13th Australasian Fluid Mechanics Conference
Monash University, Melbourne, Australia
13-18 December 1998

SMOOTH PARTICLE HYDRODYNAMICS MODELLING OF VERTICAL JET IMPINGEMENT

P.J. Reichl, P. Morris, S.A.T. Stoneman [1], M.C. Thompson and K. Hourigan

Fluid-dynamics Laboratory for Aeronautical and Industrial Research (FLAIR)
Department of Mechanical Engineering
Monash University, Clayton, Victoria 3168, AUSTRALIA

[1] Department of Civil Engineering
University of Wales
Swansea, SA2 8PP, UNITED KINGDOM.

ABSTRACT

Despite recent theoretical and computational advances, the calculation of free surface behaviour is still a particularly difficult problem to solve. Complications arise as the positioning of the free surface is not known in advance, but must also be found in the solution procedure. Developing unified schemes for the simulation of such flows by Lagrangian particle methods is the aim of the current work. The scheme is tested and compared with experiments on the case of jet impingement on a vertical wall in two and three dimensions.

INTRODUCTION

Free-surface flows occur over a wide range of scales and problems in the universe, from the accretion and collapse of stars to the fall of raindrops. In the industrial sphere, there is an enormous array of important problems involving free-surface behaviour, including sloshing in tanks, splashing, surface coating, continuous casting, and rock cutting. The latter class of problems often involve geometrically complicated boundaries as well.

Published literature for techniques to handle the computational modelling of free-surfaces in complex domains is rare. Both features create specific difficulties from a computational point of view. There are various finite-difference/volume approaches presented in the literature especially treating the free-surface nature of problems (Harlow & Welch, 1966 and Nichols & Hirt, 1971). These methods can be efficient for regular domains in two-dimensions, however, it is difficult to see how they could be readily extended to complex geometries in three-dimensions, as many of the algorithms rely on the uniformity of the grid. The grid-based methods are also naturally diffusive. There are algorithms based on flux-corrected transport to overcome this effect (see Oran & Boris, 1987), but again these generally rely on the grid being regular.

Apart from those methods mentioned above with their inherent problems, there are also Lagrangian (grid-free) techniques. They model the flow as a collection of particles which represent parcels of fluid. An attractive alternative for free-surface problems is one such technique called Smoothed Particle Hydrodynamics (SPH). In addition to being grid free, SPH is robust and easy to code. However, it has not been clear previously that it could be extended to the simulation of pseudo-incompressible flows nor that boundaries could be incorporated easily. This technique has been extensively applied to theoretical three-dimensional compressible astrophysical problems for the last twenty years, for example, Lucy (1977), Gingold & Monaghan (1977), Monaghan (1988 & 1992). In the previous few years the method has been tested on standard benchmark incompressible flow problems, such as Couette flow, an elliptical drop undergoing acceleration, the bursting dam, Monaghan (1994), and free jets falling under gravity Reichl *et. al.* (1998). Typically, these tests indicate that the achievable accuracy is typically of the order of a couple of percent given moderate computational resources. Compared with the grid-based techniques, this method can model free surfaces naturally without the need to introduce special tracking schemes. It can also, in principle, cope with complex geometries. Boundaries can be added by using either boundary particles or a repulsive force.

On the other hand, in comparison with grid-based methods, SPH has a lower theoretical convergence rate (Monaghan, 1992). There is also *idiosyncratic* behaviour such as the tendency of particles gathered along lines (for example, in directional flows like jets) to remain aligned even when this is unphysical. This seems to be associated with the development of non-isotropic (and non-physical) stresses due to particle positioning. There are also other *structure* effects leading to irregularities at solid boundaries.

In the following, the SPH method will be applied to the test case of a jet impinging on a vertical wall in both two and three dimensions. This furthers the work on the numerical simulation of jets, and adds to the previously studied horizontal wall cases (see Reichl *et al.* 1998).

FORMULATION

The governing equations in SPH determine the values possessed by each of the particles, and a formal derivation using the theory of integral interpolants can be found in Monaghan (1988).

Momentum

The momentum equation for particle i can be written

$$\frac{d\mathbf{v}_i}{dt} = -\sum_j m_j \left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2} + \Pi_{ij} \right) \nabla_i W_{ij} + \mathbf{F}_i, \tag{1}$$

where the summation on i is over all particles other than particle i (although in practice only near neighbours contribute), P is the pressure and ρ is the density, Π_{ij} produces a shear and bulk viscosity, $\mathbf{F}_i$ is a body force (gravity in this case), W_{ij} is the interpolating kernel, and ∇_i denotes the gradient of the kernel taken with respect to the coordinates of particle i.

The viscous term Π_{ij} is that given in Monaghan (1992) which has the general form

$$\Pi_{ij} = \begin{cases} \dfrac{-\alpha \bar{c}_{ij} \mu_{ij} + \beta \mu_{ij}^2}{\bar{\rho}_{ij}} & ; \mathbf{v}_{ij} \cdot \mathbf{r}_{ij} < 0 \\ 0 & ; \mathbf{v}_{ij} \cdot \mathbf{r}_{ij} > 0 \end{cases}$$

where

$$\mu_{ij} = \frac{h\, \mathbf{v}_{ij} \cdot \mathbf{r}_{ij}}{\mathbf{r}_{ij}^2 + \eta^2}.$$

Continuity

To find the smoothed density, the continuity equation is used. Writing the continuity equation in the form

$$\frac{d\rho}{dt} = -\nabla \cdot (\rho \mathbf{v}) + \mathbf{v} \cdot \nabla \rho, \tag{2}$$

and applying the theory of integral interpolants, the rate of change of the density of particle i becomes

$$\frac{d\rho_i}{dt} = \sum_j m_j (\mathbf{v}_i - \mathbf{v}_j) \cdot \nabla_i W_{ij}. \tag{3}$$

Kernel

The cubic spline based kernel (Monaghan & Lattanzio, 1985) is used in these calculations. This kernel has compact support and vanishes for separations greater than $2h$. The errors introduced by SPH interpolants, that is by using particles, is $O(h^2)$.

Equation of State

The equation of state given by Batchelor (1983) for water, and which describes sound waves accurately is used here and has the form.

$$P = B \left(\left(\frac{\rho}{\rho_0} \right)^{\gamma} - 1 \right) \tag{4}$$

In the simulations undertaken here γ was taken to be 7. The choice of B determines the speed of sound. In practice it is chosen from dimensional arguments so that the Mach number is small for the entire simulation period.

Treatment of Boundaries

For the simulations the boundary was modelled using a Lennard-Jones force, which in the absence of any additional viscous model for boundary behaviour, acts as a free slip boundary. The form for the Lennard-Jones force is as given in Monaghan (1994). For a boundary and fluid particle separated by a distance r the force $f(r)$ has the form

$$f(r) = \frac{D}{r} \left(\left(\frac{r_0}{r} \right)^{p_1} - \left(\frac{r_0}{r} \right)^{p_2} \right), \tag{5}$$

but is set to zero if $r > r_0$ so that the force is purely repulsive. The indices p_1 and p_2 are taken as 5 and 3 respectively, although other values have been tried with similar results.

The length scale r_0 is taken to be the initial spacing between the particles. The coefficient D is chosen by considering the physical configuration.

Implementation Issues

For the calculations described, the particles were set up initially on a Cartesian lattice. The mass of particle i is given by $m_i = \rho \Delta A$ where ΔA is the area per particle.

Time evolution was carried out using an adaptive time stepping Runge-Kutta-Fehlberg scheme for both the 2D and 3D results. The time-step is controlled by both the Courant condition and the tolerance set in the Runge-Kuttu-Fehlberg scheme.

RESULTS AND DISCUSSION

In the current investigation two cases are considered, namely, the impingement of a two dimensional jet, and a three dimensional jet, onto a vertical wall.

For both cases, the particles are fed in at the inlet with a uniform velocity profile when existing particles have moved through the initial inter-particle separation. To limit particle numbers, they are removed from the domain when they have moved sufficiently far from the nozzle exit. The number of particles required depends on the amount of splashing and surface wave motion that occurs after the boundary collision, and hence depends on both the impingement angle and Froude number ($U/\sqrt{L\,g}$ where g is gravity). Typically the 2D results had approximately 5000

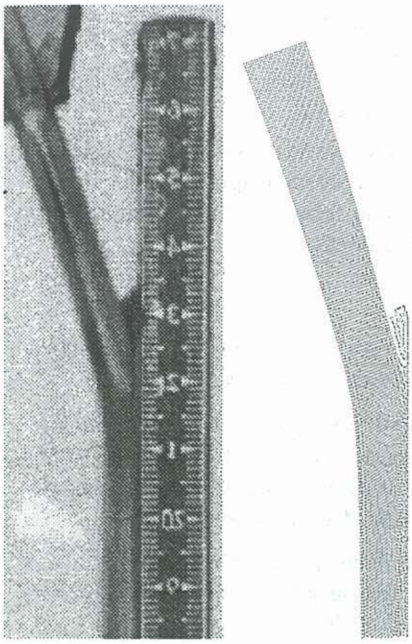

Figure 1: A comparison of the experimental and numerical plane jet, for an impingement angle of 20 degrees to the vertical.

particles in the domain of evolution, while the 3D results had approximately 100000 particles.

Some 2D results are shown in figures 1 and 2 for angles to the vertical of 20 and 30 degrees respectively. For the flow at 30 degrees to the vertical, the fluid rising up the wall forms a 'fountain' which reaches a maximum height and then falls back towards the jet. Figure 2 shows the case in which the fluid has just moved back to the nozzle. In both the experiments and the computations, this return flow interferes with that issuing from the nozzle, and disrupts the clean flow of the jet for angles less than approximately 40 degrees to the vertical. The behaviour is somewhat unsteady and the height reached varies with time. Some of the numerical and experimental results are compared, as shown in figures 1 and 2, with these snapshots being representative of the general behaviour.

Test Cases for the Three-Dimensional Code

At this time, the code is still undergoing validation; however, preliminary results are available for the three-dimensional impinging jet problem to verify that it is producing consistent predictions. Future simulations will be validated against experimental results obtained by the circular jet impingement rig at the University of Wales as they become available.

A typical end-on view of a circular jet impinging against a vertical wall under gravity is given in figure 3.

The jet is circular and the inlet impingement angle is 90^o to the vertical. The perpendicular distance of the inlet from the wall is three nozzle diameters, while the Reynolds number and Froude number are 100 and 2.0 respectively.

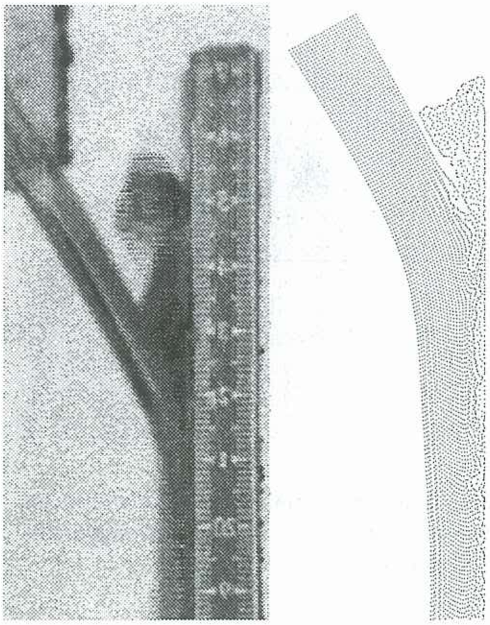

Figure 2: A comparison of the experimental and numerical plane jet, for an impingement angle of 30 degrees to the vertical.

A comparison of the spreading width at the point of impingement and Froude number was then considered for a set of different Froude numbers. The results of the comparison are shown in Figure 4. It should be noted that while all the numerical results are for a Reynolds number of 100, the experimental results have differing values of the Reynolds number (as the same fluid was used for each of the experimental results). In general, the agreement between the numerical and experimental results is acceptable, especially when one considers that the width at the point of impingement tends to increase slightly with Reynolds number for the numerical results.

The end-on and side-on views (figures 3 and 5) show the shape of the spreading jet. The sheet of fluid attached to the wall is in most cases quit thin, with some splashing being observed. At larger distances from the impingement point, fluid attached to the wall shows a tendency to break up into 'streams'. Towards the outer boundary of the fluid noticeable void regions can be observed as the layer of fluid becomes thinner. Details of this downward flow may depend on surface tension and wetting which are not implemented in this model.

CONCLUSION

Numerical simulations have been performed for both two and three-dimensional jets impinging against a vertical wall. This again adds a further test of the capabilities of Smooth Particle Hydrodynamics to han-

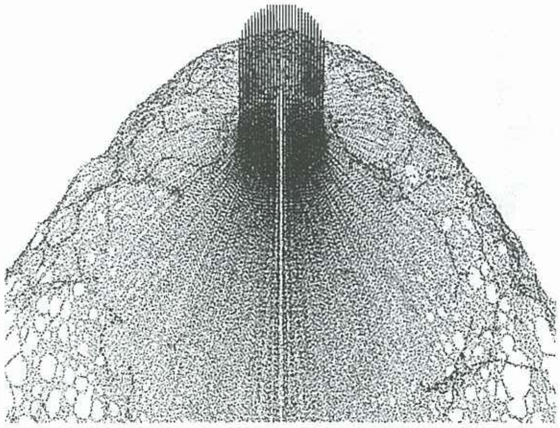

Figure 3: Numerical visualisation of a round jet impacting onto a vertical wall for an impingement angle of 90 degrees.

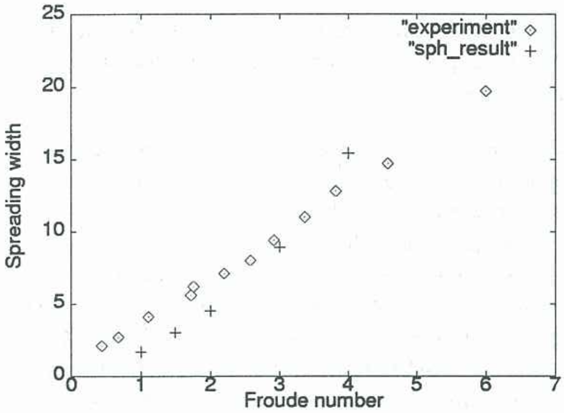

Figure 4: A comparison of the width of spread at the point of impingement between the experimental and numerical results for a series of different Froude numbers.

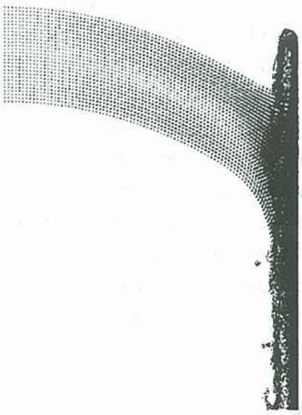

Figure 5: Side-on view of a jet impinging onto a vertical wall for a Froude number of 2.0.

dle nearly incompressible flows that contain complicated free surfaces.

ACKNOWLEDGEMENTS

Paul Reichl would like to acknowledge the Australian Postgraduate Award which helped to fund this research. The funding from the Australian Research Council is also thankfully acknowledged.

REFERENCES

Batchelor, G.K., "An Introduction to Fluid Mechanics", Cambridge Press, 615pp, 1983.

Gingold, R.A. and Monaghan, J.J., "Smoothed Particle Hydrodynamics: theory and application to non spherical stars", *Monthly Notices of the Royal Astronomical Society*, **181**, 375-389, 1977.

Harlow, F. and Welch, J.E., "Numerical study of large-amplitude free-surface motions", *Physics of Fluids*, **9**, 842-851, 1966.

Lucy, L., "A numerical approach to the testing of the fission hypothesis", *Astronomical Journal*, **82**, 1013-1024, 1977.

Monaghan, J.J., "An introduction to SPH", *Computational Physics Communications*, **48**, 89-96, 1988.

Monaghan, J.J., "Smoothed Particle Hydrodynamics", *Annual Reviews of Astronomy and Astrophysics*, **30**, 543-574, 1992.

Monaghan, J.J., "Simulating free surface flows with SPH", *Journal of Computational Physics*, **110**, 399-406, 1994.

Monaghan, J.J. and Lattanzio, J.C., "A refined particle method for astrophysical problems", *Astronomy and Astrophysics*, **149**, 135-143, 1985.

Nichols, B.D. and Hirt, C.W., "Improved free surface boundary conditions for numerical incompressible-flow calculations", *Journal of Computational Physics*, **8**, 434-448, 1971.

Oran, E.S. and Boris, J.P., "Numerical simulation of reactive flow", Elsevier, New York, 601pp, 1987.

Reichl, P.J., Morris, P., Hourigan, K., Thompson, M.C., and Stoneman, S.A.T., "Free surface flows using smooth particle hydrodynamics", 1997 ASME fluids engineering division summer meeting. Vancouver, Canada. FEDSM97-3403, 1997.

Reichl, P.J., Morris, P., Hourigan, K., Thompson, M.C., and Stoneman, S.A.T., "Smooth Particle Hydrodynamics Simulation of Surface Coating", *Journal of Applied Mathematical Modelling*, **22** No. 12, the December 1998