FSI-88-TN10

NOTES ON F-ADVECTION AND F-PACKING

C.W. Hirt and R.P. Harper
Flow Science, Inc.
April 1988

BACKGROUND

In codes using the VOF technique to track free surfaces it has sometimes been observed that small fluid-fraction values will propagate and disperse through large regions of a computational mesh. When this so-called "foam", appears, it can ruin an otherwise good calculation. Usually the appearance of foam is associated with situations involving large fluid distortions and relatively coarse mesh resolution resolving the distortions.

Another problem sometimes encountered in VOF calculations is their failure to close up small void regions within the interior of fluids (i.e., to attain fluid fraction values of unity in mesh cells treated as full of fluid). This situation usually occurs when fluid strikes a solid boundary or when two fluid masses collide. Some people refer to this as the "F-packing" problem.

The foam and F-packing symptoms are not unrelated because lack of packing near a free surface may encourage the production of isolated bits of fluid that later degenerate into foam.

Recognizing these difficulties, Flow Science has undertaken a lengthy development effort to improve the methodology. As a result of this effort several major changes have been incorporated into the 88-Version of the FLOW-3D program (i.e., FLOW-3D/88). A summary of these changes is given here.

VOF ENHANCEMENTS

To address both the foam and F-packing problems it has been necessary to modify several subroutines in the FLOW-3D program. Specifically, we have modified the following functions:

Free surface boundary conditions,

Selection of neighbors to surface cells (NF), Pressure iteration routines,

Fluid fraction advection routine, and

Miscellaneous subroutines used in above routines.

Neighbor Selection

The selection of NF indicating the neighbor to a surface cell has been simplified. In previous code versions this quantity was selected by estimating surface normals with respect to each coordinate direction and then choosing the one that seemed to most nearly represent the correct normal. In the new procedure we have eliminated all the complicated normal calculations (except when needed for the surface tension model) and simply select the neighbor cell as that cell opposite an empty cell and having the maximum F value of all such cells. This means the NF flag always points away from an empty cell. It also eliminates spurious NF values that sometimes occurred when surface normals could not be accurately calculated (e.g., in poorly resolved regions or regions of foam). A consequence of this new procedure is that it helps reduce the dispersion of foam when used in conjunction with the new F-advection routine.

F-Advection

The F-advection routine used in FLOW-3D/87 contained several refinements over the original VOF scheme. These refinements were primarily associated with special situations, although there was one particular modification introduced to limit advection into empty cells. This latter modification was meant to reduce foaming, which it did to some extent, but recent experience has shown that it was not enough. In FLOW-3D/88 we have modified and expanded the special tests to make them stronger and more symmetric. We have also added a test to eliminate F-fluxes into empty cells under special circumstances. These tests significantly reduce foam dispersion. It should be noted, however, that the proper operation of the newly-added test relies on the new NF selection method.

Tests have been added to increase F fluxes into full cells and to reduce F fluxes away from full cells. One consequence of these tests is that in an interior region, away from all surfaces, the F value in a cell can only change by having a nonzero velocity divergence. Another consequence is that surfaces remain better defined and there is less tendency for bits of F to be left in the wake of a receding surface.

Another modification that has been added is one to throw away fluid in isolated mesh cells, that is, fluid in cells that have all empty neighbors. The reasoning behind this is that the code cannot accurately resolve these isolated drops and they almost always lead to a dispersed foam. Thus, eliminating such drops reduces foam generation.

F-Packing

As noted above, the only way F values in interior cells can change is for the velocity divergence to be nonzero. Therefore, in interior cells with F values less than one, the only way to raise F to a value of one (i.e., to pack the cell) is to generate a negative cell divergence corresponding to a net flow of fluid into the cell. A mechanism to do this can be constructed by treating an F value less than one as representing the presence of vapor in the cell and then condensing the vapor to locally reduce the pressure and force packing. To make this process work we must treat the cell as though it has some limited compressibility, but with a net pressure change that insures a negative velocity divergence.

It is relatively easy to modify the pressure iteration function to produce a negative divergence, but it is not easy to do this in a way that keeps pressure changes within limits that give reasonable force calculations. Our approach has been to modify the pressure time derivative term used in the limited compressibility model in two ways. First, we calculate the coefficient of the pressure term, which is a reciprocal bulk compressibility, in a way that keeps the flow near the incompressible limit. The coefficient is based on the cell BETA value (the SOR relaxation coefficient) and the value of F in the cell. This gives a scaling that is sensitive to obstacles, local mesh cell sizes and the time-step size. With this model one can still include a limited compressibility for the liquid, a practice that is recommended for strong liquid/solid impacts.

The second modification is to add a positive pressure "shift" to the pressure time derivative. This shift provides a net negative divergence, but one that is moderated by the scaled coefficient of the pressure term. The magnitude of the shift is proportional to the dynamic head (actually, to absolute pressure plus kinetic energy) in the cell. In this way the shift is also scaled to the local conditions, which is necessary to keep force calculations reasonably accurate. In practice, since a constant pressure can be added to flow without changing the dynamics, we modify the "dynamic head" by subtracting out the average pressure in the mesh.

The F-packing algorithm is controlled by the input parameter IFPK. Using IFPK=0 will turn off F-packing, while IFPK=1 (the default) invokes the F-packing algorithm. F-packing is only allowed in one material free surface problems.

Foam Reduction

Use of the above modifications greatly reduces the production and dispersal of foam. However, in some extreme cases these measures may still not be sufficient. For extreme cases, therefore, we have introduced one additional option. By setting IFPK=2 the new F-advection algorithm further limits F fluxes into empty cells and enhances F flux away from empty cells. In well-behaved calculations this option leads to slightly more "lumpy" free surfaces, so we do not recommend its use in general.

Free Surface Boundary Conditions

Several modifications have been made to the free surface boundary conditions. These changes improve the free surface dynamics and aid in the reduction of foam production. First, in setting velocities on the open sides of surface cells we assume zero velocity derivatives normal to the open sides. This is the same as in the old code, but instead of then adjusting one open side velocity to give a zero velocity divergence, we now use a pressure-like adjustment to all open side velocities. This adjustment, however, is proportional to the amount of F in the cell so that we do not force zero surface cell divergences, as in previous versions of the code, unless the cell is sufficiently full of fluid that it might completely fill in the next time step. The consequences of this new technique are that it takes the burden off one velocity component having to be changed excessively to make divergences zero and distributes surface cell momentum to open sides more realistically.

An example of how this setting of surface velocities can help reduce foam is a surface cell with a small F value and having a positive flow of fluid entering the cell from two or more sides. In the old procedure one open face of the cell would require a large outflow velocity to make the divergence of the cell zero. In the new procedure this task may be shared by other faces and, in any case, the velocity is less because we do not enforce a zero divergence when the F value is low. Thus, in the old procedure the larger velocity was more likely to advect fluid into empty cells, while in the new procedure this is less likely to happen.

Another modification has been introduced in the free surface boundary conditions to preserve symmetry. In particular, we have changed the way in which velocities are set outside a free surface. Instead of simply setting exterior velocity components equal to nearby interior values, we use an averaging method that accounts for all nearby interior velocity components. This method now insures a symmetric setting of these velocities whereas the previous method did not.

SAMPLE CALCULATIONS

We first illustrate the effect of these new modifications with a problem of fuel reorientation. For simplicity, a two-dimensional case with constant acceleration is considered.

As initial conditions, consider a cylindrical tank of radius 0.57 m with a cylindrical ullage having radius 0.4 m and offset toward the right hand side of the tank. A relatively low acceleration of 0.03 m/s/s is applied to the tank in the vertical (z) direction. The density of the fluid in the tank is 1440 kg/m^3. Figure 1 shows this setup and the finite-difference mesh used for all calculations, which consisted of a 20 by 20 arrangement of square cells. No surface tension was applied in the test cases presented here. Also, all calculations used the same input file data (except for the IFPK option, which was used in the third case).

Because of the acceleration, fluid at the top of the tank will slosh toward the bottom. Along the top fluid surface the fluid is subject to a Rayleigh-Taylor type of instability that tends to break up the surface, but the breakup should be somewhat retarded by the proximity of the tank wall. Calculations, in fact, show that fluid to the right of the 12 o'clock position tries to flow along the wall to the right, while fluid to the left flows along the left wall, and only a modest breakup is observed.

Figure 2 shows a comparison of computed results at t=10.0 s. The left plot in this figure is the result obtained from the original code, FLOW-3D/87. In this figure velocities are displayed in all cells containing nonzero F values in order to detect foam regions. (Usually velocities are only displayed in cells having F values greater than one half.) Between 11 and 3 o'clock there is a continuous region of foam that has been detached from the upper surface. This region is quite broad and as time proceeds it will continue to disperse over an even larger region. In fact, some foam has already collided with the fluid sloshing in the bottom of the tank and forced some of this fluid to make a 90 degree turn to the left.

The middle plot of Fig. 2 shows the results obtained with the new code version, FLOW-3D/88. Here we see less foam in the 12 o'clock to 3 o'clock region. Some fluid has been detached from the top and is falling downward, but there is little dispersal associated with this motion. The code modifications introduced into the new version have clearly made an improvement in the control of foam in this example.

Finally, in the right plot of Fig. 2 are the results obtained with the new IFPK=2 option. In this case the improvement in foam control is even better, for there is virtually no foam in this calculation at this time.

Another check of the improvements made to the new code can be seen from a comparison of forces generated, as shown in Fig. 3. The x and z forces computed with the original code are quite good in that they exhibit few oscillations except during the last couple of seconds of the calculation. If the new F-packer is working properly, then it should produce comparable force histories. That this is the case can be seen from Fig. 3, which includes the forces calculated with the new code and with the extra foam control option, IFPK=2. The principal differences observed with the new code are mild force oscillations at about 7 s to 8 s and a reduction in force oscillations at late times. The new force oscillations are the result of fluid flowing around the top boundary and striking the lower mass of fluid near the 3 o'clock location.

Another example of fuel reorientation in the same geometry can be used to illustrate the improvements made in F-packing and in the preservation of symmetry properties. In this example the cylindrical ullage bubble is located in the center of the tank so that there is left/right symmetry with respect to a vertical mid line. A vertical acceleration of 0.2 m/s/s is applied to the tank. Initial symmetry should be preserved at all times and no x forces should be computed.

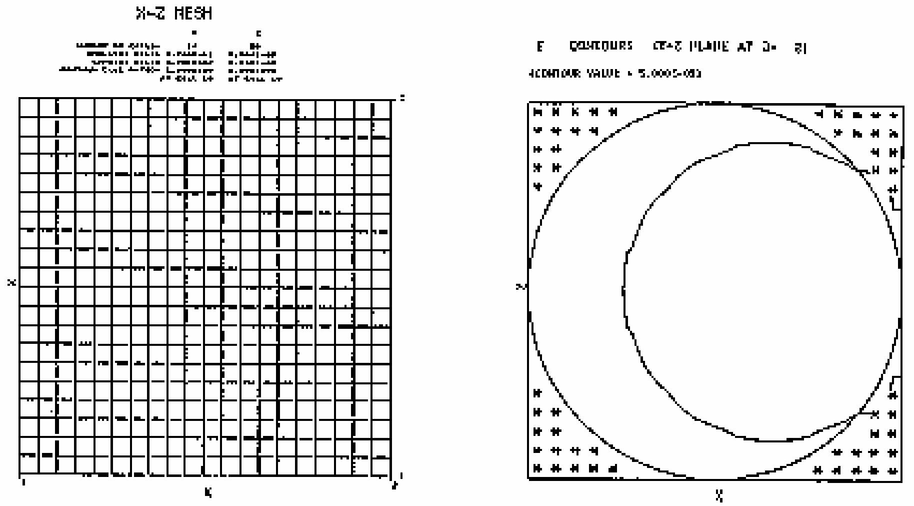
Results from the original code, FLOW-3D/87, shown in Fig. 4 (top) indicate that symmetric conditions were not maintained and that the asymmetries caused significant x forces to be generated. Both the x and y force histories exhibit large force spikes. We also note that there are numerous void regions trapped within the main body of fluid.

In Fig. 4 (bottom) are the calculational results from the new FLOW-3D/88 code for the same problem. We now see that symmetry has been quite well preserved, the x forces have low values, and there are no voids appearing in the main body of the fluid.

Based on these examples, the improvements in FLOW-3D/88 are quite significant with respect to the reduction of the foam problem and in the preservation of symmetry. There have been equally impressive improvements in other examples and, in general, we find that the new code runs more smoothly and efficiently than its predecessors.

Fig. 1.

Mesh and initial setup for foam test.



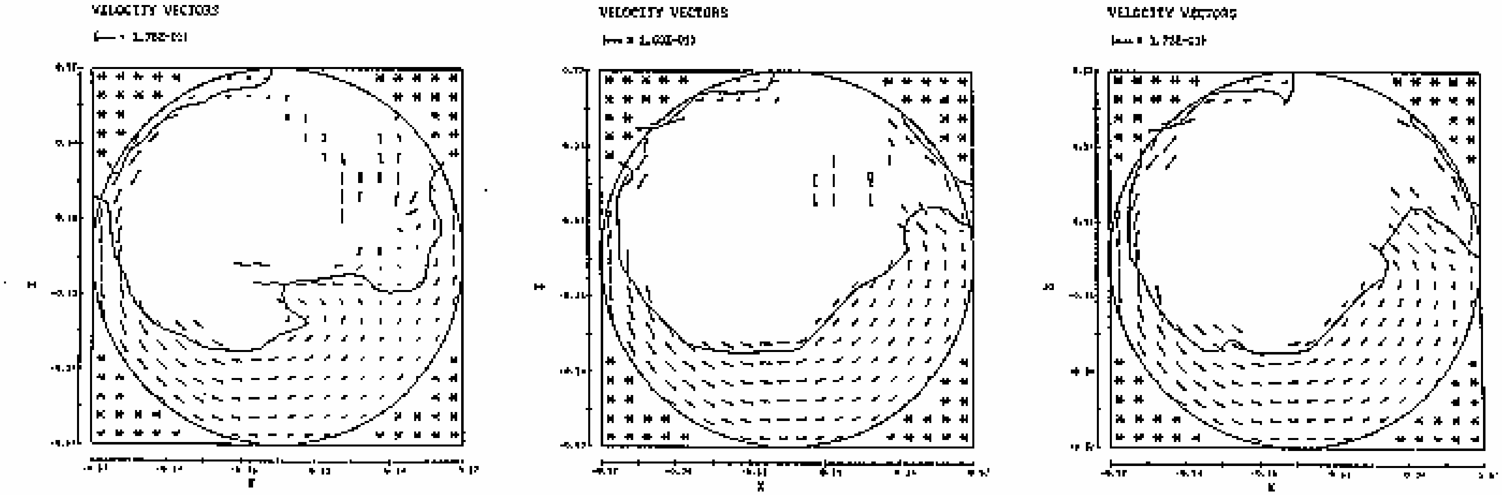


Fig. 2. Comparison of calculated results at t=10.0 s. Original code (left), new code (middle) and IFPK=2 option (right). All fluid containing cells plotted.

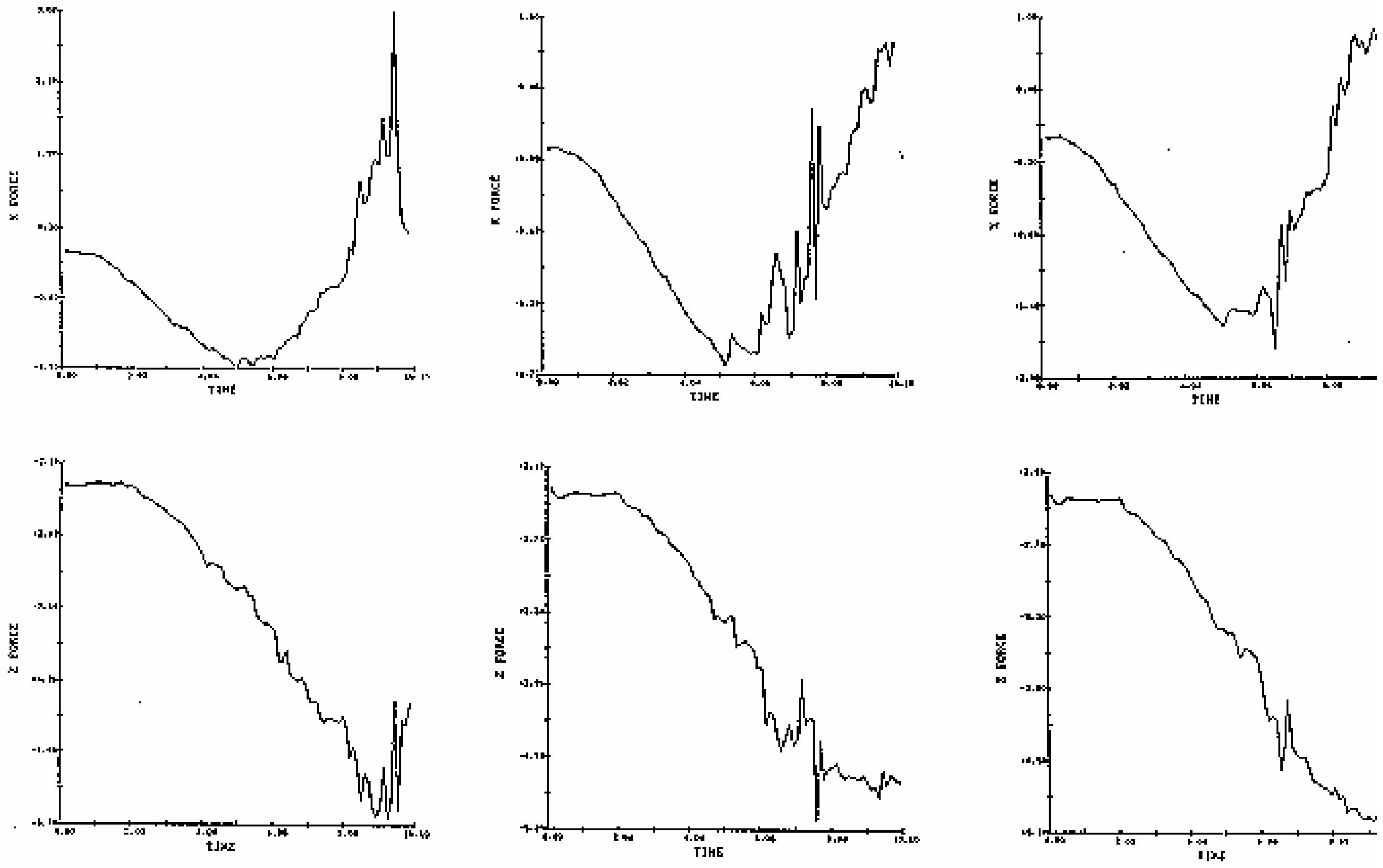


Fig. 3. Comparison of computed x and z forces. Original code (left), new code (middle) and IPPK=2 option (right). Note different vertical scales.

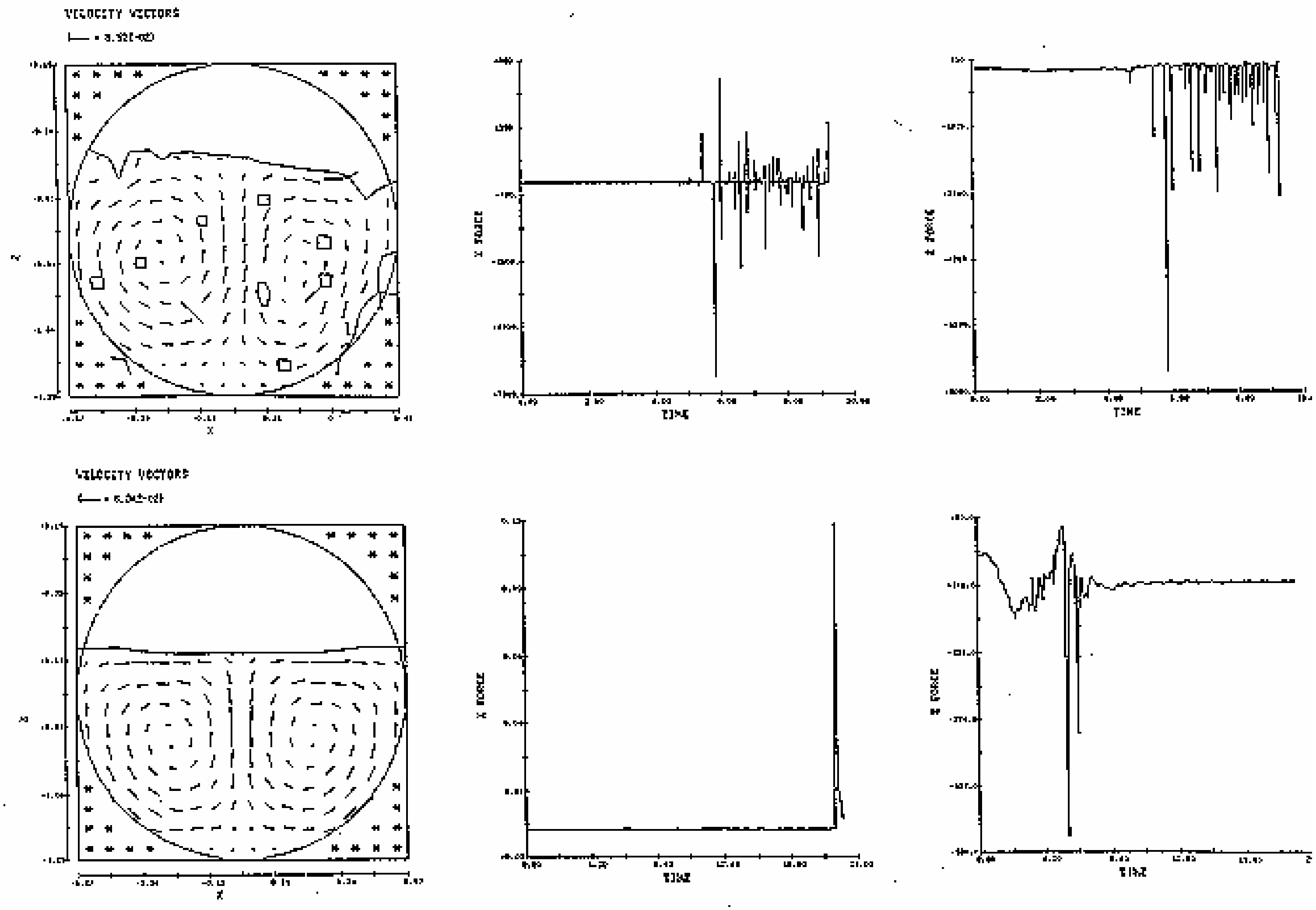


Fig. 4. Comparison of calculated results for symmetry test. Original code (top) and new code (bottom), where forces in x direction arise from small, temporary asymmetry. Plots at left are at t=20.0 s.