Experimental Data Confirms CFD Models of Mixer Performance

The Problem

Over the years, manufacturers of mixing systems have presented computational fluid dynamics (CFD) calculations to claim that their technology can achieve adequate mixing in water storage tanks and to estimate pumping rates for their equipment. CFD is a powerful technique for estimating fluid dynamics within water tanks (and for generating lots of colorful diagrams and charts). However CFD experts caution that any CFD calculation should be verified and calibrated by experimental data.

In 2009, scientists at PAX Water Technologies, Stanford University, and operators at Redwood City Water (Redwood City, CA) collaborated on an ambitious project to obtain high-quality experimental data of the mixing performance of the PAX Water Mixer inside a 500,000 gallon water storage tank. These data were then compared to CFD models developed independently by PAX Water and Stanford to determine the real pumping rates for the PAX system. With these calibrated CFD models, other mixing technologies can also be evaluated on an apples-to-apples basis.

Tank-scale Measurements

The most reliable means of evaluating an active mixer is to study its performance in a full-scale water storage tank. The easiest approach is to measure the time required to take a thermally stratified tank to a condition of uniform temperature. Submersible temperature sensors can be suspended at various points within a water tank and temperature readings can be recorded as the mixer is turned on and allowed to operate. Disinfectant concentrations as a function of depth can also be measured to determine mixing.

The 500,000 gallon Cambridge Street water tank in Redwood City, CA, which has had a PAX Water Mixer since the summer of 2006, was selected for study. Temperature probes were suspended on a cable inside the tank at various depths within the water and temperatures were recorded every 10 minutes over the course of a month. The mixer was left off for 3-7 days to allow for thermal stratification to develop. Then, the mixer was turned on and the probes measured the time required to mix the tank at various depths. This procedure was repeated 10 times.

Figure 1 shows the results of a typical thermal de-stratification experiment. Before starting the mixer (at \( t=0 \) hour), there was an initial temperature gradient of about 5 degree Celsius between water at the bottom and the top of the tank. After starting the mixer, cold water from the bottom of the tank was transported to the top of the tank and the temperature in the topmost water began to fall. After 4 hours, the entire tank was thermally homogeneous, indicating a fully mixed condition.
Experimental Data Confirms CFD Models of Mixer Performance

Tank-scale Measurements

Figure 1: Measured thermal de-stratification of a 500,000 gallon ground storage tank

CFD Model

Extensive research efforts have gone into predicting the hydrodynamics of agitated tanks in a variety of industries using CFD (Paul, Atiemo-Obeng, & Kresta, 2004). In spite of all these efforts, the simulation of agitated tanks is still complex and challenging, especially at the industrial scale. In particular, a suitable strategy is required to handle the transient geometry due to the impeller rotation and the turbulence involved in an agitated tank. Therefore, mathematical simplifications are commonly required to conduct these simulations, leading inevitably to approximation of the real physical phenomenon.

In addition to these general difficulties in modeling agitated tanks, the simulation of the mixing impeller in this study (the Lily impeller manufactured by PAX Water Technologies, Inc.) in a water storage tank poses several additional challenges because of the very large size of the tank and the complex geometry of the small impeller (Figure 2).

Figure 2: A CFD model of active mixing in a large storage tank. The complex geometry of the Lily impeller and large differences in length and velocity scales make this calculation extremely challenging.
Experimental Data Confirms CFD Models of Mixer Performance

CFD Model

Because of the large volume of water involved in storage tanks (typically several MG), the CFD simulation requires the resolution of several million or even billion equations. These calculations require several days of dedicated calculation time on high-end parallel supercomputers (our study utilized a 128 parallel processor cluster).

The CFD simulation of active mixing requires careful attention in the choice of the mesh generator and the solver. The large difference in the velocity scale prevailing in the tank is also an important challenge to address. Close to the impeller, the velocity of the water is high (about 8 m/s) while far from the mixer the water velocity is relatively slow (the bulk average velocity is typically 1 cm/s or less). Therefore, 1% error in the maximum velocity predicted by the CFD solver lies in the same magnitude as the bulk average velocity in the tank. In that context, the numerical techniques used to solve the underlying mathematical problem have to be particularly robust to provide a reasonable prediction of the hydrodynamics inside the tank. Finally, the CFD software has to generate meshes capable of handling complex geometries with very large differences in length scale.

In our models, the Navier-Stokes equations are solved using AcuSolve, a commercial software package developed by ACUSIM (now Altair). This software is used in a wide variety of applications and industries (Corson, 2009). The solver is based on the finite element method and its numerical algorithms are specifically designed for parallel execution on multiprocessor machines. AcuSolve is also designed to address models with large differences in velocity scales and offers numerical strategies suitable to model complex geometries with moving parts, such as an impeller rotating in a tank.

In order to better simulate the action of the mixer at the full scale of a water tank, we conducted a series of experiments measuring the pumping power of a scaled-down version of the mixer in a 200-gallon lab-scale tank. We used an Acoustic Doppler Velocimeter (ADV) (model Vectrino from Nortek) to measure the velocity created by the impeller in various directions away from the spinning impeller. The ADV was mounted on a rail system allowing us to precisely change the position where the measurements were taken.

At the same time, we simulated this experimental set-up using the CFD model presented in the previous section and compared the calculated velocities to those we measured in the tank. Our initial CFD mesh consisted of 6.8 million nodes and 39 million tetrahedral elements. A finer mesh size was used near the impeller where larger velocity gradients were expected. This CFD simulation required about 7 days of calculation on a 16 processor computer.
An example of the velocity magnitude calculated using CFD is shown in Figure 3. The highest velocities computed are several meters per second and correspond to the axial jet close to the mixer. However, most of the rest of the water moves at much slower velocities. Notice that Figure 3 uses a threshold value of 20 cm/s (in red) for the maximum velocity magnitude. This reveals the large difference in velocity magnitude prevailing in this tank and also illustrates the challenge of visualizing and interpreting the simulation results.

Figure 3: Velocity magnitude of the Lily impeller in a 200 gallon tank (m/s).

The axial velocity generated by the impeller along the centerline in line with the horizontal shaft was used to compare experimental measurements to those predicted by the CFD model. Moreover three meshes were generated by refining the mesh sizes in the entire volume of the tank in order to study the sensitivity to the mesh size on the axial velocity profiles (Table 1).

<table>
<thead>
<tr>
<th>Case</th>
<th>Number of nodes (millions)</th>
<th>Number of elements (millions)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>3.7</td>
<td>21</td>
</tr>
<tr>
<td>B</td>
<td>6.8</td>
<td>39</td>
</tr>
<tr>
<td>C</td>
<td>8.8</td>
<td>51</td>
</tr>
</tbody>
</table>

Table 1: Mesh information for the 200 gallon tank CFD models

The CFD predictions of axial velocity are compared to those from ADV measurements in Figure 4. Results indicate the calculations from the two finer meshes are in good agreement with the experimental data, while the results provided by the coarse mesh slightly underestimate the axial velocity.

Our results show that our CFD model accurately predicts the axial jet velocity generated by the impeller using a high-fidelity mesh. The nearly identical results obtained with the two finer meshes shows that the model is insensitive to the mesh if a sufficiently fine mesh size is employed. As expected, it confirms the critical importance of using an appropriate mesh size to accurately predict the velocity field in the tank.
The mesh sensitivity analysis also suggests that a coarse mesh tends to underestimate the true velocity and the intensity of the flow created in the tank. Therefore, our CFD model of full-scale tanks is likely to be conservative: we expect the performance of the PAX mixer in a full-scale tank to be equal or greater than the results predicted by the CFD simulations.

These observations of the mesh sensitivity are particularly important because scaling-up of these high-fidelity meshes used for the 200 gallon tank to a tank several million gallons in size is impossible – we must use a coarser mesh for modeling full-scale tanks. Nevertheless, the CFD model is conservative and the use of a coarser mesh for large storage tanks should generate conservative estimates of the performance of the mixer in full-scale water tanks.

Examples of the velocity vector field computed for the 500,000 gallon Cambridge tank are shown in Figure 5. The velocity vectors illustrate the typical flow pattern produced by this impeller in a large storage tank. A strong axial jet is generated by the mixer and pushes the water from the bottom to the top of the tank and then the water re-circulates down by the sides of the tank. A secondary flow pattern located in the region between the axial jet and the tank boundaries is also noticeable.

The highest velocities computed by CFD are about 8 m/s and corresponds to the axial jet close the mixer while most of the rest of the water moves at slower velocities. Figure 5-A,B use a threshold value of 5 cm/s for the maximum velocity magnitude to help distinguish the presence of the strong axial jet above the mixer (in red).
One goal of our CFD simulations is to determine whether complete circulation of the water is achieved by the mixer. Figure 5-C uses a threshold value of 1 cm/s and shows that almost all the water is experiencing a velocity higher or equal to 1 cm/s. The mixer generates flow circulation through the entire volume of water in the tank and there is no dead zone where the water is stagnant in the tank. This is in agreement with the experimental observations gathered for this tank. An example of the streamlines predicted by the CFD model is shown in Figure 6, showing the circulation of the water in the entire volume of the tank.

Figure 5: Velocity vectors computed for 200,000 gallons of water in the Cambridge Street tank. Cross-section in the middle of the tank. A) 3D view, B) & C) Side views.
Experimental Data Confirms CFD Models of Mixer Performance

Results: CFD Matches Experiment

Non-isothermal cases of the CFD were run to compare the predicted de-stratification results to those obtained from the Cambridge Street tank. Figure 7 summarizes the results. Our CFD model matches the experimental results closely and shows the same general features in the experimental data.

Figure 6: Streamlines computed for the Redwood City Cambridge tank (200,000 gallons of water)

Figure 7. Experimental data (top) compared to CFD results (bottom) for de-stratification of the Cambridge tank are in close agreement.
Having calibrated our CFD model with data from the Cambridge Street tank, we can apply it to other tank sizes and shapes, in particular elevated tanks and large reservoirs.

Figure 8 presents the velocity vectors computed for a 1 million gallon elevated tank. For that tank, the mixer is not located in the center because of the presence of a central inlet riser pipe, which is common in elevated tanks. Results indicate a somewhat different flow pattern when the mixer is off-centered. In that configuration, the jet produced by the impeller still pumps the water to the top of the tank, but the asymmetry changes the circulation pattern compared to cylindrical ground storage tanks. Nevertheless, the CFD prediction suggests that the mixer achieves complete circulation of the water in the tank.
Experimental Data Confirms CFD Models of Mixer Performance

Figure 9 shows the CFD model of a 2,750,000 gallon reservoir similar to one at Ontario City in California (See our case study entitled Mixing Eliminates Stratification and Boosts Chlorine Residual in Square In-Ground Concrete Tank). Field measurements of the mixer performance were gathered for this reservoir showing the thermal de-stratification achieved by the active mixer. Despite the presence of 36 supporting columns inside the reservoir, the velocity vectors computed by the CFD indicate that the mixer is able to completely circulate the water in the reservoir without creating any significant stagnant zones.

For the case of large reservoirs with supporting columns and other obstructions, our CFD model is particularly useful in determining the optimal position and number of mixers to eliminate dead zones and to ensure complete mixing.

Figure 9: Velocity vectors for Ontario reservoir.

Conclusion

These results show that accurate CFD modeling of active mixers in water storage tanks is possible. CFD results that are not calibrated with experimental data cannot be used to reliably predict mixer performance. Customers who are evaluating different mixing technologies for water storage tanks should always insist on calibrated CFD data and not just the verbal assurances of equipment manufacturers!