Numerical Calculation of Circulation Flow Rate in the Degassing Rheinstahl-Heraeus Process

Young-Geun PARK, Won-Chul DOO, Kyung-Woo YI and Sang-Bok AN
School of materials Science & Eng., Seoul National University, Seoul 151-742 Korea.
(Received on October 4, 1999; accepted in final form on April 11, 2000)

The circulation flow rate of a steel melt, which is one of the important factors which determine decarburization rate of RH system, was calculated by using a 3-dimensional numerical simulation technique. A new model for evaluating the shape and volume of the plume zone, formed by Ar gas blown through multiple nozzles on the wall of the snorkel, was proposed to determine the driving force of melt circulation.

Circulation flow rates were measured in the 1/10 scale water model and in a real RH system to verify the calculation results with good agreement with the calculated circulation flow rates. Because the computer program can calculate the circulation flow rates at various operating conditions and dimensions of the RH system, it may be useful in determining the optimum operating conditions and designing new RH systems.

KEY WORDS: RH process; plume zone; numerical simulation; water model.

1. Introduction

Recently, in order to meet the demands for sheet steel that has good cold workability, the production of ultra-low carbon steel has increased. To produce ultra-low carbon steel, steel melt tapped from the converter to the ladle must be decarburized before it is conveyed to the continuous casting process. Among the various decarburization processes, the RH (Rheinstahl-Heraeus) vacuum system is widely used.

The refining process of the RH system is classified as degassing and decarburization of a steel melt which circulates between the ladle and vacuum vessel by blowing with Ar gas. Although the RH system can treat a larger volume of melt compared with other refining processes, the decarburization rate of a steel melt with a low carbon content has a tendency to be low.

One of the dominant factors which influence the decarburization rate in the RH process is circulation flow rate (the melt flow rate per unit time through the snorkel). It is generally accepted that the decarburization rate increases as the circulation flow rate increases. The circulation flow rate in the RH process depends upon the shape of the system such as snorkel diameter, radius of the vacuum chamber, nozzle position and number, and operation conditions such as the pressure of the vacuum vessel, Ar gas flow rate and the submerged depth of the snorkels. Because direct measurement of the circulation flow rate is difficult, many studies on the RH system have been performed to predict the circulation flow rate using water model experiments, mixing time measurements in real process and numerical methods.

Water model experiments have an advantage in being able to measure the flow velocity more easily than other experiments. Water model experiments have been done on a 1/10–1/5 scale model in order to investigate the influence of circulation gas flow rate, snorkel submerged depth, snorkel diameter and the number of nozzles, on circulation flow rate. From these water model experiments, several empirical equations to predict the circulation flow rate have been proposed. However, there is a limitation in their applicability to a real RH system due to the large difference between the conditions of the water model and the real system.

Kurokawa proposed an empirical equation to calculate the circulation flow rate using mixing time which is more easily measured than flow rate. However, mixing time measurements are time consuming and costly. Mixing time measurements also have the problem of melt contamination with the tracer material.

Numerical calculations are alternative method to overcome experimental limitations. Fluid flow and concentration variations were calculated by numerical simulations. Previous numerical studies did not calculate the circulation flow rate but calculated the variation in concentration or the decarburization rate with assumed values for the circulation flow rates. The reason for this is that previous studies did not calculate the buoyancy force of Ar gas.

In this study, the volume of the two phase region (plume zone) and buoyancy force caused by blowing Ar gas was calculated. Using this buoyancy force as a source of fluid flow, the velocity field and circulation flow rate of the RH system were calculated. The results were compared with the experimental result of the water model and the real sys-
tem with good agreement. Furthermore, using this numerical method, the behavior of the circulation flow rate with increasing Ar gas flow rates was calculated in a real RH system.

2. Numerical Calculation Method

The purpose of the numerical calculation is to simulate the fluid flow in a RH system shown in Fig. 1. Because the real fluid undergoes 3 dimensional motion, the 3-components of velocity (u, v, w) should be calculated. In addition, turbulent motion of the fluid must be considered because of the large size of the ladle and vacuum equipment and the high velocity of the melt. This study has calculated fluid turbulent motion using the k-ε model.

2.1. Governing Equation

The general governing equation for calculating 3-dimensional fluid flow in a cylindrical coordinate is expressed as

\[
\frac{\partial (\rho \phi)}{\partial t} + \frac{\partial (\rho u \phi)}{\partial r} + \frac{1}{r} \frac{\partial (\rho v \phi)}{\partial \theta} + \frac{\partial (\rho w \phi)}{\partial z} = \frac{1}{r} \frac{\partial}{\partial r} \left( r \Gamma_\phi \frac{\partial \phi}{\partial r} \right) + \frac{1}{r} \frac{\partial}{\partial \theta} \left( \Gamma_\phi \frac{\partial \phi}{\partial \theta} \right) + \frac{\partial}{\partial z} \left( \Gamma_\phi \frac{\partial \phi}{\partial z} \right) + \frac{\partial}{\partial t} \left( \Gamma_e \frac{\partial \phi}{\partial t} \right) + S_\phi
\]

(1)

In Eq. (1), \( \phi \) represents the three components of velocity (u, v, w), turbulent energy (k) and energy dissipation (ε). \( \rho \) is the fluid density and \( \Gamma_\phi \) is the transport constant of each variable as shown below, where \( \mu_i \) is viscosity and \( \mu \) is turbulent viscosity. The turbulent viscosity is calculated by Eq. (2) from the calculated values of turbulent energy and energy dissipation.

\[
\mu = C_{\mu} \rho k^2 \varepsilon, \quad \text{ ...........................................(2)}
\]

where \( C_{\mu} \) is the experimental data which is listed in Table 2. And in Table 1, \( \sigma_i \) and \( \sigma_0 \) are the turbulent Schmit number for turbulent energy and energy dissipation respectively. These values vary with the intensity of turbulence and the fluid properties. The data used in this study are listed in Table 2. \( S_\phi \) is the source term of each variable. \( S_\phi \) of each velocity component is listed below.

\( S_\phi \) for radial velocity:

\[
S_r = \frac{\partial P}{\partial r} + \frac{\rho v^2}{r} - \frac{1}{r} \frac{\partial \mu}{\partial r} \left( \frac{\partial u}{\partial r} + \frac{\partial v}{\partial \theta} + \frac{\partial w}{\partial z} \right)
+ \frac{1}{r} \frac{\partial}{\partial \theta} \left( r \mu \frac{\partial u}{\partial \theta} \right) + \frac{1}{r} \frac{\partial}{\partial z} \left( \mu \frac{\partial w}{\partial z} \right)
\]

(3)

\( S_\phi \) for azimuthal velocity:

\[
S_v = \frac{\partial P}{\partial \theta} + \frac{\rho \mu v}{r} + \frac{2 \mu u}{r \theta} - \frac{\partial \mu}{\partial r} \left( \frac{\partial u}{\partial r} + \frac{\partial v}{\partial \theta} + \frac{\partial w}{\partial z} \right)
+ \frac{1}{r} \frac{\partial}{\partial \theta} \left( r \mu \frac{\partial v}{\partial \theta} \right) + \frac{1}{r} \frac{\partial}{\partial z} \left( \mu \frac{\partial w}{\partial z} \right)
\]

(4)

\( S_\phi \) for axial velocity:

\[
S_w = \frac{\partial P}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r} \left( r \mu \frac{\partial u}{\partial r} \right) + \frac{1}{r} \frac{\partial}{\partial \theta} \left( \mu \frac{\partial v}{\partial \theta} \right) + \frac{1}{r} \frac{\partial}{\partial z} \left( \mu \frac{\partial w}{\partial z} \right)
\]

(5)

In Eqs. (3)-(5), \( F_r \), \( F_\theta \) and \( F_z \) are external forces such as gravitational force.

\( S_\phi \) for turbulent energy:

\[
G - C_{\phi} \rho \varepsilon = \frac{G - C_{\phi} \rho \varepsilon^2}{k}
\]

(6)

\( S_\phi \) for energy dissipation:

\[
C_1 G \frac{\varepsilon}{k} - C_2 \rho \frac{\varepsilon^2}{k}
\]

(7)

where \( C_1, C_2 \) are listed in Table 2 and \( G \) is defined as,

\[
G = \mu \left[ 2 \left( \frac{\partial u}{\partial r} \right)^2 + \left( \frac{\partial v}{\partial \theta} + \frac{\partial u}{\partial r} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right]
+ \left( \frac{\partial v}{\partial r} - \frac{\partial u}{\partial \theta} - \frac{\partial w}{\partial z} \right)^2 + \left( \frac{\partial w}{\partial r} - \frac{\partial u}{\partial \theta} + \frac{\partial w}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial r} - \frac{\partial w}{\partial \theta} + \frac{\partial v}{\partial z} \right)^2
\]

(8)

Since the pressure gradients are the sources of the velocity components, the pressure field also should be solved. This field is calculated from the following continuity equation,
\[ \frac{\partial \rho}{\partial t} + \frac{1}{r} \frac{\partial}{\partial r} \left( \rho u_r \right) + \frac{1}{r} \frac{\partial}{\partial \theta} \left( \rho u_\theta \right) + \frac{\partial}{\partial z} \left( \rho u_z \right) = 0 \quad \text{(9)} \]

2.2. Boundary Condition

Wall Boundary: the section of the wall where the melt makes contact with the ladle, vacuum vessel and snorkels. The implementation of wall boundary conditions in turbulent flows starts with the following equation

\[ y^+ = \frac{\rho C_{\mu}^{1/4} \sqrt{k}}{\mu_1} y, \quad \text{.................(10)} \]

where \( y \) is the distance from the near wall node to the solid wall. The wall shear stress value (\( \tau_w \)) is obtained from\(^8\)

\[ y^+ \leq 11.63, \quad \tau_w = \frac{\mu u}{y}, \quad \text{..............(11)} \]

\[ y^+ > 11.63, \quad \tau_w = \frac{\rho C_{\mu}^{1/4} \sqrt{k u}}{\ln(Ey)} , \quad \text{..............(12)} \]

where \( u \) is the velocity which is parallel to the wall at the grid node. \( k \) and \( E \) are constants of which the values are 0.4187 and 9.793 respectively.

Free Surface: the melt surface which is exposed to either air or vacuum. The influence of slag was ignored. The velocity which is perpendicular to surface was set to zero and with the parallel component, it was assumed that the gradient of them was zero.

Center Line: The radial velocity was determined from the average of the neighboring grid values, and the other values were assumed to have the same boundary conditions as the free surface.

Buoyancy Force by Ar Gas: The driving force for melt circulation is the buoyancy force of the Ar gas which is blown in the up-snorkels. This force is included in the source term \( (F_s) \) for calculating axial-velocity. The volume and the buoyancy force of an Ar plume zone are calculated to determine the driving force for melt circulation. The procedure for calculating the buoyancy force is explained in Sec. 3.

2.3. Details of Numerical Scheme

The finite difference method is used to obtain the difference representation of Eq. (1). The grids for the velocities and the scalar variables, such as pressure, turbulent energy and energy dissipation, were staggered to prevent a checkerboard pressure field.\(^9\) The algorithm for calculating the variables is the SIMPLE algorithm.\(^9\) Total grid number was 42\( \times \)40\( \times \)50 (ladle: 42\( \times \)40\( \times \)20, snorkel: 5\( \times \)8\( \times \)15, vacuum vessel: 42\( \times \)24\( \times \)15).

3. The Behavior of Ar Gas and Calculation of Buoyancy Force

To simulate the velocity fields of the melt, the buoyancy force operated by blowing Ar gas must be calculated. The force operated by the gas in a liquid may be expressed as\(^9\)

\[ f = g(\rho_l - \rho_g) \cdot V_j \cdot \alpha (1 - \alpha), \quad \text{..............(13)} \]

where \( g \) is acceleration due to gravity, \( V_j \) is the volume of the plume and \( \alpha \) is the average volume fraction of the gas. To calculate the buoyancy force using Eq. (13), the volume and the average gas fraction of the plume should be calculated.

3.1. Calculation of the Plume Volume

Ar gas in the RH operation is blown by nozzles on the wall of the up-snorkel. In case of side gas blowing, the trajectory and shape of the plume is not well known. To calculate the shape of the plume, the plume zone was divided into 2 regions; near the nozzle throat and the region where gas rises vertically. Themelis et al.\(^10\) calculated the trajectory of the plume centerline, sketched in Fig. 2, with the assumption that the vertical force exerted on the melt is caused by the buoyancy force and horizontal force caused by the inertia force of the blown gas. The equation for the centerline trajectory is

\[ \frac{d^2 y_s}{dx_t^2} = \frac{4}{N_{iv} \rho_g} \left[ \tan^2 \left( \frac{\theta_c}{2} \right) / \cos \theta_o \right] \left[ 1 + \left( \frac{dy_s}{dx_t} \right)^2 \right]^{3/2} x_t^2 \alpha_p, \quad \text{..............(14)} \]

where \( y_s = y/d_g \), \( x_t = x/d_o \), \( N_{iv} = \rho_l \rho_g g (\rho_l - \rho_g) d_g \), \( d_g \) is the diameter of nozzle (m), \( \rho_l \) is the density of melt (kg/m\(^3\)), \( \rho_g \) is the density of gas (kg/m\(^3\)), \( d_o \) is the velocity of gas at the nozzle throat and \( \theta_c \) is the angle between the centerline of the plume and the horizontal line.

The boundary conditions are written as

\[ y_s = 0, \text{ at } x_t = \frac{1}{2 \tan \left( \frac{\theta_c}{2} \right)} , \]

\[ \frac{dy_s}{dx_t} = \tan \theta_o, \text{ at } x_t = \frac{1}{2 \tan \left( \frac{\theta_c}{2} \right)} , \]

where \( \theta_o \) is the angle between the nozzle and horizontal line. \( \theta_c \) is calculated from the viscosity \( (\mu_g) \) and density \( (\rho_g) \) of the gas using following equation.\(^11\)

\[ \tan \left( \frac{\theta_c}{2} \right) = 0.238 \left( \mu_g / \rho_g \right)^{0.133} \mu, \rho: \text{CGS unit} \quad \text{..........(15)} \]
The volume fraction of gas in plume is calculated from the momentum and the continuity equation as

\[
\alpha_p = \frac{d}{d} \left( \frac{d^2}{d} + \frac{\rho_l}{\rho_g} (1 - \alpha_p) \right)^{1/2}, \quad \ldots \ldots \quad (16)
\]

where \( d \) is the diameter of the plume area perpendicular to the centerline. \( d \) is expressed as

\[
d = 2 \times \tan(\theta_c/2) \quad \ldots \ldots \quad (17)
\]

The plume area, calculated using Eq. (17), are consistent with experimental results before the jet curves to the vertical direction. However, when the bubble rises vertically, the calculated plume width becomes smaller than the experimental result.

At the region where the bubble rises vertically, the plume shape was calculated using the experimental results of the plume by vertical blowing. The plume area which is perpendicular to the plume centerline calculated using Themelis’s model was calculated using the following equations proposed by Iguchi \(^{12}\) were adopted.

\[
\begin{align*}
\beta_A &= \beta_A(z_0(z/z_0)^n), \quad \ldots \ldots \quad (18) \\
\beta_A(z_0) &= 0.26(G^2/g)^{0.2}(\rho_l/\rho_g)^{0.07}, \\
z_0 &= 0.77d_{in} \left( \frac{\rho_l}{\rho_g} \right)^{0.28} Fr^{0.89}(\rho_l/\rho_g)^{-0.16}, \\
Fr' &= \frac{\rho_l G^2}{\rho_l gd_{in}^5}, \\
r_p &= 1.7b_a.
\end{align*}
\]

In Eq. (18), \( b_a \) is the radial distance where the gas volume fraction is the half the centerline gas fraction. \( \rho_l \) is the density of the melt (kg/m\(^3\)), \( \rho_g \) is the density of the gas (kg/m\(^3\)), \( G \) is the gas flow rate (m\(^3\)/s), \( g \) is the gravitational acceleration (m/s\(^2\)), \( d_{in} \) is the diameter of nozzle (m) and \( r_p \) is the radius of plume area.

As Ar is blown at the wall of the up-snorkel in the RH operation, the plume area was calculated using Themelis’s equation near the nozzle and Iguchi’s equation at the vertical plume zone. The boundary of two equations is where the calculated radius of the plume (Szekely: \( d/2 \), Iguchi: \( r_p \)) is same for both calculation methods. In Fig. 3, the shape of the plume zone calculated using these methods is shown. The centerline represented in Fig. 3 was calculated using Themelis’s equation. It can be seen that the plume area calculated by Iguchi’s equation is symmetrical to centerline of plume.

### 3.2. The Superposition of Plume Zones

In the RH operation, 12–24 nozzles are used to blow Ar gas into the melt from the wall of the up-snorkel. Because the plume zone area widens with the bubble rising as shown in Fig. 3, the plume zones superpose upon one another, as shown in Fig. 4. Figure 4 shows a section of snorkel when the nozzle number is 12. A small circle represents the plume zone formed by a single nozzle. As a result of superposition, the total volume of plumes is smaller than the sum of each of the plumes. The method of calculating the volume of superposed plumes is as follows.

Firstly, lines are drawn from the center of the snorkel to every mid-point of the adjacent nozzles, the snorkel section is then divided into 12 parts, same as the number of nozzles. Each part corresponds to each nozzle which is located at the center of the arc of the fan-shaped region. The effective area of the plume formed by one nozzle, which is located in corresponding part, is then calculated. As shown in Fig. 4, the hatched region is the effective area of one plume. The effective volume of a plume formed by a nozzle is calculated by integrating each effective area from the nozzle position to the free surface of the melt in a vacuum vessel. The total volume of plumes (\( V \)) is the number of nozzles multiplied by the effective volume of one plume.

The average volume fraction of gas (\( \alpha \)) is the total volume of residual gas in the melt divided by the total plume volume. The volume of residual gas was calculated as

\[
G_m = G \times t_{\text{melt}} = G \times \frac{H_{\text{melt}}}{u_{\text{melt}} + u_{\text{slip}}}, \quad \ldots \ldots \quad (19)
\]
where \( G \) is the gas flow rate \((m^3/s)\), \( t_{melt} \) is the residual time of the gas, \( H_{melt} \) is the distance from the nozzle to the vacuum surface, \( u_{melt} \) is the velocity of the melt and \( u_{slip} \) is the relative velocity of gas in the melt. Therefore, the average volume fraction of gas (\( \alpha \)) is \( G_m/V_j \).

### 3.3. The Effect of Vacuum Vessel Pressure

In water model experiments, the difference between the surface pressure and the pressure at the nozzles is small. However, the density of melt is so large that the hydrostatic pressure of the melt should be considered. Moreover, the melt surface in a RH system is in a vacuum vessel. Thus the pressure of melt surface is same with that of a vacuum vessel. Therefore, bubbles grow larger than that of the 1 atm model experiment.

The bubble volume can be calculated from the gas equilibrium equation with ease. The gas equilibrium equation, \( PV=\text{constant} \), says that the volume of gas is inversely proportional to pressure.

However, it is doubtful that this equilibrium assumption is realistic. For example, if the pressure of vacuum vessel is 1 torr, the volume of gas which rises from nozzle will increase approximately 1 000 times. In reality, the residual time of gas in the melt is only about 1 second and the bubble volume near the surface has to rapidly increase and push the melt to meet the equilibrium.

Szekely et al. proposed the following equation regarding rising bubble size under reduced pressure from the continuity, momentum and gas equilibrium equation.

\[
R_b^3 + \frac{3}{2} \frac{R_b^2}{R_b} = \frac{1}{\rho_i R_b} \left[ P_0 \left( \frac{R_b}{R_b} \right) \right]^{\frac{3}{2}} P_0 + \rho_i g u_{slip} \frac{t}{R_b} \\
(\text{Eq. 20})
\]

In Eq. (20), \( R_b \) is the radius of the bubble, \( \rho_i \) is the density of the melt, \( P_0 \) is the pressure of the vacuum vessel and \( H \) is the height of the melt. Using this equation, the effect of reduced pressure on bubble volume is calculated and the expansion ratio is multiplied by the driving force, Eq. (13).

### 4. Result and Discussion

#### 4.1. Fluid Flow Calculation

Using the 3-dimensional fluid flow calculation program, the fluid flow in the RH system was calculated. The calculated buoyancy force expressed in Eq. (13) was assumed to act on the control volumes which correspond to the melt in and above the up snorkel from nozzle to vacuum vessel surface. The force acting on the each control volume is calculated as

\[
f_{C.V} = f_{tot} \times V_{C.V} \quad \text{...........................(21)}
\]

In Eq. (21), \( f_{tot} \) is the calculated total buoyancy force, \( V_{tot} \) is the total volume of control volumes on which the buoyancy force act and \( V_{C.V} \) is the volume of each control volume.

**Figure 5** shows the calculated flow results in the snorkel sectional plane when the vacuum pressure is 10 torr and the Ar gas flow rate is 200 Nm3/hr. The melt in the ladle rises to the vacuum vessel by the Ar gas blown in the up-snorkel and moves to the down-snorkel. **Figure 6** shows the fluid flow in the surface of the vacuum vessel. The melt spreads widely from the up-snorkel to down-snorkel.

The result of fluid flow analysis in the RH system may provide important data in calculating the local behavior of the decarburization rate and mixing. To verify the numerical models, the local velocities should be compared with experimental results. However it is nearly impossible to measure the local velocity of the melt. Therefore the circulation flow rate was measured in a water model and in a real RH system in order to verify the numerical model.

#### 4.2. Comparison with Water Model Experiment

A 1/10 scale water model was set up to measure the velocity in the down snorkel and these results were compared with calculations. **Figure 7** represents the water model system. 4 nozzles were set up at each of 90° intervals with a nozzle diameter of 0.003 m. The vacuum vessel pressure was 743 torr, the submerged depth was 0.045 m and the air flow rate was 1.5–4.0 l/min. The velocity in the down-snorkel was measured using a CCD camera and image ac-
The water model was made of transparent acrylic plastics and the tracer was made of a small rubber balloon (about 0.005 m) which was filled with the water. The sequential images were acquired using computer interfaced image board in the down-snorkel.

Figure 8 shows sequential images in the down-snorkel. The velocity was the tracer interval between two sequential images divided by the elapsed time. Because the difference in the velocities along the radial direction in a section of snorkel is small,\(^3\) the flow rate can be calculated using the measured velocities.

Figure 9 shows the velocity in the down-snorkel measured by this method. As the gas flow rate was increased from 1.5 l/min to 4.0 l/min, the water flow in the down-snorkel increased from 0.115 m/s to 0.155 m/s.

The calculated result is also shown in Fig. 9 with the water model result for comparison. The calculated flow rate is the same as that measured by the water model within experimental error. Therefore the circulation flow rate of water can be predicted by this numerical model.

4.3. Comparison with a Real RH System

To verify the numerical model, experiments in a real RH system were performed and the result compared with the calculations. The circulation flow rate in the melt was measured from mixing time. Mixing time was measured using a Cu tracer and circulation flow rate was evaluated from the following equation proposed by Kurokawa,\(^4\)

\[ Q = 3.33 \cdot D^{4/3} \cdot W^{1/3} \cdot T^{-0.74} \] \hspace{1cm} (22)

In Eq. (22), \( Q \) is the circulation flow rate (ton/min), \( D \) is the diameter of the snorkel (cm), \( W \) is the weight of the melt (ton) and \( T \) is the complete mixing time (sec). 125 kg of Cu was injected into the melt when the melt circulation reached a steady state. After Cu injection, a sample was taken from the melt every 12 or 30 sec. Cu was analyzed from the sample by emission spectroscopy. The complete mixing time was assumed to be the time when the Cu concentration was uniform within 3% error.

The gas flow rates in each experiment was 120, 175 and 210 Nm\(^3\)/hr respectively. The experimental and calculated results are shown in Table 3. The circulation flow rate evaluated from mixing time show good agreement within 5–7% with that of the numerical model.

4.4. Advantage of the Numerical Model

To estimate the circulation flow rate of melt, several empirical equations have been proposed. For example,

\[ Q = 11.4D^{1.33}G^{0.33} \left( \frac{P_1}{P_2} \right)^{0.33} \] \hspace{1cm} (23)

where \( Q \) is the circulation flow rate (ton/min), \( G \) is the gas flow rate (Nm\(^3\)/min), \( D \) is the snorkel diameter (m), \( P_1 \) is the vacuum vessel pressure (torr) and \( P_2 \) is the pressure where the gas is blown (torr).\(^2\)

Equation (23) can evaluate the circulation flow rate with the operation conditions (for example, the vacuum vessel pressure and gas flow rate) and the shape condition (the diameter of snorkel) of a RH system. If only knowledge of the circulation flow rate is needed, the numerical simulation needs more time than the empirical Eq. (22) or (23). However, to obtain the circulation flow rate using Eq. (22), the mixing time should be measured through tracer experiments which are more difficult and time consuming than the numerical simulation. Because circulation flow rate depends on many other factors (number of nozzles, nozzle height, shape of vacuum vessel, etc.), the empirical Eq. (23) has limitations when used generally.

However, when using this numerical model, the effects of the operation conditions, such as gas flow rate, the melt quantity, vacuum level, snorkel submerged depth etc. and the shape conditions such as the shape of vacuum vessel, the number of nozzle in addition to the shape of snorkel etc.
on circulation flow rate, can be evaluated quantitatively. Therefore the numerical model can overcome the limitations of previous empirical equations such as Eqs. (22), (23) and give greater understanding of the RH operation systematically.

4.5. Applications of the Numerical Simulations: Effect of Gas Flow Rate on the Circulation Flow Rate

Using the numerical simulation, the effect of Ar gas flow rate on the circulation flow rate was investigated in a real RH system. Figure 10 shows circulation flow rates of the melt at different Ar flow rates. The vacuum vessel pressure was 10 torr, the snorkel diameter was 60 cm and gas flow rate was increased from 100 Nm\(^3\)/hr to 260 Nm\(^3\)/hr.

As the gas flow rate increased, the circulation flow rate also increased. However, there exists a saturated circulation flow rate at 220 Nm\(^3\)/hr as shown in Fig. 10. If the gas flow rate is increased over 220 Nm\(^3\)/hr, the circulation flow rate decreases. This behavior was reported in a water model study\(^3\) and in a real RH process.\(^4\) Empirical equations such as Eq. (23) cannot explain this behavior.

This can be explained as following: as the gas flow rate increases over the saturated gas flow rate, the superposed volume of plumes increase. Therefore, total plume volume (\(V\)) decrease, though the volume of a plume by one nozzle which is calculated using Eq. (18) increase. On the other hand, the average gas volume ratio increases as gas flow rate increase. However, the effect of increase of average gas volume ratio (\(\alpha\)) on driving force for melt circulation is even less than the effect of decrease of total plume volumes.

Therefore, using the numerical model, the optimum gas flow rate can be calculated and the effects of other conditions on circulation flow can also be investigated in order to obtain the optimum conditions for a RH operation. Furthermore, it may be useful for designing new RH systems.

5. Conclusion

In this study, a simulation program which can calculate the fluid flow in a RH system was developed. Using this numerical model, this study has found that:

1. A model for the plume shape was established and the plume volume and gas volume fraction in plume zone were calculated.
2. The fluid flow in a RH system could be simulated through calculating the force that the gas exerts on the melt.
3. The result of this numerical model was compared with the experimental results and good agreement was shown.
4. This numerical model can predict the decreasing tendency of the circulation flow rate when the gas flow rate exceeds the saturated value.

REFERENCES