Journal of Mechanical Science and Technology (KSME Int. J.), Vol. 20, No. 10, pp. 1765~1772, 2006 **1765**

Simulation of Water Flows in Multiple Columns with Small Outlets

Yong Kweon Suh*, Zi Lu Li, Jong Hyun Jeong
Department of Mechanical Engineering, Dong-A University, 840 Hadan-dong, Saha-gu, Busan 604-714, Korea
Jun Hee Lee
Department of Metallurgical Engineering, Dong-A University, 840 Hadan-dong, Saha-gu, Busan 604-714, Korea

High-pressure die casting such as thixocasting and rheocasting is an effective process in the manufacturing automotive parts. Following the recent trend in the automotive manufacturing technologies, the product design subject to the die casting becomes more and more complex. Simultaneously the injection speed is also designed to be very high to establish a short cycle-time. Thus, 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 bed cylinder. We will show how the very small opening of the outlet 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

* Corresponding Author,
E-mail : yksuh@dau.ac.kr
TEL : +82-51-200-7648; **FAX** : +82-51-200-7656
Department of Mechanical Engineering, Dong-A University, 840 Hadan-dong, Saha-gu, Busan 604-714, Korea. (Manuscript **Received** August 26, 2005; **Revised** July 6, 2006)

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 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).

In this paper, we 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 bed-cylinder 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 the experimental set-up built for flow visualization. We choose water as the working fluid to mimic the real die-casting process, 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.

The experimental conditions are; the bed-cylinder is 20 mm in diameter and 300 mm long, each of the vertical columns is 10 mm in diameter and 300 mm long, the air-vent diameter is 1 mm,

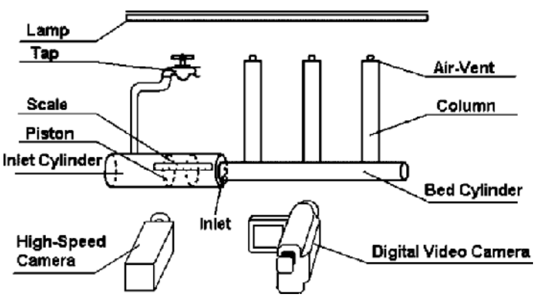


Fig. 1 Experimental set-up

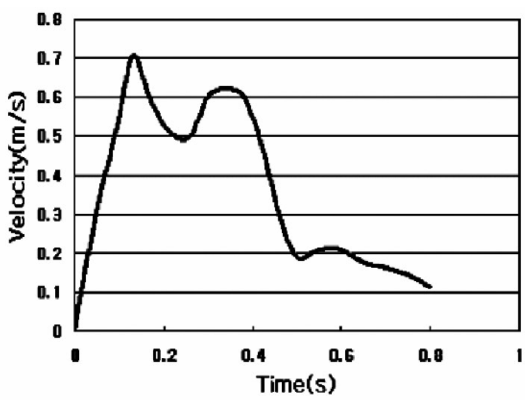


Fig. 2 A typical piston-velocity variation

the outlet pressure is 0 Pa (gauge pressure) and the initial free surface height is 15 mm measured from the centerline of the bed cylinder.

Velocity of the piston measured by the high speed camera 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.

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 20 mm in diameter and 300 mm long. Fluid is injected from the inlet. Diameter and length of the three columns are 10 mm and 300 mm, respectively. The air-vent diameter in each column is 1 mm. Instead of the hot molten metal, water is used as the working fluid in this study to follow the experimental conditions. It is true that the molten metals have changeable viscosity due to temperature. Therefore it must be considered in the numerics for more practical purposes. However, this study aims at the fundamental understanding of the liquid flow while filling in the three column cylinders connected to a common bed cylinder, in particular the friction effect on the fill-in process. We expect that at high Reynolds numbers, as is the case of the present study, the effect of the viscosity on the overall flow friction may be less depend on Re than the case with lower Reynolds numbers.

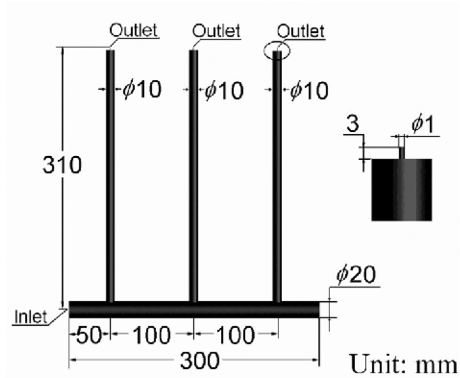


Fig. 3 Simulation model for the three-column structure

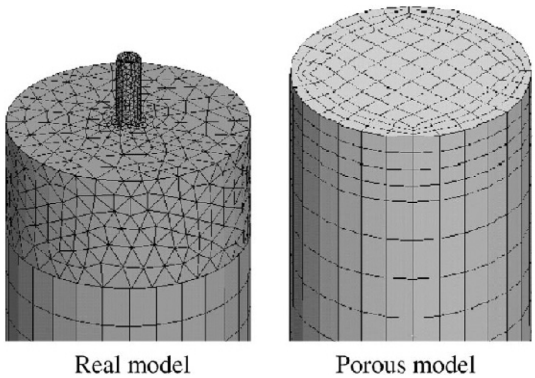


Fig. 4 Comparison of two different outlet (air-vent) models

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).

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.

Table 1 compares those two structures in terms of the grid type and the number of grids built,

Table 1 Grid configuration for real and porous parts

Outlet model	Grid type	Number of elements	Number of nodes
Real	Tetrahedral	5993	1837
Porous	Hexahedral	428	288

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. We checked the time saving by performing calculations of our code for the L-pipe model of Fig. 6. We set the same time step (0.001s) for both cases, and for the real case it took 13000 s and for the porous 9800 s ; we can save 33% run time. We believe that the saving would be increased if the length of the outlet is larger than the present model, 3 mm.

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

$$\frac{\Delta p}{\Delta L}=KU_1^2, \tag{1}$$

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). It can be shown that the dimensional constant K is given by

$$K=\frac{\rho k}{2\Delta L}\left(\frac{D_1}{D_2}\right)^4, \tag{2}$$

where k is an empirical dimensionless constant.

The value of k is suggested as 0.42 in the manual of CFX. We checked the validity of this value by calculating for a simple L-pipe model (see Fig. 6). The simulation conditions we imposed in CFX are ; the inlet velocity 0.2 m/s, the pipe diameter 10 mm and the air-vent diameter 1 mm.

Because of the mass conversation, the water level remains the same on the average for both cases in this single pipe model. But discrepancy can be found in the air pressure depending on the value of k. Fig. 7 shows the numerical results

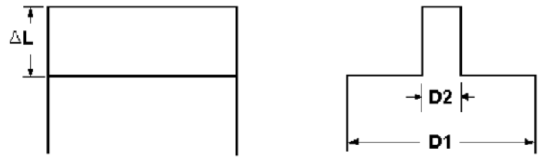


Fig. 5 Notations for derivation of formula

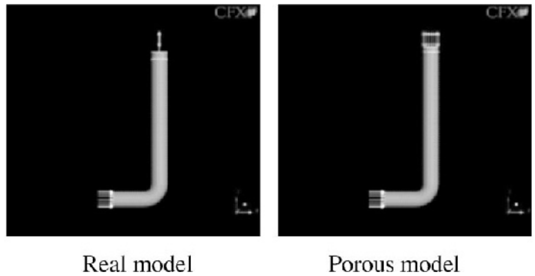


Fig. 6 L-pipe model

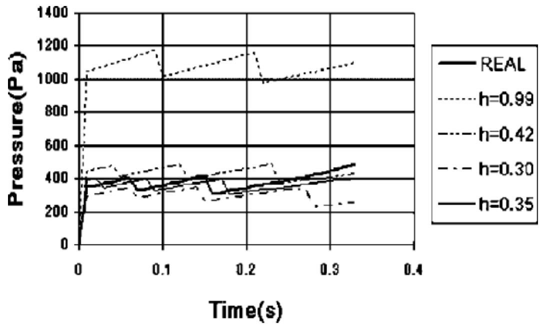


Fig. 7 Variation of the air pressure within the column of the L-pipe model obtained from the real and porous models with various k values

for various k values.

This indicates that the result with k=0.42 overestimates the pressure predicted by the simulation for the real air vent. When k equals to 0.35, the pressure value obtained from the porous model is in the best agreement with that given by the real model. So hereinafter we fix k=0.35 in the porous model.

As for the correct inlet condition, we may think two types ; one is a constant-pressure and the other is a constant-velocity. Actually which type is more relevant is dependent on the design of the actual pumping mechanism in the die-casting machine. But for simplicity, we have only employed a constant-velocity condition for most simulation cases presented in this paper except Fig. 9 which was obtained by applying the result of Fig. 2.

4. Results and Discussions

Figure 8 shows the mesh configuration in the numerical computation of three-column model. We use hexahedral mesh in the entire fluid domain including the junctions. The number of nodes for this configuration is roughly 120,000.

Figure 9 shows the comparison of the free-surface levels obtained from the visualization experiment and numerical simulation. The seemingly different thickness of the liquid between the two results is caused by the light refraction of the Plexiglas. The free surface in the left column rises the fastest at the beginning and then after 0.3 s 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.

Next, we performed a series of numerical simulations by changing the bed cylinder diameter under the same water volume flow rate to investigate the effect of the bed-cylinder size. The parameter values used in the CFD are as shown in Table 2.

We have observed that the free surface especially in the first column involves a severe distortion and even the air bubble is entrapped inside the column. Such distortion of course leads to bad quality of the die-casting product. So we have checked this distortion, Δh, as well as ΔH the maximum difference of the free-surface-height among the three columns from the numerical results (see Fig. 10).

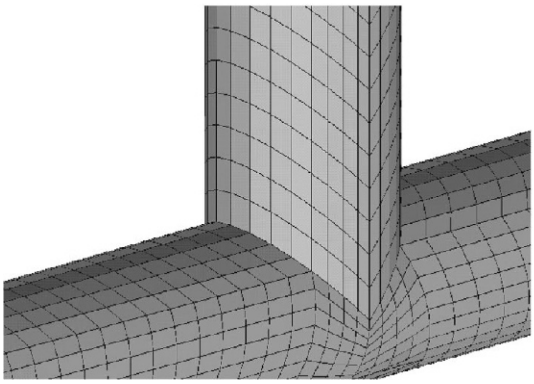


Fig. 8 Mesh configuration

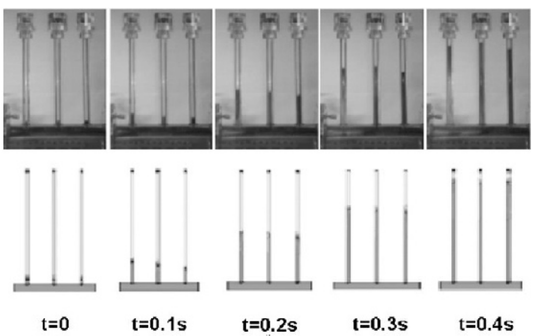


Fig. 9 Development of the free-surface level given from the experiment (top) and computation (bottom)

Table 2 Simulation conditions for parametric study

Case name	Inlet diameter (m)	Inlet velocity (m/s)	Flow rate (m^3/s)
D10	0.010	4.00	0.000314
D15	0.015	1.78	0.000315
D20	0.020	1.00	0.000314
D25	0.025	0.64	0.000314
D30	0.030	0.44	0.000311

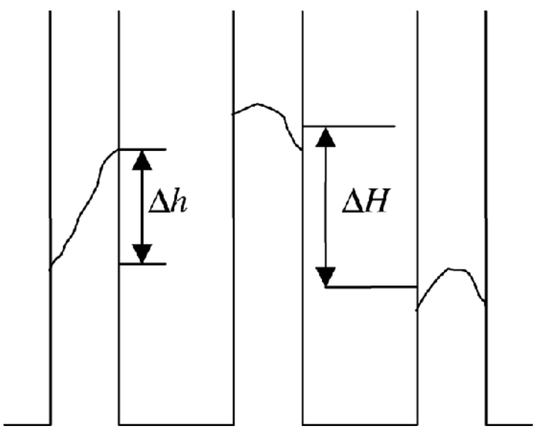


Fig. 10 Definition of Δh and ΔH

Shown in Fig. 11 is development of Δh. A large value of Δh means a big splashing in the free surface motion. We can see that at smaller size of the bed cylinder the splashing becomes bigger. As explained previously, such big splashing is harmful for the quality of the final product. On the other hand, a larger bed cylinder means larger scrap materials that bring additional manufacturing cost caused by re-melting of those materials. Fig. 12 shows development of ΔH. It also shows that a larger bed cylinder brings a smaller level-difference.

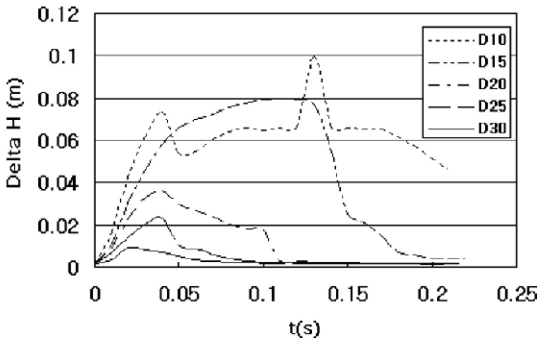
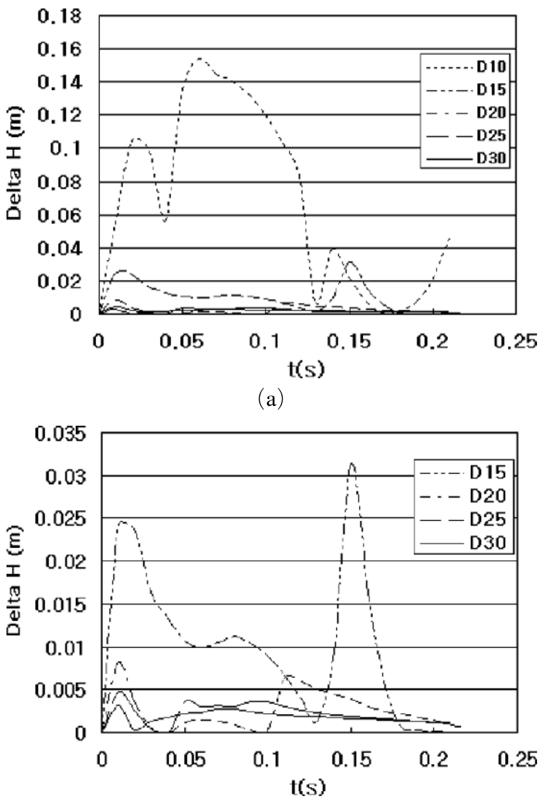


Fig. 11 Development of the free-surface deformation in the 1st column, Δh



(a)

(b)

Fig. 12 Maximum free-surface-height difference among the three columns, ΔH, (a) for all the 5 cases in a large scale of ΔH and (b) for 4 cases in a small scale of ΔH

In particular, from these figures we can see that the bed-cylinder diameter should be larger than or equal to 20 mm if we set the limit of ΔH as 10 mm for qualified products.

Figure 13 shows the development in time of the pressure of the air chamber above the free surface, the pressure being averaged all over the chamber rooms of the three columns. After 0.02 s, the pressure becomes constant and the same for all 5 cases. This pressure can be estimated theoretically by using Eq. (1). If the air density is selected as $\rho = 1.2\ \mathrm{kg/m^3}$, it gives $p_a = 3{,}800$ Pa. We can see from Fig. 13 that this prediction is very accurate.

Figure 14 shows the development of the total pressure at the inlet of the bed cylinder, the pressure being averaged over the inlet section. After 0.05 s the total pressure increases linearly in time. The reason for such linear increase is explained in terms of the hydrostatic pressure. As the water level is increased in each vertical column, the hydrostatic pressure must also increase linearly. So, the inlet pressure must be increased to withstand the increased hydrostatic pressure at the bottom of each column. The rate of the inlet-

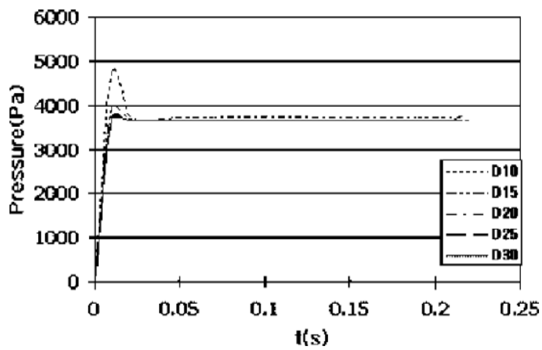


Fig. 13 The spatially averaged pressure in the air chamber of the vertical column

pressure increase in time due to the level rising and its hydrostatic effect may be given by

$$\frac{d\Delta p_g}{dt}=\gamma V, \qquad (3)$$

where γ is the specific weight of water and V is the speed of the free-surface level. For all cases shown in Table 2, $V=1.33$ m/s. Substituting this together with $\gamma=9800$ kg/m^3, we get 13,000 Pa/s as the rate of the pressure increase. So, for instance, the time elapse of 0.2 s will result in the increase of the pressure as much as $\Delta p_g=2{,}600$ Pa, and this is in good agreement with that measured from Fig. 14.

The inlet total pressure p_0 at t_0 extrapolated from the linear profile occurring after 0.05 s in Fig. 14, which is measured to be about 5,100 Pa, can be considered to be the sum

$$p_0=p_{g0}+(p_a+\rho_w V^2/2)$$

where the last term within the bracket ( ) corresponds to the total pressure at the free surface, and p_a is the air pressure in the air chamber, which has been estimated to be 3,800 Pa. The hydrostatic pressure p_{g0} due to the initial water level 15 mm is $p_{g0}=9{,}800\times0.015$ or 147 Pa. Then

$$p_{g0}=147+(3{,}800+1000\times1.33^2/2)\cong4{,}840 \text{ Pa}$$

So this theoretical value is less than the numerical prediction 5,100 Pa as much as 5% difference for the cases with the bed-cylinder diameter larger than or equal to 20 mm. This difference arises of course from the pressure loss due to the pipe friction and the local loss occurring at the junctions between the columns and bed cylinder. It is to be noted that for the case with the bed-cylinder diameter less than 20 mm, the pressure loss is significant.

In the above analysis we have assumed that the total pressure inside the bed cylinder may be uniform from the law of energy conservation. Fig. 15 shows the total pressure distribution for the case of D20 after 0.15 s. We can find that the pressure in the bed cylinder region is uniform indeed. The reason for such uniformity in the total pressure distribution comes from the fact that the dynamic pressure change due to the velocity difference through the bed cylinder is not significant for the cases with the bed-cylinder diameter larger than or equal to 20 mm.

As far as the power needed for the injection of liquid through the passages in the die is concerned, it is worth finding out which effect is dominant. The power is obtained by the flow rate times the total pressure at the inlet section. The flow rate is given by dividing the total volume of the passages within the die where the molten liquid should be filled in, including the bed cylinder, by the injection time. On the other hand, the inlet pressure p_0, as analyzed previously, is composed of p_{g0}, the initial hydrostatic pressure, p_a, the air pressure, $\Delta p_d=\rho_w V^2/2$, dynamic pressure within the column, Δp_g, the increase of the hydrostatic pressure due to the level-up of fluid

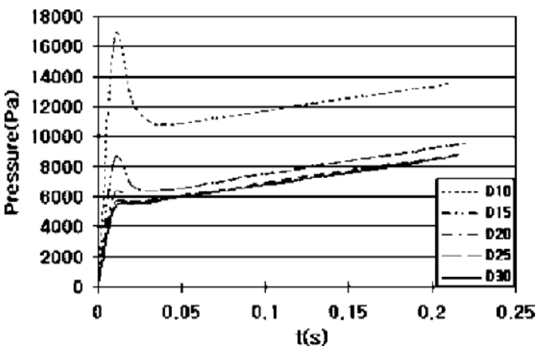


Fig. 14 The spatially averaged total pressure in the inlet section of the bed cylinder

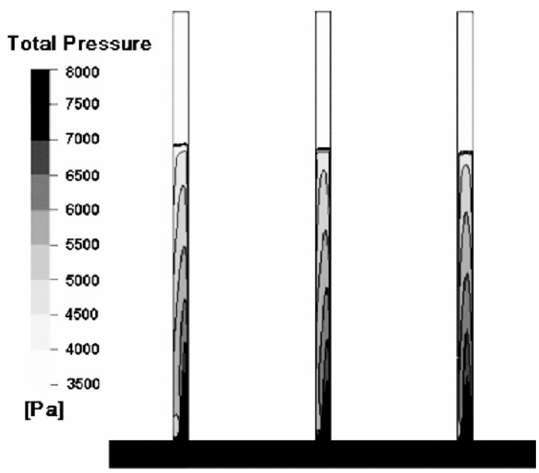


Fig. 15 Total pressure distribution in D20 model at $t=0.15$ s

in the columns and finally p_f, that due to the fluid friction. Among these all pressures are constant in time except the hydrostatic pressure Δp_g. So, to assess each pressure's contribution, we must set the time. For instance, at the time 0.2 s, $p_{g0}=$ 147 Pa, $p_a=$3,800 Pa, $\Delta p_d=$890 Pa, $\Delta p_g=$2,600 Pa, and p_f ranges from 0 to 5,000 Pa (see Fig. 14) in the present study. So, we can see that, except for the frictional effect, the most dominant contribution to the power is the air pressure and the hydrostatic pressure is the second one. On the other hand, the pressure increase due to the fluid friction becomes very significant when a small bed cylinder is selected in order to reduce the recycling material. The reason for such a sharp increase in the inlet pressure due to the friction effect for the small bed-cylinder case is because the pressure loss is proportional to the square of the velocity, or inversely proportional to the fourth power of the cylinder diameter.

As seen from Fig. 14 and as mentioned previously, the frictional effect is almost negligible for the case with the bed-cylinder diameter larger than or equal to 20 mm. So, from the point of view regarding the power consumption and the free-surface distortion, we can conclude that the optimum diameter of the bed cylinder is 20 mm for qualified products and the low cost of production.

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 inlet pressure that is directly connected with the power consumption of the die-casting process, and the theoretical prediction turns out to be in good agreement with the numerical results.

(5) It has been shown through a parameter study (the bed-cylinder diameter being the parameter) that the optimum diameter for the gentle and uniform flow within the columns exists, and for the case of 10 mm-diameter columns treated in this study the optimum value is 20 mm, that also leads to the low power consumption.

Acknowledgements

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