Flow Science Report 02-14

# ADDING FLOW LOSSES TO FAVOR™ METHODOLOGY

C.W. Hirt
Flow Science, Inc.
June 2014

## Background

From time to time it has been observed that large velocities may appear in the vicinity of grid elements that are severely, but not completely, blocked by solid obstacles. On some occasions these regions even exhibit an instability in which these velocities continue to grow with time. In this note the reason for these problems, or at least for some of these problems, is discussed and a solution is given.

## Problems Associated with Abrupt Area Changes

Instabilities that sometimes arise in a flow solution can often be traced to locations where there is a large change in flow area across a grid element. For example, consider an element having a small area fraction on its left side and a full open area on its right side. When flow is moving fluid from the left to the right there is a large reduction in velocity at the right in order to have a conserved volume of flow (i.e., uA should be constant in the direction of flow, where A is the fractional area open at the side of an element). At the right side the velocity receives a larger advective flux from the left side than it should because no mechanism is in place to allow for an expansion of the flow between the left and right sides. To recover the correct velocity at the right side the pressure in the element must decrease more than it otherwise should. In some circumstances this decrease may then pull in a cross--flow that grows with time.

It is well known that there should be significant flow losses at abrupt area expansions and contractions. Such losses were never considered in the original development of the FAVOR™ method, so it is not surprising that their absence could cause some distortions in the flow. In the next section a brief discussion of flow losses is given, which is then followed by a proposed addition to the advective flow that aims to account for these losses.

## Flow Losses at Abrupt Area Changes

A good discussion of flow losses is given in the book “Elementary Mechanics of Fluids,” by Hunter Rouse, Dover Pub., N.Y., (1946). The discussion given here is largely based on Rouse’s book.

The Bernoulli theorem relating velocities and pressures along a streamline comes from the steady momentum equation for a constant density fluid,

$$\vec{u} \bullet \nabla \vec{u} = -\frac{1}{\rho}\nabla p + \nabla(gz). \tag{1}$$

It is assumed here that gravity is in the z direction and that there are no other body forces or dissipative forces (e.g., viscosity). If the dot product is taken between this equation and the velocity the result is,

$$\bar{u} \bullet \nabla\left(u^2/2\right) + \bar{u} \bullet \nabla\left(p/\rho\right) - \vec{u} \bullet \nabla\left(gz\right) = 0, \quad (2)$$

or

$$\vec{u} \bullet \nabla\left(u^2/2 + p/\rho - gz\right) = 0 \quad (2a)$$

This result says that along a streamline the following quantity is equal to a constant,

$$\frac{u^2}{2} + \frac{p}{\rho} - gz = const., \quad (3)$$

a relation known as the Bernoulli theorem.

In the presence of an abrupt expansion this relation is not satisfied because there are flow separations that generate turbulence and subsequent viscous dissipation. At an abrupt area expansion the average mass and momentum fluxes before and after the change are conserved, but the average kinetic energy is not and there is a “loss” that is manifested by an extra drop in pressure.

The object of this Note is to add such losses explicitly to the flow equations so that the local fluid pressure does not have to decrease so much to account for the losses. In this way the disturbance of neighboring regions by pressure forces should be reduced. It has also been noted that this addition eliminates a variety of numerical instabilities that may arise from certain combinations of fractional area/volume fractions.

**Application of Flow Losses to FAVOR™ Method**

Within a grid element, if the flow area on one side is different from the flow area on the opposite side then in principle there should be some flow loss associated with that abrupt change in area. In the FAVOR™ method it is always assumed that we have no knowledge of a flow distribution within a cell and at an element edge the normal velocity component there is assumed to represent an average uniform flow across that cell edge. By not including some realistic losses we introduce velocity perturbations that are often undesirable. Of course, the worse cases are likely to be those associated with the largest area changes, which tend to be in elements having little open volume but lying adjacent to fully open elements. It is in these locations that stability problems have occasionally been observed.

In practice it is found that the loss terms can be too large because the flow may require some distance downstream of an area change before the full loss in flow pressure can be realized. Also, there are situations where area changes in the FAVOR™ method are not likely to result in flow losses. For these reasons the loss terms must be applied with some restraint.

## Illustrative Examples

### Flow off a Step

Why a change in pressure resulting from an area change may be important can be appreciated from a simulation that led to this development: a 2D flow of a layer of liquid off a backward facing step. The step obstacle was initiated such that it was not coincident with grid lines. In particular, the top surface of the step was inside a row of mesh elements and its right side was inside a column of elements, see Fig.1. Thus, the grid element containing the corner of the step has abrupt area expansions in both the horizontal and vertical directions. A consequence of this arrangement is shown in Fig.2, which contains snapshots of the flow at three times separated by 0.2s. The flow jetting off the step exhibits an oscillation that continues more or less periodically.

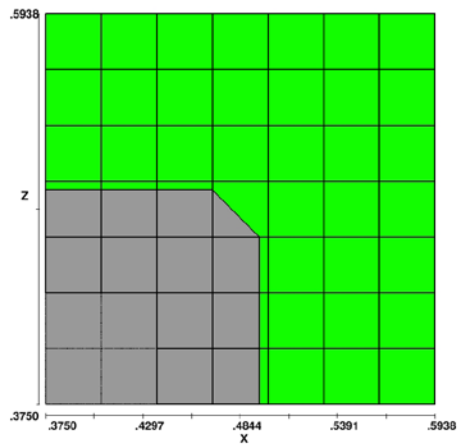


Figure 1. Close up of grid in vicinity of step corner. Element side areas change in both x and z directions in the element containing the obstacle corner.

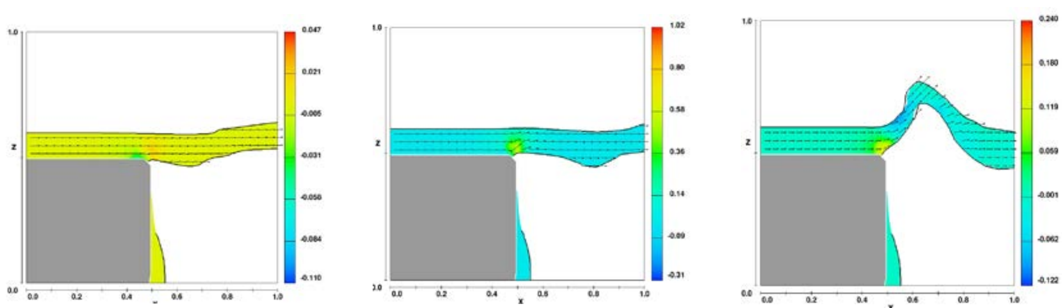

Figure 2. Snapshots of jet off a step showing an unphysical oscillatory behavior. Color indicates pressure.

A detailed study of the mechanism reveals that a low pressure in this corner element, partially caused by the area expansion in the horizontal direction, sucks flow in from the element below when that element contains a free surface having a constant pressure. The area expansion in the vertical direction then results in an excess transfer of vertical momentum across the corner element leading to the redirection of the jet upwards.

By introducing flow losses associated with abrupt expansions we can reduce both the unnaturally low pressure and the excess transfer of momentum, as shown in Fig. 3, which is a repeat of the simulation in Fig. 2, but with the loss mechanisms described in this note. The Fig. 3 result is steady and does not oscillate or otherwise exhibit fluctuations.

Variations on this test case using smaller and larger flow blockages in the corner cell, including viscosity and gravity effects all show a nice stable jet flowing off the step.

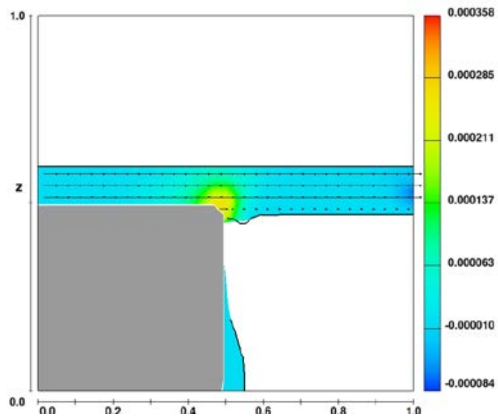


Figure 3. Flow off a step simulated with the addition of abrupt area flow losses. Fluid at base of step does not flow down because there is no gravity.

## Flow in Diagonal Duct

One of the key tests of the FAVOR™ method is a uniform flow in a duct lying at an angle to the computational grid, Fig.4. Without some consideration to boundary conditions at solid walls, there will be an artificial viscous-like drag along the duct walls because the velocity advection terms pick up zero velocity values in the wall areas. The FAVOR™ method uses a special technique to eliminate these artificial drag effects.

One worry about the inclusion of flow losses is that in a diagonal duct there are cut cells having both area enlargements and contractions where the side walls of the duct cut through a mesh. Adding losses may now alter the zero drag construction. Tests show that the losses do, in fact, modify the results of this test case and should not be used. Further, this example does not suffer from instabilities so losses are unnecessary in any case.

To prevent the addition of unnecessary flow losses, a flag has been added to the program input that may be used to select the FAVOR™ Loss addition as an option. This flag is

***ifloss*** where the default value, ***ifloss=0***, means do not use the loss option. A value of ***ifloss=2*** signifies that losses are to be used at all area changes.

Finally, for situations where the user is not sure whether or not the addition of losses would be useful, inputting ***ifloss=1*** activates a test in the Mentor routine to look for possible instabilities and if it detects such a possibility it turns on the losses by resetting ***ifloss****=2*. This option is desirable because it does not apply losses unless they appear to be necessary. For example, in many complex high-pressure die casting simulations this choice can keep a simulation running smoothly without having to be repeated because it failed with an instability.

In contrast, using the ***ifloss=1*** option for the Diagonal Duct problem works well because the Mentor detects that no instabilities are developing and so no losses are introduced into the simulation. The results are identical to the case with no losses added, ***ifloss=0***, as seen in Fig.4.

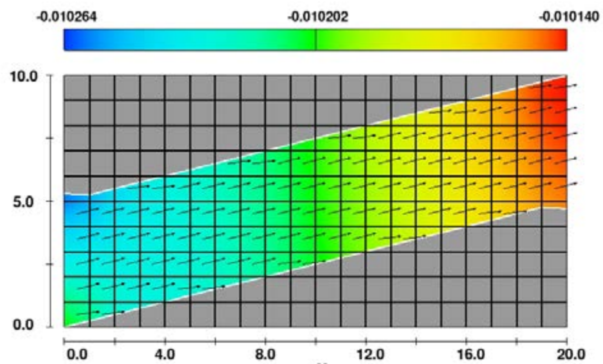


Figure 4. Flow through a diagonal channel that cuts cells creating both abrupt expansions and contractions. Flow shows no artificial viscous drag at walls with ***ifloss=1***. Color indicates pressure, which has a very small variation across the duct, a further indication of no drag resistance to the flow.

## Flow in a Wave Breaker

An example of a situation that exhibits numerous flow instabilities due to many complicated regions with large variations in element area fractions is shown in Fig.5, courtesy of XC Engineering. This is a two-dimensional, vertical slice through a three-dimensional region that was designed to dissipate wave energy from random waves striking a shore. A rock bed at the bottom of the structure has been modeled by many irregular solids (i.e., rocks). Fluid can flow in and around these solids, but because of the limited resolution of the grid this entails extensive use of the FAVOR™ method's area and volume fractions to represent the complicated geometry.

Initial simulations of this problem led to instabilities associated with large velocities developing in the small irregular passages in the rock bed. However, when flow losses are added the simulation runs without difficulty for many wave periods. Either the ***ifloss=1*** or ***ifloss=2*** option may be used for this problem, but the ***ifloss=1*** option was used and offers a good example of how the Mentor program can monitor a simulation and make changes to keep it running smoothly. In Fig.6 the history of the time-step size is plotted. The arrow indicates the time at which the Mentor detected a possible instability and activated FAVOR™ flow losses. The developing instability is indicated by a rapid and uniform decline in time-step size. When the flow losses have been added the time step immediately recovers and the simulation continues to run well (the plot shows the time history to 50s; the simulation was actually run to 250s).

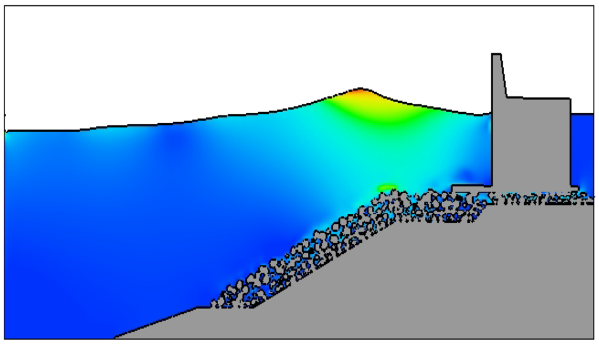

Figure 5. A slice through a wave breaker consisting of a bottom rock bed and a vertical sea wall. Simulation is fully three-dimensional. Color indicates velocity magnitude.

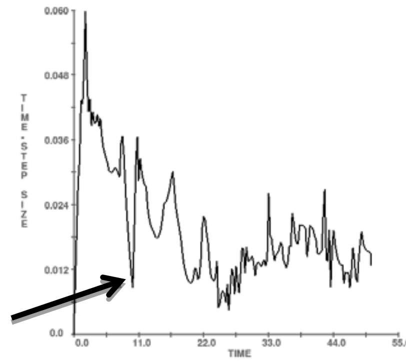


Figure 6. Time-step size history. Arrow marks time that Mentor activated flow losses.

## **Summary**

In practice, the addition of loss terms to momentum advection can be viewed as variations in the finite-difference expressions for advection much like using a donor cell method as opposed to a central difference or higher order approximation. The losses only occur at localized positions in a grid where there are area changes in the direction of flow. Refinement of a grid means that these loss adjustments shrink with the grid size because they are confined to cells only where FAVOR™ area fractions are changing. These losses also help bring neighboring velocities across an area change together as a grid is refined, a necessity for fluid continuity. In fact, in the limit of vanishing grid size the difference equations, including the flow loss terms, reduce to the correct governing conservation equations for a continuum fluid.

Finally, as an aid to users in situations where the need for adding losses is not *a priori* evident, the Mentor function in the program can be used to look for possible instabilities and to only activate the addition of losses when they appear to be necessary to maintain stability.