Global Nest: the Int. J. Vol 2, No 2, pp 179-187, 2000
Copyright© 2000 GLOBAL NEST
Printed in Greece. All rights reserved

GAS INJECTION INTO AERATION TANKS

M. LATSA[1]
A. STAMOU[2]
N. MARKATOS[1]
D. ASSIMACOPOULOS*[1]

[1]*National Technical University of Athens*
Department of Chemical Engineering
[2]*National Technical University of Athens*
Department of Civil Engineering

Received: 20/01/00
Accepted: 06/07/00

*to whom all correspondence should be addressed
e-mail: assim@chemeng.ntua.gr

ABSTRACT

Aeration is one of the most common stages in the liquid waste clarification process. Most aeration tanks are equipped with aeration devices that inject oxygen to enhance the biodegradation of the liquid waste. In this work an aeration process is simulated, where the air is injected from the bottom of a cylindrical tank. Two numerical models are developed and two different geometries are considered. Their results are verified against published data. Both models solve the momentum, continuity and k-ε equations for the relevant phases. The first model considers liquid and gas phases as homogeneous fluid. The calculated velocity refers to the mixture of the phases. The second model assumes liquid and gas as two distinct phases. Velocity and volume fraction profiles portray the induced motion of the liquid and the extent of the aeration process. Regarding the two different geometries, the first assumes a flat free liquid surface, while the second a liquid surface free to swell according to the gas injection rate. It is shown that the second geometry gives more accurate results.

KEY WORDS: Aeration tank, gas-injection, two-phase turbulent modelling

INTRODUCTION

Gas injection is widely used in wastewater treatment processes. Gas bubbles, rising through the liquid, enhance mixing, minimize concentration inhomogeneities in the treatment tank and promote chemical and biological reactions. The effectiveness of gas injection in promoting these processes depends on the hydrodynamic behaviour of gas and liquid phases and their interaction with each other. The coupling between the gas and liquid phase creates the motion in the bulk of the liquid. As bubbles rise in the tank, due to buoyancy and kinetic energy at the nozzle exit, they induce a circulation of the fluid in the tank.

The importance of gas injection into liquid baths, not only for wastewater treatment but in metallurgy as well, is the reason for the high intensive research in this area. The difficulty of the experimental prediction of the flow field makes important the numerical simulation of the liquid behavior under gas injection. Considerable progress has been made, first with analytical models (Bravik and Killie, 1996) and more recently with numerical predictions based on single-phase (homogeneous) (Deb Roy *et al*., 1978; Deb Roy and Majumdar, 1981; McKelliget *et al*., 1982; Grevet *et al*., 1982; Bernard *et al*., 2000) or two-phase models (Delnoij *et al*., 1997; Cross and

Markatos, 1984; Turkoglou and Farouk, 1990; Turkoglou and Farouk, 1992; Schwarz, 1996; Smith, 1998; Hua and Wang, 2000; Latsa *et al*., 1999).

Analytical models employ pre-defined (Gaussian) distribution of bubble concentration and liquid velocity. Single- and two-phase simulations compute bubble concentration and liquid velocity from conservation equations. Single-phase simulations assume a predetermined two-phase region, on the basis of experimental data. Two-phase simulations have no such limitations and examine each phase separately. In general they fall into two categories; those which track bubbles as discrete particles (Eulerian-Lagrangian) (Delnoij *et al*., 1997) and those which assume each phase to be continuous and distinct (Eulerian-Eulerian) (Cross and Markatos, 1984; Turkoglou and Farouk, 1990; Turkoglou and Farouk, 1992; Schwarz, 1996; Smith, 1998; Hua and Wang, 2000; Latsa *et al*., 1999).

Earlier single-phase models assumed fixed cylindrical or conical plumes and they model turbulence through the k-ε model or simple ad-hoc viscosity formulas. Deb Roy *et al*., (1978) presented a homogeneous single-phase turbulent model for the aeration process, in which turbulence was modeled by an effective viscosity equation, with no other special treatment at the boundaries (wall functions etc.). The volume fraction of the gas phase was a function of the gas inflow rate and liquid velocity. The extent and location of the two-phase region was supposed to be known by experimental observations. An extension of that model followed in 1981, when Deb Roy and Majumdar simulated turbulence by the k-ε model. McKelliget *et al*. (1982) introduced an additional equation in order to calculate the gas dispersion, while viscosity was predicted by a simple ad-hoc viscosity formula. Grevet *et al*. (1982) used the model proposed by Deb Roy and Majumdar (1981) and calculated the gas volume fraction in the two-phase region by a mass balance approach. Recently, Bernard *et al*. (2000) implemented a drift-flux model for bubble plumes and the k-ε model for turbulence using empirical correlations of the plume radius and the bubble slip velocity.

All the above-mentioned models achieve a qualitative agreement between numerical predictions and experimental results in gas-liquid systems. However, their applicability is confined to flow systems, in which the distribution of the gas volume fraction has been determined experimentally or can be described by an empirical correlation. Two-phase simulations provide a more detailed description of bubble systems and can faithfully reproduce the near-field dynamics of bubble plumes.

Cross and Markatos (1984) presented a two-phase turbulent model and their results were in good agreement with Grevet *et al*. (1982) experiments. They used a simplified correlation for the interface friction force. Turkoglou and Farouk (1990), employed a more complex correlation for the intephase friction force and used the two-phase turbulent model proposed by Cross and Markatos (1982). They examined the effect of turbulence on heat transfer, and they compared the estimated height of the free surface with experimental data. The effect of gas inflow rate on the velocities and the gas volume fractions was analyzed in a later work by Turkoglou and Farouk (1992). Schwarz (1996) examined the surface swelling during the gas injection process. The main disadvantage of his model was the assumption of a constant interface friction, which neglects the bubble diameter. Smith (1998) compared a number of two-phase models with his experimental measurements and concluded that a more detailed model for the bubble/liquid interaction gives more accurate results. Hua and Wang, (2000) extended the previous models by adding a turbulent mass diffusion in the continuity equation and the viscous forces in the gas-momentum equation.

The present study formulates a single-phase turbulent model and a two-phase model, which solves the standard two-equation turbulence model for the liquid phase. The viscous forces are included in the gas phase equations as well as the turbulent mass diffusion. The interface friction is modeled through an empirical correlation using the bubble diameter and the calculated relevant phase velocity. Furthermore, the swelling of the water surface is modeled, with a variable interface friction term (instead of the constant term, which was used by Schwarz (1996)). The results of a flat free water surface single-phase and two-phase model are compared with the full two-phase model, which makes no specific assumptions for the free surface.

PROBLEM FORMULATION

The examined system consists of a vertical cylindrical tank of height H=1.0 m, and radius R=0.3 m, filled with water of height H_w=0.6 m. Air is injected at a point through the bottom of the tank, and the radius of the bubble hole is R_{bubble}=0.00635 m (Figure 1).

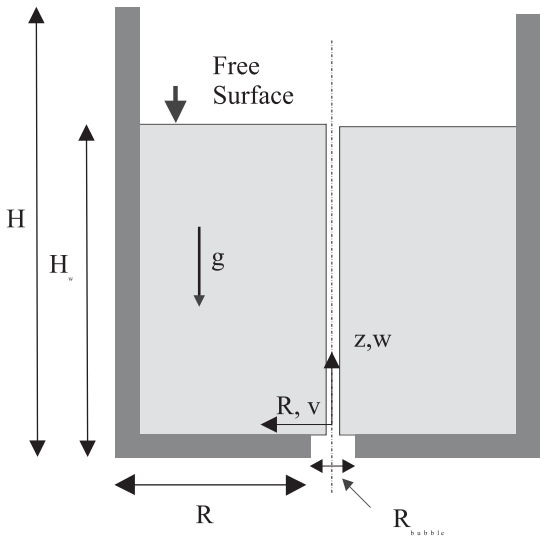

Figure 1. Geometry of the tank

Two-Phase Model

The two-phase formulation assumes that both gas and liquid coexist in the same computational cell at the same time. Each phase is a continium that may interact with the other one in the domain. Phases can exchange mass and momentum. The governing equations (mass, momentum and k-ε conservation equations) obey the general form:

$$\frac{\partial}{\partial t}\left(R_i \rho_i\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(rR_i \rho_i v_i\right) + \frac{\partial}{\partial z}\left(R_i \rho_i w_i\right) = div\left(\Gamma_i grad\left(R_i \varphi_i\right)\right) + R_i S_i \quad (1)$$

where i denotes the state of the phase (g for gas and l for liquid), R_i is the volume fraction of phase i, v_i and w_i the velocities of phase i, $\Gamma_{\varphi i}$ the exchange coefficient (viscosity, mass diffusion coefficient etc.) and $S_{\varphi i}$ the source terms, the form of which depends on the conservation equations (Table 1).

The mass conservation equations are formed by setting $\varphi=1$, $S_{\varphi i}=0$ and $\Gamma_{\varphi i}=D_t$ in the general conservation equation (1).

The term $div\ (\varrho_i D_t grad\ R_i)$ represents the dispersion of gas and liquid in the bath due to turbulence. The diffusion coefficient D_t is related to the turbulent viscosity through a turbulent dispersion Prandtl number σ_d as:

$$\Gamma_d = \frac{\mu_t}{D_t}$$

where ν_t is the turbulent kinematic viscosity calculated by the k-ε turbulent model. The value of σ_d varies from 0.1 to 1.0. In air-water systems the most common value used is 0.1 (Turkoglou and Farouk, 1990), which is also used in the present work.

In the momentum equations, C_f is the interphase friction coefficient. The effective viscosity ($\mu_{eff,i}$) is predicted by the k-ε model, which is applied only for the liquid phase. The application of the turbulence model to the gas phase has little effect on the results, as the gas phase has a small density and viscosity.

According to the k-ε model, the effective viscosity in the momentum equations is equal to:

$$\mu_{eff,i} = \mu_i + \mu_t$$

where μ_t is given by

$$\mu_t = C_D \frac{\rho_i k^2}{\varepsilon}$$

k is the turbulent kinetic energy and ε is the rate of dissipation of the turbulent kinetic energy and C_D a constant.

The k and ε conservation equations are obtained by transforming the general conservation equation (1) according to Table 1, where:

$$G_k = \mu_{t,l}\left\{2\left[\left(\frac{w_l}{z}\right)^2 + \left(\frac{v_l}{r}\right)^2 + \left(\frac{v_l}{r}\right)^2\right] + \left(\frac{w_l}{r} + \frac{v_l}{z}\right)^2\right\}$$

The k-ε constants C_1, C_2, C_μ, σ_k, σ_ε are equal to (Mohammadi, B., Pironneau, O., 1994):

C_1	C_2	C_μ	σ_k	σ_ε
1.44	1.92	0.09	1.0	1.3

Boundary conditions
Along the axis

$$v_g = v_l = 0$$

$$\frac{w_g}{r} = \frac{w_l}{r} = \frac{k}{r} = \frac{\varepsilon}{r} = 0$$

Along the walls

$v_g \quad v_l \quad w_g \quad w_l \quad 0$

For the values of k and ε the wall logarithmic law was used (Deb Roy and Majumdar, 1981).

At the injection point

The velocities of the gas are set equal to the injection velocities while the volume fraction of the liquid is zero and that of the gas is set at one. The k and ε are assumed to have zero gradient at the gas entrance.

At the free surface

Zero gradients are assumed for k and ε, as well as for all the velocities of both phases and for the volume fraction of the gas.

When the computational boundary coincides with the water surface the following practice is adopted for the liquid phase:

The gas leaving the liquid bath draws away an amount of liquid with it, elevating the surface of the bath and resulting in the non-satisfaction of the mass balance of the liquid. The calculation domain cannot simulate the elevation of the free surface. In order to ensure the satisfaction of the liquid mass balance, the mass of liquid that leaves the calculation domain is calculated, and then added to the free surface as a mass inflow.

This practice accelerates the rate of convergence.

Auxiliary relationships

To complete the model, information on the interphase friction, C_f, is necessary. Several relationships are studied in this work (Table 2 and 3), in order to assess their ability of those methods to model the friction between gas and liquid phase in this particular process.

The diameter of the bubble was assumed to be 3mm, according to experimental observations (Castillejos and Brimacombe, 1987; Anagbo and Brimacombe, 1990). A special handling was adopted for the last method (2f, Table 3), because the flow regime changes along the tank height. Whether a phase is considered dispersed or not, depends on the value of the volume fraction. Following Turkoglou and Farouk (1992), a phase is considered as dispersed, if its volume fraction is greater than 0.5.

Table 1. Values of terms of the general conservation equation for the two-phase model.

Equation	φ_i	$\Gamma_{\varphi i}$	$div\left(\quad _i \quad _i grad\left(R_i \quad \right)\right) \quad S_i$
Mass conservation	1	D_t	0
r-momentum	v_i	$\mu_{eff,i}$	$R_i \frac{P}{r} \quad \frac{2}{r}\frac{}{r}\left(R_i r \quad _{eff,i}\frac{v_i}{r}\right) \quad \frac{}{z}\left(R_i \quad _{eff,i}\frac{v_i}{z}\right)$ $\frac{}{z}\left(R_i \quad _{eff,i}\frac{w_i}{z}\right) \quad \frac{2R_i v_i \quad _{eff,i}}{r^2} \quad C_f\left(v_j \quad v_i\right)$
z-momentum	w_i	$\mu_{eff,i}$	$R_i \frac{P}{z} \quad \frac{1}{r}\frac{}{r}\left(R_i r \quad _{eff,i}\frac{w_i}{r}\right) \quad 2\frac{}{z}\left(R_i \quad _{eff,i}\frac{w_i}{z}\right)$ $\frac{1}{r}\frac{}{r}\left(rR_i \quad _{eff,i}\frac{v_i}{z}\right) \quad R_i \quad _i g \quad C_f\left(w_j \quad w_i\right)$
k-equation	k	$\frac{_{eff,l}}{_k}$	$div\left(\quad _l \quad _{kl} grad\left(R_l k_l\right)\right) \quad R_l\left(G_k \quad _l\right)$
ε-equation	ε	$\frac{_{eff,l}}{}$	$div\left(\quad _l \quad _l grad\left(R_l \quad _l\right)\right) \quad R_l \frac{}{k}\left(C_1 G_k \quad C_2 \quad _l\right)$

Table 2. Interphase friction coefficient, C_f

	Equation	Source
1	$C_f = C_D r_g r_l \bar{}$, C_D is a constant	Schwarz, 1996
2	$C_f = \frac{3}{8}\frac{r_g \ l}{d_{bubble}} C_D \lvert u_g \ u_l \rvert$ variable C_D	Lo, 1985

Single-phase Model

Turbulent one-phase flow is assumed and the whole mixture has one velocity. The governing equations obey the general form of equation (1), and the values of φ, Γ_φ and S_φ are summarized in Table 4.

The density of the homogeneous mixture is given by Grevet *et al*., 1982:

$$ _l, r \quad r_c$$

$$\bar{a} \ _g \quad (1 \quad \bar{a}) \ _l, r \quad r_c$$

where r_c is the radius of the two-phase region and $\bar{a}$ is the gas fraction, computed by the mass balance equation in each computational cell (Grevet *et al*., 1982):

$$\bar{a} = \frac{1}{2}\frac{Q_g \quad r_c^2 \bar{a}(1 \quad \bar{a}) w}{\int_0^r r w dz}$$

and $w_¥$ is the rising velocity of a characteristic single bubble equal to 0.4 m s^{-1} (Grevet *et al*., 1982). The k-ε equations are obtained from the two-phase k-ε model by setting the liquid volume fraction equal to unity. The constants C_1, C_2, C_μ, σ_k, σ_ε have the same values as for the two phase turbulent model.

Re, in Table 3, is the characteristic Reynolds number of a bubble and is defined as:

$$\mathrm{Re} = \frac{ _l \lvert U_l \quad U_g \rvert d_b}{ _l}$$

Table 3. Drag coefficient (C_D)

	Equation	Source
a	$C_D = \frac{8}{\mathrm{Re}}\frac{2 \quad 3\frac{g}{l}}{1 \quad \frac{g}{l}}$	Lo, 1985
b	$C_D = f(\mathrm{Re})$	Lo, 1985
c	$C_D = 4 \quad 7.2*10^{-4}\mathrm{Re} \quad 4.4*10^{-8}\mathrm{Re}^2$	Lo, 1985
d	$C_D = \frac{24}{\mathrm{Re}}(1 \quad 0.15\mathrm{Re}^{0.687})$	Ishii, 1979
e	$C_D = 110\frac{d_{bubble}}{2}(1 \quad r_g)^3$	Cheng *et al*., 1985
	$C_D = \frac{8}{3}(1 \quad r_g)^2, r_{dispersed} \quad 0.3$	
f	$C_D = \frac{24(1 \quad 0.15\mathrm{Re}_d^{0.687})}{\mathrm{Re}_d} \quad \frac{0.42}{1 \quad \frac{(4.25 \ 10^4)}{\mathrm{Re}_d^{1.16}}}, r_{dispersed}$	Turkoglou and Farouk 1992 0.3

Table 4. Terms in the conservation equations for the single-phase model.

Equation	φ	Γ_φ	$div(\quad grad(\ )) \quad S$
Mass conservation	1	*0*	0
r-momentum	*v*	μ_{eff}	$\frac{P}{r} \quad \frac{2}{r}\frac{}{r}\left(r \ _{eff}\frac{v}{r}\right) \quad \frac{}{z}\left(\ _{eff}\frac{v}{z}\right) \quad \frac{}{z}\left(\ _{eff}\frac{w}{z}\right) \quad \frac{2v \ _{eff}}{r^2}$
z-momentum	*w*	μ_{eff}	$\frac{P}{z} \quad \frac{1}{r}\frac{}{r}\left(r \ _{eff}\frac{w}{r}\right) \quad 2\frac{}{z}\left(\ _{eff}\frac{w}{z}\right) \quad \frac{1}{r}\frac{}{r}\left(r \ _{eff}\frac{v}{z}\right) \quad g\bar{}$

Boundary Conditions

At the axis

$$r = 0 \quad v = 0$$

$$r = 0 \quad \frac{w}{r} = 0$$

At the walls

$$v = w = 0$$

At the surface

$$z = H \quad w = 0$$

$$z = H \quad \frac{v}{z} = 0$$

At the orifice

$$r = 0, z = 0 \quad w = w_{in}$$

NUMERICAL METHOD

The differential equations are integrated over finite control volumes. A staggered grid is employed in the calculations, so that each velocity grid node lies between two scalar volumes. The volume fractions of each phase are calculated on the scalar nodes. An iterative solution procedure is used to solve the finite volume equations, based on the Inter-Phase Slip Algorithm, IPSA (Spalding, 1981), with an upwind differencing scheme. The computer code, TFLOW-2D, which was used in the present study is a modified version of the TEACH code (Gosman and Ideriah, 1976). TFLOW-2D was developed at the Computational Fluid Dynamics Section of the Department of Chemical Engineering, at the National Technical University of Athens for solving two-phase two-dimensional problems, of plane or axisymmetric systems, involving parabolic or elliptic, laminar or turbulent flows, with or without heat transfer.

RESULTS

Cases Examined

In order to determine the best physical configuration and the best model nine cases have been studied. In the first eight cases, the upper part of the computational domain coincides with the water surface. For the last case (Table 5, Case 9), the computational domain exceeds the water surface and the analysis is carried out for the values of the parameters that produce the better results in comparison to the experimental data. The error is calculated as follows:

Table 5. Examined cases

Case	Model
Case 1	Single-phase, no swell
Case 2	Two-phase, C_f=const, no swell
Case 3	Variable C_f (a), no swell
Case 4	Variable C_f (b), no swell
Case 5	Variable C_f (c), no swell
Case 6	Variable C_f (d), no swell
Case 7	Variable C_f (e), no swell
Case 8	Variable C_f (f), no swell
Case 9	Variable C_f (f), surface swelling

$$\%error = \sum_{n} \frac{\left| computed - experimental \right|}{experimental} 100 \frac{1}{n}$$

where n is the number of the experimental data (Grevet *et al*., 1982).

Discussion of Results

Table 6 presents the % error in each of the simulations.

The velocities predicted with the one-phase model (Case 1) differ 40% from the experimental values while the velocities predicted by the two-phase model differ 28 % (Case 8), and 22 % (Case 9) when the surface swelling is taken into account. The results obtained for cases 1, 8 and 9 fit the experimental values better than those obtained by the simulations of Grevet *et al*, (1982), whose results differ by 52%.

Table 6. Difference of computed results with experimental data

Examined Cases	% error
Case 1	40
Case 2	63
Case 3	60
Case 4	48
Case 5	70
Case 6	50
Case 7	72
Case 8	28
Case 9	22
Grevet *et al*., (1982)	52

The best results obtained with the two-phase model (Cases 8 and 9) using the method f (Table 5) for the interphase friction force, and the results of the one-phase model are shown in Figure 2. The effect of the gas injection becomes larger (the velocities of the liquid phase are greater) as the height of the liquid increases. This can be easily explained as the gas phase diffuses in the bulk of the liquid when the tank height increases (Figure 5). The results obtained from Case 9 are in better agreement with the experimental measurements even for small heights (z=0.06 m).

Furthermore, the results of Case 9, especially near the water surface, are significantly better than those obtained by the two-phase model with a flat water surface (Figure 3). It is clear that the flat surface assumption has the disadvantage of poor velocity predictions near the surface.

Figure 4 shows that the surface height reaches a maximum height of 0.695 m at the injection point.

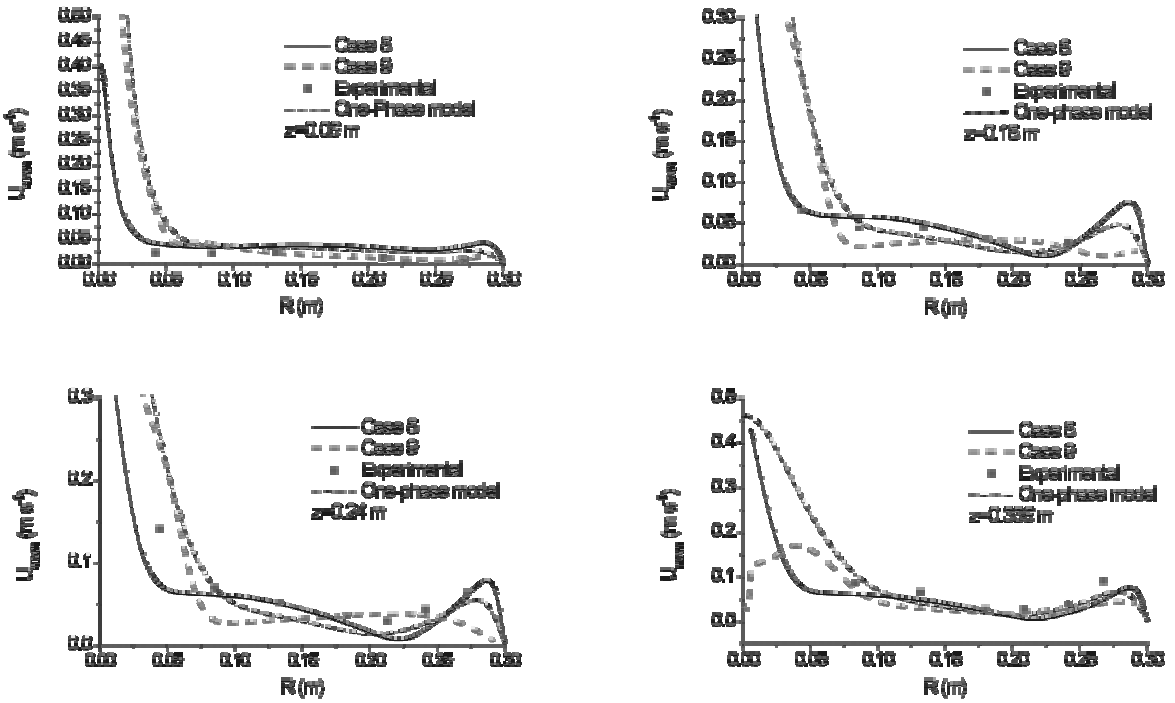

Figure 2. Radial dependence of the absolute value of the velocity at various axial positions (Case 1, Case 8 and Case 9).

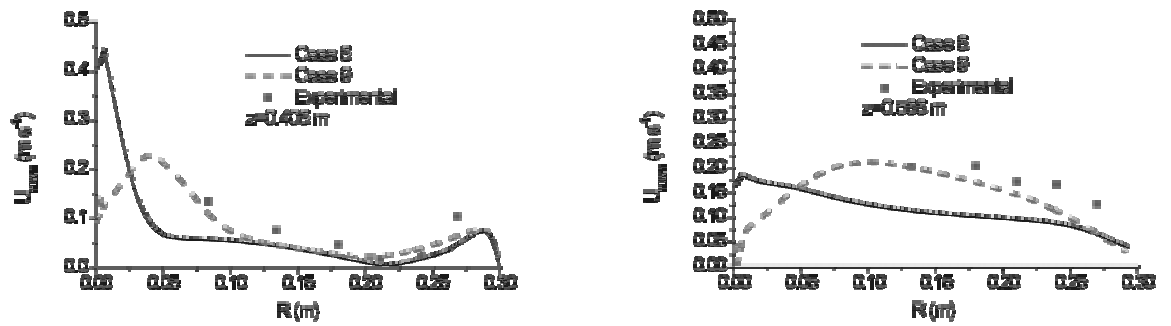

Figure 3. Radial dependence of the absolute velocity value near the free surface (Case 8 and Case 9).

Furthermore, the velocity and volume fraction fields at steady state, when surface swelling is considered (Figure 6), are not identical to those computed when a flat surface is assumed (Figure 5). This is clarified in Figure 7, where a second recirculation region is formed near the bath wall. In the region above the free surface, downward liquid velocities are predicted, indicating a downward motion of the liquid droplets (due to gravity). The liquid fraction profiles indicate a slight elevation of the water surface as well as a wider two-phase region. These observations lead to the conclusion that apart from the predicted velocities, phase volume fractions differ also significantly, when no flat surface is assumed. The surface swelling becomes clearer when gas enters the bath at higher injection rates.

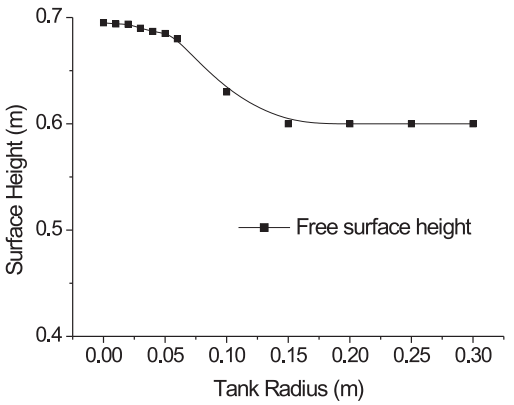

Figure 4. Computed height of water surface (Case 9)

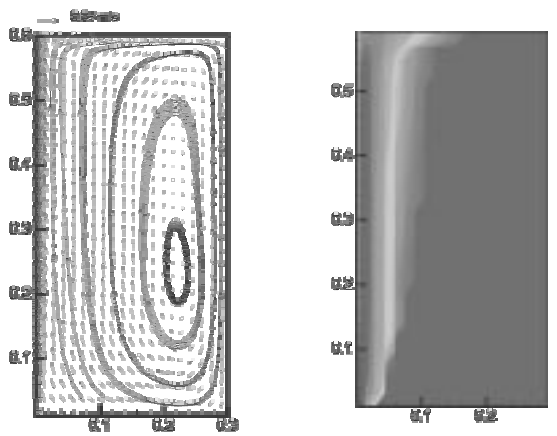

Figure 5. Liquid velocities field (left) and gas volume fraction profiles (right). Two-phase model with flat free surface (Case 8).

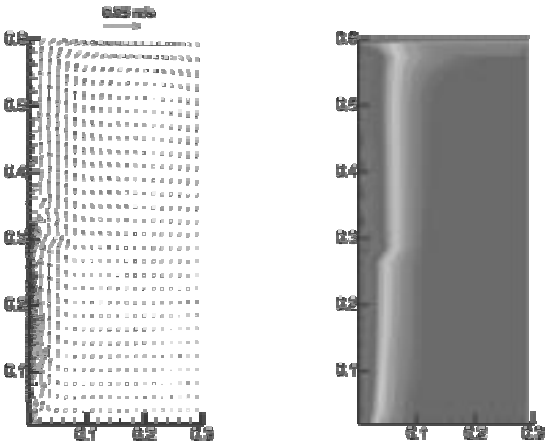

Figure 6. Liquid velocities field (left) and gas volume fraction profiles (right). Two-phase model with surface swelling (Case 9).

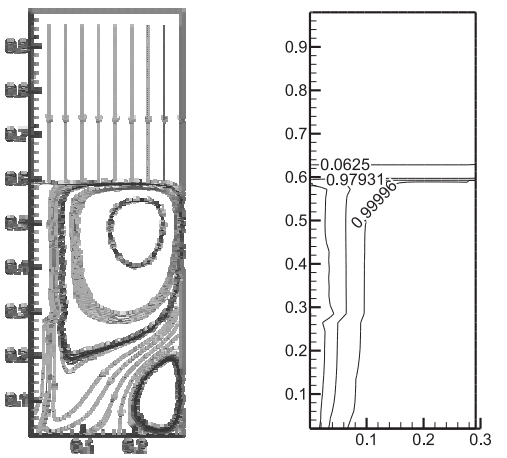

Figure 7. Streamlines (left) and liquid volume fraction (right). Two-phase model with surface swelling (Case 9).

CONCLUSIONS

The present study compares the proposed two-phase turbulent model with a single-phase turbulent model for the prediction of the behavior of gas-injection systems. The two-phase model produces more detailed and accurate results than the single-phase model. A series of simulations has also been carried out, in order to find the most adequate correlation for modeling the interface friction force in such systems. The one suggested by Turkoglou and Farouk (1992) was the best for this particular process.

Furthermore, two possible configurations of

studying the injection of gas into liquid baths during the aeration processes are considered:
a) a flat liquid surface and
b) a liquid surface, which is free to swell.
The results obtained with the second configuration appear to be in better agreement with experimental measurements. The differences of the computed volume fraction and velocity profiles of the two models cannot be ignored.
From the results of the present study it is clear that the best configuration for modeling gas injection in aeration processes is the assumption of an empty area above the liquid surface.

REFERENCES

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

Bernard, R.S., Maier, R.S. and Falvey, H.T. (2000), A simple computational model for bubble plumes, *Applied Mathematical Modelling*, **24**, 215-233.

Bravik, I. and Killie, R. (1996), Phenomenological description of the axisymmetric air-bubble plume, *International Journal of Multiphase Flow*, **22**, 535-549.

Castillejos, A.H. and Brimacombe, J.K. (1987), Structure of turbulent gas-liquid plumes in vertically injected jets, *Metallurgical Transactions B*, **18B**, 649-658.

Cheng, L.Y., Drew, D.A. and Lahey, R.T. (1985), An analysis of wave propagation in bubbly two-component, two-phase flow, *ASME Trans*, **107**, 402-408.

Cross, M. and Markatos, N.C. (1984), Mathematical modeling of gas injection into liquid metals, *4th Process Technology Conference*.

Deb Roy, T., Majumdar, A.K. and Spalding, B.D. (1978), Numerical prediction of recirculation flows with free convection encountered in gas-agitated reactors, *Applied Mathematical Modelling*, **2**, 146-150.

Deb Roy, T. and Majumdar, A.K. (1981), Predicting fluid flow in gas-stirred systems, *Journal of Metals*, 42-47.

Delnoij, E., Lammers, F.A., Kuipers, A.M. and Van Swaaij, W.P.M. (1997), Dynamic Simulation of dispersed gas-liquid two-phase flow using a discrete bubble model, *Chemical Engineering Science*, **52**, 1429-1458.

Gosman, A. D. and Ideriah, F. J. K. (1976), Teach-2E: A general computer program for two-dimensional, turbulent, recirculating flows, Department of Mechanical Engineering, Imperial College, London.

Grevet, J.H., Szekely, J. and El-Kaddah (1982), An experimental and theoretical study of gas bubble driven circulation systems, *International Journal of Heat and Mass Transfer*, **25**, 487-497.

Hua, J. and Wang, C. (2000), Numerical Simulation of bubble-driven liquid flows, (2000), *Chemical Engineering Science*, **55**, 4159-4173.

Ishii, M. and Zuber, N. (1979), Drag Coefficient and Relative Velocity in Bubbly, Droplet or Particulate Flows, *AlChE Journal*, **25**, 843-855.

Latsa, M., Kyrkilis, D., Stamou, A. and Assimacopoulos, D. (1999), Simulation of Circulation in aeration Processes, *6th International Conference on Environmental Science and Technology*, **A**, 233-242, Samos, Greece.

Lo, S.M. (1985), Development of a new interphase friction law for air-water flows in inclined pipes, PhD Thesis, Imperial College London.

McKelliget, J.W., Cross, M. and Gibson, R.D. (1982), A turbulent fluid flow model of gas agitated reactors, *Applied Mathematical Modelling*, **6**, 469-480.

Mohammadi, B., Pironneau, O. (1994), Analysis of the k-epsilon Turbulence Model, John Wiley and Sons, Paris.

Schwarz, M.P. (1996), Simulation of gas injection into liquid melts, *Applied Mathematical Modelling*, **20**, 41-51.

Smith, B.L. (1998), On the modelling of bubble plumes in a liquid pool, *Applied Mathematical Modelling*, **22**, 773-197.

Spalding, D.B. (1981), IPSA 1981; New Developments and Computed Results, *CHAM HTS/81/2*.

Turkoglou, H. and Farouk, B. (1990), Numerical Computations of fluid flow and heat treanfer in a gas-stirred liquid bath, *Metallurgical Transactions B*, **21B**, 771-781.

Turkoglou, H. and Farouk, B. (1992), Effect of gas injection velocity on mixing and heat tranfer in molten steel baths, *Numerical Heat Transfer*, *Part A*, **21**, 377-399.