ELSEVIER

Computational Materials Science 22 (2001) 200–212

COMPUTATIONAL MATERIALS SCIENCE

www.elsevier.com/locate/commatsci

Simulating a double casting technique using level set method

Qiang Du [a,*], Dianzhong Li [a], Yiyi Li [a], Ruo Li [b], Pingwen Zhang [b]

[a] *Institute of Metal Research, Chinese Academy of Sciences, 72 Wen Hua Rd, Shenhe District, Shenyang 110015, China*

[b] *School of Mathematical Science, Peking University, Beijng 100871, China*

Received 20 December 2000; received in revised form 15 February 2001; accepted 3 May 2001

Abstract

Level set method is an appropriate mathematical tool for solving two-phase flow problems. It is used here for the first time to track the interface evolution during the process of casting a second alloy into mold, partly displacing the first to achieve a casing with a different alloyed skin. Projection method for Navier–Stokes equations was adopted and the algorithm was implemented in finite element method. The numerical example shows that though in those cases where the layer of liquid metal above is lighter than that below, the interface is not always stable for density ratios less than one of the two liquid metals. The stability of the interface is sensitive to the density ratio and inlet velocity and relatively insensitive to the viscosity ratio or other parameters. It is believed that level set method has potential to be further used in real double casting processes. © 2001 Elsevier Science B.V. All rights reserved.

Keywords: Level set method; Two-phase flow; Double casting; Alloyed surface

1. Introduction

The process of mold filling in the metal forming industry had long been based on the intuition and experience of foundry engineers and designers. In order to bring the industry to a more scientific basis, many researchers have tried to integrate the design process with numerical simulation, such as fluid flow and heat transfer. The difficulty of simulating mold filling is to track the evolution of free surface. In general, Mark and Cell (MAC) [1] or Volume of Fraction (VOF) [2] algorithm is adopted. But insufficient precision and computational instability hindered their application, especially for two-phase flow arising in casting a second liquid metal. In this technique, the mold is filled with the first liquid metal, which is allowed to solidify to a desired layer. After that, the second liquid metal is poured, flushing the remaining first un-solidified liquid metal away, and leaving the second liquid metal occupying the space left by the first metal.

This process, which is a typical two-phase flow, is often applied for the production of cast rolls. Sometimes because the processing parameters are incorrect, the interface is broken down, two kinds of liquid metal mix, and cause waste of materials. It is difficult to determine how much the inlet velocity should be and when to make the second pour using only experiences. With the help of filling simulation, engineers will visualize the evolution of the interface between the two kinds of liquid metal. Although it may not be easy to simulate because of the complexity of the flow pattern

* Corresponding author. Tel.: +86-24-2384-3531.
E-mail address: qdu@imr.ac.cn (Q. Du).

0927-0256/01/$ - see front matter
PII: S0927-0256(01)00190-2

and the interaction of two kinds of fluid, now it is feasible to realize with the aid of the powerful mathematical tools, level set algorithm.

Level set method, developed especially for tracking the interface between two kinds of fluid, can compute the interface change, and interaction between two kinds of fluid. It is a good way for interface tracking in solving two-phase flow problems compared with other methods. The core idea of level set method is to introduce a transition layer at the free interface of the two different fluids, and try to connect the two fluid fields as an integrate field. The main virtue of level set is that the fluid field is treated as a single one, so special code for the free boundary is not needed, and it's efficient to deal with complex interface, even if topology changes. Level set method incorporate the boundary condition at the interface into the governing equation, and it is not necessary to locate the position of interface as VOF or MAC method. The method has been applied in simulations such as the falling of water drops in air and the rising of air bubbles in water [3].

In this paper, level set method is used for the first time in simulating the flushing of a second alloy through a mold filled with a first alloy. The effects of the density ratio of the two kinds of liquid metal, the viscosity and inflow velocity are investigated. Although only a simple test is demonstrated in this first case, level set promises to be a powerful aid for the design of real parameters of the double casting technique.

2. Level set method

Here we give the scheme for applying the level set method on the incompressible two-phase flow. The governing Navier–Stokes equation for the incompressible two-phase flow in two dimensions, which describes the case of mold filling, is as:

$$\frac{\partial \vec{u}}{\partial t} + (\vec{u}\cdot\nabla)\vec{u} = \vec{g} + \frac{1}{\rho}\left(-\nabla p + \frac{1}{Re}\nabla\cdot(2\mu D) + \frac{1}{B}\kappa\delta(d)\vec{n}\right), \quad (1)$$

$$\nabla\cdot\vec{u} = 0 \quad (2)$$

in which

$$D = \frac{1}{2}(\nabla\vec{u} + (\nabla\vec{u})^{\mathrm{T}}),$$

where $\vec{g}$ is the gravity, μ is the viscosity coefficient, κ is surface tension coefficient, $\delta(d)$ is the delta function, d is the distance from the interface and $\vec{n}$ is the normal vector of the interface. The boundary condition is set as $\frac{\partial \vec{u}}{\partial t}|_{\partial\Omega} = 0$ according the no-slip boundary condition on the mold wall and the constant boundary condition on the inflow and outflow boundary. The Webb number, which is the ratio of the fluid inertia and the surface tension, is about 200 [6]. This rather large value tells us that the surface tension will not significantly influence the velocity field, and is hence ignored in our computation as other authors did [6,7]. The level set method introduces a function ϕ, called level set function, whose physical meaning is the signed distance from the current position to the interface, which implies

$$|\nabla\phi| = 1. \quad (3)$$

Its governing equation is

$$\frac{\partial\phi}{\partial t} + \vec{u}\cdot\nabla\phi = 0 \quad (4)$$

from which we can see that the zero contour of ϕ will follow the interface of the two kinds of fluids. The level set function cannot be kept as distance with the evolution of the governing equation (4). So a so-called "re-initialization" operation is adopted so that Eq. (3) is satisfied.

Re-initialization for a given function ϕ_0 is implemented by solving the following equation to the stable state:

$$\frac{\partial\phi}{\partial\tau} = \mathrm{sign}(\phi_0)(1 - |\nabla\phi|) \quad (5)$$

with the initial condition

$$\phi|_{\tau=0} = \phi_0.$$

Then ρ, μ are dependent on ϕ, by a given heaviside function H as:

$$\rho = \rho_1 + H(\phi)(\rho_2 - \rho_1), \quad (6)$$

$$\mu = \mu_1 + H(\phi)(\mu_2 - \mu_1) \quad (7)$$

in which the subscripts 1 and 2 mean the physical amounts for the two kinds of fluids, respectively. The heaviside function should be a function which increases from 0 to 1 smoothly when the current position moves across the interface, and will be 0 or 1 when ϕ is less than $-\alpha$ or greater than α for a previously given small positive number α. The same heaviside function as in [3] is adopted in our computation such that

$$\rho = \begin{cases} 1 & \text{if } \phi > \alpha, \\ \bar{\rho} + \Delta\rho \sin(\pi\phi/2\alpha) & -\alpha \leqslant \phi \leqslant \alpha, \\ \rho_2/\rho_1 & \text{if } \phi < -\alpha \end{cases}$$

$$\text{in which } \begin{cases} \bar{\rho} = (\rho_1 + \rho_2)/2\rho_1, \\ \Delta\rho = (\rho_1 - \rho_2)/2\rho_1, \end{cases}$$

$$\mu = \begin{cases} 1 & \text{if } \phi > \alpha, \\ \bar{\mu} + \Delta\mu \sin(\pi\phi/2\alpha) & -\alpha \leqslant \phi \leqslant \alpha, \\ \mu_2/\mu_1 & \text{if } \phi < -\alpha \end{cases}$$

$$\text{in which } \begin{cases} \bar{\mu} = (\mu_1 + \mu_2)/2\mu_1, \\ \Delta\rho = (\mu_1 - \mu_2)/2\mu_1. \end{cases}$$

Generally, α is given as $\alpha = \frac{3}{2}dh$ in which dh is the typical element size [3].

The projection method is used for Navier–Stokes equation, which is an often-adopted technique for two-dimensional fluid computation. It can remove the term of the gradient of the pressure from the Navier–Stokes equation, which is a conventional difficulty in fluid dynamics. Though there are other methods that can smooth the same difficulty, such as the vortex-stream function method and SIMPLER method, we did not adopt these two methods because the boundary condition given is of velocity, while vortex-stream function method needs the boundary condition of vortex, and the SIMPLER method is suitable for the staggered difference scheme, while we should work on an un-constructed mesh (see part 3). According to the Hodge–Helmholtz decomposition, every two-dimensional vector field can be divided into two parts, one of which is divergence free, and the other is curl free, and these two parts are orthogonal. At first, the Navier–Stokes equation in two-dimensional case is re-written as

$$\frac{\partial \vec{u}}{\partial t} = -\frac{1}{\rho}\nabla p + L(\vec{u}) \tag{8}$$

in which

$$L(\vec{u}) = -\vec{u}\cdot\nabla\vec{u} + \vec{g} + \frac{1}{Re}\nabla(2\mu D). \tag{9}$$

It can be seen that the left-hand side of (8) is divergence free and the gradient of the pressure is curl free. So the right-hand side of (8) should be the divergence free part of $L(\vec{u})$. By introducing the stream function

$$\vec{\Psi} = (0, 0, \Psi) \tag{10}$$

the projection operator can be expressed as

$$P : L(u) \mapsto P(L(u)) = \nabla \times \vec{\Psi} \tag{11}$$

in which Ψ can be obtained by solving the Poisson equation

$$-\nabla(\rho\nabla\Psi) = \nabla \times (\rho L(\vec{u})) \tag{12}$$

with boundary condition

$$\Psi|_{\partial\Omega} = 0.$$

This boundary condition is obtained from the zero boundary condition for $\partial\vec{u}/\partial t$.

3. Implementation of the level set algorithm

There are plentiful references for level set method. We will concentrate on the finite element method implementation. The finite element method which is used mainly for the computational domain is fairly irregular, and the tangential viscosity plays an important role for the problem at the boundary of the mold so that the zigzag by finite difference method at the boundary is not acceptable. The weak formation of the Navier–Stokes equation, governing equation of ϕ, and an upwind scheme for the re-initialization equation are given.

We triangulate the domain Ω into simplex elements and denote the nodes as X_i, and elements as T_i; we choose the finite element space on the discretized domain $V_h(\vec{u})$, $V_h(\phi)$, $V_h(\Psi)$ for $\vec{u}$, ϕ, Ψ, respectively. In our numerical example, we choose $V_h(\vec{u})$ and $V_h(\Psi)$ as $H^1_h(\Omega)$, and $V_h(\phi)$ as $H^0_h(\Omega)$. Because what the engineers need is the pattern of the flow, we would like to choose lower order finite element space. And for the convenience to construct an upwind scheme required by the re-initialization equation, the level set function is set as piecewise constant.

Then the weak formation of the projection is:

$$\int_\Omega \rho \nabla \Psi \nabla \omega \, \mathrm{d}x = \int_\Omega \nabla \times (\rho L(\vec{u})\omega) \, \mathrm{d}x = -\int_\Omega \rho L(\vec{u}) \cdot (\nabla \times \omega) \, \mathrm{d}x, \quad (13)$$

$$\int_\Omega \frac{\partial \vec{u}}{\partial t} \lambda \, \mathrm{d}x = \int_\Omega \nabla \times \vec{\Psi} \lambda \, \mathrm{d}x \quad (14)$$

in which the test function $\omega \in V_{h,0}(\Psi)$ and $\lambda \in V_{h,0}(\vec{u})$.

The weak formation of Eq. (4) is

$$\int_\Omega \frac{\partial \phi}{\partial t} \chi \, \mathrm{d}x = -\int_\Omega \{\vec{u} \cdot \nabla \pi_h(\phi)\} \chi \, \mathrm{d}x \quad (15)$$

in which $\chi \in V_h(\phi)$, ϕ is a piecewise constant function according to our choice for $V_h(\phi)$ so that Eq. (15) needs no boundary condition and the scheme we obtained from it turns into an explicit one at last. There is an interpolation operator π_h from piecewise constant space to piecewise linear on ϕ so the gradient operator can be applied. We solve the re-initialization equation (5) of the level set function with an upwind scheme as

$$\frac{\partial \phi_i}{\partial \tau} = \mathrm{sign}(\phi_{0,i})(1 - |\phi_i|), \quad (16)$$

$$|\phi_h|_{T_i} = \sum_{j=1}^{3} \max(\mathrm{sign}(\phi_i - \phi_{i_j})\mathrm{sign}(\phi_{0,i}), 0)) \times \frac{|\phi_i - \phi_{i_j}|}{|C_i - C_{i_j}|} \quad (17)$$

in which T_{i_j}, $1 \leqslant j \leqslant 3$ are the neighbor elements of T_i, and C_{i_j}, $1 \leqslant j \leqslant 3$ are its centers. Numerically, $\mathrm{sign}(\phi_{0,i})$ is approximated as $\phi_{0,i}/\sqrt{\phi_{0,i}^2 + \varepsilon^2}$ with a given positive number ε. When the time step length is small enough, scheme (16) and (17) is upwind, and will converge to the solution uniformly with one order accuracy. In our computation, the step length of time is 0.2 times the step length of space and the scheme is converge.

The three-stage Runge–Kutta scheme is adopted in the temporal direction to discretize Eqs. (14) and (15). That is

$$\frac{\vec{u}^{*,1} - \vec{u}^{n}}{\Delta t} = \frac{1}{3} P(L(\vec{u}^{n})),$$
$$\frac{\vec{u}^{*,2} - \vec{u}^{n}}{\Delta t} = \frac{1}{2} P(L(\vec{u}^{*,1})), \quad (18)$$
$$\frac{\vec{u}^{n+1} - \vec{u}^{n}}{\Delta t} = P(L(\vec{u}^{*,2})),$$

and

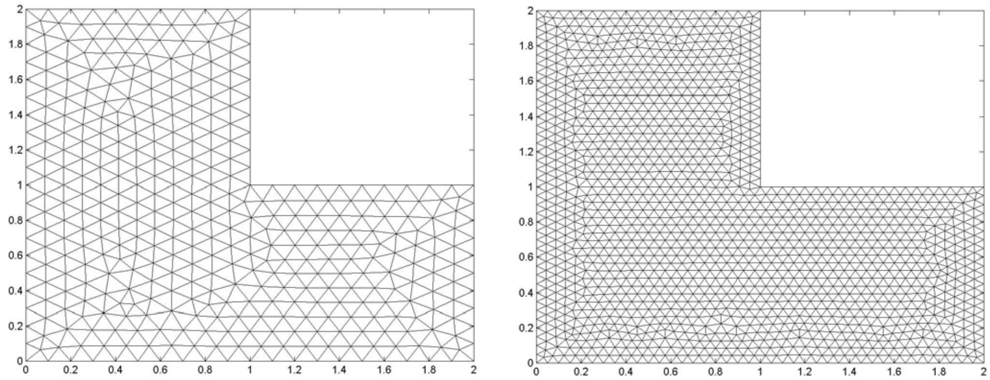

Fig. 1. Meshes in L-shape domain with different mesh densities.

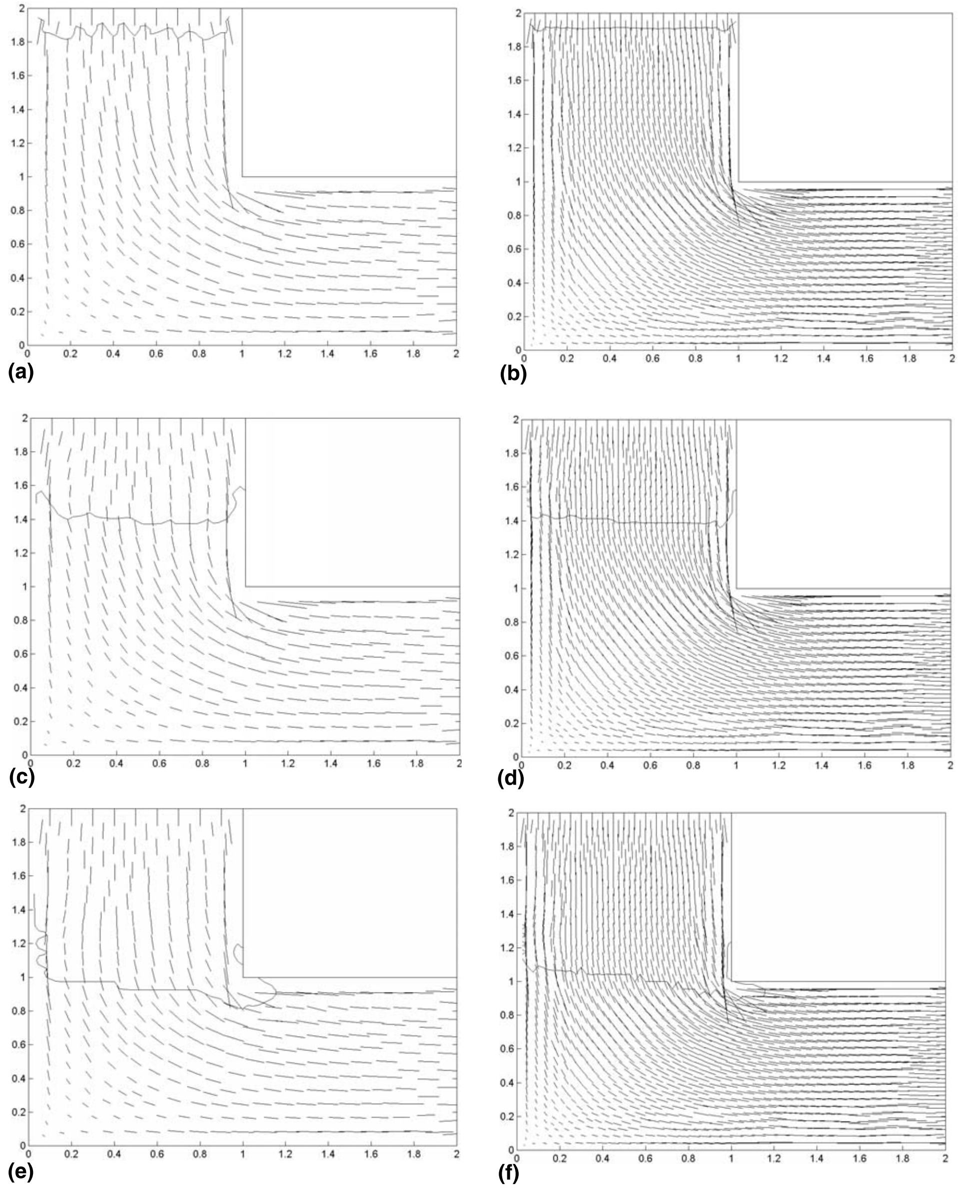

Fig. 2. The velocity field and free interface at several moments.

$$\frac{\phi^{*,1}-\phi^n}{\Delta t}=\frac{1}{3}\vec{u}^n\cdot\nabla\phi^n,$$
$$\frac{\phi^{*,2}-\phi^n}{\Delta t}=\frac{1}{2}\vec{u}^{*,1}\cdot\nabla\phi^{*,1}, \qquad (19)$$
$$\frac{\phi^{n+1}-\phi^n}{\Delta t}=\vec{u}^{*,2}\cdot\nabla\phi^{*,2}$$

in which Δt is set as in [3] as

$$\Delta t_v=\min_{\Omega}\frac{3}{14}\rho Reh^2/\mu,$$
$$\Delta t_c=\min_{\Omega}\frac{h}{|\vec{u}|}, \qquad (20)$$
$$\Delta t=\min(\Delta t_v,\Delta t_c).$$

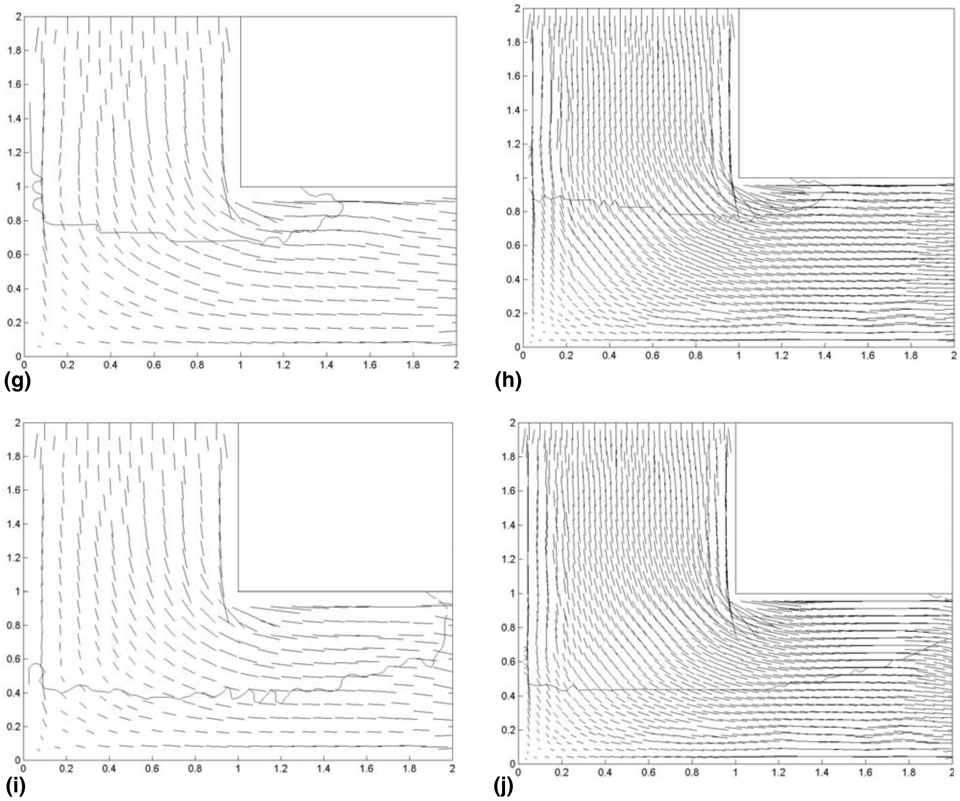

Fig. 2. (Continued).

High order Runge–Kutta scheme is nearly A-stable. This is the reason why we use it. In Eqs. (18) and (19), the projection is operated on the right-hand side term only, then the continuity equation (2) will not be satisfied after a period of time because of the accumulation of the residual error of the projection. We introduce a correction procedure when this happens. Similar to the projection method, we use the vortex-stream function method to implement the procedure. The vortex function $\zeta = \nabla \times \vec{u}$, and the stream function Ψ satisfy

$$-\nabla(\rho\nabla\Psi) = \zeta \qquad (21)$$

with the Neumann boundary condition

$$\nabla \times \Psi|_{\partial\Omega} = \vec{u}|_{\partial\Omega}.$$

Then the velocity can be restored with $\vec{u} = \nabla \times \Psi$. This procedure can be implemented after one or several step forward operations.

4. Numerical convergence study and numerical test with physical data

We give a numerical example in a very simple L-shape domain, and compute on different meshes. Then the meshes are as in Fig. 1. The inflow speed is set as 1, and outflow speed is 1 too. The inlet boundary is the top boundary, and outlet boundary is the right down boundary. The gravity is 980. The velocity field and the free interface are in Fig. 2. From the numerical results, we can see that the method is really grid convergent.

The computation domain is triangulated as in Fig. 3. This mold is often used as the benchmark experiment [6] for the mold filling. Though the mold is far from the shape of practical interest, the roll casting process, it is chosen as a numerical test domain because a far more complex flow pattern is expected to appear in such geometric region, thus

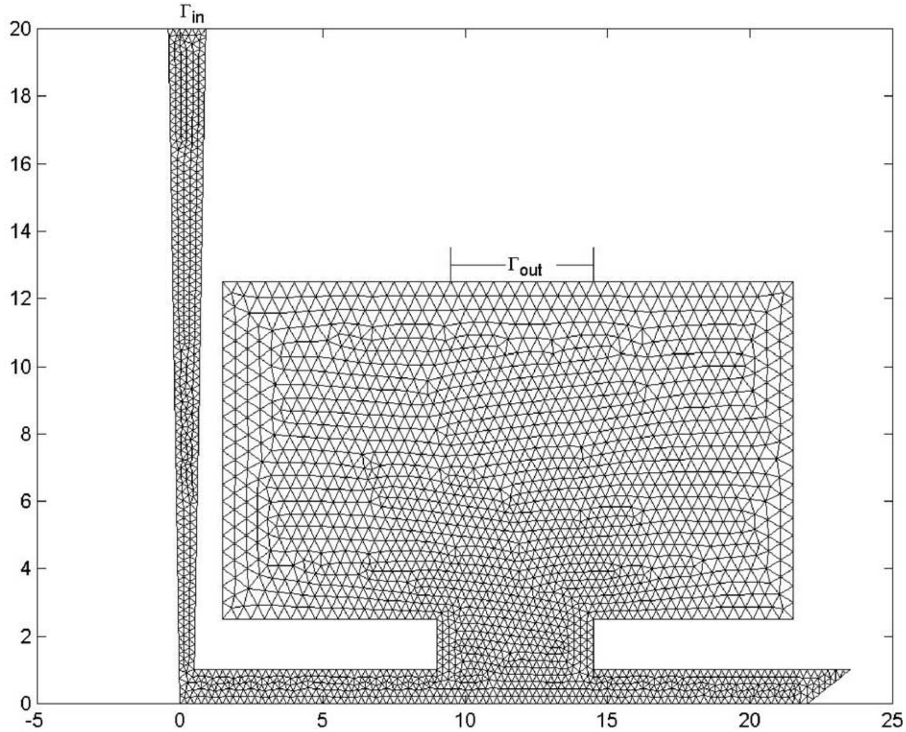

Fig. 3. The physical domain of computation.

constituting a good test of the level set simulation approach.

The velocity of inlet is 5 m/s, and gravity is 9.8 m/s^2. The boundary condition is

$$\vec{u} = \begin{cases} (0, -500)^T & \text{on } T_{in}, \\ (0, 130)^T & \text{on } T_{out}, \\ (0, 0)^T & \text{elsewhere.} \end{cases} \quad (22)$$

Other data are generally as following:

$$\rho_1 = 2500 \text{ kg/m}^3,$$
$$\rho_2 = 2569 \text{ kg/m}^3 \text{ s},$$
$$\mu_1 = 1.0 \times 10^4 \text{ kg/m s},$$
$$\mu_2 = 1.2 \times 10^4 \text{ kg/m s},$$
$$Re = 100.$$

The height of the mold is 200 mm, and the width of the grating is 13 mm. It means the flow in the mold is like a typhoon in our common scale with a speed of about 100 m/s. For simplification, the heat transfer between metal and mold is not considered, which means the solidified shell is assumed to be zero. After the mold is filled with the liquid metal, then the second liquid metal begins to pour, and first kind of liquid metal will be displaced, and overflow along the outlet.

5. Results and discussion

Level set algorithm is used for a check of the processing parameters of the liquid displacement process in the mold. Figs. 4–8 illustrate the evolution of the free interface, and the velocity changes under gravity.

The interface shown in Fig. 4 is seen to be stable under such parameters as density ratio 1:1.025, and inlet velocity 5 m/s. Most of the first liquid metal will be flushed away, which is the case expected by the engineer. The interface in Fig. 5 is not so stable; when the density ratio was increases to 1:1.02625, the two liquid metals are mixed, and

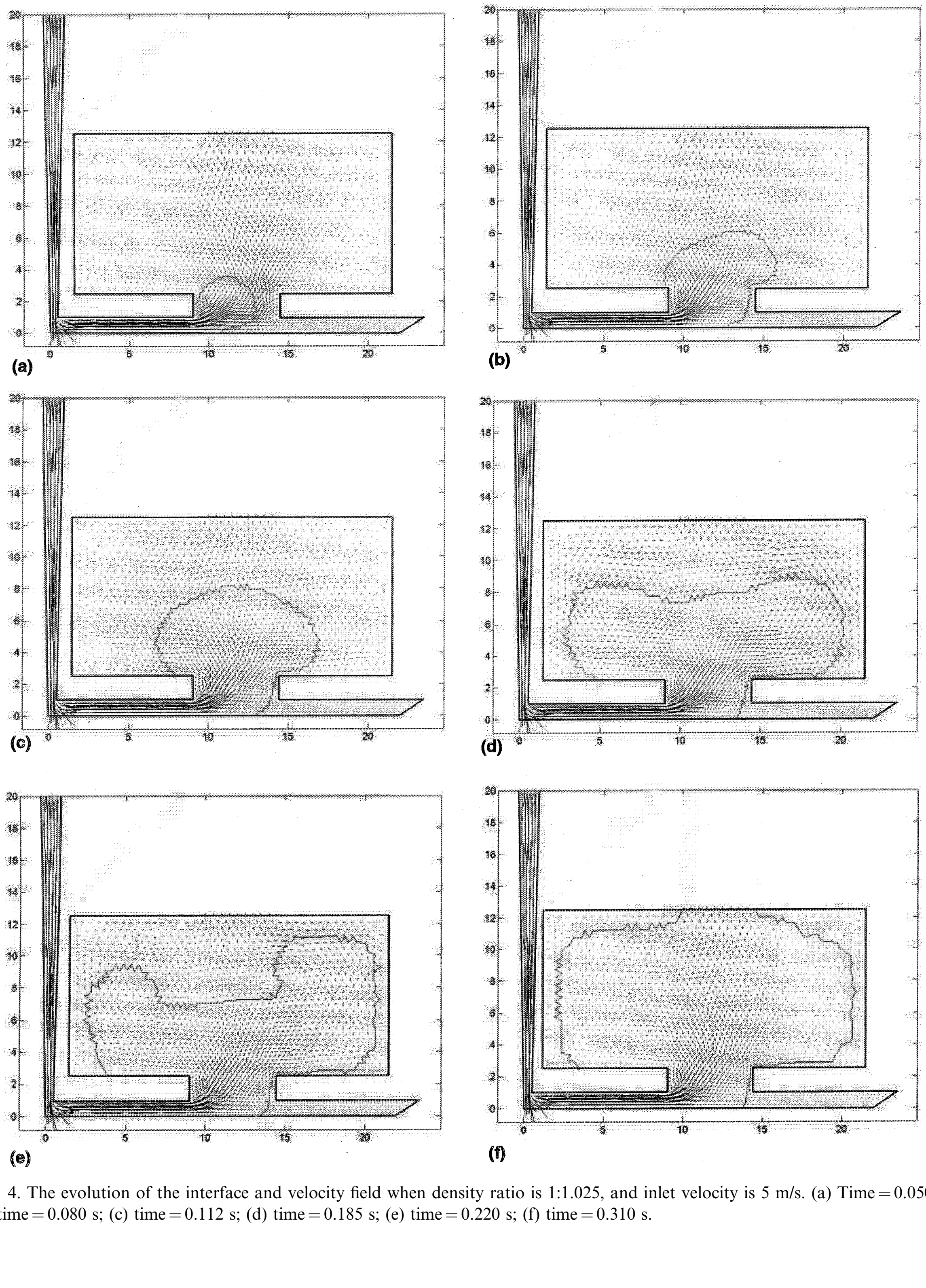

Fig. 4. The evolution of the interface and velocity field when density ratio is 1:1.025, and inlet velocity is 5 m/s. (a) Time = 0.050 s; (b) time = 0.080 s; (c) time = 0.112 s; (d) time = 0.185 s; (e) time = 0.220 s; (f) time = 0.310 s.

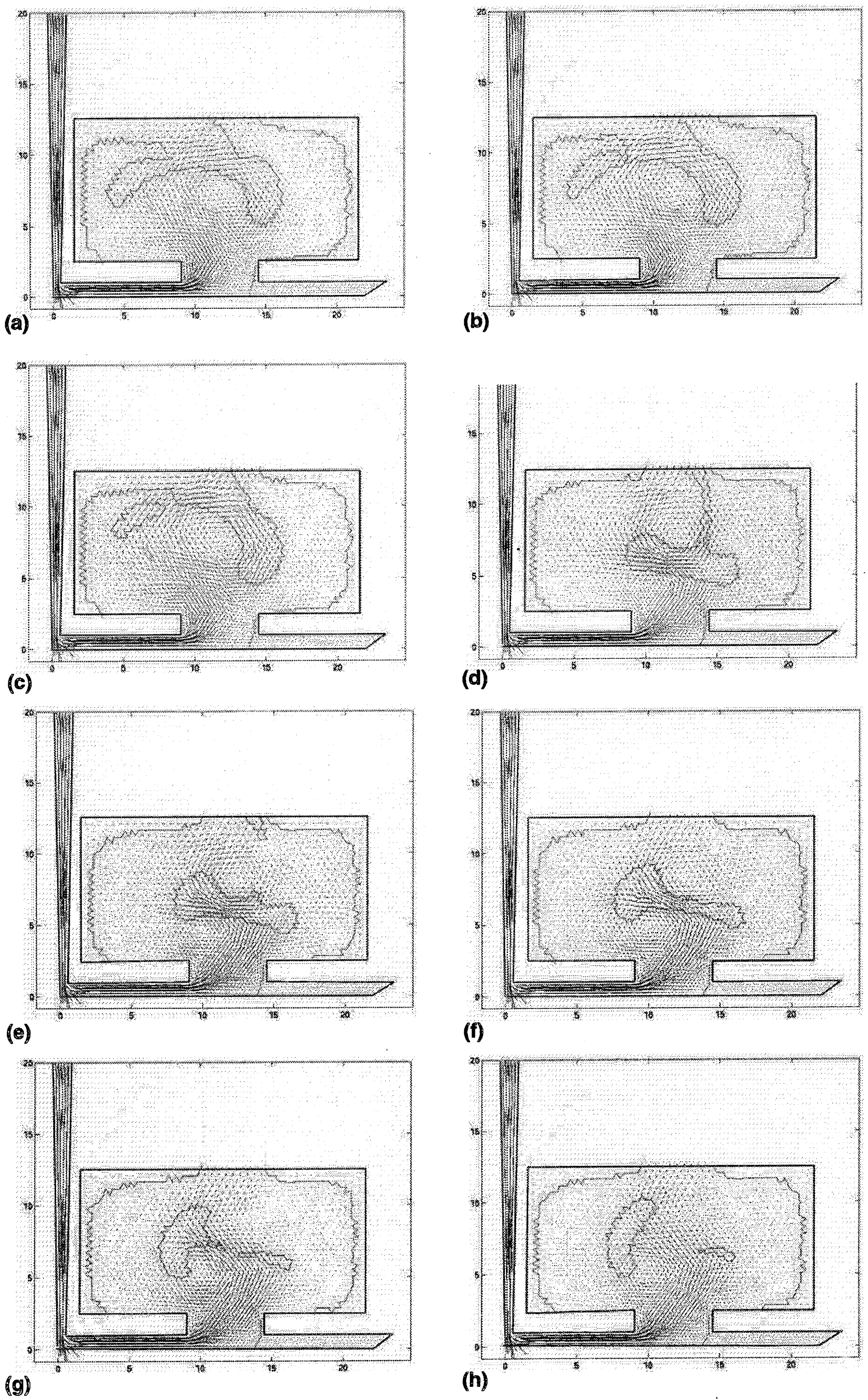

Fig. 5. Topological change of the interface occurred when the density ratio is 1:1.02625, and the inlet velocity is 5 m/s. This density ratio is very close up to the stability threshold for such inlet velocity. (a) Time = 0.276 s; (b) time = 0.279 s; (c) time = 0.282 s; (d) time = 0.297 s; (e) time = 0.300 s; (f) time = 0.303 s; (g) time = 0.309 s; (h) time = 0.312 s.

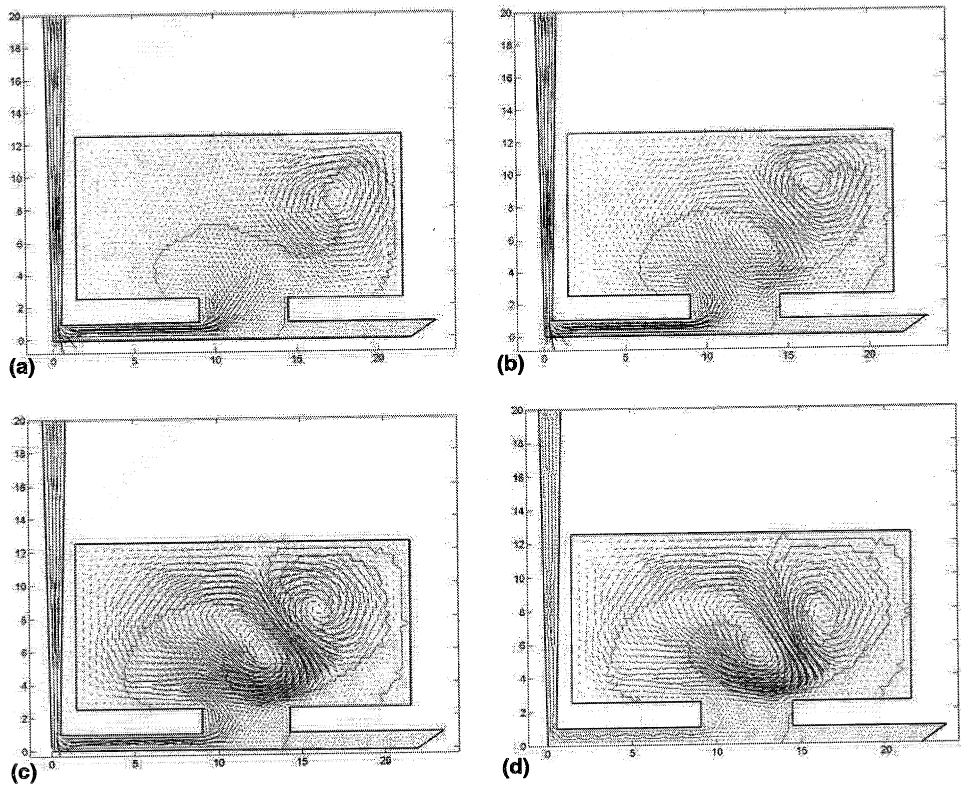

Fig. 6. The interface evolves unstably when the density ratio is 1:1.05, and the inlet velocity is 5 m/s. (a) Time = 0.140 s; (b) time = 0.146 s; (c) time = 0.152 s; (d) time = 0.155 s.

complex topological changes of the interface occur. The first liquid metal is seen to contaminate the second liquid metal. In Fig. 6, when the density ratio increases to 1:1.05, the second liquid metal rushes to the right wall of the mold. The interface breaks down and the violent mixing of the fluids occurs, destroying the possibility of a separate and distinct core and surface alloy regions. In Fig. 7, the density ratio is comparably large at 1:1.15. When the second liquid metal enters the rectangle volume of the mold, the velocity field changes dramatically, causing the splash of the second fluid into the first. In Fig. 8, the inlet velocity is decreased to a low value, 0.1 m/s. With such a low inlet velocity the fluid is far more stable. Gravity plays an important role, conferring stability on the interface so that the second liquid metal spreads out quietly in the bottom of the rectangular part of the mold. This implies that, if productivity and solidification time allow, the inlet velocity should be set at a low value. In practice casting, a comparative lower casting temperature is needed to avoid complex solidification pattern. Such consideration makes engineers preferring a fairly high inlet velocity to prevent premature solidification. Then with the guarantee to prevent premature solidification, lower inlet velocity is our advice to the engineers. From the figures above, it is seen that though the metal above is lighter than that below, the surface is not always stable. There is a threshold of density ratio for the surface to be stable at about 1:1.0275 for that particular mold and inlet velocity, 5 m/s. The fact of the existence of a threshold is new finding that it is hoped to explain many of the previously baffling features of attempt at double casting.

The threshold is sensitive to the inflow velocity more than on the viscosity, etc. In the calculation,

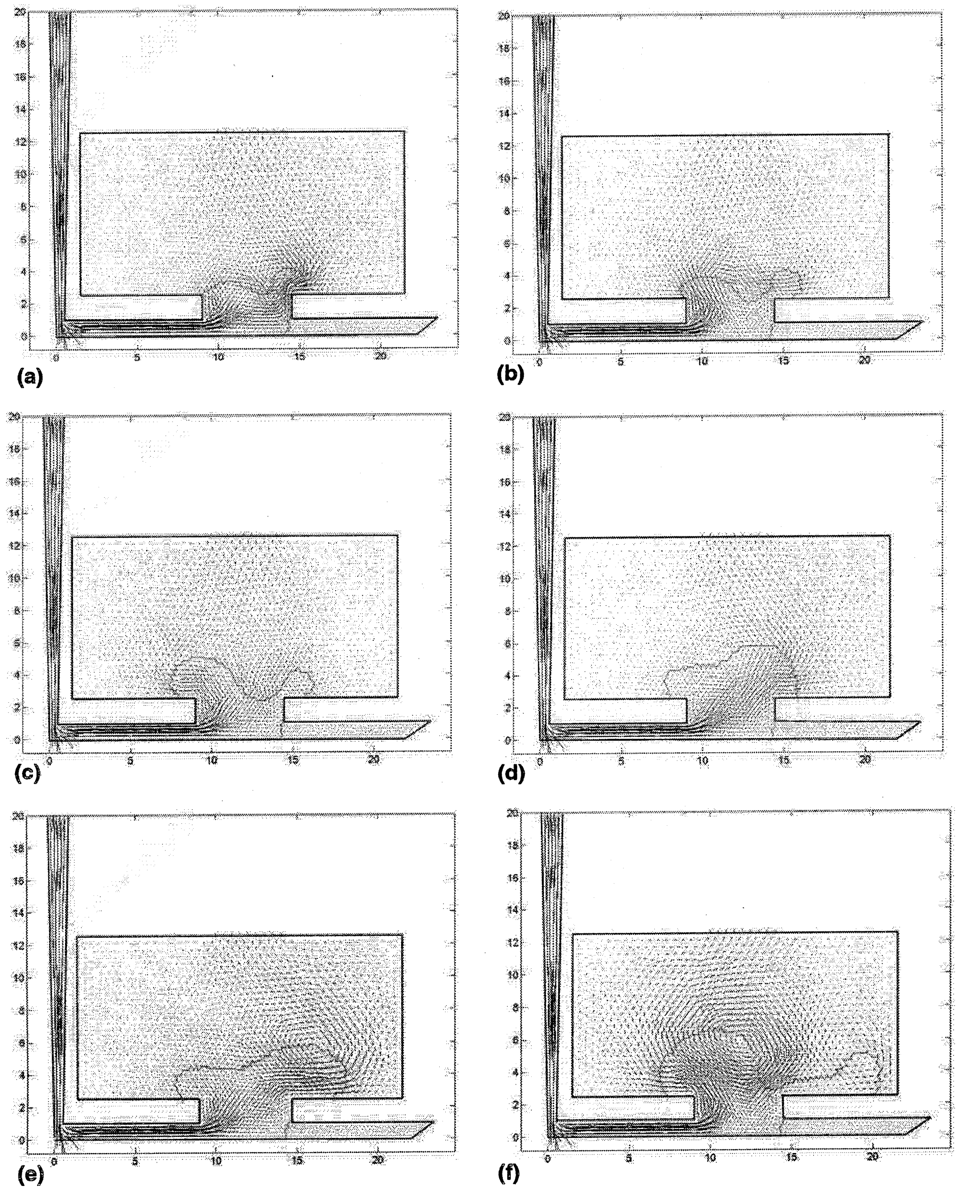

Fig. 7. The flow field wash away from right to left like a faucet when the density ratio is 1:1.15, and the inlet velocity is 5 m/s. (a) Time = 0.058 s; (b) time = 0.061 s; (c) time = 0.067 s; (d) time = 0.076 s; (e) time = 0.082 s; (f) time = 0.091 s.

viscosity ratio of 1:1, 1:10, and 1:100 with Reynolds number 100, and 1000, respectively, has been checked, the velocity field and interface stability has little change. This shows that viscosity and its ratio plays little role in determining the interface stability.

6. Conclusion

Level set algorithm has been used for the first time in simulating double casting technique. It seems now possible to give some insight into the design of the mold filling technique and choice of

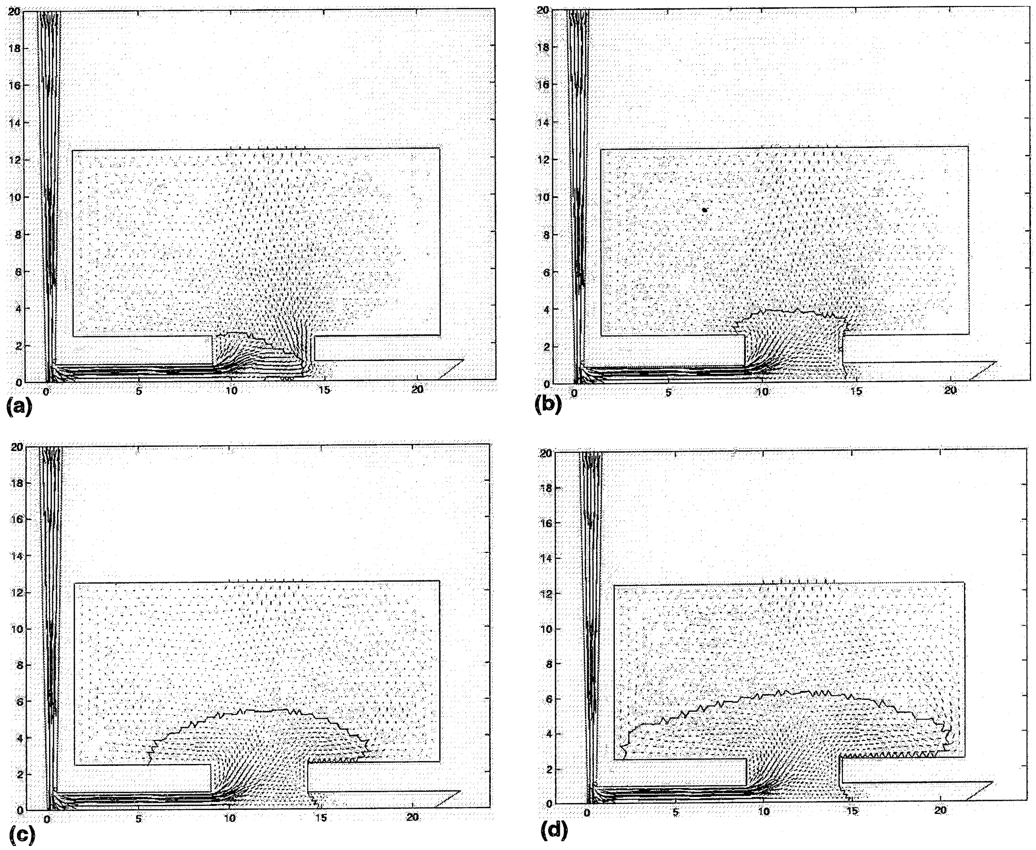

Fig. 8. The interface and velocity field evolution when density ratio is 1:1.05, and the inlet velocity is 0.1 m/s. (a) Time = 2.999 s; (b) time = 3.919 s; (c) time = 5.999 s; (d) time = 9.498 s.

processing parameters. This method is preferable to VOF and MAC method in tracking the interface evolution of two phases. In this paper, a benchmark numerical test is adopted to visualize the mold filling process. The following initial results are drawn:

1. Level set algorithm is suitable method to deal with two-phase flow. The flushing procedure can be given in detail by the algorithm. Because the fluid fields are combined into one whole, the work of programming is less, while the code is more efficient. By adapting the width of the transition layer, a numerical result can be obtained with more stable fluid field or with less mass exchange and higher resolution.
2. It is found that for the flushing through a mold, density ratio is the critical parameter for the forming of a stable interface, and viscosity is unimportant. With the same density ratio, when inlet velocity increases, an unstable interface will form, and the interface will break down.
3. Further research work is planned to apply the level set method to the practical process of cast roll, and the energy conservation equation, coupled with Navier–Stokes equations, will be solved.

7. Uncited references

[4,5].

Acknowledgements

Many Thanks to Prof. John Campbell from Inter-disciplinary Research Center (IRC) of the University of Birmingham for good advice.

References

[1] B.D. Nichols, C.W. Hirt, R.S. Hotchkiss, SOLA-VOF: a solution algorithm for transient fluid flow with multiple free boundaries, Los Alamos National Laboratory Report, 1980.

[2] D.B. Kothe, R.C. Mjolsness, M.D. Forre, RIPPLE: a computer program for incompressible flows with free surface, Los Alamos National Laboratory Report, 1997.

[3] M. Sussman, P. Smereka, S. Osher, A level set approach for computing solutions to incompressible two-phase flow, J. Comput. Phys. 114, pp. 146–159.

[4] V. Girault, P. Raviart, Finite Element Methods for Navier–Stokes Equations, Springer, New York, 1986.

[5] A. Baker, Finite element computational fluid mechanics, Series in Computational Methods in Mechanics and Thermal Science.

[6] M. Barkhudarov, K. Williams, Simulation of "surface turbulence" fluid phenomena during mold filling, AFS Trans. (1995) 669–674.

[7] J. Zhu, J. Sethian, Projection methods coupled to level set interface techniques, J. Comput. Phys. 1 (102) (1992) 128–138.