Second International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
6-8 December 1999

THREE DIMENSIONAL MODELLING OF HIGH PRESSURE DIE CASTING

Paul W. CLEARY and Joseph HA

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

ABSTRACT

This paper reports on the development of 3D smoothed particle hydrodynamics for high pressure die casting simulations. Numerical simulations of two realistic dies are presented.

INTRODUCTION

High pressure die casting (HPDC) is an important process for manufacturing high volume and low cost automotive components, such as automatic transmission housings and gear box components. Liquid metal (generally aluminium) is injected at high speed (50 to 100 m/s) and under very high pressures through complex gate and runner systems and into the die. The geometric complexity of the dies leads to strongly three dimensional fluid flow with significant free surface fragmentation. Crucial to forming homogeneous cast components with minimal entrapped voids is the order in which the various parts of the die fill and the positioning of the gas exits. This is determined by the design of the gating system and the geometry of the die.

Smoothed particle hydrodynamics (SPH) has previously been used by Cleary and Ha (1998), Ha, et al. (1998) and Ha and Cleary (1999) to successfully predict two dimensional filling patterns in water analogue experiments for several dies of modest geometric complexity. Here we report the results of applying 3D SPH to the filling of two realistic dies. The methodology used to construct the simulation configuration starting from CAD input for the cast component through mesh generation to the SPH initial conditions will also be described.

SPH is a Lagrangian method and it does not need a grid to compute the spatial derivatives. The particles are the computational framework on which the fluid equations are solved. SPH will automatically follow complex flows. This makes the method particularly suited for fluid flows involving complex free surface motion. The method only performs calculations in the regions where mass is located. No computational time will be spent in empty regions. These attributes allow the method to relatively easily to handle three-dimensional problem. It is often stated that it is relatively easy to write an SPH computer program and to change a 2D computer code to handle 3D computation. This is certainly true for certain problems, such as those that do not include physical boundaries or those which are geometrically simple such as in astrophysics applications where SPH originates (Monaghan, 1992). For problems of industrial and engineering interest, there are three areas causing significant additional complexity that must be addressed in the 3D simulation code. The major difficulties of changing from 2D to 3D are in the handling of boundary conditions, nearest neighbour search and the initial set-up. Although these difficulties also apply to 2D simulations, they are much more acute in 3D. For example, many defects in the CAD model can be extremely difficult to detect in 3D and its repair can be an extremely laborious, manual-intervention process. This can be particularly difficult when trying to automatically assign boundary normals for SPH particles at surface intersections.

The aim of this paper is to report on the progress of our effort in developing a 3D SPH computer program for industrial and engineering applications, and to present results of a couple of 3D die filling simulations.

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 are associated with its centre, which is then interpreted as a particle. A particle b has mass m_b, position $\boldsymbol{r}_b$, density ρ_b and velocity $\boldsymbol{v}_b$. In SPH, the interpolated value of any field A at position $\boldsymbol{r}$ is approximated by:

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

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

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

In this way, the SPH representation of the hydrodynamic governing equations can be built from the Navier-Stokes equations. These equations of motion are given in Cleary and Ha (1998). The simulation progresses by explicitly integrating this system of ordinary differential equations. These SPH equations at a particle position are made up of terms similar to those of equations (1) and (2). The forms of these equations are the same regardless of the dimension of the governing equations. There is no reference to a computational grid. The particle position is the only geometric term in the equations. The computation of the sums in the equations requires only the identification of the particles' neighbours.

Boundary Conditions

To simulate confined fluid flow, such as die filling in high pressure die casting, it is necessary to prevent the fluid penetrating the physical boundary. One approach that has proved to be flexible and applicable to many problems is to replace the boundaries by boundary particles, which interact with the fluid by forces that are dependent on the orthogonal distance of the particle from the boundary. Arbitrary boundary surfaces can be readily represented by boundary particles. They have a further advantage that it is easy to simulate the motion of boundary particles.

To work with boundary particles, it is necessary to find a way to ensure the fluid particles feel a continuous boundary when two straight/curved boundaries are joined at edges or corners. If the boundary force is not continuous then nearby particle motions are unphysical and generally catastrophic for the simulation. The present implementation of the normal boundary force is described in Monaghan (1995) and involves the use of a repulsive Leonard-Jones potential force field that connects the adjacent boundary particles and repels the fluid particles. As fluid particles approach the boundary, the repulsive force rises rapidly and prevents them from penetrating. This approach is very flexible allowing arbitrary, smoothly varying boundary shapes. In the tangential direction, the particles are included in the summation for the shear force to give non-slip boundary conditions for the walls. Details are contained in Cleary and Monaghan (1993).

Neighbour Search

The summations in the SPH equations are over all particles within a radius $2h$ of $\boldsymbol{r}_a$ (the position of particle a). One of the basic requirements of an SPH computer code is the identification of the neighbouring particles of a given particle. In contrast to grid-based method, where the locations of neighbouring grid-cells are found directly, SPH is dependent on fast techniques for finding the neighbouring particles that contribute to SPH summations. Without an efficient method, it will degrade into a direct summation and computational time will scale as N^2, where N is the number of particles. One approach is to use a searching grid with linked lists to store the particles within each cell. This method works well when a constant smoothing length is used and is described in detail in Hockney and Eastwood (1988).

Initial Set-up

In a SPH calculation, one needs to initially specify particle masses, positions, velocities and other necessary quantities. All of these except the positions and masses are usually straight forward to specify according to the PDE initial conditions. Our strategy is to take the geometric description of a casting component from industry as input, to feed it through a commercial mesh generator and to then produce the initial set-up for SPH simulation using an in-house pre-processor operating on the FEM mesh produced by the mesh generator. This geometric description of the die is usually in the form of a computer-aided design (CAD) file. This file is parsed by the mesh generator (such as FEMAP) to produce the surface mesh of the die and volume meshes for the die volume and for the liquid metal. The nodes of the mesh become the positions of the SPH boundary particles. For the surface meshes, the boundary normals are calculated. For all particles, masses are calculated, material properties and other state variables are set to give the complete initial set-up for SPH simulation. The boundary normals are required for computing the boundary force described above. If it is required, the mesh generator will also produce a volume mesh in the selected region of the 3D object. The nodes of the volume mesh are positions of the initial fluid particles.

NUMERICAL RESULTS

In this section, SPH simulations of two dies are presented. The two dies are the C-shaped mould (see Figure 1) and a die cast object (see Figure 3) chosen to exhibit typical features of cast objects. The C-shaped mould is geometrically simple, enabling its initial set-up to be built by hand. The simple geometry also allows any programming problem that is associated with the geometry to be identified relatively more readily. Furthermore, it allows easier capture of "clean" experimental images from water analogue modelling. The second die shown in Figure 3 is more representative of industrial and engineering casting objects. It is a testing example for our methodology for constructing the initial set-up for a SPH simulation.

C-shaped Mould

Figure 1 shows the geometry of the die used in previous experimental and numerical simulations (Ha, et al., 1998 and Cleary, et al., 1999). It is 50 mm long, 20.9 mm high and 20 mm deep. It is connected to the runner by a gate of width 2.9 mm. The width of the vertical sections is 4 mm and the width of the connecting horizontal section is 4.9 mm.

A resolution of 2.6 particles per millimetre was used giving a total of 208,054 particles for the simulation. The fluid speed through the gate was 0.62 m/s. Two perspective views of the filling pattern at 6 different times are shown in Figure 2. The fluid particles are shown coloured by their speed.

The flow in the xy-plane is broadly similar to that predicted by the 2D SPH simulation and observed in the matching experiments (Ha, et al., 1998). In particular, the separation from the right angle bends and the back filling down the left side of the first vertical section are reproduced.

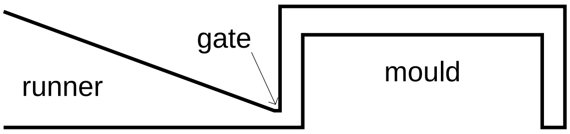

Figure 1: Cross section of the die.

Concentrating on the flow after 28 ms when the horizontal section of the die cavity is beginning to be filled, the two views show the highly fragmented nature of the fluid and the three dimensionality of the fluid flow pattern. These features continue as the die cavity is increasingly filled, with many voids in the fluid. These voids move in an irregular and complex pattern and would lead to significant porosity in the final cast component.

Die Cast Object

The geometry of the die, the gate, the runner and the cylindrical plunger section is shown in Figure 3. In real die casting, the cylinder orientation would be horizontal and the part vertical but for the modelling we use a reference orientation as shown here. The fluid initially fills the cylindrical column and is pushed downward by a plunger at the top of the fluid moving at 15 m/s. In the simulation, a resolution of 1 particle per millimetre was used giving a total of 243,576 particles.

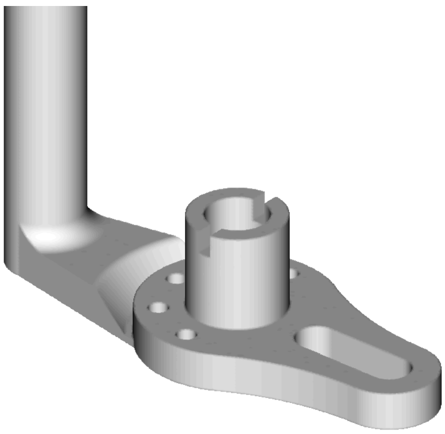

Figure 3: A 3D mould with cylindrical plunger on the left leading to a divergent runner, through a curved gate into the die of a 3D machine component.

Two perspective views of the filling pattern at 3 different times are shown in Figure 4. The fluid particles are shown coloured by their speed. The first frame at 1 ms shows the system just as the fluid enters the runner. The second frame at 2.8 ms shows the fluid having mostly filled the runner and the leading fluid splashing through the gate. This leading material consists of fast moving fragments and droplets generated by splashing when the leading fluid flowed around the right angle turn as it entered the runner. In the final frame (4.52 ms) the fluid has entered the die proper and has split into separate jets around each of the vertical cylindrical cut-outs. After collision with these cut-outs and central cylindrical core, the fluid begins to flow vertically up into the upper part of the die cavity. At the time the upper part of the runner is filled by a reverse flow reflecting from the surface above the gate back towards the shot sleeve. Note the rapid acceleration of the fluid particles as they pass through the gate produced by the converging surfaces of the runner.

Visualisation of particle systems in 3D is quite difficult with the three dimensionality of the flow difficult to perceive when using just the particles. An alternative is to construct iso-surfaces based on the underlying particle data. A substantial difficulty is that the data is non-structured with no inherent geometrical relationship between particles. In principle, a given particle can be anywhere in the computational domain. Figure 5 shows a rendered surface mesh coloured by fluid velocity which has been created from the particles and shows the fluid distribution as the fluid front passes through the gate and enters the die cavity. The fluid surface conveys much more information to the viewer than do the particles, in particular depth perception and variations in the surfaces are much easier to identify. In this case, it can be seen that the fluid separates from the front wall of the cylinder as it enters the runner producing a thinning sheet of fluid that flows into the gate. At the same time it spreads sideways and flows up the side walls of the runner to the top surface, giving a U-shaped profile in a cross section through the runner parallel to the gate.

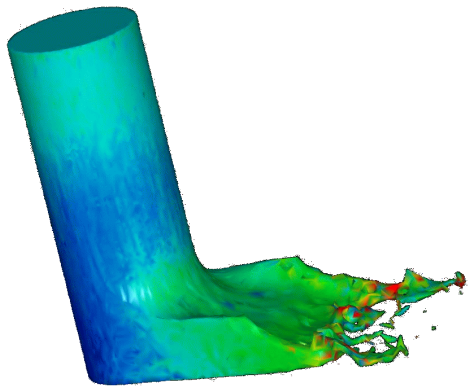

Figure 5: Liquid metal surface (coloured by fluid velocity) as the fluid passes through the gate visualised using a surface mesh calculated from the SPH particles.

Using another visualisation option, the particles are mapped to a rectangular grid which is then used to display the fluid surface. Figure 6 shows four views of the flow as die cavity fills. Note the introduction of the staircase artefacts in the surface. These are produced by the visualisation grid and are not present in the SPH data. This indicates that mesh based descriptions of the surface better represent the fluid than interpolation to an underlying grid.

CONCLUSION

In this paper, we have described the SPH method and the application of SPH to simulate the 3D die filling in high pressure die casting. The methodology for 3D industrial SPH modelling involves the CAD specification of die geometry, the construction of FEM meshes using comercial mesh generators and the conversion of this to SPH input data. The difficulty associated with visualisation of complex 3D free surface flows using non-structured particle data were also highlighted.

The two examples demonstrate the complex 3D flow patterns in the die filling process. The complex free surface behaviour include splashing and surface breakup are handled naturally by SPH. This results from the Lagrangian nature of SPH and the superior mass conservation properties of this particle method.

ACKNOWLEDGEMENTS

This project is funded by the Cooperative Research Centre for Alloy and Solidification Technology (CAST).

REFERENCES

CLEARY, P.W. and HA, J., (1998), "SPH modelling of isothermal high pressure die casting", *Proc. 13th Australasian Fluid Mechanics Conference*, Melbourne, Australia, December 663-666.

CLEARY, P.W., HA, J. and AHUJA, V., (1999), "High pressure die casting using smoothed particle hydrodynamics", *Int. J. Cast Metal Res.,* in press.

CLEARY, P.W. and MONAGHAN, J.J., (1993), "Boundary interactions and transition to turbulence for standard CFD problems using SPH", *Proc. 6th International Computational Techniques and Applications Conference,* Eds. Stewart, D., Gardener, H., and Singleton, D., Canberra ACT, pp. 157.

HA, J., CLEARY, P. and AHUJA, V., (1998), "Comparison of SPH Simulation of High Speed Die Filling with Experiment", *Proc. 13th Australasian Fluid Mechanics Conference*, Melbourne, Australia, December 901-904.

HA, J. and CLEARY, P., (1999), Comparison of SPH simulations of High Pressure Die Casting with the experiments and VOF simulations of Schmid and Klein, *Int. J. Cast Metals Res.*, submitted.

HOCKNEY, R.W. and EASTWOOD, J.W., (1988), "Computer Simulation Using Particles", Institute of Physics Publishing Ltd.

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

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

MONAGHAN, J.J., (1995), "Improved modelling of boundaries", SPH Technical Note #2, CSIRO Division of Mathematics and Statistics, *Technical Report DMS - C 95/86*.

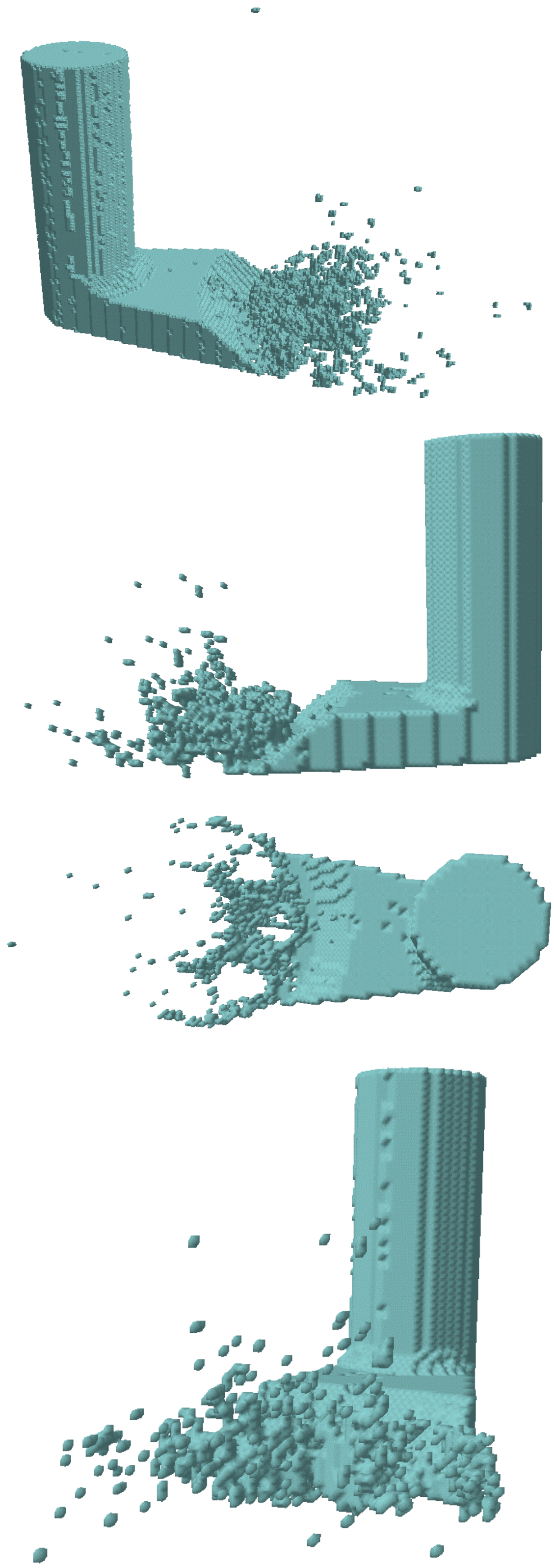

Figure 6: Liquid metal surface when the die cavity is partially filled using a rectangular grid for surface visualisation.

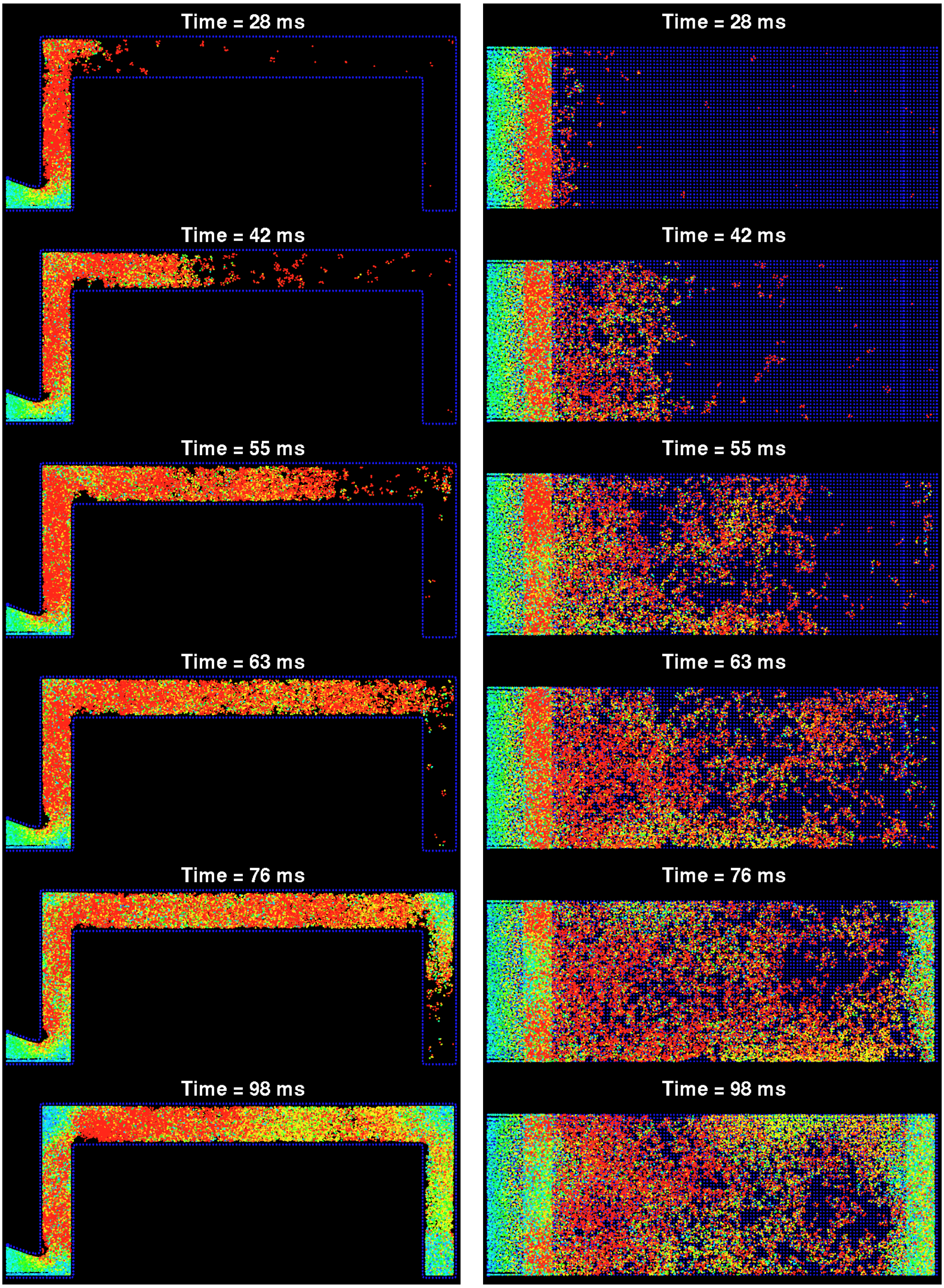

Figure 2: Two perspective views of the filling of the C-shaped mould. The particles are coloured by velocity with red being 60 m/s and dark blue being stationary. Left: xy-plane. Right: xz-plane.

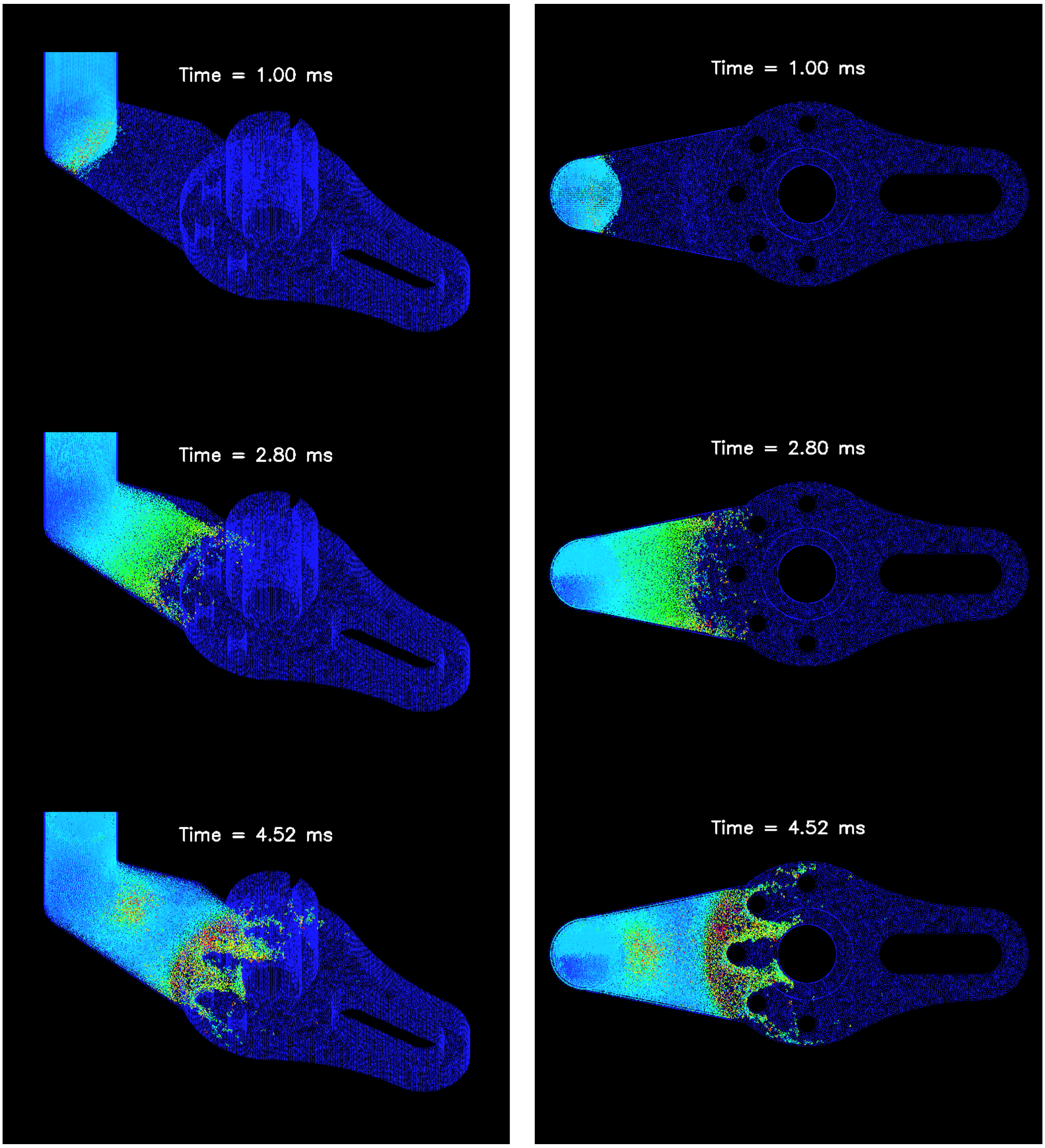

Figure 4: Two perspective views of the filling process at three times, as indicated. The particles are coloured by velocity with red being 60 m/s and dark blue being stationary.