CFD Modeling of Activated Sludge Mixing and Sedimentation

Randal W. Samstag, P.E., B.C.E.E
Principal Technologist
Ed Wicklein, P.E.
Senior Technologist

Computational Fluid Dynamic Modeling (CFD) is a method of analyzing and solving fluid flow problems using numerical methods:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0
\]

\[
\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla P + \nabla \cdot \sigma + \mathbf{F}.
\]

\[v = C \frac{1}{\varepsilon}
\]

CFD is well established for analysis of hydraulic components and flow splitting.

Carollo has long history of CFD analyses for fluid problems:
- Pump Intake Hydraulics
- UV Disinfection
- Flow Split
- Clearwell Performance

CFD is increasingly used for analysis of processes:
- Solids Transport
- Vesilind settling
- Density couple
- Viscosity impact
- Ongoing Enhancement
  - Multi-component sedimentation rates
  - Flocculation model

Carollo has Incorporated Solids Transport into Flow Solution by User Defined Function (UDF):

Current Model features:
- Solids Transport
- Vesilind settling
- Density couple
- Viscosity impact
- Ongoing Enhancement
  - Multi-component sedimentation rates
  - Flocculation model
Future Model Enhancements to Include More Chemical and Biological Interaction

- Biokinetic Models such as the ASM Models
- Advanced oxidation Models
- Disinfection models

Case Studies Using Solids Transport Modeling

1. Clarifier Inlet Comparison for Sedimentation
2. Activated Sludge Mixing in a SBR
3. Activated Sludge Mixing in an Oxidation Ditch Process Conversion

Case Study 1
Clarifier Inlet Comparison

Clarifier Inlet Comparison Existing Configuration

Alternative Inlet Configurations

Alternative Velocity Plans
Alternative Velocity Profiles

- Target Baffle
- Baffled EDI
- Tangential Inlet
- Concentric Tub EDI

Comparison Solids Profiles

- Target Baffle: 30 mg/l at 11.5 feet
- Baffled EDI: 30 mg/l at 11.5 feet
- Tangential Inlet: 30 mg/l at 13.2 feet
- Concentric Tub EDI: 30 mg/l at 10.9 feet

Case Study 2
Jet mixing and aeration in a sequencing batch reactor (SBR)

Jet nozzles for mixing pump suction outlets
Air modeled with multiphase model
Solids transport, settling, and density impact modeled by UDF

Min / Max Deviations from Average

<table>
<thead>
<tr>
<th>Layer</th>
<th>Average TSS Concentration (mg/L)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top</td>
<td>1.208 1.404 2.192 2.155</td>
</tr>
<tr>
<td>2</td>
<td>2.385 2.331 2.285</td>
</tr>
<tr>
<td>3</td>
<td>2.519 2.374 2.302 2.308</td>
</tr>
<tr>
<td>4</td>
<td>2.536 2.422 2.448 2.387</td>
</tr>
<tr>
<td>5</td>
<td>2.554 2.518 2.526 2.443</td>
</tr>
<tr>
<td>6</td>
<td>2.504 2.620 2.511 2.456</td>
</tr>
<tr>
<td>Bottom</td>
<td>3.006 2.606 2.559 2.500</td>
</tr>
<tr>
<td>Average</td>
<td>2.402 2.353 2.302 2.282</td>
</tr>
<tr>
<td>CoV (%)</td>
<td>50% 45% 12% 9%</td>
</tr>
</tbody>
</table>
**Comparison of Power Levels at Different Jet Velocities**

<table>
<thead>
<tr>
<th>Jet Velocity</th>
<th>Mix Criterion</th>
<th>Power Level (hp/MG)</th>
<th>Power Level (W/m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.5 m/sec jet velocity</td>
<td>50% Max Deviation</td>
<td>39</td>
<td>7.7</td>
</tr>
<tr>
<td>3.0 m/sec jet velocity</td>
<td>40% Max Deviation</td>
<td>66</td>
<td>13.0</td>
</tr>
<tr>
<td>3.5 m/sec jet velocity</td>
<td>12% Max Deviation</td>
<td>105</td>
<td>20.7</td>
</tr>
<tr>
<td>4.0 m/sec jet velocity</td>
<td>&lt; 10% Max Deviation</td>
<td>156</td>
<td>30.8</td>
</tr>
</tbody>
</table>

To meet a 10 percent deviation criterion would require four times more power than currently installed.

**Frequent CFD practice for mixing is to assume neutral density.**

The influence of solids on the velocity pattern is ignored.

A velocity profile is then calculated assuming clear water.

It is then assumed that a given minimum velocity (2.5 ft/min) will be sufficient to provide mixing.

But it is SOLIDS that we are trying to mix. They aren’t typically modeled.

**Comparison of density-coupled and neutral density simulations**

**Density-coupled**

Solids transport model calculates the local solids concentration based on flow regime.

The influence of the local solids concentration on the local density is then iteratively calculated.

This approach was verified by the field solids profile test data.

**Neutral density**

Neutral density simulation dramatically over-predicts the degree of mixing.

**Case Study 3**

Mechanical and air mixing in an oxidation ditch process conversion

Ditch converted to once through reactor
Walls added to isolate Zones
Anoxic and Zone 1 have mechanical mixers
Fine bubble aeration in Zones 1 to 4
Two mixer types compared, MRF model used for rotating components
15 Hp Submersible
Horizontal Propeller
10 Hp Vertical
Hyperboloid

Horizontal mixers generate high velocity along mixer axis and sides

Keeping solids in suspension

Vertical mixers generate high velocity near bottom

Keeping solids swept up

Solids mixing results showed similar coefficient of variation for both types of mixer

<table>
<thead>
<tr>
<th>Mixer</th>
<th>Anoxic Zone</th>
<th>Zone 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horizontal</td>
<td>1.9</td>
<td>1.5</td>
</tr>
<tr>
<td>Vertical</td>
<td>2.2</td>
<td>1.5</td>
</tr>
</tbody>
</table>
Vertical mixers needed less power for this facility

<table>
<thead>
<tr>
<th>Mixer</th>
<th>Horizontal</th>
<th>Vertical</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>7.8</td>
<td>4.5</td>
</tr>
<tr>
<td>2</td>
<td>7.8</td>
<td>4.6</td>
</tr>
<tr>
<td>3</td>
<td>9.4</td>
<td>4.6</td>
</tr>
<tr>
<td>4</td>
<td>9.5</td>
<td>6.7</td>
</tr>
<tr>
<td>5</td>
<td>6.6</td>
<td>6.8</td>
</tr>
<tr>
<td>Total</td>
<td>34.8</td>
<td>27.0</td>
</tr>
</tbody>
</table>

Air introduced on flat sections of the bottom creates rolling pattern over sloped sides

<table>
<thead>
<tr>
<th>Zone</th>
<th>Air Flow (cfm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1285</td>
</tr>
<tr>
<td>2</td>
<td>812</td>
</tr>
<tr>
<td>3</td>
<td>560</td>
</tr>
<tr>
<td>4</td>
<td>200</td>
</tr>
</tbody>
</table>

The air induced rolling pattern carries through the 180 degree bend

Aerated zones have relatively high velocity, keeping everything well mixed

Comparison of Power Levels to Other Mixing Devices

<table>
<thead>
<tr>
<th>Mixer</th>
<th>Reference</th>
<th>Max. Deviation</th>
<th>Power Level (hp/MG)</th>
<th>Power Level (W/m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.0 m/sec jet</td>
<td>This study</td>
<td>≤15% Max Deviation</td>
<td>156</td>
<td>30.8</td>
</tr>
<tr>
<td>Large Propeller in Racetrack Carollo Field Visit</td>
<td>Little MLSS separation</td>
<td>5</td>
<td>~1</td>
<td></td>
</tr>
<tr>
<td>Surface Mixing Impeller</td>
<td>Centrotest Witnessed Test</td>
<td>0.8 m/sec (2 fps) bottom velocity</td>
<td>39</td>
<td>7.6</td>
</tr>
<tr>
<td>Hydrofoil mixer</td>
<td>Otun et al. (2009)</td>
<td>&lt; 30% Max Deviation</td>
<td>39</td>
<td>7.6</td>
</tr>
<tr>
<td>Hyperboloid mixer</td>
<td>Otun et al. (2009)</td>
<td>&lt; 11% Max Deviation</td>
<td>20</td>
<td>4.0</td>
</tr>
</tbody>
</table>

Summary

CFD is well established tool for analysis of hydraulic components.
Growing appreciation that CFD can be a powerful tool for analysis of the impact of geometry and hydrodynamics on process performance.
Results from studies of radial flow clarifier inlets, and activated sludge mixing show that CFD can establish important conclusions for process engineering.
Questions?

- ewicklein@carollo.com - 206-538-5166
- rsamstag@carollo.com - 206-851-0094