Numerical Study of Fluid Flow and Heat Transfer Behaviors in a Physical Model Similar in Shape to an Actual Glass Melting Furnace and Its Experimental Verification

Chien-Chih Yen and Weng-Sing Hwang*

Department of Materials Science and Engineering, National Cheng Kung University, Tainan, Taiwan, R. O. China

In this study, a three dimensional numerical model based on the SOLA-VOF method, which incorporates a Quasi-two Phase method to consider the gas bubbling phenomena, was developed to investigate the fluid flow and heat transfer behaviors in a glass melting furnace. A reduced physical model with heating electrodes and air bubbling devices was also constructed to validate the numerical model. The reduced physical model was made of an acrylic tank similar in shape to an actual glass melting furnace, but reduced to one-tenth in size. The gas flow rate was determined at $6.67 \times 10^{-7} \text{Nm}^3/\text{s}$ by the similarity conversions. The electrode temperatures were set between 298 K to 373 K. The flow trajectories of tracer particles and temperature field were measured to validate the accuracy and reliability of the numerical model. The results show that the temperature and trajectories of tracer particles predicted by the numerical model were consistent with experimental observations/measurements from the physical model. [doi:10.2320/matertrans.M2010215]

(Received June 24, 2010; Accepted August 4, 2010; Published September 25, 2010)

Keywords: numerical model, glass melting furnace, solution algorithm-volume of fluid, marker-and-cell (MAC), physical model, gas bubbling, electrode heating

1. Introduction

The quality requirement for glass substrates in liquid crystal displays (LCD) is very stringent, especially for the alkali-free glass process. The production of glass is a very complicated process, which involves a number of physical and chemical phenomena. However, the quality of glass is decisively influenced by the fluid flow and heat transfer behaviors of the molten glass in the tank. Therefore, understanding of the fluid flow and heat transfer behaviors of molten glass is a significant issue. The conventional approach to studying the effects of the design and operating conditions of a glass melting furnace is to employ an empirical method based on the measurements and observations of an actual furnace. However, it is extremely difficult to observe the glass flow patterns and temperature fields inside a furnace because of the high temperature environment. Full-scale experiments in an actual furnace are thus very costly and dangerous. In addition, although the information obtained for the molten glass is direct, it is also incomplete.

Due to the fast development of computer technology, numerical modeling for the actual glass melting furnaces is becoming a valuable tool for improving both their design and operation. Viscontti reviewed the three-dimensional mathematical modeling of glass melting furnaces and compared the advantages and disadvantages of physical and mathematical models. Choudhary outlined the significant advances in the mathematical modeling of flow and heat transfer phenomena in glass furnaces, describing developments in both the fundamental/scientific and practical aspects of modeling. Ungan et al. developed a mathematical model to predict the effects of electric boosting and air bubbling separately on circulation and heat transfer in a glass-melting tank. Dzyuzer investigated the effect of the profile of the floor on hydrodynamics and heat exchange in the melting tank of a glass furnace. The effects of the length and depth of the tank on the formation of convection currents and heat exchange in different parts of the tank were established. Schill et al. used two methods, coupled and decoupled, to solve differential equations and discuss various important phenomena, including heat and mass transfer as well as melting kinetics for a real glass furnace. However, the mathematical models in previous studies were mostly focused on traditional glass processes, and few of them considered the effects of electrode heating and gas bubbling at the same time. In addition, the validation of the physical models for the mathematical models in previous studies was deficient. Therefore, this study mainly focuses on an alkali-free glass furnace that produces glass used in LCD. A three dimensional numerical model based on the SOLA-VOF method was developed to investigate the fluid flow and heat transfer behaviors in the glass tank. This 3-D model can deal with the effects of electrode heating and gas bubbling simultaneously. A reduced physical model was also established according to similarity theory to validate the accuracy and reliability of the numerical model.

2. Mathematical Method

2.1 Description of the physical system

In order to understand the melting process, a schematic illustration of the furnace for alkali-free glass is shown in Fig. 1. Batch materials are fed continuously from the inlet to the batch zone located between the combustion space and molten glass zone. Fuel is burned with air or pure oxygen in the combustion zone to provide heat to the batch blanket for the fusion of the batch materials and to the glass melt surface to obtain a low-viscous melt. Heat from the combustion space can be transferred by radiation, convection of the hot combustion gases, and conduction. Another source of energy to melt the glass batch is an electrode array immersed in the
molten zone. Electric boosting is an effective method for increasing productivity and enhancing glass-melting efficiency. By appropriately arranging the electrodes, glass melt circulation is enhanced as well as lowering the load and temperature on the combustion space to obtain longer superstructure life. Heat flux at the batch-glass melt interface is also increased, and hence a higher pull rate can be obtained. A higher temperature near the electrodes will cause density gradients, driving free convection flows in the affected zone. The convection of mass in the melt is essential for mixing and heat transfer, in particular for melts with high thermal resistance. Therefore, all-electric melting furnace is a new tendency to replace the traditional gas combustion glass furnace in recent years. The bubbling gases move through bubbling tuyeres mounted on the bottom of the tank, which enables agitation by bringing relatively cold glass melt from the bottom to the surface, and hence the surface temperature of the glass melt is decreased. This enhances the circulation in the molten glass zone.

As described above, the fluid flow and heat transfer behaviors play a significant role in a glass melting furnace. The mathematical model should thus be capable of dealing with the free surface flow, radiation, heat transfer, and convection induced by electrode heating and gas bubbling, and the gas-liquid mixture situation. Due to all-electric melting furnaces is a new development tendency recently, this study mainly focuses on the effects of electrode heating and gas bubbling. The effects of radiation for glass furnace were not taken into consideration. As the quality of glass is decisively influenced by the fluid flow and heat transfer behaviors of the molten glass zone, in this study, a three-dimensional numerical model based on the SOLA-VOF method, which incorporates a Quasi-two Phase method to consider the gas bubbling phenomena, was developed to investigate the fluid flow and heat transfer behaviors in the molten glass zone. A schematic diagram of the numerical model is shown in Fig. 2.

2.2 Governing differential equations

In order to solve partial differential equations in the molten glass zone, certain assumptions were made: (1) Molten glass is an incompressible Newtonian fluid; (2) Molten glass enters continuously from the inlets; (3) The properties of fluid are constant except for density; (4) Viscous heat dissipation effects are negligible; (5) Fluid flow in the tank is laminar due to the high viscosity and low velocity of the fluid. With these characteristics, the energy equation can be written as:

\[ \rho_l C_p \left[ \frac{\partial T}{\partial t} + \mathbf{V} \cdot \nabla T \right] = k \nabla^2 T \]  

where \( \rho_l \) (kg m\(^{-3}\)) is the density of liquid; \( C_p \) (J kg\(^{-1}\) K\(^{-1}\)) is the heat capacity; \( T \) (K) is the temperature; \( t \) (s) is the time; \( \mathbf{V} \) (m s\(^{-1}\)) is the velocity; \( k \) (W m\(^{-1}\) K\(^{-1}\)) is the thermal conductivity. The density variations of the liquid are considered by the following equation:

\[ \rho_l = \rho_0 [1 + \beta_l (T - T_0)] \]  

where \( \rho_0 \) (kg m\(^{-3}\)) is the density of liquid at room temperature; \( \beta_l \) (K\(^{-1}\)) is the coefficient of thermal expansion; \( T_0 \) (K) is the reference temperature.

The Navier-Stokes equation of momentum is:

\[ \rho_l \left[ \frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla) \mathbf{V} \right] = -\nabla P + \mu \nabla^2 \mathbf{V} + \rho_l \mathbf{g} \]  

where \( P \) (kg m\(^{-1}\) s\(^{-2}\)) is the pressure; \( \mu \) (Pa s) is the dynamic viscosity; \( \mathbf{g} \) (m s\(^{-2}\)) is the gravitational acceleration; and \( \rho \) (kg m\(^{-3}\)) is the density of liquid, gas or gas/liquid mixture. If the mesh element is pure liquid, \( \rho \) is equal to \( \rho_l \). If the mesh element is pure gas, \( \rho \) is equal to the density of gas. If the mesh element is gas/liquid mixture, \( \rho \) is equal to the density of gas/liquid mixture. The density of the gas-liquid mixture was calculated by the weighted average method. The continuity equation (conservation of mass) for incompressible flow is:

\[ \nabla \cdot \mathbf{V} = 0 \]  

From the law of mass conservation, the volume fraction of fluid, \( F \), is governed by the following equation:

\[ \frac{\partial F}{\partial t} + (\mathbf{V} \cdot \nabla) F = 0 \]  

With the VOF method, a field variable, \( F(x, y, z, t) \), is designated to each computational element to indicate the volume fraction of liquid in that particular cell. When \( F \) is equal to 1, it means the cell is full of liquid. When \( F \) is equal to 0, it means the cell is full of gas (or empty of liquid). When \( F \) is between 0 and 1, the element contains both liquid and gas and an interface is then allocated in that particular cell. The \( F \) value can thus indicate the domain of fluid flow, and it is a step function.
2.3 Boundary conditions

The boundary conditions include (1) no-slip condition at the walls, (2) free surface condition, (3) temperature condition. Let \( u_n \), \( u_m \) and \( u_m2 \) denote the normal and tangential velocities to the boundary respectively. Then, a no-slip boundary can be written as:

\[
\begin{align*}
  u_n &= 0 \\
  u_m &= 0 \\
  u_m2 &= 0
\end{align*}
\]

The approximate boundary conditions on the free surface\(^1\) are

\[
\begin{align*}
  n \cdot \tau \cdot n &= 0 \\
  m1 \cdot \tau \cdot n &= 0 \\
  m2 \cdot \tau \cdot n &= 0
\end{align*}
\]

where \( \tau = \tau_{ij} \) is the stress tensor given by

\[
\tau_{ij} = -P\delta_{ij} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right), \quad i, j = 1, 2, 3
\]

and \( \vec{n} = (n_1, n_2, n_3) \) is the local unit normal vector; \( m1, m2 \) are local tangential vectors; \( P \) (kg m\(^{-1}\) s\(^{-2}\)) is the pressure; and \( \delta_{ij} \) is the Kronecher delta. \( \delta_{ij} \) is equal to 1 as \( i = j \) and \( \delta_{ij} \) is equal to 0 as \( i \neq j \).

The temperature boundary conditions at the interfaces between the molten glass and atmosphere in the furnace as well as furnace walls and ambient environment are as follows.

\[
-\frac{k}{\delta n} \frac{\partial T}{\partial n} = (T - T_{\infty})
\]

Subsequently the virtual temperature of the mesh, \( T_\varphi \) (K), which is close to the free surface mesh or wall mesh in any direction, can be derived from the equation above and results in the differential equation, as follows.

\[
T_\varphi = \frac{\left( \frac{k}{\delta n} - \frac{h}{2} \right) T_{i,j,k} + hT_{\infty}}{\left( \frac{k}{\delta n} + \frac{h}{2} \right)}
\]

where \( k \) (W m\(^{-1}\) K\(^{-1}\)) is the thermal conductivity; \( h \) (W m\(^{-1}\) K\(^{-1}\)) is the heat transfer coefficient; \( T_{\infty} \) (K) is the temperature of the furnace atmosphere or ambient temperature outside the walls; and \( \delta n \) is the mesh size in \( x, y \) or \( z \) directions.

2.4 Treatments for the gas bubbling

The Quasi-two Phase method was used to consider the rising gas-liquid mixture as a homogenous fluid of variational density. A buoyancy force of gas bubbles was introduced by the density differences among pure gas, glass-liquid mixture, and pure liquid. Surface tension effects were not yet taken into consideration. In this study, a spherical gas bubble was released from each of the six tuyeres once every three seconds, and this frequency was obtained from experimental observations. The radius of the released bubble, \( R_b \) (m), was then calculated by the following equation:

\[
R_b = \frac{Q_b}{4\pi \cdot \frac{t_b}{3}}
\]

where \( Q_b \) (m\(^3\) s\(^{-1}\)) is the flow rate of gas bubbles, and \( t_b \) (s) is the release time of gas bubbles. As the bubble size is significantly larger than that of the mesh size, the mesh element specified can then be pure liquid, pure gas, or a gas-liquid mixture. The density of the element was then determined accordingly. The density of the gas-liquid mixture was calculated by the weighted average method. The gas/liquid interface and the motions of gas bubbles were considered using a marker-and-cell (MAC\(^12\)) technique, and the modeling method is shown in Fig. 3. In this study, the virtual markers that were used to deal with gas bubble mixture are referred to as bubble markers. First, the location of the bubble should be determined, and the radius of the bubble can be calculated by the method mentioned above, as shown in Fig. 3(a). Secondly, the elements covered by the bubble were divided into equal parts, as shown in Fig. 3(b), and virtual bubble markers were then placed at these dividing parts, as shown in Fig. 3(c). Finally, the location of all the bubble markers was checked and those that were outside the bubble were eliminated, as shown in Fig. 3(d). The gas/liquid regions can then be obtained. The velocity of a bubble marker was calculated by the weighted average method according to the nearest eight velocities of elements. The new interface and bubble shapes in the next time step can then be determined. The motion of gas bubbles is shown in Figs. 4(a) to 4(n).

2.5 Treatments for particle tracing of fluid

In order to understand the fluid flow behavior in the glass furnace, the MAC technique was also used to observe the fluid flow trajectories from inlet to outlet. In this study, the virtual markers that used to deal with the flow trajectories are referred to as fluid tracer particles. As the steady state flow field without gas bubbling or the transient flow field of gas bubbling were obtained for the furnace operation in the numerical model, the temperatures and the velocities of the fluid flow in all three directions in every grid cell can be...
known. The fluid tracer particles were placed underneath the inlet as the temperatures and the velocities of the fluid flow were obtained. The fluid tracer particles were then allowed to flow with the fluid, and the particle flow path can be traced by the MAC technique.

3. Experimental Method

3.1 Experimental setup

A one-tenth-sized physical model with an air bubbling and electrode heating system, as illustrated in Fig. 5, was established to study the fluid flow and heat transfer behaviors in the glass furnaces and validate the numerical model. In order to observe the flow behaviors, the tank was made of transparent acrylic, with the dimensions of 0.9 m in length, 0.29 m in width and 0.15 m in height. A row of six gas blowing tuyeres; 0.04 m in diameter, was installed at the bottom along the center line of an actual glass furnace. Since the physical model is one-tenth the scale of the actual furnace, the diameter of the tuyere in the physical model is 0.004 m. Compressed air at room temperature was used as the bottom blown gas. Four rows of electrodes with three electrodes on the first three rows and four electrodes on the fourth row were installed on the bottom and connected to a heating control system. Each electrode was 0.0075 m in diameter and 0.045 m in height, and attached to a thermocouple. This corresponds to actual electrodes of 0.075 m in diameter and 0.45 m in height in the actual furnace. Five thermocouples were installed on the top of the physical model, which could measure the surface or internal temperature of the liquid inside. Silicon oil with a viscosity of 20 Pa·s was selected to simulate the molten glass. Acryl particles with a density similar to silicon oil were used as tracers. Tracers were inserted at the inlet at steady state, and the flow behaviors were recorded by a video camera.

3.2 Liquid flow rate conversion

Due to the high viscosity of glass melt, the Reynolds number (Re) is used to convert the fluid flow rate between the physical model and glass furnace. Applying a Reynolds criterion between the physical model and glass furnace, this indicates that

$$\text{Re}_{\text{pm}} = \text{Re}_{\text{gf}}$$

Then

$$\frac{Q_{\text{pm}} \cdot \rho_{\text{pm}}}{\pi \cdot r_{\text{pm}}^2 \cdot \mu_{\text{pm}}} = \frac{Q_{\text{gf}} \cdot \rho_{\text{gf}}}{\pi \cdot r_{\text{gf}}^2 \cdot \mu_{\text{gf}}}$$

(13)

$Q_{\text{pm}}$ can then be calculated in the following manner.

$$Q_{\text{pm}} = \left( \frac{L_{\text{gf}}}{L_{\text{pm}}} \right) \left( \frac{r_{\text{pm}}^2}{r_{\text{gf}}^2} \right) \left( \frac{\rho_{\text{gf}}}{\rho_{\text{pm}}} \right) \left( \frac{\mu_{\text{gf}}}{\mu_{\text{pm}}} \right) \cdot Q_{\text{gf}}$$

(14)

where $Q_{\text{pm}}$ is the Reynolds number of the physical model; $\text{Re}_{\text{gf}}$ is the Reynolds number of the glass furnace; $Q_{\text{pm}}$ (m³·s⁻¹) is the flow rate of silicon oil; $Q_{\text{gf}}$ (m³·s⁻¹) is the flow rate of molten glass; $L_{\text{pm}}$ (m) is the characteristic length of the physical model; $L_{\text{gf}}$ (m) is the characteristic length of the glass furnace; $\rho_{\text{pm}}$ (kg·m⁻³) is the density of silicon oil; $\rho_{\text{gf}}$ (kg·m⁻³) is the density of molten glass; $r_{\text{pm}}$ (m) is the radius of the tuyere in the physical model; $r_{\text{gf}}$ (m) is the radius of the tuyere in the glass furnace; $\mu_{\text{pm}}$ (Pa·s) is the viscosity of silicon oil and $\mu_{\text{gf}}$ (Pa·s) is the viscosity of molten glass.

3.3 Gas flow rate conversion

In order to correlate the gas flow rate between the physical model and the actual glass furnace, the modified Froude number ($N'_{Fr}$) was used to convert the gas flow rate. Applying a modified Froude criterion between the physical model and the actual glass furnace, the following relationship has to be obeyed.

$$N'_{Fr_{\text{pm}}} = N'_{Fr_{\text{gf}}}$$

(15)

Then,

$$\frac{\rho_{\text{air}} Q_{\text{air}}^2}{\rho_{\text{gf}} \left( \pi r_{\text{gf}}^2 \right) \cdot L_{\text{gf}} \cdot g} = \frac{\rho_{\text{air}} Q_{\text{air}}^2}{\rho_{\text{pm}} \left( \pi r_{\text{pm}}^2 \right) \cdot L_{\text{pm}} \cdot g}$$

(16)
4. Results and Discussion

As the flow field and temperature field were obtained for the physical and numerical model, fluid and acryl tracer particles were placed underneath the inlet to observe the fluid flow pattern. For numerical simulations, the computational domain is shown in Fig. 2 with the related sizes depicted. A uniform mesh was adopted, and the domain was divided into a rectangular mesh system of 33 × 19 × 93.

4.1 Effects of electrode heating

4.1.1 Effects of electrode heating on heat transfer behavior

For the heat transfer behavior, the 3-D temperature contours at the electrode temperature of 373 K by numerical modeling for the investigated region shown in Fig. 6(a) are shown in Fig. 6(b). Figure 6(c) is the 2-D sectional view of Fig. 6(b). The marked positions 1, 2, and 3 in this figure are 0.04 m above the bottom. In the physical model, the temperatures of the silicon oil, which are located in positions 1, 2, and 3 as marked in Fig. 6(c), at the electrode temperature of 373 K were also measured. From the numerical results of Fig. 6(c), the calculated temperatures at positions 1, 2, and 3 is approximately 343 K, 346 K and 342 K. From the measurement of experimental results in the physical model, the temperature of the silicon oil at positions 1, 2, and 3 is 340 K, 345 K and 342 K, respectively. These experimental temperature results are close to the temperatures calculated from the temperature field by numerical modeling. The parameters that used in this section are listed in Table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet velocity</td>
<td>12.916 × 10⁻⁴ m⁻¹ s⁻¹</td>
</tr>
<tr>
<td>Outlet velocity</td>
<td>6.458 × 10⁻³ m⁻¹ s⁻¹</td>
</tr>
<tr>
<td>Overflow velocity</td>
<td>6.458 × 10⁻³ m⁻¹ s⁻¹</td>
</tr>
<tr>
<td>Air flow rate</td>
<td>0 N m⁻³ s⁻¹</td>
</tr>
<tr>
<td>Thermal conductivity</td>
<td>0.5 W m⁻¹ K⁻¹</td>
</tr>
<tr>
<td>Heat capacity</td>
<td>1800 J kg⁻¹ K⁻¹</td>
</tr>
<tr>
<td>Heat transfer coefficient between liquid and air</td>
<td>45 W m⁻² K⁻¹</td>
</tr>
<tr>
<td>Heat transfer coefficient between liquid and wall</td>
<td>1 W m⁻² K⁻¹</td>
</tr>
</tbody>
</table>

4.1.2 Effects of electrode heating on flow behavior

For the effects of electrode heating process condition, the flow fluid behaviors were observed at the electrode temperatures of 298 K and 373 K without gas bubbling. From the numerical model, Fig. 7 shows the 2-D section view of the steady-state velocity field in the vicinity of the electrodes at 298 K without heating or bubbling. It can be seen that the velocity vector in the tank is horizontal toward the outlet. This is believed to be due to the absence of a stirring mechanism, as such free convection induced by electrode heating or strong forced convection induced by gas bubbling. After the fluid tracer particles were placed underneath the inlet, their flow paths, as shown in Fig. 8, can then be obtained from the steady-state velocity field. Figure 8 shows the flow paths of fluid tracer particles in the tank. As can be seen from this figure, the flow trajectory of fluid tracer particles moved horizontally along the free surface to the outlet, and did not shift much in the y and z directions. This could be due to the absence of a stirring mechanism, as mentioned above. The corresponding experimental observa-
tions are shown in Fig. 9, which reveals the flow paths of acryl tracers in the physical model at 298 K without heating or bubbling. It can be seen that the acryl tracer particles move along the free surface from inlet to outlet, as shown in Figs. 9(a) to 9(c). Acryl tracer particles did not shift much in the y and z directions. This is believed to be due to the high viscosity of silicon oil and the absence of a stirring mechanism in the physical model. From Figs. 8 and 9, it can be seen that the simulated results are consistent with the experimental observations.

With regard to the effects of the electrode heating process condition, it can be observed that the fluid close to the electrode would rise upward due to free convection induced by the heating electrodes. The free convection induced by the heating electrodes would cause small-scale circulations near certain electrodes. From the results of the numerical model, Fig. 10 shows the 2-D section view of the velocity field in the region between the inlet and the first row of electrodes at the electrode temperature of 373 K. As can be seen from this figure, there was an upward velocity field near the electrode, and this would drive the nearby flow of fluid upwards to the surface and hinder the forward velocity from the inlet. Consequently, the fluid in the vicinity of the inlet started to flow downward, and therefore induced small-scale circulation phenomena in this region. Figure 11 shows the 2-D section view of the steady-state velocity field in the region between the third row of electrodes and the bubblers at the electrode temperature of 373 K. It can be seen that there was also an upward velocity field near the electrode, and this velocity field would drive the nearby flow of fluid upwards to the surface, and cause the fluid in the vicinity of the bubbler to flow backward to the bottom of the electrode. Consequently, another small-scale circulation

Fig. 8 Schematic illustrations of flow paths of fluid tracers in the numerical model at 298 K without heating and bubbling. (a) Side view; (b) Top view.

Fig. 9 Flow paths of acryl tracers in the physical model at 298 K without heating and bubbling in the vicinity of (a) inlet, (b) bubbling zone, (c) outlet. (The inserts show the sketches of the investigated regions in the physical model)

Fig. 10 (a) Investigated region in the numerical model; (b) 2-D section view of the velocity field in the region between the inlet and the first row of electrodes at the electrode temperature of 373 K.

Fig. 11 (a) Investigated region in the numerical model; (b) 2-D section view of the velocity field in the region between the third row of electrodes and the bubblers at the electrode temperature of 373 K.
occurs because of the velocity field induced by electrode heating.

As the steady state flow and temperature field were obtained, as shown in Figs. 10 and 11, fluid tracer particles were placed underneath the inlet and allowed to flow with the fluid to obtain the flow pattern in the tank, as shown in Figs. 12 and 13. Figure 12 shows the 3-D flow path of one fluid tracer, which was recorded every 100 seconds in the region between the inlet and the first row of electrodes at the electrode temperature of 373 K. Figure 13 shows the 3-D flow path of one fluid tracer, which was recorded every 100 seconds in the region between the third row of electrodes and the bubblers, also at the electrode temperature of 373 K. As can be seen from these two figures, small-scale circulations were observed in these two regions due to the free convection induced by electrode heating. The corresponding experimental observations are shown in Figs. 14 and 15. Figure 14 shows the flow paths of acryl tracers in the region between inlet and the first row of electrodes at the electrode temperature of 373 K. Figure 15 shows the flow paths of acryl tracers in the region between the third row of electrodes and the bubblers at the electrode temperature of 373 K. From the experimental results of the physical model, two small-scale circulations were also observed near the electrodes, the first one was in the region between the inlet and the first row of electrodes, and the other one was in the region between the third row of electrodes and the bubblers. As can be seen from these two figures, acryl tracer particles close to the electrodes would rise upward and small-scale circulations occur due to the free convection induced by the heating electrodes. From Figs. 12 to 15, it can be seen that the simulated results are consistent with the experimental observations.

4.2 Effects of gas bubbling

With regard to the effects of gas bubbling, the flow fluid behavior was observed at the gas flow rate of $6.67 \times 10^{-7}$ Nm$^3$/s without electrode heating. From the numerical results, Fig. 16(b) shows a 2-D section view of the velocity field in the cross-section of the bubbler near the bubbling zone. As the stirring range in the bubbling zone stabilized, a transient flow field as shown in Fig. 16 was selected as the time that spherical gas bubble released from each of the six tuyeres. Fluid tracer particles were then be placed underneath
the inlet and allowed to flow with the fluid to obtain the flow pattern in the tank, as shown in Fig. 17. Figure 17 shows the 3-D flow path of a fluid tracer particles which was recorded every 20 seconds in the region near the bubbling zone, and it can be seen that there was a long range flow circulation in this region. Initially, the fluid tracer particles moved towards the bottom near the third row of electrodes. As the particles tracer approached the bubbler, they rose to the surface and then some of them moved backwards to the third row of electrodes. After several circular flows between the third row of electrodes and the bubblers, the fluid tracer particles moved forward to the fourth row of electrodes at the surface and then flowed several times in circles between the fourth row of electrodes and the bubblers. The moving direction mainly depends on the surface velocity field of the bubbling zone, as shown in Fig. 16(c). From this figure, it can be observed that there were opposing directions of velocities and these were the influential factor that caused the fluid tracer particles to move backwards or forward. Specifically, as the particles passed through the bubbling line, they would move forward, but they would move backwards if they did not pass through the line. The corresponding experimental observation is shown in Fig. 18.
From the experimental results of gas bubbling in the physical model, a long range flow circulation was also observed near the bubbling zone. This circulation would cause some acryl tracer particles to reside inside the bubbling zone for many cycles. From the flow pattern of tracer particles, the acryl tracers initially moved along the free surface from the inlet. In the region shown in Fig. 18(a), they then began to aggregate near the third row of electrodes and move towards the bottom, as shown in Figs. 18(b)–(c). As the tracers approached the bubbler, they rose quickly to the surface due to fluid current induced by air bubbling, as shown in Fig. 18(d). At the surface some of the acryl tracers moved backward to the third row of electrodes, and others moved forward to the fourth row of electrodes as shown in Fig. 18(e). From Figs. 17 and 18, it can be seen that the simulated results are consistent with the experimental observations.

5. Conclusions

In this study, a numerical model for a glass melting furnace was developed, and a reduced physical model with heating electrodes and air bubbling devices was constructed to validate it. The following conclusions are made.

1. With regard to the effects of electrode heating, the simulated temperatures were close to the temperature readings measured in the reduced physical model.

2. With regard to the effects of electrode heating, two small zones of circulation were observed in the vicinity of the electrodes in the numerical model. The first one was in the region between the inlet and the first row of electrodes, and the other one was in the region between the third row of electrodes and the bubblers. These results can be also observed in the reduced physical model.

3. With regard to the effect of air bubbling, a long range circulation was observed between the third and fourth rows of electrodes in the numerical model. These results could be also observed in the reduced physical model.

From the above, it can be found that the results from the numerical and physical models are similar. Therefore, the numerical model developed in this study is considered reliable as a tool to aid the design and operation of a glass melting furnaces.

Acknowledgments

The authors would like to acknowledge the financial supported provided by the National Science Council in Taiwan (NSC 98-2221-E-006-066-MY3).

REFERENCES