CONF-800661--2

LA-UR - 80 - 1631

TITLE: A FRACTIONAL VOLUME OF FLUID METHOD FOR FREE BOUNDARY DYNAMICS

AUTHOR(S): B. D. Nichols
C. W. Hirt
R. S. Hotchkiss

SUBMITTED TO: Seventh International Conference on Numerical Methods in Fluid Dynamics
Stanford University, Palo Alto, CA
June 23-27, 1980

University of California

LOS ALAMOS SCIENTIFIC LABORATORY
Post Office Box 1663 Los Alamos, New Mexico 87545
An Affirmative Action/Equal Opportunity Employer

Form No. 836 R3
St. No. 2629
12/78

UNITED STATES
DEPARTMENT OF ENERGY
CONTRACT W-7405-ENG. 36

A FRACTIONAL VOLUME OF FLUID METHOD FOR FREE BOUNDARY DYNAMICS*

B. D. Nichols, C. W. Hirt, and R. S. Hotchkiss
Theoretical Division, Group T-3
University of California
Los Alamos Scientific Laboratory
Los Alamos, NM 87545

I. INTRODUCTION

In this paper we describe an exceptionally versatile Eulerian method to track free boundaries that undergo large deformations. The method follows regions of fluid, defined by a volume of fluid function, in contrast to a direct representation of free boundaries, such as marker particle chains. This new scheme is superior to previous free boundary methods because it requires minimum computer storage, avoids logic problems associated with the creation or destruction of disjoint fluid regions, is applicable to arbitrarily contorting flows, and is directly extendible to three-dimensional problems.

II. THE VOF METHOD

The volume of fluid (VOF) method is based on a function F whose value is unity at any point occupied by fluid and zero otherwise. The average value of F in a computational cell represents the fractional volume of the cell occupied by fluid. In particular, a unit value of F corresponds to a cell full of fluid, while a zero value indicates that the cell contains no fluid. Cells with F values between zero and one must then contain a free boundary.

In addition to defining which cells contain a boundary, the VOF method defines where fluid is located in a boundary cell. The normal direction to the boundary lies in the direction in which the value of F changes most rapidly. Because F is a step function, however, its derivatives are computed in a special way. Finally, knowing both the normal direction and the value of F in a boundary cell, a line cutting the cell can be constructed that approximates the interface there. This boundary location can then be used in setting boundary conditions. In addition, surface curvatures can be computed from the F distribution when surface tension forces must be considered.

The time dependence of F is governed by a kinematic equation stating that the F values flow with the fluid. If standard finite-difference approximations were used to compute the advection of F, excessive numerical smearing of the F function would occur and interfaces would lose their definition. Fortunately, the fact that F is a step function with values of zero or one permits the use of a special flux approxi-

*This work was supported by the Electric Power Research Institute under contract RP-965-3, and in part by the Office of Naval Research, contract NA-onr-6-75, NR 062-455, with the cooperation of the U. S. Department of Energy.

mation that preserves its discontinuous nature. In particular, a type of donor-acceptor flux approximation is used that uses information about F downstream as well as upstream of a flux boundary. In this way a crude interface shape is established that can be used to compute the flux of F.

III. THE SOLA-VOF CODE

The VOF method is applicable to Eulerian or the more general Arbitrary-Lagrangian-Eulerian (ALE) numerical formulations. We have verified the accuracy and versatility of the method by incorporating it into a two-dimensional, finite-difference, Eulerian scheme that uses a solution algorithm (SOLA) by Hirt et al. (1975) based on the well-known Marker-and-Cell (MAC) method (Harlow and Welch, 1965). The combined code, SOLA-VOF, (described by Hirt and Nichols, 1980 and Nichols et al., 1980), uses an Eulerian mesh of rectangular cells having variable sizes δx_i for the ith column and δy_j for the jth row. The fluid equations solved are the Navier-Stokes equations in either Cartesian or cylindrical coordinates. These equations are supplemented with either the continuity condition for incompressible fluids or the continuity condition for fluids with limited compressibility effects (e.g., acoustic waves).

The SOLA-VOF code also has a variety of additional options. Any combination of cells in the mesh can be defined as obstacle cells into which fluid cannot flow. Instead of one fluid with a free surface, two fluids can be defined with different density ratios separated by a free interface. Surface tension forces are optional at the fluid interface for both the one and two fluid cases.

The basic procedure for advancing a solution through one increment in time consists of three steps:

(1) Explicit approximations of the momentum equations are used to compute the first guess for new time-level velocities using the initial conditions or previous time-level values for all advective, pressure, and viscous accelerations.

(2) To satisfy the incompressible fluid continuity equation, pressures are iteratively adjusted in each cell and velocity changes induced by each pressure change are added to the velocities computed in step 1. An iteration is needed because the change in pressure needed in one cell to satisfy the continuity equation will upset the balance in the four adjacent cells. On the other hand, when the limited compressibility option is used the SOLA-VOF program automatically, and continuously, switches from an implicit to an explicit solution for pressures as the time step is reduced below the Courant stability limit. This feature permits more accurate results to be obtained with less computational work!

(3) Finally, the F function defining fluid regions must be updated to give the new fluid configuration.

Repetition of these steps will advance a solution through any desired time interval. At each step, of course, suitable boundary conditions must be imposed at

IV. SAMPLE PROBLEMS

We have chosen several calculational examples to illustrate the capabilities of the SOLA-VOF method.

A. Broken Dam Problem

A simply executed problem, for which experimental data is available, is the "broken dam" problem. An initially rectangular block of fluid, in hydrostatic equilibrium, collapses under the force of gravity and flows across a dry horizontal floor, as shown in Fig. 1. A comparison of the experimental data reported by Martin and Moyce (1952) with the calculated leading edge of the fluid as a function of time, Fig. 2, shows the greatest deviation is everywhere less than a calculational cell width.

B. Collapse of a Cylindrical Fluid Column

The collapse of a cylindrical column of fluid is similar to the "broken dam" problem, but uses the cylindrical coordinate system. Several additional features of the SOLA-VOF code are also illustrated by this calculation. Flow visualization is typically realized by plotting the velocity field with velocity vectors drawn from the center of each mesh cell containing fluid and by depicting the free surfaces with the volume fraction (F=1/2) contour, as seen in Fig. 1. However, marker particles can be used to follow the fluid flow as in Fig. 3. In addition, this calculation demonstrates the use of obstacle cells. The capability of the code to handle highly contorted fluid configurations is exceptionally well illustrated here.

C. A Reactor Safety Application

Many boiling water reactors use a large pool of water to condense steam should a major steam leak occur. In some designs, steam would be forced into the pool through vertical pipes extending several pipe diameters below the surface of the pool. Before steam enters the pool, however, air initially in the pipes must be pushed out. The ejection of this noncondensable air forms large bubbles in the pool and displaces the pool surface upward. Several small scale experimental programs have been conducted to understand the hydrodynamic forces generated during the process. We have numerically calculated these hydrodynamic forces and compared with laboratory test data (Nichols and Hirt, 1980).

To model these experiments it was necessary to supplement the SOLA-VOF code with calculations for the gas pressure in the pipe and for the pressure in the closed space above the pool surface. These pressures are then used as free surface boundary pressures. A sequence of calculated results illustrating the fluid dynamics associated with the air-clearing process are contained in Fig. 4.

D. Instability of a Liquid Column

For some applications, such as the breakup of a thin liquid jet, surface tension forces must be considered. A classic problem of this type concerns the instability of a cylindrical column of fluid. When its free surface is perturbed by radial displacements, an exponential growth in perturbation amplitude may result that eventually causes the cylinder to break up into a series of discrete drops. Figure 5 illustrates a SOLA-VOF calculation of this type of surface tension driven instability. The cylinder is initially perturbed with an axisymmetric displacement of its free surface that is sinusoidal in the axial direction. Two axial wave lengths, λ, are followed, with $\lambda = 4.598$ D, where D is the diameter of the undisturbed liquid column. The initial perturbation amplitude is 0.001 D. A comparison of the computed amplitude growth with linear theory shows the interesting result that the linear theory is valid for large amplitude displacements. Nevertheless, nonlinear effects are important, for they are the cause of the small satellite drops that develop between the large drops.

Many additional calculations have been performed that validate various capabilities of the SOLA-VOF code not mentioned in the above examples. Included in these are a study of bubble growth and collapse, which made use of the limited compressibility feature, and the passage of an immiscible liquid drop through a constriction in a tube, in which the two-fluid and surface tension options were utilized together.

SUMMARY

We present the volume of fluid (VOF) technique as a simple and efficient means for numerically treating free boundaries embedded in a calculational mesh of Eulerian or Arbitrary-Lagrangian-Eulerian cells. It is particularly useful because it uses a minimum of stored information, treats intersecting free boundaries automatically, and can be readily extended to three-dimensional calculations.

REFERENCES

Harlow, F. H. and Welch, J. E., 1965, "Numerical Calculation of Time-Dependent Viscous Incompressible Flow," Phys. Fluids 8, 2182; Welch, J. E., Harlow, F. H., Shannon, J. P., and Daly, B. J., 1966, "The MAC Method: A Computing Technique for Solving Viscous, Incompressible, Transient Fluid Flow Problems Involving Free Surfaces," Los Alamos Scientific Laboratory report LA-3425.

Hirt, C. W., Nichols, B. D., and Romero, N. C., 1975, "SOLA - A Numerical Solution Algorithm for Transient Fluid Flows," Los Alamos Scientific Laboratory report LA-5852.

Hirt, C. W. and Nichols, B. D., 1980, "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries." J. Comp. Phys., accepted for publication.

Martin, J. C. and Moyce, W. J., 1952, Phil. Trans. Roy. Soc., London, A244, 312.

Nichols, B. D. and Hirt, C. W., 1980, "Numerical Simulation of BWR Vent-Clearing Hydrodynamics," Nuc. Sci. Engr. 73, 196.

Nichols, B. D., Hirt, C. W., and Hotchkiss, R. S., 1980, "SOLA-VOF: A Solution Algorithm for Transient Fluid Flow with Multiple Free Boundaries," Los Alamos Scientific Laboratory report LA-8355.

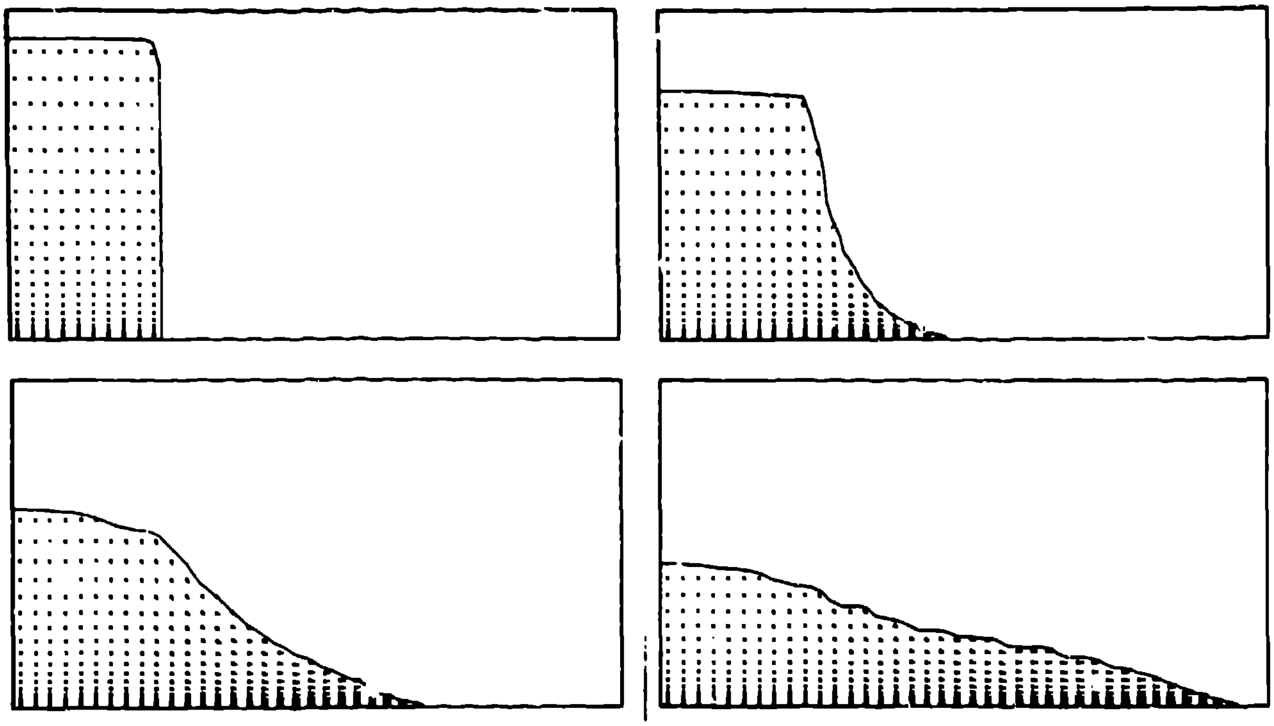

Fig. 1. Velocity vectors and free surface configurations computed for the broken dam problem at times 0.0, 0.9, 1.4, and 2.0.

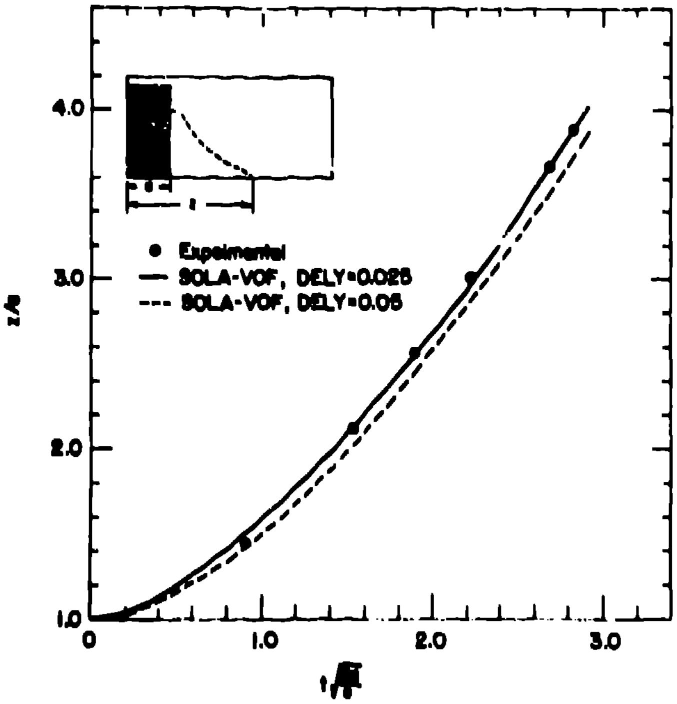

Fig. 2. Comparison of calculated results with experimental data for the broken dam problem.

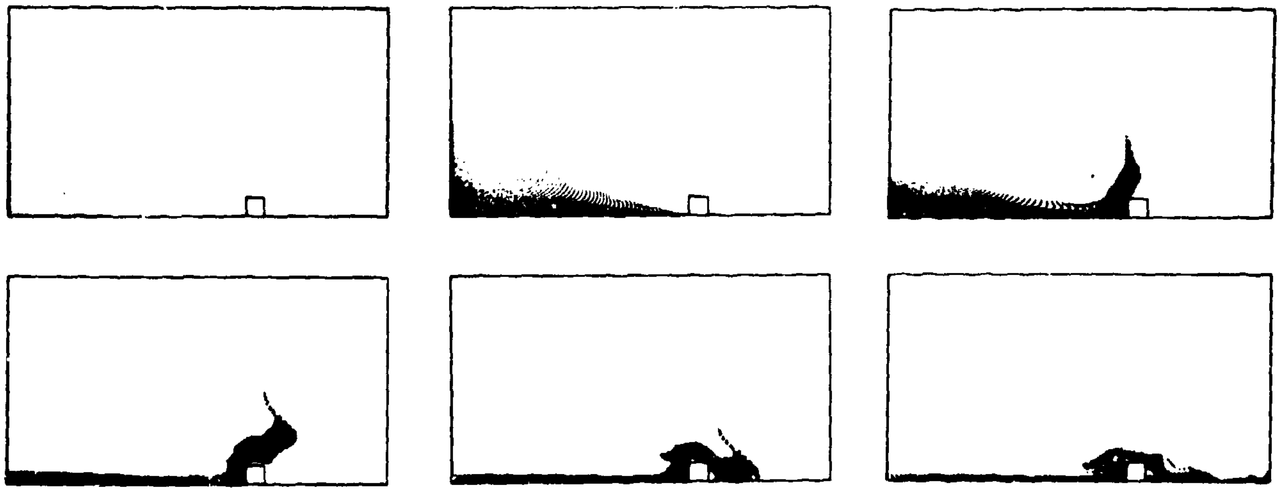

Fig. 3. Calculational results showing the collapse of a cylindrical column of fluid surrounded by a low retaining wall. Times shown are 0.0, 1.6, 2.5, 3.6, 4.2, and 4.6

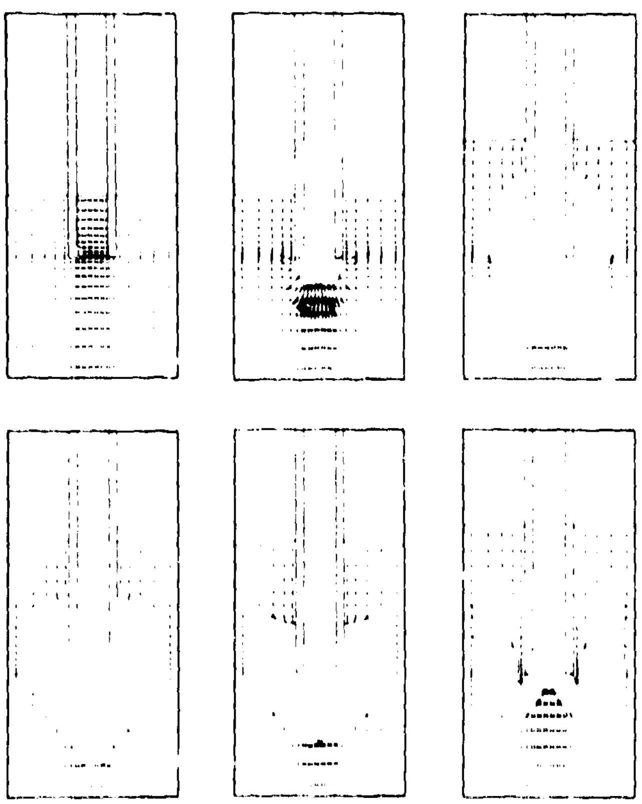

Fig. 4. Velocity vectors and free surface configurations computed when air is forced through submerged vent pipe.

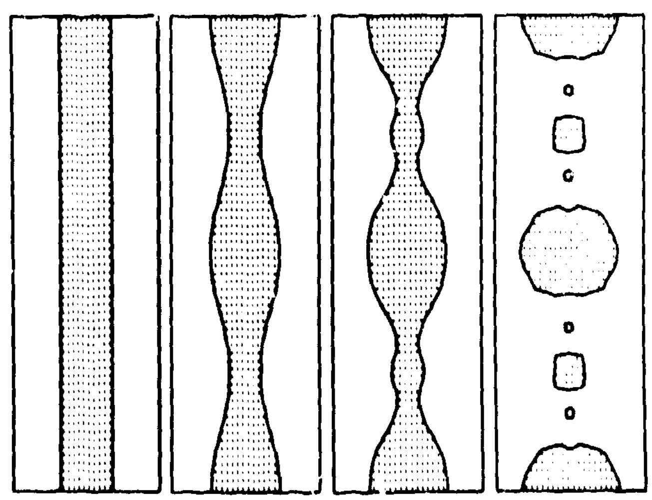

Fig. 5. Fluid configurations calculated for a surface tension driven instability of a cylindrical column of fluid. Times are approximately 0.0, 6.03, 6.49, and 7.0