Second International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
6-8 December 1999

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF GAS STIRRED LADLES

J.F. DOMGIN, P. GARDIN and M. BRUNET

IRSID
Heat Transfer, Electromagnetism & Fluid Dynamics Department
Voie Romaine BP 30320
57283 MAIZIERES-LES-METZ Cedex, FRANCE

ABSTRACT

This paper describes IRSID's work to predict hydrodynamics of gas stirred ladles, with one or two injectors. This two-phase flow study is divided in two main parts : measurements in water model with plume characterisation and numerical simulation with Fluent CFD package, using either Euler-Euler or Euler-Lagrange approaches.

NOMENCLATURE

- C_D drag coefficient
- D_P particle diameter [m]
- F momentum transfer [kg m/s²]
- F_D drag force [N]
- F_q additional terms in momentum equation [kg/m²s²]
- g acceleration of gravity [m/s^2]
- I average interfacial momentum transfer rate [kg/m²s²]
- K turbulent kinetic energy [m^2/s^2]
- K_{pq} momentum exchange coefficient [kg/m^{3}s]
- P static pressure [Pa]
- q index =1 (continuous phase), =2 (dispersed phase)
- Q_v volumetric gas flow rate [m^3/s]
- Q_{air} volumetric air flow rate [m^3/s]
- r radial abscissa of the ladle [m]
- R radius of the ladle [m]
- Re Reynolds number
- t time [s]
- U mean fluid velocity [m/s]
- u instantaneous fluid velocity [m/s]
- $\langle u_i' u_j' \rangle$ fluctuating velocity correlation [m^2/s^2]
- v instantaneous bubble velocity [m/s]
- x position [m]
- α void fraction
- Δt time step [s]
- ε dissipation rate of K [m^2/s^3]
- μ fluid dynamic viscosity [Pa.s]
- ρ fluid density [kg/m^3]
- ρ_P bubble density [kg/m^3]

INTRODUCTION

Ladle treatment is a steelmaking process devoted to improve inclusion elimination or mixing of additive, just before the feeding of a tundish. Usually, pneumatic stirring is used. A lot of papers have already been published dealing with the use of CFD to design ladle (Ilegbusi and Szekely (1990), Mazumdar and Guthrie (1994), Sheng and Irons (1995), Schwarz (1996)). But experimental information is often incomplete (gas injection geometry and porosity, plume morphology …) and the assessment of the origin for disagreement between measurements and calculations is usually impossible.

The main interest in conducting experiments in some different water models is to identify physical mechanisms that CFD codes have to cope with. In this way, complexity of two-phase flow is characterised and it becomes possible to use appropriate models and to explain their limitations.

The aim of this paper is to review basic configurations of water models that we would like to simulate and to apply Fluent CFD code (version 4.5) for some of them.

VISUALISATION ON WATER MODELS

Before performing calculations with Fluent CFD code, some visualisations were carried out when air is injected in a water tank (0.5 m diameter and 0.5 m height). The objective is first to get experimental values that can be used for CFD validation and also to get an insight into the plume behaviour and the free surface stability for different gas flow-rates.

One plume

A lot of papers have already been published on single plume behaviour for different gas flow-rates. Unfortunately, most of the time there is a lack of experimental data and numerical simulations cannot be supplied with the correct values (initial velocities, bubbles diameters …). This is one of the reasons why we decided to carry out measurements for this classical configuration. Gas flow-rates, when transformed to industrial values with similarity laws, are representative of in site values. Flows are quite similar to what has already been published (Anagbo and Brimacombe, 1990).

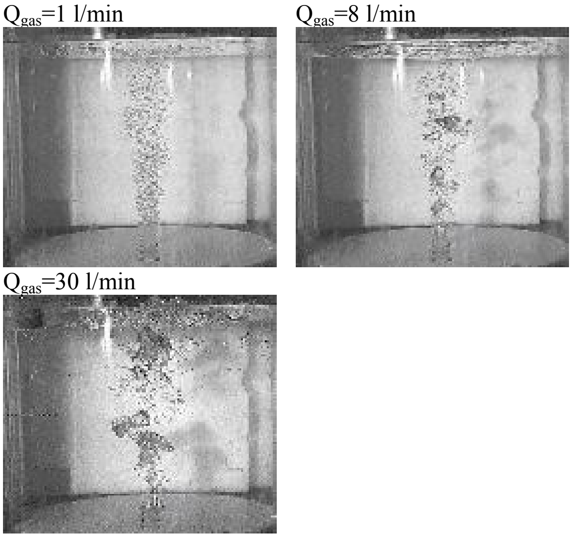


Figure 1 : Plume behaviour for different air flow-rates.

Whereas bubbles can be considered as spherical or slightly ellipsoidal for small air flow-rate, important heterogeneity in bubble phase morphology is observed when flow-rate increases (Tacke et al., 1985). In the same way, free surface disturbance is all the more pronounced as air flow-rate is high.

For CFD, it is a major challenge to deal with bubble coalescence or break up using standard commercial package. As far as we know, there is no available model implemented in any commercial package to cope with interacting bubbles, although very promising work is under development (Sheng and Irons, 1995, Laux and Johansen, 1999). We shall limit our comparison to flow-rates lower than 8 l/min ; the free surface can be considered as flat.

Two plumes

In this situation, only 2 air flow-rates were investigated : 1 and 5 l/min flowing through each of the porous plugs, which are separated either by a distance of R/2 (distant position) or by R/4 (close position).

The following table summarises the main results deduced from visualisations :

	Air flow-rate : 1l/min	*Air flow-rate : 5 l/min*
Distant position	No noticeable mutual influence of plumes.	Each plume development is practically independent from the other one (a slight inclination towards walls is noticed).
Close position	Plumes seems to be attracted by each other, but the mutual influence remains quite small. (Figure 2)	Plumes behaviour is clearly non stationary : from time to time plume are either repulsive or attractive (with important inclination of plumes, Figure 3).

Table 1 : Mutual influence of plumes.

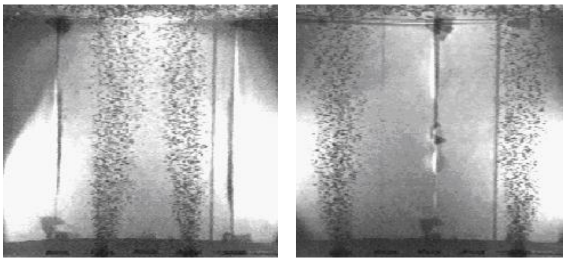


Figure 2 : 2 plumes configuration at different locations – flow-rate : 1l/min

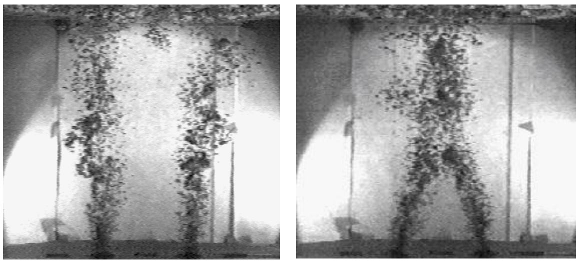


Figure 3 : 2 plumes configuration at different instants – flow-rate : 5l/min

When calculations should be performed, it appears that we have to establish whether the flow is stationary or not, since small air flow-rates (lower than 5 l/min) can lead to important transient phenomenon. As a practical disturbance, Becker and Oeters (1998) mentioned that important concentration fluctuations are observed, when tracing a ladle water mode, probably due to fluctuation in plume position. Schwarz (1996) already published interesting results showing that oscillation can occur, provided that bath depth is within a certain range. Xie and Oeters (1992), observed that, for single plume, the main parameter for the oscillation period is the vessel diameter. Multi-plume configurations are probably more difficult to simulate than single ones, due to possible strong mutual interaction of plumes.

Wall effect

Another water model was used here. It consists of a square section water channel (200*200 mm^2 section), filled with approximately 400 mm height water. Porous plug can come closer to the wall : 2 positions were studied

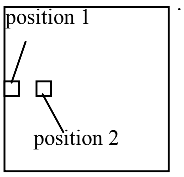
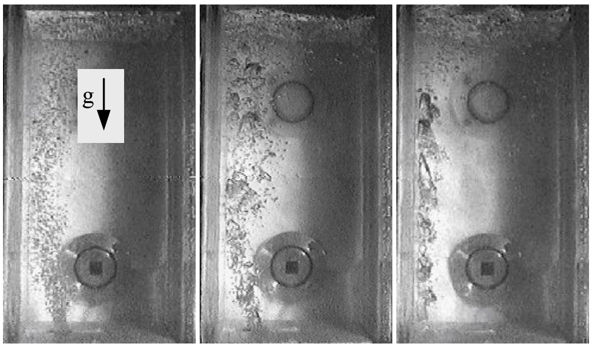


Figure 4 : Water model – top view.

When air is injected at position 2, a systematic attraction of plume by wall is observed (see Figure 5, first 2 figures) ; this looks like a Coanda effect and was already mentioned by Iguchi et al. (1998). For the smallest air flow-rate (1l/min per injector), it seems there is a bounce of the plume on the wall which was not observed in Iguchi's paper. This stationary phenomenon was not visualized for higher flow-rates.

For gas injection at position 1, the most striking effect is the elongation of bubbles along the wall and their very limited lateral dispersion for high flow-rate (8l/min and

10l/min). This behaviour is very similar to what is observed in two-phase flow in a vertical pipe, for the so-called plug and churn flow regimes. It is likely that difficulties will appear when this kind of situation is tackled by CFD, in particular for prediction of regime transition.



1 l/min-position 2   8 l/min-position 2   10 l/min-position 1

Figure 5 : Wall influence in plume development

In conclusion, it turns out that the modelling of two phase flow is all the more difficult as there are many injection positions and plumes are close to the walls.

TWO PHASE FLOW MODELLING

Currently, there are two approaches for the numerical calculation of multiphase flows : the Euler-Euler approach, commonly called the Eulerian approach and the Euler-Lagrange approach called the Lagrangian approach. In this section, we define the numerical basis of these two different methods and present some examples of numerical results obtained with these two methods applied to the different experimental configurations.

Eulerian approach

In the Eulerian approach, the different phases are treated mathematically as interpenetrating continua. Since the volume of a phase cannot be occupied by the other phase, the concept of phasic volume fraction is introduced. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations which have similar structure for all phases. These equations are presented below :

continuity equation (1)

$$\frac{\partial}{\partial x_j}\alpha_q \rho_q U_{q,j} = 0$$

In our case, no mass transfer between the 2 phases and stationary situation are considered.

momentum equation (2)

$$\alpha_q \rho_q U_{q,j} \frac{\partial}{\partial x_j} U_{q,i} = -\alpha_q \frac{\partial P}{\partial x_i} + \alpha_q \rho_q g_i$$

$$-\frac{\partial}{\partial x_j}\alpha_q \rho_q \left\langle u_i' u_j' \right\rangle_q + I_q + F_q$$

where the average interfacial momentum transfer rate $I_{q,i}$ between the 2 phases is based on the value of the exchange coefficient given by :

$$K_{pq} = \frac{3}{4} C_D \frac{\alpha_p \rho_q \left| \vec{u}_p - \vec{u}_q \right|}{D_p} \qquad (3)$$

Appropriate interphase drag coefficient is introduced using the user-defined subroutines of the CFD Fluent code. Formulations proposed by Moore (1968) and Magnaudet et al. (1995) for bubble Reynolds number less than 470 (spherical regime), and supplemented by Comolet (1979) correlation for large bubbles diameter are tested.

F_q is an additional term which takes into account the virtual mass effect and is given by (Drew and Lahey, 1993) :

$$F_q = 0.5 \alpha_p \rho_q \left(\frac{\partial u_q}{\partial t} - \frac{\partial u_p}{\partial t} \right) \qquad (4)$$

To take into account the effects of turbulence, a model based on the (K-ε) model is used. Turbulent predictions for the continuous phase are obtained using the standard (K-ε) model supplemented with extra terms that include interphase turbulent momentum transfer (Elgobashi et al., 1983). On the other hand, predictions for turbulence quantities of the dispersed phase are achieved using the Tchen theory for dispersion of discrete particles by homogeneous turbulence (Hinze, 1975). Those quantities are deduced from mean characteristics of the primary phase and the ratio between bubble relaxation time and eddy-bubble interaction time (Simonin & Viollet, 1990). Turbulent momentum diffusivity is then obtained and the drift velocity concept is also used.

An example of the predicted air volume fraction, computed with this method, is presented in figure 6. It concerns the simulation of the experimental configuration shown in figure 5, where the plume is injected near the wall at position 2 and the flow-rate is 1 l/min. From a qualitative point of view, when we compare these 2 different results, a good agreement is observed : the bouncing phenomenon is reproduced by the numerical calculation, but is probably not pronounced enough.

(1)

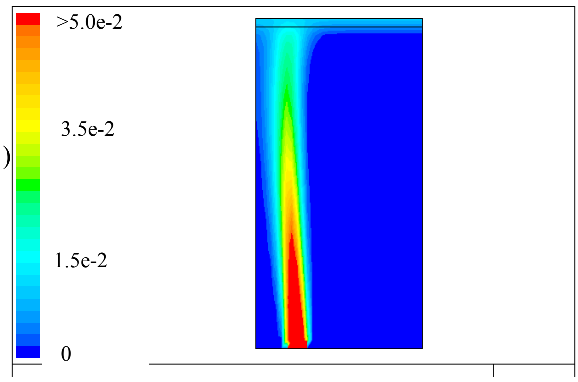


Figure 6 : Predicted air volume fraction for wall-plume configuration - Eulerian approach.

Lagrangian approach

In this second approach, the fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of bubbles through the calculated flow field. The bubble trajectories are computed individually at specified intervals during the fluid phase calculation, and can exchange momentum with the carrier phase (two-way coupling).

The continuity and the momentum equations used in this method and solved only for the continuous phase are similar to those proposed previously for the Eulerian approach.

However, the momentum transfer from the continuous phase to the dispersed phase is computed by examining the change in momentum of a bubble as it passes through each control volume and is given by :

$$F = \sum \frac{3\mu C_D R_e}{4D_p^2}(v-u)Q_v \Delta t \qquad (5)$$

It appears as a momentum sink in the continuous phase momentum balance in any subsequent calculations of the carrier phase flow field.

Turbulent predictions for the continuous phase are computed with a standard (K-ε) model. However, this model implemented in Fluent 4.5 assumes that the bubbles have no direct impact on the generation or dissipation of turbulence in the carrier phase.

Bubble trajectory is predicted by integrating the equation of motion written below :

$$\frac{dv}{dt} = F_D(u-v) + g(\rho_p - \rho)\Big/\rho_p + \frac{1}{2}\frac{\rho}{\rho_p}\frac{d}{dt}(u-v) + \frac{\rho}{\rho_p}\frac{du}{dt} \qquad (6)$$

This force balance equates the bubble inertia with the forces acting on it. In this equation, the most significant forces for gas-liquid two-phase flow (drag, buoyancy, added mass and pressure gradient) are taken into account (Domgin et al., 1998).

The dispersion of bubbles due to turbulence in the fluid phase is predicted using a 'random walk' model which includes the effect of instantaneous turbulent velocity fluctuations on the bubble trajectories through the use of stochastic methods (commonly called the 'eddy lifetime' model).

An example of a predicted water turbulent kinetic energy field, computed with this method, is presented in figure 7. It shows that maximum value of kinetic energy is located near plume impingement at free surface. This situation also occurs when Eulerian approach is selected. As it has been already observed for jet impingement, turbulent kinetic energy is probably overestimated at stagnation point using K-ε model (Behnia et al., 1998) : production of turbulent kinetic is directly connected to local mean velocity gradient (which is very important at stagnation point), whereas it should display some relaxation. A more detailed study is probably necessary to describe turbulence near the free surface, taking into account the local deformation of free surface. This step is vital for the modelling of mass transfer through interface between liquid metal and slag layer.

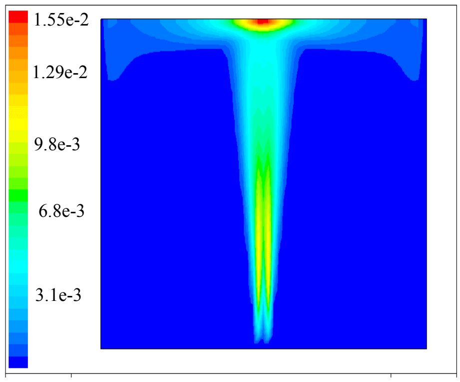


Figure 7 : Predicted water turbulent kinetic energy for one plume configuration - Lagrangian approach.

Description of the simulated configuration

The two different methods described previously are tested on the two-phase flow configuration proposed in figure 8. In addition to the boundary conditions specified in this figure, we define, for the Lagrangian approach only, reflection (elastic collision with a coefficient of restitution equal to 1) for the bubbles touching the walls and we assume that the bubbles are instantaneously captured at the free surface.

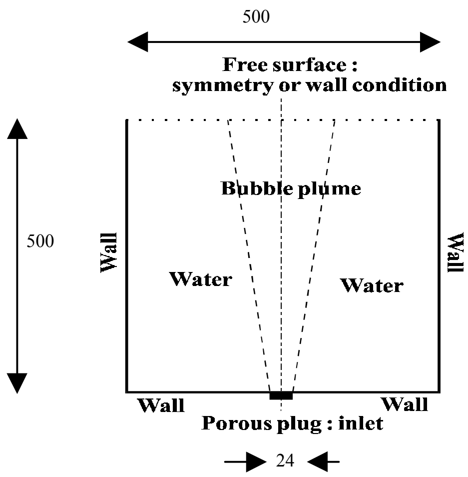


Figure 8 : Water model characteristics and boundary conditions illustration.

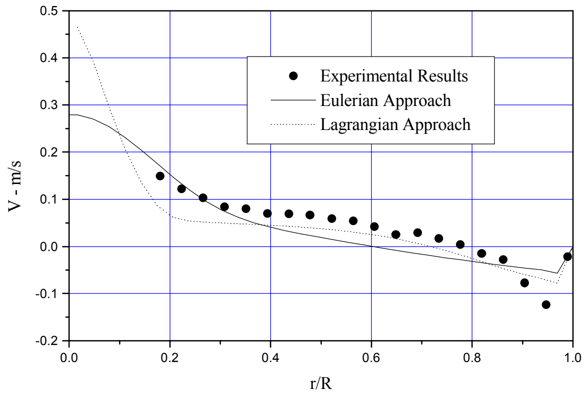
The 3D numerical simulations consist in determining the main characteristics of the flow field (velocity and turbulence) generated by the rising of spherical bubbles of 2 mm in diameter, injected continuously at the bottom of the ladle (air flow rate = 5 l/min). No interaction between the bubbles and no coalescence are considered here.

Results and discussion

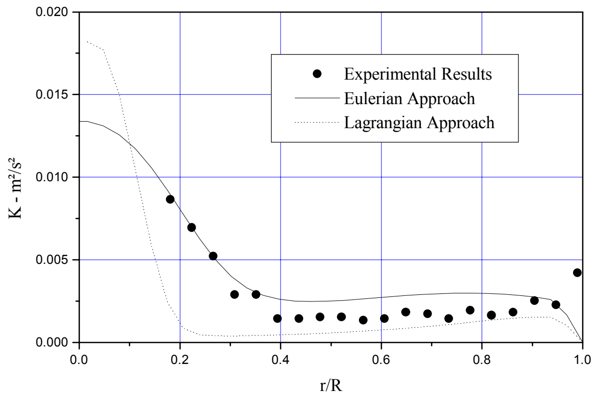
Finally, in this last section we compare the numerical results, obtained with the 2 approaches detailed previously, with the experimental ones. These experimental data are provided by a Laser Doppler Anemometer, measuring the axial and radial mean velocities and their root mean square values. To determine the total turbulent kinetic energy, we assume that the radial and tangential components are equal.

Comparisons are presented in figures 9 and 10. They concern mean axial velocity and turbulent kinetic energy along the non dimensional radius of the vessel r/R. Profiles are located 300 mm above the bottom of the tank.

Two different boundary conditions at free surface of the water tank were tested : a symmetry and a wall condition (see figure 8). When symmetry condition is considered (see figures 9), we observe some discrepancies between experimental and numerical results. The both eulerian and lagrangian approaches underestimate the mean axial velocity, particularly near the wall (r/R>0.9), while the turbulent kinetic energy is overestimated by the eulerian method and largely underestimates by the lagrangian one (particularly in the centre region : r/R<0.3).



a- Water vertical velocity

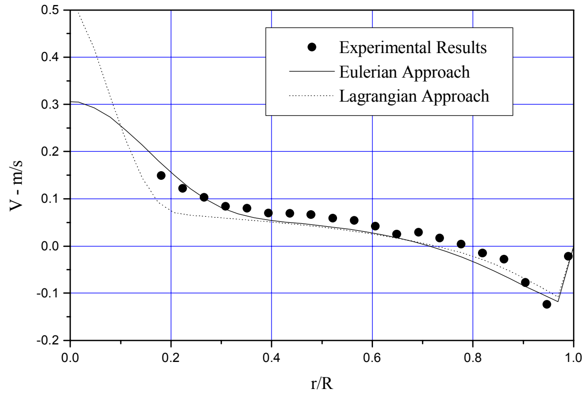


b- Water turbulent kinetic energy

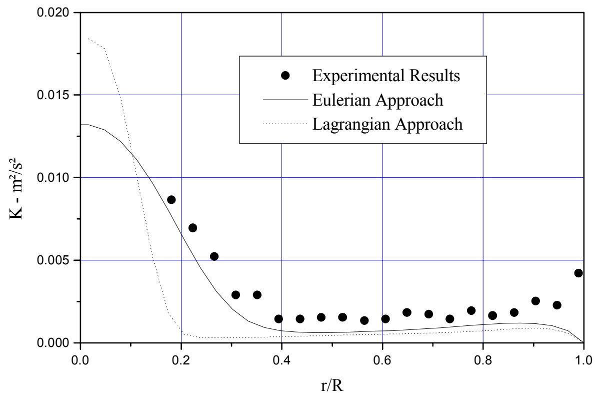
Figure 9 : Comparison between predicted and experimental data - Symmetry condition.

Then, in order to improve these comparisons, we decide to modify the boundary condition at the top of the water tank. As a matter of fact, the free surface can be considered as a very complex system with fluctuating energy coming from turbulent eddies. So, kinetic energy is likely to decrease at free surface and a zero condition for this variable was tested. The only way to do that with this CFD software is to set a wall type condition in this region. Comparisons, obtained in such a configuration, are presented in figure 10.

According to our numerical results, we can note that the new boundary condition has a positive influence on the numerical results : the Eulerian computed velocities display now a better agreement with the experimental data. On the contrary, the Lagrangian results seem to be more affected by this change, particularly for the turbulent kinetic energy which is underestimated. However, in this method, no extra source terms are added for the generation and dissipation of turbulence, which can explain problems observed with Lagrangian approach.



a- Water vertical velocity



b- Water turbulent kinetic energy

Figure 10 : Comparison between predicted and experimental data - Wall condition

Although some qualitative improvements have been made for the predicted mean axial velocity by modifying boundary condition at free surface, the computed turbulent kinetic energy is underestimated. Transport equations for K and ε need to be improved in conjunction with boundary conditions at free surface. Assumption that radial and tangential fluctuation velocities are equal has also to be discussed.

CONCLUSION

Flow visualisations realised in water models show that plume behaviour can be very complex :

◊ For important gas flow-rate (but remaining representative of industrial stirring), plume is composed of a wide range of bubbles : from small and spherical to large and distorted ; taking into account bubble interactions in CFD is probably one of the big challenge for the next decade ; at present, this cannot be properly tackled with commercial package. Some promising attempts are currently under development, with transport equation for average bubble diameter (Laux and Johansen, 1999) with the limitation of small bubbles. But there is nothing available and reliable at the moment for break-up and coalescence in commercial packages.

◊ When plumes are close to each other, important non stationary effects are created for moderate gas flow-rate ; as far as we know, there is no CFD results reproducing this phenomenon, although interesting theoretical results have already been obtained by Schwarz (1996).

◊ Walls can affect bubble plume when located in the immediate vicinity of the porous plugs, leading to bubble elongation and limiting their lateral dispersion. In addition to results published by Iguchi et al. (1998), it appears that the bubble plume can detached from the wall for low gas flow-rate.

From a numerical point of view, comparisons with experimental results show that CFD Fluent code supplies good prediction for the hydrodynamics of a gas-liquid two phase flow : computational results obtained with an Eulerian approach show a correct agreement with the experimental ones, particularly the mean velocity, while the turbulent kinetic energy is generally underestimated whatever the formulation used.

Moreover, a clear influence of boundary condition set at free surface of the bath is detected. It seems that a wall condition in this region improves the numerical results for mean velocity but not in terms of turbulent kinetic energy.

REFERENCES

ANAGBO P.E. and BRIMACOMBE J.K., (1990), “Plume characteristics and liquid circulation in gas injection through a porous plug”, *Metallurgical and Materials Transactions B*, **21B**, 637-648.

BECKER J.U., OETERS F., (1998), “Model experiments of mixing in steel ladles with continuous addition of the substance to be mixed”, *Steel research 69*, **1**, 8-16.

BEHNIA M., PARNEIX S., DURBIN, P.A., (1998), “Prediction of heat transfer in an axisymmetric turbulent jet impingement on a flat plate”, *Int. J. Heat Mass Transfer*, **41**(12), 1845-1855,

COMOLET R., (1979), “Sur le Mouvement d’une Bulle de Gaz dans un Liquide”, *La Houille Blanche*, **1**.

DOMGIN J.F, HUILIER D., KARL J.J, GARDIN P. and BURNAGE H., (1998), “Experimental and Numerical Study of Rigid Particles, Droplets And Bubbles Motion in Quiescent and Turbulent Flows - Influence of the History Force -”, *ICMF98, Lyon*.

DREW D.A. and LAHEY R.T., (1993), “Particulate Two-Phase Flow”, *Butterworth-Heinemann*, *Boston*, 509-556.

ELGOBASHI S.E. and ABOU-ARAB T.W., (1983), “A Two-Equation Turbulence Model for Two-Phase Flows”, *Phys. Fluids*, **26**, 931-938.

GREVET J.H., SZEKELY J., EL-KADDAH N., (1982) “An experimental and theoretical study of gas bubble driven circulation systems”, *Int. J. Heat Mass Transfer*, **25**(4), 487-497.

HINZE J.O., (1975), “Turbulence”, *McGraw-Hill Publishing Co., New-York.*

IGUCHI M., SASAKI K., NAKAJIMA K., KAWABATA H., (1998), “Effect on bubble characteristics in a bubbling jet rising near the side wall of a cylindrical vessel”, *ISIJ International*, **38**(12), 1297-1303.

ILEGBUSI O.J., SZEKELY J., (1990), “The modeling of gas-bubble driven circulation systems”, *ISIJ International*, **30**(9), 731-739.

LAUX H., JOHANSEN S.T., (1999), “A CFD analysis of the air entrainment rate due to a plunging steel jet combining mathematical models for dispersed and separated multiphase flows”, *Fluid Flow Phenomena in Metal Processing, Edited by N. El-Kaddah et al., The Minerals, Metals and Materials Society*, 21-30.

MAGNAUDET J., RIVERO M. and FABRE J., (1995), “Accelerated Flows Past a Rigid Sphere or a Spherical Bubble. Part 1 : Steady Straining Flow”, *J. Fluid Mech.*, **284**, 97-135.

MAZUMDAR D., GUTHRIE R.I., (1995), “The physical and mathematical modelling of gas stirred ladle systems”, *ISIJ International*, **35**(1), 1-20.

MOORE D.W., (1963), “The Boundary Layer on a Spherical Gas Bubble”, *J. Fluid Mech.,* **16**, 161-170.

SCHWARZ M.P., (1996), “Simulation of gas injection into liquid melts”, *Appl. Math. Modelling*, **20**, January, 41-51.

SHENG Y.Y., IRONS G.A., (1995), “The impact of bubble dynamics on the flow in plumes of ladle water models”, *Metallurgical and Materials Transactions B*, **26B**, 625-635.

SIMONIN O. and VIOLLET P.L., (1990), “Predictions of an oxygen droplet pulverisation in a compressible subsonic co-flowing hydrogen”, *Numerical Methods for Multiphase Flows*, FED, **91**, 65-82.

TACKE L., SCHUBERT H., WEBER D., SCHWERDTFEGER K. , (1985), “Characteristics of round vertical gas bubble jets”, *Metallurgical and Materials Transactions B*, **16B**, 263-275.

XIE Y., OETERS F., (1992), “Experimental studies on the bath oscillation during gas blowing into liquids, part1 : measurements using a single nozzle”, *Steel research 69*, **6**, 227-233.