Modelling of Gas-Liquid Two-Phase Flows in Vertical Pipes using PHOENICS

Vladimir Agranat, Masahiro Kawaji, Albert M.C. Chan*
Department of Chemical Engineering and Applied Chemistry
University of Toronto, 200 College Street, Toronto, Ontario, Canada, M5S 3E5
*Ontario Power Technologies, 800 Kipling Avenue, Toronto, Ontario, Canada, M8Z 6C4
E-mail: vladimir.agranat@utoronto.ca

Computers used:	SGI workstation (Unix) and Pentium PC (Windows-95)
PHOENICS versions:	2.0, 2.1.3 and 2.2.1

ABSTRACT

The PHOENICS code is employed to perform numerical simulations of gas-liquid two-phase bubbly flows in large vertical flow channels. Air-water and steam-water flows in large diameter pipes are simulated using the built-in two-fluid option of the code where liquid and gas are modeled as two interacting continua. The code is utilized in conjunction with advanced two-phase flow models, accounting for the interfacial drag, lift, pressure and virtual mass forces and the bubble induced turbulence. Numerical results obtained for both air-water and steam-water bubbly flows are in reasonably good agreement with experimental data available for large-diameter pipes from the open literature. The sensitivity study, showing the effects of various model parameters on flow characteristics, has been conducted. There is a significant effect of bubble diameter and lift force coefficient on the predicted lateral void fraction profiles. The void fraction peaking in the pipe wall region is discussed in more detail. The numerical results show that a further development of constitutive relations for interfacial transfer terms is needed to validate the two-fluid model under churn bubbly flow conditions.

CONTENTS

1. INTRODUCTION

Air-water and steam-water two-phase flows are important in nuclear reactors, heat exchangers, chemical processing and energy engineering. Furthermore, the two-phase flow characteristics in full-scale components such as 50-cm diameter pipes used in nuclear power plants are different from those observed in channels of smaller dimensions due to scaling and multidimensional effects. Therefore, a detailed analysis of gas-liquid two-phase flows in large flow channels and the development of multidimensional two-fluid models of such flows are much needed.

Information on steam-water two-phase flows in large diameter pipes (50-cm or larger) at elevated pressures is limited. However, there are many applications, in the chemical, nuclear and petroleum industries among others, where two-phase flows in large diameter pipes at elevated pressures and temperatures are encountered routinely or under accident scenarios. In the nuclear industry, understanding of the two-phase flow behavior in full-scale components and piping systems is important for safety analysis of existing nuclear reactors. The next generation of advanced reactors also emphasizes the elimination of active safety components and the introduction of passive systems in order to improve their reliability and safety. To further develop the natural circulation type advanced reactors and to increase the accuracy of safety analyses for the existing nuclear reactors, it is highly important to fully understand and to be able to predict the two-phase flow characteristics in large diameter pipes and non-circular channels.

Analyses of thermal-hydraulic phenomena often require extensive testing, however, experiments involving complex geometries, full-scale components and realistic conditions are too costly to perform. A reliable multidimensional two-fluid model, which can perform three-dimensional two-phase flow simulations, is a needed tool to conduct such analyses. The present work addresses the application of a multidimensional two-fluid model of the PHOENICS code to analyze and predict the gas-liquid flow characteristics for the plant size circular channels.

The approach, using the Computational Fluid Dynamics (CFD) numerical evaluation of a properly formulated multidimensional two-fluid model, has been recently developed for use in nuclear safety analysis. Lahey et al. [1] have found that a 3-D two-fluid CFD model is capable of predicting the void distribution data for air-water two-phase flow in a small tube under low void fraction, bubbly flow conditions. This approach has a promise [1-3] towards the prediction of two-phase flows at higher void fractions, however, additional work is necessary to develop the suitable constitutive relations and to verify the model predictions. Lahey et al.'s approach [1] is adopted in the present work by applying their model equations within the PHOENICS code [4].

The objective of the present study is to apply the PHOENICS two-fluid model for the gas-liquid two-phase flows in large vertical flow channels and validate it against experimental data on air-water flows in large pipes [5,6] and the data obtained in Ontario Power Technologies (OPT) [7] on steam-water two-phase flow parameters (local void fractions and fluid velocities) in a vertical 51-cm inner diameter pipe at pressures up to 7.5 MPa. The model can also be useful for predicting the void distributions in square channels for applications to the Simplified Boiling Water Reactor (SBWR) chimney design [8].

The reminder of this paper consists of three sections. Section 2 is a brief description of the mathematical model and the solution method (a complete description is available in [1-3] and the PHOENICS documentation, e.g., [4]). Section 3 presents the results of the simulation of air-water and steam-water flows in large vertical pipes and makes comparison with the available experimental data. This section is mostly based on the previous results [9,10]. The final section makes concluding remarks and recommendations for the future study.

2. MULTIDIMENSIONAL TWO-FLUID MODEL

In recent years, multidimensional two-fluid Eulerian-Eulerian models of two-phase flows have been significantly developed [1-6, 11-15], in particular, for nuclear engineering applications. They have been implemented into the general-purpose commercial CFD codes, such as PHOENICS and CFX, and some special-purpose CFD codes like ACE-3D [5,6] and CFD-TWOPHASE [15]. These models have become commonly used modelling tools among researchers and engineers from various engineering fields.

In a two-fluid model, two sets of governing conservation equations expressing the balance of mass, momentum, and energy are solved to find the pressure, phase volume fractions, phase velocities and phase temperatures [1-4]. These equations are well described in [1-4, 11-14]. The phase conservation equations are interdependent and linked by the coupling interfacial source terms. To derive the coupling terms the so-called constitutive equations are applied with a number of empirical coefficients [1-4, 11-14]: C_D, C_L, C_p, C_{vm}, C_μ, $C_{\mu b}$, which are the coefficients for the interfacial drag, lift, pressure, virtual mass forces, respectively, the shear induced turbulence coefficient and the bubble induced turbulence coefficient. To apply a multidimensional two-fluid model in a particular case with sufficient confidence, the constitutive equations and the specific values of the above coefficients need to be tested and validated against the experimental data available.

One of the most challenging tests for a two-fluid model is its ability to predict the two different lateral gas phase distributions, observed in bubbly upflows in vertical pipes: the wall void peaking and the core void peaking (the void accumulation near the pipe wall and the pipe centreline, respectively) [1,5,13,14]. It is important to note that the PHOENICS code was capable of predicting very well the lateral void distribution phenomena (the wall void peaking for the bubbly air-water upflows and the core void peaking for the bubbly air-water down-flows) in the case of a small diameter vertical pipe [1,13,14]. The code was also validated [9] against the average void distribution data for a vertical steam-water bubbly upflow in a large diameter pipe at high pressures [7]. In this paper, the two-phase turbulence model and the existing correlations [1-3,13,14] for the interfacial drag, lift, pressure and virtual mass forces, implemented in the PHOENICS code, are tested in order to predict the wall void peaking and the core void peaking in bubbly and churn bubbly air-water upflows in a large diameter pipe [5].

The two-phase flow is simulated using the standard two-fluid option of the PHOENICS code where liquid and gas are modeled as two space-sharing interspersed continua. The code solves two complete sets of Navier-Stokes equations via an interphase-slip algorithm known as IPSA. The fast parabolic option of the PHOENICS code is employed as the flow considered is a fully developed flow without any re-circulation patterns. As boundary conditions the following are used: at the inlet the superficial liquid and gas velocities, J_l and J_g, are specified and no-slip condition is assumed for the two phases (the phase velocities are equal), at the pipe centreline the symmetry condition is applied, and at the pipe wall the no-slip condition is used.

2.1. Comparison of Interfacial Drag and Lift Forces

The interfacial drag force, $\mathbf{F_D}$, between liquid and bubbles is introduced as the force, acting on isolated, spherical particles. The particle number-density (as a function of the gas volume fraction, α_g, and the bubble diameter, D_b) is $6\alpha_g/(\pi D_b^3)$, and hence the total drag force per unit volume, acting on the dispersed phase, is

$$\mathbf{F_D} = -0.75\, C_D/D_B\, \rho_l \alpha_g |\mathbf{V_r}|\, \mathbf{V_r} \tag{1}$$

where ρ_l is the liquid phase density, and $\mathbf{V_r} = \mathbf{V_g} - \mathbf{V_l}$ is the relative velocity vector.

There are various built-in interfacial drag coefficient models available in the PHOENICS code [4]. In this study, the interfacial drag coefficient is taken to be the 'dirty water' model given by Wallis [4]:

$$C_D = 6.3/Re_b^{0.385} \tag{2}$$

where the bubble Reynolds number, $Re_b = D_b|\mathbf{V_r}|/\nu_l$.

Usually, there is a considerable uncertainty in the values of bubble diameter, D_b. To obtain the first estimate of D_b in a bubbly up-flow, the following equation can be applied [4]:

$$D_b = 2\sqrt{\frac{\sigma}{g(\rho_l - \rho_g)}} \tag{3}$$

where σ is the surface tension, g is the acceleration due to gravity, ρ_l and ρ_g are the densities of liquid and gas, respectively. For air-water flows under standard conditions, equation (3) leads to 5 mm. For steam-water flows at high pressures, it will give about 2 to 3 mm. In the present work, the bubble diameter, D_b, was specified as 1 or 2 mm for air-water flows and 3 mm for the steam-water flow. In general, D_b can vary with the pipe diameter, the liquid phase subcooling [11] and the void fraction.

The interfacial lift force, $\mathbf{F_L}$, acting on the dispersed phase and induced by a velocity gradient of the continuous phase in the lateral direction (the vorticity of the liquid phase), is defined as [1,13,14]

$$\mathbf{F_L} = -C_L\rho_l\alpha_g\mathbf{V_r} \times \nabla \times \mathbf{V_l} \tag{3}$$

where x denotes the vector cross product.

For the axisymmetric pipe flows, considered in this paper, equation (3) becomes [13]:

$$F_{L,z} = -C_L\rho_l\alpha_g V_{r,z}\, dV_{l,z}/dr \tag{4}$$

where $V_{r,z} = V_{g,z} - V_{l,z}$ is the axial component of the relative velocity (in z direction), and r is the radial distance from the pipe centreline.

It is seen that the lift force, $\mathbf{F_L}$, is proportional to C_L and does not depend on the bubble diameter, D_b. However, its relative contribution with respect to the drag force, i.e., the ratio, $|\mathbf{F_L}|/|\mathbf{F_D}|$, which is proportional to $C_L D_b^{1.385}$ (for the drag model selected), increases with D_b. As a result, the relative effect of accounting for the lift force on the predicted flow characteristics can vary with the bubble diameter. This is discussed in more detail in the next section.

There is a significant uncertainty in the value of lift force coefficient, C_L. Various constant values of C_L from 0.01 (highly viscous flows) to 0.5 (inviscid flows) are used in the literature (see [11,13]). In this paper, the value of 0.1, recommended in [1,13] for bubbly flows, is applied in the base cases.

Other interfacial forces (pressure, virtual mass, etc.), acting on the bubbles in the gas-liquid two-phase flows, are not discussed here, as they are explained well in the literature [1-4, 11-13], where the two-phase turbulence models, accounting for the bubble induced turbulence, can also be found.

3. SIMULATION RESULTS

The two-fluid PHOENICS model, briefly described in the previous section, has been applied to simulate steam-water and air-water bubbly upflows in large vertical pipes. The emphasis of study is on the capability of the model to predict the lateral void distribution data. More details of the study can be found in [9,10].

3.1. Modelling of a Steam-Water Bubbly Up-Flow in a Large Vertical Pipe

In this section, the numerical results [9], obtained for a steam-water up-flow in a large vertical pipe at elevated pressures (up to 7.5 Mpa), are briefly reviewed. The versions 2.0 and 2.1.3 of the PHOENICS code were used to simulate the above flow. The computational results were in reasonably good agreement with OPT's experimental data available for a 51-cm inner diameter pipe [7].

The void distributions, measured in a 24-inch (50.8cm) inner diameter vertical pipe with steam/water flowing at 230°C and 280°C [7], have been predicted, using the versions 2.0 and 2.1.3 of the PHOENICS code. The code solved a set of two-fluid model equations and accounted for interfacial drag forces (version 2.0) and lift, pressure and virtual mass forces (version 2.1.3). A standard k-ε model of turbulence was also employed.

Calculations were performed in both two dimensions (r, z) and three dimensions (r, θ, z), for steady state conditions. Some transient simulations were also performed. The local void fraction, phasic velocities and pressure were predicted for several test conditions. The inlet gas and liquid velocities were estimated from the total mass flux and measured void fraction data using a homogeneous flow model. The estimated superficial velocities were assumed to exist at the exit of the flow straightener section near the pipe inlet. The solution domain ranged from the exit of the flow straightener to the measurement station 4.2 m downstream [7,9].

The test conditions simulated are summarized in Table 1. Three cases with inlet void fractions of $\alpha_{g,inlet}$ = 0.205, 0.43 and 0.59 at 280°C and three cases with inlet voids of 0.106, 0.205 and 0.46 at 230°C have been simulated. In Table 1, the inlet void fraction, $\alpha_{g,inlet}$, and the liquid and gas mass fluxes, G_l and G_g, at the inlet are given as well as the measured and calculated pipe-average void fractions. The standard calculations were performed with the bubble diameter , D_b, set equal to 3 mm.

Table 1: Simulation Conditions and Results [9]

Run No.	T (°C)	G_l (kg/m^{2}s)	G_g (kg/m^{2}s)	$\alpha_{g,inlet}$	$\overline{\alpha}_{g,measured}$	$\overline{\alpha}_{g,calculated}$
1	280	2107	24.1	0.205	0.211	0.195
2	280	1652	55.1	0.430	0.409	0.420
3	280	1239	79.3	0.590	0.593	0.583
4	230	1445	2.9	0.106	0.105	0.094
5	230	1119	4.9	0.205	0.206	0.182
6	230	836	12.2	0.460	0.465	0.436

In all cases (Runs 1-6), the radial void fraction distribution was predicted to be almost flat across the pipe diameter (in agreement with the measured data), except for the boundary layer region near the pipe wall. In the latter, the predicted void fraction increased slightly or significantly depending on the interfacial force model used. The void fraction profiles predicted using the simplified model of PHOENICS 2.0, where only the interfacial drag force is taken into account, showed that the gas phase does not migrate significantly toward the wall region and does not cause the void peaking near the wall, which was previously observed and predicted in small-diameter tubes [1,13]. There was no significant effect of bubble size on the computed velocities and the void fraction profiles.

However, for the calculations with the advanced model of PHOENICS 2.1.3, which accounts not only for the drag force but also for the interfacial lift, pressure and virtual mass forces, the positive radial steam velocities are typical. They lead to a steam flow from the central part of the pipe towards the pipe wall and ultimately to the wall void peaking. The wall void peaking phenomenon was found in small diameter pipes [1,13] both experimentally and numerically. For the flows in large pipes, such a phenomenon has not been observed yet. The measurements in the OPT's experiments [7] did not cover in sufficient detail the pipe wall region, where the above effect is possible. Therefore, the additional validation of the advanced two-fluid model predictions described is necessary (in the near wall regions).

The pipe-average void fraction values predicted by PHOENICS are shown in Table 1 along with the measured void fractions. The predictions were in good agreement with the measurements.

3.2. Modelling of Air-Water Bubbly Up-Flows in Large Vertical Pipes

In this section, the numerical results [10] obtained for air-water bubbly upflow in a large vertical pipe are described. The simulation results are in good agreement with the experimental data available [5,6]. The sensitivity study, showing the effects of various model parameters on flow characteristics, has been conducted. The effects of the interfacial lift force and the bubble size on the computed velocities and the lateral void fraction profiles are examined in more detail. The void fraction peaking in the pipe near-wall regions is predicted and discussed.

The PHOENICS code was validated for bubbly air-water flows in small vertical pipes, having inner diameters of about 4 to 6 mm, in various papers, e.g., in [1,12-14]. However, it was not tested as extensively for gas-liquid flows in larger pipes. The study [10] was performed to investigate the capability of the two-fluid PHOENICS model to predict the effects of interfacial lift force and bubble diameter on the lateral gas phase distribution in a turbulent bubbly air-water upflow in a larger diameter vertical pipe (the internal pipe diameter, D=0.2 m). The PHOENICS 2.2.1 was used for the simulation.

The flow parameters investigated in detail in [10] were as follows: $D = 0.2$ m, $L/D = 60$, $J_l = 1.06$ m/s, $J_g = 0.033$, 0.11, and 0.26 m/s, where L is the pipe length, J_l and J_g are the superficial velocities of liquid and gas, respectively. The following values of the constitutive coefficients recommended in [1] were used in the base case: $C_{vm} = 0.0$, $C_p = 1.0$, $C_L = 0.1$, $C_\mu = 0.09$, $C_{\mu b} = 1.2$. The computational results were evaluated and compared with the gas phase distribution data [5,6].

The grid employed in the computations consisted of 20x120 cells in the radial and axial directions, respectively. The grid size was chosen based on the grid independence test results. The numerical study was aimed at testing the constitutive equations incorporated into the PHOENICS code under both uniform bubbly and churn bubbly flow conditions in a large vertical pipe air-water upflow [5]. Figures 1-4 show the main computational results obtained.

Figure 1 shows the lateral gas phase distribution calculated under the bubbly flow conditions (J_g = 0.033 m/s). The experimental data [5] are shown by circles. It is seen that the agreement between the simulation and experimental results is good. The effect of the bubble diameter is also shown. The difference between the phase distributions predicted for D_b = 1 and 2 mm is significant particularly near the pipe wall (r = 0.1 m). For the higher D_b, the predicted wall void peaking is more pronounced than for the lower D_b. It can be partially explained by a greater effect of the interfacial lift force (see Section 2) on the void distribution in the near-wall region. The lateral void profile predicted for D_b = 1 mm agrees with the experimental distribution better than that for D_b = 2 mm.

Figure 2 shows the lateral gas distribution under higher void fraction conditions (J_g = 0.11 m/s). There is good agreement between the computational results and the experiment (circles) in the pipe core, however, the difference between them near the pipe wall (r = 0.1 m) is larger than that shown in Figure 1 in the case of the lower void fraction (J_g = 0.033 m/s). The same values of the empirical constitutive coefficients are used in both cases (J_g = 0.033 m/s and 0.11 m/s).

Figure 3 shows the computational results under churn bubbly flow conditions where J_g = 0.26 m/s. In this case, the qualitative and quantitative agreement with the experimental gas phase distribution is seen only for a negative value of the lift force coefficient (C_L = -0.2). A similar result was obtained in [5] with the ACE-3D code. This result shows that the conventional interfacial lift force model with a positive lift force coefficient, C_L, proposed in [1-3,13,14] and implemented in PHOENICS and CFX, is not capable of predicting the void core peaking phenomenon in the churn bubbly air/water upflow considered in [5]. Only negative values of C_L allow to achieve it.

Figure 4 shows that the effect of the interfacial pressure force on the lateral void distribution is not significant. Indeed, the computational results obtained with and without accounting for the interfacial pressure force (C_p = 1.0 and 0.0) are very close.

Thus, in the case of bubbly flow, a good agreement between the PHOENICS predictions and the experimental data [5] has been shown under different gas flow rate conditions, while using the same lift force coefficient value (C_L = 0.1). There is a significant effect of the bubble diameter, D_b, on the lateral gas distribution: the wall void peaking due to the lift force effect is greater for larger bubbles than for smaller ones. The interfacial pressure force (the coefficient C_p) does not have a significant effect on the lateral void distribution.

However, in the case of churn bubbly flow, satisfactory agreement between the computational results and experimental gas phase distribution [5] can be obtained only for the negative value of the lift force coefficient (C_L=-0.2). Similar problems have been reported in the literature [5,6]. Thus, further development of the interfacial lift force model is needed to extend the two-fluid model applications to the churn bubbly flow conditions where the coalescence of large bubbles and the core void peaking are observed [2,5]. Under such conditions, where large bubbles can deform from a spherical shape, the advanced interfacial drag and lift force models for deformed bubbles might be useful (e.g., see [16,17]).

4. CONCLUSIONS

- The multidimensional two-fluid model, integrated into the PHOENICS code, is capable of predicting turbulent two-phase flow characteristics of both air-water and steam-water bubbly up-flows in large vertical pipes, including the lateral void fraction distribution data.
- Under bubbly flow conditions and for different gas flow rates, the PHOENICS two-fluid model was able to predict quite well the wall void peaking phenomena caused by the lift force effect on the dispersed phase distribution. However, the conventional lift force model with a positive lift force coefficient was not able to predict the transition from the wall void peaking regime to the core void peaking one under higher gas volume fraction conditions (churn bubbly flow).
- Further research is needed to develop and validate more advanced constitutive equations, in particular, the lift force implementation and the bubble size model, for the applications of two-fluid model to churn bubbly flows and other two-phase flow regimes. It will enable the model to be used for the detailed analyses of the two-phase flow behavior in full-scale nuclear reactor components as well as various other engineering systems.

5. ACKNOWLEDGEMENTS

This work was sponsored by the Natural Sciences and Engineering Research Council of Canada (NSERC) under the IOR grant.

6. REFERENCES

1. R.T. Lahey, Jr., M. Lopez de Bertodano and O.C., Jr. Jones*, Phase distribution in complex geometry conduits*, Nuclear Engineering and Design, 1993, Vol. 141, pp. 177-201.
2. M.Z. Podowski, *Toward Next Generation Multiphase Models of Nuclear Thermal-Hydraulics,* Proceedings of the Eighth International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, Kyoto, Japan, September 30-October 4, 1997, Vol.1, pp. 53-68.
3. A. Alajbegovic, N. Kurul, M.Z. Podowski, D.A. Drew and R.T. Lahey, Jr., *A CFD Analysis of Multidimensional Phenomena in Two-Phase Flow Using a Two-Fluid Model,* ANS Proceedings of 1996 National Heat Transfer Conference, August 3-6, 1996, Houston, Texas, HTC-Vol. 9, pp. 387-395.
4. H.I. Rosten and D.B. Spalding, The PHOENICS beginner's guide. TR/100. CHAM Ltd., London, 1984.
5. A. Ohnuki, H. Kamo and H. Akimoto, *Developed Flow Pattern and Phase Distribution under Gas-Liquid Two-Phase Flow in a Large Vertical Pipe and Prediction of Phase Distribution by Multidimensional Two-Fluid Model,* Proceedings of the Eighth International

Topical Meeting on Nuclear Reactor Thermal-Hydraulics, Kyoto, Japan, September 30-October 4, 1997, Vol.3, pp. 1670-1676.

6. A. Ohnuki, H. Kamo and H. Akimoto, *Prediction of Developing Bubbly Flow along a Large Vertical Pipe by Multidimensional Two-Fluid Model (Development of Multidimensional Two-Fluid Model Code and Analysis under a Low Velocity),* Proceedings of The Japan-U.S. Seminar on Two-Phase Flow Dynamics, July 15-20, 1996, Fukuoka, Japan, pp. 75-82.

7. H.A. Hasanein, A.M.C. Chan, M. Kawaji and Y. Yoshioka, *Steam-Water Two-Phase Flow in Large Diameter Vertical Piping at High Pressures and Temperatures,* Proceedings of the ASME/JSME 4th International Conference On Nuclear Engineering (ICONE-4), New Orleans, Louisiana, March 10-14, 1996, Vol. 1- Part B, pp. 675-685.

8. B.S. Shiralkar, Md. Alamgir and J.G.M. Anderson, *Thermal Hydraulic Aspect of the SBWR Design,* Proceedings of the International Conference on Design and Safety of Advanced Nuclear Power Plants, Atomic Energy Society of Japan, Volume III, pp. 31.1-1-31.1-7, October 1992.

9. V. Agranat, M. Kawaji, H. Hasanein, A.M.C. Chan and Y. Yoshioka, *Mathematical Modelling of Steam/Water Two-Phase Flows in Large Diameter Pipes at Elevated Pressures,* Proceedings of the Fourth Annual Conference of the CFD Society of Canada, Ottawa, Ontario, June 2-6, 1996, pp. 481-488.

10. V. Agranat, M. Kawaji and A.M.C. Chan, *Modelling of Steam/Water Two-Phase Flows in Large Vertical Channels at High Pressures,* Proceedings of the Sixth Annual Conference of the CFD Society of Canada, Quebec City, Quebec, June 7-9, 1998, pp. XI.13-XI.18.

11. H. Anglart and O. Nylund, *CFD Application to Prediction of Void Distribution in Two-Phase Bubbly Flows in Rod Bundles,* Nuclear Engineering and Design,, 1996, Vol. 163, pp. 81-98.

12. F.A. Hamad, M.K. Khan and H.H. Bruun, *A Comparison of Predicted and Experimental Phase Distribution and Turbulence in Two-Phase Flow*, The PHOENICS Journal, 1999, Vol. 12, No. 1, pp. 19-41.

13. R.T. Lahey, *The Prediction of Phase Distribution and Separation Phenomena Using Two-Fluid Models,* Boiling Heat Transfer/R.T. Lahey, Jr. (Editor), 1992, pp. 85-120.

14. M. Lopez de Bertodano, R.T. Lahey, Jr. and O.C. Jones, *Phase Distribution in Bubbly Two-Phase Flow in Vertical Ducts*, International Journal of Multiphase Flow, 1994, Vol. 20, No. 5, pp.805-818.

15. N. Vaidya and M. Lopez de Bertodano, *A Two-Fluid Model Computational Analysis of Bubble Columns*, ANS Proceedings of 1996 National Heat Transfer Conference, August 3-6, 1996, Houston, Texas, HTC-Vol. 9, pp. 379-386.

16. A. Tomiyama et al., *Effects of Eotvos Number and Dimensionless Liquid Volumetric Flux on Lateral Motion of a Bubble in a Laminar Duct Flow,* Proceeding of the Second International Conference on Multiphase Flow, Kyoto, Japan, 1995, Vol. 1, PD1-11-PD1-18.

17. A. Kariyasaki, M. Kawaji, M. Misawa and N. Ichikawa, *Bubble Detachment and Motion in a Flowing Liquid,* Proceeding of the Third International Conference on Multiphase Flow, Lyon, France, June 8-12, 1998.

Fig. 1. Prediction of wall void peaking under bubbly flow conditions

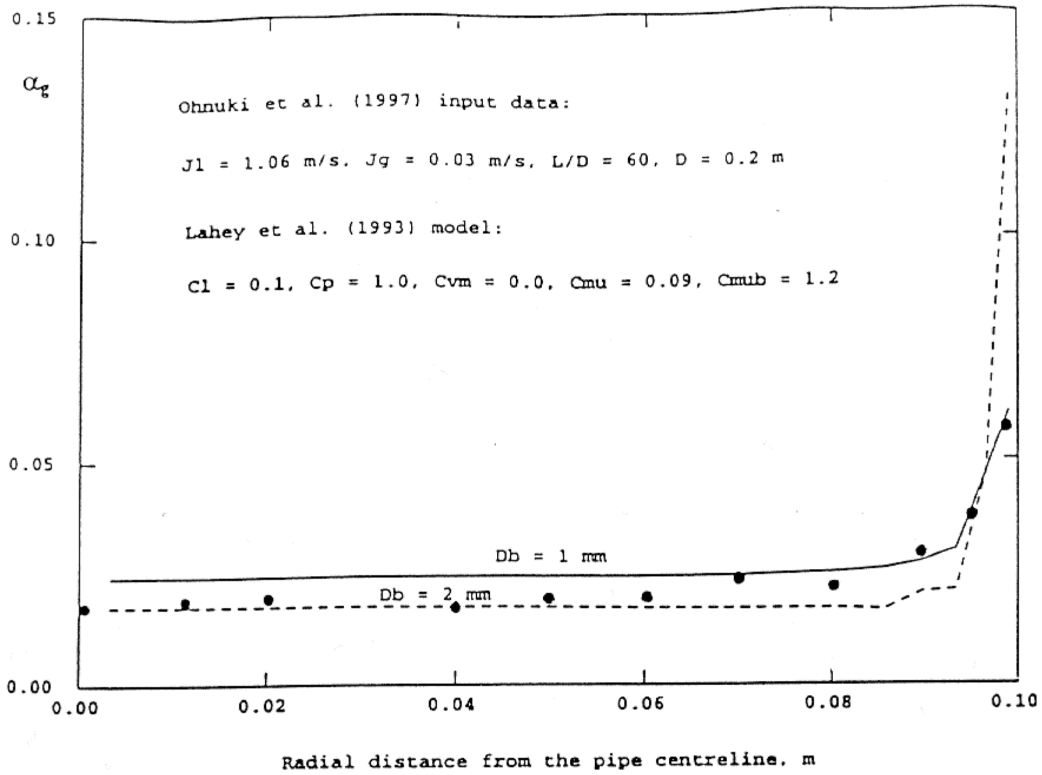

Fig. 2. Wall peaking for a higher void fraction (Jg = 0.11 m/s)

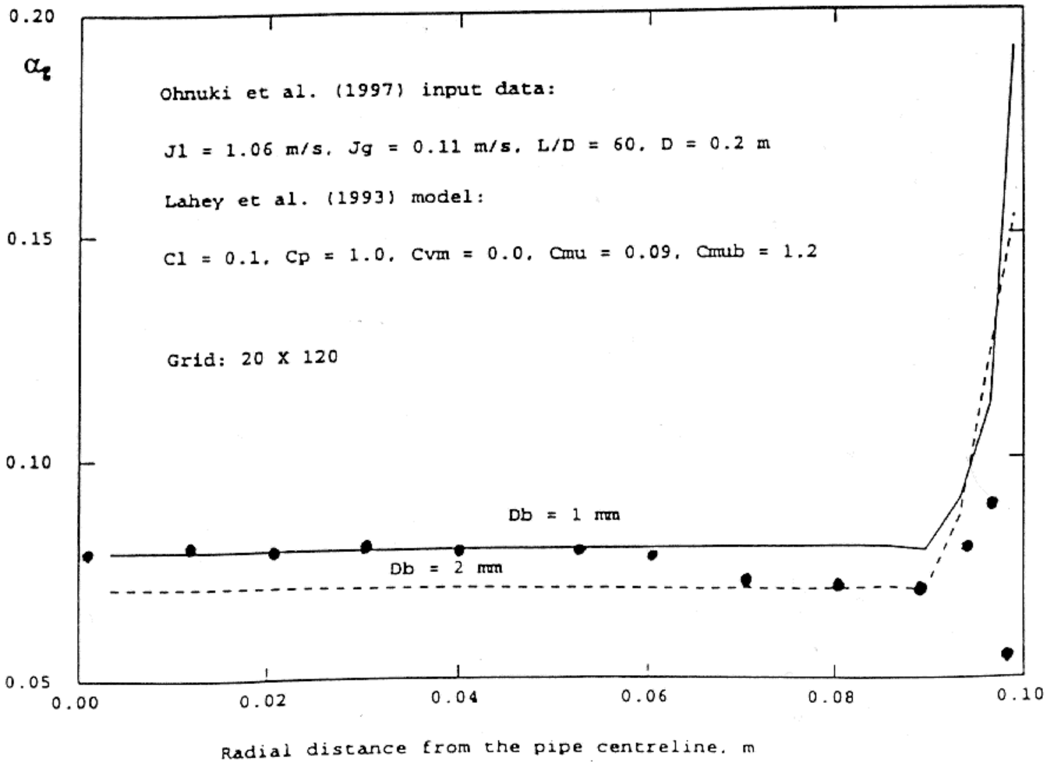

Fig. 3. Core void peaking under churn bubbly flow conditions

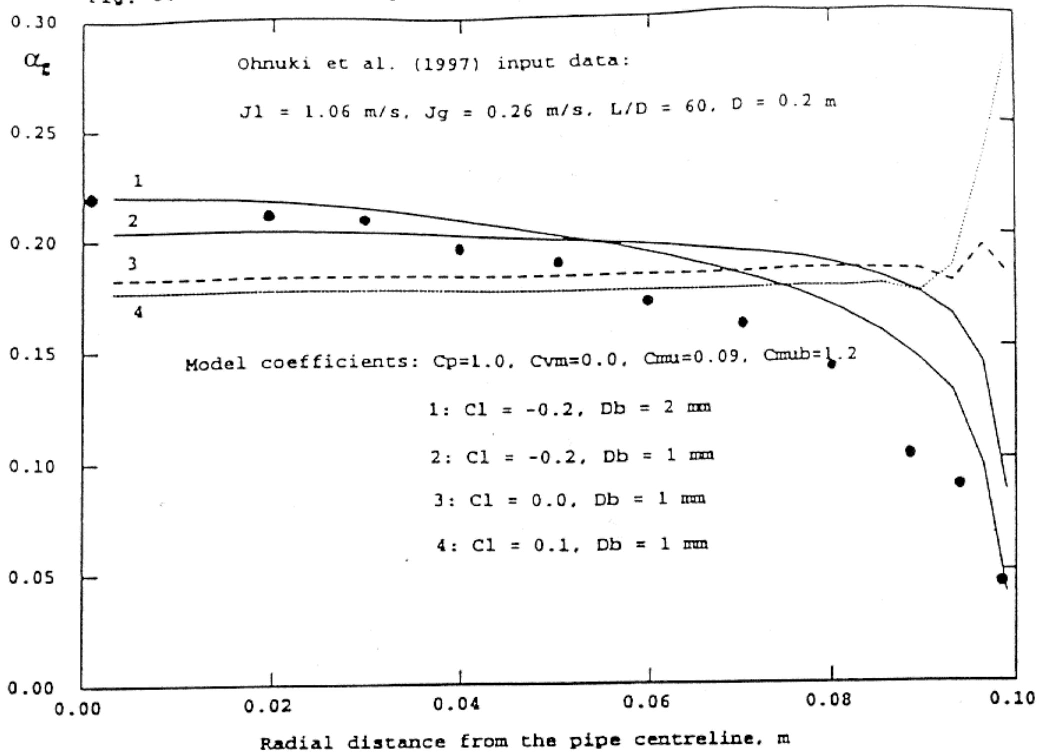

Fig. 4. Effect of interfacial pressure force on wall peaking