Mathematics and Computers in Simulation XXVI (1984) 475–484
North-Holland

TWO DIMENSIONAL MODELLING OF THE HOMOGENIZING EFFECTS OF FLOW OBSTRUCTIONS IN TWO FLUID FLOW *

M.B. CARVER

Applied Mathematics Branch, Aromic Energy of Canada Limited, Chalk River Nuclear Laboratories, Chalk River, Ontario, K0J 1J0, Canada

M.A. SALCUDEAN

Department of Mechanical Engineering, University of Ottawa, Ottawa, Ontario, Canada

It is well known that obstructions in two fluid flow have the effect of homogenizing the two fluid mixture, resulting not only in a more uniform phase distribution, but also in a significant reduction of the ratio of fluid velocities. This paper shows that these effects can be modelled using a two fluid computer program, and compares results to those obtained in a recent experiment.

1. Introduction

As a nuclear reactor system relies entirely on fluid circuits for energy transport, mathematical modelling of thermalhydraulic phenomena plays an important role in reactor design and development, and methods of improving the accuracy and efficiency of thermalhydraulic computations are sought continually. A simplified fluid circuit diagram of a CANDU reactor is shown in Fig. 1. Throughout most of the piping network the fluid behaviour may be adequately described by one-dimensional models such as those described in [8,9]. However, in the reactor fuel channel, flow must distribute itself amongs the intricate flow passages of the fuel bundle. In the secondary side of the steam generator, and in the calandria, the flow distribution is also complex. One-dimensional analysis is adequate to simulate overall or bulk energy transfer, but multi-dimensional analysis is necessary to model detailed local distribution of flows and temperatures in any of these geometrically complex components. The secondary side of the steam generator develops two-phase flow, but this flows vertically, so the homogeneous or single

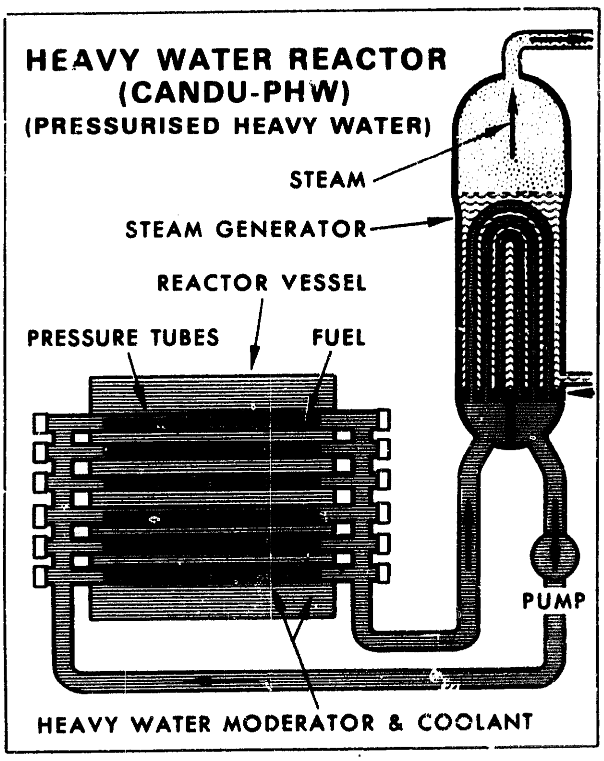


Fig. 1. Simplified view of CANDU reactor showing flow paths.

* This paper was presented at the IMACS International Symposium on Computer Methods for PDE's, Bethlehem, PA, U.S.A., June 1984.

fluid model of two-phase flow is reasonable [5]. In the CANDU fuel bundle, however, which is shown in Fig. 2, the flow passages are horizontally oriented. Under boiling conditions and lower flows there is, therefore, a tendency for the steam and water to flow in opposite directions with respect to gravity. A model of two-phase flow, which permits the water and steam phases to flow with different velocities, is required to simulate this gravity induced separation tendency. The Chalk River Nuclear Laboratory (CRNL) is involved in an extensive program of research and development of non-equilibrium two-phase flow modelling and multi-dimensional computational analysis of flow in fuel bundles and related geometries. Development of prototype codes incorporating advanced techniques is underway at CRNL and the University of Ottawa [6,10,11], and models and numerical methods used are discussed in a number of publications [5,15,16]. Parallel phenomenological investigation is required to ensure that the computer codes incorporate the mechanism necessary to simulate experimentally observed trends. The development program, therefore, contains a number of coordinated experimental and analytical projects, each of which acts as an auxiliary building block, providing information essential for the central project. This paper concentrates on one of these fundamental projects, which examines the effect of flow blockages on two fluid flow in horizontal channels. This study is relevant to the effect of the structural endplates in a CANDU bundle, which, as clearly seen in Fig. 2, introduce a blockage in some of the flow channels. It is well known that such a flow blockage tends to redistribute flow and introduce a homogenization effect. Currently, in a subchannel code, such as ASSERT [6], this effect is modelled by introducing local loss and diffusion parameters and an empirical modification to the drift flux model [6]. However, studies are needed to quantify this effect in further detail. A recent experiment reported by Salcudean et al. [12,13] studied the effect of flow obstructions on pressure drop and void distribution in horizontal air-water flow, and confirmed that a flow obstruction causes several separately identifiable but related local phenomena; a pressure drop, a mixing of the two fluids, a reduction of the ratio of air to water velocities, and an eventual return toward stratification. It is of interest to know whether a two fluid model contains the basic mechanisms necessary to simulate these phenomena, and the current study investigates this question. Obviously, a one-dimensional two fluid model cannot simulate either the tendency towards stratification or the homogenizing effects of the blockage. The effect is three dimensional in nature, but can be sufficiently characterized in two dimensions to provide an adequate proof-of-concept study. The two-dimensional two fluid code TOFFEA [3] has been used for this purpose.

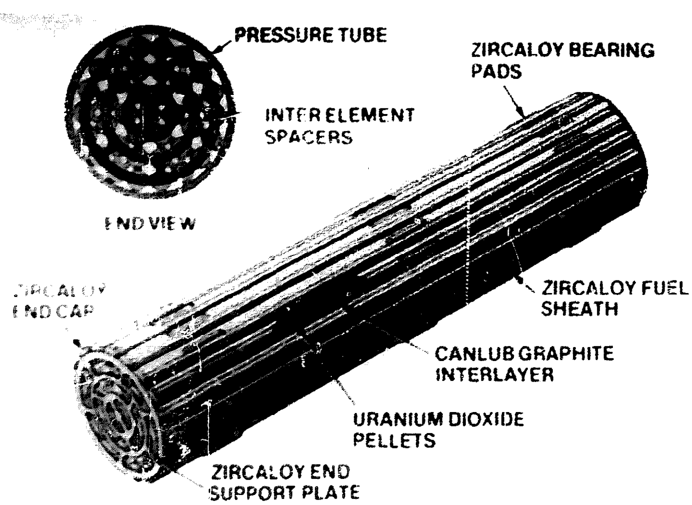
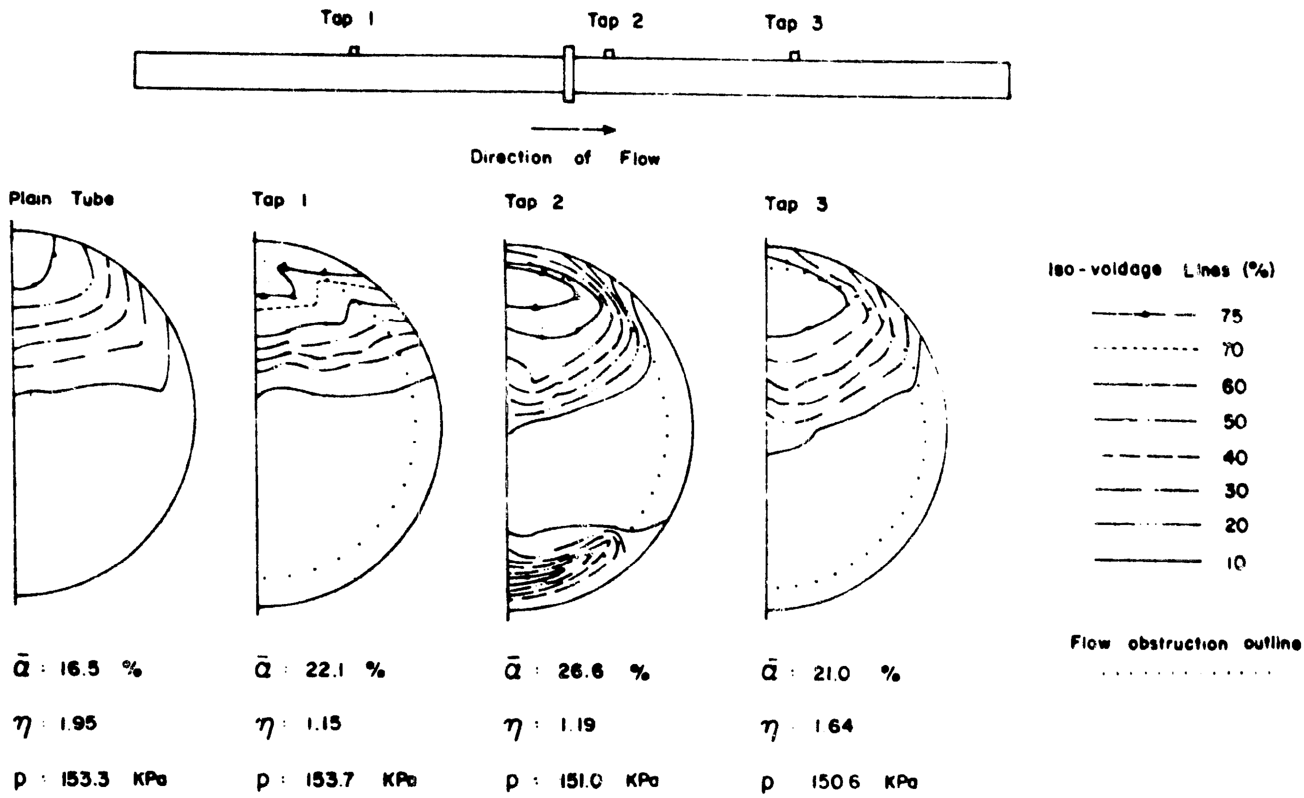
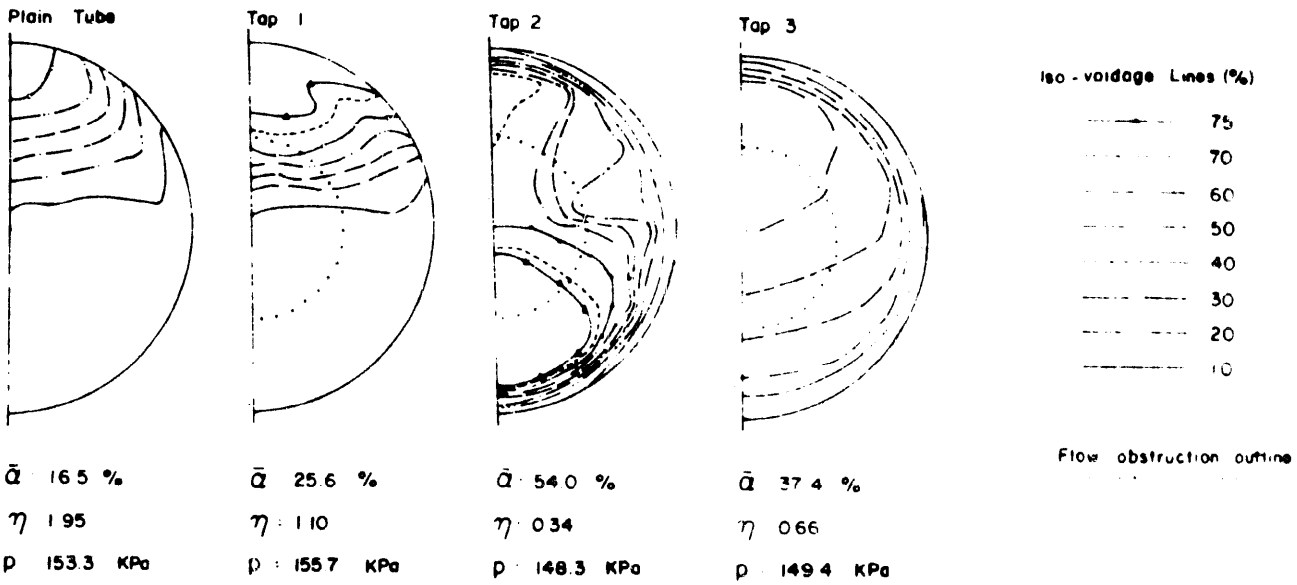


Fig. 2. A CANDU fuel bundle.



(a). Change of void distribution. Peripheral obstruction. Bubbly flow. Water: 1.1 kg/s. Air: 0.000775 kg/s. Quality: 0.0704%.



(b). Change of void distribution. Central obstruction. Bubbly flow. Water: 1.1 kg/s. Air: 0.000775 kg/s. Quality: 0.0704%.

Fig. 3. Results from the flow blockage experiments.

2. The flow blockage experiment

The test section was a 25.4 mm i.d. 3 m long horizontal plexiglass tube. Air and water flow at selected rates through a mixer and a 3 m long approach to establish flow prior to the test section. Void fraction was measured at three stations using an optical probe. Typical results for peripheral and central flow obstructions blocking 25% of the flow area are shown in Fig. 3 and illustrate the phenomena mentioned above.

3. The TOFFEA computer program

The TOFFEA (Two Fluid Flow Equation Analysis) is a two dimensional computer program that is based on the same principles as the well known TEACH [7] single phase code, and was developed from the University of Ottawa version [11]. The early development of TOFFEA is described in [1] and [4]. The main features of the numerical scheme are now discussed.

3.1. Governing equations

The conservation equations governing adiabatic two fluid flow can be written in cartesian coordinates for fluid 1 as follows:

Conservation of mass:

$$\frac{\partial}{\partial x}(\alpha\rho u)_1 + \frac{\partial}{\partial x}(\alpha\rho v)_1 = 0;$$

Conservation of momentum:

$$\frac{\partial}{\partial x}(\alpha\rho u^2)_1 + \frac{\partial}{\partial x}(\alpha\rho uv)_1 = -\alpha_1\frac{\partial P}{\partial x} + \Gamma_{1x} + \phi_{12}(u_2 - u_1) + (\alpha\rho)_1 g_x, \quad (2a)$$

$$\frac{\partial}{\partial x}(\alpha\rho uv)_1 + \frac{\partial}{\partial y}(\alpha\rho v^2)_1 = -\alpha_1\frac{\partial P}{\partial y} + \Gamma_{1y} + \phi_{12}(u_1 - u_2) + (\alpha\rho)_1 gy. \quad (2b)$$

Here, α denotes volume fraction, ρ is density, u and v are velocity components, g is gravitational acceleration, and ϕ and Γ are functional relationships for interphase drag, and friction and viscous stresses respectively.

For brevity, these may be generalized in vector notation for each fluid k, $k = 1, 2$:

$$\left[\frac{\partial}{\partial x_j}(\alpha\rho u_j)\right]_k = 0, \quad (3)$$

$$\left[\frac{\partial}{\partial x_j}(\alpha\rho u_i u_j)\right]_k + \alpha_k\frac{\partial P}{\partial x_i} = S_{ik}. \quad (4)$$

The velocity component along coordinate x_i is denoted by u_i and $\partial u_j/\partial x_j \equiv \partial u/\partial x + \partial v/\partial y$; S_i contains the remaining terms from (2), and (4) comprises both (2a) and (2b).

3.2. Numerical solution

The solution scheme used in TOFFEA is a novel extension of the TEACH scheme for solution of the single fluid equations. The equations are first discretised by integrating over the control volumes for each fluid k, using the usual staggered grid concept:

$$\sum_i\left[(\alpha\rho uA)_{i+} - (\alpha\rho uA)_{i-}\right]_k = 0, \quad (5)$$

$$\sum_i\left[(\alpha\rho u_i u_j A)_{i+} - (\alpha\rho u_i u_j A)_{i-}\right]_k + A\alpha_k(P_{i+} - P_{i-}) = S_{ik}V, \quad (6)$$

A and V denote cross sectional area, and volume of the appropriate control volume and $i-$ or $i+$ denote upstream and downstream values.

If an initial pressure field is assumed, (6) can now be reduced to a form which gives a first estimate of mass flux in terms of the neighbouring mass fluxes and the relevant pressure gradient:

$$\left[a(\alpha\rho u)_1 + \sum_n(b\alpha\rho u)_n + e(P_{i+} - P_{i-})\right]_k = S_{ik}V. \quad (7)$$

The coefficients a, b and e contain the appropriate expressions reduced from (6), but (7) is now linearized, and can be solved in point or matrix form for a new matrix of mass fluxes. However, as the pressure field was not necessarily guessed correctly, the new mass fluxes will not satisfy the continuity equation (5) but yield a mass imbalance D:

$$\sum_i [(\alpha\rho uA)_{i+} - (\alpha\rho uA)_{i-}]_k = D_k. \tag{8}$$

As there are two continuity equations, the question of how they may be used most efficiently to extract the remaining two variables, pressure and volume fraction, must be addressed. Shah et al. [14] have suggested combining the two equations (8) to form a total mixture mass balance:

$$\sum_{k=1}^{2} \sum_i [(\alpha\rho uA)_{i+} - (\alpha\rho uA)_{i-}]_k = D_1 + D_2 = D_s. \tag{9}$$

A pressure equation can then be derived in the classical 'SIMPLE' [3] manner. It is necessary to find a new pressure filed that will drive D_s to zero. This can be done most efficiently by the Newton–Raphson technique, which yields

$$\mathrm{d}P = -D_s/(\mathrm{d}D_s/\mathrm{d}P). \tag{10}$$

Thus the required equation for the change in the pressure field, $\mathrm{d}P$, is obtained from rearranging (10):

$$\left[\frac{\mathrm{d}D_s}{dP}\right]\mathrm{d}P = -D_s. \tag{11}$$

As expression for $\mathrm{d}D_s$ is readily obtained by differentiating (8):

$$\begin{aligned}\mathrm{d}D_s &= \sum_{k=1}^{2} \sum_i [A\mathrm{d}(\alpha\rho uA)_{i+} - A\mathrm{d}(\alpha\rho uA)_{i-}]_k \\ &= -D_s,\end{aligned} \tag{12}$$

and from (7)

$$\begin{aligned}\mathrm{d}(\alpha\rho U)_{ik} = \frac{-1}{a_k}\bigg[&\sum_n b_n \mathrm{d}(\alpha\rho U)_n \\ &+ e(\mathrm{d}P_{i+} - \mathrm{d}P_{i-})\bigg]_k.\end{aligned} \tag{13}$$

Equation (13) is handled more readily by neglecting the variations in neighbouring mass fluxes, thus relating changes in each mass flux component at each point to the changes in pressure upstream and downstream, as in SIMPLE. Substituting the simplified equation (13) into (11) now yields a definitive matrix equation for the changes in the pressure field required to drive D_s towards zero, in the form

$$(A_1 + A_2)\mathrm{d}P = A\,\mathrm{d}F \quad -D_s = -(D_1 + D_2). \tag{14}$$

This is a Poisson equation and relates, for both fluids, the pressure correction required at each node to reduce the mass flux imbalances existing between nodes.

The pressure correction resulting from (14) will not precisely satisfy continuity because neighbouring flux variations were neglected in (13) and void fraction variations were also neglected. Shah et al. [14] obtain a further expression for void fraction by solving (8) for α_1 with D_1 imposed as zero.

Numerical experiments at CRNL using this approach for air-water flow have shown that the lighter fluid is poorly conserved. This can be explained by noting that (14) tends to dictate a pressure equation based on $D_1 + D_2 = 0$, not on $D_1 = D_2 = 0$. When $\rho_1 \ll \rho_2$, as for air and water, the lighter fluid hardly influences the pressure field at all, as $D_1 \ll D_2$.

This problem is circumvented in the TOFFEA code by normalizing each continuity equation (8) by the reference density ρ_0 to give a volumetric conservation equation equivalent to (9):

$$\sum_{k=1}^{2} \sum_i [(\alpha uAr)_{i+} - (\alpha uAr)_{i-}] = D_1' + D_2' \tag{15}$$

where $r = \rho_k/\rho_{0k}$, $D_k' = D_k/\rho_{0k}$.

In this case as $r_k \approx 1$, the phases and their contributions D' to the pressure equation are equally weighted, and a pressure field that imposes continuity in each is computed.

Further tests showed that while the volumetric pressure equation improved conservation, the rate of convergence was still slow when the continuity equation (8) for one fluid was used to solve for the volume fraction α_k, $k = 1$. The reason for this was identified as the absence of information about fluid 2 in (8).

As two linearized continuity equations are to be solved, any linear combination will yield solutions. The sum of the volumetric continuity equations was used to derive a pressure equation implicit in both fluids. The equation used to compute volume fraction must also be implicit in both fluids for efficient convergence. The continuity equations (8) may be normalized as before, and then subtracted. Noting $\alpha_1 = 1 - \alpha_2$, requiring $D_1 - D_2 = 0$, and using Q for volumetric flux uA, this yields

$$\sum_i \left[\alpha_1(Q_1 + Q_2)_{i*} - \alpha_1(Q_1 + Q_2)_{i}\right] = (Q_{i*} - Q_{i})_2. \tag{16}$$

This is a further linear matrix equation implicit in both fluids, and defining the field necessary to drive $D_1' - D_2'$ to zero. Hence the combination of (15) and (16) drives both $D_1' + D_2'$ and $D_1' - D_2'$ to zero and thus ensures both D_1' and D_2' are also driven to zero individually.

A final novel features is the modification of the iterative solution of (16) which ensures $\alpha \in (0, 1)$ at all times. If the bounds are merely imposed subsequent to solving (16), the integrity of the entire remaining solution sequence is degraded, the constraints must therefore be imposed implicitly. The method for this has been published separately [2].

This combination of the implicit volumetric pressure equation (15) and implicit volumetric volume fraction equation (16) has been incorporated in a two dimensional, two fluid computer program known as TOFFEA [3]. A three-dimensional two fluid program has also been developed using the same principles and is at prototype stage. The TOFFEA code has been used successfully to simulate separation of two fluid flows due to centrifugal forces in elbows [4] and due to gravity in horizontal communicating subchannels [3]. In this paper it is used to simulate the counter effect of homogenization caused by flow blockages. As the experiments shown were conducted in the bubbly flow regime, the constitutive relationships for wall and interphase friction suitable for bubbly flow described in [3] were used in TOFFEA.

4. Simulation of the experiments

Because the intent is not yet to simulate the experiment exactly, but to demonstrate first that the two fluid model has the ability to simulate the mechanisms involved, a two dimensional simulation was used with the same flow field height. Similar water and air flow rates and volume fraction were used and the code computed subsequent pressure drops and phase and velocity distribution. To simulate the mixer, uniformly distributed void was assumed at entry, subsequent void fraction profiles, velocity and slip profiles were then extracted.

4.1. Velocity and phase distribution fields

In order to illustrate the features of the flow field, demonstration problems at 50% entry void fraction were completed, as this yields more easily interpreted flow field graphs. Examples are shown in Figs. 4 and 5 for both the peripheral and the central flow blockages.

The figures show vector fields of volumetric flow ($u\alpha$) for each fluid, and a bar chart of void fraction α. Specific computed values of void fraction and air/water velocity ratio are shown in Figs. 6 and 7 both for local centre line conditions and overall cross sectional average conditions.

As the initial void fraction was uniform the tendency toward stratification can be observed in Figs. 4 and 5, but only the test section length was used in the computation, so 'fully developed' stratified flow was not established. The stratification produces quite a large air flux near the top of the test section, even though zero slip is imposed at the wall.

The obstructions do affect the upstream flow patterns. As expected, the central obstruction causes a divergent deflection upstream, while the peripheral obstruction causes a convergent deflection. Downstream, the restoring tendencies are, of course, opposite. For the central obstruction, the downstream restoring velocities are generally downward in the upper half of the channel where most of the air resides, tending to drive this air back towards the centre of the channel and produce the enhanced mixing observed in the experiment. By contrast, the peripheral obstruction generates upward downstream restoring velocities and the mixing effect is less. The diagrams also clearly show that effects of the central obstruction persists

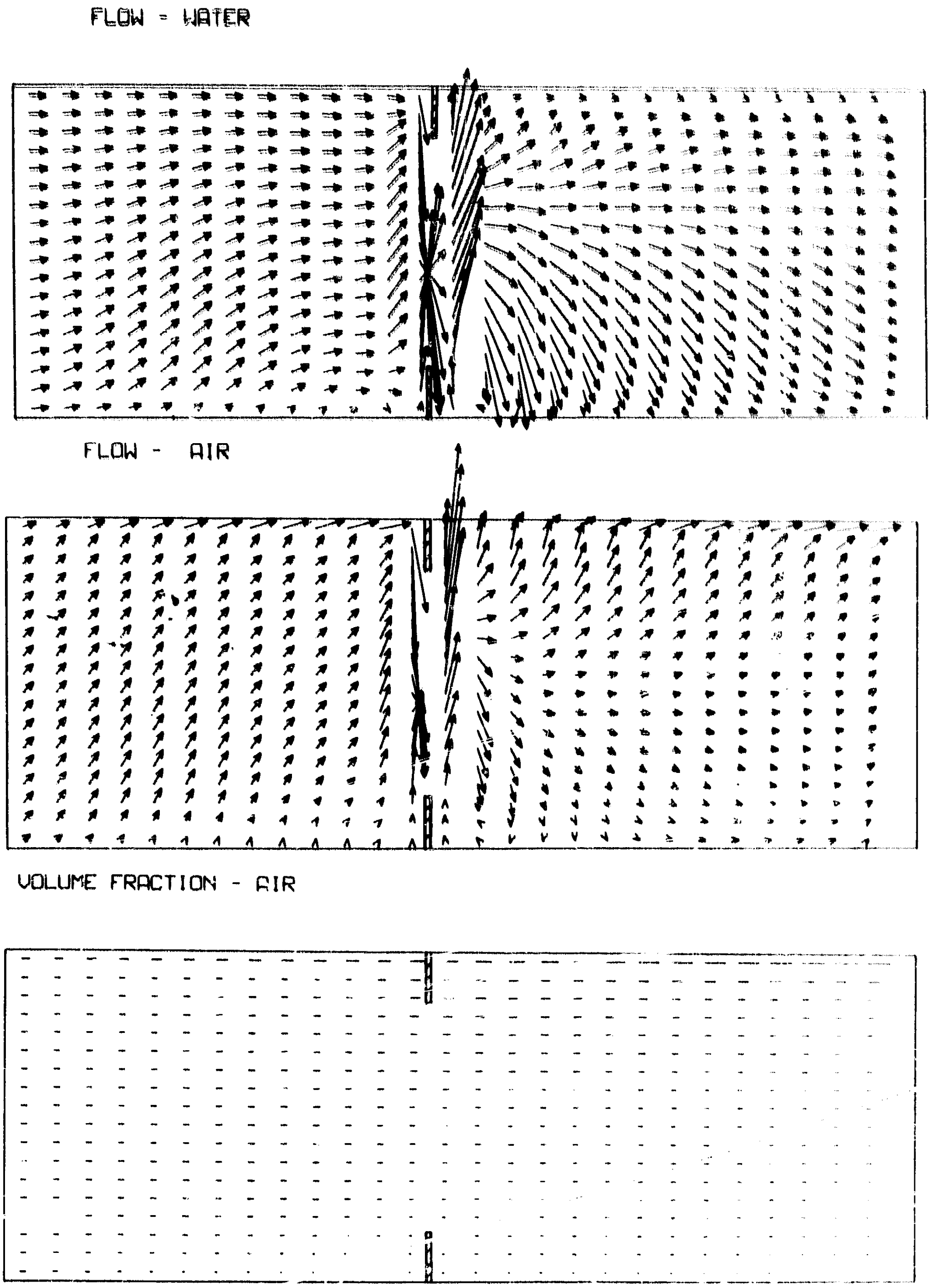


Fig. 4. Computed volumetric flow fields and void fraction profile for the peripheral blockage.

much further downstream than for the peripheral obstruction, again as observed in the experiment.

In these, the vertical dimension has been augmented in scale to accentuate clearly the distinctly different velocity components in the two phases. The variation in slip along and across the channel can be deduced qualitatively from these vector fields, quantitative figures are given below.

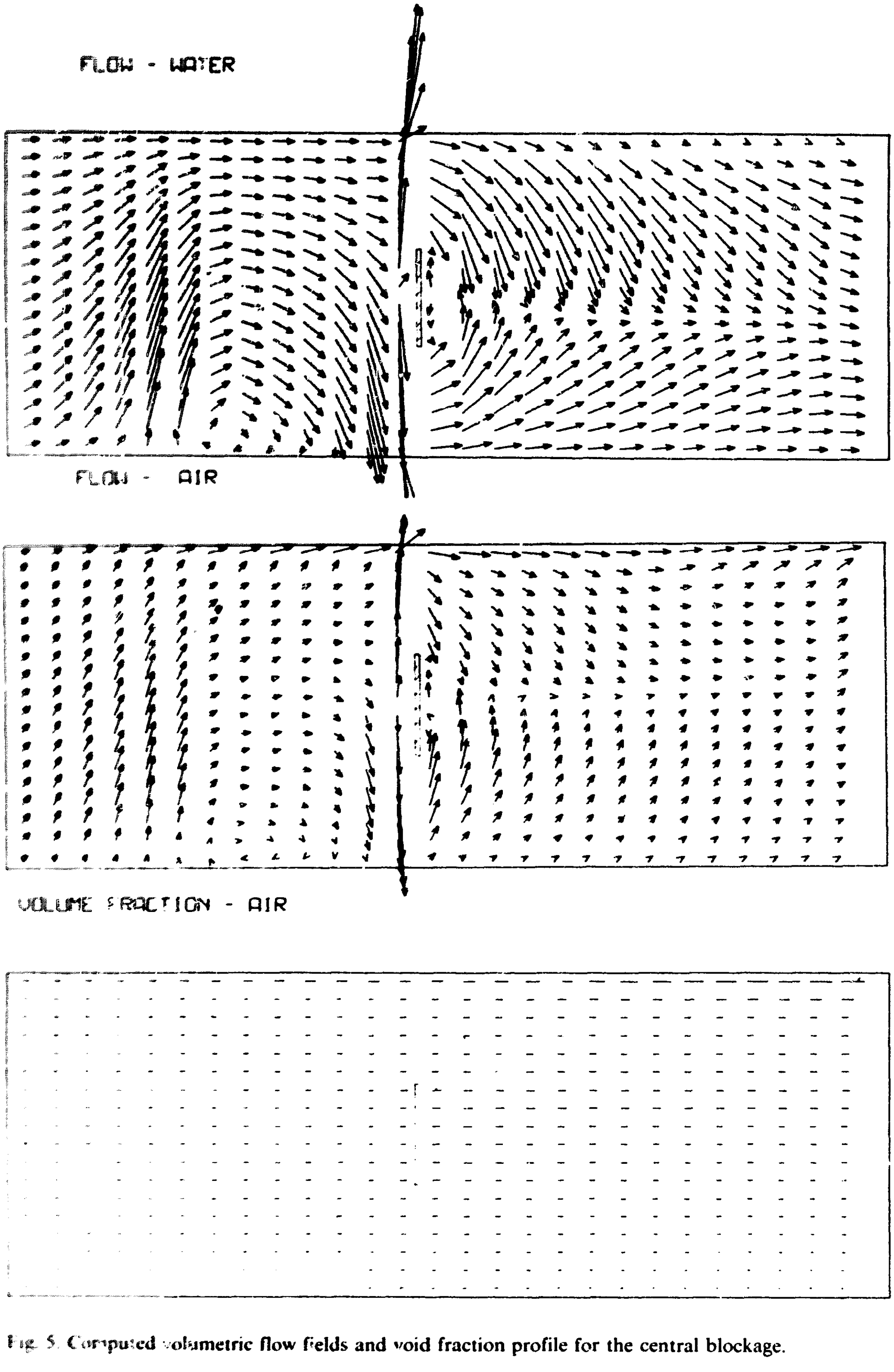


Fig. 5. Computed volumetric flow fields and void fraction profile for the central blockage.

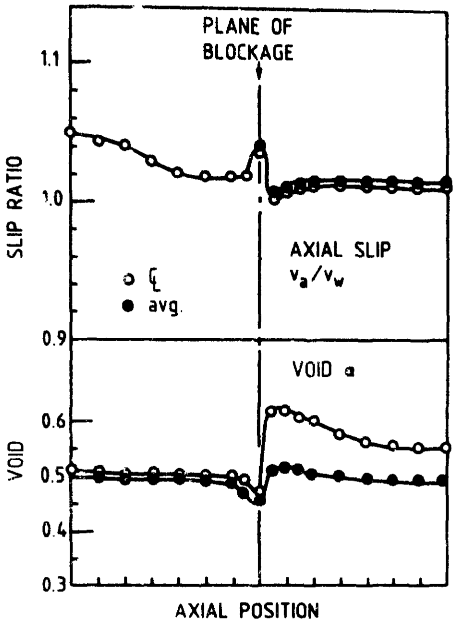


Fig. 6. Computed void fraction and slip ratio, average and centre line values (peripheral).

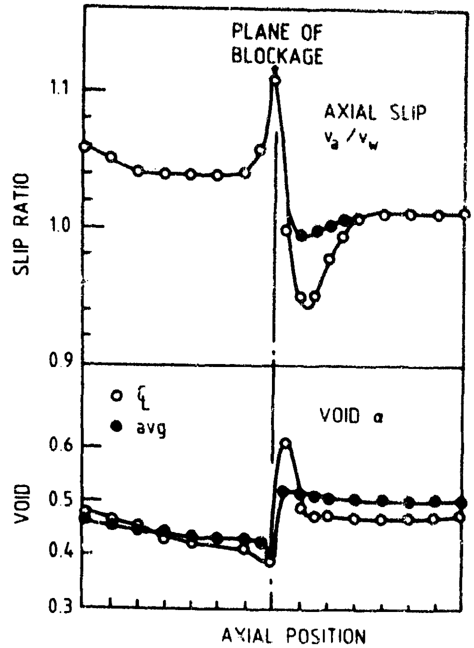


Fig. 7. Computed void fraction and slip ratio, average and centre line values (central).

4.2. Particular analysis of velocity and phase distribution

An overall impression of the nature of velocity and phase distribution can be gleaned from the flow fields depicted, particular features are now discussed. Graphs of computed cross sectional average void fraction and axial velocity ratio (air/water) are given for the peripheral and central obstruction cases in Figs. 6 and 7. Note that the trends in void fraction and slip are again in agreement with the experiments of Figs. 3 and 4. In particular, the slip can be seen to increase in the plane of the obstruction and decrease markedly just downstream, with the overall void fractions varying inversely as expected. The increase in slip and slight decrease in void in the plane of the constriction are due to the greater acceleration induced in the lighter phase. The subsequent deceleration causes an opposite tendency, but the mixing effects cause comparably larger variations, the average void increasing above upstream values, and the average slip decreasing proportionally. This is also compatible with the experiment. The computed centre line values do not show strict proportionality, but are not necessarily constrained to, as this is a local measurement in a varying field.

Again as in the experiment, the central obstruction causes a significantly greater redistribution than does the peripheral one. The flow around the central obstruction shows the liquid phase behaving in a similar way as single phase flow past an obstruction. A zone of recirculation is formed in the wake, and air flowing into this zone causes a slow moving air pocket, giving a velocity ratio smaller than one. This accounts for the high void region observed in both experiments and calculation. The effect is less pronounced in the computation due to the two-dimensional treatment of the flow field.

Further downstream, pressure recovery occurs with an associated increase in slip and decrease in void. The recovery is computed to be much faster for the peripheral obstruction than for the central one, again as observed in experiment.

5. Conclusion

As stated, the intent of the study was to determine whether the two fluid model contains the necessary flexibility to simulate the several physical phenomena which occur in two phase flow around a constriction. Because this was a feasibility study no attempt was made to simulate the three dimensional nature of the flow, and as this is a major influence no attempt was made to precisely reproduce experimental conditions. In spite of these approximations it is clear that the numerical simulation reproduces the main phenomena of interest and therefore contains the necessary mechanisms. It would be counterproductive to attempt to tune a two dimensional model to agree with the experiment in more detail, but the agreement is sufficient to promise reasonable success for a three dimensional simulation. One feature on which a discrepancy exists between the computed and experimental results is the precise amount of mixing. In the central blockage experiment, the mixing appears to temporarily invert the void fraction profile. As the blockage is symmetric, this is quite surprising and may be due to the inducement of swirl, a phenomenon that would require multidimensional simulation.

References

[1] M.B. Carver, Conservation and the pressure continuity relationship in multidimensional two fluid computation, in: R. Vichnevetsky, Ed., *Advances in Computer Methods for Partial Differential Equations, IV* (IMACS Press, 1981) 168–172.

[2] M.B. Carver, A method of limiting intermediate values of volume fraction in iterative two fluid computations, *J. Mech. Eng. Sci. 24* (4) (1982) 221–224.

[3] M.B. Carver, Numerical computation in two fluid flow using the two dimensional TOFFEA code, AECL-8066 (1983).

[4] M.B. Carver, Numerical computation of phase separation in two fluid flow, *J. Fluids Eng. 106* (1984) 147–153.

[5] M.B. Carver, L.N. Carlucci and W.W.R. Inch, Thermalhydraulics in recirculating stream generators, THIRST Code User's Manual, Atomic Energy of Canada Limited report AECL-7254 (1981).

[6] M.B. Carver, A. Tahir, D.S. Rowe, A. Tapucu and S.Y. Ahmad, Computational analyses of two-phase flow in horizontal rod bundles, *Nucl. Eng. & Des. 66* (1984) 101–123.

[7] A.D. Gosman and W.M. Pun, Lecture notes for course in calculation of recirculating flows, Imperial College Report HTS/74/2.

[8] W.T. Hancox and S. Banerjee, Numerical standards for flow boiling analysis, *Nucl. Sci. & Eng. 64* (1977) 106–123.

[9] W.G. Mathers, W.W. Zuzak, B.H. McDonald and W.T. Hancox, On finite difference solutions to the transient flow-boiling equations, *CSNI Specialists Meeting on Transient Two-Phase Flow*, Toronto (1976).

[10] D.S. Rowe, C.A. McMonagle and S.Y. Ahmad, Numerical solutions for two-phase flow in horizontal nuclear fuel assembles, in: R. Vichnevetsky, Ed., *Advances in Computer Methods for Partial Differential Equations, IV* (IMACS Press, 1981) 351–358.

[11] M. Salcudean and Z. Abdelrehim, Heat transfer in turbulent recirculation flows affected by buoyancy in rectangular cavities, *Chem. Eng. Comm. 4* (1980) 249–268.

[12] M. Salcudean, J.H. Chen, D.C. Groeneveld and L. Leung, Effect of flow obstruction geometry in pressure drop in horizontal air water flow, *Internat. J. Multiphase Flow 9* (1) (1983) 73–85.

[13] M. Salcudean, D.C. Groeneveld and L. Leung, Effect of flow obstruction on void distribution in horizontal air-water flow, *Internat. J. Multiphase Flow 9* (1) (1983) 91–96.

[14] V.L. Shah et al., Numerical procedures for calculating steady/unsteady single/two phase three dimensional fluid flow, NUREG-CR-0789 (1979).

[15] M. Shoukri, A. Tahir and M.B. Carver, Numerical simulation of two phase flow in horizontal interconnected subchannels, *CNS/ANS International Conference on Numerical Methods in Nuclear Engineering*, Montreal (1983).

[16] A. Tahir and M.B. Carver, ASSERT & COBRA prediction of flow distribution in vertical bundles, *Proc. CNS/ANS International Conference on Numerical Methods in Nuclear Engineering*, Montreal (1983) 371–387.