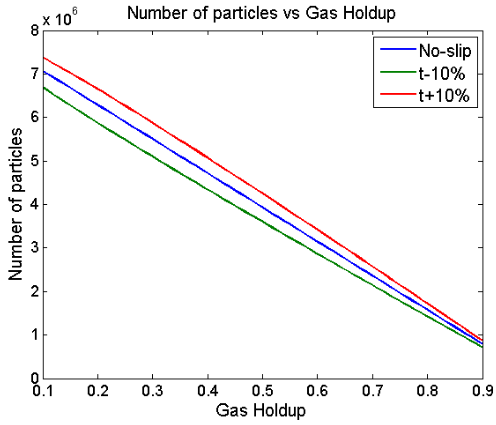

Figure 7: Number of SPH particles for a spatial resolution of 1mm. Note the (much smaller) volume of liquid in the tangential inlet is not included here.

The centrifugal acceleration experienced by the fluid is plotted against gas holdup in Figure 8.

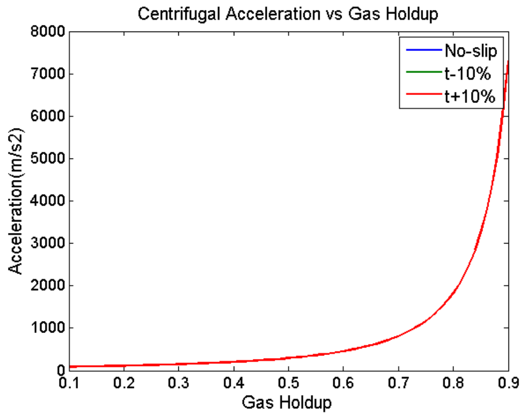

Figure 8: Centrifugal acceleration versus gas holdup for no slip and perturbed cases. Note that the lack of dependence on film thickness makes the three solutions identical.

Computational Cost

It is difficult to make accurate predictions of run-times without knowing every detail of the simulation. However, it is possible to make a rough estimate by comparing the results of this study with the results of an existing test case. The latter have been provided by a 3D dam break simulated with the WCSPH code SPHYSICS.

To extrapolate from the dam case, first assume we are to simulate only a few seconds as suggested by the short residence time. Next assume the time steps will be comparable to those in the dam case and that the run time is proportional to NLn(N) where N is the number of particles. The dam case has 30000 particles and a run time of 7000s on a serial workstation. Scaling these values to the rotating flow test case with a 5mm particle separation suggests run times between approximately 1 and 3 hours. This is entirely feasible and would yield results of sufficient resolution for code development.

For the higher resolution given by 1mm particle separation, scaling suggests run-times ranging from 3 days to 29 days on a serial workstation. The upper end of this range is not feasible and so some form of parallelism would be required, either cluster computing or Graphics Processing Units. For example, a cluster of 16 processors with 90% parallelisation efficiency would reduce the 29 day run-time to just 2 days.

Compressibility

This study estimates the centrifugal acceleration to be between 50m/s^2 and 500m/s^2. As described above, it is possible to model the effect of centrifugal acceleration using a static test case. Hence the gravitational acceleration in the 2D bucket test case was set to 100m/s^2 and the density of the liquid observed after 5s. The resulting contours for density are shown in Figure 9. The original 1mx1m square of water has reduced in height indicating a failure of volume conservation. The colouration indicates there is significant density stratification between the top and bottom of the bucket.

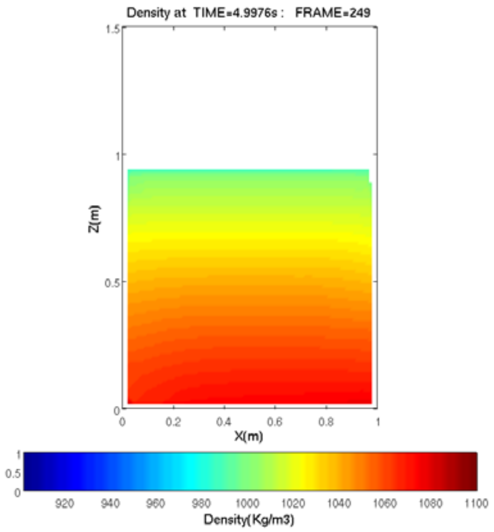

Figure 9: Density contours after 5s for bucket test case with g=100m/s^2.

The test was repeated for values of g set to 9.81m/s^2 and 50m/s^2. The results for all three cases together with the analytical solution are shown in **Figure 10**. It can be seen that with g=100m/s^2 there is an 80kg/m^3 density range with a 10% density error at the base. Such errors are unacceptable and will worsen with increasing acceleration. It should be noted that these values of acceleration are at the lower end of the range predicted by this study; higher values cause such large compressions in the bucket test case that the code fails.

Stratification could be reduced by decreasing the compressibility of the fluid as set by the equation of state. However, this would also increase the artificial speed of sound towards the true sound speed, generating shorter time steps. If the resulting prohibitive increase in computational cost is to be avoided, a shift from WCSPH to truly incompressible SPH (ISPH) will be required.

CONCLUSIONS

There is a strong industrial case for the development of Multiphase Computational Fluid Dynamics (MPCFD) for applications in the oil and gas industry. These problems are also of significant technical and academic interest.

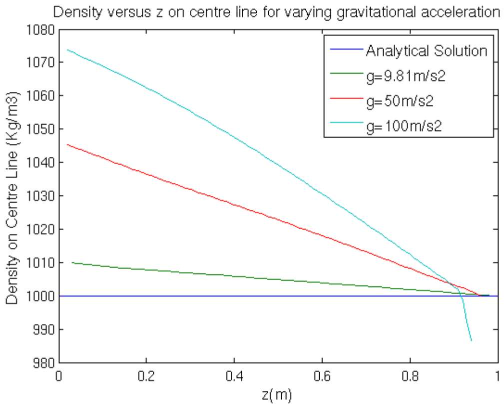

Figure 10: Density variation on centre line of bucket test case with varying g.

From the many MPCFD techniques available, Smoothed Particle Hydrodynamics (SPH) has been chosen as a good match to the requirements of many oilfield problems. It automatically handles topology changes and has the inherent ability to model free surfaces. SPH also has scope for fundamental development, perhaps allowing it to surpass current methods in the future.

Rotating gas/liquid systems have been chosen as a test case due to their industrial relevance and good match to SPH. The feasibility of the rotating flow test case has been evaluated using a no-slip solution for the position of the gas-liquid interface. Results indicate that the computational cost with a serial code is acceptable for moderate spatial resolution, but a parallel code will be required for higher resolutions. The accelerations seen in the feasibility study suggest the compressibility present in the WCSPH formulation may cause large density errors. Use of Incompressible SPH is therefore preferable.

Overall, the modelling of gas/liquid rotating flows using Smoothed Particle Hydrodynamics appears relevant, interesting, feasible and challenging.

ACKNOWLEDGEMENTS

This work is jointly funded by an EPSRC Doctoral Training Award and Schlumberger Cambridge Research. Thanks to Prof W N Dawes, PhD supervisor, and Chris Lenn, Gary Oddie and Richard Mills at Schlumberger for many useful discussions and suggestions.

REFERENCES

DICKENSON, P., (2009), “A Survey of Multiphase CFD Techniques”, *Grand Review in the State-of-the-Art in the Numerical Simulation of Fluid Flow 2*, IMechE.

HEWITT, G. G. and HALL-TAYLOR, N. S., (1970), “Annular Two-Phase Flow”, Pergamon.

ISHII, M., (1975), “Thermo-Fluid Dynamic Theory of Two Phase Flow”, Byrolles, Paris.

MONAGHAN, J. J., (1988), “An Introduction to SPH”, *Computer Physics Communications*, **48**: 89-96.

OSHER, S. and FEDKIW, R., (2003), “Level Set Methods and Dynamic Implicit Surfaces”, Springer.

PROSPERETTI, A. and TRYGGVASON, G., (2007), “Computational Methods for Multiphase Flow”, Cambridge University Press.

