Numerical and Experimental Analysis of Flow Phenomenon in Centrifugal Flow Tundish

Yun WANG, Yunbo ZHONG, Baojun WANG, Zuosheng LEI, Weili REN and Zhongming REN

Shanghai Key Laboratory of Modern Metallurgy & Material Processing, Shanghai University, No. 149 Yanchang Road, Shanghai, China 200072. E-mail: yunboz@staff.shu.edu.cn

(Received on April 2, 2009; accepted on June 16, 2009)

Fluid flow phenomenon in Centrifugal Flow (CF) tundish is investigated using water modeling and numerical simulation techniques. The effect of the dam spacing and rotation speed on the flow structure has been analyzed in detail. Results reveal that the bias flow, originating from the rotary outflow, leads to the formation of transversal circulation behind the dam. Such transversal flow can effectively diminish the conventional dead volume. Meanwhile, small dam spacing helps to produce a large-scale transversal circulation, and thus the prolonged flow path and relatively low velocity results in an increased plug volume. With the increase of dam spacing, the intensity of transversal circulation decreases and the increased fluid velocity causes a diminished plug volume. The highest ratio of plug to dead volume is obtained under the dam spacing when transversal circulation is strongest. Furthermore, under lower magnetic intensities, the weaker fluid momentum leads to relatively large dead volume. With the increasing of magnetic intensity, the fluid mixing becomes better. However, much larger magnetic intensity will lead to decreased ratio of plug to dead volume. Therefore, a medium rotation speed (around 30 r/min) should be recommended.

KEY WORDS: centrifugal flow tundish; flow phenomenon; water modeling; numerical simulation; bias flow; transversal flow.

1. Introduction

The tundish, working as a buffer and distributor of liquid steel between the ladle and continuous casting (CC) molds, plays a key role in affecting the performance of the CC machine, solidification of liquid steel, quality, and productivity. As the demand for higher cleanliness becomes greater, separation of inclusion particles from molten steel has been pursued in the continuous casting process by various means, such as increasing the tundish volume size, adopting H-type tundish, using flow control devices including dams, weirs, impact pads and turbulence inhibitors, injecting argon and installing filters, etc. However, for the next generation of clean steels, it is urgent to propose positive methods for inclusion separation.

Major advances have been made in the application of magneto-hydrodynamics in metals and materials processing in recent years. The application of electromagnetic forces to inclusion separation has been proposed. Centrifugal Flow (CF) tundish, in which the molten steel is horizontally rotated by electromagnetic force, has been successfully exploited by chiba works to improve the productivity and cleanliness of the continuous-cast steel. This process has been proved to show excellent performance of deoxidation and inclusion separation in plant trials. The mechanism of inclusion separation in the rotation chamber has been experimentally and mathematically studied by Miki et al., but the flow characteristic in a whole CF tundish containing the distribution chamber was not studied. In addition, Wang et al. have employed the large eddy simulation method to study the flow structure in the centrifugal flow tundish, their work mainly emphasized on how to enhance the swirling flow in rotation chamber, while the detailed flow characteristic in the distribution chamber was less mentioned.

It is undoubted that the rotational flow in the rotation chamber plays a leading role in separating inclusions, while the melt flow in the distribution chamber also exerts considerable influence on further removal of inclusions and meanwhile the temperature and composition homogenization. Therefore, a detailed knowledge of the molten steel flow in this kind of tundish must be explored to gain better insight into the flow behaviour and achieve favourable flow conditions. So in the present work, physical and mathematical modeling study of flow phenomena in CF tundish has been carried out. The effect of two important factors, including the position of dam and rotation speed, on the flow pattern is investigated by the two methods, and the flow characteristic in the CF tundish is discussed in detail.

2. Description of the Physical Problem

A schematic view as well as the physical dimensions of an actual one-strand centrifugal flow tundish is shown in Fig. 1. This tundish system is constructed based on a conventional one-strand tundish belonging to a steel corporation and devised to the production of stainless steel slabs.
with high-quality. It consists of a cylindrical rotation chamber and a cuboid distribution chamber. A narrow tunnel on the bottom of weir connects these two chambers. The rotating magnetic field generator is placed at the outer side of the rotation chamber. The total capacity of the centrifugal flow tundish is 18 ton, the bath depth is kept at 760 mm and submerged depth of the nozzle is 200 mm. The inlet flow rate of the tundish is 1.1 ton/min. Besides, considering that the ladle stream from a center-located ladle nozzle will prevent the inclusions converging at the rotating center, an off-centered nozzle which deviates 300 mm from the center is employed. The molten steel is poured into the tundish through the long ladle nozzle, and when the molten steel passes through the rotation chamber, the inclusions in the molten steel will be greatly removed under the effect of centripetal force. Then the purified liquid steel will flow into the distribution chamber through the connecting tunnel. Residual inclusions in the liquid steel will further decrease by floating up to the top slag.

3. Water Model

To study the dynamics of the molten steel flow in the centrifugal tundish, a 1/2 scale water model satisfying the Froude number similarity criterion is constructed based on the actual tundish system. Figure 2 shows the experimental setup which consists of a perspex glass tundish, water supply, tracer injection system and the instrumentation to record the tracer concentration at tundish exit.

Water was used for simulating the liquid steel, and the density and viscosity of water are 1000 kg/m$^3$ and 0.001 Pa·s respectively. To simulate the rotational flow in the rotation chamber, stirring bars driven by the variable-frequency electromotor were adopted. The stirring bars were placed close to the inner wall of rotation chamber, and the rotation speed of the stirring bars can be freely adjusted by changing the frequency of the electromotor. Figure 3 shows the measured relationship between the frequency of electromotor and the revolution of stirring bars. Good linear relationship is obtained by fitting the measured data. Since the step length of the motor frequency can be controlled at 0.01 Hz, random revolution (such as 20 r/min, 30 r/min etc.) of the stirring bars can be achieved by adjusting to the corresponding frequency based on the linear expression in Fig. 3.

The flow behavior of water in the tundish is studied by the stimulus-response-technique. Methylene blue is used for flow visualization and photography. In addition, potassium chloride (KCl) is used for the measurement of residence time distribution. Two-hundred milliliter saturated KCl solution were added as a pulse after stable fluid flow conditions were established. The change in the electrical conductivity of the fluid was measured by an electrical conductivity probe and recorded into a computer in the form of

![Fig. 1. Two-dimensional view of centrifugal flow tundish. (a) Top view. (b) Front view.](image1)

![Fig. 2. Experimental setup.](image2)

![Fig. 3. Variation of the revolution of stirring bars with the motor frequency.](image3)
4. Mathematical Model

To conduct an analysis of fluid flow phenomena in the centrifugal flow tundish, assumptions are made to ease the complexity of the task. First, the fluid flow is isothermal. Second, the free surface of the liquid in the tundish is flat.

4.1. Governing Equations

For the three-dimensional turbulent flow calculations, the governing equations used in the Cartesian vector notation are as follows:

Continuity:
\[
\frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad \text{..........................(1)}
\]

Momentum:
\[
\frac{\partial}{\partial x_i} (\rho u_i u_j) = \frac{\partial}{\partial x_j} \left[ \mu_{\text{eff}} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{\partial P}{\partial x_j} + \rho g_i + F_i \right] \quad \text{..........................(2)}
\]

Turbulence kinetic energy:
\[
\frac{\partial}{\partial x_i} \left( \rho u_i K - \frac{\mu_{\text{eff}}}{\sigma_K} \frac{\partial K}{\partial x_i} \right) = G - \rho e \quad \text{..........................(3)}
\]

Rate of dissipation:
\[
\frac{\partial}{\partial x_i} \left( \rho u_i \varepsilon - \frac{\mu_{\text{eff}}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) = (C_1 G - C_2 \rho e) \varepsilon / K \quad \text{..........................(4)}
\]

Where
\[
\mu_{\text{eff}} = \mu_l + \mu_t = C_1 \rho K^2 / \varepsilon \quad \text{..........................(5)}
\]

\[
G = \mu_{\text{eff}} \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad \text{..........................(6)}
\]

In the above equations, \( \rho \) is the density of fluid (water/molten steel), \( u_i \) and \( u_j \) is the mean velocity of fluid in the \( i, j \) direction, and \( i, j = x, y, z \) denotes the three Cartesian coordinates direction. \( P \) is pressure, \( g_i \) is gravity acceleration in the \( i \) direction and \( F_i \) is the volumetric Lorenz force. \( \mu_l \) and \( \mu_t \) are laminar viscosity and turbulent viscosity respectively, and the sum of both the quantities is known as effective turbulent viscosity \( \mu_{\text{eff}} \). \( K \) is turbulent kinetic energy, \( \varepsilon \) and \( G \) are dissipation rate and generation of turbulent kinetic energy respectively.

Values of the five empirical constants \( C_1, C_2, C_\mu, \sigma_K \) and \( \sigma_\varepsilon \) in the turbulence model are 1.44, 1.92, 0.09, 1.0 and 1.3 respectively, suggested by Launder and Spalding.\(^7\)

4.2. Lorentz Force

The rotary magnetic field induces a Lorentz force in the electrically-conducting liquid, and the Lorentz force can be calculated using the Maxwell’s equation and Ohm’s law. In this work, an analytical solution derived by Spitzer et al\(^9\) can be used for the approximation of Lorentz force:

\[
F_{\theta} = \frac{1}{2} B^2 (\omega_j - \omega_m) \sigma r \quad \text{..........................(7)}
\]

\[
F_r = -\frac{1}{8} B^2 (\omega_j - \omega_m)^2 \sigma^2 \mu_m r^3 \quad \text{..........................(8)}
\]

For the above equations, \( \omega_m = v_j / r \) is the rotation speed of molten steel and \( v_j \) is the tangential velocity. \( \omega_r = 2 \pi f / s \) is the angular velocity of rotating magnetic field, where \( f \) is the magnetic frequency and \( s \) is pole-pair number of the electromotor. In the present work, \( f = 8 \) Hz and \( s = 1 \). \( \sigma \) and \( \mu_m \) are the conductivity and magnetic permeability of the molten steel respectively. For the liquid stainless steel, \( \sigma = 0.789 \times 10^5 \) S/m and \( \mu_m = 1.257 \times 10^{-8} \) H/m. \( B = B_0 \times e^{-\left(\frac{r}{\delta} \right)} \) is the magnetic induction in the melt, where \( B_0 \) is the amplitude of magnetic induction \( B \) at external surface of the melt, \( R_0 \) is external radius of melt, \( \delta = 4/3 (\sigma_\theta \mu_m r_0) \) is the skin depth. The Cartesian components of Lorentz force were added to the source term of momentum equation.

4.3. Boundary Conditions

In this study, at the tundish solid walls, the velocity components and the turbulence transport properties were imposed with no-slip wall boundary conditions. For the free surface, the shear stress boundary (zero shear stress) condition was used\(^9-13\). At the inlet, the mean vertical average velocity is assumed to be uniform in its transversal section, and it was obtained by dividing the steel flow rate into the transversal area. A value of zero was established for the other two components of velocity. The turbulence kinetic energy, \( K \), and its dissipation rate, \( \varepsilon \), were also assumed to be uniform and calculated as proposed in Ref. 14.

4.4. Numerical Procedures

The commercial package CFX was employed to solve the continuity, momentum, and turbulence equations, the electromagnetic force was added to the momentum equation as a source term. Second-order difference scheme was used for discretization of these equations to provide higher order accuracy. For all solved variables, RMS (root mean square) normalized residuals of 1×10^-5 was considered. In order to ensure the iteration convergence and solving precision, regular hexahedral mesh was adopted and the grid system used in current work was shown in Fig. 4.

5. Results and Discussion

5.1. Comparison of Flow Characteristics in the Tundish with and without Stirring

The flow characteristics in the tundish with and without stirring are compared using physical and mathematical
model method. For a better comparison, the ladle nozzle is centrally placed. The mathematical model is tested on the 1/2 scale water model to calculate the flow patterns of water, and the simulated results are compared with the experimental observations for the verification of reliability.

5.1.1. Mathematical Simulation

To ease the complexity of modeling the rotational stirring bars, momentum source term method (as described in Sec. 4.2) still was used to produce equivalent rotational flow. For a comparison of the calculated flow pattern with experimental observations in the case of stirring, an approximately equal rotation speed of the fluid should be ensured. So the mathematical simulation was first done and then through the post-processing, the rotation speed of the fluid was approximately calculated. The rotation speed at a certain point is defined as:

$$\omega_i = \frac{(v_i)}{r_i}$$

Averaged rotation speed is calculated through the following expression:

$$\psi = \frac{\sum_{i=1}^{N} \omega_i} {N}$$

Where \(v_i\) and \(r_i\) are the tangential velocity and rotation radius at a point respectively, and \(N\) is the number of points used in the computations (all node points contained in the rotation chamber were considered, except for a small part of points around the rotation center which will produce a high value of rotation speed due to the very small rotation radius. In this paper, the node points within the radius of 0.05 cm were excluded). In the present case, the rotation speed is approximately calculated as 40 r/min. So, the same rotation speed of the stirring bars is transported when carrying out the water modeling.

**Figure 5** shows the calculated 3-D streamlines in the tundish with and without stirring. It is obvious that the rotational flow has greatly changed the overall flow pattern. When the rotary outflow passes through the outlet of rotation chamber (as schematically shown in **Fig. 6**), it hits and moves along the rear wall of the connection tunnel owing to the large inertial force, and a relatively low-velocity recirculation region is present in the rest part of the tunnel. Meanwhile, due to the rebound effect of the tunnel wall, the outflow will deflect when it enters into the distribution chamber. And this uneven flow distribution causes biased flow in the front side of the dam. The bias flow at the outlet port of rotation chamber can be characterized (in terms of deflection angle and jet speed) based on the formula proposed in Ref. 15), which has been employed to qualify the characteristics of jet flow at the SEN outlet port.

The rotary outflow changes the way how the fluid enters into the distribution chamber and thus leads to a very different flow pattern. It can be seen from the figure that in the absence of stirring, the high-velocity fluid from the ladle nozzle directly impact on the bottom wall of the tundish, then part of the stream will enter into the second chamber. When this stream arrives at the dam, a forced upward surface flow is formed. And when this surface flow moves forward and meets the left sidewall, longitudinal recirculating flow is induced. Under this flow condition, weak flow region is present in the downward side of dam. While in the presence of stirring, the flow asymmetry brought by the bias flow results in the formation of a large horizontal circumfluence. Such a transversal flow is the main flow character of CF tundish, and it boosts fluid mixing to a large extent and thus provokes the flow exchange in the downstream side of the dam.

5.1.2. Water Modeling

In water modeling, the dye was injected at different positions to visualize the flow path. **Figures 7 and 8** show the dye dispersion in the tundish, and the overall flow pattern reflected by these photos is similar with the computation results. Focusing on the cylindrical chamber, it can be observed that rotational flow produced in the case of stirring can restrain the flow in the vertical direction, consequently preventing the short-circuiting and weakening the impaction depth of ladle stream. In the distribution chamber, dead volume is seen clearly on the downward side of dam in the absence of stirring. And this may spoil the overall...
residence time and hence the mixing in the tundish. While in the presence of stirring, the induced transversal circumfluence promotes the propagation of flow in a large range, and thus the common stagnant zone is activated. By injecting the dye at different locations and tracing its diffusion path, such transversal circulation flow has been outlined. So, the fluid movement acquired by the numerical simulation qualitatively matches well with the experimental observations.

Figure 9 shows the measured RTD curves which are plotted between dimensionless concentration and dimensionless time. It can be seen that the RTD curve shows a very sharp peak in the absence of stirring, indicating very poor mixing. Whereas in the presence of stirring, the concentration curve becomes smooth and the peak value decreases, which means better mixing has been achieved. Meanwhile, in the presence of stirring the accelerated fluid velocity leads to a decreased $t_{\text{min}}$, while the $t_{\text{peak}}$ increases a lot. Figure 10 shows the RTD characterization results. It is obvious that when the stirring is present, the dead volume fraction ($V_d$) is greatly reduced. Meanwhile, the mixed volume fraction ($V_m$) increases significantly and this change is accompanied by a minor decrease in the plug volume fraction ($V_p$).

5.2. Flow Pattern under Different Positions of Dam
Flow characteristics in the CF tundish with the dam at various positions are numerically and physically studied. In the computation, flow pattern of molten steel in the industrial-scale tundish with an off-centered nozzle was investigated. The density and viscosity of liquid stainless steel was kept at 6 842 kg/m$^3$ and 0.007 Pa·s respectively.

5.2.1. Mathematical Simulation of Steel Flow
Under the same magnetic intensity (an equivalent rotation speed of 40 r/min), flow patterns under different dam positions are investigated. The “dam spacing” was defined as the distance between the dam and the outlet port of the rotation chamber and labeled as “D” (as shown in Fig. 11(a)). Figures 11(a) to 11(c) shows the two-dimensional flow pattern through the outlet plane of rotation chamber under different dam spacing. Owing to the large velocity difference in this plane, the section is divided into three sub-zones (respectively covered by the rectangle with different line type) and velocity vector in each zone is plotted separately according to different reference velocity. These figures show that the dam spacing has an important effect on the evolution of bias flow. When the dam spacing is small, the dam has strong restriction on the bias flow and this will aggravate the bending of bias flow toward the front sidewall, so there exists serious flow asymmetry in the anterior region of the dam. With the increase of dam spacing, the bias flow can move a longer distance and it arrives at the dam in an increased deflection angle. Hence, the impact point on the dam gradually moves in the direction toward the front sidewall. When the dam spacing increases to a very large value, the rapid bias flow will impact on the tundish wall and thus lead to strong flush of the wall and serious flow.
flow asymmetry simultaneously.

**Figure 12** shows the corresponding 3-D streamlines in the tundish. It can be seen that when the bias flow meets the dam, part of the stream is recirculated backward in the direction toward the left sidewall, and a recirculation tends to form in front of the dam. In the condition of small dam spacing, the recirculating flow before the dam has more space to evolve owing to the weakened restriction of the left sidewall. So the vortex in front of the dam becomes more and more complete. And in the backside of the dam, transversal circulation also occurs due to the existence of flow asymmetry. But compared with the small dam spacing case, the intensity of transversal mixing decreases and longitudinal flow emerges. The coexistence of transversal and longitudinal flow results in an inclined circulation.

The phenomenon of flow asymmetry in the tundish is investigated by comparing the mean velocity in the front and rear half-domain of distribution chamber. The distribution chamber is divided into two sub-domains with equal volume by the vertical center-plane, and the mean velocity in each domain is calculated. **Figure 13** shows the calculated mean velocity in the two sub-domains under different dam spacing cases. It can be found that under small dam spacing, velocity difference between the two sub-domains is small and uniform flow is produced. When dam spacing increases, the velocity difference obviously increases which indicates non-uniformity of the flow. Although the bias flow leads to flow asymmetry and possibly will cause uneven flow on the whole, large transversal circulation forms under small dam spacing can create a relatively uniform flow and thus eliminate such flow asymmetry.

### 5.2.2. Water Modeling

The flow characteristics in the tundish under various dam spacing are also studied by water modeling. **Figure 14** shows the measured RTD curves. Obviously, the RTD curve for the too small dam spacing shows an apparent high peak value which indicates a poor mixing. And this peak value will descend and fluid mixing becomes better when increasing the dam spacing.

The characteristic time $t_{\text{min}}$ and $t_{\text{peak}}$ as well as their ratio are plotted in **Figs. 15(a)** and 15(b) respectively. As shown in these figures, smaller dam spacing exhibits relatively larger $t_{\text{min}}$ due to the smaller fluid velocity and the prolonged flow path. And with the increase of the dam spacing the $t_{\text{min}}$ shows apparent decreasing tendency, since larger dam spacing weakens the restriction of dam on the bias flow and thus leads to higher fluid velocity and lower $t_{\text{min}}$. 
The ratio of \( t_{\text{min}} \) and \( t_{\text{peak}} \) can reflect the degree of axial dispersion in the tundish.\(^{16,17} \) When the dam spacing is small, the \( t_{\text{min}}/t_{\text{peak}} \) value is large and this reveals that the axial dispersion decreases and uniform flow can be achieved. This is confirmed by the numerical results that transversal circulation formed under small dam spacing weakens the axial dispersion and contributes to attaining a uniform flow. With the increase of dam spacing, the \( t_{\text{min}}/t_{\text{peak}} \) reduces which indicates the increase of axial dispersion, and this is due to the change of flow pattern from transversal to longitudinal. The RTD characterization result is shown in Fig. 16. The result shows that small dam spacing is beneficial to achieve an increased plug volume. But dead volume will enlarge simultaneously, especially for too small dam spacing where lower fluid velocity is present. With the augment of dam spacing, the increased average velocity helps to reduce the dead volume. While the enlarged mixed-volume yields a diminished plug volume. Large ratio of plug to dead volume and plug to mixed volume, which can ensure large inclusion separation ratio,\(^{16} \) is achieved at the dam spacing of 0.2 m (equivalent to 0.4 m in the prototype tundish). Under this dam spacing, large-extent transversal circulation will be induced. So the transversal circulation is validated by water model to be capable of increasing the plug volume and thus promoting the inclusion removal.

5.3. Flow Pattern under Different Rotation Speeds

5.3.1. Mathematical Simulation of Steel Flow

The effect of magnetic intensity on the flow structure has been investigated under different dam spacing cases. Here, only the results for case of \( D=0.6 \) m is given for illustration. Figure 17 shows the velocity profile in the plane through the outlet of rotation chamber for different magnetic intensities. The figure reveals that the magnetic intensity influences the deflection angle of bias flow through affecting the magnitude of outflow inertia. With the augment of magnetic intensity, the bias flow meets the dam in an increased deflection angle. But with the further increase of magnetic intensity above 0.02 T (about 30 r/min), the orientation of bias flow keeps almost unchanged possibly due to the constraint of geometry configuration.

The corresponding 3-D streamlines in the tundish are shown in Fig. 18. It can be seen that in the rotation chamber, with the increase of magnetic intensity, the molten steel rotates in a more and more axis-symmetric pattern. And in the distribution chamber, the changing orientation of bias flow leads to different degree of asymmetrical flow and thus different flow pattern. Under lower magnetic intensity,
transversal circulation is predominated. Since the transversal flow increases the path line of flow, the weak fluid momentum generated under a low magnetic intensity will fail to induce a global circulation. In this situation, the flow asymmetry caused by the bias flow seems to be an unfavorable factor for deteriorating the overall mixing. With the magnetic intensity rising up, the increased fluid momentum will create a better circulation. Meanwhile, the transversal mixing weakens and an inclined longitudinal circulation is induced. When the magnetic intensity exceeds 0.03 T, the flow pattern remains almost the same (just as shown in Fig. 18(d)) owing to minor change of deflection angle. Once the flow path keeps unchanged, further increase in the magnetic intensity will result in higher velocity throughout the domain, thereby causing the mean residence time to be lower and thus the mixing becomes less efficient. Under the other dam spacing cases, the magnetic intensity is found to have similar effect on the evolution of bias flow and the resulting flow pattern.

5.3.2. Water Modeling

Figure 19 shows the experimental RTD curves measured under different rotation speeds. It can be seen that at very low rotation speeds (10 rpm), the concentration curve achieves higher peak value and fluid mixing is poor. However, when the rotation speed is beyond 20 rpm, the peak value in the RTD curves decreases significantly and a better mixing can be obtained. In addition, these RTD curves apparently show similar statistical dispersion. The variation of $t_{\text{min}}$ and $t_{\text{peak}}$ as well as their ratio is plotted in Fig. 20. The figure shows that with the rotation speed rising up the $t_{\text{min}}$ decreased, since larger rotation speeds result in higher flow velocity and thus leads to fast appearance of the tracer at the exit. The variation of $t_{\text{min}}/t_{\text{peak}}$ shows that smaller rotation speeds shows a high value of $t_{\text{min}}/t_{\text{peak}}$ which indicates decreased longitudinal dispersion, while large rotation speed will cause the descend of this value which indicates an enhanced longitudinal dispersion. And this can be rightly explained by the aforementioned simulation results, which have showed that transverse flow is predominant under smaller rotation speeds and while longitudinal circulation strengthens at larger rotation speeds.

The quantitative flow results are shown in Fig. 21. It can be found that at very low rotation speed, the flow behavior is less favorable since large dead volume and small piston volume is presented. And with the rotation speed augments, the mixed volume is significantly enlarged and the dead volume fraction increased.

Fig. 18. Calculated path lines in the tundish under different magnetic intensities. (a) $B_0=0.01$ T, (b) $B_0=0.015$ T, (c) $B_0=0.02$ T, (d) $B_0=0.03$ T.

Fig. 19. Experimental RTD curves measured under different rotation speeds.

Fig. 20. Variation of $t_{\text{min}}$, $t_{\text{peak}}$ and $t_{\text{min}}/t_{\text{peak}}$ with the rotation speed.

Fig. 21. Flow quantification results derived from the RTD curves.
volume is reduced to a low level. Meanwhile, when the rotation speed increases beyond 30 r/min, similar quantitative results have been obtained owing to the similarity in the flow pattern. Nevertheless, the increased rotation speeds will result in slight decrease in plug volume and thus low ratio of plug to dead volume. Based on the analysis above, medium rotation speeds (which are around 30 r/min) should be recommended.

6. Conclusions

In present work, fluid flow characteristics in a centrifugal flow tundish have been physically and mathematically modeled and the main conclusions derived from this study are as follows.

(1) The bias flow, originating from the rotary outflow, leads to uneven flow and thus results in the formation of transversal circulation. Such transversal flow is beneficial to diminish the conventional dead volume.

(2) Small dam spacing will promote the formation of large-scale transversal circulation, and thus the prolonged flow path and the relatively low velocity will result in an increased plug volume. While dead volume enlarges simultaneously, especially for too small dam spacing case. With the augment of dam spacing, the intensity of transversal circulation decreases. In addition, the averaged fluid velocity increases and the enlarged mixed-volume yields a reduced plug volume. The highest ratio of plug to dead volume is obtained under the dam spacing when the transversal circulation is strongest.

(3) Under low magnetic intensities, the weak fluid momentum causes a relatively large dead volume. With the rise of magnetic intensity, the fluid mixing becomes better and meanwhile shows similar characterization results. But much larger magnetic intensity will result in higher fluid velocity and thus low ratio of plug to dead volume. Therefore, medium rotation speeds (which are around 30 r/min) should be recommended.

Acknowledgements

The author wish to thank the Shanghai Scientific and Technological Committee for support of this research, Project No. 065212079.

REFERENCES

1) Y. Miki, H. Kitaoka, N. Bessho, T. Sakuraya, S. Ogura and M. Kuga: Tetsu-to-Hagané, 82 (1996), No. 6, 498.
2) Y. Miki, H. Shibata, N. Bessho, Y. Kishimoto, K. Sorimachi and A. Hirot: Tetsu-to-Hagané, 86 (2000), No. 4, 239.
3) Y. Miki, H. Kitaoka, T. Sakuraya and T. Fujii: ISIJ Int., 32 (1992), No. 1, 142.
4) F. Wang, B. K. Li and F. Tsukihashi: ISIJ Int., 47 (2007), No. 4, 568.
5) Y. Sahai and T. Emi: ISIJ Int., 36 (1996), No. 6, 667.
6) S. Lopez-Ramirez, J. de J. Barreto, P. Vite-Martinez, J. A. Romero-Serrano and C. Durant-Valencia: Metall. Mater. Trans. B, 35B (2004), 957.
7) B. Launder and D. Spalding: Comp. Meth. Appl. Mech. Eng., 3 (1974), 269.
8) K. H. Spitzer, M. Dubke and K. Schwertfeger: Metall Trans. B, 17B (1986), 119.
9) A. Tripathi and S. K. Ajmani: ISIJ Int., 45 (2005), No. 11, 1616.
10) P. K. Jha, S. K. Dash and S. Kumar: ISIJ Int., 41 (2001), No. 12, 1437.
11) A. Kumar, S. C. Koria and D. Mazumdar: ISIJ Int., 44 (2004), No. 8, 1334.
12) Y. He and Y. Sahai: Metall. Mater. Trans. B, 18B (1987), 81.
13) R. Pardeshi, S. Basak and A. K. Singh: ISIJ Int., 44 (2004), No. 9, 1534.
14) D. B. Spalding: Int. J. Num. Math. Eng., 4 (1972), 551.
15) F. M. Najjar, B. G. Thomas and D. E. Hershey: Metall. Mater. Trans. B, 26B (1995), 749.
16) S. Singh and S. C. Koria: ISIJ Int., 33 (1993), No. 12, 1228.
17) S. C. Koria and S. Singh: ISIJ Int., 34 (1994), No. 10, 784.
18) Y. Sahai and R. Ahuja: Ironmaking Steelmaking, 13 (1986), 241.

© 2009 ISIJ