FSI-89-TN19R

A SIMPLE MOVING OBSTACLE METHOD FOR FLOW-3D

C.W. Hirt and R.P. Harper
Flow Science, Inc.
September 1989

OVERVIEW

Users of the FLOW-3D computer program have occasionally expressed a need for a moving obstacle capability; for example, the capability to model the flow and hydrodynamic forces associated with pistons, valves, mixers, gates, etc. In this note we present a scheme for implementing a simple moving obstacle capability into FLOW-3D. By "simple" is meant an obstacle of elementary shape in rectilinear motion.

The scheme introduced here is based on a modification of the Darcy-Drag Law used for flow in porous media. Several simplifying approximations will be assumed in this note. Only incompressible flow will be considered and no free surfaces or material interfaces will be allowed adjacent to the moving bodies. We have also "hard coded" the modifications needed in FLOW-3D to model an example test problem. That is, these changes have not been made permanent additions to the code. As experience is gained, some sort of moving obstacle will surely be added to FLOW-3D, but its precise form and capabilities cannot be predicted at this time.

In the following section we outline the general idea of the present scheme. A sample test problem illustrating the new method is presented in a subsequent section. This problem involves an oscillating plunger mechanism that could be incorporated into an ink jet printing device.

MOVING OBSTACLE CONSIDERATIONS

FLOW-3D is based on an Eulerian grid of rectangular cells. Within this grid it is possible to have obstacles and baffles defined through the use of fractional areas/volumes for each mesh cell (FAVOR technique). Such a scheme makes it tempting to introduce moving obstacles by making these area/volume fractions time-dependent. While this approach has some advantages, it also suffers from a variety of disadvantages.

For example, the use of time-dependent area/volume fractions makes it difficult to distinguish moving obstacles from those that are stationary because all obstacles must be based on the same set of area/volume fraction values. Furthermore, in this method any change of cell volume must be accounted for by the addition of sources or sinks for all cell-centered scalar quantities (e.g., density, internal energy, turbulence quantities). In particular, the sources and sinks of fluid volume in a confined flow must exactly sum to zero, otherwise the pressure-velocity iteration method will not converge. Another problem is how to initialize dependent variables in cells that have been newly uncovered by a moving obstacle.

In view of these potential difficulties it seems worthwhile to consider other possible techniques for modeling moving obstacles. One idea is to use a modification of the implicit drag mechanism already incorporated into FLOW-3D for modeling flow in porous media and solidification/melting processes. To be more specific, let us assume that a moving obstacle is represented by a region of infinite drag much like a viscous fluid whose viscosity is so large that it cannot be deformed. Under the restrictions of this note we need only consider what this assumption means for the momentum equations and the incompressibility condition.

In the usual Darcy type drag the retarding acceleration is directly proportional to the velocity, $Drag = -k\vec{u}$. This means that in the limit of infinite drag coefficient, k, the flow velocity must be zero. More generally, in a porous obstacle moving with velocity $\vec{U}$ the corresponding drag contribution to the momentum equation is,

$$Drag = -k(\vec{U}-\vec{u}) \ , \tag{1}$$

so that in the limit of an infinite drag coefficient the fluid velocity is equal to the obstacle velocity.

The numerical treatment of Darcy drag must be implicit so that a balance between pressure accelerations and drag forces can be achieved. In FLOW-3D this is accomplished by writing the momentum equation estimate for the flow velocity at time step $n+1$ as,

$$\vec{u}^{n+1} = D(\vec{u}^{n} - \nabla p + \ldots) + (1-D)\vec{U} \ , \tag{2}$$

where

$$D = 1/(1+k\delta t) \ .$$

and the ... on the right side represents all other contributions to the change in velocity. Observe that if k is infinite, then D is zero and the $n+1$ velocity reduces to the obstacle velocity.

For non-zero D the pressure p is adjusted to insure that the final $n+1$ time velocity, $\vec{u}^{n+1}$ satisfies the incompressibility condition. In the interior of the fluid, away from obstacles or other boundaries defined by area fractions, this condition is

$$\nabla \cdot \vec{u} = 0 \ , \tag{3}$$

and is used in FLOW-3D as an implicit equation for the pressure.

If the region consists of a mixture of ordinary fluid plus our supposed infinite-viscosity fluid, this condition must be generalized to

$$\nabla \cdot (f\vec{u} + (1-f)\vec{U}) = 0 \ , \tag{4}$$

where f is the volume fraction of fluid and $(1-f)$ the volume fraction of "solid".

A key to the present moving obstacle method is to observe that if we identify f with the coefficient D appearing in Eq. (2), then the incompressibility condition, Eq. (3), becomes identical to the mixture incompressibility condition, Eq. (4). In other words, with the proper definition of a drag coefficient the usual incompressibility condition is identical to a two-fluid incompressibility condition in which one "fluid" is rigid and moving with a specified velocity $\vec{U}$. By defining the drag coefficient to suitably follow a moving obstacle we can simulate the flow generated as it moves.

This observation can be better understood by examining how this drag method compares to a moving obstacle method based on time-dependent area/volume fractions. In the FAVOR technique it is assumed that velocities located in zero area/volume regions are identically zero. In particular, for stationary obstacles the incompressibility condition is written as,

$$\nabla \cdot A\vec{u} = 0 \ , \tag{5}$$

where A is the fractional area open to flow.

If the obstacles are moving, this relation must be modified to account for a change in the fractional volume, V, open to flow,

$$\partial V / \partial t + \nabla \cdot A\vec{u} = 0 ,$$

where the first term represents a volumetric source. Similar source terms must be added to all other scalar equations.

Now, if the area/volume fractions are associated with an obstacle moving with velocity $\vec{U}$, the time rate of change of solid volume fraction, $(1-V)$, must satisfy the relation

$$\partial(1-V)/\partial t + \nabla \cdot (1-A)\vec{U} = 0 .$$

The easiest way to understand this equation is to integrate it over a small control volume. Employing Gauss' theorem to convert the divergence term into a surface integral, we see that the change in obstacle volume fraction in the control volume is simply equal to the flux of obstacle volume across the surface of the control volume. (Note that incompressibility of the obstacle flow is assured because $\vec{U}$ is constant.)

Combining this relation with the fluid incompressibility condition we have

$$\nabla \cdot A\vec{u} + \nabla \cdot (1-A)\vec{U} = 0 ,$$

which is identical to Eq. (4) except that in that equation the fractional area for fluid flow was denoted by f instead of A.

This discussion has shown that the mass continuity relations in the two methods are equivalent. There are, however, significant differences in their numerical implementations. For instance, in the drag method the need for source terms in the scalar transport equations is eliminated by the automatic inclusion of volume flux terms.

Although the continuity condition can be the made the same in the two approaches, there is another significant difference. The velocity computed in the drag-based method is an area-weighted average of the fluid and "solid" velocities, Eq.(2). In the area-fraction method, on the other hand, the computed velocity is the velocity of the fluid alone,

$$\vec{u}^{n+1} = \vec{u}^{n} - \nabla p + \ldots$$

One consequence of this difference is that the drag-based method may exhibit some numerical slowing down (or speeding up) of velocity components that are located adjacent to a moving boundary. The problem arises from the advection of the averaged velocity into a region of pure fluid. Since the advection process carries some of the "solid" velocity (which is included in the velocity average) into the adjacent fluid region, this region will have its velocity slowly relaxed toward that of the solid.

Where the moving obstacle boundaries coincide with grid lines this effect is absent. Thus, the best results are obtained with this method when obstacles remain aligned with the grid lines. For example, a rectangular plunger having its vertical sides coincide with vertical grid lines and moving in the vertical direction will only suffer from this numerical effect at the face of the plunger, and then, only when the plunger is moving into the fluid rather than withdrawing.

With respect to viscous stresses, the standard finite-difference approximation used elsewhere within the fluid also applies at the boundary of a drag-defined moving obstacle because these regions are treated as though they are continuous fluid regions. Thus, it is not possible to have a free-slip condition imposed in this technique without some additional code modification. In the present application, which involves low Reynolds number flows, this is not required.

Another difference between the two moving obstacle techniques is associated with the advection of scalar quantities. At a cell boundary embedded in a moving obstacle the drag-based method can have a non-zero velocity. Since the volume fraction is also non-zero in this method, a flux must be computed at this boundary for every scalar quantity. In contrast, in the time-dependent area/fraction method the velocities on boundaries within an obstacle are zero and no fluxes are computed. But, this absence of a flux must then be accounted for by the presence of scalar source/sink terms as mentioned earlier.

As long as we are dealing with incompressible flow having no free surfaces, no heat transfer, and no turbulence transport in the vicinity of the moving obstacles there are no scalar equations to worry about, so the drag-based method needs no further consideration in this regard.

FLOW-3D CODE MODIFICATIONS FOR A MOVING OBSTACLE

To implement the drag-based moving obstacle method for the present application only two simple code changes in FLOW-3D are necessary. First, in the momentum equations (Tilde Routine) a term must be added to the right hand side of the w-equation,

$$\text{Right Side Addition} = +(1-D)W \quad ,$$

where W is specified as the time-dependent velocity of the plunger in the z direction. In this case,

$$W = a\omega \mathrm{Sin}(\omega t) \quad ,$$

in which ω is the frequency of oscillation and a is the amplitude.

The second code change is to define the drag function D used in Eq.(2). Since D is a cell-centered quantity, its value is set equal to the fractional area of the cell that is open to flow, i.e., not covered by the plunger. This change is made to routine DRGCAL following the statement "300 CONTINUE". To exercise this drag option one specifies IDRG=4 in the input file.

OSCILLATING PLUNGER APPLICATION

Statement of Problem

To illustrate this simple moving obstacle method we shall consider the two-dimensional, axisymmetric flow induced by an oscillating plunger. The plunger is located at the bottom of a chamber. At the top of the chamber there is an orifice opening to the atmosphere. When filled with ink the chamber/plunger device may be considered a possible mechanism for an ink jet printer. In fact, the geometry and time scales have been adapted from the article, "Thermodynamics and Hydrodynamics of Thermal Ink Jets," by R.R. Allen, J.D. Meyer, and W.R. Knight in the Hewlett-Packard Journal, Vol. 21, page 21, May 1985. In the original article the authors used an expanding vapor bubble as the mechanism to force ink through the orifice. In this note the bubble has been replaced by the plunger.

In keeping with the concept of an ink jet printer, the overall dimensions of the device are quite small, Fig. 1. An assumption of axisymmetric symmetry means that only one-half of a diametrical cross section must be modeled. All plots, however, will be presented with a full cross section showing. The orifice has a minimum diameter of 0.006 cm with its exit plane located 0.015 cm above the floor of the chamber. The diameter of the chamber is 0.021 cm, while the plunger extending through the floor has a diameter of 0.017 cm.

Initially the plunger is at rest and extends 0.0005 cm above the bottom of the chamber. A single stroke of the plunger consists of the sinusoidal motion,

$$h = 0.0020 - 0.00075\mathrm{Cos}(\omega t)$$

where h is the height of the top face of the plunger above the bottom of the chamber and $\omega = 2.094E+5$ rad/s corresponding to a plunger period of 0.03 ms. In any real ink jet there must be some way to replenish the ink supply in the chamber before another stroke is initiated. Here we have omitted this feature in the interest of simplicity. Our goal is to investigate the flow dynamics in the chamber and in the initial jet formation generated by a single stroke of the plunger.

Another approximation introduced for simplicity is to assume that the initial ink surface coincides with the exit plane of the orifice. In reality, surface tension and wall adhesion forces would generate a curved surface as the equilibrium surface shape.

The ink is treated as an incompressible fluid with density of 1.0 gm/cm^3, kinematic viscosity of 0.06 cm^2/s, surface tension coefficient of 20.0 dynes/cm and a wall contact angle of 15 degrees. If the average speed of the plunger and its maximum displacement are used as characteristic values, then the Reynolds number of the flow is 9.98. In other words, because of the small size of this device, we are dealing with a rather viscous flow process.

Computational Model

All computations have been performed with the FLOW-3D program as modified by the two changes described earlier.

The width of the device, including a one cell obstacle boundary at the outer radius, has been subdivided into 21 mesh cells. The vertical extent of the chamber and orifice is represented by 20 cells with an additional 20 cells located above the orifice to resolve the early formation of the ink jet. A nonuniform mesh was used to increase the radial resolution of the jet and the vertical resolution near the plunger. The minimum cell size in the radial direction was 0.00043 cm, while in the vertical direction the minimum cell size was 0.0005 cm. A picture of the computing mesh is given in Fig. 2.

Figure 3 contains the entire input file for the calculation. Although the bottom boundary of the mesh was defined to be a continuative boundary, a baffle has been used to close off that portion of the boundary outside of the plunger. This procedure allows the plunger to move in and out across the bottom of the mesh.

A seemingly large pressure iteration convergence parameter of EPSI=500 is consistent with the rule-of-thumb value $(2.0E-4)(u/\delta x)$, where u and δx are a typical large velocity and small cell size, respectively.

Computational Results

Figure 4 contains the computed fluid configurations and velocity fields at quarter cycle times during one complete cycle of the plunger. Velocity vectors are plotted without arrowheads, but small plus signs, "+", are placed at the bases of the vectors to mark the location of fluid even where its velocity may be zero. This is necessary in order to distinguish void regions from fluid regions. Vector bases are at mesh cell centers while the magnitude and direction of a vector reflects the computed velocity in the cell.

A strong ink jet has been formed and has partially left the computing mesh by the end of the calculation. Although surface tension may be smoothing the surface of the jet somewhat, the major contribution to its shape is the motion of the plunger. During the second half cycle the plunger is retreating and pulling fluid back into the orifice, which causes the jet diameter to narrow substantially.

The jet does not pinch off during the plunger cycle. To see if pinch-off would occur in another half cycle, the calculation was continued with the plunger held stationary. This result is shown in Fig. 5A. The jet within the orifice has significantly narrowed, but pinch-off has not yet occurred. A plot of pressure contours in the narrow portion of the jet is shown in Fig. 5B. The highest pressures in this region are in the jet, and a simple analysis shows that this pressure is the value expected from surface tension due to its axisymmetric curvature. While it might be expected that surface tension forces would dominate much of the jet motion, this is not the case for the period of flow covered by the calculation because of the high material velocities within the jet. For instance, at the end of the plunger cycle a rough Weber number for the flow in the jet is 40, which indicates that inertial effects are about 40 times more significant than surface tension forces. On the other hand, at the time shown in Fig. 5 the Weber number is on the order of 5 so that surface tension is reaching the point where it will start dominating the jet's behavior. In fact, in the exit plane of the orifice the jet velocity is very low and surface tension will eventually cause the jet to pinch off.

Flow Near Plunger

A better idea of the plunger motion and the flow it generates in its vicinity is given by the plots in Fig. 6. Here only that portion of the chamber extending up to the entrance of the orifice is plotted. The plunger boundary has been drawn in by hand.

An interesting secondary flow can be seen in the half cycle plot of Fig. 6 when the plunger has reached its top dead center position. The upward inertia of the flow is trying to pull fluid away from the face of the currently stationary plunger. This causes fluid to be sucked in from the sides and is the source of the secondary recirculation seen along the outer sides of the chamber. This secondary flow appears strongest along the side wall of the plunger because flow was earlier being dragged along by the plunger through viscous effects and because of a magnification of the flow speed due to radial convergence.

SUMMARY AND COMMENTS

Modifications to FLOW-3D have been described and used to model the flow generated by an oscillating plunger. The moving obstacle modeling technique used here was based on an adaptation of a time-dependent, porous-media drag coefficient. For the present application, this technique was shown to be a simple and effective method.

Calculation Summary

A calculation was carried out for a model ink jet device. The ejection of ink from a reservoir through an orifice was generated by a moving plunger located at the bottom of the reservoir. Only one cycle of plunger motion was followed because no allowance was made for replenishment of ink in the reservoir. During this time a strong jet was generated with velocities exceeding 1400 cm/s, but the jet does not pinch off. An extended calculation for another half cycle of time, but with the plunger held stationary, demonstrated that pinch-off requires a considerable length of time. An earlier pinch-off time could be engineered, perhaps by altering the motion of the plunger to have a more rapid withdrawal stroke. It is in designing such variations that FLOW-3D simulations can be extremely useful.