COUPLED FLUID-STRUCTURE METHOD FOR PRESSURE SUPPRESSION ANALYSIS

W. H. McMASTER, D. M. NORRIS, Jr., G. L. GOUDREAU

University of California, Lawrence Livermore Laboratory,
P.O. Box 808, Livermore, California 94550, U.S.A.

ABSTRACT

We have coupled an incompressible Eulerian hydrodynamic algorithm to a Lagrangian finite-element shell algorithm for the analysis of pressure suppression in boiling water reactors. The computer program calculates loads and structural response from air and steam blowdown and the oscillating condensation of steam bubbles in a water pool. The fluid, structure, and coupling algorithms have been verified by the calculation of solved problems from the literature and from air and steam blowdown experiments.

The foundation of the program is the semi-implicit, two-dimensional SOLA algorithm. The shell structure algorithm uses conventional thin-shell theory with transverse shear. The finite-element spatial discretization employs piecewise-linear interpolation functions and one-point quadrature applied to conical frustra. We use the Newmark implicit time-integration method implemented as a one-step module. The algorithms are strongly coupled in the iteration loop using the iterated pressure in the fluid to drive the structure.

The coupling algorithm requires normal velocity compatibility at the fluid-structure interface and incompressibility of the computational Eulerian zone overlaid by the structure. This is accomplished by iterating on the pressure field which is applied to the structure during each iteration until both conditions are satisfied.

---

*Work performed under the auspices of the U.S. Department of Energy by the Lawrence Livermore Laboratory under contract number W-7405-ENG-48.

1. Introduction

This report describes a method for calculating the loads and the structural response resulting from blowdown suppression in boiling-water reactor containment vessels. The method uses a computer code that couples an incompressible Eulerian fluid algorithm with a Lagrangian finite element shell algorithm. The fluid physics is based on the SOLA algorithm [1]. The Navier-Stokes equations provide a trial velocity field that is corrected iteratively, at each time step, to satisfy the incompressibility condition and the fluid-structure interface velocity boundary conditions. The air blowdown loads are obtained by pressure prescriptions applied in the downcomer pipes. The oscillatory steam loads are described by simple condensation models and by upper-bound pressure excursions in the steam bubble taken from experiments.

These fluid and fluid-structure-interaction algorithms are verified by comparison with solved problems from the classical literature and from recent air and steam blowdown experiments.

The calculations are done by PELE-IC [2], which is a two-dimensional semi-implicit Eulerian hydrodynamics code written in both plane and cylindrical coordinates. The coupling algorithm developed here is general enough to handle a wide variety of structural shapes.

The structural motion is computed by a finite element shell code [3] from the pressure applied at the fluid-structure interface. The fluid algorithm then uses the position and velocity of the structure as boundary conditions. The structural response and the fluid motion are corrected iteratively until the boundary conditions and the incompressibility conditions are both satisfied.

2. The SOLA Algorithm

The differential equations to be solved are the Navier-Stokes equations for the conservation of momentum and the continuity equation for the conservation of mass. The incompressible assumption for the conservation of mass requires that the divergence of the velocity field be zero. These equations require pressure and velocity boundary conditions at free surfaces and structural interfaces. Due to the velocity coupling technique used, we are not required to set explicit boundary conditions on pressure at structural interfaces. The capability of the code to solve a wide variety of structural problems depends on the proper specification of these boundary conditions for use by the SOLA algorithm.

The solution strategy is to first solve the Navier-Stokes equations explicitly using values from the previous time step with all boundary conditions applied. This yields trial values for the flow field $(\tilde{u},\tilde{v})$, where the tilde denotes that these velocities are tentative. Since these velocities do not necessarily satisfy the continuity equation and the fluid-structure interface boundary conditions, we iterate on the pressure field until all conditions are satisfied. During the iteration, the boundary conditions imposed by free surfaces and structures are applied. When all conditions are satisfied, the new values of the field variables (p, u, v) at the new time level $t + \delta t$ are then used to locate all free surfaces and update the boundary conditions.

3. The Free Surface Algorithm

The amount of liquid in a cell is tracked by the use of a void fraction. Surface location and orientation are determined by use of the free-surface algorithm which performs the following four functions:

3.1 Cell Centered Pressure

The cell-centered pressure used in the equations of motion is determined from a linear interpolation or extrapolation based on the prescribed surface pressure and the current fluid pressure in the adjacent full liquid cell. If there is more than one adjacent full cell, then the average of all the interpolated values is used. This surface boundary pressure can either be a constant or a prescribed function of time.

3.2 Surface Orientation

The surface orientation is specified by its intercepts on two sides of the cell. These points are determined by the void fractions of the cell and its four nearest neighbors. Within the cell, the interface is considered to be a straight line segment. The cell side intercept of a free surface is determined from the relative void fractions in each of the two mixed cells with a common side. Thus the surface is tracked by its intersection of grid lines.

3.3 Fluid Advection

The fluid is advected across all cell boundaries using the donor cell method where the amount of liquid advected is determined from the contents of the upstream cell. The amount of donor cell liquid advected is determined from the fluid velocity and the orientation of the fluid-void interface. This method provides an accurate tracking of the free surface without the requirement of using marker particles to define the surface.

3.4 Cell Incompressibility

The incompressibility of the cell is maintained by setting the void side velocity such that $\nabla \cdot v = 0$. This velocity is not used in the calculation but is necessary to maintain continuity in the flow field when the surface motion causes the interface to cross a grid line.

4. The Finite Element Shell Module

The structural module is designed for a shell-of-revolution or cylinder which is coupled to the fluid through the fluid-structure-interaction algorithm. The module implicitly advances the dynamic shell response one time step using the old interface geometry and the current iterated nodal velocities and pressure distribution. It returns the new interface positions and velocities for use by the fluid algorithm. The module is applied during each step of the iteration and results in a strong coupling to the fluid field.

The module uses the thin shell equations originally derived by Love but modified to include deformation due to transverse shear by Kraus [4]. These equations are discretized spatially by use of the finite element method and temporally by the Newmark implicit time integration scheme. The element formulation is similar to that described by Hughes and Taylor [5] for plates and extended to a shell by Goudreau [3] and Zienkiewicz et al. [6].

The element is a two-node, conical frustrum with three degrees-of-freedom per node. Element shape functions are piecewise linear for displacements and rotations. The shear "locking" associated with the thin shell limit is removed by the use of one-point quadrature. Large displacements are accounted for by updating the initial coordinates and reformulating the stiffness matrix every time step. This gives an incremental small strain formulation that, although not exact, provides excellent accuracy for displacements up to three shell thicknesses.

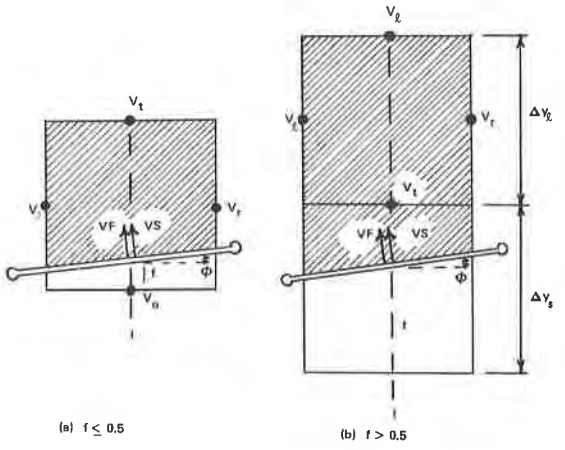
5. The Fluid-Structure-Interaction Algorithm

This algorithm couples the fluid motion to the structure's motion by requiring that the normal velocities be equal at the interface and that the computational cell remain divergentless. This is accomplished in the iteration process by the adjustment of the pressure field.

The algorithm applies equally well to both rigid and moving surfaces. For rigid surfaces, the normal velocity is set to zero. For moving surfaces, the finite element module uses the pressure field supplied by the fluid and provides the fluid code with the resultant position and velocity of the interface.

The velocities are coupled at the intersection of an I or J line with the structure where an I line is defined as a vertical line through the cell center and a J line is a horizontal line through the cell center. The choice of I or J line coupling is determined from the angle the structure makes at the intersection. For angles of 45 degrees or less we use I line coupling and J line coupling for larger angles.

The normal fluid velocity is found by interpolation between all four of the cell side velocities and the structure velocity is found by interpolation between nodal values. We then use the structure's normal velocity to set the cell side velocity which lies outside the structure. This then causes the cell not to satisfy the divergence criteria and the pressure must then be adjusted using the SOLA algorithm. This change in the pressure field causes a different structural response; therefore, the iteration proceeds until both conditions are satisfied. Within a single iteration, all Eulerian zones are adjusted one by one, and then all the shell nodes are simultaneously adjusted by the implicit step.



ORIENTATION FOR I LINE COUPLING

The fluid velocities are specified at cell sides and the structure velocities are specified at nodes. The normal velocity of the structure at the intersection is given by

$$VS = \cos\phi\ (-U_s \tan\phi + V_s)$$

while that of the fluid is

$$VF = \cos\phi\ (-\bar{U}\ \tan\phi + V_i)$$

where ϕ is the angular orientation of the structure, $\bar{U} = 0.5\ (U_r + U_\ell)$ is the interpolated x-velocity along the I line, and V_i is the interpolated vertical fluid velocity at the I line intersection. These two velocities must be equal which gives us the basic coupling equation

$$V_i = V_s + \tan \quad (\bar{U} - U_s) \ .$$

Using this equation we can determine the cell velocity outside the fluid as

$$V_o = V_t + (1-f)\ (V_s - V_t) + \tan \quad (\bar{U} - U_s) \ .$$

Note that as f approaches 1, V_o becomes almost equal to V_t and the coupling to the structure becomes very weak. To avoid this condition we blend cells whenever f is greater than 0.5.

In this case the velocity coupled to the finite element is V_t instead of V_o. We find the coupled cell side velocity from an interpolation between V_t and V_ℓ with the result

$$V_t = V_\ell + \frac{\Delta y_\ell}{\Delta y_\ell + (1-f)\Delta y_s} \left[(V_s - V_\ell) + \tan\phi\ (\bar{U} - U_s)\right]$$

and we note that as f approaches 1 then V_t approaches V_s which gives us the strong coupling desired.

By performing this blending we avoid discontinuities as the structure crosses a grid line as well as maintaining a strong coupling regardless of the structure's location in the fluid cell.

6. Program Verification

The objective of the program verification was to check the PELE-IC code results against analytical solutions. Three types of verification calculations were chosen: A draining tank, an expanding spherical bubble, and a vibrating submerged plate. The problems were designed to test different boundary conditions needed for applications. These calculations demonstrated remarkable ability to reproduce the solutions from the literature.

The problem of a tank draining through an outlet tube at the tank bottom was chosen since the pipe flow pattern is similar to that of a downcomer pipe during vent clearing following a loss-of-coolant accident. These calculations require rigid internal structures such as pipes and straight walls. The calculation checked mass conservation, general flow behavior, and the free-surface algorithm.

The expansion of a spherical bubble was calculated to test the logic of the air-water interface used to describe bubble expansion. The expanding (and contracting) bubble is similar to the bubble phenomenon at the end of a downcomer pipe accompanying blowdown.

To test the fluid-structure coupling algorithm, a series of submerged vibrating plate calculations were done. We considered two cases: The fluid is unconfined, and the fluid is confined in a cylindrical tank with rigid side walls. The unconfined fluid solutions showed the correct change in vibrational frequency as a function of water depth thus confirming the strong coupling between the fluid and structure.

To further assess the capability of the PELE-IC code we have calculated the results of small scale experiments by Anderson, et al. [7] and Chan et al. [8]. These experiments simulate wetwell dynamics by blowing air into a vertical pipe submerged in a rigid vertical cylindrical tank.

References

[1] Hirt, C. W., Nickols, B. D., Romero, N. C., "SOLA - A Numerical Solution Algorithm for Transient Fluid Flow," Los Alamos Scientific Laboratory Report LA-5552 (1975).

[2] McMaster, W. H., Norris, Jr., D. M., Goudreau, G. L., Quiñones, D. F., Gong E. Y., Moran, B. and Macken, N. A., "Coupled Fluid-Structure Method for Pressure Suppression Analysis," Lawrence Livermore Laboratory Report UCRL-52620 (NUREG/CR-0607) (1979).

[3] Goudreau, G. L., "A Computer Module for One Step Dynamic Response of an Axisymmetric or Plane Linear Elastic Thin Shell," Lawrence Livermore Laboratory Report UCID-17730 (1978).

[4] Kraus, H., Thin Elastic Shells, John Wiley & Sons, Inc., New York (1967).

[5] Hughes, T. J. R., Taylor, R. L., "A New Finite Element for Plate Bending Analysis," SMiRT Proc. M 2/1 (1977).

[6] Zienkiewicz, O. C., et al., "A Simple and Efficient Element for Axisymmetric Shells," Int. J. Numerical Methods in Eng., 11, (10), 1545 (1977).

[7] Anderson, W. G., Huber, P. W., Sonin, A. A., "Small Scale Modeling of Hydrodynamic Forces in Pressure Suppression Systems," Massachusetts Institute of Technology, Final Report, NUREG/CR-0003, prepared for the U.S. Nuclear Regulatory Commission, Dec. (1977)

[8] Chan, C. K., et al., "Suppression Pool Dynamics and Annual Report," University of California, Los Angeles, Report NUREG-0264-3 prepared for the U.S. Nuclear Regulatory Commission, June (1977).