Physical and Mathematical Modelling of Inert Gas Shrouding in a Tundish

Kinnor CHATTOPADHYAY, Mihaieia ISAC and Roderick I. L. GUTHRIE

McGill Metals Processing Centre, Department of Mining and Materials Engineering, McGill University, Montreal, Quebec, H3A 2B2 Canada. E-mail: roderick.guthrie@mcgill.ca

(Received on November 16, 2010; accepted on January 13, 2011)

Inert gas shrouding practices in water model tundishes were mathematically simulated using the finite volume based program ANSYS 12. The fluid within the tundish and the ladle shroud was assumed to be Newtonian and incompressible (water), and the flow within the shroud was assumed to be predominantly bubbly i.e. discrete bubbles are formed, and these move down the shroud along with the down-flowing water. Thus, the numerical model was developed using the Discrete Phase Modeling (DPM) approach, along with the standard $k$-$\varepsilon$ turbulence model with two way turbulence coupling. Predicted flow fields, slag behaviour and bubble trajectories were investigated and then compared with experiments in the full scale and one third scale water model tundishes. Various experimental measurements compared well with predictions from the mathematical model, which was shown to slightly over-estimate depths of penetration of the bubble column by 5–15%.

KEY WORDS: physical modelling; mathematical modelling; tundish; inert gas shrouding; Discrete Phase Modelling.

1. Introduction

Mathematical modelling has always been a powerful tool to simulate multiphase gas-liquid flow systems, and inert gas shrouding is a very practical example of that. A number of researchers have reported on modelling gas injection in ladle shrouds and Submerged Entry Nozzles (SEN) entering tundishes and moulds. In early work, D. Bolger et al.\textsuperscript{1)} identified the transitions between bubbly flow, churn-turbulent, and separated flows, as the gas flow injected into the ladle shroud is gradually increased. That work was in relation to the development of a slag detector system for slag carry-over from a ladle into a tundish. In other work, G. M. Evans et al.\textsuperscript{2)} also mentioned that for applications such as SEN’s, too much gas injection can result in a transition from the bubbly flow regime to the churn-turbulent flow regime, which is undesirable. Thus, it is essential to have a scientific understanding of the gas shrouding phenomenon, for ladle shrouds and SEN’s, in order to optimize the process. In this respect, most steel companies practise inert gas shrouding, but are unable to quantify the amount of gas entering the liquid steel versus that leaking from the sealing joints into the surrounding air outside. As such, overly high flow rates of inert gas are possible, and this will be detrimental to steel quality, owing to entrainment of tundish slag around the ladle shroud, or mould powder around the SEN. Bai and Thomas\textsuperscript{3,4)} studied the flow of liquid steel and argon gas bubbles in a SEN during the transfer of liquid steel from the tundish into a mould. They developed a Eulerian multiphase mathematical model, using the finite difference program CFX, and studied 3D turbulent flow of liquid steel with entrained gas bubbles. CFX\textsuperscript{5)} was used to simulate the time averaged flow of argon bubbles within the liquid steel. In CFX modeling, each phase has its own set of continuity and momentum equations. Coupling is achieved through an empirical inter-phase drag between liquid steel and argon bubbles. The model predictions agreed both quantitatively and qualitatively with measurements conducted using Particle Image Velocimetry (PIV) on a 0.4-scale water model. In the work of G. M. Evans et al.,\textsuperscript{2)} the authors developed a model based on a one-dimensional drift flux analysis, and a critical Weber number for stable bubble size. Their model was used to predict the bubble size and gas void fraction as a function of the gas and liquid flow velocities within the bubbly flow regime. The model could also be used to predict the gas and liquid flow conditions at which the transition from bubbly to churn-turbulent flow occurs. Wang et al.\textsuperscript{6)} developed a mathematical model for inclusion removal in molten steel by inert gas shrouding. They discussed the probabilities of collisions and subsequent adhesion of bubbles with inclusions, and pointed out that bubble adhesion is an important parameter for inclusion removal. They also mentioned that finer bubbles and larger inclusions are more favourable for inclusion removal. Zhang et al.\textsuperscript{7)} reported on the effect of nozzle diameter on bubble sizes formed in a water model. The finer the injection nozzle, the smaller are the bubbles formed. Hae-Geon Lee and co-workers\textsuperscript{8–10)} investigated the removal of inclusions from “liquid steel” using “fine” gas bubbles in a water model. The bubbles were created by injecting air into a model ladle shroud, immediately below the slide gate. They reported that the governing factor is one of wettability, as quantified by the contact angle of the inclu-
sion with the water, the larger the contact angle, the higher was the efficiency. They discussed bubble formation and dispersion in the model ladle shroud, and mentioned that bubbles in the size range 0.5–1 mm can be formed in water. However, no exact measurement of bubble size is given in their work, nor has there been any comparison to bubbles formed in steel melts. In summary, no researchers have reported on the trajectory of these bubbles within a water model of a steel tundish, nor how they affect the liquid flow field within the model tundish. Similarly, the formation of an exposed ‘eye’ of hot steel and/or slag, due to gas injection through the shroud, has not been considered in any of the work mentioned above.

Chattopadhyay et al.\textsuperscript{11)} have reported on the effect of inert gas shrouding on fluid flow within the tundish, and associated slag movements. For this, they used a 2D mathematical model. However, a 2D numerical model has definite drawbacks versus a 3D model, especially when modelling a three dimensional delta-shaped tundish. Similarly, turbulence is three dimensional. Perhaps the only advantages are the shorter computational times, and its relative ease. For the present computations, the ANSYS 12.0 package was used, and the results compared with water model experiments, using a full-scale delta shaped tundish, and a third scale equivalent.

2. Mathematical Modelling

The following steps were involved in the mathematical modelling using FLUENT ANSYS 12.0:\(a\) drawing of the tundish (the calculation domain) using basic CAD tools,\(b\) discretization of the whole domain into tiny tetrahedral cells (commonly called meshing),\(c\) exporting the mesh to an appropriate CFD package,\(d\) setting up of material properties and boundary conditions,\(e\) setting up of solution parameters,\(f\) iterative solution of the discretized equations until convergence is achieved,\(g\) post processing and graphical representation.

The mathematical model of the two phase gas/liquid flows in the ladle shroud was based on the following assumptions:

1. The liquid passing down the shroud (water) is an incompressible, Newtonian fluid.

2. The two phase flow within the shroud is predominantly bubbly; discrete bubbles are formed at the entry location, which then are evenly distributed as they move down the shroud, in conjunction with the down-flowing water. This assumption of bubbly flow holds true because the volume fraction of gas in the shroud was not greater than 10%. Above that, gas curtains are formed instead of gas bubbles and hence the discrete phase approach is incorrect.

The standard \(k-\varepsilon\) model of Launder and Spalding\textsuperscript{12)} was used, coupled with the discrete phase model. Two-way turbulence coupling was used to capture the effects of the discrete phase on the primary liquid phase. In the \(k-\varepsilon\) model, \(k\) is the kinetic energy of turbulence per unit mass of fluid, while \(\varepsilon\) is the rate of turbulence energy dissipation. Thus,

\[
k = \frac{1}{2} \sum u_i^2 \quad \varepsilon = \frac{u_i}{Re} \quad \frac{Dk}{Dt} = \frac{\nu}{\sigma_k} \nabla^2 k + G_k - \varepsilon \quad \frac{D\varepsilon}{Dt} = \frac{\nu}{\sigma_\varepsilon} \nabla^2 \varepsilon + \frac{\varepsilon (C_k G_k - C_\varepsilon \varepsilon)}{k} \quad (2)
\]

Here \(G_k\) is the rate of production of \(k\) and is given by the following equation:

\[
G_k = \nu \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] \frac{\partial u_i}{\partial x_j} - \left[ \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} \right] \quad (3)
\]

The turbulent and effective viscosity is calculated by the following equations:

\[
\mu_t = C_\mu \rho k^2 \quad \varepsilon \quad \mu_{\text{eff}} = \mu + \mu_t \quad (5)
\]

The recommended values of the constants adopted in this study were \(C_\mu=1.44, C_\varepsilon=1.92, \sigma_k=1\) and \(\sigma_\varepsilon=1.3\) as proposed by Launder and Spalding.\textsuperscript{12)}

In the discrete phase model, the particle trajectories are computed in a Lagrangian frame of reference. The equations solved in addition to the basic conservation equations in FLUENT ANSYS12, are as follows.\textsuperscript{13)}

\[
\frac{du_p}{dt} = \frac{18 \mu C_p Re}{24 \rho_p d_p^2 u_{rel}} u_{rel} + \frac{g (\rho_p - \rho_p)}{\rho_p} + \frac{1}{2} \rho \frac{d}{dt} u_{rel} \quad (7)
\]

\[
Re = \frac{\rho d_p}{\mu} \left| u_{rel} \right| \quad (8)
\]

\[
u_{rel} = u - u_p \quad (9)
\]

\[
C_d = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \quad (10)
\]

An appropriate derivation of Eq. (7) is given in Appendix I. In the two way turbulence coupling, as the trajectory of a particle is computed, a track of the momentum gained or lost by the particle stream that follows that trajectory is kept, and these quantities are then incorporated in the subsequent continuous phase calculations. Thus, while the continuous phase always impacts the discrete phase, we can also incorporate the effect of the discrete phase trajectories on the continuum phase. This two-way coupling is accomplished by alternately solving the discrete and continuous phase equations, until the solutions in both phases have stopped changing.\textsuperscript{14)}

As noted, the present calculations were carried out using the ANSYS12 package. The drawing of the system and meshing were performed using GAMBIT 2.4.6. Calculations were performed in one half of the tundish, assuming a vertical symmetry plane between the two sides of the tundish, and ignoring any potential large scale transient turbulent motions, for which the RANS model of turbulence would have been necessary. This was done to save on computational time. All velocity components were set to zero at the walls by using the no-slip boundary condition. At the free surface of the liquid, the shear stress was set as zero. The primary fluid was taken to be water and the discrete
phase material was set as air.

In the numerical modeling, the steady flow field solution was first obtained in the absence of any gas injection. After obtaining the steady state velocity field, gas injection was started under transient conditions, using a time step size of 1 millisecond (\(10^{-3}\) s).

During the initial steady state calculations, the SIMPLE\(^{15,16}\) algorithm, along with a first order upwind scheme for momentum, \(k\), and \(\epsilon\) equations, was used. Default values of the under relaxation factor were used i.e. unity for body forces, density and turbulent viscosity; 0.8 for the \(k\) and \(\epsilon\) equations; 0.7 for the momentum equation. For pressure, the standard scheme was used, with an under relaxation of 0.3. After obtaining the steady state velocity field, the inert gas injection was started, at which point the under relaxation factors for \(k\) and \(\epsilon\) were changed to 0.5. This value was finalised, after having performed many prior convergence tests.

3. Physical Modelling

In the complementary physical modeling research, a full-scale water model and its one third scale equivalent was used to simulate the gas-shrouding process. Liquid steel was replaced by its low-temperature aqueous analogue (water), while argon was substituted by compressed air. Helium would be a better option over compressed air, but since the latter is much cheaper, it has been used. A schematic diagram of our full-scale model and its dimensions are shown in Figs. 1 and 2. The square tank above the tundish was used to provide the 3-m head pressure of water, and was used to control the inflow rate of water into the tundish. A flow rate of 0.17 m\(^3\)/min was maintained through the ladle shroud, so as to maintain a steady-state height of 500 mm of water within the tundish. The immersion depth of the ladle shroud was 60 mm. Compressed air was injected from just below the slide gate, at volumetric flow rates ranging between 2 and 10 pct of water entry flows. The flow rate of air was controlled with the help of a flow meter (Cole Palmer Instrument Company, Barrington, IL). The slag phase was simulated using polyethylene beads (density=920 kg m\(^{-3}\), diameter=2.5 mm to 3 mm) that were poured uniformly over the free surface of water in the tundish, prior to the experiment. The thickness of the polyethylene bead layer in the vicinity of the ladle shroud was 0.02 m\(\pm\)0.002 m. For the one third scale model, a flow rate 0.01 m\(^3\)/min was maintained from the inflow, so as to obtain a steady state height of 167 mm, based on Froude \(\left(\frac{U^2}{gL}\right)\) similitude. A 0.01 m thick layer of mineral oil (density=870 kg m\(^{-3}\), viscosity=0.017 Pa.s) was used to simulate the slag phase in the small scale model. High Definition video photography was used to visualize bubble tracks, together with slag layer movements and disruptions.

4. Results and Discussion

Bubble trajectories: The 3D numerical model proved very effective in predicting bubble trajectories within the tundish. The gas flow rate was varied from 4% to 10% of the volumetric flow rate of water. The bubble tracks in the full scale tundish for 6% gas flow are shown in Fig. 3. As seen, at first the bubbles come down the ladle shroud together with the liquid and touch the base of the tundish. Then, due to their buoyancy, they rise almost straight back up to the surface, forming a bubble plume within the tundish around the ladle shroud. These rising bubbles disrupt the protective slag phase (polyethylene particles or mineral oil), creating a reverse flow, and forming an exposed ‘eye’ of water (steel). Figure 4 shows the temporal evolution of the

![Image](https://example.com/image1)

**Fig. 1.** Schematic representation of the full scale water model.

![Image](https://example.com/image2)

**Fig. 2.** Key Dimensions (mm) of the full scale delta-shaped tundish.

![Image](https://example.com/image3)

**Fig. 3.** Numerically predicted temporal evolution of the bubble plume for gas injection at 6% of water entry flows in the full scale tundish.

![Image](https://example.com/image4)

**Fig. 4.** Numerically predicted temporal evolution of the bubble plume for gas injection at 10% of water entry flows.
bubble tracks with 10% gas injection. Here, the greater initial gas flow rate, and the larger buoyancy of the forming column, prevents the initial bubbles from reaching the bottom of the tundish. Also, once steady state conditions are reached, the area over which the bubbles spread out around the shroud is very large, as compared to that for 6%. This is due to the overall buoyancy force, which for 10% gas injection is almost double that of 6%. When the mathematical modelling results are compared with the physical modelling results, the same trend was observed. These are shown in Figs. 5 and 6. The depth of penetration of the bubble column varies with the volume fraction of gas injected. This was measured from the full scale water model experiments and compared with those predicted from the mathematical model. The percentage error was also calculated and was found to be within +15%. Figure 7 shows the numerically predicted maximum depths of penetration of the bubble column (in the full scale tundish), which decreases as the amount of gas injected in the entering liquid is increased. The corresponding experimental and computed penetration values, as a function of the percentage of gas blown, are given in Table 1. As seen, the model tends to slightly over-predict the degree of penetration.

Another interesting observation was made from this 3D

Table 1. Depth of penetration of the bubble column from the free surface of the tundish.

| Volume fraction of gas injected | Experimentally measured penetration depth of the bubble column from the free surface of the tundish (m) | Numerically predicted penetration depth of the bubble column from the free surface of the tundish (m) | Percentage Error |
|---------------------------------|---------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------|------------------|
| 4%                              | 0.5                                                                                               | 0.5                                                                                               | 0%               |
| 6%                              | 0.416                                                                                             | 0.433                                                                                             | +4.09%           |
| 8%                              | 0.324                                                                                             | 0.355                                                                                             | +9.57%           |
| 10%                             | 0.267                                                                                             | 0.294                                                                                             | +10.11%          |

Fig. 5. Bubble plume in the full scale water model for gas injection at (a) 4% (b) 6% (c) 8% (d) 10% of water entry flows.

Fig. 6. Bubble plume in the one third scale water model for gas injection at (a) 2% (b) 4% (c) 6% (d) 8% (e) 10% of water entry flows.

Fig. 7. Numerically predicted penetration depth of the bubble column in the full scale water model for gas injection at (a) 4% (b) 6% (c) 8% (d) 10% of water entry flows.

Fig. 8. (a) Predicted bubble trajectories when micro bubbles of size 150–400 μm are formed in a tundish. (b) Experimentally observed micro bubble cloud movement in the full scale water model tundish.
model. The trajectories of micro bubbles in the size range 150–500 μm are totally different from those of the 3–5 mm diameter gas bubbles. Since micro bubbles have much lower buoyancy forces compared to 3–5 mm sized bubbles in steel, they tend to be in the Stokesian flow regime, with the bubbles fully entrained within the entering flow of liquid. They therefore spread out within the tundish with the liquid (Fig. 8(a)), rather than decoupling to form a rising bubble column. This was not reported by previous researchers. The mechanism of spreading of the bubbles is much clearer from the 3D model than that predicted by our previously developed 2D model. Recall it is probable that this bubble spreading phenomenon could be very beneficial for removing smaller inclusions from steel baths, so as to produce super clean steel. The fluid flow modifications were validated with some water model experiments, in which surfactants were injected simultaneously with compressed air into the ladle shroud, thereby generating 0.05–0.3 mm micro-bubbles. The trajectories of these micro-bubbles were similar to our numerical predictions (Fig. 8(b)), and are quite different from the larger bubbles that are normally associated with gas bubbling in steelmaking operations. For these, the high surfacet tension and non-wetting properties make the stability of injected micro-bubbles most unlikely.

Flow fields: The flow fields predicted from this 3D numerical model are next considered. Figure 9 shows the predicted flow field (ms⁻¹) on the surface of the tundish (around the ladle shroud) with gas shrouding at (a) 4% (b) 6% (c) 10% of water entry flows.

![Fig. 9. Predicted flow field (ms⁻¹) on the surface of the tundish (around the ladle shroud) with gas shrouding at (a) 4% (b) 6% (c) 10% of water entry flows.](image)

![Fig. 10. Velocity field (ms⁻¹) on a plane just beside the shroud measured by 2D PIV.](image)
flow field at the top surface of the tundish with different amounts of gas injected. It is clearly seen that the fluid vectors are pointing away from the shroud and thus reversed flows are formed which tend to push the slag layer away from the shroud. The higher the amount of gas injected, the stronger are these reverse flows. This fact was also validated with 2D PIV experiments in the full scale water model. Reverse flows were detected on a plane just beside the ladle shroud and these are shown in Fig. 10. The velocity values were next compared with those predicted by the 2D numerical model (Fig. 11). As seen, the values are comparable, and are in the range of 0.06 to 0.067 m s\(^{-1}\) from the 2D PIV experiments and 0.0613 to 0.0648 m s\(^{-1}\), as predicted from the 2D numerical model. Figure 12 shows the numerically

Fig. 10. Predicted flow field (ms\(^{-1}\)) on the exposed liquid surface of the tundish (around the ladle shroud) without gas shrouding (no reverse surface flows).

Fig. 11. Velocity field (ms\(^{-1}\)) beside the shroud predicted from a 2D mathematical model.

Fig. 12. Predicted flow field (ms\(^{-1}\)) on the exposed liquid surface of the tundish (around the ladle shroud) without gas shrouding (no reverse surface flows).

Fig. 13. Showing exposed ‘eye’ around the shroud in the full scale water model for gas injection at (a) 4% (b) 10% of water entry flows.

Fig. 14. Showing exposed ‘eye’ around the shroud in the one third scale water model for gas injection at (a) 2% (b) 4% (c) 6% (d) 8% (e) 10% (f) 12% of water entry flows.
predicted flow fields around the ladle shroud on the surface of the tundish without any gas injection. It is seen that the vectors now point towards the shroud and thus prevents the formation of an "eye" around it. The area of the exposed "eye" varies with gas flow rate, and slag thickness, and this is shown in Figs. 13 and 14. Figure 13 represents full scale water modelling with polyethylene beads as the slag phase, whereas Fig. 14 represents one third scale water modelling with mineral oil as the slag phase. It is interesting to note that both systems behave similarly. Initially, without gas injection, the simulated slag phase (polyethylene beads or mineral oil) completely covers the water in the tundish. Then with increasing gas flow rate, the size of the exposed eye increases, and then when gas flow is stopped, they are restored back to their original positions. For low gas flow rates, the reverse flows are not so strong and hence the area of the exposed eye is small. For high gas flow rates, there are stronger reverse flows and so the area of the exposed eye is greater. The size of the exposed 'eye' was measured from the full scale water model and was compared with that predicted numerically. To measure the area of the exposed 'eye' from the numerical model, regions of high surface velocity were identified. For this, we expressed the edge as being located where the velocity was ten times higher than the rest of the tundish (Fig. 15). The area over which this high velocity region extends should correspond to the area of an exposed 'eye', depending on slag thickness. The values of the average radius of the exposed 'eye' are given in Table 2 along with the percentage errors. The error was always less than +15%.

### Table 2. Average radius of the slag 'eye'.

| Volume fraction of gas injected | Experimentally measured average slag 'eye' radius (m) | Numerically predicted average slag 'eye' radius (m) | Percentage Error |
|-------------------------------|----------------------------------------------------|--------------------------------------------------|------------------|
| 4%                            | 0.227                                              | 0.255                                            | +12.55%          |
| 6%                            | 0.3063                                             | 0.323                                            | +5.45%           |
| 10%                           | 0.43                                               | 0.474                                            | +10.23%          |

5. Conclusions

The present 3D numerical model can efficiently predict bubble tracks and flow fields in the tundish, and has been validated with water model experiments. In terms of quantitative measurements, the mathematical model is quite robust and the error with respect to experimental measurements is less than 15 percent in all cases. The 3D model takes into account the delta shape of the tundish, and gives a better picture of the bubble tracks as compared to the previous 2D model. The spread of the bubble column can be correlated to the area of the exposed eye, using the 3D model. This was not possible with the 2D model. While it is very true that the amount of shroud gas should be optimised it is very difficult in practise. From the results, it is seen that at high gas flow rates, the area of the exposed eye is more and so are the chances of greater re-oxidation. Also, higher gas flow rates will increase slag-metal interactions and the slag droplets thereby formed may become entrained into the final product, by passing through the SEN's. An optimum amount of 6% gas by volume injected would be good for plant operations, because the bubble column does not spread too much, and the area of the exposed eye is smaller. Finally, as micro bubbles could be a very efficient means to cleanse the steel of smaller inclusions (<50 μm approx.) during tundish operations, their generation in a real steelmaking tundish, represents a difficult, but rewarding, challenge to steelmakers.
If we start with the familiar form of Newton’s Second Law of Motion, for a submerged particle entrained within a liquid, the acceleration term on the left, is balanced by the sum of the relevant forces acting on the particle, $F_{\text{in}}, F_{\text{fb}},$ and $F_{\omega}$.

\[ m_p \frac{du}{dt} = F_D + F_{\omega} + Fa \]

\[ m_p \frac{du}{dt} = \frac{1}{2} \rho \left[u - u_p \right]^2 C_D A + m_p \frac{g(\rho_p - \rho)}{\rho_p} \]

\[ + \frac{1}{2} \rho \left[u - u_p \right]^2 C_L \frac{d}{dt}(u - u_p) \]

\[ m_p \frac{du}{dt} = \frac{1}{2} \rho \left[u - u_p \right]^2 C_D A + m_p \frac{g(\rho_p - \rho)}{\rho_p} \]

\[ + \frac{1}{2} \rho \left[u - u_p \right]^2 C_L \frac{d}{dt}(u - u_p) \] (Replacing $v_p$ by $\frac{m_p}{\rho_p}$)

This equation states that the acceleration of an entrained particle in a fluid is related to the relative velocity of the particle, to its buoyancy, and to the acceleration of its added mass.