Water & Wastewater

CFD as a Mixing Application Tool

Steve Saunders

ABSTRACT

Computational Fluid Dynamics, commonly referred to as CFD, has emerged as a viable design tool in the pumping industry over the past decade. During this time, ITT Flygt has successfully employed CFD in the realm of hydraulic end design for both pumps and mixers. Now the focus also includes mixing applications. A typical Flygt mixing application involves a bulk flow generated in a tank by one or more submerged mixers; CFD modeling is a means of predicting what this bulk flow will look like. This ability to predict bulk flow is particularly useful to an applications engineer who needs to determine mixer orientation or to demonstrate effective mixer placement to a customer.

1. INTRODUCTION

Old hands in the world of fluid dynamics were quite pleased with themselves when they came up with the nickname Colorful Fluid Diagrams for CFD. It is true that one can produce dazzling color pictures—sometimes of questionable utility—with CFD, but the gain in insights into flow phenomena and the ease of illustrating them through CFD is now winning over the skeptics. The foothold CFD has gained in a variety of industries speaks for its increasing acceptance as a design tool. Examples include the automobile industry, where it is used under the hood for both combustion and cooling analyses, in the passenger compartment for climate control, and outside the vehicle for aerodynamic shaping. The home appliance industry uses CFD to design vacuum cleaners and the cooling cycles of refrigerators, among other applications.

Mixing has not been exempt from the develoments in CFD as shown below in Figure 1. This figure, supplied by Fluent Inc., is an illustration of the output from MixSim, a program developed for modeling top-entry stirred vessels (Fluent Inc., 1998). In the meantime at ITT Flygt, we have been busy applying CFD to our own type of mixing: a submerged jet, or jets, oriented to produce a bulk flow in the vessel requiring the mixing.

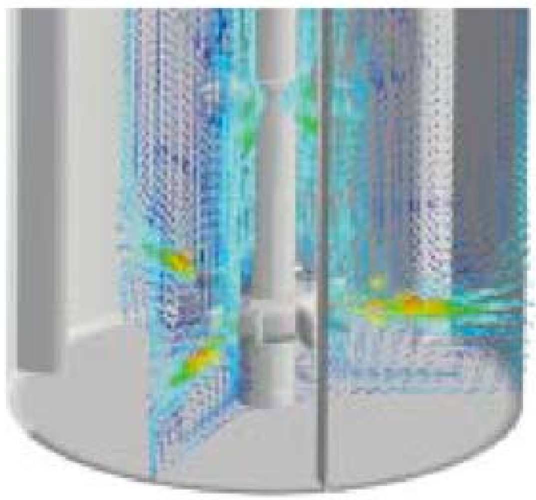

Figure 1. Baffled cylindrical tank stirred by a vertical turbine (bottom) and a pitched-blade turbine (mid-depth). Vectors show flow direction and are colored by velocity magnitude (dark blue for lowest velocity and red for highest velocity).

2 . ESTABLISHING THE DEGREE OF RELIABILITY

Establishing how closely CFD simulations model their real-life prototypes is a primary concern. To this end, two models were built to determine the degree of reliability of CFD modeling within the realm of submerged jet mixing. One model was a Plexiglas tank equipped with a model mixer plus thrust and velocity measurement equipment. The other was its CFD counterpart.

2.1 LAB MODEL

The model used for collecting experimental data is shown in Figure 2. This model mixing vessel was a 1.0-m by 0.7-m rectangular Plexiglas tank filled with water to a depth of 0.5 m. The mixer was a scale model of a production submersible Flygt unit.

The input to the system was measured in Newtons of thrust determined from a load cell incorporated into the mixer mount. Flow data were collected using an ultrasonic velocity probe that recorded velocity components Vx, Vy and Vz at a sampling rate of 2 Hz over an interval of 300 s for each point. Velocity components at each point were then determined by finding the average of the 2-Hz samples taken over this 300-s time interval. The grid pattern of data-point locations (total 350) is illustrated in Figure 2.

2.2. CFD MODEL

The CFD model was prescribed with geometry and boundary conditions simulating the lab model as closely as possible. An RNG k-e turbulence model was applied, and the equations set for a steady-state solution. The mixer was modeled using a momentum source, that is, flow conditions on a face representing the mixer were set in such a way that momentum input would be equal to that measured in the lab model. Tangential velocity at the mixer face was based on laser-Doppler velocimetry data from earlier tests on a similar propeller (Petersson, 1996).

The grid used in the CFD model was a hybrid type, a portion of which is shown in Figure 3. The hybrid-grid type allows for the use of both hexahedral cells (brick-shaped cells with six facets) and tetrahedral cells (tetrahedrons with four triangular facets).

The hybrid-grid type is particularly practical for submerged-jet mixing. From a computationalstandpoint, the best performance is achieved with the grid orthogonal to the dominant flow pattern, particularly in regions of high gradients (Fluent Inc., 1997). It is therefore desirable to have a hexahedral grid in the region of the mixer momentum source and its developing jet. The remainder of the tank volume does not need a hexahedral grid, however, and can be easily filled with tetrahedralcells. The hybrid grid illustrated in Figure 3 is a view of the CFD mesh sectioned through the mixer centerline showing both hexahedral and tetrahedral regions.

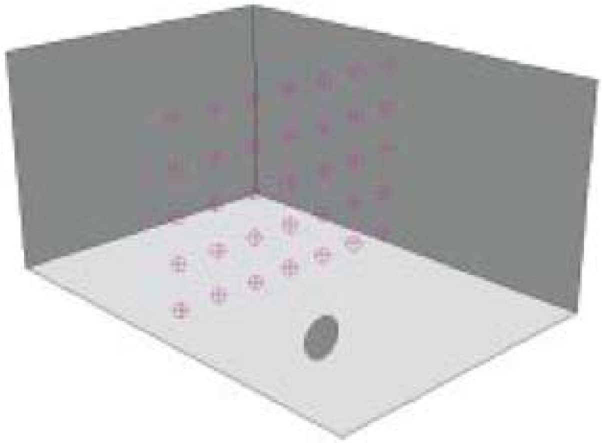

Figure. 2.A Figure 2. A CAD rendering of the model tank showing the velocity measurement points located on a plane at the tank midpoint. Altogether, data were measured on 10 of these planes spaced at equal intervals along the tank length. The mixer is represented by the dark ellipse in the lower near corner.

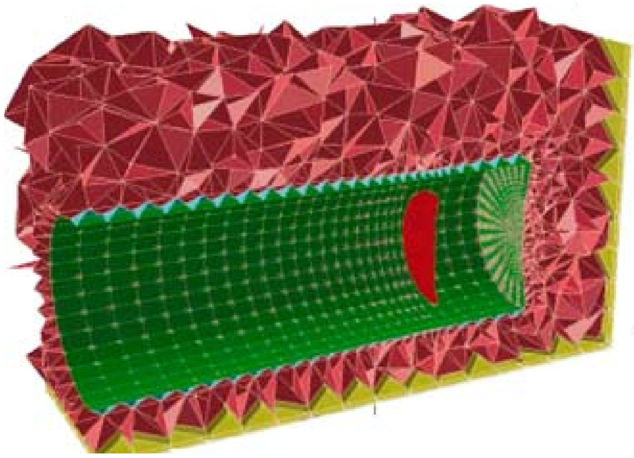

Figure 3. View of the CFD mesh sectioned vertically through the mixer centerline. The tetrahedral cells (magenta) comprise most of the tank volume. The green cylindrical shape is the shell of the hex mesh with the mixer location shown in dark red.

2.3 COMPARISON OF CFD AND MEASURED VELOCITY

A representative sample of velocity component Vx (parallel to the mixer centerline) is shown in Figure 4. In this figure are traverses taken across the center of the tank at three elevations: close to the tank floor at 0.05 m off the bottom, mid-depth at 0.25 m off the bottom and near the water surface at 0.45 m off the bottom. These are the bottom, middle, and top lines of dots shown previously in Figure 2. Agreement between the predicted and measured values is quite good particularly along the mid-depth and near-surface traverses that are in the bulk-flow region. The near-floor traverse, at 0.05 m off the bottom, shows that the CFD solution is predicting different velocities in this region, however, as the traverses pass through dz = 0.1 m, both show an increase in velocity near the centerline of the mixer jet. Here, the lab data points appear to bracket a spike in the velocity predicted by the CFD solution.

Error in the lab measurements was determined by integrating velocity data on each measurement plane (Fahlgren, 1998) and employing mass conservation. Since the model tank is an enclosed volume, the flow crossing each of the measurement planes must be zero. Then any non-zero values for total measured flow across a plane indicate the measurement error (Saunders, 1997, Fahlgren, 1998). In Figure 4, error bars of magnitude ±0.04 m/s have been included on off-bottom data points for 0.25 m. These error bars are representative of all the lab data but have been left off the traverse points of 0.05 and 0.45 m for the purpose of clarity.

2.4 VOLUMETRIC VELOCITY DISTRIBUTION

Curves of the cumulative average velocity magnitude throughout the tank volume are shown in Figure 5. Instead of focusing on individual spatial points, these curves represent the distribution of the velocity data in the entire volume of each model. Interpretation of the distributions is a matter of following the horizontal grid line from a given velocity to its intersection with the curves. For example, in both models, about 40% of the tank volume has a velocity magnitude equal to or greater than 0.1 m/s.

Agreement between the CFD simulation and the

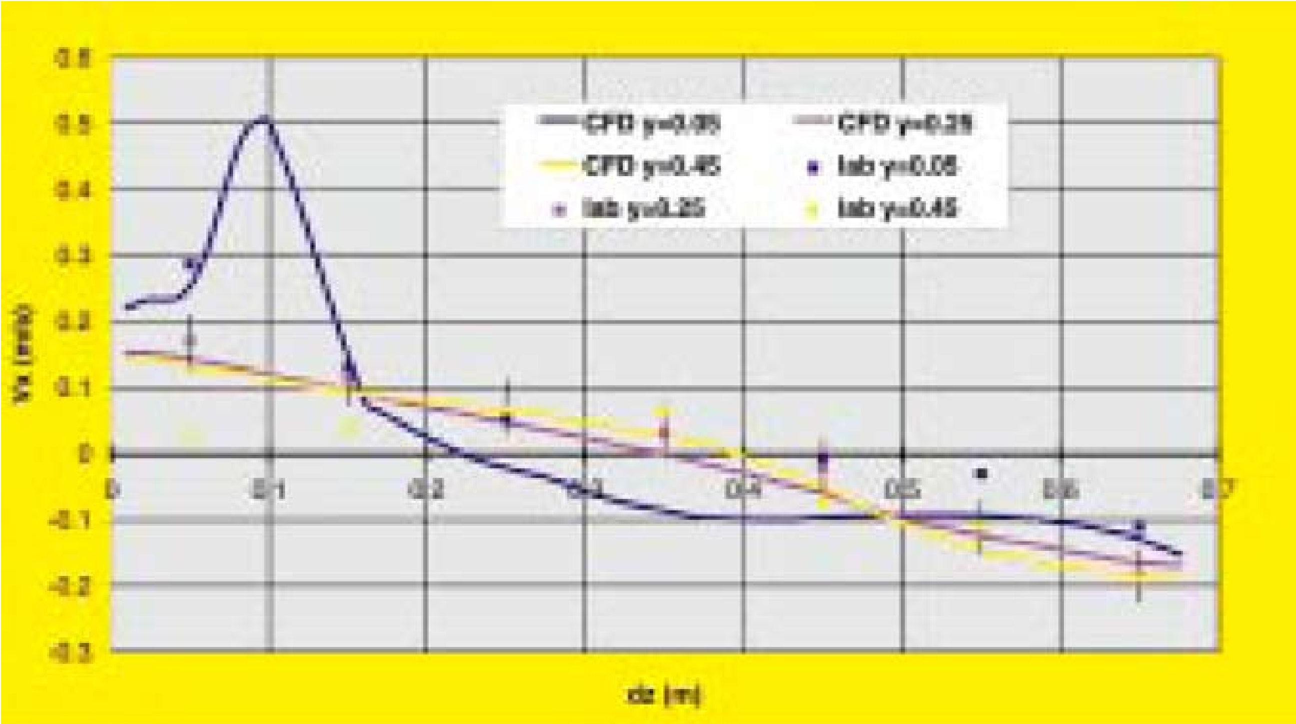

Figure 4. Horizontal traverses of Vx taken across the tank midsection. The square symbols are lab-data points taken at the positions indicated. The solid lines represent the CFD prediction. Error bars on the traverse data points, y=0.25 m represent a V_x measurement error of ±0.04 m/s.

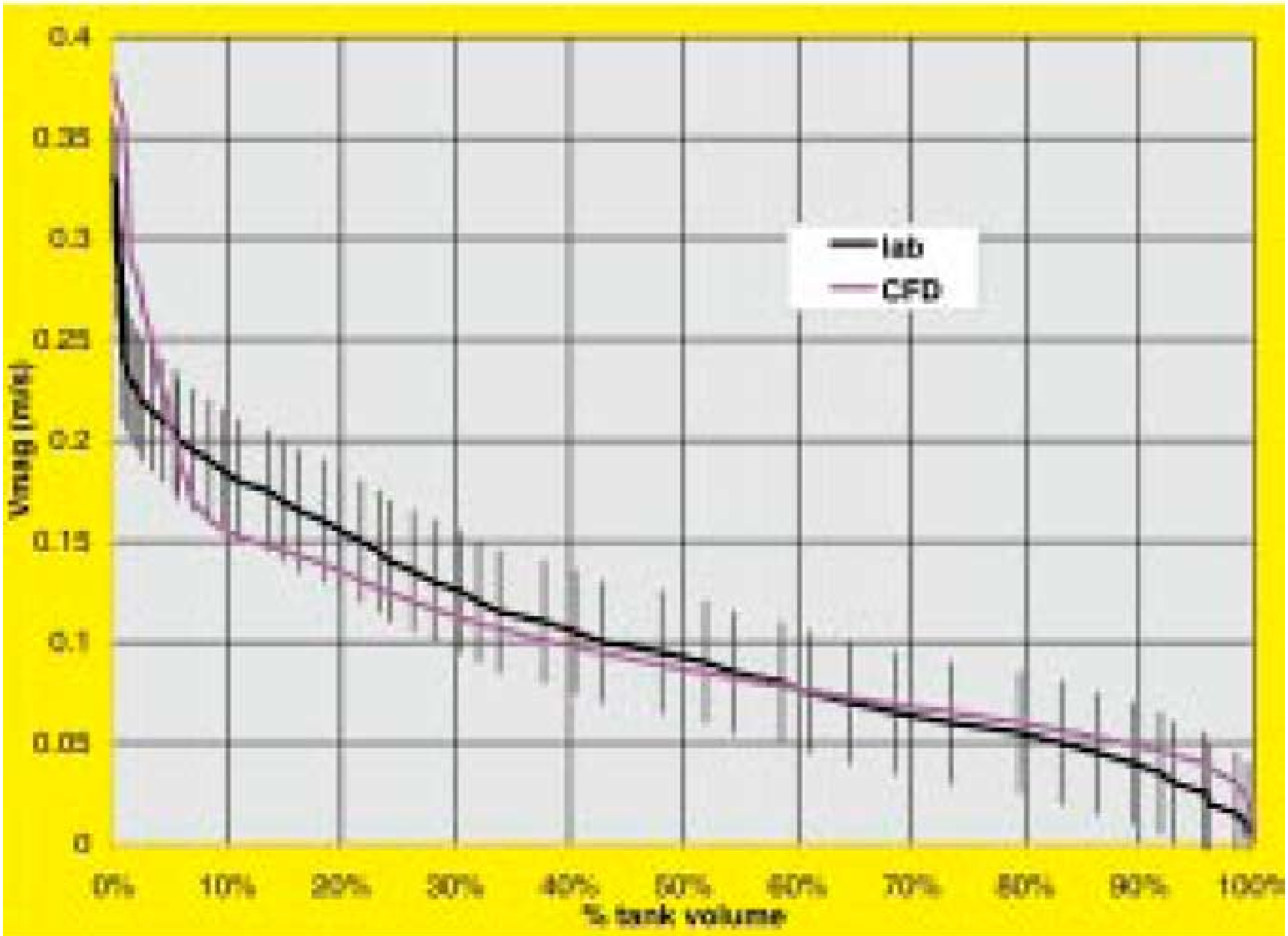

Figure 5. Cumulative distributions of velocity magnitudes for measured data and the CFD simulation. Error bars of magnitude ±0.04 m/s are shown on the lab-data curve.

lab model is excellent. Except for approximately 5% of the tank volume, predicted velocity magnitude falls within the error margin of measured data described in section 2.3. Examining individual data points again showed that the largest discrepancies came from the high gradient region in or near the mixer jet. Data were not collected for several diameters downstream of the mixer propeller in the lab model owing to interference between the velocity probe and the moving propeller blades. Consequently, the highest velocities in the lab model were missed. These missing velocities are most likely the explanation for the discrepancy between the two curves near their intercepts with the y-axis in Figure 5.

2.5 FLOW VISUALIZATION

Post-processing the results obtained from the CFD solution revealed a flow field similar to what had been observed in the lab model using dye and particle-flow-visualization techniques. The jet and bulk-flow patterns are illustrated in Figure 6; a plot of fluid-path lines. Path lines are started from the origin of the mixer jet and allowed to follow paths that the fluid takes as it circulates throughout the tank. The path lines in this plot are shaded according to elapsed time. Here, the color of the lines 20 s. Both the lab and CFD models showed that the swirling flow in the mixer jet dissipated after about five mixer propeller diameters. This dissipating swirl behavior is consistent with laser Doppler data taken from a similar propeller (Petersson, 1996). Unlike the CFD model, however, the flow in the lab model was unsteady. Some vortices would appear and disappear while others would migrate around the tank volume. During the time interval the lab model was observed, these occurrences appeared to be random.

The random behavior in the lab model is considered to be a significant factor behind the error of ±0.04 m/s presented in Sections 2.3 and 2.4. In an attempt to duplicate this random behavior, the CFD simulation was set up and run as an unsteady, time-dependent case. The resulting simulation did not show any flow instability, however. Flow patterns did not change from one time step to the next and were identical to the earlier steady state solution.

2.6 ASSESSMENT OF THE LEVEL OF CFD RELIABILITY

At its current state of development, it is reasonable to assume that a CFD simulation of a submerged jet in a tank will predict a bulk-flow pattern well. In terms of direct comparison of parameters such as velocity, the CFD simulation and collected data differ by as much as 30% in the extreme cases. This is due, in part, to the inherent statistical errors involved in collecting velocity data in a turbulent flow (Fahlgren, 1998). The spacing of the data recording points may have been too great to record regions of high velocity gradients effectively, a point well illustrated by Figures 4 and 5 in the near-jet region.

From the CFD standpoint, some refinements may be necessary in simulating a mixer by using a steady momentum source. An important note, however, is the large portion of the tank volume over which CFD velocity predictions and lab data are in good agreement with each other. The CFD prediction for velocity magnitude lies within the uncertainty envelope of the lab data for approximately 95% of the model volume!

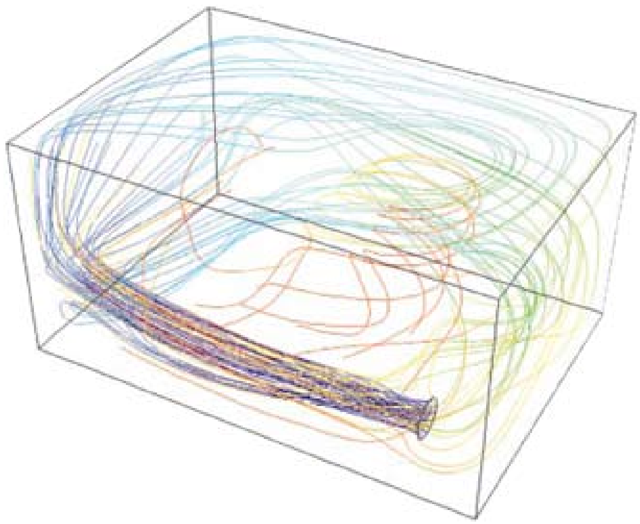

Figure 6. Flow-path lines emanating from the mixer. Path lines are colored by elapsed time starting with blue at t = 0.0 s and turning to red as t approaches 20.0 s. Some flow phenomena made visible by these path lines are the vortex in the tank center and the swirl in the mixer jet that dissipates after about five propeller diameters.

3. CASE STUDY

Having established what can be expected in terms of level of modeling accuracy from CFD, the simulations can be put to work in evaluating several alternative mixerc onfigurations for a given application. At Flygt, the axial thrust produced by a mixer is used as a sizing criterion. This means that a given tank geometry and mixing application will have a mixer axial thrust input level above which the mixing task at hand should be performed adequately. Here is an example of a cylindrical tank having a height-to-width ratio of 1.5:1 for which two equal mixer thrust alternatives have been chosen. In Case 1, shown in Figure 7, a single mixer is positioned near the top of the tank and oriented toward the tank bottom. The Case 2 alternative, shown in Figure 8, has one mixer in the
same location and orientation as in Case 1 and an additional mixer near the tank bottom. The jet from this mixer is oriented parallel to the tank bottom and 30 degrees off tank center. The results of these simulations are compared using two mixing criteria: particle suspension and mixture homogenization.

3.1 PARTICLE SUSPENSION

In a typical suspension case, solid particles having higher specific gravity (SG) than the surrounding fluid will settle to the bottom of the vessel they happen to be in if left undisturbed. One or more mixers can be employed to keep the particles in suspension or re-suspend those that may have settled.

A criterion employed in particle suspension and re-suspension comes from Shields, who, through experimental work on open channel flows, determined the minimum

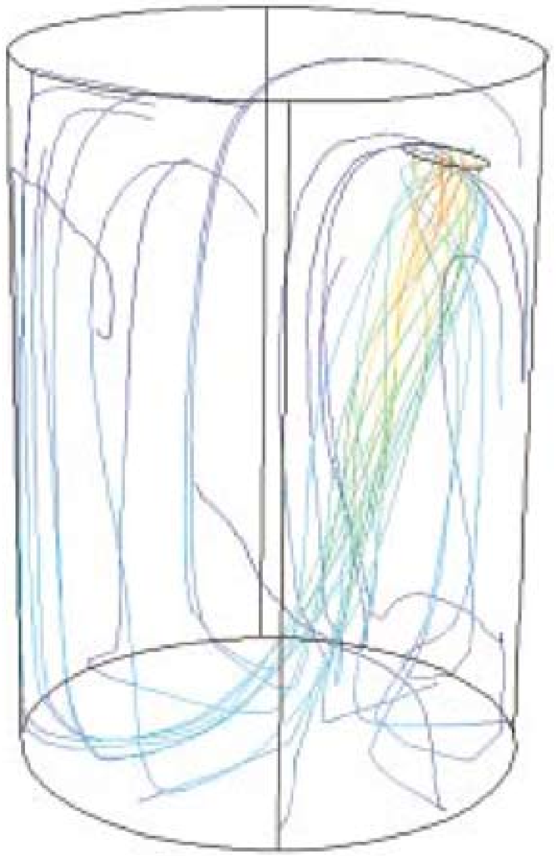

Figure 7. Single mixer in cylindrical tank. The path lines are colored by velocity magnitude (dark blue for lowest velocity and red for highest velocity).

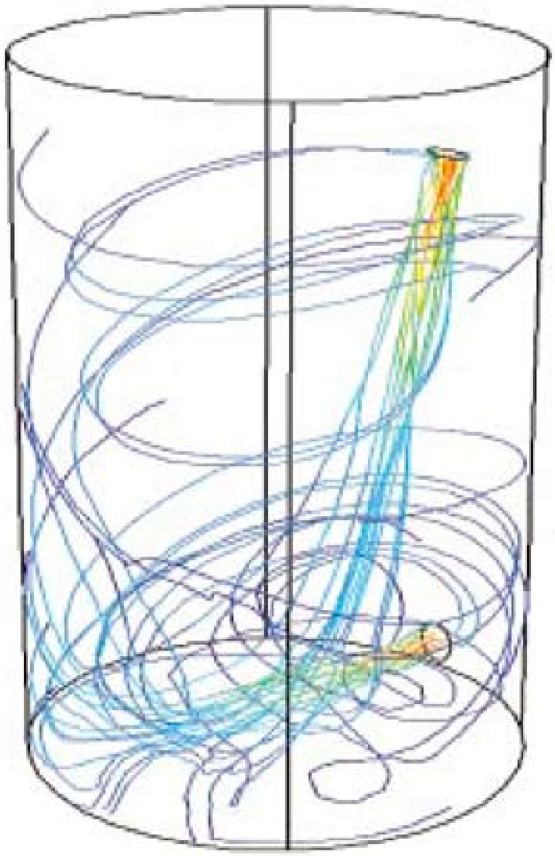

Figure 8. A pair of mixers in a cylindrical tank. The path lines are colored by velocity magnitude (dark blue for lowest velocity and red for highest velocity).

shear stress required to suspend solid particles of given size and density (Shields, 1936). This minimum required shear stress is the shear threshold (τ_l). In this example, the CFD simulations provide a shear stress value (τ) for each of the cell faces on the tank bottom.
These have been normalized against τ_l such that a value of τ/τ_l equal to or greater than 1.0 means the shear stress on that cell face meets or exceeds the Shields criterion. The distribution of normalized shear stress is shown in Figure 9, plotted against percentage of bottom area.

The curves in Figure 9 illustrate that Case 1, the single mixer, is a better arrangement for suspensioning than the mixer pair in Case 2 for this particular example. For Case 2, only 20% of the tank bottom area has shear stress that meets or exceeds the Shields criterion, whereas 50% of the tank bottom area meets or exceeds the criterion in Case 1.

3.2 MIXTURE HOMOGENIZATION

Often, there are mixing applications wherein the requirement is that the solid particle contents of a tank be mixed uniformly throughout the liquid in that tank, or *homogenized*. The distribution of solids throughout the mixed tank is not an easy parameter to measure, and it is therefore not uncommon for customers to specify a velocity threshold (v_t) for a given percentage of tank volume. The assumption is that homogenization should occur if v_t is met, or exceeded, in a sufficiently large enough portion of the tank.

The same simulation solutions from Cases 1 and 2 in the previous section are used to develop distributions of the velocity magnitudes. This time, however, the CFD solution provides a velocity magnitude (v) for each cell in the tank volume and normalizes it against v_t. Shown in Figure 10 are curves for normalized velocity magnitude (v/v_t) versus percentage of tank volume.

Now, from a velocity distribution standpoint, the pair of mixers from Case 2 is performing better than the single mixer of Case 1. The percentage of tank volume that meets v/v_t equal to or greater than 1.0 is approximately 30% for Case 1; for Case 2, this number is close to 70%. If the assumption is correct that a larger portion of the tank meeting or exceeding v_t means a more even distribution of solids, then the mixture in Case 2 should be more homogenized than Case 1.

3.3 CASE STUDY ASSESSMENT

The cylindrical tanks used for Cases 1 and 2 above were identical. The total mixer thrust (or momentum input) was also equal for both. Nevertheless, the CFD simulations dramatically showed that the mixing results for the single unit in Case 1 were better for suspensioning than the two mixers in Case 2, but the opposite was true when the desired mixing result was homogenization. It would have been difficult, time consuming, and expensive to

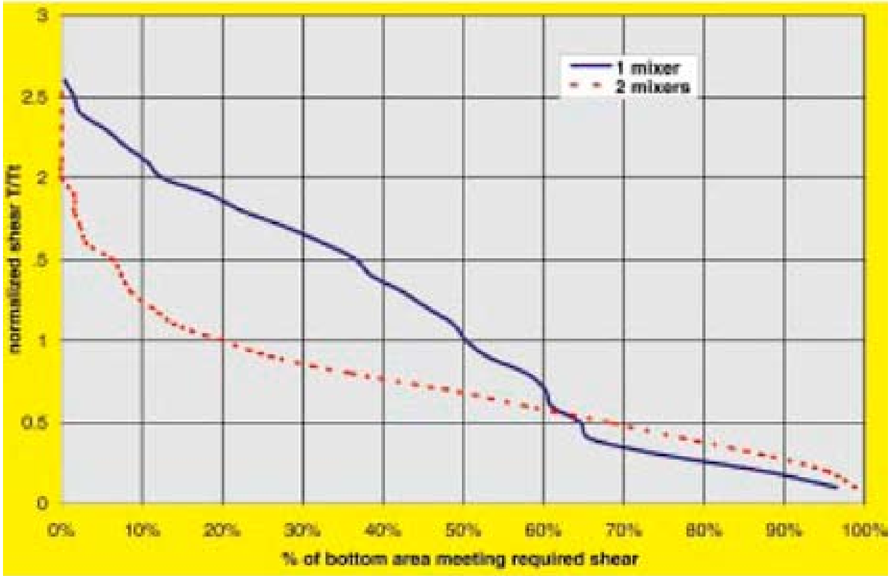

Figure 9. Graph of normalized shear stress versus percentage of bottom area meeting required shear stress.

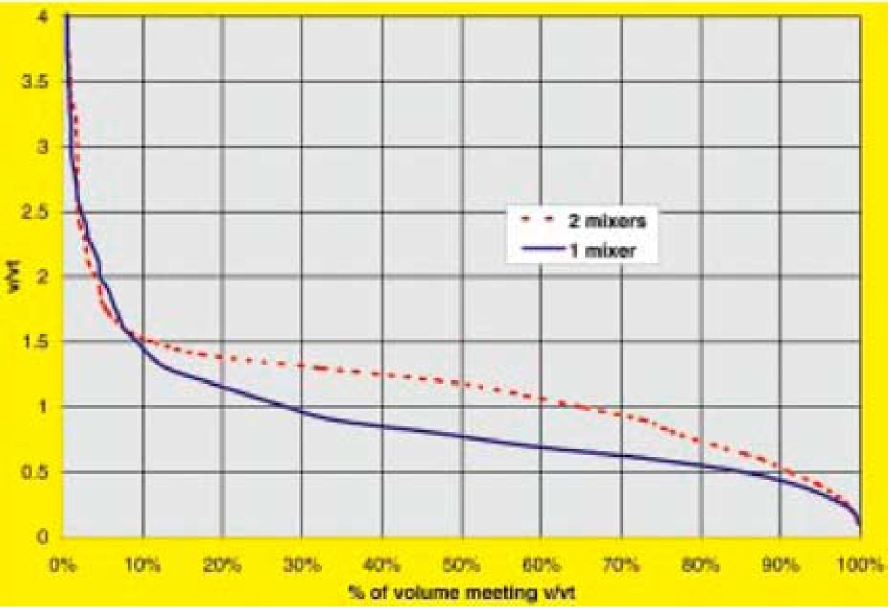

Figure 10. Graph of normalized velocity magnitude versus percentage of tank volume meeting required velocity magnitude.

establish—either on site or in a lab model—which of these two alternatives might be the most suitable for the given application. The total time required to complete both cases, starting with entering the tank-and-mixer geometries and finishing with printing the distribution curves, was about two days with the CFD solver running through both nights.

4. CONCLUSIONS

Computational Fluid Dynamics is proving itself to be a practical tool for analysing submerged-jet mixing applications. In particular, CFD flow-field predictions have sufficient accuracy to provide an engineer with an insight into how jet and bulk flows behave within the volume being mixed. The ability to predict accurately and then interpret flow phenomena means that the designer can work in an informed, systematic manner toward the elusive "optimal mixer configuration" for a particular application.

Article from Scientific Impeller, 1998

Reference

1. Fahlgren, M. (1998)
Velocity Measurements in Mixer Applications. ITT Flygt AB, Solna, Sweden.
2. Fluent Incorporated. (1996) *MixSim User's Guide*. Fluent Incorporated, Lebanon, NH.
3. Fluent Incorporated. (1997) *Tgrid 3 User's Guide.* Fluent Incorporated, Lebanon, NH.
4. Petersson, P. (1996) *Laser Doppler Velocity Measurements in an Impeller-Generated Turbulent Jet.* Department of Water Resources Engineering, Lund University, Sweden
5. Saunders, S. (1997) *Submerged Jet in a Rectangular Tank: A Comparison Between a Fluent and a Lab-Scale Model Fluent Users Group Meeting*. Fluent Incorporated, Lebanon, NH.
6. Shields, A. (1936) *Anwendung der Ählichkeitsmechanik und der Turbulenzforschung auf die Geschiebebewegung.* Hydrodynamics Laboratory Publication No. 167, California Inst. of Tech., Pasadena, CA.