Numerical Investigation of Liquid Flows in Parallel Columns for Use in the Al or Mg Die-Casting

Yong Kweon Suh *, Zi Lu Li and Jong Hyun Jeong

Department of Mechanical Engineering, Dong-A University,

840 Hadan-dong, Saha-gu, Busan 604-714, Korea

Following the recent trend in the automotive manufacturing technologies, the product design subject to the die casting becomes more and more complex. The requirement of the die design becomes more demanding than ever before. In some cases the product's shape can have multiple slender manifolds. In such cases, design of the inlet and outlet parts of the die is very important in the whole manufacturing process. The main issues required for the qualified products are to attain gentle and uniform flow of the molten liquid within the passages of the die. To satisfy such issues, the inlet cylinder ('bed cylinder' in this paper) must be as large as possible and simultaneously the outlet opening at the end of each passage must be as small as possible. However these in turn obviously bring additional manufacturing costs caused by re-melting of the bed cylinder and increased power due to the small outlet-openings. The purpose of this paper is to develop effective simulation methods of calculation for fluid flows in multiple columns, which mimic the actual complex design, and to get some useful information which can give some contributions to the die-casting industry. We have used a commercial code CFX in the numerical simulation. The primary parameter involved is the size of the air-vent. We will show how the very small opening of the outlet, i.e. the air-vent, can be treated with the aid of the porous model provided in the code. To check the validity of the numerical results we have also conducted a simple experiment by using water.

Key Words: Multiple Columns, Water Flows, Porous Model, Thin Outlet

1. Introduction

High-pressure die casting is an effective process in the manufacturing of high-volume and low-cost automotive components such as automatic transmission housings and gear box components etc. Liquid metal (generally magnesium or aluminum) is injected into the die chamber at a high speed and fills in the inner room of the die immediately. The two representative casting technologies, thixocasting and rheocasting, use high-pressure to increase the production rate (McLaughlin and Kim, 2003).

The geometric complexity of the die chamber leads to strongly three-dimensional fluid flow with significant free surface fragmentation and splashing. The order in which the various parts of the die are filled with liquid metal and the locations as well as the area of the air-vents are crucial to forming homogeneous cast products with minimal voids entrapped. This is influenced by the design of the gating system and the geometry of the die (Han and Xu, 2005). Numerical simulation offers a powerful and cost-effective way to

---

*Corresponding Author,
E-mail :yksuh@dau.ac.kr
TEL : (51)200-7648; **FAX** : (51)200-7656
Department of Mechanical Engineering, Dong-A University,
840, Hadan-dong, Saha-gu, Busan 604-714, Korea

estimate the effect of the die design and filling processes, ultimately leading to improvement in both product quality and process productivity via more effective control of the die filling and die thermal performance (Cleary et al, 2004). It is also necessary to predict the design parameters such as the friction effect or the pressure drop and so on (Jamialahmadi et al., 2005).

Our target is to investigate the fill-in process with a simple model which is composed of 3 vertical long columns attached to a horizontal bed cylinder (Fig. 1). Inlet is at one side of the bed cylinder, and each of three vertical parallel columns has an air-vent at the outlet. This is a typical structure found in symmetrical or non-symmetrical casting model, although in reality the shape of the columns may be more complex. Hot liquid metal in principle should fill the fluid space gently and uniformly, but depending on the die design it may not be satisfied. Such uniform and gentle behavior of the fluid fill-in is assumed to give a contribution to improving the mechanical properties as well as the micro-structure of the final products. In order to attain such uniform and gentle fill-in, the air-vent should be as small as possible and the bed cylinder should be as large as possible. Besides, the inlet speed of the molten metal should be as small as possible. However, all the above requirements are contradictory to those necessary for the increase of the production rate. So, as for the die design, we must compromise between the quality and the quantity of the die-casting process. Thus in this paper we investigate the effect of parameter, i.e. the air-vent size, on the flow pattern within the vertical columns mounted on the horizontal bed cylinder, which has a direct relationship with the products' quality as well as the manufacturing cost in the die-casting process.

2. Experimental Set-up and Methods

Figure 1 shows a simple experimental set-up built for flow visualization, and the purpose of this experiment is to check if the numerical results will be reliable.

We inject water into the left-hand side of the inlet cylinder to push the piston that in turn drives the water into the bed cylinder and then into the three columns. We fast open the valve to start this experiment. A high-speed camera located in front of the piston captures the piston movement and a digital video camera captures the free-surface rising in the columns.

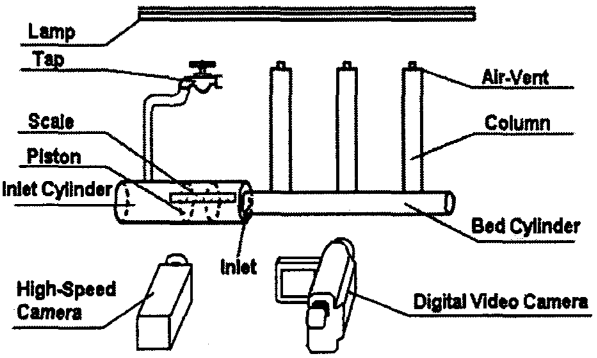


Fig. 1 Experimental set-up

The experimental conditions are; the bed-cylinder is 20mm in diameter and 300mm long, each of the vertical columns is 10mm in diameter and 300mm long, the air-vent diameter is 1mm, the outlet pressure is 0Pa (gauge pressure) and the initial free surface height is 15mm measured from the centerline of the bed cylinder

Velocity of the piston measured after the beginning of the experiment is typically as shown in Fig. 2. The fluctuation of the piston speed seems to be caused by the elastic property of the air in the columns, inertial effect of the liquid mass and resistance of the solid surface contacting the liquid.

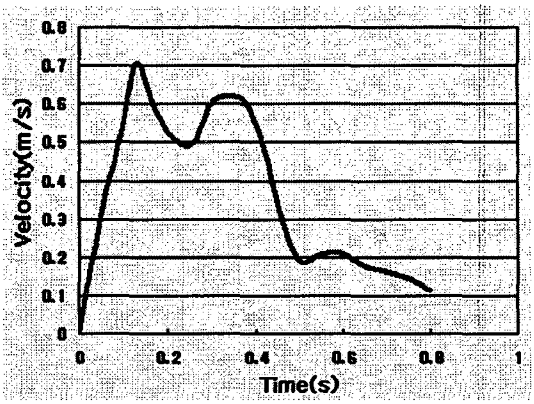


Fig. 2 A typical piston velocity

In the experiment, the water-level height in each column is monitored by a high-speed camera. Due to the difference in the flow resistance of the fluid in each column or due to the inertial motion of the fluid mass, the fluid fronts are obviously not the same with each other.

3. Numerical Method

We used the commercial package CFX for the numerical simulation and selected a two-phase model which can treat the motion of the interface between the water and the air. Our first aim is to justify whether the code can calculate the high-speed flow in a reasonable accuracy. Our model is composed of the horizontal bed cylinder with three long vertical columns as explained in the previous section. The bed cylinder is 20mm in diameter and 300mm long. Fluid is injected from the inlet. Diameter and length of the three columns are 10mm and 300mm, respectively. The air-vent diameter in each column is 1mm. Instead of the hot molten metal, water is used as the working fluid in this study to follow the experimental conditions. We use k-ε turbulent flow model and VOF (volume of fluid; Hirt and Nichols, 1981) method to treat the interface motion (See Fig. 3).

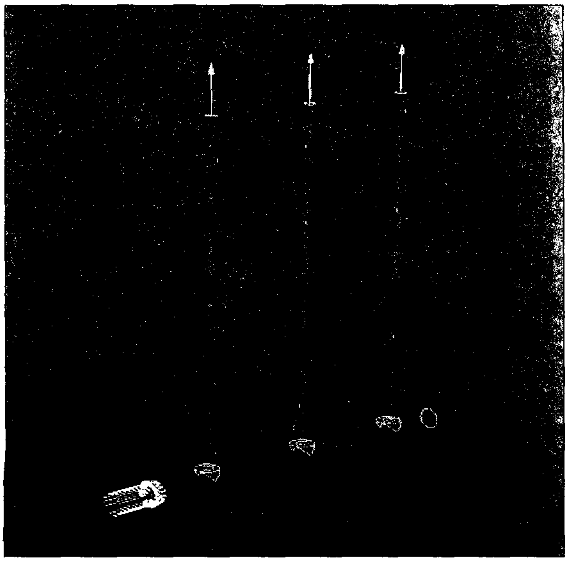
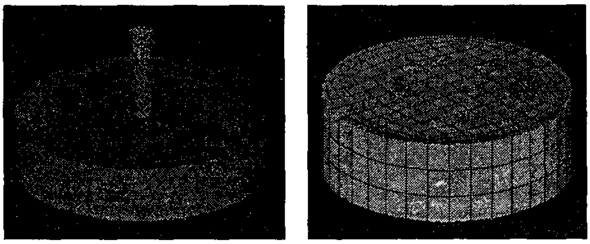


Fig. 3 Simulation model for the three-column structure

In our calculation, however, we have found that it took a long time due to the clustered grids near the air-vent that has a very small opening. To save the computational time, therefore, we designed a porous part in lieu of the air-vent (Fig. 4). Compared with the small air-vent, the porous part can yield normal-size grids and simultaneously it can supply an equivalent resistance. So it is expected to give the same effect as the real structure. This method has been justified by applying it to a simpler, L-pipe flow model. Fig. 4 shows meshes for the original air-vent structure and the porous structure.



Real Model Porous Model

Fig. 4 Comparison of two different outlet (air-vent) models

Table 1 compares those two structures in terms of the grid type and the number of grids built, and it surely reveals that the porous-part model necessitates grids far less than the real air-vent. Thus evidently a larger time step can be employed in CFD with the porous-part model, which can indeed save the calculation time.

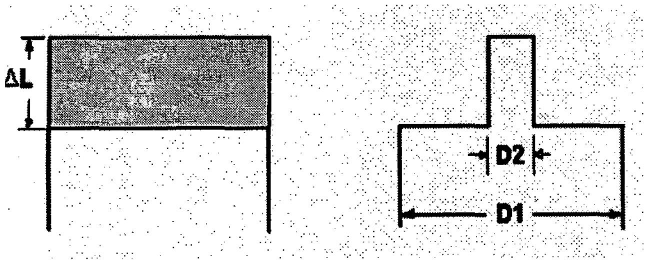
Table 1. Grid configuration for real and porous parts

Outlet Model	Grid Type	Number of Elements	Number of Nodes
Real	Tetrahedral	2231	7007
Porous	Hexahedral	832	1133

To use the porous model, however, we need to define the resistance coefficient K in CFX. Fig. 5 reveals some notations used in derivation of K formula. Here, K is defined as

$$\frac{\Delta P}{\Delta L} = KU_1^2, \qquad (1)$$

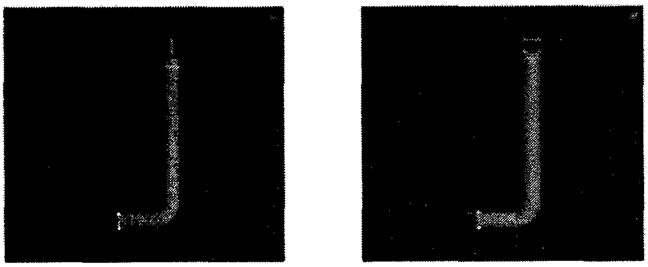
where ΔP is the pressure drop across the porous part with length ΔL and U_1 is the fluid velocity through the porous region(Fig. 5).



Porous Part Real Part

Fig. 5 Notations for derivation of K formula

In formula (1), parameter K is unknown. We employ a single pipe model as shown in Fig. 6 which is named L-pipe model to estimate the K value, and we will see how this parameter is changed depending on the flow velocity and the ratio of the air-vent diameter and the main pipe diameter.



Real Model Porous Model

Fig. 6 L-pipe model

In the L-pipe model calculation the diameter of the main pipe is 10[mm], the total height is 155[mm] and the length of the air-vent at the upper end of the pipe is 3[mm]. We get ΔP value from the calculation for the real part model (left-hand side of Fig. 6), and then compute the K value in order to apply it to the porous part model (right-hand side of Fig. 6). We have done a series of numerical computations by changing the inlet velocity and the area of the air-vent. We use 1[m/s], 1.33[m/s], 1.5[m/s] and 2[m/s] as the inlet velocity, and we change the air-vent diameter with 1[mm], 1.5[mm] and 2.5[mm].

When we apply the Bernoulli equation, we can show that the dimensional constant K is given by

$$K=\frac{\rho h}{2\Delta L}\left(1-\left(\frac{D_2}{D_1}\right)^2\right)\left(\frac{D_1}{D_2}\right)^4 \qquad (2)$$

where h is an empirical dimensionless constant. It is supposed that the parameter h is not so much changed depending on the diameter of the air-vent or the fluid velocity through the passage. Table 2 shows the results of ΔP, K and h value for three ratios D2/D1 and three inlet velocities.

Table 2 Numerical Results of L-pipe Model

(a) D2/D1=10%

$U1\,[m/s]$	1.33	1.5	2
$\Delta P\,[Pa]$	19000	24200	42800
K $[kg/m^4]$	3.57×10^6	3.59×10^6	3.57×10^6
h	1.82	1.83	1.82

(b) D2/D1=15%

$U1\,[m/s]$	1	1.5	2
$\Delta P\,[Pa]$	2300	5160	9190
K $[kg/m^4]$	7.66×10^5	7.64×10^5	7.65×10^5
h	1.97	1.96	1.97

(c) D2/D1=25%

$U1\,[m/s]$	1	1.5	2
$\Delta P\,[Pa]$	270	602	1070
K $[kg/m^4]$	9.00×10^4	8.92×10^4	8.90×10^4
h	1.90	1.88	1.88

In table 2, we can find that h number is almost the same in every case. Compare with the other two cases, h number in 15% diameter ratio is a little larger. We will find the reason in our further work and investigate how does h change during changing the other parameters.

Figure 7 compares the air pressure within the column given from the real air-vent model and the porous model, where in the latter case K is chosen as 766000$[kg/m^4]$ which comes from Table 2(b) for the case with D2/D1=15% and U1= 1[m/s]. Two results are in a very good agreement with each other.

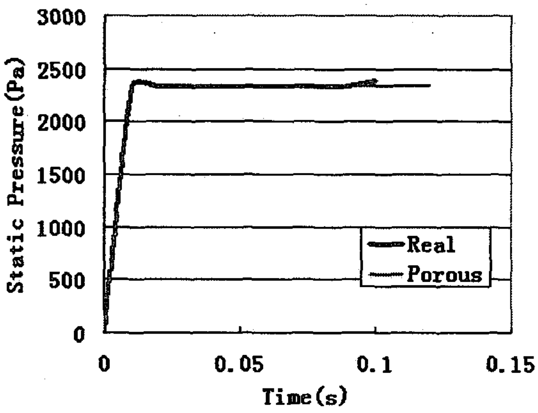


Fig. 7 Variation of the air pressure within the column of the L-pipe model obtained from the real and porous models with K=766000 in 15% diameter ratio and 1m/s inlet velocity.

4. Results and Discussions

Figure 8 shows the mesh configuration in the numerical computation of three-column model. The number of nodes for this configuration is roughly 120,000.

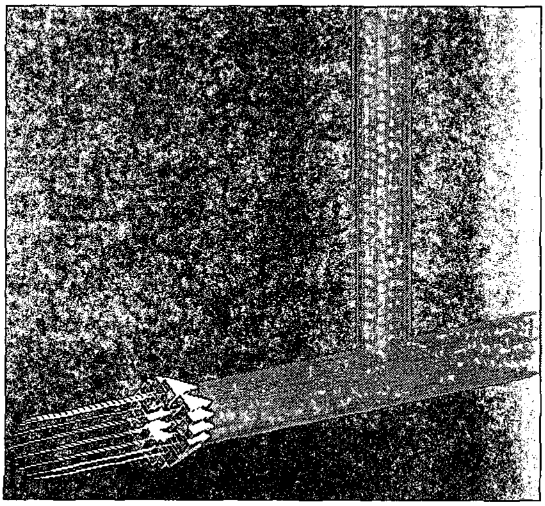


Fig. 8 Mesh configuration

Figure 9 shows the comparison of the free-surface levels obtained from the visualization experiment and numerical simulation. The free surface in the left column rises the fastest at the beginning and then after 0.3s the middle column has the highest level. In overall the experimental results agree well with the numerical results indicating that the numerical methods are reliable.

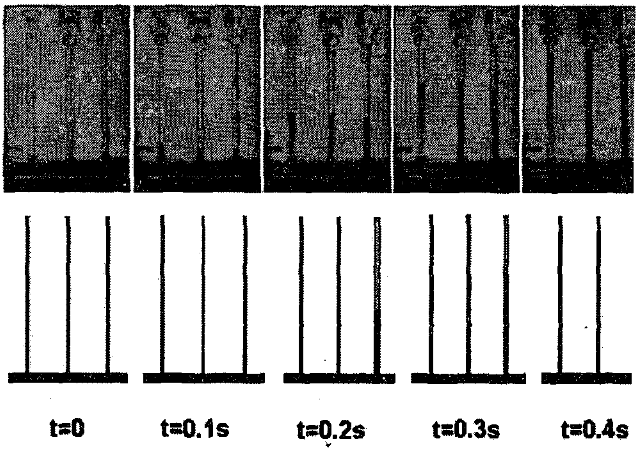


Fig. 9 Development of the free-surface level given from the experiment (top) and computation (bottom)

5. Conclusions

(1) We have successfully developed numerical methods that can compute the details of the flow field in the multiple columns for use in the die casting.

(2) The porous model used as the outlet of each column necessitates far less grids near the outlet compared with the original thin air-vent, and thus it permits a larger time step leading to saving of the computational time.

(3) We have assured the validity of the numerical results by comparing those with the results of a simple flow-visualization experiment.

(4) A very simple analysis has been performed as to the development of the inner air pressure that is directly connected with the air-vent size of the die chamber.

Acknowledgment

This work was supported by grant No. RTI04-01-03 from the Regional Technology Innovation Program of the Ministry of Commerce, Industry and Energy (MOCIE).

References

Cleary, P., Prakash, M., Ha, J., Sinnott, M., Nguyen, T. and Grandfield, J., 2004, "Modeling of cast systems using smoothed-particle hydrodynamics", http://www.tms.org/pubs/journals /JOM/ 0403/Cleary-0403.html.

Han, Q.Y. and Xu, H.B., 2005, "Fluidity of alloys under high pressure die casting conditions", *Scripta Materialia*, Vol. 53, No.1, pp. 7~10.

Hirt, C. W. and Nichols, B. D., 1981, "Volume of fluid (VOF) method for the dynamics of free boundary", *J. Comput Phys*, Vol. 39, pp. 201-225.

Jamialahmadi, M., Müller-Steinhagen, H. and Izadpanah, M.R., 2005, "Pressure drop gas hold-up and heat transfer during single and two-phase flow through porous media", *Int. J. Heat Fluid Flow* Vol. 26, No.1, pp. 156-172.

McLaughlin, M. and Kim, C.W., 2003, "The importance of trapped gas analysis in magnesium filling simulations", http://www.ekkinc.com/dce2003 /DCEarticle2003 final.html