Prediction of Circulation Flow Rate in the RH Degasser Using Discrete Phase Particle Modeling

P. Anil KISHAN and Sukanta K. DASH

Department of Mechanical Engineering, IIT, Kharagpur, India, 721 302. E-mail: sdash@mech.iitkgp.ernet.in

(Received on October 2, 2008; accepted on January 15, 2009)

Conservation equations for mass and momentum with a two equation \( k-e \) model are solved for the continuous phase along with a discrete phase particle modeling (representing gas bubbles) in the RH degasser to predict the circulation flow rate of water in a scaled down model and then the numerical solution has been extended to the real plant case for the prediction of steel circulation flow rate in the actual RH degasser. The prediction of the circulation flow rate of water from the present numerical solution matches reasonably well with that of the experimental observation, taking into account various uncertainties those have been imbedded in the numerical model. RH operation for multi up legs and single down leg for a water model shows that the circulation flow rate falls with the number of up legs and there is an optimum number of down legs for which the circulation flow rate is the maximum for the case of a single up leg. For the actual RH operation in plant it was seen that the circulation flow rate increases with the increase in snorkel diameter and snorkel immersion depth (SID). However, it is apparent that there is existence of optimum SID for maximum circulation flow rate. For different down leg immersion depth the circulation flow rate in the RH depends heavily on the up leg immersion depth. The actual RH operation of the plant for the multi up leg and down leg cases was found to be exactly similar in nature to that of the water model cases.

KEY WORDS: circulation flow rate; multi leg RH; discrete particle modeling.

1. Introduction

Direct study of full RH degasser is difficult to be computed as it involves two phase flow which is conceptually difficult and extremely time consuming to solve on a computer (typical computer time for an actual RH process to achieve steady circulation flow rate is of the order of 20 d). In order to understand the RH degasser operation experiments were conducted with 1/5 to 1/10 water models.\(^1\)\(^-\)\(^5\) In water models, attention was focused on the different parameters such as snorkel diameter, gas injection rate, number of nozzles etc., affecting the liquid circulation rate which provides only flow field and mass flow rates of water. The circulation rate obtained by water models cannot be adopted for the real RH systems as thermal interactions are normally neglected in the model studies.

Mixing time measurements in the real RH degasser was also carried out by few researchers.\(^6\) But carrying the extremely neglected in the model studies.\(^1\)\(^-\)\(^5\) In two dimensional models, the short circuiting of flow between the legs was observed which does not represent the real situation, but the three dimensional models never showed short circuiting. In all these ladle studies, the presence of the vacuum vessel was ignored (RH operation was not taken in to picture) because it was too difficult to simulate the flow in both the ladle and the RH vessel. This made the above studies limited to only ladle portion where the two phase flow simulations were absent.

The gas–liquid two phase flow in the up leg snorkel plays a decisive role in creating the circulation in the RH degasser. As the flow of molten steel in the ladle can not be separated from liquid flows in the vacuum vessel and snorkels, results obtained by the above studies are only qualitative.

Later, some researchers\(^1\)\(^-\)\(^13\) used pseudo single fluid (homogeneous fluid) model to study the full RH system which includes ladle, snorkels and vacuum vessel. In these models the shape and position of gas bubbles/plumes and gas hold up were considered and the buoyancy force was included in the momentum equations as source terms to obtain the circulation flow rate. However, these models could not be generalized in a CFD code to predict the circulation flow rate, because of presence of lot of empiricism in the model.

In order to properly analyze the full RH degasser operation, two phase analysis is needed. The shortcomings arising due to the above methods can be overcome by using multiphase models. There are two main methods used for modeling two-phase flows namely the Eulerian–Eulerian and the Eulerian–Lagrangian approaches. The Eulerian–Eulerian approach is based on the concept of interpenetrating continua, for which all the phases are treated as continuous media with properties analogous to those of a fluid. Both phases are coupled by inter phase forces. From the mathematical modeling point of view both fluids are described by similar equations of conservation. In the Eulerian–Lagrangian approach the particles, bubbles and droplets are considered as discrete phase and the trajectory of each individual particle is tracked in space using the equations of motion based on Newton’s second law. This method is physically realistic than the Eulerian–Eulerian approach for dilute flows.

Recently an attempt was made to analyze the full RH degasser using the Eulerian–Eulerian model\(^14\)\(^,\)\(^15\) and the present authors have made a contribution to it. The thermal and pressure variations on the gas bubbles were neglected in these studies. The flow predicted by these models is similar to those obtained in previous studies and the circulation flow rate predicted for the cold cases could match reason-
ably well with that of the experimental observations. But for the actual RH degasser, this model predicts the circulation rate far away from the actual circulation rate because the thermal effects of the gas expansion could not be taken into account in the model. The reason for this will be discussed later in the text and can be read from the article of the present authors.\(^5\)

All the above studies have so far considered two leg RH systems which consist of one up leg and one down leg. Li and Tsukihashi\(^13\) studied the effect of multi leg RH system on circulation rate. In that study, one down leg was kept at the centre, and three up legs were placed surrounding the down leg at equal angles. The circulation flow rate of multi leg RH system was found to be more compared to conventional RH system.

In the present study, the recirculation in the RH degasser with gas injection was solved using Eulerian–Lagrangian approach which is well suited for dilute flows. In this method, liquid phase is considered to be a continuum and gas phase is treated as dispersed phase in the form of bubbles. Each computational cell contains the continuous and dispersed phase to some fraction. Emphasis was made on the effect of different snorkel sizes, snorkel immersion depths along with multi up legs and down legs for the circulation flow rate of the liquid in the RH system.

2. Governing Equations with the Assumptions

The unsteady mass and momentum transfer of rising gas bubbles to liquid is considered using the following assumptions.

1. The flow is assumed to be isothermal. So the energy equation need not be solved. In reality, the argon gas which is injected into steel attains the temperature like that of steel within a much shorter time compared to its average residence time in the up leg and in the RH vessel.

2. The presence of slag on top surface of the liquid steel in the ladle was neglected in the study.

3. The flow is essentially Newtonian and incompressible with constant properties.

4. Momentum transport between the continuous phase and discrete phase can be represented by the inter phase coupling terms.

5. Bubble–bubble interactions were assumed to be negligible and hence, the drag coefficient correlation, applicable to a single bubble–fluid system, was applied for the appropriate range of Reynolds number.

6. Turbulence is limited to be a property of the liquid phase, and it may be described using the modified two equation \(k–\varepsilon\) model.

7. Discrete mono size spherical bubbles were assumed to form at the nozzle tip. Thus the size of the bubbles forming at the nozzle or orifice was assumed to be known a priori from the available correlation reported in the literature\(^16\)

\[
d_b = 0.35 \left( \frac{Q^2}{g} \right)^{0.2} \quad \text{................................(1)}
\]

8. The size of the bubble was estimated on the basis of the gas flow rate and was considered to remain invariant during its rise through the liquid.

9. Collapse of gas bubbles near the free surface and the splashing due to them is neglected. The free surface of the liquid changes little bit due to splashing and this does not affect the circulation flow rate.

With the above assumptions, the governing equations for continuous phase are written as follows.

2.1. Continuous Phase Equations

The governing equations for the continuous phase of multiphase flows can be derived based on the Navier–Stokes equations for single-phase flows. Considering the existence of dispersed particles, a volume-averaging technique\(^17\) was used to develop a set of partial differential equations to describe the mass and momentum conservation. The volume-averaged forms of incompressible, unsteady conservation equations can be written as\(^16\)–\(^18\)

\[
\frac{\partial}{\partial t} (\alpha_i \rho_i) + \frac{\partial}{\partial x_j} (\alpha_i \rho_i U_i) = 0 \quad \text{...........(2)}
\]

\[
\frac{\partial}{\partial t} (\alpha_i \rho_i U_i) + \frac{\partial}{\partial x_j} (\alpha_i \rho_i U_i U_j) = - \alpha_i \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \alpha_i \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right) - \frac{\partial}{\partial x_j} \alpha_i \rho_i u_i u_j' + \frac{1}{V} \sum F_i + \alpha_i \rho_i \theta_i \quad \text{...........(3)}
\]

The velocity fluctuations are related to the mean stress field through the Boussinesq hypothesis:

\[
rho_i u_i u_j' = 2 \frac{2}{3} \rho_k \delta_{ij} - \mu_i \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad \text{...........(4)}
\]

Based on Newton’s third law of motion, the forces acting on the particles by liquid is equal to the reaction force on the liquid. Therefore, the momentum transfer from particles to liquid is taken into account by adding the particle–fluid interaction force \(\sum F_i\) which can be obtained by summing the forces that are acting on all the particles in the cell by the fluid. This force includes drag force, virtual mass force and other forces acting on the particles by the fluid which is explained in the inter phase coupling.

The 3-D model of unsteady, liquid phase equations must be supplemented with additional relationships to achieve the closure. The closure relationships used in the present model refer to two-phase bubbly flow, with continuous liquid phase and dispersed gas phase.

2.2. Dispersed Phase Equations: Equations of Motion for the Bubbles

The analysis of the bubbles has been carried out via a Lagrangian approach, in which the trajectories of individual bubbles are determined in space and time. The time rate of change of momentum (proportional to mass*acceleration) of a discrete bubble is the result of various forces acting on the bubble. Forces acting on a particle include inter phase forces between fluid and particle and forces imposed by external fields. The appropriate form of Newton’s second law of motion for the bubble, is represented as

\[
m_b \frac{dv_i}{dt} = F_{G,i} + F_{b,i} + F_{D,i} + F_{VM,i} + F_{Int,i} \quad \text{...........(5)}
\]

Right hand side of Eq. (5) represents the forces acting on the particle/bubble which include gravity force, buoyancy force, drag force, virtual mass force, and other fluid–particle interaction forces which include Basset force, Lift forces, Brownian force etc., those were neglected in the present study. Drag force, virtual mass force and other interaction forces are the inter phase forces between the liquid and gas phase. The calculations of different forces are shown below.

The gravity force and buoyancy force on the particle is given as follows.
The position equation for the trajectories of the bubbles are defined as
\[
\frac{dx_i}{dt} = v_i \quad \text{(7)}
\]

2.3. Inter Phase Coupling

The interaction between fluid and particle is known as phase coupling or phase exchange. The coupling of the discrete Lagrangian and continuous Eulerian phases is the most difficult part of the modeling. There are three kinds of phase couplings namely, mass coupling, momentum coupling and energy coupling. Various models have been proposed for these couplings. If the particle concentration is sufficiently low, then the so-called one-way coupling is enough. One-way coupling means that particles are affected by fluid but fluid is not affected by particles. In this case, the theoretical treatment is simple where the equations of fluid motion are same as single phase flows.

As the concentration increases, two-way coupling is needed. In the two-way coupling, both phases are affected by each other. The fluid phase is affected by particles and particles are affected by fluid. As the concentration of discrete phase increases further, in addition to the two-way coupling, we have to consider the particle–particle interaction. In this work, mass and energy coupling are not described anymore because there is no phase change taking place. In the present case, we are using two way coupling where both phases are coupled with each other, because the concentration of bubbles is not insignificant in the medium.

2.4. Types of Coupling

2.4.1. Volume Fraction Coupling

In two phase bubbly flow, the dispersed phase bubbles can become quite large due to coalescence, merging to form extended cylindrical bubbles which can have transverse diameters approaching the pipe diameter. For this reason, the volume occupied by the dispersed particles cannot be neglected. In the present simulations, the diameters of bubbles are not negligible compared to the size of the cell. Therefore volume fraction of the discrete phase bubbles in the computation cells has been considered. The volume fraction in the continuous phase equations results from these spatially distributed discrete bubbles. However, in the continuous phase equations, the volume fraction is a continuous field variable with a spatially smooth distribution, as in the classical two-fluid model.

The volume fraction of dispersed phase is calculated as
\[
\alpha_g = \frac{NV}{V_{\text{cell}}} \quad \text{(8)}
\]

assuming all the bubbles are of uniform size.

And the volume fraction of the liquid phase is calculated from
\[
\alpha_g + \alpha_l = 1 \quad \text{(valid for each computational cell)} \quad \text{..(9)}
\]

2.4.2. Momentum Transfer Coupling

The momentum exchange term in the liquid phase equation of motion can be evaluated from the concept that the inter phase exchange forces experienced by the particles act with equal magnitude but in opposite directions in the liquid phase. The drag force acting on a suspended particle is proportional to the relative velocity between the phases and is given as
\[
F_{G,j} + F_{B,j} = \nu \eta \left( \rho_g - \rho_l \right) \quad \text{(6)}
\]

where \( F_{G,j} \) is the drag force acting on a suspended particle in the direction of the incoming flow, \( F_{B,j} \) is the drag coefficient, which is a function of the particle Reynolds number, \( Re \). For rigid spherical particles, the drag coefficient, \( C_D \), can be estimated by the following equations
\[
C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \quad \text{(11)}
\]

where \( a_1, a_2, \) and \( a_3 \) are constants applicable for smooth spherical particles over several ranges of \( Re \) given by Morsi and Alexander.\(^{20}\)

The forces due to acceleration of the relative velocity can be divided into two parts: virtual mass effect and the basset force. The virtual mass effect relates to the force required to accelerate the surrounding fluid. This force is some times called the apparent mass force because it is equivalent to adding a mass to the sphere. The added mass force accounts for the resistance of the fluid mass that is moving at the same acceleration as the particle. For a spherical particle, the volume of the added mass is equal to one-half of the particle volume, so that
\[
F_{VM} = \frac{1}{2} \rho_l V \left( \frac{du}{dt} - \frac{dv_i}{dt} \right) \quad \text{(13)}
\]

The Basset term describes the force due to the lagging boundary layer development with changing relative velocity. But such forces are normally not present in a bubbly flow, so we do not describe them here. So the inter phase coupling force calls for drag force and virtual mass which has been used through Eqs. (10), (13). The bubble rotation and bubble coalescence during the bubble motion are also ignored.

2.5. Turbulence Modeling

Several alternatives have been proposed to estimate the effective viscosity of turbulent liquid phase in gas–liquid flows. Modified form of standard \( k-\varepsilon \) model performs satisfactorily. The governing transport equations for turbulent kinetic energy, \( k \) and its dissipation rate, \( \varepsilon \) is represented as follows.\(^{16,18,19}\)

\[
\frac{\partial}{\partial t} \left( \alpha_l \rho U_k \right) + \frac{\partial}{\partial x_j} \left( \alpha_l \rho U_k U_j \right) = \frac{\partial}{\partial x_j} \left( \alpha_l \frac{\mu_l}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + \alpha_l \left( -\rho_l \frac{\mu_l}{\sigma_k} \frac{\partial U_j}{\partial x_i} - \rho_l e + P_b \right) \quad \text{(14)}
\]

\[
\frac{\partial}{\partial t} \left( \alpha_l \rho \varepsilon \right) + \frac{\partial}{\partial x_j} \left( \alpha_l \rho U_j \varepsilon \right) = \frac{\partial}{\partial x_j} \left( \alpha_l \frac{\mu_l}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + \alpha_l \left( -C_{\varepsilon 1} \rho_l \frac{\mu_l}{\sigma_k} \frac{\partial U_j}{\partial x_i} + P_b \right) \frac{\varepsilon}{k} - C_{\varepsilon 2} \rho_l \frac{\varepsilon^2}{k} \quad \text{(15)}
\]

The distribution of turbulent viscosity is obtained from
\[ \mu = \rho \xi C_{e} \alpha^{2} / \gamma \]  
\[ \sigma = 0.09 ; \quad C_{e} = 1.44 ; \quad C_{e} = 1.92 ; \quad \sigma = 1.0 \]  
\[ \sigma = 1.3 \]  
\[ \xi = 0.0001 \]  
\[ \gamma = 1002 \]

Where \( C_{p} \), \( C_{e} \), \( \sigma \), and \( \xi \) are constants and their values are as follows. \( C_{p} = 0.09 \); \( C_{e} = 1.44 \); \( C_{e} = 1.92 \); \( \sigma = 1.0 \) and \( \xi = 1.3 \) according to Lauder and Spalding.\(^{21}\)

It is to be noted that the mass conservations (Eq. (2)), momentum conservations (Eq. (3)) and the turbulent \( \kappa \) and \( \epsilon \) equations (Eqs. (14), (15)) are the most general in form. If \( \sigma \) is zero or too small (means no gas bubble being injected to the system or too less gas being injected) then these equations become the governing equations for a single fluid flow in any system.

\( \gamma \) is the production rate of turbulence energy which is due to the interaction between the mean flow field and the bubble motion. In literature\(^{18}\) it was reported that one may neglect the contribution of extra turbulence generation due to the interaction between fluid flow and bubble motion. Therefore, in the present work, \( \gamma \) was set to zero. As a result, the modified \( \kappa-\epsilon \) model reduces to standard \( \kappa-\epsilon \) model.

In the present study, the effect of fluid turbulence on bubble motion is not considered. As a result, average liquid velocity is used in the calculation of source terms instead of instantaneous velocities of the liquid. If the fluid turbulence on particle trajectories is to be considered, then we have to use the instantaneous velocity instead of average velocity in the calculation of the source terms. The instantaneous velocity consists of average velocity and fluctuating component. Calculation of fluctuating components is available in literature\(^{6,18}\) but is not required in the present numerical simulation.

3. Boundary Conditions

No slip boundary condition was used at all the impermeable walls for continuous phase while bubbles were allowed to reflect at these impermeable walls. Wall functions were considered at the wall for computations of near wall \( \kappa \) and \( \epsilon \) values. The gas bubbles enter the solution domain from the nozzles at a rate determined by the gas flow rate. The gas bubbles are allowed to escape from the degasser at the rate at which it reaches the free surface. The free surface of the liquid in the RH was considered to be flat with zero shear stress condition (the free surface being the domain boundary) and the bubbles were allowed to escape from this surface. It is to be mentioned here that the exact shape of the free surface in the RH vessel is immaterial if one wishes to determine the circulation flow rate in the RH system. At the inlet of the nozzle the gas velocity is prescribed because it is known from the experiment and actual operation and the turbulence intensity was set to 2% with a turbulent viscosity ratio of 10 to start with.

4. Numerical Modeling

The equations to be solved are the mass and momentum conservation equations for the liquid (Eqs. (2) and (3)) and the equations which describe the transport of turbulence quantities \( \kappa \) and \( \epsilon \) (Eqs. (14) and (15)). These partial differential equations are coupled with equations of motion of bubbles which are solved to obtain bubble trajectories and velocities, volume fraction and momentum exchange between phases. The momentum equation of liquid phase along with turbulence closure equations are integrated over a control volume and then discretized to yield algebraic equations by using finite volume method. The discretized equations thus obtained were solved by the algebraic multi grid solver of Fluent 6.3 CFD code by incorporating the above boundary conditions. In discretization process, first order upwind scheme was used for convective variables and central differencing for diffusion terms. The combination of discrete particle simulation and continuous phase simulations was achieved in the following way.

First, velocity and pressure fields are assumed throughout the computational domain. The discrete phase bubble velocities for each particle are obtained by integrating the particles equation using Trapezoidal integration. From these the volume fraction and fluid–particle interaction forces in each computational cell are calculated. The continuous phase velocity distribution is calculated by using the momentum source terms (which are the sum of fluid–particle interactions of all particles in each computational cell). The continuity equation is used to correct the pressure field in the domain. The pressure and the velocities of continuous phase are then updated for the next iteration to continue. The solutions of continuous phase equations involve several iterations before convergence for each time step. Once the solution is converged, the positions of particles are determined using the position equation (Eq. (7)) for the next time step. The computation returns to the beginning of the loop where the particle motion equations are integrated. In this way, both dispersed phase and continuous phases are coupled via inter phase coupling. The simulations were carried till steady state conditions for mass circulation in the RH were achieved.

In this way continuous and discrete phases are coupled by inter phase coupling forces while satisfying momentum conservation equations. The convergence of the equations was monitored by whole field residual of each variable and when this variable fell below \( 10^{-3} \) the solution was assumed to have converged for that time step. Steady state solution for the circulation flow rate could be achieved by performing the simulation for more than 10 s which takes about a computer time of more than 20 d.

Near the nozzle inlet, grids were tetrahedral and away from the nozzles hexahedral grids could be placed to have better accuracy. Due to the connection problems of the nozzles, with up leg tetrahedral cells were placed near the nozzles although these type of cells reduce the accuracy of solution to some extent. The nozzle arrangements for different cases are shown in the corresponding figures (Fig. 1, and Figs. 3 and 4). The time step used in the simulation was 0.0001 s during the entire period of simulation. The total number of cells for a particular case was always more than 70 000 with finer cells near the nozzle and in the up leg. Grid independent test was carried for almost all the cases of the water model since the simulation time was limited to only 4 d. But for the case of actual plant RH system it was impossible to carry out grid independent test for all the cases as because each run was taking more than 20 d. Only one grid independent test for the plant RH system was done 90 000 cells and the mass circulation rate was almost same like the case where 70 000 cells were used. The properties of the fluids used in the simulation can be read from Table 1.

5. Results and Discussions

At the beginning of a typical calculation, bubbles of known diameter (from Eq. (1)) were introduced into the domain through 10 nozzles and their position tracked as a function of time till the gas bubbles reached the free surface where these bubbles were allowed to escape from the system. In the present calculations, the bubble size created at the nozzle tip is around 3–6 mm for water–air model and 8–16 mm for steel–argon system depending on the flow rate at ambient conditions. The bubbles drag the liquid along with it inside the up leg. Thus the liquid from the ladle gets sucked in to the RH vessel through the up leg and then returns to the ladle again through the down leg. This way the liquid movement creates mixing in the ladle as well as in
5.1. One Up Leg and One Down Leg for the Water Model

In order to get a numerical confidence we first started a case for a water model experiment. The schematic diagram and details of geometry of the water model are given in Fig. 1(a) and Fig. 1(b). Figure 1(c) shows the nozzle arrangement in the up leg. There are two rows of nozzles in the up leg with 5 nozzles in each row spaced at 72° to each other in a symmetric manner. The circulation flow rate of water was numerically computed from the present CFD computation and has been plotted against the air injection rate in Fig. 2 (triangles are numerically computed and line is a curve fit) where a comparison with the experimental observation can be seen. The experimental measurement was done by adding 50 g of KCl solution to the setup when the setup was almost in steady state. Then the concentration of the tracer (C) (KCl solution) was measured with time, and plotted as C/\(C_\infty\) against time where \(C_\infty\) was the final concentration in the vessel. The peak area method (essential idea of Nakanishi et al.) was used to compute the circulation flow rate, details of which can be seen from Appendix-A. We did not use any velocity measurement in the down leg to compute the circulation flow rate because that would have been a very expensive setup. The height of water in the vacuum vessel was controlled by the amount of vacuum being produced by the pump in the vessel. The experiment was not directly done by us, rather was done in an industrial R&D of Tata Steel. Almost all the data are classified, so we can not show the details of the raw readings.

Only the end result was given to us for comparison purposes. The number and size distribution of bubbles in the experiment could not be measured because any such measurement will call for very elaborate setup which again will increase the cost of the experiment tremendously. The comparison with the experimental observation is not very bad if we think that the governing equations describing the physical process is not all that simple. It can be seen from Fig. 2 that the circulation flow rate has not reached a state of saturation. After reaching a state of saturation the circulation flow rate will decrease with the gas flow rate because at high gas flow rate there will be too many bubbles present in the up leg which will choke the up leg and this will not allow the liquid to rise in to the up leg for which the circulation flow rate will decrease. The present study agrees with the results of Park et al. where there is a saturation value for circulation rate.

5.2. Multiple Up Legs and Multiple Down Legs for the Water Model

Figure 3 shows the diagram for a two up legs and one down leg case with the nozzle arrangement while Fig. 4 shows the diagram for three up legs and one down leg for a scaled down RH system. From Figs. 3 and 4 it can be clearly seen that there is only one row of nozzles used for each up leg instead of two which was used for the one up leg and one down leg RH system. The diameters of the up legs and down legs in multi leg RH system is chosen on the basis of equal cross sectional area for up leg and down leg compared to a two leg RH system (one up leg and one down leg). The same RH system can be interchanged to get new cases like one up leg and two down legs along with one up leg and three down legs. Figure 4(d) shows the velocity vector at a cross section AA passing through the up leg and one down leg. Figure 4(e) shows the vector at a cross section of BB. From the field vector it is clear that there is enough mixing in the ladle as well as in the upper RH vessel. It must be noted here that the RH vessel height may be anything (as seen from Fig. 4) but the field vectors will be only seen up to the height where the liquid is present in the vessel. The height of the liquid in the vessel will depend on the amount of suction being applied to the RH vessel. Figures 5(a) and 5(b) show the field vectors at cross section AA and BB when the RH system has one up leg and three down legs. In any situation the mixing in the ladle is very clearly visible due to the RH operation.

Figure 6 shows (symbols are CFD computation and lines are curve fitted) the circulation flow rate in the water model with multiple up legs and only one down leg. It can be seen from the Figure that with the increase of up legs the circulation flow rate in the system has decreased. The amount of
gas being injected for one up leg system is the same as that for three up leg system in our present computation. But from practical operation point of view the diameter of the up legs has to be reduced. This introduces extra resistance to the flow of gas in the up leg and hence the velocity of gas falls in the up leg as a result and hence the drag force imparted to the fluid on turn also decreases. This is the reason for which the circulation flow rate falls in the RH system with the increase with the up legs. Li and Tsukihashi\(^{13}\) found more circulation rate for three up leg and one down leg RH system in their computational model because their model had incorporated lots of empirical formula from the experiments. With such empiricism the model can not be in general applied to a CFD study with a general CFD package which has been done in the present case.

**Figure 7** shows (symbols are CFD computation and lines are curve fitted) the circulation flow rate in the water model RH system with one up leg and many down legs. To accommodate the down leg in the fixed ladle we have to reduce the size of the down legs when the number of down legs increases (according to the above equivalent cross sectional area concept). When the gas flow rate in to the up leg is fixed the driving force for the circulatory motion in the RH system is fixed. But the circulation flow rate in the RH system is seen to vary with the number of down legs. When the down legs are two, the circulation flow rate is the maxim-
Flow rate through it, resistance depends on the pipe diameter (length of each pipe). The system resistance is the lowest we get highest flow rate. The system resistance rises due to small diameter pipes but smaller in diameter then in which case we get more flow rate? Surely when the system resistance increases to three. This can be explained in the following manner. Through the up leg gas gets pushed into the vacuum vessel and thereby creates a head for the fluid to move in the system. The fluid flow will be highest if the system encounters less resistance. This happens in the case of two down leg system compared to a three down leg system where the system resistance rises due to small diameter down legs being present more in number. This phenomenon can also be thought this way. If we want to discharge water from a dam (on the other side of which there is a constant head of water available) through a pipe or through many numbers of pipes but smaller in diameter then in which case we get more flow rate? Surely when the system resistance is the lowest we get highest flow rate. The system resistance depends on the pipe diameter (length of each pipe) is equal here and also in the RH system) and also on the flow rate through it, via the friction factor being influenced by the Reynolds number. So there are always an optimum number of pipes with specified diameters which gives the highest flow (unless someone totally opens the dam which is not the case we are considering). Our present RH system is exactly a replica of the dam problem where we get highest circulation rate with only two down legs.

5.3. Actual RH System Used in the Plant

After having a water model study done, we extended the numerical simulation to a real plant case. The RH system used in the plant is shown in Fig. 8(a) with all its dimensions and Fig. 8(b) shows the three dimensional view of the system. The objective is to compute the steel circulation flow rate in the RH system. The circulation of steel is shown in Fig. 9 as a function of argon injection rate and snorkel size. It can be seen from Fig. 9 that as the snorkel size increases the circulation flow rate in the RH system increases. When the diameter of the up leg or down leg (snorkel size) increases it offers less flow resistance so the circulation flow increases. But in any case this numerical simulation will not be able to predict the actual circulation flow rate that is encountered in the plant, because argon when injected into the up leg will expand tremendously which will cause more drag force on the fluid and hence the circulation flow rate will be much more in real practice. But the formulation we did here is capable of doing it provided we use the energy equation to be solved and then predict the circulation flow rate. But for the time being this type of computation seems to be almost impossible. The reason for this will be discussed little later from now. But the present model can predict the circulation flow rate for a cold case very well as we have already seen in the case of the water model.

5.4. Effect of Snorkel Immersion Depth

Figure 10 shows the circulation flow rate when the SID is changing from 500 to 1500 mm. When the SID is 1500 mm, the liquid column above the nozzle level to the free surface in the RH vessel is higher than that for the case when the SID is 500 mm. For a particular argon blowing rate the driving force created by the argon bubble to drag the liquid along with it will depend on the height of the liquid column that is available in the up leg (from the nozzle to the free surface in the RH vessel). If the liquid column height is higher, then the bubbles will impart more kinetic energy to the liquid column before they come out at the free surface. This is because the bubbles get enough length to impart the energy to the fluid because they take more time to travel up the liquid column. Although the mass of the liquid column for 1500 mm SID is higher than that for the case of 500 mm SID but it has acquired lot of kinetic energy so it gets more upward velocity which helps it to move up and hence the circulation flow rate becomes higher for the case of 1500 mm SID compared to that for the case of 500 mm SID. For the case of 500 mm SID the length of the liquid column is much shorter compared to that of 1500 mm SID and hence the bubbles get very short time to escape out at the free surface and they can not impart all the kinetic energy to the fluid because their surface escape velocity is much higher here compared to the case of 1500 mm SID. So the circulation flow rate for 500 mm SID remains lower compared to the case of 1500 mm SID for lower argon flow rate. When the argon flow rate increases then the kinetic energy that is imparted by the argon bubbles to the liquid column also increases but this time the liquid in the up leg of the 500 mm SID will acquire higher velocity because of its lower mass (lower column length), so the circulation flow rate at 1600 lpm of argon flow rate is higher for the case of 500 mm SID compared to the case of 1500 mm SID. The circulation flow rate for the 1000 mm SID remains in between the 500 mm and 1500 mm SID till the argon flow rate of 1200 lpm and falls after an argon flow rate of 1400 lpm compared to the case of 500 mm SID.

So far we discussed about the drag force being imparted by the argon bubbles to the liquid column in order to give it a circulation flow rate in the RH system. However, the drag force imparted by the bubbles normally depends on the slip velocity of the bubbles compared to the liquid mass surrounding it. So the slip velocity can be zero when the bubble velocity and the liquid velocity in the up leg are about
the same. Then the circulation flow rate can fall in the system which can be seen to be happening for the case of 1 000 mm SID at 1 600 lpm of argon flow rate.

5.5. Variable Immersion Depth for up Leg and Down Leg

If the down leg immersion depth is varied along with the up leg immersion depth then the circulation flow rate of steel in the RH system can be very peculiar. Such a plot of circulation flow rate at an argon injection of 1 600 lpm is shown in Fig. 11 where the down leg and up leg immersion depth varies. At low down leg SID of 100 to 500 mm, an up leg SID of 500 mm produces highest circulation flow rate compared to the case of 1 000 mm and 1 500 mm of up leg SID. But at 1 000 mm of down leg SID the case just reverses completely and again at a down leg SID of 2 000 mm the case again reverses completely. The up leg immersion depth has very less role to play in the circulation flow rate. It is the down leg which injects the fluid in to the ladle. When the down leg depth is low the jet coming out of it will hit the bottom of the ladle with less strength which will create lesser circulation in the ladle but the jet will get enough suction length in the ladle to suck more fluid with it. So the strength of the recirculation in the ladle will be a combined effect of the impact of the jet and the length of the plume suction created by the jet. So it is expected that the recirculation strength will vary with the immersion depth of the down leg and one can expect maxima and minima on the curve in such type of a combined variation of the circulation strength. On top of this effect, the immersion depth of the up leg will also add some minor effect and hence the curve can be a wave shape curve which we have obtained in Fig. 11.

5.6. Effect of Multiple Up Legs and Down Legs for Plant Size RH Operation

Figure 12(a) shows the plant size RH system with two up legs and one down leg with SID of 500 mm. Figure 12(c) shows the velocity field at a cross section of AA and Fig. 12(d) at a cross section of BB. Both the vector plots show strong circulation in the ladle. Figure 13 shows the velocity field in the RH system with one up leg and two down legs with a SID of 500 mm (this has been obtained by just interchanging the down leg and up legs in Fig. 12). The vector plots at cross sections of AA and BB are shown for this case in Fig. 13(a) and Fig. 13(b).

Figure 14 shows the circulation flow rate of steel versus the argon flow rate when there are two up legs one down leg and one up leg and two down legs. We have discussed earlier for the water model that the circulation flow rate will be less if we increase the number of up legs (argon flow remaining same and up leg size decreasing). Exactly the same phenomenon is repeated here for the circulation flow rate of steel in the plant size RH system which can be seen from Fig.14. We have also seen earlier for the water model that
the circulation flow rate becomes the highest for two down legs and one up leg and again here for the steel circulation rate we see the same phenomenon exactly repeating (Fig. 14).

Figure 15 shows the circulation flow rate of steel for plant size RH when there are two down legs and one up leg. When the down leg size is higher the circulation flow rate of steel is higher. For the higher down leg size with a SID of 1 500 mm the circulation flow rate is again higher compared to lower SID of 500 mm and 1 000 mm.

6. Difficulty to Reproduce Plant Result

In the actual plant practice the circulation flow rate in the RH vessel is much higher compared to what has been presented here in Fig. 14. In the present CFD model the expansion of the argon gas due to sudden heating in the liquid steel is not taken into account due to several difficulties. Had that been taken into account then the circulation flow rate would have been much higher. In order to take the expansion of the gas in the liquid steel one has to solve the energy equation along with the equation of state for the gas which considerably adds to computer time, and was found almost impossible with our computers. The gas within a very short time attains the temperature of liquid steel (about 1 823 K) inside the bath and hence expands to about 15 times inside the liquid bath. So in order to take into account this effect one has to use a very fine grid at the nozzle inlet and use extremely small time step, which considerably increases the computational time and also the computation becomes unstable for even 800 Lt/min of argon flow which is the bare minimum flow for an industrial setup. So it was almost impossible to get a feasible solution with the energy equation being activated along with all other equations in place. Otherwise also one could try to put higher nozzle inlet velocity for argon to take into account higher volume flow of argon inside the bath. This process also calls for instability because the inlet velocities become as large as 250 to 300 m/s at the inlet of each nozzle for the minimum argon flow of 800 Lt/min in a real plant case. So a practical solution of the plant case with or without the energy equation being solved looks very difficult for the time being. However, the present method is a step towards that which can predict at least the circulation flow rate in a RH vessel from first principles with the use of CFD with very little empiricism in the CFD model (only empirical formula being used is the drag law).

7. Conclusions

The conservation of mass and momentum equations (with suitable source terms to represent interfacial drag, lift and virtual mass forces) have been solved numerically with the k–ε model; using an unstructured grid; to predict the flow field and the distribution of the liquid and gas bubbles inside the experimental set up as well as for the RH ladle and vessel. By using the flow field the circulation flow inside the devices could be computed for various gas flow rates. It was found from the present numerical simulation that multiple up legs do not help in augmenting the circulation flow rate in a RH vessel where as one up leg and two down legs help to increase the circulation flow rate. At low argon injection rate the lower SID will produce lower circulation rate but at higher injection rate the lower SID will produce higher circulation rate only for two leg RH system. However, for the multi leg RH system, lower SID may not produce higher circulation flow rate at higher argon flow rate because the circulation flow rate becomes too sensitive to the diameters of the down legs. Circulation flow rate increases with the increase in the size of the down leg and up leg. Although the cold model study produces circulation flow rate closer to the experimental observation but the present model could not produce circulation rate of steel in the RH vessel to the actual observation of the plant due to several difficulties, one of those is the inability to take argon expansion in the RH system. But the present computation is a realistic step towards the exact computation of the plant case because the present simulation is a computation from the first principle without much empiricism imbedded in it.
Nomenclature

- \( A \): Exposed frontal area of the bubble in the direction of incoming flow (m²)
- \( C_p \): Drag coefficient
- \( d_b \): Diameter of bubble or particle (m)
- \( F \): Interaction forces between liquid and gas phases (N)
- \( g \): Acceleration due to gravity (m/s²)
- \( k \): Turbulent kinetic energy (m²/s²)
- \( m_c \): Mass of the bubble (kg)
- \( N_b \): Number of bubbles in the computational cell
- \( P_d \): Production of turbulence due to particle drag (kg/m³/s)
- \( Q \): Gas injection rate (m³/s)
- \( Re \): Relative Reynolds number
- \( U_0 \): Average velocity of liquid (m/s)
- \( U_s' \): Fluctuating velocity component of liquid (m/s)
- \( V \): Volume of the bubble (m³)
- \( V_{cell} \): Volume of the computational cell (m³)
- \( v \): Velocity of bubble (m/s)
- \( x_i \): Position vector of bubble (m)
- \( \alpha_d \): Volume fraction of dispersed phase
- \( \alpha_{lp} \): Volume fraction of liquid phase
- \( \varepsilon \): Turbulent energy dissipation rate (m²/s³)
- \( \mu_l \): Viscosity of the liquid phase (Pa s)
- \( \rho_b \): Density of bubble (kg/m³)
- \( \rho_g \): Density of liquid (kg/m³)
- \( \rho_{ke} \): Density of bubble (kg/m³)
- \( \rho_{ke} \): Volume fraction of dispersed phase
- \( \rho_{ke} \): Volume fraction of liquid phase
- \( \rho_{ke} \): Viscosity of the liquid phase (Pa s)
- \( \rho_{ke} \): Density of bubble (kg/m³)
- \( \rho_{ke} \): Density of liquid (kg/m³)

Appendix A

50 g solution of KCL is introduced to the system by a syringe and the tracer concentration is measured with time by the probe which is inserted to the down leg. The probe actually measures the conductivity of the solution at that point, but the computer converts the conductivity to the concentration by reading a predefined conductivity versus concentration chart which is normally calibrated very carefully. A “C" versus “t" curve can be obtained this way (Fig. A-1). When the system comes to steady state the final concentration in the ladle will be \( C_\infty \) which can be straight read from the “C" versus “t" plot. Another plot for \( C/C_\infty \) versus “t" has to be drawn which will look exactly the same like that of “C" versus “t" but will have different scale on the y axis because the y axis is now divided by \( C_\infty \) (see Fig. A-1)

\[
m \int_0^\infty C \, dt = m_{\text{KCl}} \quad \text{(A-1)}
\]

The net amount of KCL solution added to the system is known and can be found out from Eq. (A-1) where the integration limit for time \( t \) and the circulation flow rate \( m \) are unknown. The \( C \) versus \( t \) curve is generated experimentally and known. The net mass of the water in the RH degasser setup at the beginning of the experiment is \( M_{\text{sol}} \), which is known. The final concentration in the system will be as per Eq. (A-2).

\[
C_\infty = \frac{m_{\text{KCl}}}{M_{\text{sol}}} \quad \text{(A-2)}
\]

If we divide \( C_\infty \) to both sides of Eq. (A-1) then we get (A-3) where the circulation flow rate \( m \) and the limit of integration \( t \) are again unknown but this gives us a relation with \( M_{\text{sol}} \) which is desired so that circulation flow rate can be found out. It is to be noted that Eqs. (A-1) and (A-3) are not separate independent equations, rather (A-3) is a scaled equation of (A-1).

\[
m \int_0^t \frac{C}{C_\infty} \, dt = \frac{m_{\text{KCl}}}{M_{\text{sol}}} = M_{\text{sol}} \quad \text{(A-3)}
\]

So, the task of determining \( t \) remains on the physics of the situation. When the tracer is added to the up leg, it comes to the down leg slowly and its concentration increase with time. But after some time the concentration reaches a lowest value, this means all the tracer have passed the down leg. Then the concentration slowly rises, which means the tracer after passing one loop in the system is again appearing at the down leg. So the point of lowest concentration in the \( C \) or \( (C/C_\infty) \) versus \( t \) plot will signify a time of one circulation in the system. Therefore, \( t_i \) is the time when the system concentration is the lowest.

After knowing \( t_i \) the circulation flow rate \( m \) can be computed from Eq. (A-3) which will be accurate to a large extent. It must be noted that without ever measuring the velocity the circulation flow could be computed.

![Fig. A-1.](image-url) A typical Concentration and normalized concentration plot for a tracer experiment.