CFD Modeling of Mixing Facilities

Steve Saunders
IBIS Group Inc.
Passive Mixing Systems

static mixer in an inflow pipe

injection lances

degree of mixing downstream of injection lances
Active (Mechanical) Mixing Systems

top entry impeller

submersible mixers

pumped jet mixers
CFD Simulation for Mechanical Mixing Systems

• In the past 20 years, CFD has emerged as an attractive choice for simulating systems that employ mechanical mixing.

• This is due to two elements in physical modeling that can drive up its cost.
  • Scaling laws are difficult to satisfy with mechanical mixers.
  • Through-flow models may require special water use permitting.
CFD Simulation for Mechanical Mixing Systems

Holy Trinity of Physical Modeling

Osborne Reynolds
Re: ratio between inertial and viscous forces
\[ Re = \frac{\rho vl}{\mu} \]
• rotating elements
• internal flows

Moritz Weber
We: ratio between inertial and surface tension forces
\[ We = \frac{\rho v^3 l}{\sigma} \]
• weir flows
• surface penetrating screens/filters

William Froude
Fr: ratio between inertial and gravitational forces
\[ Fr = \frac{v}{\sqrt{gl}} \]
• open channel flows
• pump sumps

gl
vFr
\[ \mu \]
\[ \rho \]
\[ \sigma \]
CFD Simulation for Mechanical Mixing Systems

• Scaling laws
  • Re, Fr and We based laws cannot be satisfied simultaneously.
  • Model mixer impeller scaling is based on Re and is rarely smaller than 4 to 1
  • Model basin scaling is based on Fr and can be as much as 10 or 20 to 1 depending on available floor space.

• Through flow models
  • No recirculation means high water consumption and large quantities of water discharging into the local sewer system
  • Receiving utility may take issue with tracer chemicals used
CFD Simulation for Mechanical Mixing Systems

Meshing

• Mixer jets are round in cross section, so using unstructured tetrahedral or polyhedral mesh is inevitable.

• If possible, use a hybrid mesh with hex cells orthogonal to the flow where steep gradients occur.
Rigid lid water surface boundary condition is adequate for most active mixing applications.

A symmetry plane best represents a planar air/water interface.

The symmetry plane acts like a wall in that no mass may pass through it, but it is frictionless.
CFD Simulation for Mechanical Mixing Systems

Boundary Conditions – Propeller Mixer Input

Sliding mesh (SM) – transient model with an accurate representation of the propeller rotating incrementally with each computational time step.

Multiple reference frame (MRF) – steady state model with an accurate representation of the propeller “frozen” in time.

Multiple reference frame with mixing plane (MRF+MP) – same steady state model with an accurate representation of the propeller as MRF, but with a mixing plane to blend out individual blade wakes.

Momentum source model (MSM) – steady state model with the impeller replaced by a “puck” shape. The fluid domain within the puck accelerates the flow and gives it the axial, radial and tangential velocity profiles produced by the rotating propeller.

Börgesson, T. Fahlgren, M. Mechanics of Fluids in Mixing, 2001
What is a mixing plane?

Developed by turbo machinery modelers who wanted to analyze individual stator/rotor passages.

Blends out wakes of individual blades by circumferentially averaging flow properties.

Facilitates the use of small steady state models for rapid optimization of blade shapes.
CFD Simulation for Mechanical Mixing Systems

Boundary Conditions – Propeller Mixer Input

Sliding mesh (SM) – best approximation of the real physics, but computationally burdensome.

Multiple reference frame (MRF) – a popular method recommended by Fluent among other CFD software vendors, but test cases performed by Börgesson and Fahlgren showed poor reproduction of the mixer jet.

Multiple reference frame with mixing plane (MRF+MP) – significant improvement to mixer jet simulation, however Börgesson and Fahlgren observed computational instability in cross current applications.

Momentum source model (MSM) – good reproduction of the mixer jet and computationally efficient. The downside is the user must obtain velocity profile data.

CFD Simulation for Mechanical Mixing Systems

Boundary Conditions – Mixer Input

Evaluation of the Momentum Source Model Against Data Collected in the Field

Velocity measurements taken by means of ultrasonic probe from a full scale denitrification basin compare quite favorably with the CFD simulation results.

Cylindrical denitrification tank 17.9m dia. filled to 4.3m depth and equipped with a 2.5m dia. Low speed mixer

CFD Simulation for Mechanical Mixing Systems

Simulation Parameters

**Solids** - Aside from identifying slow moving regions where solids may settle and accumulate, many clients do not include treatment of suspended solids in their CFD modeling scope. There is, however, a caveat…

Studies of mixing systems running with clear water will predict higher mixing efficacy than the same systems with suspended solids included in their simulation parameters.

If suspended solids content is known, their presence should be simulated by means of a user defined function that includes density coupling.

**Turbulence** - For free shear flows, that is, jets acting in tanks or basins, the Realizable K-Epsilon equations have been demonstrated to be robust and reliable.
The real world basin has 7 or 8 (depending on brand selected) top entry mixers.

Scale model was run without mixers. The purpose of the concurrent physical and CFD models was to verify the CFD simulation was replicating the flow splits among the 10 aeration lanes.
Hybrid mesh with structured hex in the cylinder and unstructured tet in the propeller domain.

Tops are pressure inlet boundaries.

Cylinder sides are pressure outlet boundaries.

Floor boundaries are walls with no slip.

Mixer propeller A

Mixer propeller B
Case Study: Common Anoxic Basin Feeding 10 Parallel Aeration Lanes

Frozen in time snapshot of sliding mesh model

Steady state momentum source model s

Rotating impellers replicated by momentum sources
Case Study: Common Anoxic Basin Feeding 10 Parallel Aeration Lanes
Looking at the Results

Mixing Efficacy Observed with Streakline Plots

Streaklines show bulk flow patterns and accelerations

With pressure, velocity and turbulence equations turned off, a time dependent simulation is run wherein a tracer (non-reacting scalar) is allowed progress through the flow continuum.

Qualitative results can be shown with the graphic output that CFD produces.

Quantitative results can be recorded by applying some statistics to the data base.
Case Study: Common Anoxic Basin Feeding 10 Parallel Aeration Lanes
Looking at the Results

Hydraulic Efficiency

\[ T_{\text{TDT}} \] - (theoretical detention time) \( \frac{\text{system volume}}{\text{inflow rate}} \)

1186 seconds for this case

\[ T_{10} \] - Time elapsed between release of tracer and 10% of steady state concentration recorded at outlet

\[ T_{90} \] - Time elapsed between release of tracer and 90% of steady state concentration recorded at outlet

Baffle Factor: \( BF = \frac{T_{10}}{T_{\text{TDT}}} \) indicates degree of short circuiting
\( BF_{\text{mix.A}} = 0.37 \quad BF_{\text{mix.B}} = 0.33 \quad \) Values 0.3 and lower indicate significant short circuiting is occurring.

Morrel Number: \( M = \frac{T_{90}}{T_{10}} \) indicates presence of long term recirculation zones
\( M_{\text{mix.A}} = 5.68 \quad M_{\text{mix.B}} = 5.01 \quad \) Values greater than 5.0 indicate some flow is getting hung up in recirculation zones.
Case Study: Common Anoxic Basin Feeding 10 Parallel Aeration Lanes

Looking at the Results

Velocity Distribution

Histogram of velocity distribution within the common mixing basin
*N.B: Cell size must be accounted for in histogram generation.*

Cumulative distribution of velocity within the common mixing basin
Questions or Comments?

Steve Saunders IBIS Group Inc.
ibisgroup@bellsouth.net

IBIS Group Inc.
http://www.ibisgroupcfd.com/