JOURNAL OF COMPUTATIONAL PHYSICS 34, 390–400 (1980)

# Adding Limited Compressibility to Incompressible Hydrocodes*

C. W. HIRT AND B. D. NICHOLS

*Theoretical Division, Group T-3, University of California, Los Alamos Scientific Laboratory, Los Alamos, New Mexico 87545*

Received April 16, 1979; revised May 31, 1979

A simple modification is described that may be used to add limited compressibility effects to incompressible hydrodynamics computer codes. Several sample calculations are discussed and used to compare the relative advantages of implicit and explicit time integration methods. It is also shown that the use of an artificially reduced speed of sound is not, in general, a good approximation for low speed fluid problems.

## I. INTRODUCTION

Although incompressible hydrodynamics codes are useful for a great variety of low speed fluid dynamic problems, there do arise situations in which it is desirable to include some compressibility effects. For example, in Boiling Water Reactor pressure suppression pools the injection of steam, arising from a normal operating transient or from a postulated loss of coolant accident, can produce steam bubbles that grow then suddenly collapse generating sharp pressure pulses in the pool. Most of the growth and collapse dynamics is well described by treating the water as an incompressible fluid [1]. However, at the instant of complete collapse the pressure pulses produced exist for times comparable to the time for sound waves to travel across the pool, so that compressibility effects are needed to accurately describe the pulse shape.

Another example arises in laboratory model tests of pressure suppression pools. When air bubbles are present in the pool water, the water-air mixture can have a large enough compressibility to significantly influence the test data [2]. In both examples, the compressibility effects are associated with acoustic waves and a fully nonlinear compressible model is not required.

Thus, to account for such effects we have devised a simple modification that may be made to many existing incompressible hydrocodes. We illustrate the method in the SOLA–VOF code [3], which uses a variant of the well-known Marker and Cell (MAC) technique [4]. This code has been successfully used [5] to model suppression pool experiments performed at MIT (Ref. [6]) and at SRI (Ref. [7]). The modification is directly transferable to most other codes originally designed only for incompressible

* Work performed under the auspices of the U.S. Department of Energy. The U.S. Government's right to retain a nonexclusive royalty-free license in and to the copyright covering this paper, for governmental purposes, is acknowledged.



0021-9991/80/030390–11$02.00/0

fluids, and which use the primary dependent variables of pressure and velocity. Several of these codes are publicly available from the National Energy Software Center [8].

The basic idea and its limitations are outlined in the next section. Finally, Section III presents the results of several test calculations illustrating the capabilities of the new method. One observation drawn from the examples is that it is not always possible or wise to approximate incompressible flows using a compressible flow model with an artificially low sound speed.

## II. Theory and Implementation

The equations of motion describing a compressible fluid are

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \mathbf{A}, \tag{1}$$

$$\frac{\partial \rho}{\partial t} + \mathbf{u} \cdot \nabla \rho + \rho \nabla \cdot \mathbf{u} = 0, \tag{2}$$

where $\mathbf{A}$ represents all gravitational accelerations, viscous stresses, or other impressed accelerations on the fluid. The exact form of $\mathbf{A}$ is not important for the following discussion. To complete this equation set we assume that the fluid pressure is only a function of density. In particular, that

$$\frac{dp}{d\rho} = c^2, \tag{3}$$

where $c$ is the adiabatic speed of sound. Although $c$ may be time- or space-dependent, in this note we shall simply assume it is a constant. Thus, our basic approximation is that only adiabatic fluid changes are to be considered. This is consistent with the standard acoustic approximation [9].

We shall also assume that density changes remain small, i.e., $\delta\rho/\rho \ll 1$. This assumption implies that $u/c \ll 1$ (see, for example, Ref. [9, p. 245]). It is the small density variation assumption that makes the inclusion of compressibility effects in SOLA–VOF especially easy. However, this assumption also requires us to refer to the method as one of "limited compressibility." Because density changes are small we expand Eqs. (1)–(2) in powers of $\delta\rho/\rho_0$ about the constant mean density $\rho_0$. Retaining only the lowest order terms, and using Eq. (3) we have

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho_0} \nabla p + \mathbf{A}, \tag{4}$$

$$\frac{1}{c^2} \frac{dp}{dt} + \rho_0 \nabla \cdot \mathbf{u} = 0. \tag{5}$$

Furthermore, because $u/c \ll 1$ we have to the same order,

$$\frac{dp}{dt} = \frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p$$
$$\approx \frac{\partial p}{\partial t}.$$

Without loss of generality we may take $\rho_0$ equal to unity, provided we remember this when scaling to any particular set of dimensional units. Thus, Eqs. (4)–(5) become

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \mathbf{A}, \tag{6}$$

$$\frac{1}{c^2} \frac{\partial p}{\partial t} + \nabla \cdot \mathbf{u} = 0. \tag{7}$$

Equations (6)–(7), which constitute the limited compressibility model, differ from the purely incompressible equations only by the appearance of the pressure term in Eq. (7); the momentum equations remain unchanged. It should also be noted that all boundary conditions involve the specification of velocities or pressures and are independent of whether the fluid is compressible or not.

To add the extra term appearing in the continuity equation, Eq. (7), to typical incompressible fluid algorithms it is necessary to introduce one additional storage array for the old-time-level pressures and to add an input constant for the sound speed.

In all incompressible codes, the continuity equation is treated implicitly. For example, in MAC codes the condition of a zero velocity divergence is treated as an equation for the pressure and is solved by iteratively adjusting the pressures and velocities [4, 8]. Alternatively, a Poisson equation for pressure may be derived by combining the incompressibility condition with the momentum equations. Iterative solutions of this equation are equivalent to the usual MAC solution method [10]. In any case, there is a difference equation and corresponding pressure for each cell of the computational mesh. A solution is usually obtained through the use of a Newton–Raphson method. For example, define $F = \nabla \cdot \mathbf{u}$ for a typical cell in the mesh. The approximate pressure change in that cell needed to drive $F$ to zero is

$$\delta p = -F \Big/ \frac{\partial F}{\partial p}. \tag{8}$$

The function $F$ is linear in the pressure and only one application of Eq. (8) is needed in each cell during a pass through the mesh. It is necessary, however, to sweep through the mesh several times, because an adjustment of the pressure in one cell will affect the divergence of all neighboring cells.

In the case of limited compressibility the $F$ function for a cell becomes

$$F = \frac{p - p^n}{c^2\, \delta t} + \nabla \cdot \mathbf{u}$$

where $p$ is the new pressure to be solved for and $p^n$ is the pressure from the previous cycle. The corresponding derivative is

$$\frac{\partial F}{\partial p} = \frac{1}{c^2\, \delta t} + \left(\frac{\partial F}{\partial p}\right)_0 \tag{9}$$

where $(\partial F/\partial p)_0$ is the relaxation parameter used in the purely incompressible case. An iterative solution using Eq. (8) will drive $F$ to zero in each cell, which insures that Eq. (7) is satisfied.

Various generalizations of this limited compressibility model are possible. For example, the sound speed could be a specified function of space and time, which would require the use of another storage array. If the speed is a function of temperature or air bubble void fraction then it might be desirable to add a full transport equation for the temperature or void fraction. The restriction to small density variations could also be eliminated, but would then require an array for the density, inclusion of the convective density changes neglected in Eq. (7), density boundary conditions, and modifications to the momentum equation. Although the same numerical solution strategy currently used, for example in the SOLA–VOF code, would be applicable in this general case, it clearly requires more extensive changes to the code.

## III. Sample Calculations and Discussion

### A. *A One-Dimensional Test Problem*

A simple one-dimensional problem demonstrates the capability of the proposed method to treat the dynamics of slightly compressible fluids. Consider a column of fluid of unity height, bounded by a rigid wall at one end and by a free surface at the other end. The fluid is initially at rest, has zero pressure, and a dimensionless sound speed of 100. At time $t = 0$, an applied pressure of unit magnitude is applied to the free surface. At time $t = 0.005$, the pressure wave should have moved a distance of 0.5 into the fluid column. Figure 1 shows the results at this time obtained by three different computations, which resolved the fluid column by 20 cells of height 0.05. In the first two cases, a time step of $\delta t = 0.00025$ was chosen. With this small time step, sound waves move a distance of 0.025 per step, which is one-half the width of a mesh cell used for the calculation. Thus, the Courant stability criterion needed for explicit integration schemes is satisfied and the implicit pressure solution used in the SOLA–VOF code is unnecessary. To provide a basis for comparison, therefore, we modified the SOLA–VOF code for one calculation to be explicit. The results of this calculation are compared in Fig. 1 with a calculation using the same small time step and the implicit SOLVA–VOF method described in Section II. Approximately three to four iterations per time step were required to satisfy the convergence criterion ($|F| \leqslant 10^{-3}$) chosen for the iteration. The two calculations provide somewhat different approximations to the wave front, which should be a step discontinuity in the present model. The explicit method exhibits oscillations trailing the front, while the implicit method shows a monotonic increase. Although the explicit front is steeper, its width including oscillations is comparable to the width of the implicit wave front. An explanation of this behavor will be given below, but first it is worthwhile to compare these results with calculational results obtained using a time step that exceeds the Courant stability

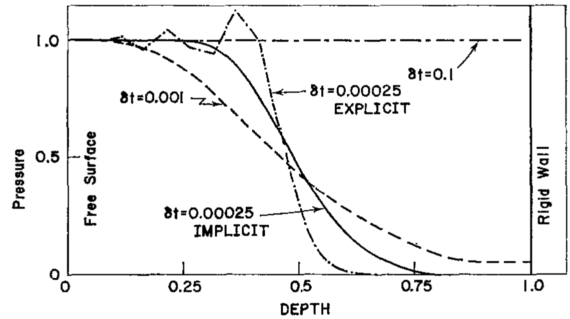


FIG. 1. Pressure profiles computed in one-dimensional test problem, using different time steps and integration methods.

limit. In Fig. 1, the curve corresponding to $\delta t = 0.001$ was obtained with twice the maximum time step that can be used in the explicit method. This calculation has moved the mid-point of the pressure wave approximately 10 cells in 5 cycles of computation, but it does so at the expense of accuracy. The pressure wave is considerably broader, and continues to broaden as the time step increases. In fact, when the time step is 100 times larger ($\delta t = 0.1$) the pressure is nearly equal to the applied value of unity throughout the entire column of fluid after one cycle of calculation, see Fig. 1. Many iterations are required to reach convergence, but the result is what one would expect in an "incompressible" fluid. That is, when $\delta t = 0.1$, the pressure wave can traverse the length of the fluid column 10 times during each time step, and with some dissipation present (as in real fluid) the wave should eventually damp out leaving a constant unit pressure everywhere in the fluid. The numerical dissipation built into the implicit integration method does this automatically. In contrast, an explicit calculation without dissipation would continue to follow the wave and it would be necessary to time average the results to obtain a uniform pressure distribution.

The above, empirical, results regarding accuracy in the explicit versus the implicit integration method can be approached in another way. If the finite-difference equations used in the explicit and implicit methods are linearized, a Fourier analysis can be performed to obtain the time amplification factor, $r$, for a mode with wave number $k$. The result for the explicit method shows that $r = 1$ when the Courant number, $c\delta t/\delta x$, is less than or equal to unity. When the Courant number exceeds unity $r$ is greater than one and the explicit method is unstable. Therefore, when stable, the explicit method exhibits no damping of pressure waves. There is, however, some dispersion and this accounts for the oscillations and spread of the pressure wave in Fig. 1. In contrast, the implicit method has the amplification factor $(1 + \mu^2)^{-1/2}$, where $\mu = 2(c\delta t/\delta x)\sin(k\delta x/2)$. The implicit method is always stable because $r$ is never greater than unity, but by the same token pressure waves are always damped. It is this feature of the implicit method that characterizes both its weakness and its strength. In particular, when Courant numbers much larger than unity are used in a calculation pressure waves are highly damped, which is the reason that the method passes to the correct incompressible limit.

On the other hand, when very small Courant numbers (much less than unity) are used, the implicit method has little damping and will give results that are at least as good as those obtained with an explicit calculation.

Unfortunately, economy usually requires the use of a Courant number near unity. To understand what happens in this case, consider a Courant number of 1/2, which is typical in explicit calculations. Then the implicit amplification factor is $[1 + \sin^2(k\delta x/2)]^{-1/2}$. Clearly disturbances having the smallest resolvable wave lengths ($\sim 2\delta x$) are highly damped, but the damping rapidly diminishes as the wave length increases. Thus, when the Courant number is near unity, the implicit method will only be accurate for waves that are spatially well resolved. This aspect of the implicit method is not all bad, however, for it means there is an automatic mechanism for removing high frequency noise (e.g., trailing oscillations) that sometimes is a problem in explicit calculations.

In summary, when it is of interest to investigate wave phenomena on short time scales, then small time steps that limit wave propagation to less than one cell per time cycle must be used for reasons of accuracy. On the other hand, for phenomena on time scales long with respect to the transit time of sound waves, the implicit method provides the correct asymptotic answer, while an explicit method must be modified to have dissipation, or its results must be time averaged. Unfortunately, for more complicated problems, the use of wave dissipation or time-averaging techniques may be difficult to apply without affecting some of the slow transient phenomena of interest. Furthermore, as the distinction between slow and fast processes gets more pronounced (e.g., as $c$ gets larger with respect to the fluid particle speed) an explicit method will require increasingly more computational time, whereas the implicit method will not. It is in this limit that implicit methods are far superior to explicit schemes.

It has been suggested that one solution to the last problem is to use an explicit compressible hydrocode with an artificially small sound speed. The idea being that a flow with Mach number $(u/c)$ on the order of 0.3 or less will closely approximate an incompressible flow. That this is not always the case may be seen from the next example.

B. *Influence of Air Bubbles on Suppression Pool Dynamics*

When air or steam is forced through a pipe (vent) whose end is immersed in a pool of water, Fig. 2, the pool container is subject to a variety of hydrodynamic forces. Several experimental programs have recently been conducted [6, 7, 11, 12] to investigate these forces to better understand similar phenomena [13] that occurs in postulated loss of coolant accidents in boiling water reactors (BWR). During the course of the MIT experimental study to confirm scaling laws for vent clearing phenomena [6] it was discovered that the presence of small air bubbles in the pool produced significant oscillations in the pressure measured on the pool floor. Moreover, these oscillations were observed to increase in magnitude as the bubble number increased. An interpretation of this phenomenon is that the bubbles provide a springiness to the fluid, which can then "bounce" on the floor.

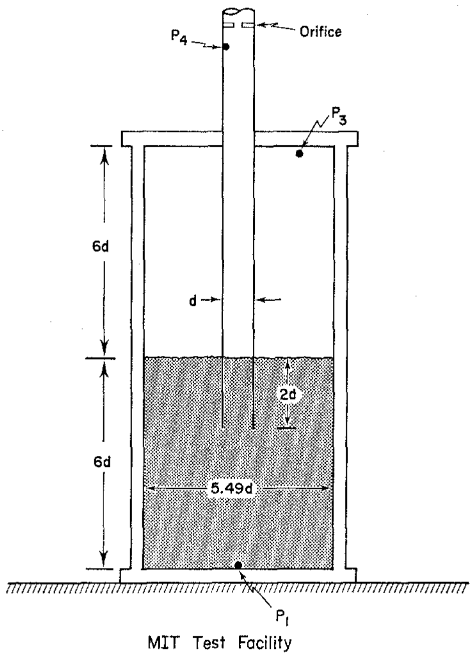


FIG. 2. Schematic of suppression pool model apparatus used at MIT. Floor pressure transducer located at $P_1$.

The springiness in a fluid caused by suspended air bubbles may be represented by asigning to the fluid a smaller velocity of sound (or bulk modulus). Thus, the new compressibility feature in the SOLA–VOF code can be used to theoretically investigate the consequences of having air bubbles in the pool during vent clearing. Results from a typical calculation simulating the MIT tests are illustrated by the fluid configurations and velocity fields shown in Fig. 3. Figure 4 shows the computed floor pressure results from three calculations in which the only difference is the fluid sound speed. In Fig. 4A,

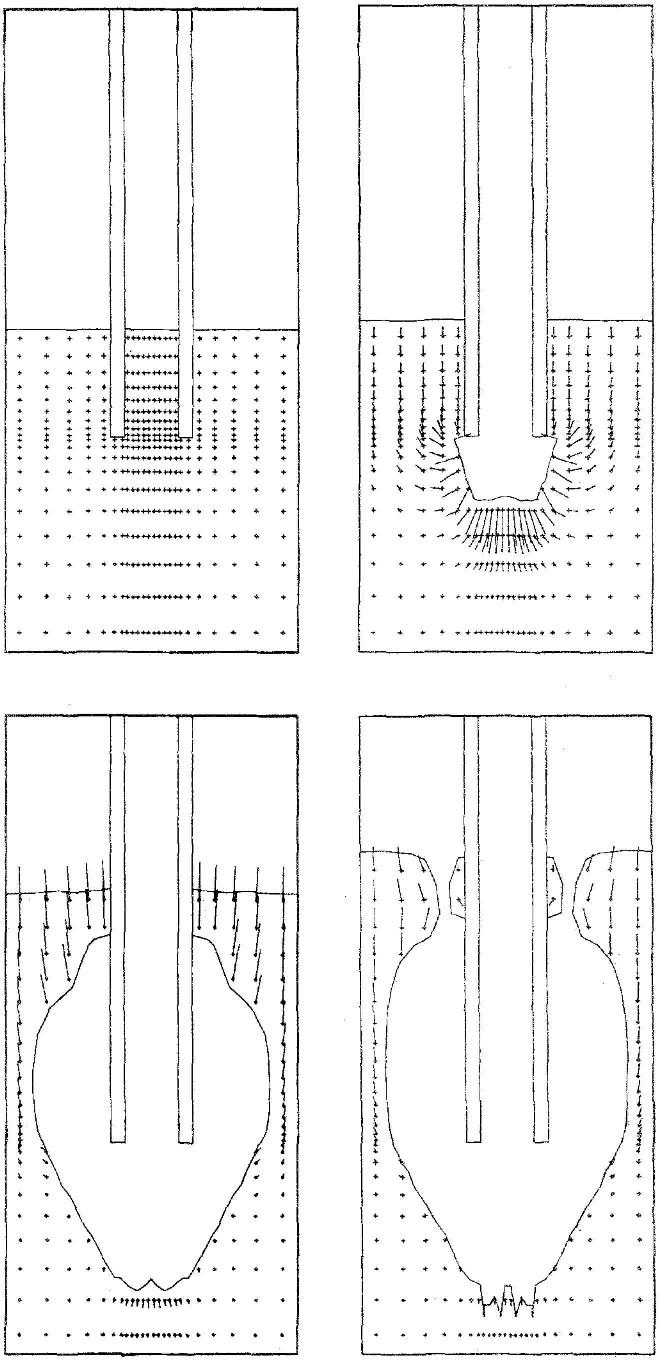

FIG. 3. Typical velocity field and fluid configuration history as computed by the SOLA-VOF code in a simulation of an MIT test.

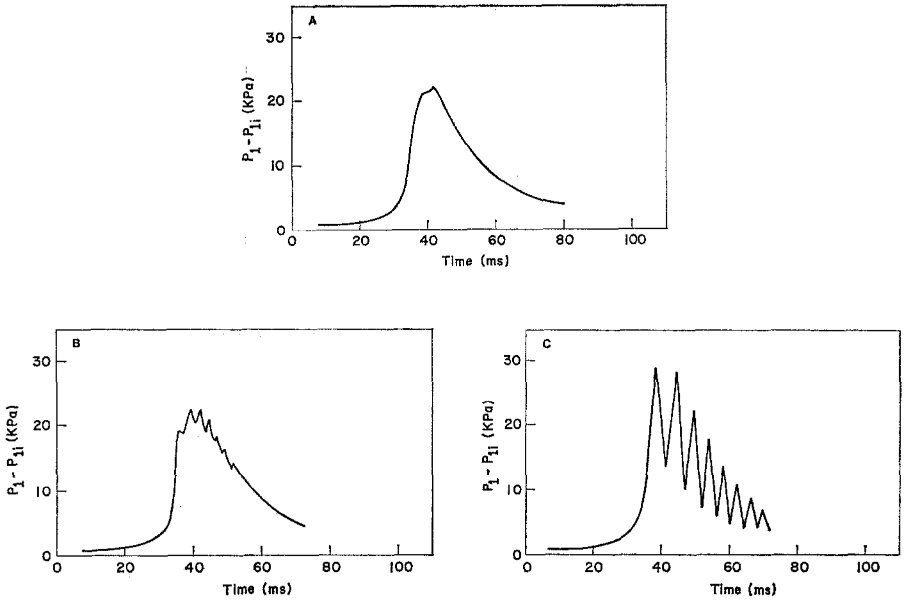


FIG. 4. (A) Floor pressure history computed during vent clearing with an incompressible fluid. (B) Floor pressure history computed during vent clearing with a fluid whose sound speed is one-fifth that of water. (C) Floor pressure history computed during vent clearing with a fluid whose sound speed is one-tenth that of water.

the fluid was treated as incompressible ($c = \infty$). In Fig. 4B, the sound speed was set at approximately one-fifth the speed of sound in water, while in Fig. 4C, it was approximately one-tenth the normal speed. A comparison of the three results shows dramatic differences. As the sound speed is lowered, oscillations in floor pressure rapidly increase in magnitude and in period. The computed results look remarkably like the MIT experimental results [6], although a direct comparison cannot be made, because we did not use the same test conditions.

The above results have important implications with regard to the proper use of compressible versus incompressible hydrodynamic computer codes. For low speed problems such as that described here, in which the fluid moves slowly with respect to the speed of sound, it would be impractical to use a compressible model. To do so would require the computation of pressure waves traversing back and forth across the pool thousands of times during the vent clearing process. Even if the computations could be carried out accurately, they would consume an unacceptable amount of computer time. As a possible way around this difficulty, some investigators have suggested reducing the sound speed in the fluid to the point where computations can be economically carried out. The justification given for this procedure is that flows at Mach numbers ($u/c$) below about 0.3 (e.g., Ref. [14, p. 9]) often behave much like incompressible flows. While this may be true for some problems, provided the right flow features are considered, it is clear from the above example that the procedure is not general.

In particular, hydrodynamic forces may exhibit unrealistic acoustic oscillations. Also in coupled fluid-structure calculations, artificially low acoustic frequencies may excite incorrect structure response. Similarly, artificially large density variations associated with reduced sound speeds could adversely affect buoyancy-driven flows. Although other examples could be cited, this list is sufficient to emphasize that the practice of approximating an incompressible flow by a compressible flow with small, but not negligible compressibility, must be employed with extreme caution.

A useful criterion covering some of the above cases has been given by Landau and Lifshitz (Ref. [9, p. 24]). In addition to the low Mach number requirement, there is another condition that must be satisfied for transient problems before a flow may be considered as incompressible. If $\tau$ and $L$ represent time and space scales in which the fluid velocity changes significantly, then this additional condition is $c\tau \gg L$. Thus, to approximate an incompressible flow with a compressible flow model requires that the sound speed be chosen sufficiently large that pressure waves will travel distances large with respect to $L$ in time $\tau$.

## IV. Acknowledgments

This study was performed under contract to the Electric Power Research Institute, Contract RP 965-3. The authors wish to acknowledge their thanks to John J. Carey, project manager, for his interest and encouragement in this work. We also wish to thank a reviewer for catching an error in the original manuscript.

## References

1. B. D. Nichols and C. W. Hirt, *in* "Proceedings, ANS 1978 Annual Meeting, San Diego, Calif., June 1978."
2. W. G. Anderson, P. W. Huber. and A. A. Sonin, *in* "Proceedings, ANS Thermal Reactor Safety Meeting, Sun Valley, Idaho, July 1977."
3. B. D. Nichols and C. W. Hirt, *in* "Proceedings, First International Conference on Numerical Ship Hydrodynamics, Gaithersburg, Md., Oct. 1975."
4. F. H. Harlow and J. E. Welch, *Phys. Fluids* **8** (1965), 2182.
5. B. D. Nichols and C. W. Hirt, "Numerical Simulation of BWR Vent-Clearing Hydrodynamics," Electric Power Research Institute Report, to be published.
6. W. G. Andersom, P. W. Huber. and A. A. Sonin, "Small Scale Modeling of Hydrodynamic Forces in Pressure Suppression Systems, Final Report," prepared for the Division of Reactor Safety Research, U. S. Nuclear Regulatory Commission, NUREG/CR-0003, 1978.
7. R. L. Kiang and P. R. Jeuck, "A Study of Pool Swell Dynamics in a Mark II Single-Cell Model," SRI International Report, Oct. 1978 (draft).
8. C. W. Hirt, B. D. Nichols, and N. C. Romero, "SOLA – A Numerical Solution Algorithm for Transient Fluid Flows," Los Alamos Scientific Laboratory report LA-5852, 1975; code available from National Energy Software Center, 9700 South Cass Avenue, Argonne, Ill. 60439.
9. L. D. Landau and E. M. Lifshitz, "Fluid Mechanics," Pergamon/Addison–Wesley, Reading, Mass., 1959.
10. J. A. Viecelli, *J. Comput. Phys.* **8** (1971) 119.

11. E. K. Collins and W. Lai, "Final Air Test Results for the 1/5-Scale Mark I Boiling Water Reactor Pressure Suppression Experiment," Lawrence Livermore Laboratory Report UCRL-52371, 1977; also NUREG/CR-0151, 1977.
12. I. Catton *et al.*, "Suppression Pool Dynamics, Quarterly Progress Report, Oct. 1–Dec. 31, 1976," prepared for the Division of Reactor Safety Research, U. S. Nuclear Regulatory Commission, NUREG/CR-0264-2, 1977.
13. T. Fernandez, *Electr. Power Res. Inst. J.*, April 1976.
14. H. Schlichting, "Boundary Layer Theory," 4th ed., McGraw-Hill, New York, 1960.