

Modeling Turbulent Entrainment of Air at a Free Surface

C.W. Hirt

Flow Science, Inc.

Overview

In free-surface flows the turbulence in the liquid may be sufficient to disturb the surface to the point of entraining air into the flow. This process is important, for example, in water treatment where air is needed to sustain microorganisms for water purification and in rivers and streams for sustaining a healthy fish population. Air entrainment is typically engineered into spillways downstream of hydropower plants to reduce the possibility of cavitation damage at the base of the spillway. Other situations where air entrainment is undesirable are in the sprue and runner systems used by metal casters, and in the filling of liquid containers used for consumer products.

The importance of being able to predict the amount and distribution of entrained air at a free liquid surface has led to the development of a unique model that can be easily inserted into **FLOW-3D**[®] as a user customization. The model has two options. One option, to be used when the volume fraction of entrained air is relatively low, uses a scalar variable to record the air volume fraction. This model is passive in that it does not alter the dynamics of the flow.

A second air-entrainment model, option two, is based on a variable density formulation. This model includes the “bulking” of fluid volume by the addition of air and the buoyancy effects associated with entrained air. However, this dynamically coupled model cannot be used in connection with heat transport and natural (thermal) convection.

In both model options the same basic entrainment process is used that is based on a competition between the stabilizing forces of gravity and surface tension and the destabilizing effects of surface turbulence. The model is described in the next section. Because turbulence is the main cause of entrainment, a turbulence-transport model must be used in connection with the air-entrainment model (i.e., *ifvis*=3 or 4). It is recommended that the RNG version of the more traditional k-epsilon turbulence model be employed. All the validation tests reported in this Technical Note were performed using the RNG turbulence model.

Model Description

Air entrainment at a liquid surface is based on the idea that turbulent eddies raise small liquid elements above a free surface that may trap air and carry it back into the body of the liquid. The extent to which liquid elements can be lifted above a free surface depends on whether or not the intensity of the turbulence is enough to overcome the surface stabilizing forces of gravity and surface tension.

Turbulence transport models characterize turbulence by a specific turbulent kinetic energy Q and a dissipation function D . A characteristic size of turbulence eddies is then

given by $L_t = c_n u (3/2)^{1/2} Q^{3/2} / D$. We use this scale to characterize surface disturbances. The disturbance kinetic energy per unit volume (i.e., pressure) associated with a fluid element raised to a height L_t , and with surface tension energy based on a curvature of L_t , is $P_d = \rho g_n L_t + \sigma / L_t$. Here ρ is the liquid density, σ its coefficient of surface tension, and g_n is the component of gravity normal to the free surface.

For air entrainment to occur the turbulent kinetic energy per unit volume, $P_t = \rho Q$, must be larger than P_d , i.e., the turbulent disturbances must be large enough to overcome the surface stabilizing forces.

The volume of air entrained per unit time, δV , should be proportional to the surface area, A_s , and the height of the disturbances above the mean surface level. All together we write $\delta V = C_{air} A_s (2(P_t - P_d) / \rho)^{1/2}$, where C_{air} is a coefficient of proportionality. If P_t is less than P_d then δV is zero. The value of C_{air} is expected to be less than unity, because only a portion of the raised disturbance volume is occupied by air. A good first guess is $C_{air} = 0.5$, i.e., assume on average that air will be trapped over about half the surface area. This value has been used in all validation tests.

Experience has shown that this model is good for most applications, but there is one situation where the model must be refined. When a liquid jet enters a pool of liquid there may be too little time (and numerical resolution) for the turbulence model to generate a sufficient level of turbulence to begin entrainment. For such cases we have added some additional logic to the model to sense when there is an intersection of two fluid surfaces in which one is moving normally into the other. At such intersections we enhance the surface turbulence level by adding to it 10% of the specific kinetic energy associated with the relative normal velocity of the intersecting surfaces. We also modify the turbulence length scale by averaging the turbulence-model length scale with a size scale for the intersecting surfaces. For the intersecting-surfaces length scale we choose the square root of the local entrainment surface area. The averaging of the scales is done using a weighting equal to the respective kinetic energies of the two contributions to surface turbulence.

Adding the Entrainment Model to **FLOW-3D**[®]

At the present time the air entrainment model is a user customization that makes use of a special version of the *qsadd.f* routine. This routine is available to any **FLOW-3D**[®] users who have an active program license. A dummy variable (dum4) is used for the value of C_{air} , and the model option that uses a scalar function assumes that this scalar is number 1. However, the scalar index can be easily changed in the user data section of the *qsadd.f* routine. Finally, a value for surface tension must also be set in the data section of *qsadd.f*; this separate setting for the surface tension coefficient is used so that surface tension is not activated in the solver where it is rarely needed for problems involving air entrainment.

Using the Model

There are two ways that this entrainment model may be used. For small amounts of entrained air a scalar variable can be defined to record the fractional volume of entrained

air. Input for this scalar should use the higher order advection option, *isclr()*=3, and the turbulent diffusion coefficient multiplier, *rmsc*=1.0, which allows turbulence to mix air into the bulk of the liquid. The customized routine *qsadd.f* locates each surface cell and adds the product of time-step size and δV to the scalar that records the fractional volume of air.

In this simple model, entrained air does not change the volume or density of the liquid. This is a reasonable approximation as long as the fractional volume of entrained air f_a remains small. To insure that the volume fraction of air does not exceed unity in this version of the model, which would make no sense and could cause computational troubles, we include a factor of $(1-f_a)$ in the air entrainment rate.

A second way to employ the air entrainment model is to use the one-fluid, variable density option by inputting *ifrho*=2 or 3 and setting *rhofs* equal to the density of air. In this case the volume of entrained air is taken into account as an increase in the total fluid (mixture) volume fraction. Furthermore, buoyancy effects associated with a variable mixture density are also taken into account. No scalar variable is required when this option is selected and there is no need for the extra factor of $(1-f_a)$ in the entrainment rate that was necessary in the scalar variable version.

The amount of entrained air as defined by its volume fraction is simply the value of the scalar variable when that version of the model is used. With the variable density model version the volume fraction, f_a , of air can be computed from the relation

$$\rho = (1-f_a)\rho_f + f_a\rho_s,$$

where ρ is the mixture density output by the program. When $\rho_s/\rho_f \ll 1$ (e.g., air/water=0.001), then a good approximation is $f_a = (\rho_f - \rho)/\rho_f$.

Validation of the Model

Four different tests have been conducted to validate the usefulness of the air-entrainment model. For all cases we have simply used a constant entrainment coefficient $C_{air}=0.5$ and the properties of water and air.

Jet into a Pool

An axisymmetric jet of water of diameter 1 cm is allowed to fall under gravity into a pool of stationary water that is 3 cm deep. The jet begins its fall 4 cm above the surface of the pool. The jet velocity where it enters the pool is approximately 1.0m/s (100 cm/s).

This problem illustrates the case where our model required the addition of logic for the intersection of a one fluid surface moving normally into another surface. Experimental data (Ervine 1980) indicates that there is a critical jet velocity of about 0.8m/s below which no air entrainment will occur. We used this critical value to guide the addition of the surface intersection treatment. In particular, the choice of combining only 20% of the mean kinetic energy associated with intersecting surfaces into the turbulence causing

entrainment. In this sense this test case is not strictly a validation, but by construction it is still a case for which *FLOW-3D*[®] gives results in agreement with data.

Strictly speaking, experiments indicate that some entrainment will take place when the jet velocity is below the critical value of 0.8m/s provided the level of turbulence in the jet is raised to a sufficiently high level. This is likely to be true in the simulations as well, but no tests of this have been made because of the limited interest in this limit of low jet velocity.

Figure 1 shows the computed flow using the scalar option for recording the volume fraction of air (indicated by color shading), which is justified by the relatively low level of entrainment. No data for the distribution of entrained air in the pool has been found with which to make comparisons. However, the general level of entrainment below the pool surface, $f_a=0.04$, seems quite reasonable (see the next example). The maximum entrainment right at the surface was about $f_a=0.4$, but this is quickly reduced at the air enters the pool.

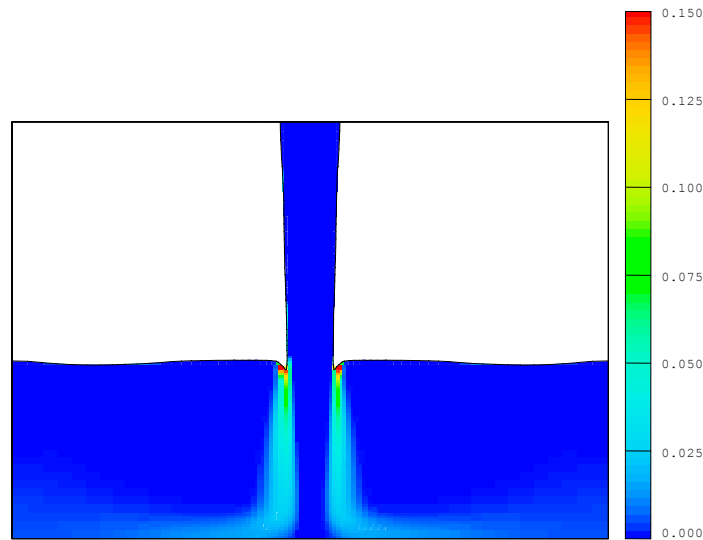


Figure 1. Jet into pool. Color shading indicates air volume fraction.

Drop Shaft

A problem closely related to a jet into a pool is that of water flowing into a vertical shaft. In this case we have simulated experimental data obtained by Ervine 1981 for rectangular, vertical drop shafts. Because the shaft was rectangular it is possible to use a two-dimensional model that corresponds to a vertical section of the shaft. In our case the shaft has a width of 0.5 m and a depth of 2.0m. The bottom end of the shaft is open and immersed in water with a pressure head of 1.0m. Only the right half of the top of the shaft is open to the flow of water.

Water enters the top and falls under gravity as a wall jet along the right side of the shaft. Although the pool height in the shaft is unsteady, it moves up and down about the center of the shaft (i.e., 1.0m above the bottom). The width of the jet where it enters the pool is about 8 cm and have a vertical velocity of about 4.5 m/s, which corresponds to a Froude number of roughly $Fr=5.7$.

Our simulation employed the variable density model because it was thought that buoyancy caused by air entrainment might have some effect on the flow in the pool. Repeating the simulation with the scalar option showed little difference in the results, presumably because the jet velocity is so large that buoyancy forces cannot compete with inertia, at least in the region modeled.

As already noted, the flow does not reach a steady state, but exhibits an oscillation in pool height, with discrete eddies generated by the jet moving downward and out the bottom of the shaft. A snapshot of the computed flow shown in Fig. 2 provides a good picture of the general distribution of mixture (air/water) density.

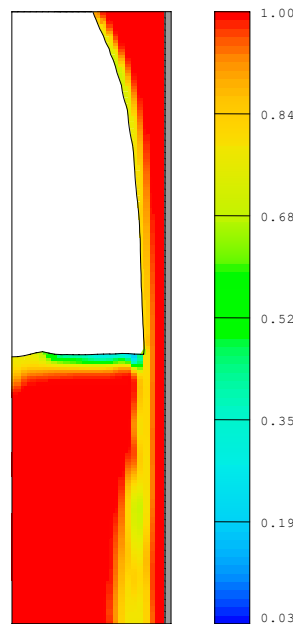


Figure 2. Variable density in drop shaft flow indicates air entrainment.

Ervine (1981) measured the ratio of the volumetric flow of air to that of water, Q_a/Q_w , and gives the empirical correlation,

$$\frac{Q_a}{Q_w} = 0.0045 Fr^2 \left(1 - \frac{u_c}{u_{jet}}\right)^3, \quad \text{with } u_c = 0.8 \text{ m/s.}$$

For the simulated case $u_{jet}=4.5$ m/s and $Fr=5.7$ so that the empirical result should be $Q_a/Q_w=0.081$. To get volumetric flow rate data from the simulation an addition was made to the end of the *qsadd.f* routine to compute and print the ratio of volume flow rate

for air versus that for water. This addition was only done for the scalar option but we have already indicated that the differences between the two model options were small.

In any case the computed results gave a mean value of Q_a/Q approximately equal to 0.08, essentially the same as the experimental result of 0.081. The computed value exhibited an oscillation of about $\pm 10\%$ because of the unsteadiness of the flow. However, the experimental data are reported to have an accuracy of $\pm 20\%$ in relation to the corresponding empirical equation. The computed values are thus seen to be in good agreement with the data for both the scalar and variable density models.

Hydraulic Jump in a Conduit

Our third test case involves a closed conduit. In this case a conduit inclined 5.71 degrees (10% slope) with respect to the horizontal. The lower end of the conduit is immersed in water whose hydrostatic head at the top edge of the outlet is 6.12 cm. At the inlet water enters with a depth of 5.0 cm. and rushes down the conduit under the action of gravity. A hydraulic jump develops about 50.0 cm down stream from the inlet.

For simplicity we have elected to model the circular conduit with a two-dimensional vertical slice through its center. The conduit has a diameter of 15 cm. (approximately 6") and a length of 100.0 cm. The scalar model option was used for the simulation.

Figure 3 shows a snapshot of the computed results at the end of the simulation, $t=3.0$ s. There is some unsteadiness associated with the hydraulic jump, as there should be, but the average flow is nearly steady.

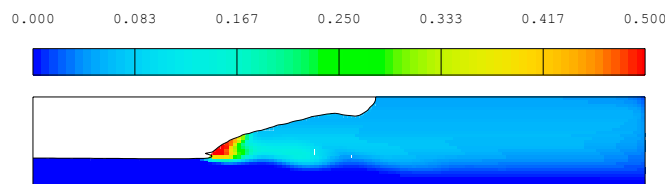


Figure 3. Hydraulic jump in conduit. Color indicates volume fraction of air.

This test case was designed to approximate an experimental study reported by Kalinske and Robertson (“Closed Conduit Flow,” by Kalinske, A.A. and Robertson, J.M., published paper in unknown Symposium proceedings is based on doctoral work of Robertson submitted to the U. of Iowa August, 1941).

The relevant data obtained in the experimental study are the air-to-water volumetric flow rate ratios versus the Froude number (Fr) of the incoming flow. The data is represented by the empirical expression,

$$\frac{Q_a}{Q_w} = 0.0066(Fr - 1)^{1.4}$$

For the case simulated the Froude number was 3.0 and the average computed ratio of flow rates was 0.018, which is to be compared to the above formula's value of 0.0174. The computational result is about 3% high, which is easily within the scatter of the experimental data. Thus, the computations are in excellent agreement with the data for a 10% sloped conduit.

Spillway

The final test involves a model spillway having a parabolic surface. The horizontal length of the spillway in the computational region is 12.5 m. At the end of the spillway its slope was about 37° with respect to the horizontal. The reservoir has a water elevation that exceeds the crest of the spillway by 1.0 m. It is assumed that the level of turbulence in the reservoir is negligible, so that any air entrainment will be generated by the build up of a turbulent boundary layer along the spillway surface. A surface roughness of 1 mm, corresponding to concrete, was defined for this purpose.

The simulation was run using the variable density model and was carried out well beyond steady state conditions, Fig. 4. That air entrainment begins where turbulence reaches the free surface can be seen in Fig.5, which shows the computed specific turbulent kinetic energy.

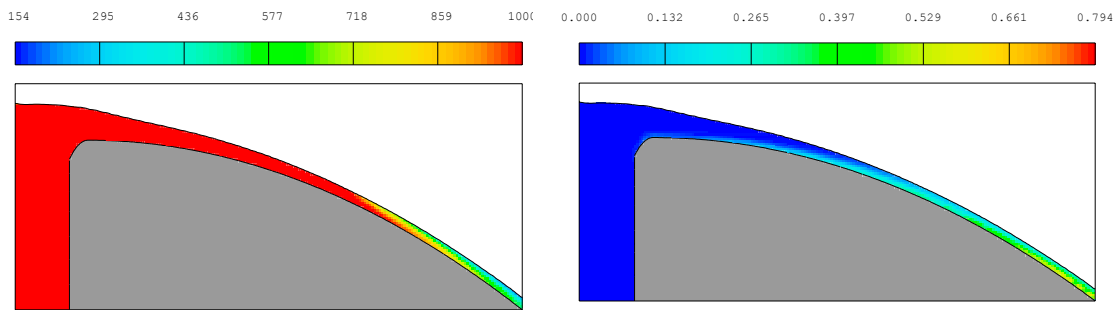


Figure 4. Air entrainment on spillway.

Figure 5. Turbulent kinetic energy.

The experimental data that we shall compare with has been taken from “Air Entrainment in Free-Surface Flows” written by I.R. Wood in the IAHR Monograph, “Hydraulic Design Considerations,” published by Balkema, Rotterdam, 1991. The data is a summary of experimental work conducted by L.G. Straub and A.G. Anderson of the St. Anthony Falls Hydraulics Laboratory at the U. of Minnesota, 1958.

We assume that the air distribution at the end of the modeled spillway has reached the observed “equilibrium” condition where it is no longer varying significantly with distance along the spillway surface. This appears to be the case, but no detailed evaluation has been attempted to prove this assertion.

The Straub and Anderson data are shown by Wood (see his Fig. 3.9) to be well fit by the function,

$$f_{air} = \frac{b}{b + e^{-ay^2}}$$

In this expression the volume fraction of air, f_{air} , is related to the nondimensional height y (normalized to be unity at the height where $f_{air}=0.9$) with two constants a and b that depend on the slope of the surface. For a 37.5° slope (the nearest value to our 37° that is reported by Wood) the values are $a=2.65$ and $b=0.638$.

To compare calculated results to the empirical expression we must first get the air volume fraction as a function of distance measured normal to the surface. To do this it was necessary to use the custom plotting feature of **FLOW-3D**[®] to determine the grid cell locations and values needed for performing a simple interpolation. The following table compares the computed and experimental values:

Distance	Experimental Air Fraction	Computed Air Fraction
0.095	0.395	0.429
0.38	0.483	0.486
0.76	0.747	0.703
0.95	0.875	0.857

The maximum error is at the lowest point and is about 8%, which is well within the experimental data scatter, an excellent result.

Summary

A new air entrainment model has been proposed for incorporation into **FLOW-3D**[®]. The model can be used in one of two options. For low air volume fractions a simple scalar function is used to record the amount of entrained air. When higher volume fractions of air are likely, or when the buoyancy of the air/water mixture might be important, a second model option can be used that records entrained air in terms of a variable mixture density.

For either model option there is one empirical coefficient, C_{air} , a nondimensional coefficient that can be adjusted, but the recommended value of $C_{air}=0.5$ was found to work for all the test cases considered.

Four different types of sample problems were used to test the new model and good results were obtained in every case. The most sensitive aspect in all the simulations was found to be the level of turbulence in an impinging jet into a pool. This sensitivity does not occur in situations where the entrainment is associated with a hydraulic jump because in those cases the instability of the jump generally over shadows any incident turbulence.

It is anticipated that the air entrainment model will become a permanent feature in future versions of **FLOW-3D**[®].