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

Comparison of SPH Simulation of High Speed Die Filling With Experiment

Joseph HA, Vikas AHUJA[†] and Paul CLEARY

CRC for Alloy and Solidification Technology (CAST)
CSIRO Mathematical & Information Sciences
Clayton, Victoria, 3169, AUSTRALIA
[†]CSIRO Manufacturing Science & Technology
Preston, Victoria, 3072, AUSTRALIA

ABSTRACT

This paper evaluates the suitability of the fluid flow simulation method called smoothed particle hydrodynamics for modelling high pressure die casting. In particular, comparisons of SPH simulations and experimental observations of water filling a mould at constant temperature are presented. The effects of turbulent viscosity and turbulent length scales on the flow are also discussed.

INTRODUCTION

High pressure die casting (HPDC) is an important industrial process by which almost finished castings with excellent surface finishes can be produced in high volumes. It plays a significant role in the making of automotive components such as gear box casings. The flow pattern of molten metal in the die cavity is one of the most important factors influencing the quality of the casting and is currently not well understood, sometimes resulting in a significant numbers of rejected castings and thus waste. Numerical simulation offers a powerful and cost effective way to increase our understanding of this difficult molten metal flow. For such simulations to be reliable, some measure of their accuracy and effectiveness must first be made. The first stage of this work, aimed at developing a computational model for HPDC, is concerned with the flow of a single fluid at constant temperature. Other effects such as heat transfer, multi-phase flow, solidification and temperature dependence of material properties will be addressed in future stages.

Smoothed particle hydrodynamics (SPH) is a Lagrangian method (Monaghan, 1992) and is therefore suitable for modelling momentum dominated flows that involves droplet formation, splashing and free surfaces such as occur in HPDC. More specifically, a variant of the incompressible extension (XSPH), originally described in Monaghan (1994) was used to compute the results presented in this paper. SPH modelling of HPDC for two moulds and Reynolds numbers from 50 to 50,000 are described in an earlier paper (Cleary and Ha, 1998). High resolution simulations in that paper confirm their accuracy. SPH modelling of HPDC for a wider range of moulds is given in Cleary (1997) for Reynolds numbers up to 2,500. Here the numerical simulation of a water-analogy of high pressure die casting is compared favourably to experimental observations for the same set up configuration.

THE SPH METHOD

SPH is a Lagrangian method that uses an interpolation kernel of compact support to represent any field quantity in terms of its values at a set of disordered points (the particles). The fluid is discretised, and the properties of each of these elements is associated with its center which is then interpreted as a particle. A particle b has mass m_b, position $\boldsymbol{r}_b$, density ρ_b and velocity $\boldsymbol{v}_b$. Any field A is then approximated by:

$$A(\boldsymbol{r}) = \sum_b m_b \frac{A_b}{\rho_b} W(\boldsymbol{r} - \boldsymbol{r}_b, h) \qquad (1)$$

where W is an interpolating kernel, h is the interpolation length and the value of A at $\boldsymbol{r}_b$ is denoted by A_b. This allows smoothed approximations to the physical properties of the fluid to be calculated from the particle information. The smoothing formalism also provides a way to find gradients of fluid properties. In this way, the SPH representation of the hydrodynamic governing equations can be built. These equations of motion are given in the preceeding paper (Cleary and Ha, 1998). The simulation progresses by explicitly integrating this system of ordinary differential equations.

THE EXPERIMENTAL METHOD

Figure 1 shows the geometry of the die used in both the experiment and the numerical simulation. This is similar to the C shaped one predominantly used in the modelling by Cleary and Ha (1998). It is 50 mm long and 20.9 mm high and is connnected to the shot sleeve by a gate of width 2.9 mm (this is the length scale used in defining the Reynolds number (Re)). The width of the vertical sections is 4 mm and the width of the connecting horizontal section is 4.9 mm.

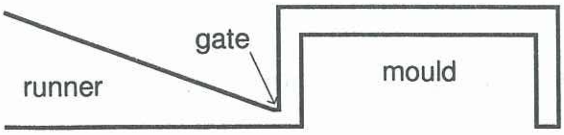

Figure 1: Cross section of the die.

The experimental apparatus is shown schematically in Figure 2. The flow re-circulates in the loop on the lower left, until the 3-way valve is electro-pnuematically switched to re-direct the flow into the mould cavity (top center). The desired gate velocity is set approximately by using the flow meter. The apparatus is designed so that the pressure drop in the re-circulation circuit and in the cavity filling line are similar. The timing unit allows synchronisation of valve switching and triggering the video camera. The video footage was captured using an Olympus Encore high speed video camera. The 240x210 pixel video images were recorded at 1000 frames per second using a shutter speed of 50 μs. The transparent acrylic mould was forward lit and the water seeded with white neutrally bouyant particles to show the flow. The captured footage was down loaded to a VCR and then digitised. The actual values of gate velocity were measured by tracing a particle or bubble in the digitised images.

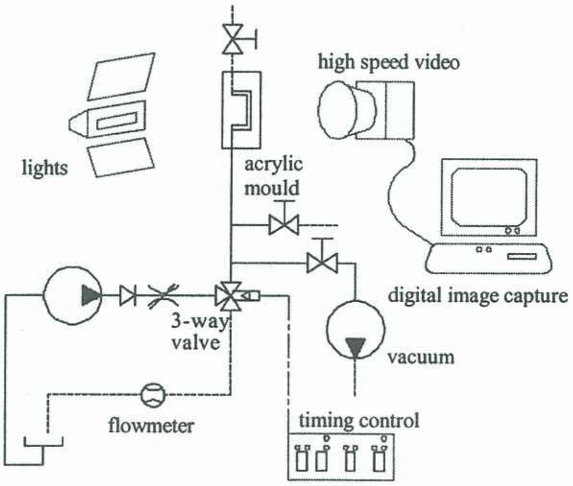

Figure 2: Experimental flow visualisation set up.

Figure 3 shows stills for six different times throughout the filling process. They are taken from the video recording of the experiment whose gate velocity was measured as 0.62 m/s.

NUMERICAL RESULTS

In these numerical simulations, 150 particles were used across the 20.9 mm height of the mould, giving rise to a total of 29,613 particles. These results were checked for convergence using 250 particles across. The water has density 1000 kg/m^3 and dynamic viscosity $\mu = 0.001$ kg/m/s.

Figure 4 shows six snap shots of the flow of water into the die predicted using the 2D SPH method. The times are the same as those shown in Figure 3. Comparison of these figures shows that the essential features of the flow are the same for both the experiment and SPH calculation. The SPH results capture the filling process in the left vertical section of the mould and the first right angle bend rather well. In particular, the shape of the jet, its trajectory along the right vertical wall, the shape and size of the void in the left part of the vertical section and the spreading of the jet to left and right after contact with the top of the mould in both experiment and SPH calculation are quite similar. Experimental results have not yet been obtained for flow beyond the left half of the die.

The comparisons between the SPH calculations and experiment in the horizontal section for 46 ms and 67 ms are slightly less favourable. The differences may be due to a range of processes not present or fully captured in the simulations. These include the effects of air pressure slowing the fluid and causing the jet to become more rounded, turbulence increasing the effective viscosity and increasing three dimensionality of the real flow producing a two dimensional projection of the fluid that covers a larger area of the mould than expected. The relative contributions are presently unknown. Overall, the comparison shows that the SPH simulations capture most of the flow features.

USING A TURBULENT VISCOSITY

The real Reynolds number at the gate is 1,798 (for gate velocity of 0.62 m/s and dynamic viscosity of 0.001 kg/m/s), so the flow either is in or near the turbulent regime. A higher value of viscosity than the kinematic one is more appropriate to capture the turbulent viscous effects with this essentially laminar calculation. Using the gate velocity and assuming the turbulent length scale is about 1% of the gate width, a turbulent viscosity of about 0.005 kg/m/s is obtained from the eddy viscosity model $\rho C_\mu k^2/\epsilon$, where k is turbulent kinetic energy and ϵ is its rate of dissipation.

Figures 5 and 6 show SPH results using the higher viscosities $\mu = 0.005$ and 0.01 kg/m/s. The flow becomes more viscous with increasing μ inhibiting droplet formation and fragmentation and leading to thicker more rounded features. The results obtained using $\mu = 0.005$ show closer resemblance with experiment than those obtained using the kinematic viscosity. In particular, note the close similarity between the experimental and SPH results at the head of the jet at 34 ms. For the highest viscosity shown, the jet features have become too rounded and the jet contacts the left wall of the vertical section, indicating that this viscosity is too high. The best match seems to occur for a viscosity five times that of the kinematic viscosity and corresponding to a turbulent

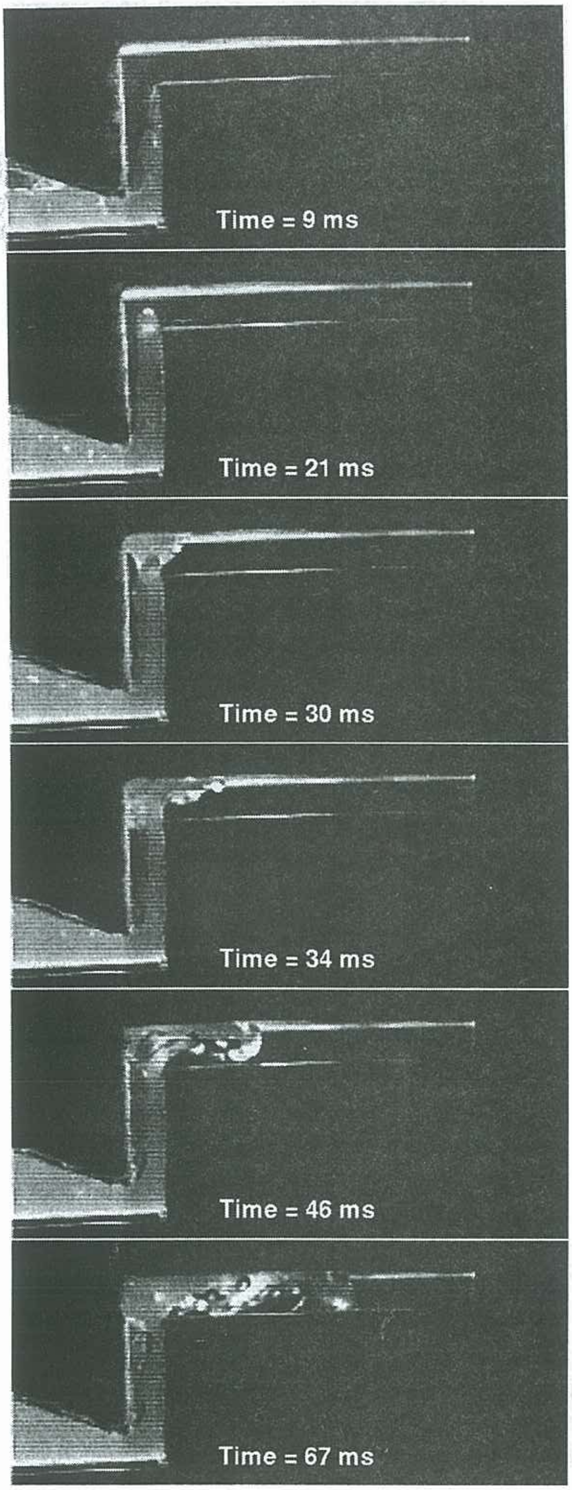

Figure 3: Experimental images

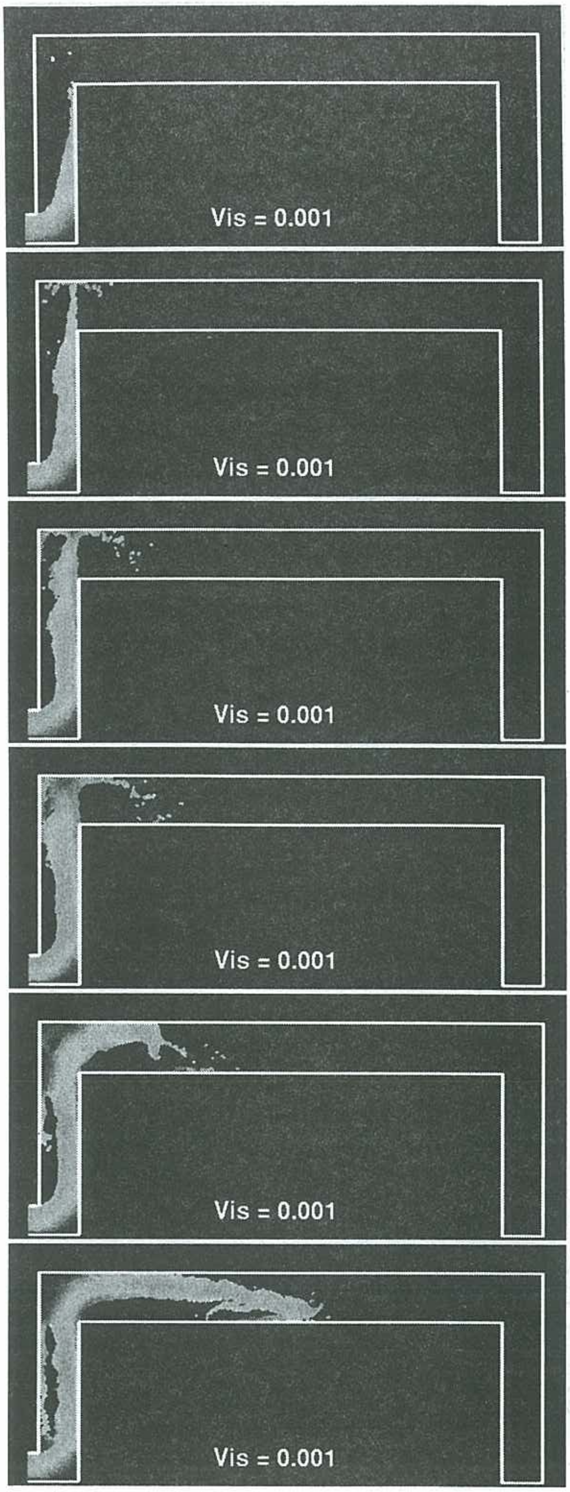

Figure 4: SPH result using $\mu = 0.001$ kg m^{-1} s^{-1}.

length scale of 0.5% of the mould width. The need to use a higher viscosity for matching a turbulent experiment was also demonstrated for the the flow of water from a breaking dam (Ha, 1997).

CONCLUSION

The results presented in this paper show that SPH is a powerful and effective method for simulating die filling. It is able to capture the essential features of the filling process. Minor differences between experiment and simulation are attributable to the turbulent, 3-dimensional and air pressure effects.

ACKNOWLEDGEMENTS

This project was funded by the Cooperative Research Centre for Alloy and Solidification Technology.

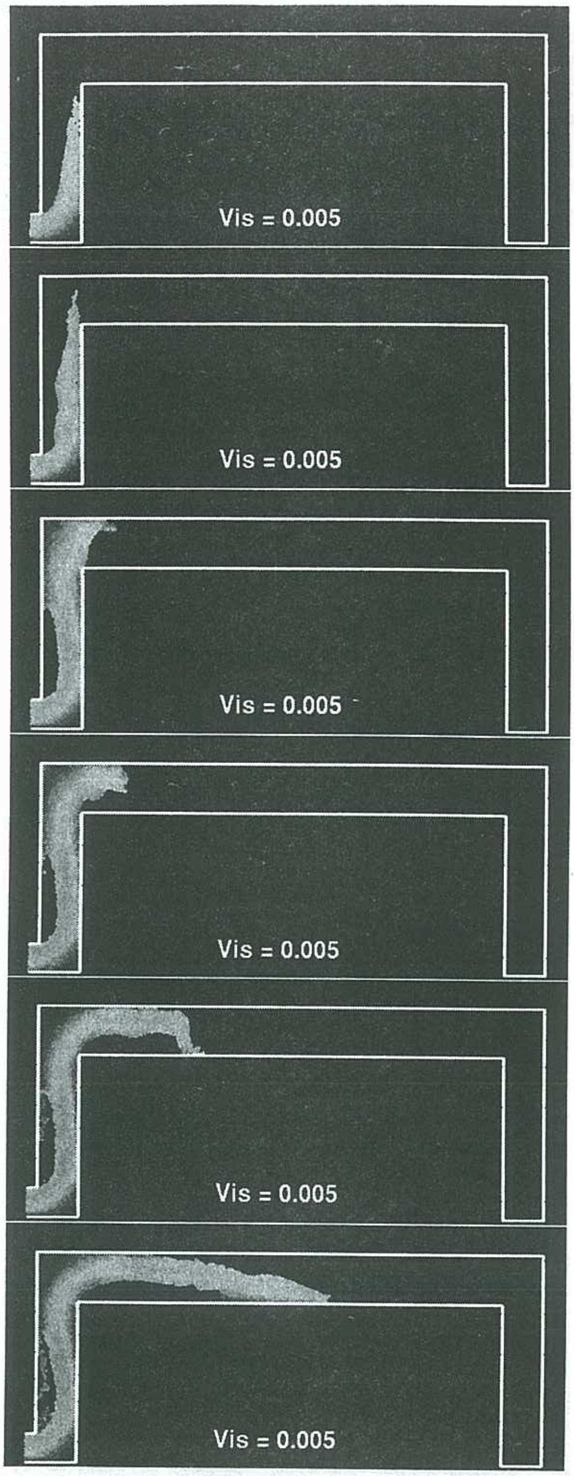

Figure 5: SPH result using $\mu = 0.005$ kg m^{-1} s^{-1}.

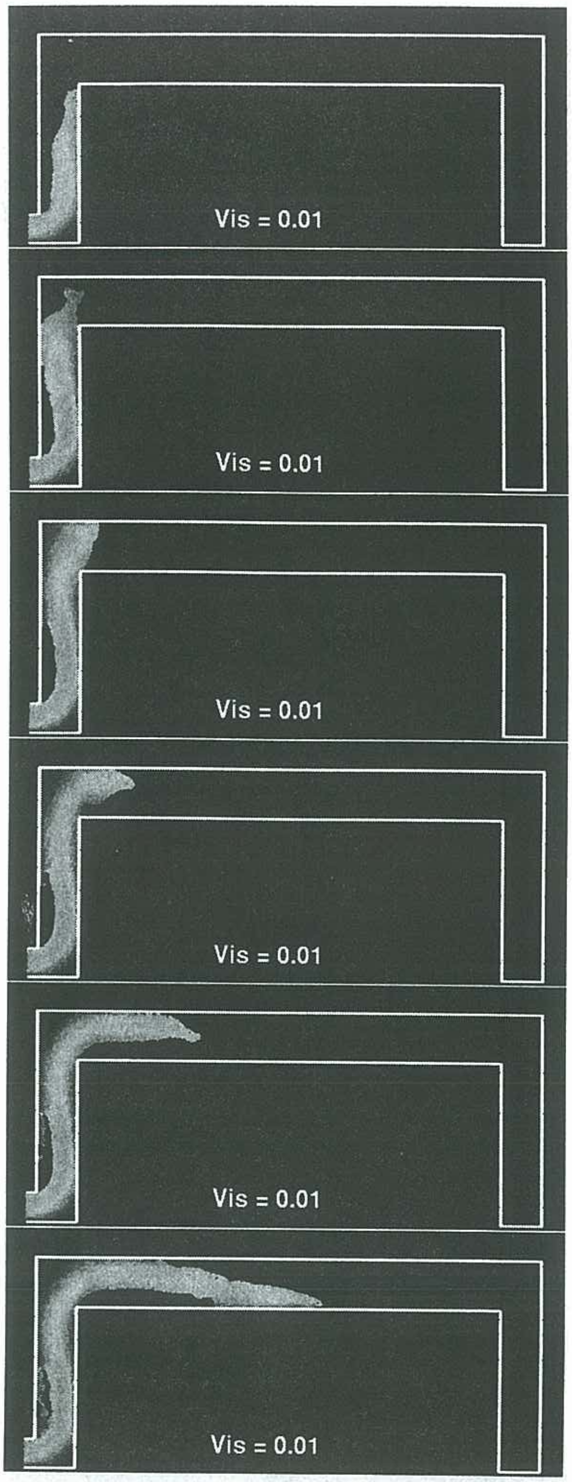

Figure 6: SPH result using $\mu = 0.01$ kg m^{-1} s^{-1}.

REFERENCES

MONAGHAN, J. J., "Smoothed particle hydrodynamics", *Ann. Rev. Astron. Astrophys.*, **30**, 543–574, 1992.

MONAGHAN, J. J., "Simulating free surface flows with SPH", *J. Comp. Phys.*, **110**, 399–406, 1994.

CLEARY, P. W., "High pressure die casting simulations with SPH: Isothermal single fluid flow", *Proc. Chemeca 97*, paper FL3B on CD Rom, 1997.

CLEARY, P. W., and HA, J., "SPH modelling of isothermal high pressure die casting", *Proc. 13th Aust. Fluid Mechanics Conf.*, 1998.

HA, J., "Comparison of SPH Simulation of Dam Breaking", SPH Technical Note #24, CSIRO Division of Maths and Info. Sciences, *Technical Report CMIS 92/97*, 1997.