11th Australasian Fluid Mechanics Conference
University of Tasmania, Hobart, Australia
14-18 December 1992

THE FILLING OF A VERTICAL PLATE MOULD

C.J. WERE and G.D. MALLINSON

Department of Mechanical Engineering
University of Auckland
Private Bag 92019, Auckland, NEW ZEALAND

ABSTRACT

The filling of a vertical plate mould, with simple two dimensional geometry, has been modelled numerically using a new "Volume of Fluid" based code developed especially for free-surface flow. A simple visualization experiment has been performed to assess the validity of the numerical results.

Comparison of the experimental results with those produced by the new code show qualitatively good agreement. In particular, the new code successfully predicts the formation of the dominant features of the flow: a large "breaking wave" (which would lead to undesirable gas entapment and oxide folding in a casting situation), a smaller secondary wave, and a large vortex.

INTRODUCTION

Free surface flow modelling finds application in a number of areas within civil and mechanical engineering, from ocean wave prediction to sloshing in tanks. While useful analytic approximations such as shallow water theory are useful for the former, the prediction of a transient free-surface of complex geometry, typical of the latter, requires the simulation of the Navier-Stokes equations, using discrete approximation (such as finite elements or finite differences).

The particular application under consideration here is the filling of foundry moulds. The accurate modelling of the filling stage in combination with a solidification model can be used to detect and eliminate costly casting defects such as "cold-shut". Free surface modelling of the pouring phase can provide useful information even in the absence of a heat transfer/solidification model, by predicting the folding of surface oxides and gas entrapment into the body of the casting.

A new computer code has been developed based on the "Volume of Fluid" technique of Hirt and Nichols. The principal reason for this development was to explore the capabilities and limitations of the technique, and if possible, enhance it. The filling of a narrow vertical plate mould, previously studied by Lin & Hwang (1988), was chosen as a benchmark problem. Other test problems, including the breaking of a dam, Rayleigh-Taylor instability, an undular bore and the surface tension driven oscillation of a small droplet were also investigated; these results are reported in detail elsewhere (Were, 1992).

THE CASTING PROBLEM

The geometry of the pouring simulation is shown in Figure 1. The thickness of the cavity is nominally 10mm, giving a characteristic aspect ratio (length:thickness) of 20:1. While the boundary geometry is very simple, the free surface configuration at the inlet velocities considered here (1.1 and 0.75ms^{-1}) is highly irregular, dictating the use of a fine computational mesh. The casting fluid is molten aluminium, though water may be used without loss of generality.

This problem has previously been examined by Lin and Hwang using the "Solution Algorithm Volume-of-Fluid" (SOLA-VOF, Hirt & Nichols 1981) code with a 20 by 20 mesh, and is thus used as a benchmark. A simple water flow visualization experiment was also performed.

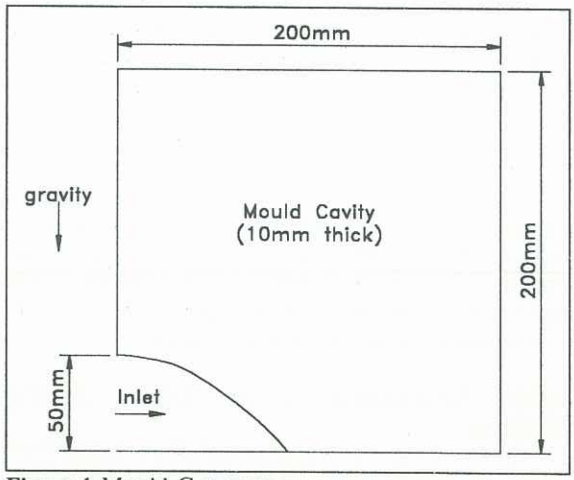

Figure 1 Mould Geometry

MODELLING PROCEDURE

The modelling technique used is based on a hybrid of the SOLA-VOF and Marker and Cell (MAC, Harlow & Welch 1965) algorithms. Essentially the free surface tracking and boundary conditions of the SOLA-VOF method (the "Volume-Of-Fluid" concept) has been combined with the Navier-Stokes approximations of the MAC technique.

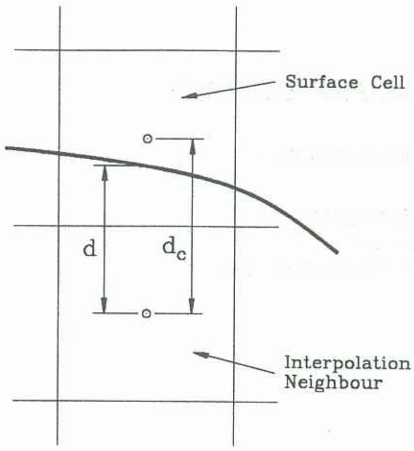

Figure 2 Surface Pressure Interpolation

Volume of Fluid Method

A Volume of Fluid (VOF) function, F, is used to trace the presence of fluid within the cavity. F has a simple physical interpretation as the fraction of a cell volume filled with fluid

$$\textit{Volume of Fluid in a Cell} = F_{cell}\,\delta x\,\delta y\,\delta z \qquad (1)$$

The time evolution of F is governed by the additional advection equation

$$\frac{\partial F}{\partial t} + u\frac{\partial F}{\partial x} + v\frac{\partial F}{\partial y} = 0 \qquad (2)$$

This must be represented as accurately as possible in the discrete approximation; in particular numerical diffusion must be minimised to prevent physically unrealistic "smearing" of the free surface. A heuristic differencing scheme known as the "Donor-Acceptor" method is used for this purpose. Essentially this method uses a standard upwind difference if the free surface is mostly parallel to the advection direction, and a downwind difference if it is mostly normal to it. In this way negative numerical diffusion is employed to maintain a sharp interface. Limits are placed on advected quantities to prevent unrealistic values of F occurring in any cell. The explicit nature of this computation phase imposes a Courant time step limitation, and this in turn suggests that an implicit or semi-implicit formulation for the momentum equations is unwarranted.

Surface Approximation in the Volume of Fluid Technique

At any time, cells are classified as being "full", "empty" or "surface" depending on their current values of F, and those of the neighbouring cells. The approximate surface orientation in a "surface" cell is obtained by evaluating spatial derivatives of F. Surface curvature may also be approximately evaluated enabling surface tension effects to be included in the model. Pressure boundary conditions are applied using an interpolation scheme to apply the external pressure at the approximate location of the surface within the cell, rather than at the centroid of the surface cell.

The surface itself is represented as a straight line segment within the surface cell. If the surface is mostly horizontal, then Y(x) may be estimated by:

$$Y_{i,j} = (F_{i,j+1} + F_{i,j} + F_{i,j-1})\,\delta y \qquad (3)$$

Here, Y is measured from the baseline of the (i,j-1) cell. Thus the height is effectively averaged over three vertical cells. A similar expression may be derived for X(y) in the mostly vertical case. In order to determine whether the surface is mostly horizontal or vertical, it is necessary to approximate the slope. Central differences are used to calculate both dY/dx and dX/dy. The relative magnitudes of these gradients are used to determine the surface orientation.

The pressure in a surface cell is given by the interpolation formula

$$p_{surf} = (1 - \eta)p_n + \eta p_{ext} \qquad (4)$$

Here p_n is the pressure in the cell containing fluid "below" the surface; this cell is referred to as the interpolation neighbour. $\eta = \frac{d_c}{d}$ is the ratio of the distance between the surface cell centre and the interpolation neighbour cell centre, over the distance from the surface to the neighbour's cell centre (see Figure 2). The neighbour cell is chosen so that the cell centre to cell centre line is most normal to the free surface.

Pressure Solution

The original SOLA-VOF code (Hirt & Nichols 1981) did not form the Poisson equation directly, but instead computed local pressure corrections applied to each cell in turn, a procedure roughly equivalent to Successive Over-Relaxation. Several workers (for example Stoehr, Wang *et al* 1986, Walther & Sahm 1988) have reported divergence problems with this method, stemming from the surface pressure interpolation. In the new code the pressure solution is formulated as a Poisson equation at each time step as in the MAC method. By means of this direct formation of the Poisson equation, more efficient and stable solution methods become available.

The most successful Poisson solution method was found to be a bi-conjugate gradient method (Van der Vorst 1992), coupled with a preconditioner based on Schneider and Zedan's (1981) Modified Strongly Implicit procedure. This provided rapid convergence and good stability during the pressure iteration enabling the practical use of large meshes (the 64×64 cell example below was advanced through 871 time steps in approximately 45 minutes on an RS6000-320). This technique will be the subject of a future paper.

The Problem of Surface "Smearing"

While the instability in the pressure iteration was eliminated by the modified Poisson solution technique, an instability arising from surface "smearing" was still a problem, particularly as the mesh was refined. Smearing can occur when two regions of the free surface collide or where the surface curvature is so extreme (relative to cell size) that the simple line segment approximation used for the surface is no longer valid. This in turn can cause the surface pressure interpolation parameter, η, to assume negative values which tend to reinforce the undesirable smearing behaviour, causing more fluid to "leak" across the surface, and leading to instability. A similar effect occurs in regions of large negative surface curvature, where void "leaks" into the interior fluid domain.

Although the new code includes a stategy to detect and correct smearing, the method is unfortunately not foolproof.

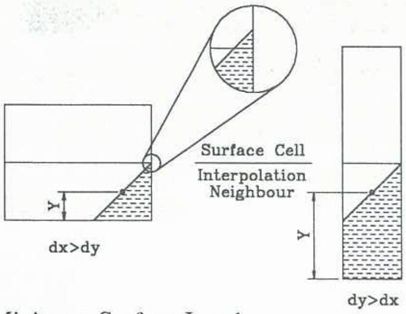

Figure 3 Minimum Surface Level

Smearing is detected geometrically. Based on the surface level and orientation, minimum acceptable values for the height function Y can be computed (a similar expression may be derived for X in the mostly vertical case). For example, the minimum value the height function Y can take, under the assumption that the surface may be represented locally by a straight line, is found by considering the surface to just enter the corner of the surface cell and travel downwards at 45 degrees (Figure 3). The minimum Y value for this cell is then (for $\Delta x \geq \Delta y$):

$$Y = \frac{\Delta y^2}{2\Delta x} \qquad (5)$$

or, for $\Delta x \leq \Delta y$:

$$Y = \Delta y - \frac{\Delta x}{2} \qquad (6)$$

These limits may be violated if there is a small amount of "unexpected" fluid in the surface cell. If this occurs, the fluid is simply added back into its interpolation neighbour; thus the old surface cell becomes empty and the interpolation neighbour (generally) becomes the surface cell. By virtue of the constraints implied by (5) and (6) there is no danger of "overflowing" the interpolation neighbour, and the amount of fluid shifted is generally very small.

RESULTS

The same mesh size (20×20) and inlet velocity (1.1m/s) were originally used to compare the new code with the SOLA-VOF results of Lin and Hwang. Very similar free surface plots were obtained, with slight differences attributable to different time step size, advection scheme and fluid properties (Lin & Hwang do not give this data).

The code was also used on more refined meshes (up to 128×128) with an inlet velocity of 0.75 ms^{-1} and water as the fluid to compare with a simple flow visualization experiment. Unfortunately the smearing instability becomes more persistent as the mesh is refined, and while the early time results appear to converge well, later time ones are affected strongly and erratically by the smearing process. Smearing was alleviated by both the smear detection/correction scheme described above and the application of a physically realistic surface tension (0.078Nm^{-1} for water); however it was finally necessary to increase the viscosity to $1.6\times10^{-4}m^2s^{-1}$ (from $1.0\times10^{-7}m^2s^{-1}$). This had only a small observable effect on the early time solution, in the form of slight thickening of surface "jets" of fluid.

The visualization experiment was performed with potassium-permanganate coloured water, which is kinematically very similar to molten aluminium with the exception that water has a much lower surface tension acceleration. As the characteristic Weber number for this problem is in any case very large, this difference is unlikely to have much effect. The inlet boundary condition and impulsive start are simulated (somewhat crudely) by a rapidly opening brass sluice gate between the perspex cavity and an entry flume connected to a header tank.

Plates 1 to 3 are taken as being representative of the generic flow pattern at a mean inlet velocity in the vicinity of 0.7 ms^{-1}; they are compiled from several different runs. Direct comparison has been performed using multiple exposure strobe photography (enabling accurate timing) and video recording. Observation of the complex and erratic free surface motion on the video footage provides ample evidence why this is a particularly difficult phenomenon to simulate.

Plate 4 is a multiple exposure photograph of the flow at approximately 0.75ms^{-1} mean inlet velocity (using a 5Hz strobe). This has been used for comparison with the numerical results.

Figure 4 shows a series of free surface/velocity vector plots from a 64×64 mesh simulation. The latter frames still show smearing occurring at the top left of the cavity, as indicated by the presence of "fluid" well away from the F=0.5 contour line used to represent the free surface, however by this stage much of the interesting flow behaviour has already occurred.

In spite of the numerical difficulties discussed above, the simulation results have several features in common with the plates. The models correctly predict the formation of the jet at left wall, evident in Plate 4, and correctly predict that the upper left corner of the mould

Plate 1

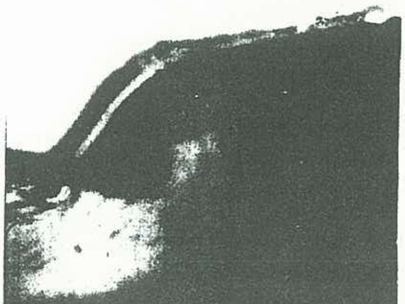

Plate 2

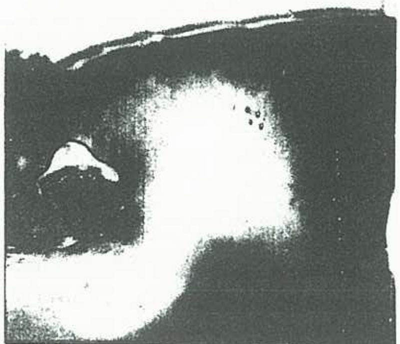

Plate 3

will be the last to fill. This is in contrast to the original higher velocity model, where the last region to fill was the upper right corner.

Importantly, the model correctly predicts the entrapment of gas by the left breaking wave, and thus the danger of folding oxides into the body of the melt. The model will not accurately predict the subsequent development of the trapped "bubbles" however as it does not conserve mass within "void" regions. A bubble will thus contract and disappear in this simulation.

CONCLUSIONS

The donor-acceptor technique has been found to "leak" fluid or void in the regions of high surface curvature. These "smearing" problems can be greatly reduced by a combination of the application of realistic surface tension, the smear detection correction algorithm introduced, and as a last resort, increased artificial viscosity.

The fine grid results could not have been obtained without the move away from the SOR-like "Solution-Algorithm" (SOLA), towards more efficient iterative solution methods, based on the direct solution of a Poisson-like equation for pressure.

The modelling of a casting-related problem, the filling of a vertical plate mould, was successfully performed using the new code. Qualitatively good agreement was obtained between simulation results and a flow visualization experiment.

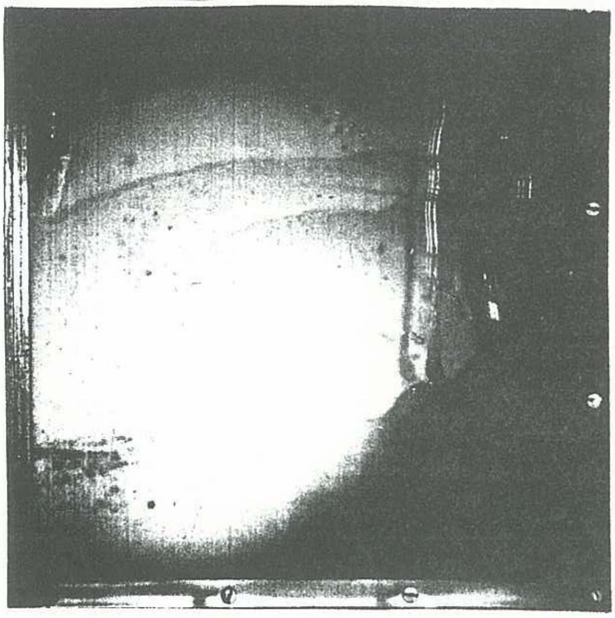

Plate 4 Strobe Photograph

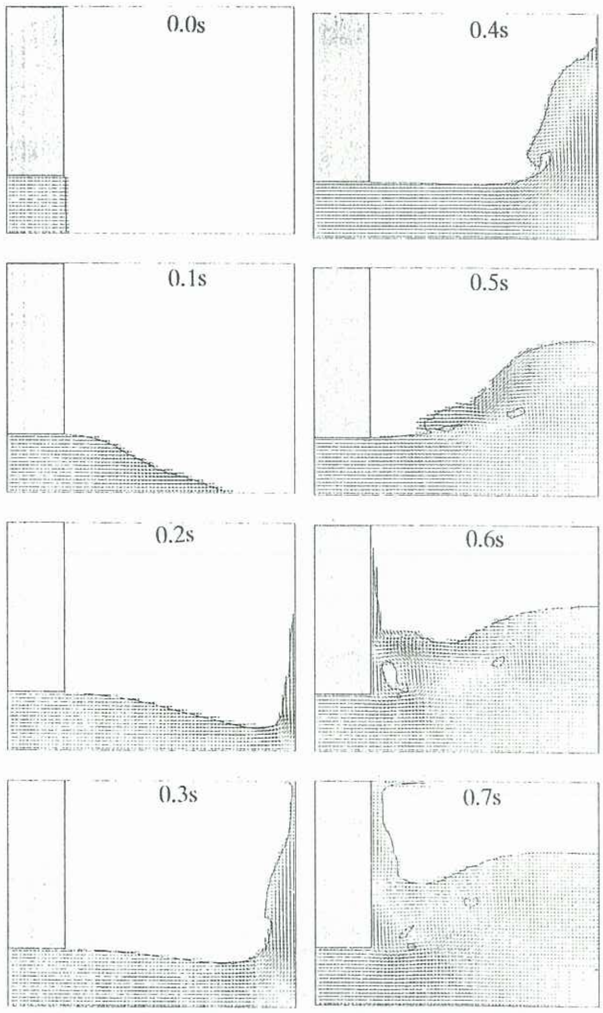

Figure 4 Numerical Simulation Results (64×64 mesh) for times 0.0s to 0.7s (0.1s increment)

REFERENCES

Harlow F.H & Welch J.E., 1965
Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free-Surface, Physics of Fluids, Vol 8 No 12 (Dec 1965), pp 2182-2189.

Hirt C.W. & Nichols B.D., 1981
Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, J. Comp. Phys., Vol 39 (1981), pp201-225.

Lin H-J & Hwang W-S, 1988
Combined Fluid Flow and Heat Transfer Analysis for the Filling of Castings, Modelling and Control of Casting and Welding Processes IV, 1988, pp487-511.

Schneider G.E & Zedan M., 1981
A Modified Strongly Implicit Procedure for the Solution of Field Problems, Numerical Heat Transfer, Vol 4 (Jan 1981), pp 1-19.

Stoehr R.A., Wang C., Hwang W.S., Ingerslev P., 1986
Modelling The Filling of Complex Foundry Moulds, Modelling and Control of Casting and Welding Processes III, 1986, pp303-313.

Walther H. & Sahm P.R., 1988
Simulating and Modeling the Filling of a Mold, Modelling and Control of Casting and Welding Processes IV, 1988, pp635-644.

Were C.J., 1992
A "Volume Of Fluid" Technique for Modelling Unsteady Free Surface Flow, ME Thesis, School of Engineering, University of Auckland.

Van der Vorst, H.A., 1992
Bi-CGSTAB: A Fast and Smoothly Converging Variant of Bi-CG for the solution of nonsymmetric linear systems, SIAM J. Sci. Stat. Comp., March 1992.