Improved Fluid-Structure Coupling

W.H. McMaster, E.Y. Gong, C.S. Landram
University of California, Lawrence Livermore National Laboratory,
P.O. Box 808, L-122, Livermore, California 94550, U.S.A.

SUMMARY

In our computer code PELE-IC, discussed at SMiRT-5, we had coupled an incompressible Eulerian hydrodynamic algorithm to a Lagrangian finite element shell algorithm for the analysis of pressure suppression in boiling water reactors. This effort also required the development of a free surface algorithm capable of handling expanding gas bubbles. These algorithms have been improved to strengthen the coupling and to add the capability for following the more complex free surfaces resulting from steam condensation. These improvements have also permitted more economical 2D calculations and have made it feasible to develop a 3D version. A compressible option using the acoustic approximation has also been added, furthering the usefulness of the code.

The coupling improvements were made in three areas which are identified as (1) preferential coupling, (2) merged cell coupling, and (3) free surface-structure coupling, and are described in this paper.

These algorithms have been additionally implemented in a three dimensional version of the code called PELE3D. This version has a free surface capability to follow expanding and contracting bubbles and is coupled to a curved rigid surface.

Work was also initiated in developing a steam condensation model for the study of the effects of chugging on containment structures. Our investigations have shown that the approach of specifying an equivalent load function is invalid since there is an effective coupling between the collapsing steam bubble and the flexible containment structures.

The development of these improved algorithms has resulted in a more versatile fluid-structure interaction code and has allowed us to develop a 3D capability to analyze the nonsymmetric aspects prevalent in existing reactor pressure suppression systems.

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

1. INTRODUCTION

This report describes the improvements made to the free surface and fluid-structure coupling algorithms since the original report at SMiRT-5 [1]. These algorithms are embedded in a computer code called PELE-IC [2,3] that couples a two-dimensional semi-implicit Eulerian fluid algorithm to a finite element shell algorithm [4]. This coupled code was developed to analyze the loads and structural response from blowdown and steam condensation chugging in reactor containment vessels.

In this code, the fluid physics equations are solved by the use of the SOLA finite difference algorithm [5], which uses a Newton-Rhapson iteration technique to solve the Navier-Stokes and continuity equations. The iteration is carried out on the pressure field until the fluid-structure interface velocity compatibility, and free-surface boundary conditions are satisfied. These fluid and fluid-structure algorithms have been extensively verified through calculations of known solutions from the classical literature, and by comparison to air and steam-blowdown experiments [3,6].

The structural motion is computed by a finite element code from the fluid pressure applied at the fluid-structure interface. The finite element shell structure algorithm uses conventional thin-shell theory which includes deformation due to transverse shear [7]. The spacial 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. Large deflections up to three shell thicknesses have been accurately calculated by updating the stiffness matrix each time step.

Compressibility effects can be modeled by use of the acoustic approximation, $\partial\rho/\partial t = 1/c^2 \; \partial p/\partial t$, in the continuity equation. Using this approach we have successfully investigated the effects of the lowered sound speed in water containing entrapped steam bubbles. A side effect is that fewer iterations are required to achieve convergence.

In November 1979, the PELE-IC code was released to the National Energy Software Center at Brookhaven National Laboratory, Upton, New York and is now available for public use.

2. FLUID-STRUCTURE COUPLING

2.1 Basic Coupling Algorithm

This algorithm couples the fluid's motion to the structure's motion within the SOLA iteration loop. Within a single iteration, all Eulerian fluid cells are adjusted one by one, using the latest values available, and then all Lagrangian shell nodes are simultaneously adjusted by the implicit time step solution. The coupling provides the boundary conditions to each algorithm: (1) the fluid pressure is the applied boundary condition to the finite element code, and (2) the structure's position and velocity provide the boundary condition to the fluid algorithm. In the fluid cell, the pressure is a cell centered quantity and the velocity components are maintained on the cell sides. In the structural elements, the velocities are nodal values and the pressure is distributed along the element.

The pressure applied to an element is determined by an interpolation along each intersecting I or J line to the neighboring full fluid cell. These interpolated values are weighted by the liquid content of the cell so that the proper pressure is applied when a free surface is in the same cell. In the algorithm, we require the normal velocities of the structure and fluid to match at the coupling points. These velocities are found by an interpolation dependent upon the structure's location and orientation. The structure is always coupled to the nearest fluid velocity along the coupling line. The tangential velocities are free to assume their respective values and are uncoupled. The fluid cell containing the structure has the additional applied condition that its velocity field remain divergence free to satisfy the mass continuity equation. These conditions are fulfilled using a Newton-Rhapson iteration to adjust the fluid pressure field until all velocities conform to the boundary restraints.

During the iteration step, i, the cell incremental pressure change is calculated from

$$\delta p_i = - \frac{(1 + \phi)\, D_{i-1}}{\delta D/\delta p}$$

where

$$D_{i-1} = \nabla \cdot u_i + \frac{1}{c^2} \frac{\partial p_i}{\partial t} \quad \text{and} \quad \frac{\partial D}{\partial p} = 2\delta t \left[\frac{F_x}{\delta x^2} + \frac{F_y}{\delta y^2} + \frac{1}{2c^2 \delta t^2} \right]$$

with ϕ, F_x, and F_y being functions dependent upon the structure. In general, ϕ varies between 0.1 and 0.9 while the structure functions F have the value of one if no structure exists. These structural dependent functions are varied by the code to speed convergence.

Since the coupling depends upon the structure's time advanced position and velocity, a strong coupling results.

2.2 Preferential Coupling

A single Lagrangian structural element may encompass several Eulerian fluid cells, and thus the convergence of the coupling algorithm is sensitive to the manner in which the coupling is calculated. In the preferential coupling technique, we now use both I- and J-line coupling for a single structural element regardless of the element orientation. This coupling between the fluid and structure is done at the element intercept with the I and J lines which define the centroid of the Eulerian fluid cells. If more than one intercept occurs in a fluid cell, then the coupling line is chosen dependent upon the angle of intercept as before. These intercepts are then ordered sequentially for each element according to the element's slope. For example, if the angle is less than forty-five degrees, then all I intercepts for the elements are processed first in the fluid algorithm followed by the J intercepts. In this manner, no intercepts are neglected. This refinement results in more coupled cells and strengthens the coupling. This new procedure becomes important when it is desirable to use non-square Eulerian cells for more economical calculations. The rectangular zoning introduces the possibility of having both I and J intercepts for a single structural element. The preferential ordering provides a more rapid convergence resulting in fewer iterations.

2.3 Merged Cell Coupling

The merged cell coupling algorithm has been improved by coupling the pressures and making the merged cell velocities dependent upon the subsequent merged cell pressure changes. Previously these velocities were assumed to be the same as those in the master cell. For I-line coupling, the pressures are gravity coupled in the vertical direction. These changes result in the merged cell being less dependent upon the neighbor to which it is merged and gives a smoother transition when a structural element crosses a grid line.

2.4 Free Surface-Structure Coupling

The free surface-structure coupling usually occurs where the coupling is along a J-line. There are two boundary conditions to be satisfied: the cell pressure is determined by interpolation from the surface boundary pressure and the fluid velocities must be coupled to the structure. Of the the two velocities outside the fluid, we choose the J-line velocity component to be coupled to the structure, and the I-line velocity is set to maintain divergence free flow. In this manner a smooth transition occurs when either the structure or free surface crosses a grid line. The pressure applied to the structural element is weighted between the fluid and gas pressures dependent upon the surface location.

3. FREE SURFACE ALGORITHM

Accurate free surface tracking is necessary to allow the application of velocity and pressure boundary conditions at all fluid interfaces with gases or structures. We track the free surface by a combination of void fraction and surface orientation in each computational cell. The void fraction provides for the conservation of mass, and the surface orientation allows us to apply the proper pressure and velocity boundary conditions and to follow the fluid advection from cell to cell. The advection routine checks the results to assure that a continuous surface is maintained. The algorithm is general enough to follow several disconnected surfaces. For example, we can follow two separate bubbles exiting from two submerged downcomer pipes or vents and also track the surge of the surface above. These bubbles can merge if they collide and break through the top water surface. These caspabilities are possible since we are following the fluid regions and then constructing the surface. Algorithms designed to follow surfaces as such generally have difficulty with these complex surfaces.

The surface orientation is specified by its intercepts on two sides of the cell. These intercepts are found by fitting a surface to its fluid fraction and those of neighboring cells. Within a cell, the interface is considered to be a straight line segment connecting the two intercepts. In this manner, each surface is tracked by its intersection of grid lines and the properties of surface location and curvature are easily computed.

The amount of fluid advected across cell boundaries is determined explicitly each time step using the surface orientation to calculate the transfer. Since we know the surface location in each cell, it is not necessary to use special donor-acceptor advection algorithms as required by other methods. The PELE algorithm uses donor cell advection with the flux calculated from the surface orientation in the donor cell using the updated velocities. When the new fluid content of all cells has been updated, we construct the new surface orientation. This method provides an accurate tracking of the free surface by grid line intercepts and allows us to follow the motion of complex surfaces. This technique has been used to investigate expanding and collapsing steam bubbles and their detachment from downcomer pipes.

4. CHUGGING SIMULATIONS

Previous investigations have revealed that the driving condensation event, augmented by downcomer vent acoustics, provides a highly coupled resonating forcing function for the fluid-structure system. Use of experimental pressure histories representing the forcing function has been shown to be invalid [3].

Recently it has been strongly suggested that available void volume data be used to derive a vent exit forcing function in representation of the condensation events occurring at the exit of the vent pipe. The volume of the void formed at the exit of the downcomer during chugging depends upon several parameters. Among these are vent pipe enthalpy rate, condensation rate, thermo-dynamic vapor state points, wetwell geometry and state, and wetwell flow field. Since these parameters governing the void volume history are system dependent, the void volume history in one system cannot be related to that in another.

A steam condensation module has been incorporated into PELE-IC for the study of the chugging phenomena. The model used is based on the work of Bornhorst [8] and Theofanous [9] and includes the effects of water vapor condensation and thermal conduction in the surrounding water. A set of integro-differential equations describe vapor energy conservation, vapor mass conservation, momentum and mass conservation in the surrounding water, and the water heat conduction.

Additionally, our shakedown calculations have revealed that total void volume is not a good indicator of void pressure history. That is, adiabatic and diabatic total void volume change little (all other conditions fixed), whereas the corresponding void pressure histories dramatically differ. Consequently, the use of a void volume measurement to obtain a void exit pressure history does not appear to be very promising. In accordance with experimental data, the mechanism for the onset of void collapse is calculated to be separation of the diabatic void from the end of the downcomer.

There have been sufficient experimental data to indicate that (1) the collapse process may be considered to occur on time scales much smaller than those associated with the period of the predominate structural vibration, and (2) the rarefaction forcing pulse (chugging forcing function) originates following void collapse. Within the framework of such evidence, the condensation model presently developed for the code could be used to determine steam bubble histories from formation to detachment from the vent followed by sudden collapse and subsequent water reentry into the downcomer pipe. We feel that the significant features of vapor bubble collapse can be calculated with reasonable computer costs and should be pursued.

5. THREE-DIMENSIONAL APPLICATIONS

A three-dimensional code, PELE3D, has been written using the techniques developed for the earlier two-dimensional code. This code uses both plane and curvilinear coordinates with either regular or proportional zoning. The free surface and bubble dynamics algorithms have been adapted to work in three dimensions. The current version is coupled to a curved rigid structure in the vertical plane.

Since the initial purpose of this code was to analyze the nonsymmetric aspects of reactor pressure suppression systems, we have developed a pipe algorithm which allows up to eight downcomer pipes. These pipes are modeled as rigid vertical structures with the requirement that there be no normal fluid velocity at their surface. Upon vent clearing the free surface algorithm will maintain separate bubbles until they merge.

The code is operable on the CRAY-1 computer and runs relatively fast using a small number of iterations per cycle. Because of the added degrees of freedom we have found that the same calculation in three dimensions requires fewer iterations than a two-dimensional calculation. Test runs have used an average of 30 microseconds for each cell-cycle-iteration and the rigid wall test problems averaged slightly over two iterations per cycle.

We have made three scoping calculations using this new code, (1) the LLNL 1/5 scale Mark I (BWR) PSE experiment, (2) the German GKSS BWR pressure suppression facility, and (3) the Japanese JAERI experimental wetwell. The latter two of these cannot be modeled with a two-dimensional code.

The three-dimensional effects of vent clearing have been demonstrated with the accurate calculation of the times for vent clearing and peak downloads when compared with the experimental data of the LLNL 1/5 scale PSE experiment. Two-dimensional calculations result in longer times for these effects thus showing the need for a three-dimensional capability.

A typical condensation chugging event in the GKSS system lasts approximately two seconds with about five seconds between events giving a seven second cycle. Our scoping calculation with a coarse grid required thirty seconds of CRAY-1 CPU time per one second of real time using a one millisecond time step. We estimate that a full cycle calculation with a finer grid would require approximately forty-five minutes of CPU time.

In the JAERI system, there is a plane of symmetry bisecting the wetwell wedge angle and thus this system can be simulated using a three-vent pipe model.

REFERENCES

/ 1 / McMASTER, W.H., NORRIS, D.M., GOUDREAU, G.L., "Coupled Fluid-Structure Method for Pressure Suppression Analysis," Transactions of the 5th International Conference on Structural Mechanics in Reactor Technology, Vol. B, paper b 1/5 (1979).

/ 2 / McMASTER, W.H., GONG, E.Y., "User's Manual for PELE-IC: A Computer Code for Eulerian Hydrodynamics," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52609 (1979).

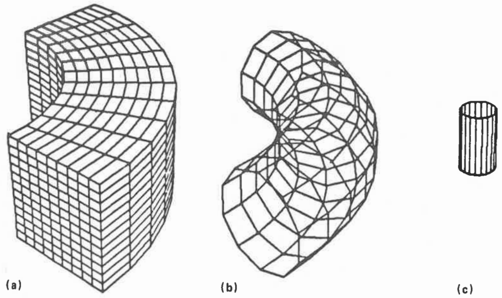
/ 3 / McMASTER, W.H., et al., "Application of the Coupled Fluid-Structure Code PELE-IC to Pressure-Suppression Analysis," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52733 (NUREG/CR-1179) (1979).

/ 4 / GOUDREAU, G.L., "A Computer Module for One Step Dynamic Response of an Axisymmetric or Plane Linear Elastic Thin Shell," University of California, Lawrence Livermore National Laboratory, Rept. UCID-17730 (1978).

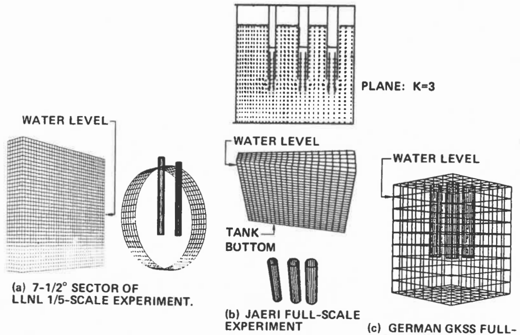
/ 5 / HIRT, C.W., NICKOLS, B.D., ROMERO, N.C., "SOLA - A Numerical Solution Algorithm for transient Fluid Flow," Los Alamos Scientific Laboratory Rept. LA-5552 (1975).

/ 6 / McMASTER, W.H., et al., "Coupled Fluid-Structure Method for Pressure Suppression Analysis," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52620 (NUREG/CR-0607) (1979).

/ 7 / KRAUS, H., Thin Shells, John Wiley & Sons, Inc., New York (1967).



(A) COMPONENTS OF THE ZONING FOR THE 90° SECTOR OF THE LLNL 1/5-SCALE PSE EXPERIMENT.



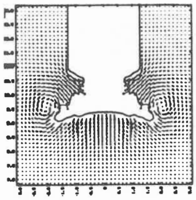
(a) 7-1/2° SECTOR OF LLNL 1/5-SCALE EXPERIMENT.

(b) JAERI FULL-SCALE EXPERIMENT

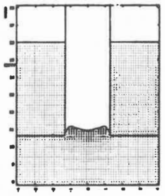
(c) GERMAN GKSS FULL-SCALE EXPERIMENT.

(B) ZONING FOR THE TEST PROBLEMS.

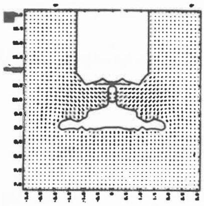
VENT PIPE BUBBLE CONDENSATION



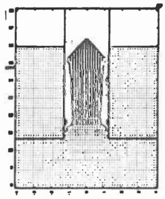
(1) Pinch Off



(2) Liquid Reentry



(3) Void Separation



(4) Reentry Jet