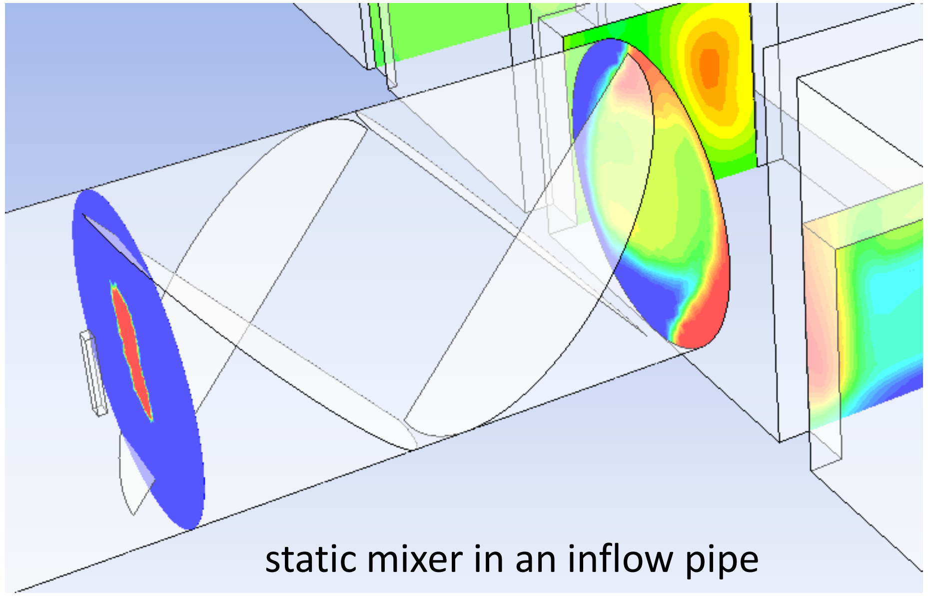
CFD Modeling of Mixing Facilities

Steve Saunders

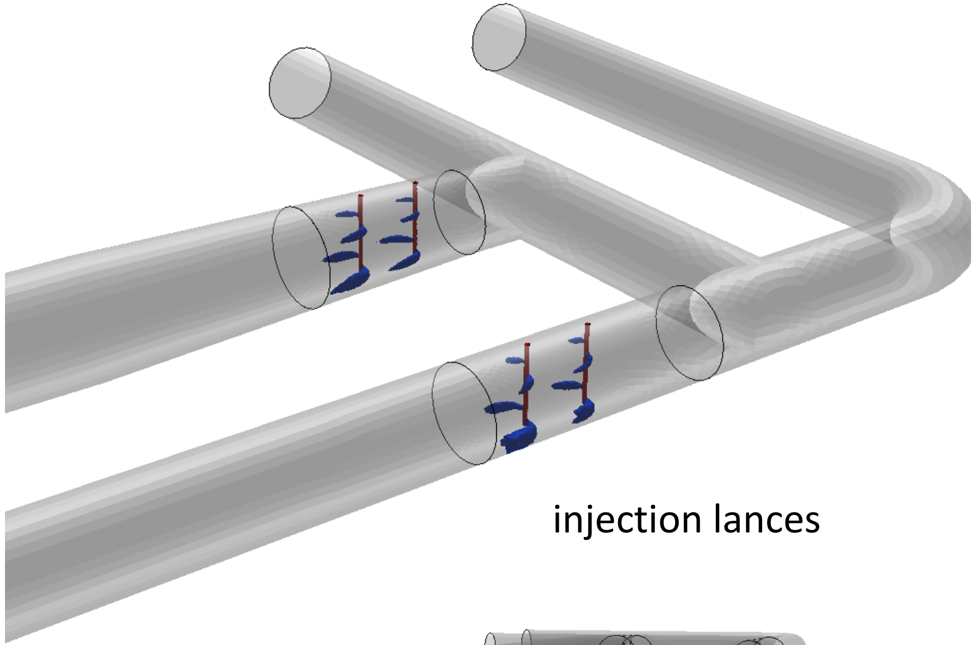
IBIS Group Inc.

weftec® 2016 WORKSHOPS

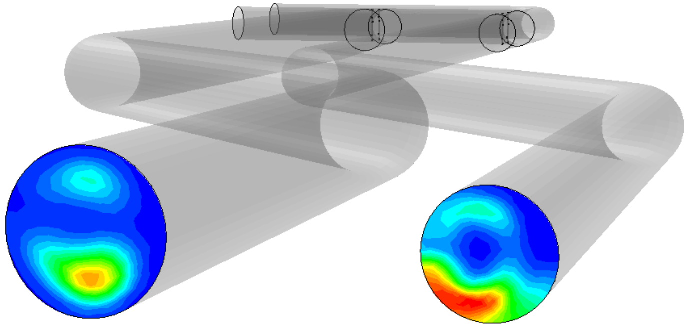
Passive Mixing Systems



static mixer in an inflow pipe

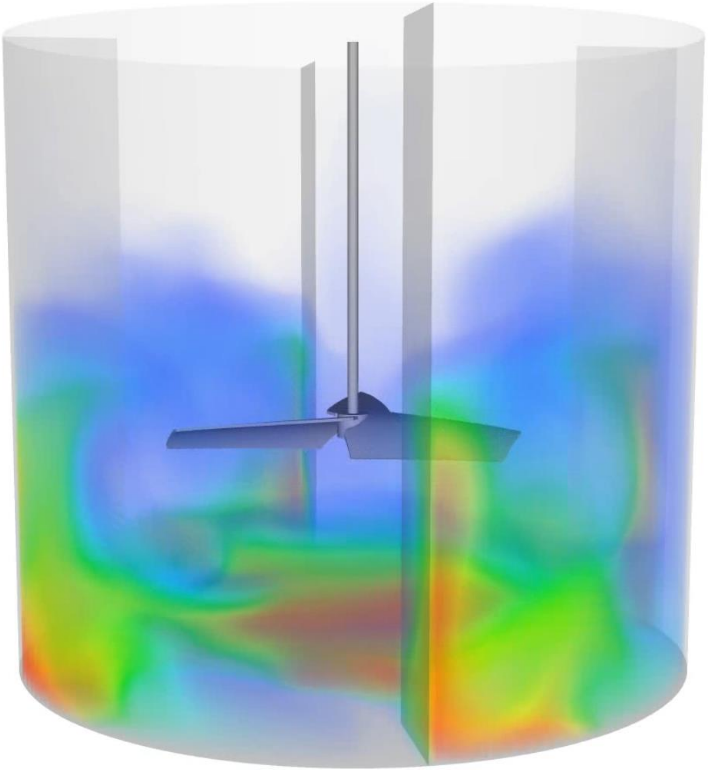


injection lances

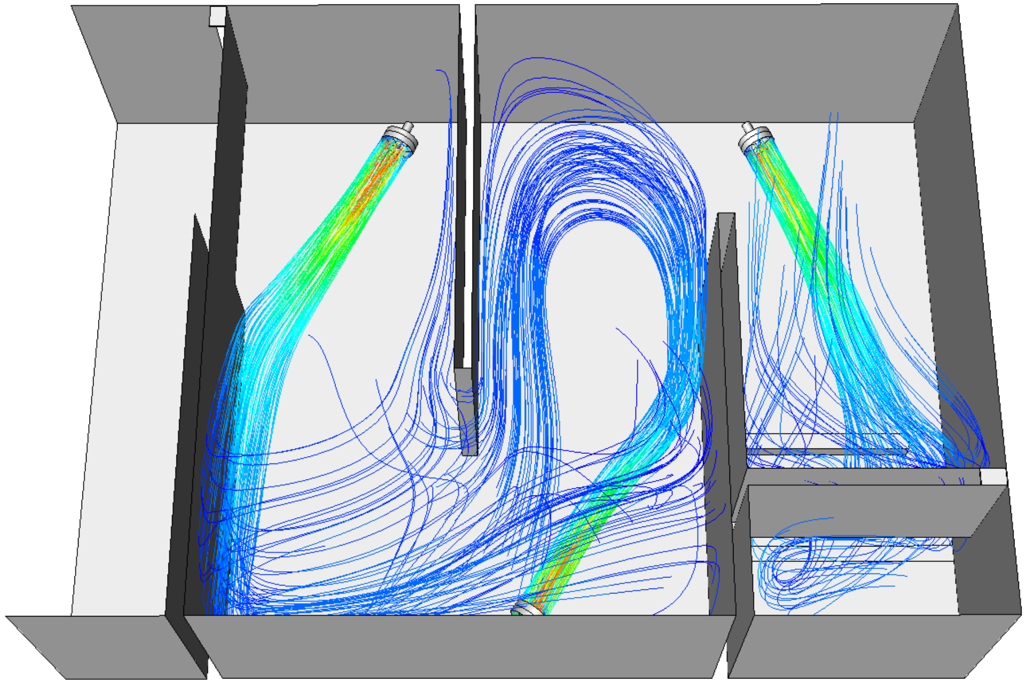


degree of mixing downstream of injection lances

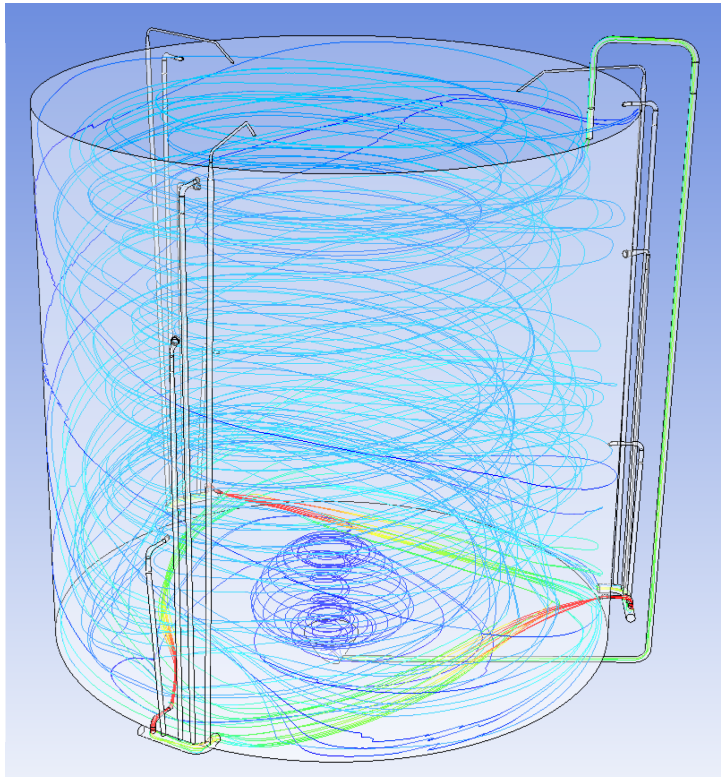
Active (Mechanical) Mixing Systems



top entry impeller



submersible mixers



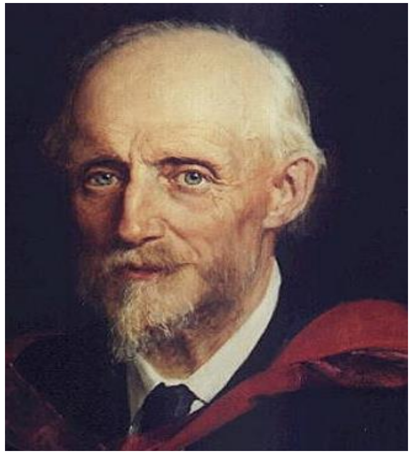
pumped jet mixers

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

$$Re = \frac{\rho v l}{\mu}$$

•rotating elements
•internal flows

Moritz Weber

We: ratio between inertial and surface tension forces

$$We = \frac{\rho v^2 l}{\sigma}$$

•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

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.

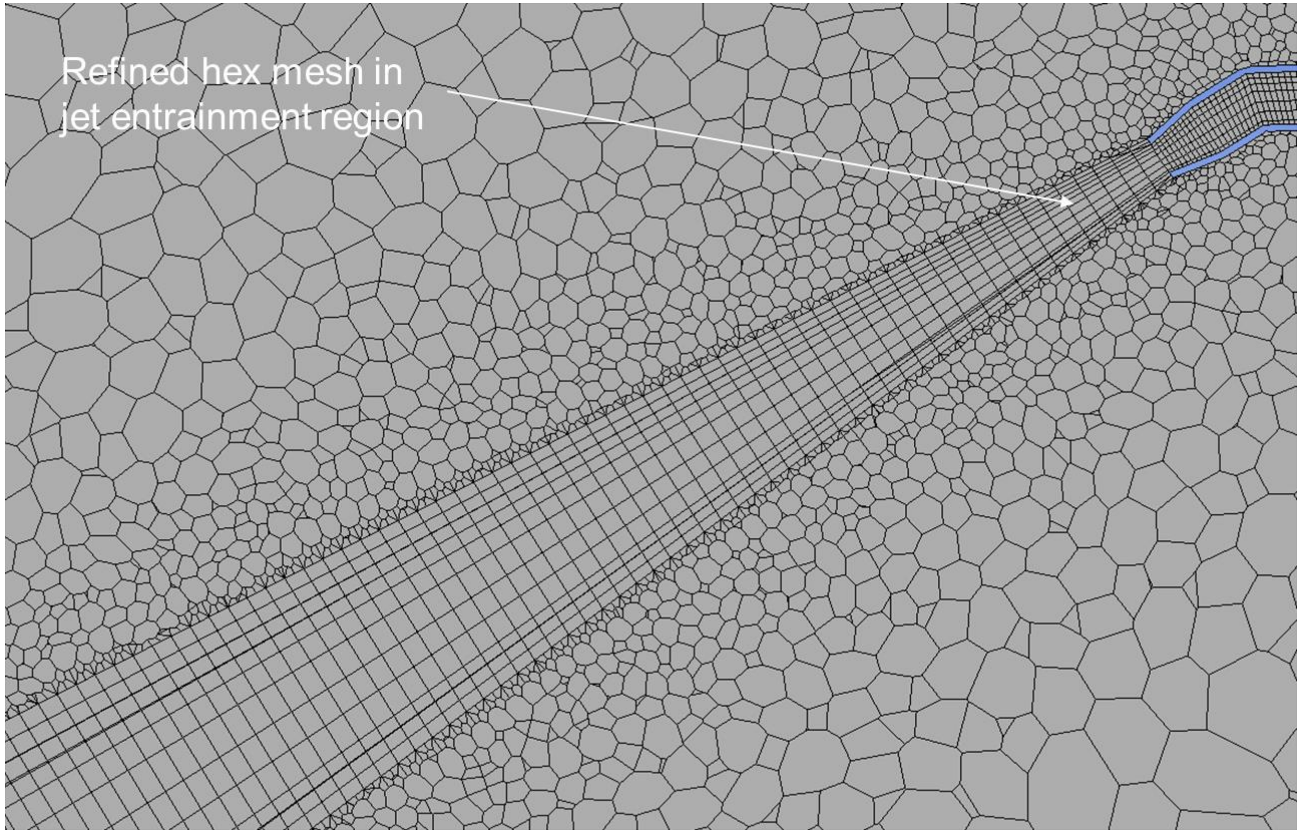
- Through flow models
  - No recirculation means high water consumption and large quantities of water discharging into the local sewer system
  - Receiving utility may take issue with tracer chemicals used

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.



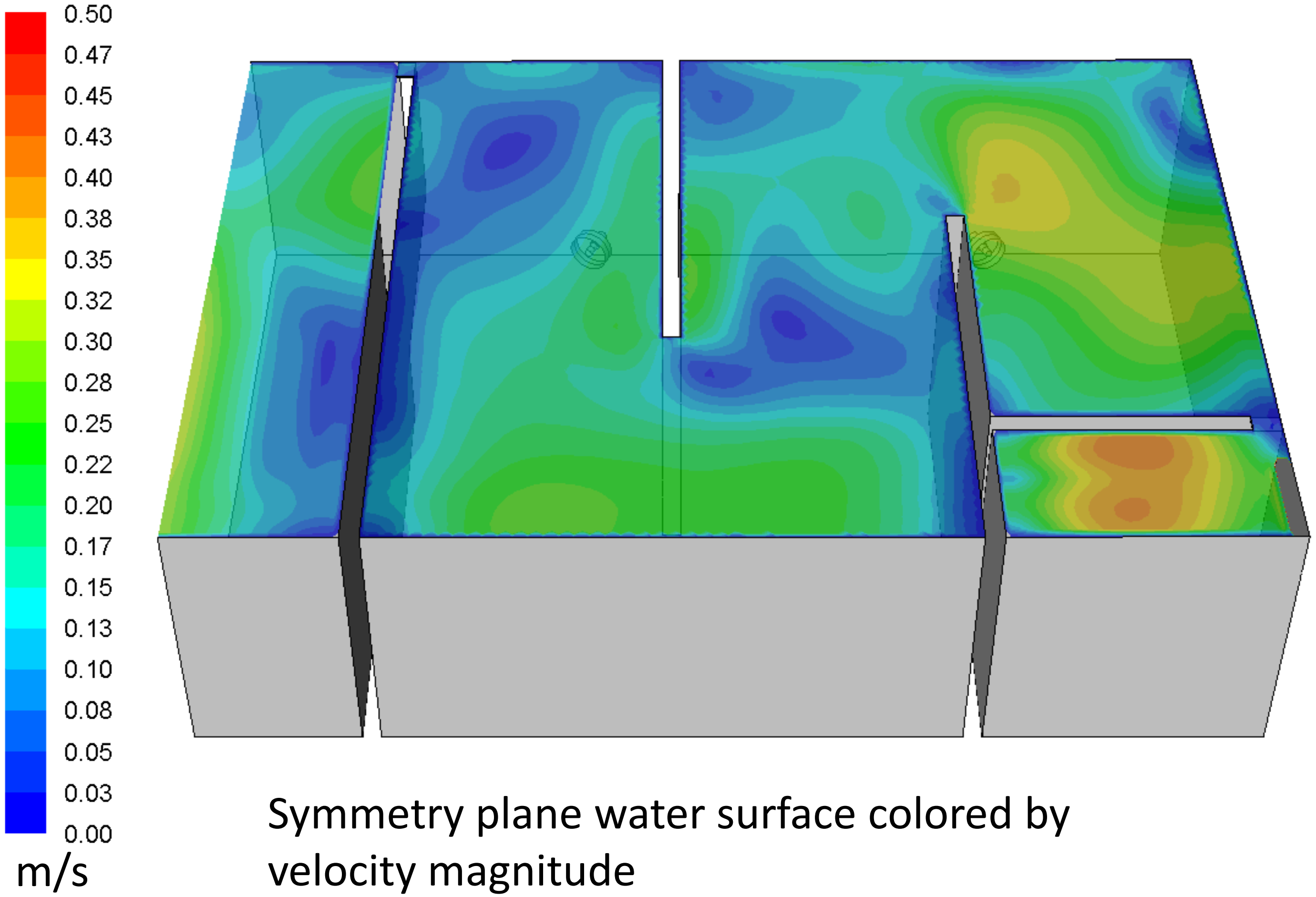
CFD Simulation for Mechanical Mixing Systems

Boundary Conditions – Water Surface

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.



Symmetry plane water surface colored by velocity magnitude

CFD Simulation for Mechanical Mixing Systems

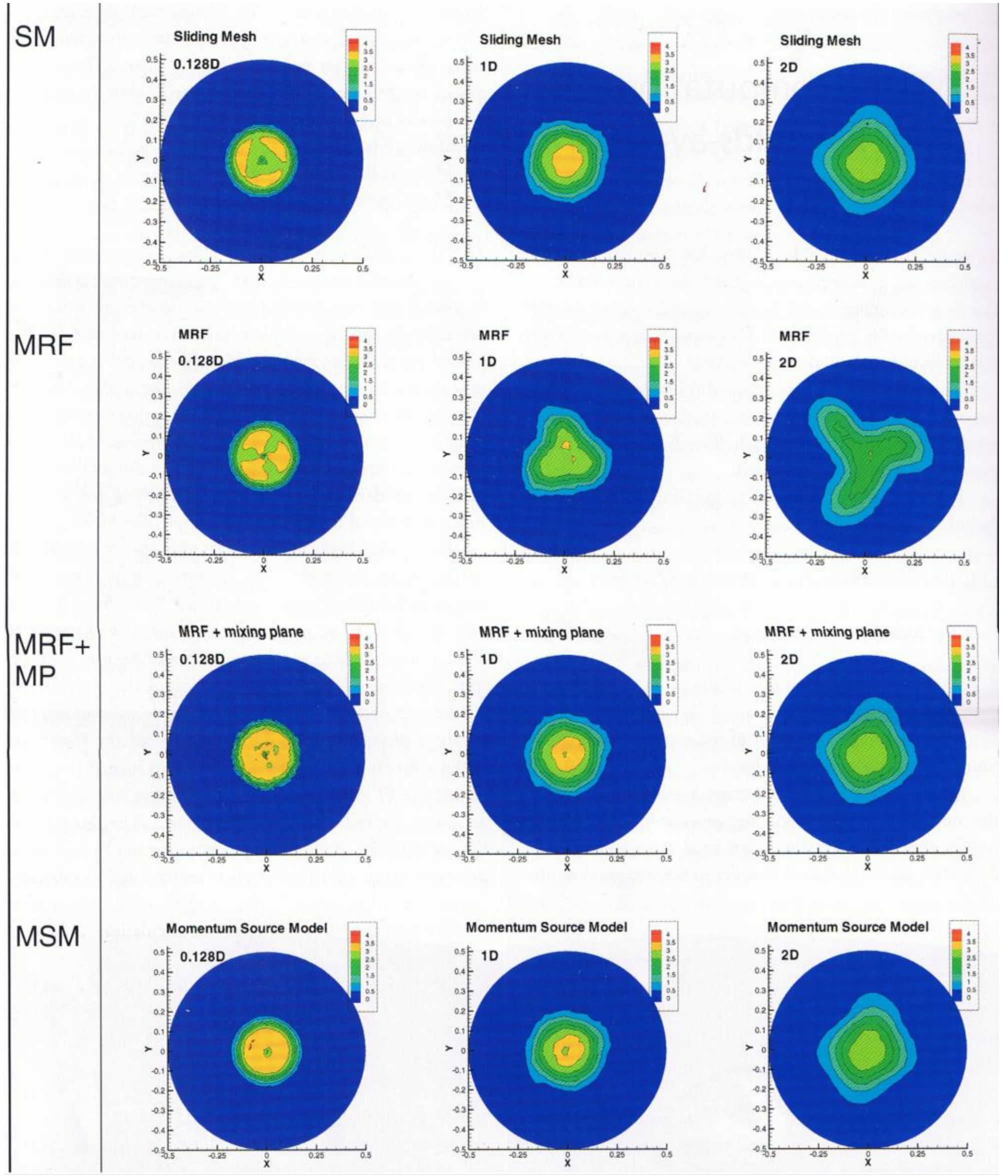
Boundary Conditions – Propeller Mixer Input

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

CFD Simulation for Mechanical Mixing Systems

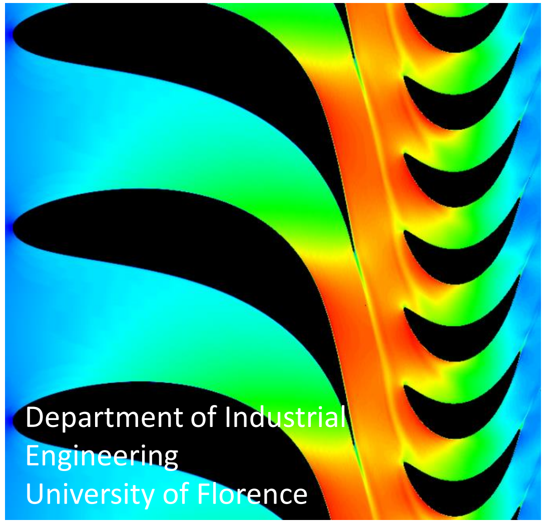
Boundary Conditions – Propeller Mixer Input

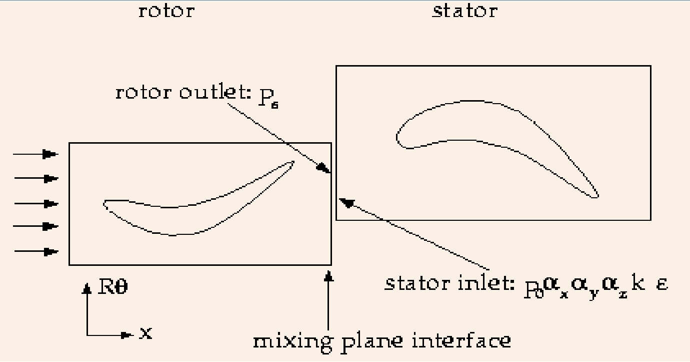
What is a mixing plane?

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

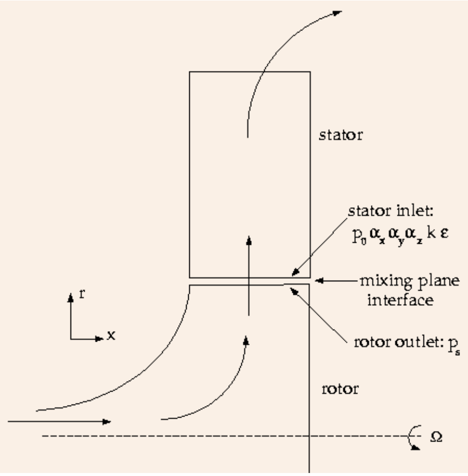
Blends out wakes of individual blades by circumferentially averaging flow properties.

Facilitates the use of small steady state models for rapid optimization of blade shapes





Mixing plane used in an axial machine



Mixing plane used in centrifugal machine

Ansys Inc. Fluent User Manual

CFD Simulation for Mechanical Mixing Systems

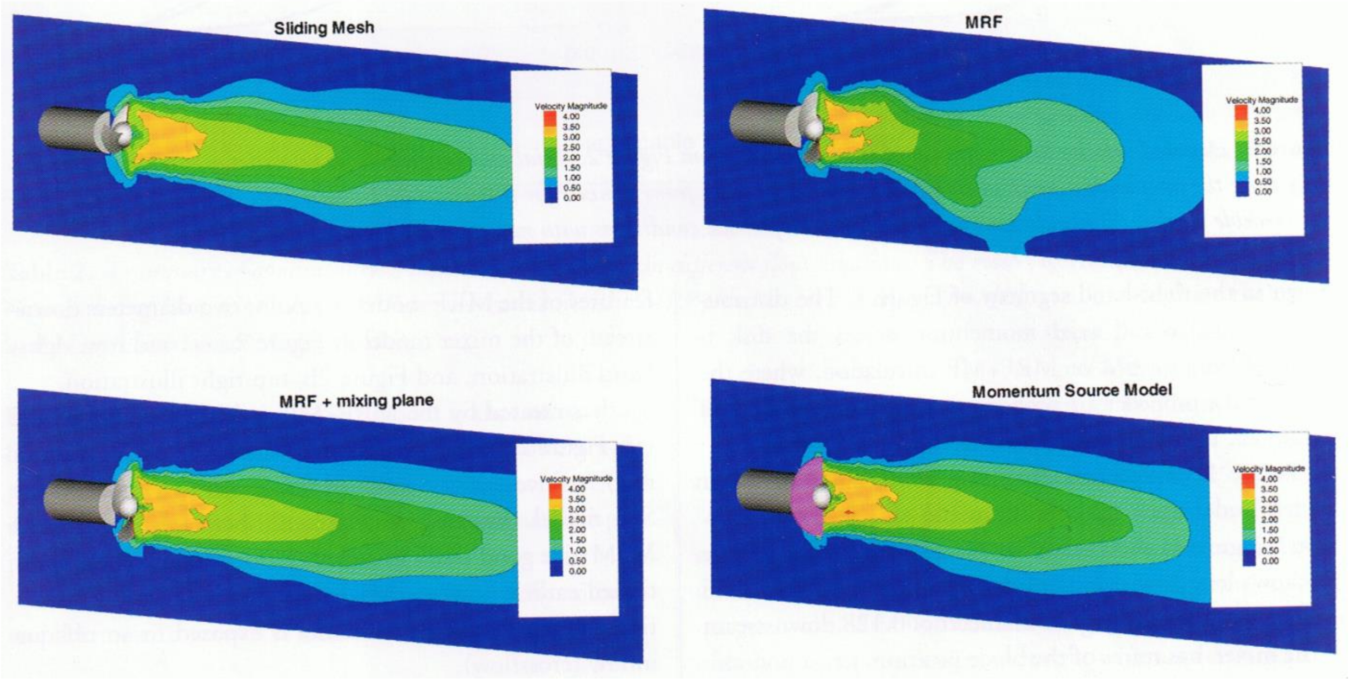
Boundary Conditions – Propeller Mixer Input

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.



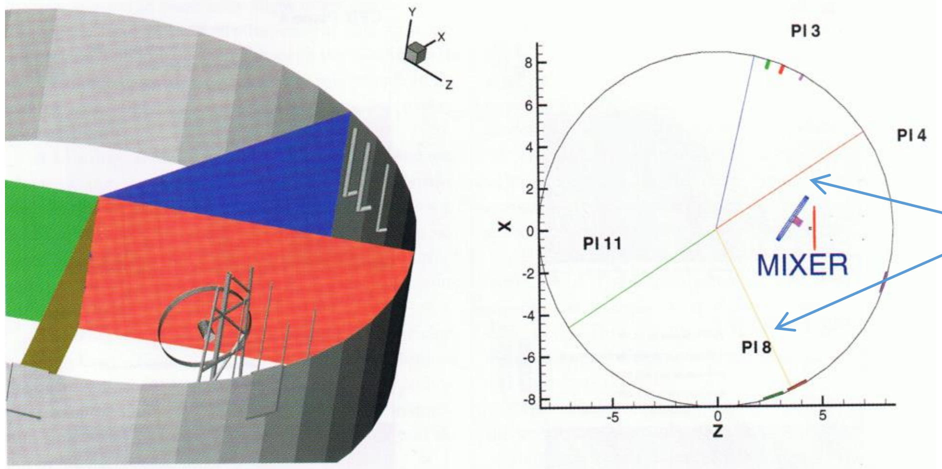
Börgesson, T. Fahlgren, M. *Mechanics of Fluids in Mixing*, 2001

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.

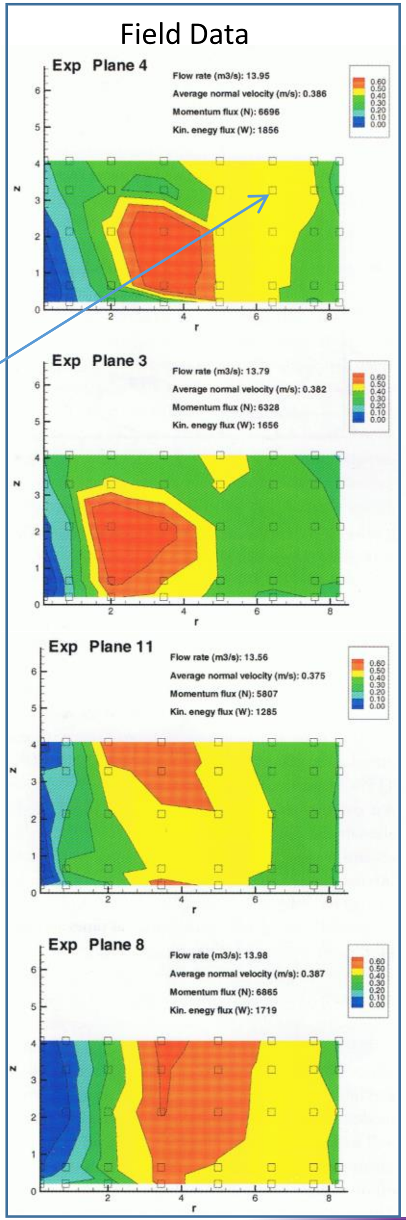


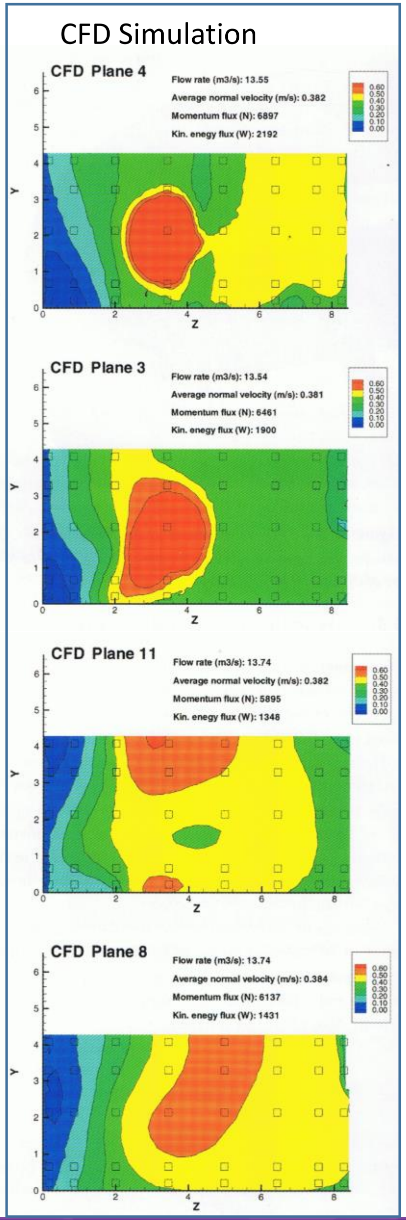
measurement planes

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

Börgesson, T. Fahlgren, M. *Mechanics of Fluids in Mixing-Bringing Mathematical Modeling Close to Reality*, Scientific Impeller No. 6, 2001

probe locations superimposed on contour plots





CFD Simulation for Mechanical Mixing Systems

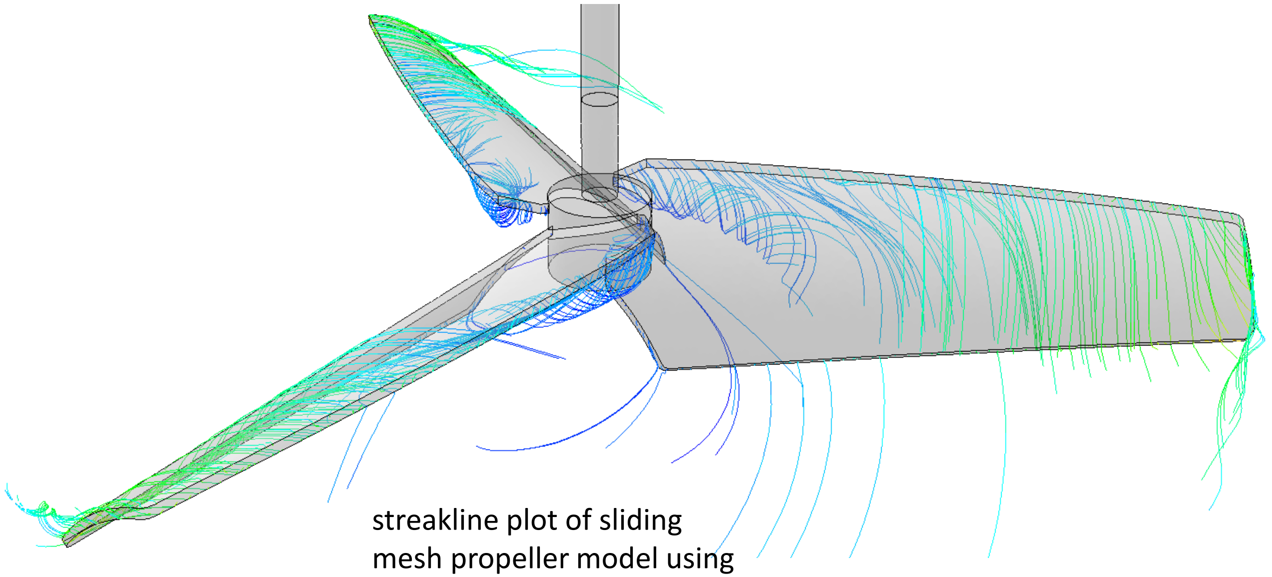
Simulation Parameters

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.

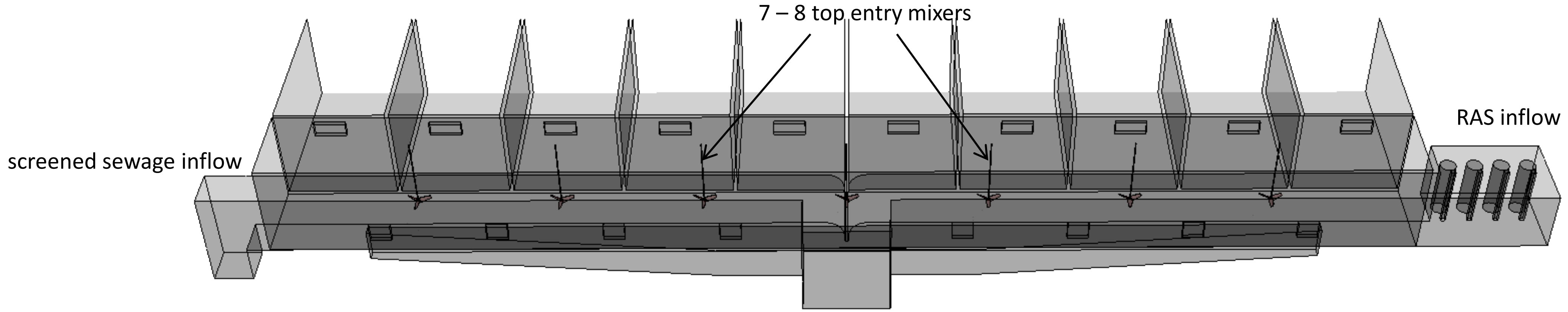
If suspended solids content is known, their presence should be simulated by means of a user defined function that includes density coupling.

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.



streakline plot of sliding mesh propeller model using RKE turbulence equations

Case Study: Common Anoxic Basin Feeding 10 Parallel Aeration Lanes



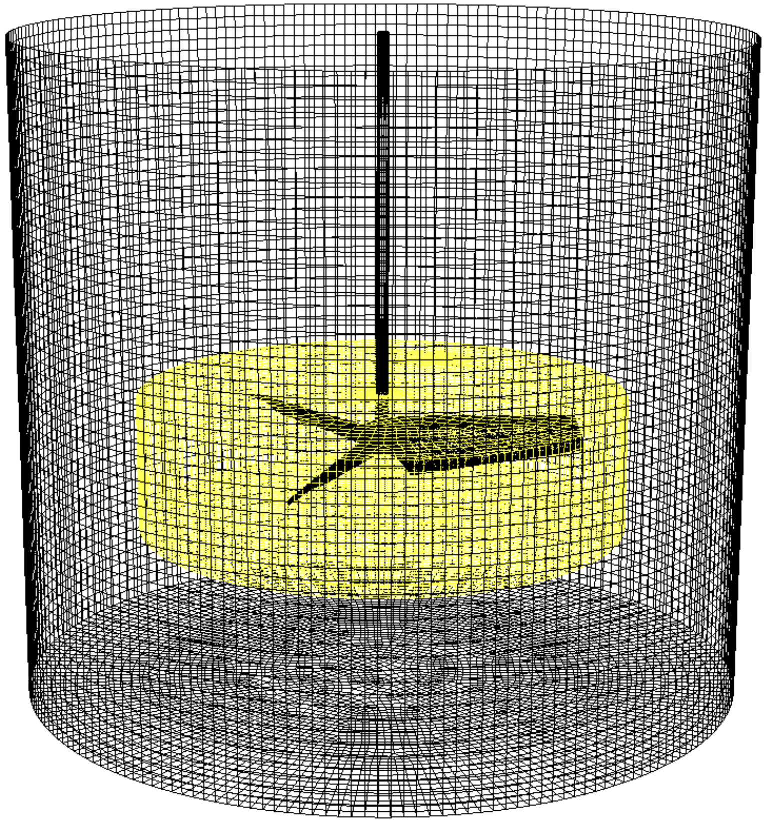
The real world basin has 7 or 8 (depending on brand selected) top entry mixers.

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.

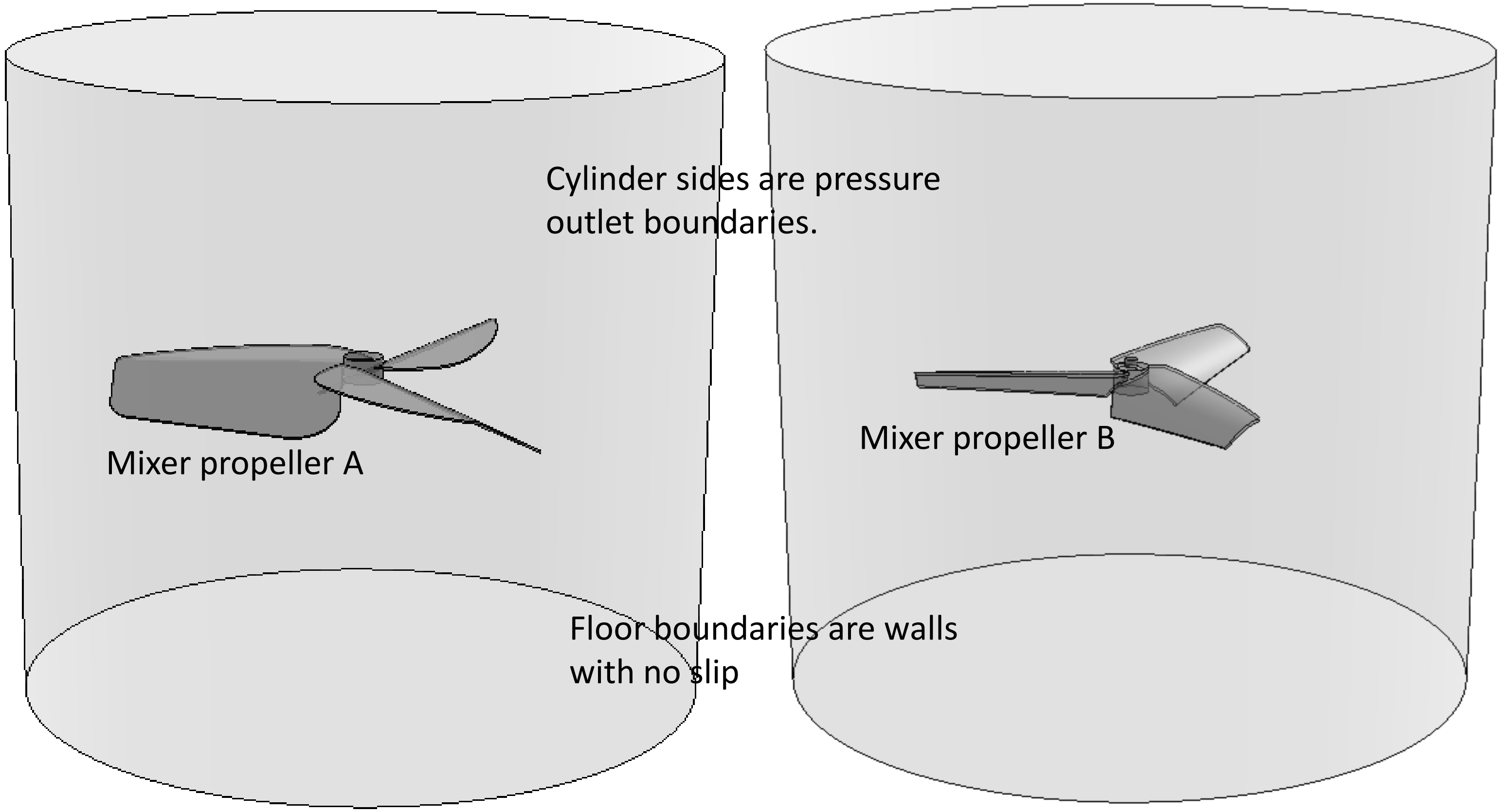
Case Study: Common Anoxic Basin Feeding 10 Parallel Aeration Lanes

Sliding Mesh Propeller Sub Models

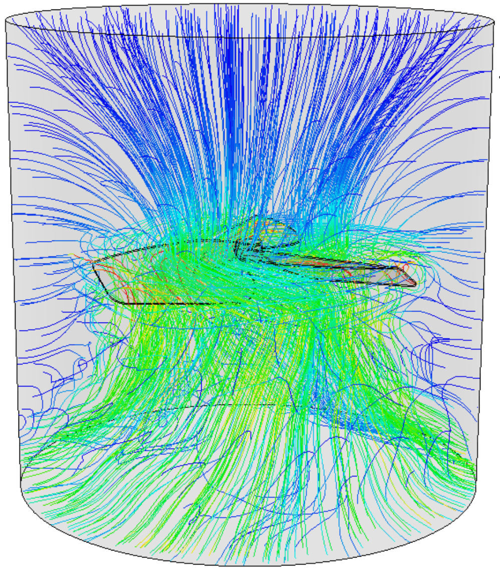
Hybrid mesh with structured hex in the cylinder and unstructured tet in the propeller domain



Tops are pressure inlet boundaries.

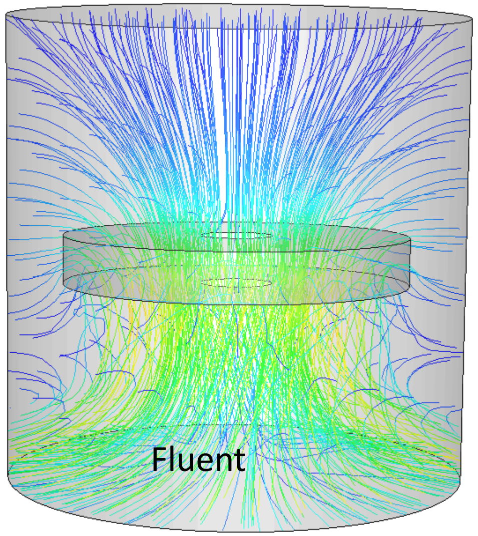


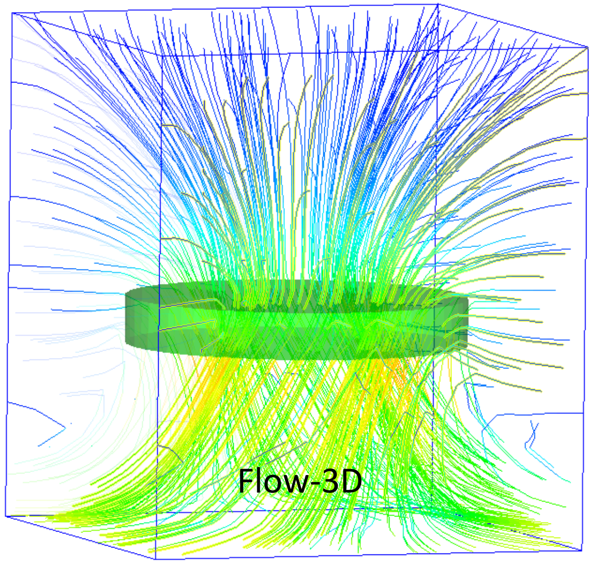
Case Study: Common Anoxic Basin Feeding 10 Parallel Aeration Lanes

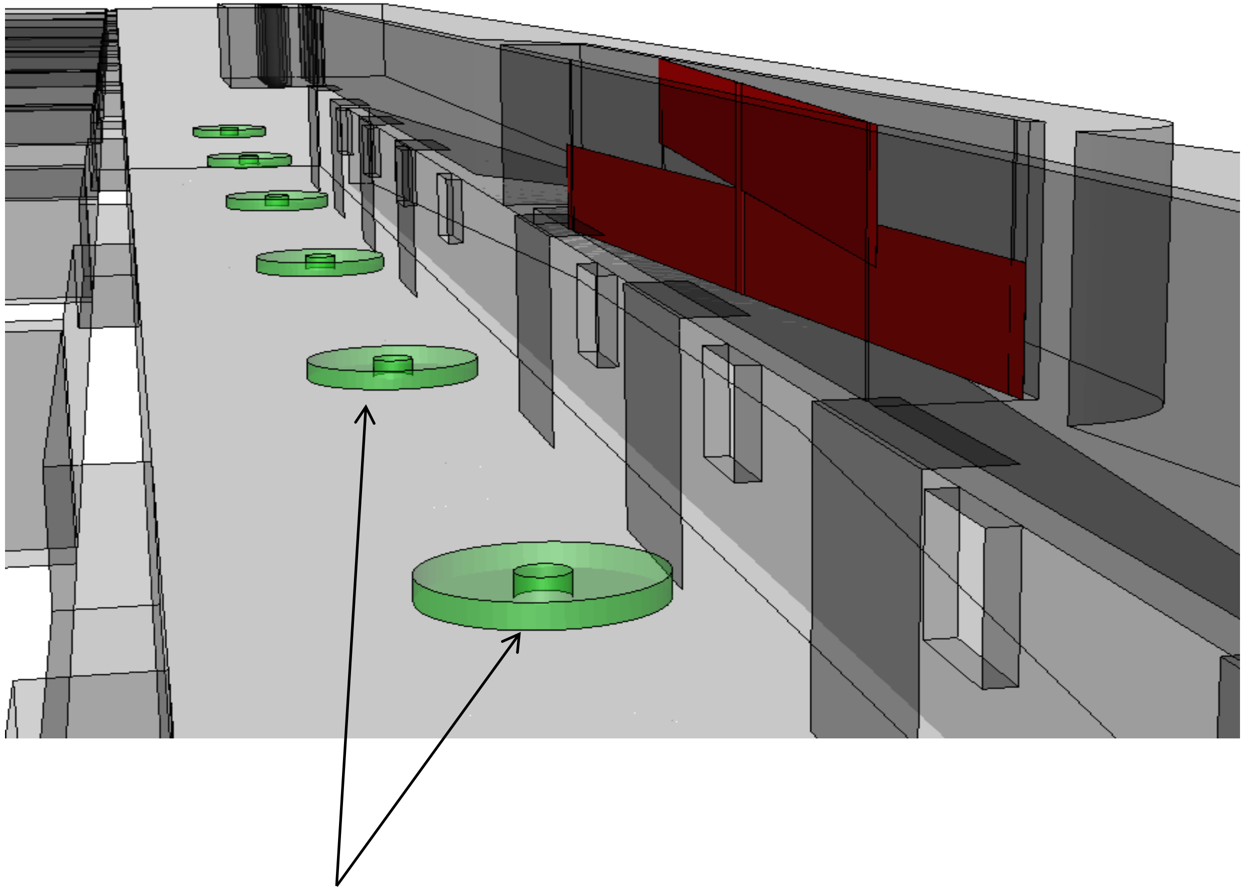


frozen in time snapshot of sliding mesh model

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

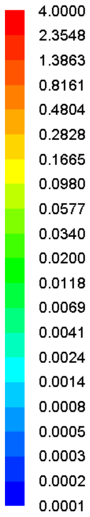
Mixing Efficacy Observed with Streakline Plots

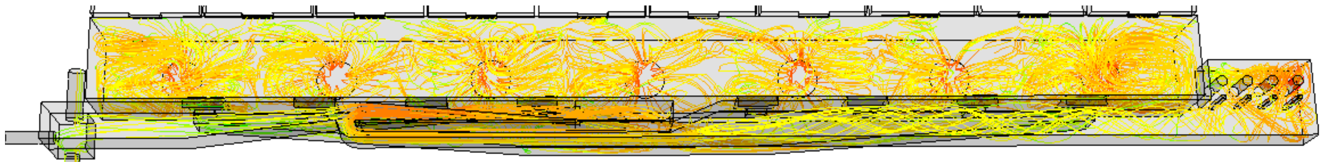
Streaklines show bulk flow patterns and accelerations

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.

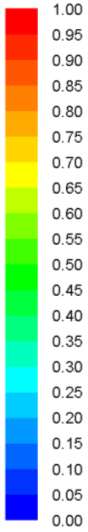
Quantitative results can be recorded by applying some statistics to the data base.

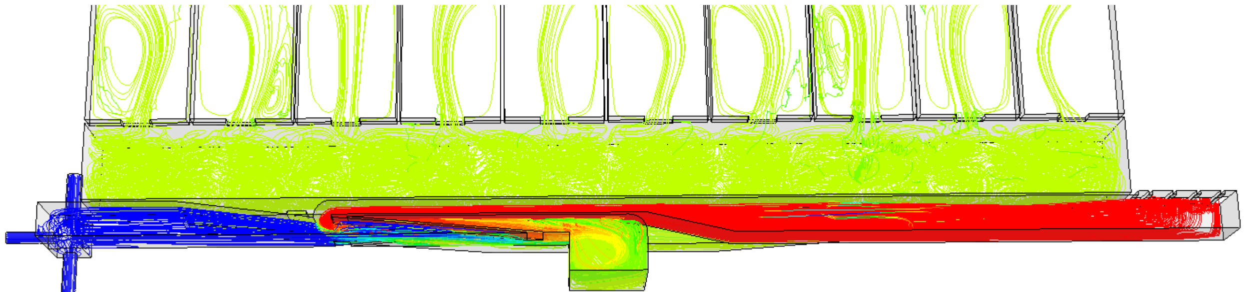




Streaklines colored by velocity magnitude

logarithmic scale allows for the illustration of a broad range of velocity magnitudes





Streaklines colored by tracer concentration

Case Study: Common Anoxic Basin Feeding 10 Parallel Aeration Lanes

Looking at the Results

Hydraulic Efficiency

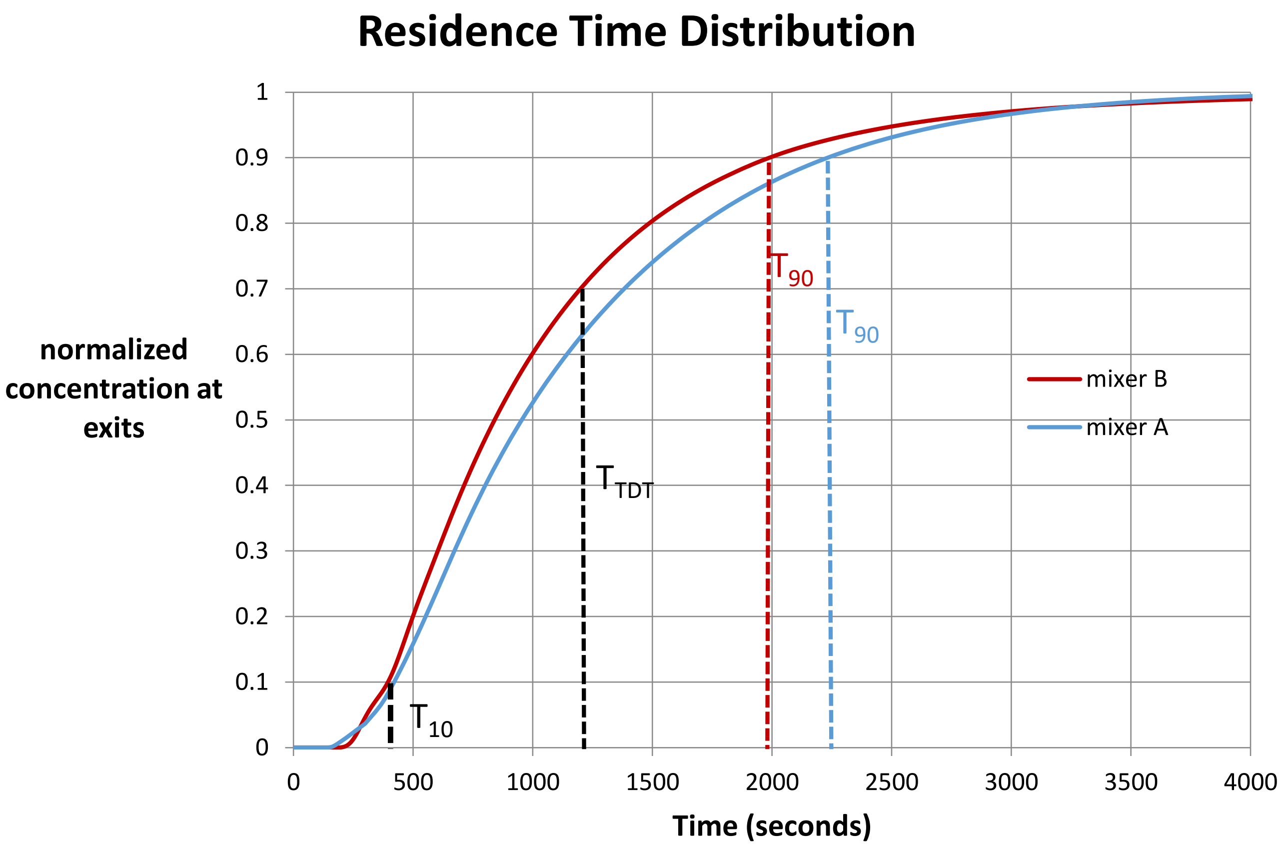
T_{TDT} - (theoretical detention time) system volume / inflow rate
1186 seconds for this case

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: $BF = T_{10} / T_{TDT}$ indicates degree of short circuiting
$BF_{mix.A} = 0.37$ $BF_{mix.B} = 0.33$ Values 0.3 and lower indicate significant short circuiting is occurring.

Morrel Number: $M = T_{90} / T_{10}$ indicates presence of long term recirculation zones $M_{mix.A} = 5.68$ $M_{mix.B} = 5.01$ Values greater than 5.0 indicate some flow is getting hung up in recirculation zones.



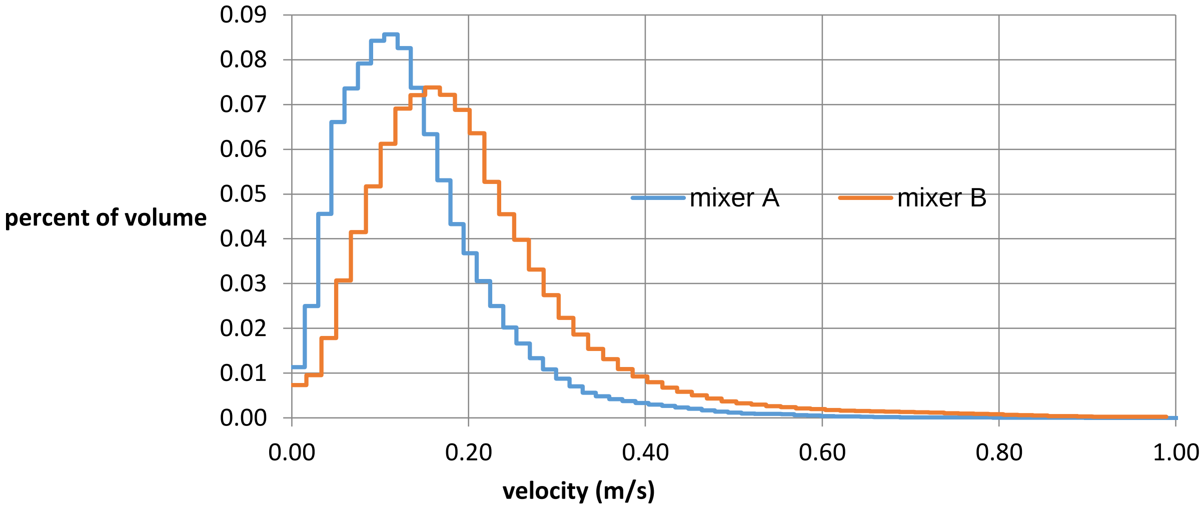
Case Study: Common Anoxic Basin Feeding 10 Parallel Aeration Lanes

Looking at the Results

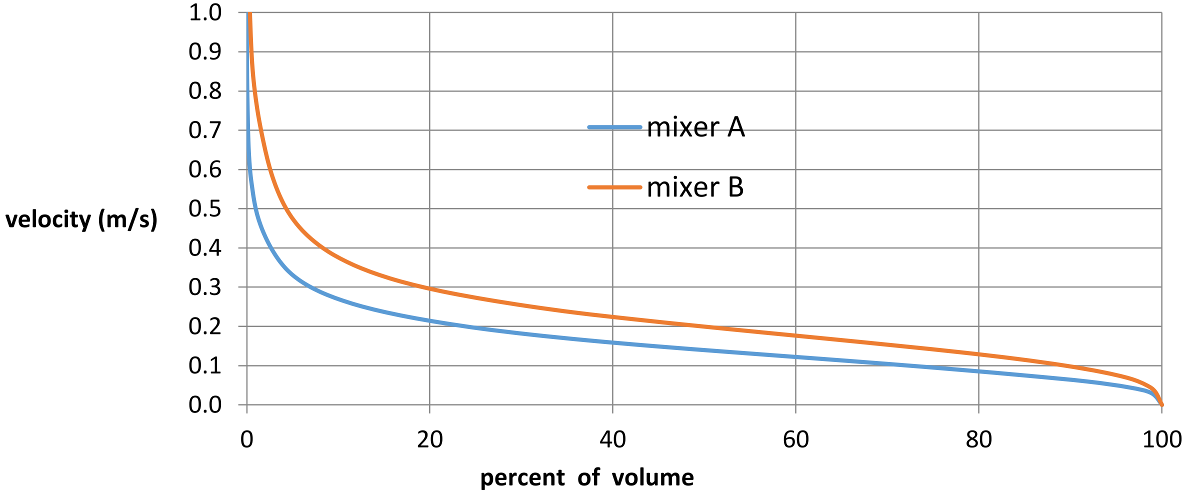
Velocity Distribution

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/