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

SPH modelling of Isothermal High Pressure Die Casting

Paul CLEARY and Joseph HA

CRC for Alloy and Solidification Technology (CAST)
CSIRO Mathematical & Information Sciences
Clayton, Victoria, 3169, AUSTRALIA

ABSTRACT

Two dimensional simulations of high pressure die casting using SPH are presented for two moulds and for Reynolds numbers between 50 and 50,000. Realistic and complex flows with recirculations, vortices, back filling, droplet and fragment formation are predicted and can explain many types of casting defects.

INTRODUCTION

High pressure die casting (HPDC) is used extensively in the automotive industries to fabricate complex items from aluminium and magnesium. Liquid metal is injected into the mould at speeds of around 50 ms^{-1} under very high pressure. The resulting flow is fast and complex with substantial droplet and fragment formation. The filling pattern is far from one dimensional (front filling) and the behaviour of the free surface is very complex. HPDC dynamics are not well understood and can have high rejection rates for cast components. Defects include trapped air bubbles, fine scale porosity, strange material microstructures and oxide materials trapped in the casting which reduces its mechanical strength and produces variations in the surface properties. Experimentation is very difficult because of the speeds, temperatures and the thinness of the moulds. Numerical modelling is therefore attractive. Conventional Eulerian computational modelling techniques such as finite element and finite volume methods have been used with reasonable success to model low pressure slow die casting (Rosbrook, 1995), but are unable to cope with the extremely complex free surface behaviour found in HPDC.

SPH METHODOLOGY

Smoothed particle hydrodynamics (SPH) is a Lagrangian method for modelling heat and mass flows. Materials are approximated by particles that are free to move rather than by fixed grids or meshes. The governing PDEs are converted into equations of motion for these particles. It has been developed over the past two decades for astrophysical applications (Monaghan, 1992). Recently the method has been extended to incompressible enclosed flows (Monaghan, 1994). SPH has a several advantages that make it particularly well suited to this type of problem:

- It handles momentum dominated flows well.
- Complex free surfaces, including break-up into fragments are modelled naturally.
- Complicated physics such as multi-phase, realistic equations of state, compressibility, radiation and solidification can be added easily.
- It extends easily to three dimensions.

The interpolated value of function A at position $\mathbf{r}$ is:

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

where the sum is over all particles b within a radius $2h$ of $\mathbf{r}$. Here $W = W(\mathbf{r}, h)$ is a spline based interpolation kernel of radius $2h$. It is a C^2 function that approximates the shape of a delta function and has compact support. The gradient of the function A is given by:

$$\nabla A(\mathbf{r}) = \sum_b m_b \frac{A_b}{\rho_b} \nabla W(\mathbf{r} - \mathbf{r}_b, h) , \qquad (2)$$

From Cleary (1997) the SPH continuity equation is:

$$\frac{d\rho_a}{dt} = \sum_b m_b (\mathbf{v}_a - \mathbf{v}_b) \nabla W_{ab} \qquad (3)$$

It is Galilean invariant, has good numerical conservation properties and is not affected by free surfaces. The SPH momentum equation used is:

$$\frac{d\mathbf{v}_a}{dt} = \mathbf{g} - \sum_b m_b \left[\left(\frac{P_b}{\rho_b^2} + \frac{P_a}{\rho_a^2} \right) - \frac{\xi}{\rho_a \rho_b} \frac{4 \mu_a \mu_b}{(\mu_a + \mu_b)} \frac{\mathbf{v}_{ab} . \mathbf{r}_{ab}}{\mathbf{r}_{ab}^2 + \eta^2} \right] \nabla_a W_{ab} \qquad (4)$$

It automatically ensures continuous of stress across material interfaces and allows multiple materials with densities and viscosities varying by up to three orders of magnitude to be accurately simulated. The pressure is given by the stiff equation of state:

$$P = P_0 \left[\left(\frac{\rho}{\rho_0} \right)^{\gamma} - 1 \right] \qquad (5)$$

where P_0 is the magnitude of the pressure and ρ_0 is the reference density. For water or liquid metals the exponent $\gamma = 7$ is used and

$$\frac{\gamma P_0}{\rho_0} = 100 V^2 = c_s^2 . \qquad (6)$$

where V is the characteristic or maximum fluid velocity. This ensures that the density variation is less than 1% and the flow can be regarded as incompressible.

The boundaries are modelled as Leonard-Jones forces applied at the boundary particles and interpolated to produce arbitrary smoothly varying boundaries.

HPDC CONFIGURATION

In this first stage of modelling, the liquid metal is assumed to be isothermal and the air is neglected. We focus on two mould geometries:

- A C shaped mould with a 2 mm strip around the left, top and right sides of a 10×25 mm rectangle.
- A rectangular mould 10 mm high and 50 mm long.

The metal begins in a shot sleeve and is pushed into an asymmetrical constriction by a piston on the left. The constriction increases the pressure and accelerates the metal from the piston speed of 3 ms^{-1} up to around 50 ms^{-1} as it enters the 1 mm wide gate before jetting into the mould. Figure 1 shows an initially rectangular body of fluid being deformed as it approaches the gate. The metal is shaded according to its velocity. At $t = 1.7$ ms the deformation is still small and the velocity of the metal front has only increased to 5 ms^{-1}. By $t = 5.1$ ms the front has passed through the gate and accelerated to 43 ms^{-1}.

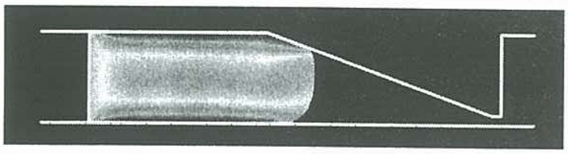

Figure 1: Piston accelerating liquid metal in the shot sleeve towards the gate.

The asymmetric shape of the shot sleeve gives the fluid a net downward motion as it enters the gate, so that it reflects upwards from the lower wall and enters the mould with a net upward velocity. The jet does not travel horizontally along the wall and creates opportunities for air to be trapped underneath. The characteristic length and velocity scales were chosen to be the gate width $L = 1$ mm and the velocity through the gate (around $V = 50$ m/s), respectively. The density was 1000 kg/m^3 and the viscosity was varied to give a range of Reynolds numbers. Simulation resolution refers to the number of particles spanning the shot sleeve vertically. Total particle numbers ranged between 12,000 and 300,000 depending on Re.

FILLING A C SHAPED MOULD

Figure 2 shows the filling at Re$=$ 500 using 100 particles between the top and bottom of the mould. The fluid enters and is forced upwards by the right wall without contacting the left one. The jet then collides head-on with the top wall and fragments. Most of the fluid moves to the right and forms an irregular horizontal jet. The rest is forced back down along the left side of the mould as a spray of droplets and is likely to trap air in the central gap in the vertical section between the upward and downward jets. This takes quite some time to fill with what would be partially solidified irregularly shaped fragments and would be expected to produce fine scale porosity and mechanical strength problems. Flakiness of the front surface of such castings is actually observed here.

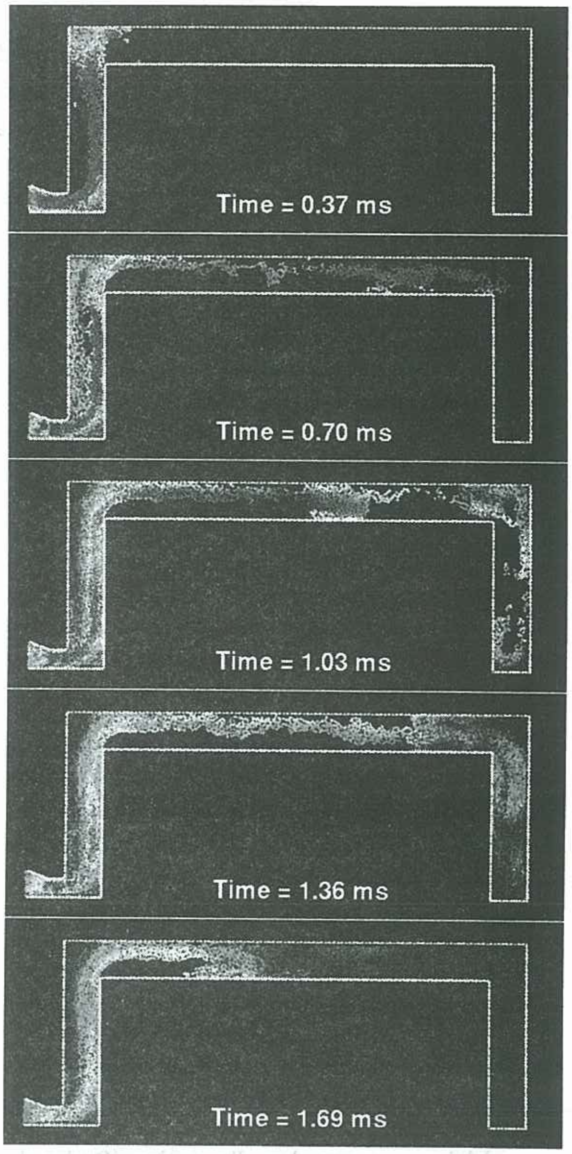

Figure 2: Filling of C shaped mould for Re$=$ 500.

Meanwhile, the jet travels along the upper part of the horizontal section to the end of the mould and is deflected downwards, again with the formation of fragments. As the end vertical section fills multiple recirculations appear. Once filled, there is a back flow and the lower part of the horizontal section fills. The region just beyond the second bend is the last to fill. Bubbles are frequently observed here in real castings.

FILLING A RECTANGULAR MOULD

The filling of a 10x50 mm rectangular mould at Re$=$ 500 is complex despite the simplicity of the mould shape. Figure 3 shows the filling at selected times. The upward inclined incoming jet contacts the right wall part way up and divides. The lower branch fills the lower right corner and generates a strong eddy. The upper branch flows around the outside of the mould and eventually re-contacts the jet, perturbing it just inside the gate. The free surface of the metal coating the right and top walls of the mould has a wavelike instability which is driven by the instability of the incoming jet. This oscillates causing the fluid to arrive at the right side of the mould in surges. Viscous effects push this material towards the centre of the mould and they develop into long filaments as they travel along the top. For a mould of this length, these filaments grow long enough to cross the centerline and collide with fragments of the destabilised incoming jet below. Such high speed collisions blast fragments around the central void. As the remaining filaments reach the bottom of the mould they further perturb the incoming jet and propagate back along the bottom. These collide violently with new ones travelling along the top surface, producing large numbers of elongated fragments and droplets, many of which tend to float in the middle of the cavity.

Once the mould is about half full two recirculating eddies form, one on either side of the central fragment filled void. When filling is complete, three eddies remain. Conventional CFD methods have difficulty with these applications because of the complexity of the free surface behaviour and the fragmentation.

This process is not an ideal way to produce uniform cast products. The large separation of the jet from the bottom wall provides good opportunities for air to be trapped there. Experimentally, a mirror point (the surface finish goes from bright to dull) is found on the lower surface of castings. The intimate contact shown between the incoming metal jet and the lower wall for the first third of the mould length would lead to the formation of such a mirror point. Many surface defects are experimentally observed along the bottom surface of the casting. These are consistent with the fragmented nature of the flow predicted in this region.

FLOW CHANGES WITH RE

For $Re < 300$ the flow is qualitatively similar with fluid in similar locations at similar times. The main differences are that high viscosities produce thicker and more rounded jets and flow structures and surface breakup is inhibited. The flow appears viscous and less free flowing and events which previously caused fragmentation now merely deform the free surface.

Figure 4 shows the filling of a C shaped mould for higher Re. The resolution required increases with Re, with 450 particles across the shot sleeve for $Re = 50,000$ (corresponding to a total of more than 300,000 particles). Qualitatively these flows are very similar to each other and to the $Re = 500$ case. Many of the differences are minor and involve a slightly greater rate of fragmentation. When a jet impacts on

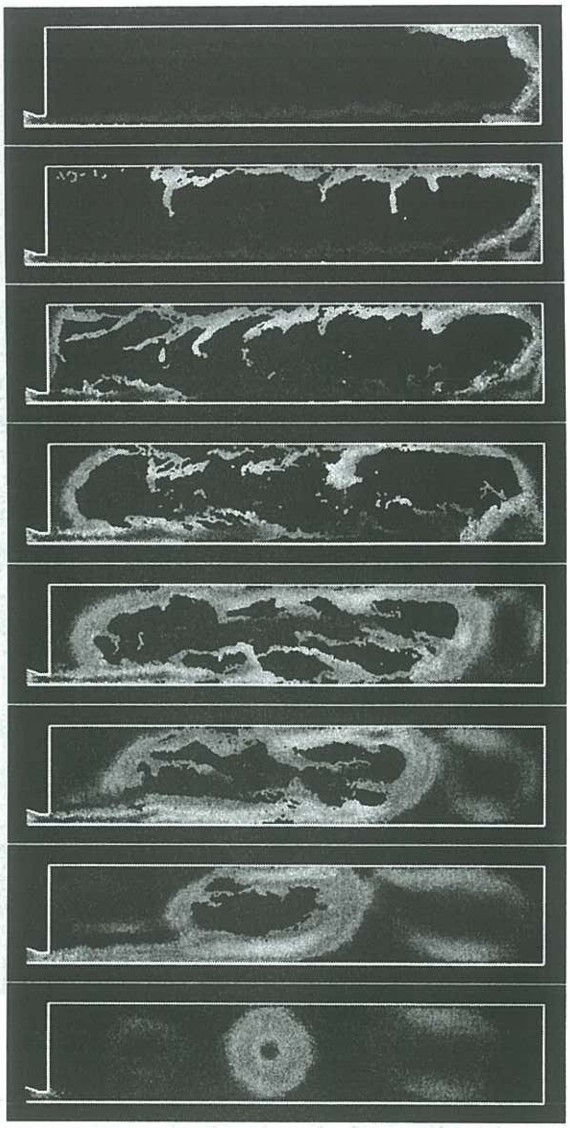

Figure 3: Filling of a rectangular mould.

a wall, the fluid is pushed to both sides. For right angle bends, one of these streams rapidly fills the closed side and the pressure increases, pushing all the fluid to the other side and producing the deflection of the jet around the corner. As the viscous forces diminish, the fluid responds faster and the deflected jet is produced earlier and is slightly narrower. The actual speed of the jet is independent of Re, but the earlier deflection of the jet means that the distance travelled along the horizontal section of the mould at a given time increases slowly with Re.

A new feature that develops with increasing Re is the remnant vortex. These are very strong eddies created in the upper and right parts of the mould by the original jet on the outside and the backfilling jet on the opposite side. These two jets are oppositely directed and introduce significant angular momentum to the flow. As the void between them collapses, the two jets break up into a sequence of vortices. As Re increases the viscous braking force declines and these vortices become very long lived. Viscous heating in such vor-

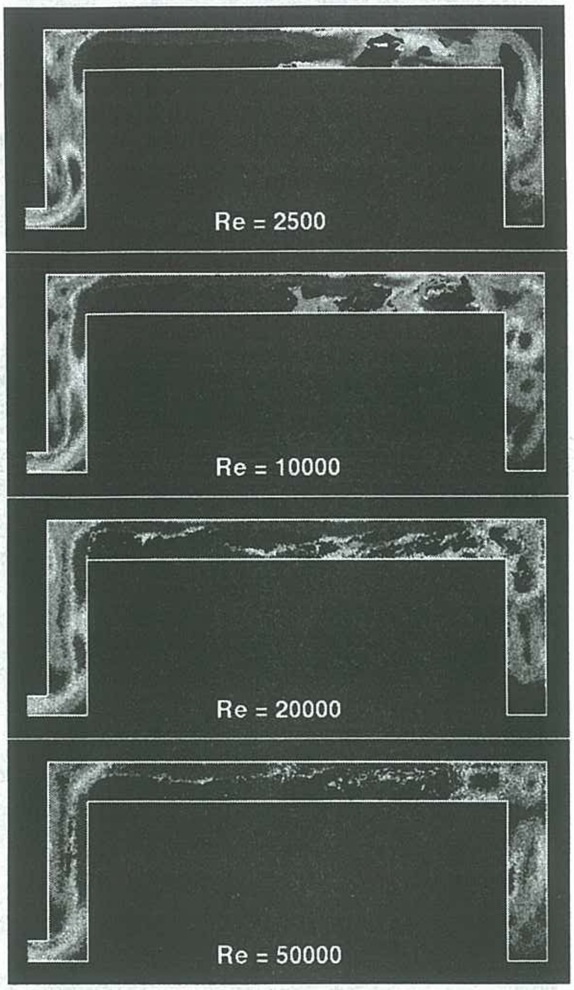

Figure 4: Filling of the C mould for higher Re.

tices is expected to produce re-melting. Strange microstructures consistent with re-melting have been found in the metal of castings in the upper part of the right vertical mould section.

EFFECT OF RESOLUTION

Figure 5 shows the mould filling at $t = 0.58$ ms for Re$= 50,000$ with 150 and 450 particles across the shot sleeve. The flows are very similar with all the essential features of the low resolution result reproduced in the high resolution one. Minor differences include the distance travelled along the horizontal section being slightly greater for the higher resolution as the pressure changes in the first turn are resolved better. The main difference is that the order of magnitude increase in the number of particles allows substantially better resolution of the undulating irregular structures in the jet. For resolution 150 the jet appears more as a spray, but at 450 it is clearly resolved as a twisting undulating jet. Importantly, they occupy similar regions of the mould and move in similar ways.

CONCLUSION

SPH has proved robust enough to simulate the difficult flows occurring in high pressure die casting. Significant breakup into drops and fragments is predicted at higher Re. This is consistent with observations from experiments. We find:

- For Re< 300 the flows are slow and viscous, whilst for higher Re they are momentum dominated and relatively invariant with Re. The main effects of increasing Re are a reduction in the smallest scales of structure in the flow and the creation of remnant vortices.
- Sharp bends lead to flow separation and the likely formation of bubbles beyond corners.
- Having the gate the same width as the mould significantly reduces the two dimensionality of the flow and improves the filling.
- 50 microns droplets are resolved for Re$= 2,500$.
- Many of the predicted flow structures are consistent with defects and surface properties observed in experiments and real castings. Three defect mechanisms were identified: bubble formation from separation, fine porosity from back-filling and strange microstructures from re-melting.
- High resolution simulations confirm the accuracy of the lower resolution ones.

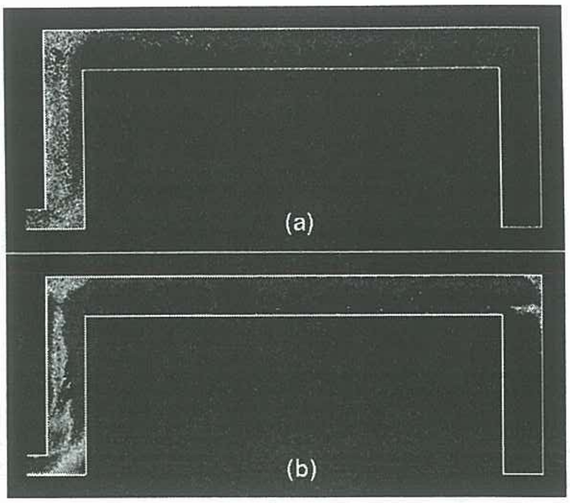

Figure 5: Flow pattern for resolutions a) 150 and b) 450 particles across the shot sleeve.

ACKNOWLEDGEMENTS

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

REFERENCES

ROSBROOK, C., "Recent advances in computer simulation of die casting – the die and beyond", *Die Casting Innovation*, Indianapolis T95-012, NADCA, 1995.

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.